RISAFloor

Rapid Interactive Structural Analysis – Floor Building Systems

Tutorials

RISA Tech, Inc.
27442 Portola Parkway, Suite 200
Foothill Ranch, California 92610

(949) 951-5815
(949) 951-5848 (FAX)
risa.com

Table of Contents

Introduction 1

How to Use this Book 1

Using the Online Help 3

Technical Support Information 4

RISA Online 4

Before You Begin 5

RISAFloor Overview 5

Hardware Requirements 5

Program Limits 6

License Agreement 7

Installation 7

First Look at RISAFloor 8

Starting RISAFloor 8

Windows and Dialog Boxes 8

Menus and Toolbars 10

Managing Windows, Model Views, and Spreadsheets 14

Working in Spreadsheets 15

Part A: Beam Supported Floors 16

Part A: Tutorial 1 – Columns and Walls 17

Starting a New File 17

Project Grid 19

Model Settings 22

Columns 24

Wall Panels 32

Part A: Tutorial 2 – Modeling Beams 39

Primary Framing 40

Infill Framing 45

Additional Beam Drawing Tips (optional) 49

Part A: Tutorial 3 – Decks & Diaphragms 53
Creating Diaphragm Edges 53
Deck Properties 57
Part A: Tutorial 4 – Loading 64
Area Loads 65
Line Loads 71
Point Loads 76
Part A: Tutorial 5 – Solution & Results 78
Load Combinations 79
Solving and Reviewing Results 82
Modifying the Model 87
Part A: Tutorial 6 – RISA-3D Integration 89
RISA-3D Lateral Load Generators 91
RISA-3D Load Combinations 98
Part A: Tutorial 7 – RISA Interoperability 102
RISA-Revit Link 102
CIS/2 Translator 105
RISAFloor & CAD 106
Conclusion 109
Part B: Elevated Slab Design 110
Part B: Tutorial 1 – Columns & Walls 112
Starting a New File 112
Drawing Grid 114
Columns 117
Wall Panels 125
Wall Panel Openings 130
Part B: Tutorial 2 – Slab & Design Strips 134
Draw the Slab 135
Modifying Your Slab 139
Creating a Slab Opening 146

Support Lines and Design Strips 150
Part B: Tutorial 3 – Loading 163
Area Loads 164
Line Loads 171
Point Loads 173
Part B: Tutorial 4 – Solution & Results 177
Load Combinations 178
Solving the Model 180
Spreadsheet Results 185
Viewing Graphical Results 187
Detail Reports 190
Part B: Tutorial 5 – RISA-3D Integration 197
Lateral Load Generators 198
RISA-3D Diaphragms 201
RISA-3D Loading 202
RISA-3D Load Combination Generator 203
Part B: Tutorial 6 – DXF Export 209
DXF Export 210
DXF Import 211
Appendix A – RISAFloor Toolbar Button Quick Reference 213
RISA Toolbar 213
Window Toolbar 215
Drawing Toolbar 219
Selection Toolbar 221

Introduction

How to Use this Book

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

Begin by reviewing First Look at RISAFloor on page 8 to familiarize yourself with the RISAFloor menus, toolbars, and shortcuts. Appendix A – RISAFloor Toolbar Button Quick Reference has also been included on page 213 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 – Beam Supported Floors will guide you step-by-step through the RISAFloor modeling process to build and analyze a model using beam supported floors; Part B – Elevated Slab Design will guide you through building and analyzing a model with elevated slab floors in RISAFloor ES. In each part, you will create a real-world building example and solve, make changes, and optimize 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.rfl**, **Tutorial A3 Starter.rfl**, etc.

After you have completed the tutorials in this guide, you can use the Help Menu and *RISAFloor General Reference* for complete, detailed information on every topic relating to RISAFloor. 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 to apply RISAFloor features such as columns, beams and walls, but the specifics of how those elements are designed are covered in the Help Menu and the *RISAFloor General Reference*.

Where to Download RISAFloor 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 https://www.risa.com/documentation.

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.

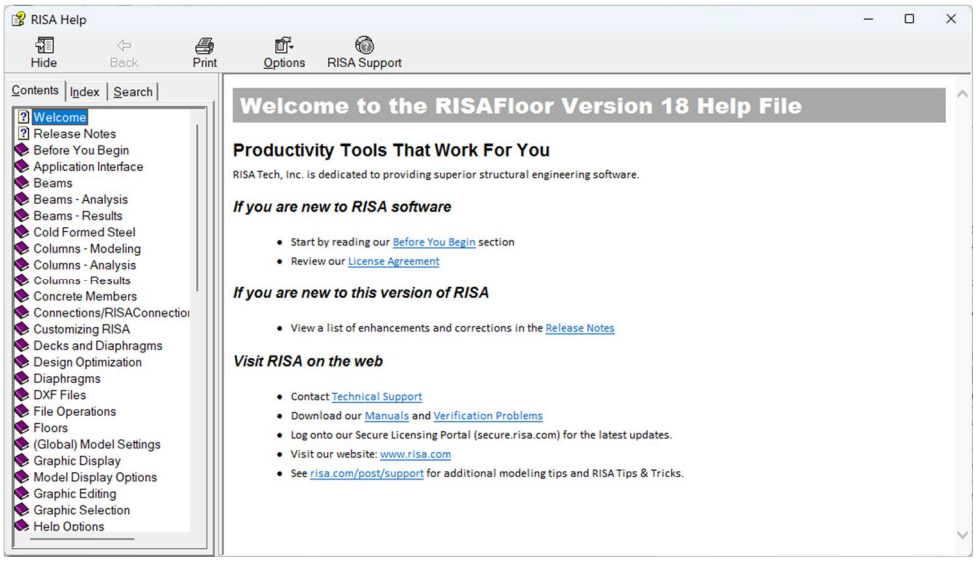
Using the Online Help

Whether you need help on general topics, specific features, or toolbars, it is all built in to the extensive RISAFloor online Help system. The RISAFloor 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.



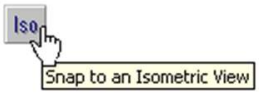
Help on a specific feature (context-sensitive help)

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 is for? 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 RISAFloor is released, the life of the previous version is considered to be ended. Technical support is a limited resource; priority will always be given to those clients whose licenses are current first.

RISA Tech, Inc. will only support the current version of RISAFloor. 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 Tips & Tricks database by visiting our website at http://www.risa.com. Click **Learn**, then **Tips & Tricks**, and then choose **RISAFloor**.

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.
- Your model (filename.rfl) 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:

- Helpful Tips & Tricks
- 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 problems

Before You Begin

RISAFloor Overview

RISAFloor has been developed to make the definition, design, and modification of building systems fast and easy. Analysis (including calculation of deflections and stresses) may be performed on simple single-story buildings or on larger multi-story structures. Plus, element design optimization is provided for columns, beams, and wall panels.

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

In RISAFloor, these two methods of entering and editing data work seamlessly together. 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.

RISAFloor ES is an optional extension of RISAFloor which includes the added feature of elevated concrete slab floor design. With this added feature, models can intermix elevated slab and beam supported floors. Reinforcement design optimization is provided on slab floors for columns, beams, wall panels, and flat slabs.

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)

Note: RISAFloor ES is only available for 64-bit operating systems

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 for 64 bit computers
- 4 GB of hard disk space

Hardware Limitations

- RISAFloor is not a multithreaded application, which means that it runs entirely within a single processor core. Therefore, the program does not take full advantage of multi-core or multi-processor machines. This limitation affects how long it takes to solve a model.

Program Limits

Floors	200
Walls (per Entire Model)	10,000
Openings/Regions (per Wall Panel)	25/100
Columns per Floor	2,500
Beams per Floor	10,000
Points per Floor	25,000
Point Loads per Floor	10,000
Line Loads per Floor	10,000
Area Load Polygons per Floor	1,000
Deck Polygons per Floor	1,000
Materials	500
Custom Wood Species	500
Load Combinations	5,000
Concrete Slabs per Floor (RISAFloor ES Only)	20
Support Lines per Slab (RISAFloor ES Only)	300

Demonstration Version: You can open and solve a model of any size, however you may not:

- Use this version for Professional Design Purposes
- Create more than 2 Concrete Floor Slabs
- Create a Concrete Floor Slab other than 8" thick, 3000 psi concrete
- Save a model with more than 40 beams per floor
- Save a model with more than 3 Floors
- Save a model with more than 1 Concrete Floor Slab
- Save a model with more than 20 columns or 4 walls
- Save a model with more than 10 Support Lines
- Import RISAConnection design results back into the RISAFloor demo.

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 RISAFloor, 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.

RISAFloor Customization–Important Assumption!

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

First Look at RISAFloor

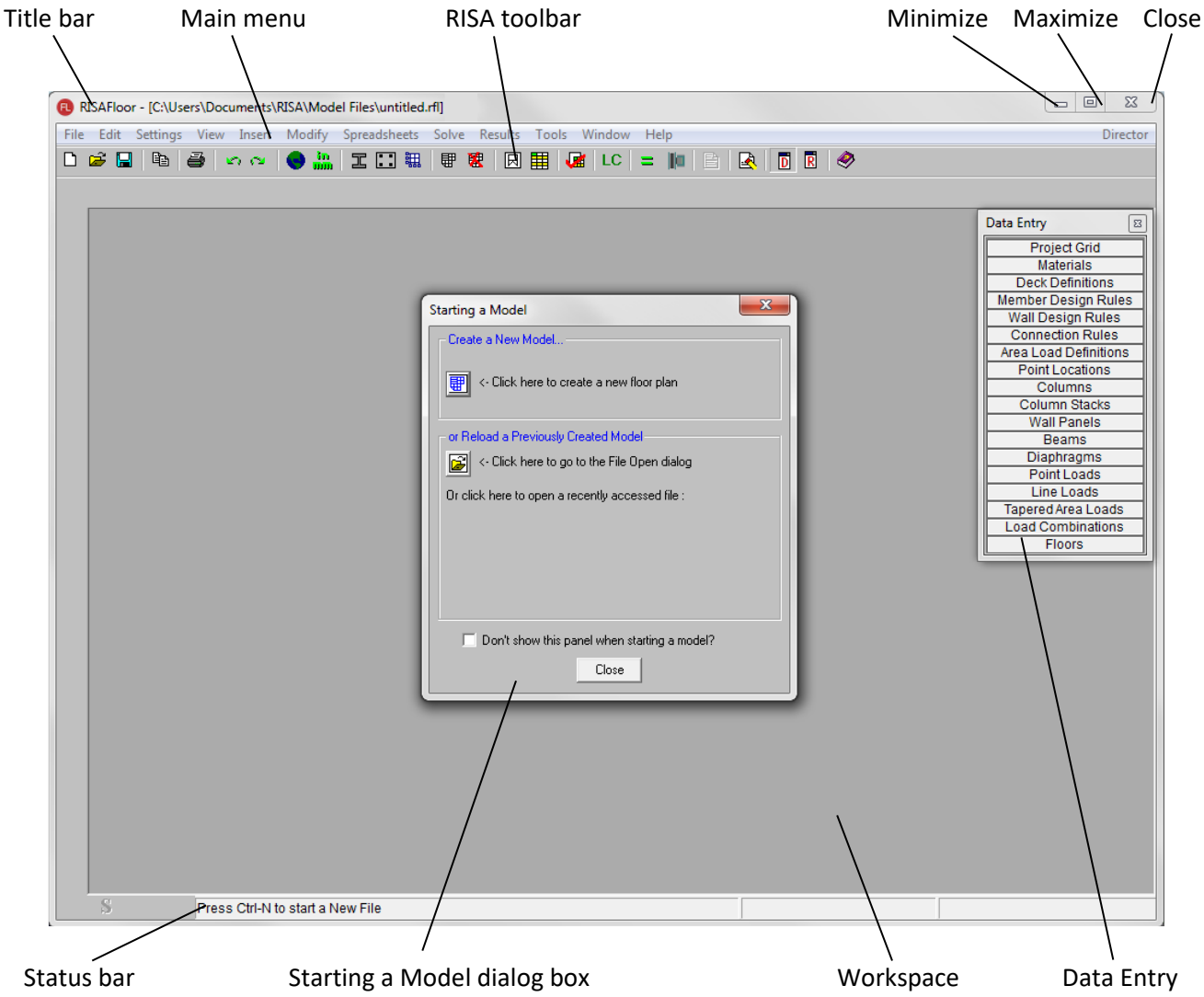
Starting RISAFloor

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

Start RISAFloor as follows:

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

Windows and Dialog Boxes



Title bar

The title bar at the top of your RISAFloor 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 RISAFloor 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 RISAFloor 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:

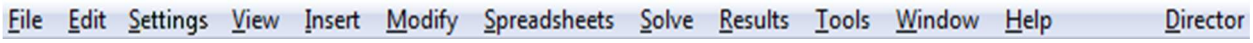
- The first status box displays general information relative to the task you are performing.
- The second (middle) status box reports information about your deck or area load.
- The third status box (on the far right) reports the cursor coordinates as you work in the model view. This will be demonstrated throughout the tutorial.

Dialog boxes

Dialog boxes are windows that help you perform a specific function within RISAFloor. For example, the **Starting a Model** dialog box is presented when you first open RISAFloor, 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 RISAFloor 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	Allows you to open model settings.
View	Allows you to open a new model view or adjust the current model view.
Insert	Used to insert drawing columns, beams, walls, 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 RISAFloor 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. Program **Application Settings** are also located here.
Window	Manages all of the windows that you have open in RISAFloor, whether they are spreadsheets or model views.
Help	Provides access to the RISAFloor online Help menu. For more information on Help, see Using the Online Help on page 3.
Director	This feature allows you to integrate your model with RISA-3D and/or RISAConnection if you also own those products.

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.

Main

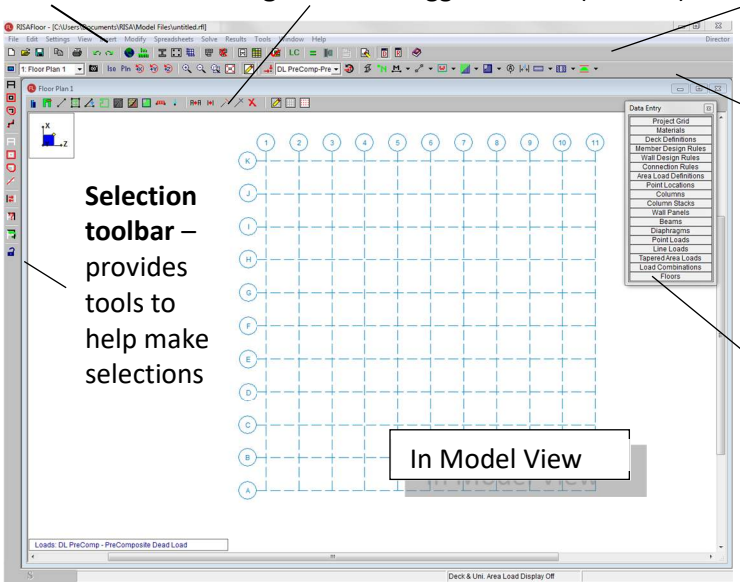
Drawing toolbar - Toggle on with (CTRL+G)

RISA toolbar – provides access to file operations, printing, changing design parameters, etc.

Window toolbar (in Model View) – contains viewing commands

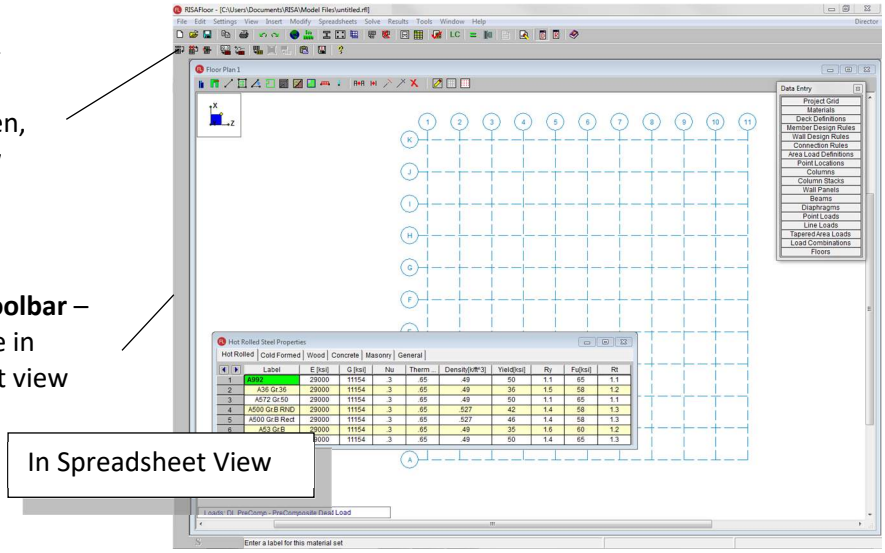
Selection toolbar – provides tools to help make selections

Data Entry toolbar - provides quick access to spreadsheets (then toolbars switch to spreadsheet view)
-AND/OR-
Results toolbar - After the model is solved, the results are displayed here



Window toolbar – any time a spreadsheet is open, notice the window toolbar changes

Selection toolbar – is not visible in 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 RISAFloor. 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

…on Beam Supported Floors

…on Concrete Floor Slabs

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.

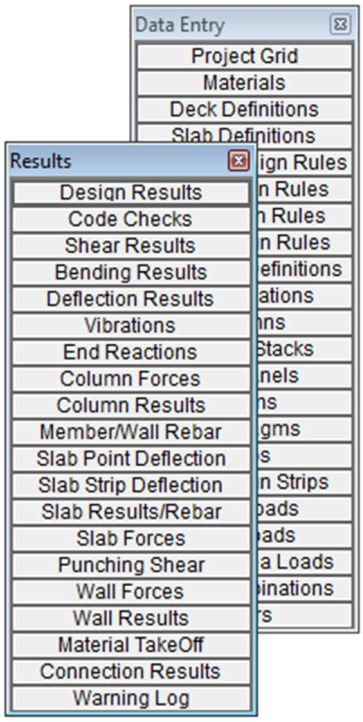
Selection Toolbar

…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.

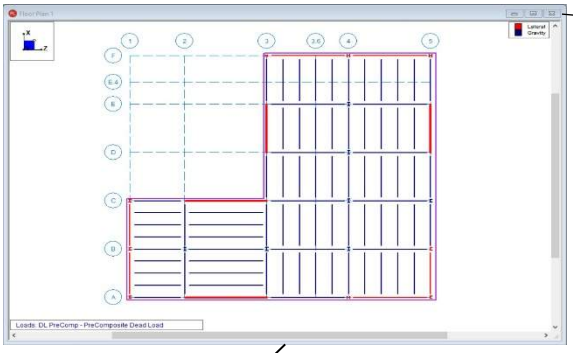
Both toolbars allow you to access the spreadsheets very quickly while building and solving your model. The buttons appear in the general order as you may need them.

Note: The toolbars above include several slab-specific spreadsheets that are only available in RISAFloor ES. For more information about these and how to use them, please see *Part B: Elevated Slab Design* on page 110.

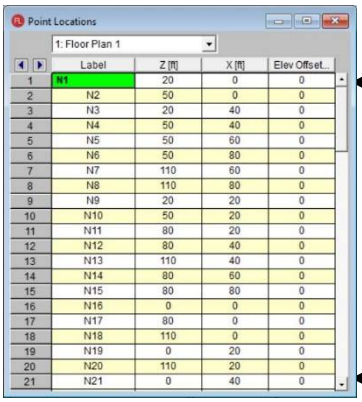
Managing Windows, Model Views, and Spreadsheets

Managing Windows

As you work in RISAFloor, 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 RISAFloor 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.



Minimize, Maximize, and Close buttons



Scroll arrows & scroll bar

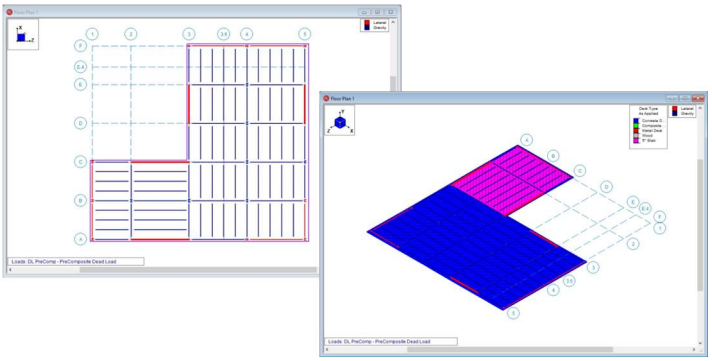
Scroll bar - to view information outside of the window's viewing area, click the scroll arrows or drag the scroll bar to move the display one way or the other.

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 RISAFloor vary depending on which window is active (the window with a colored title bar is the active window).

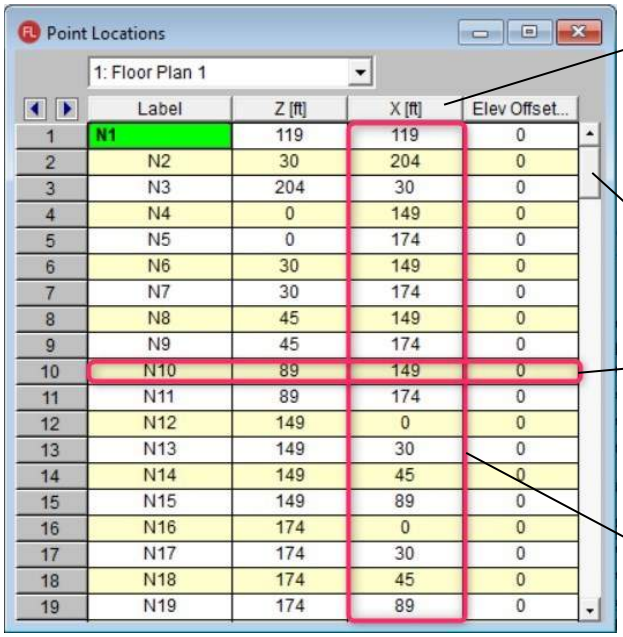
For example, if your active window is a spreadsheet, and you are looking for the zoom toolbar, you will not be able to locate it until you click your model, switching to model view. Then you will be able to access the zooming tools, and all the other tools related to modeling.



Working in Spreadsheets

Spreadsheets are composed 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.



Column label – Click the column label to select the entire column

Scroll bar

Row of cells – Click the row label (at left) to select the entire row

Column of cells

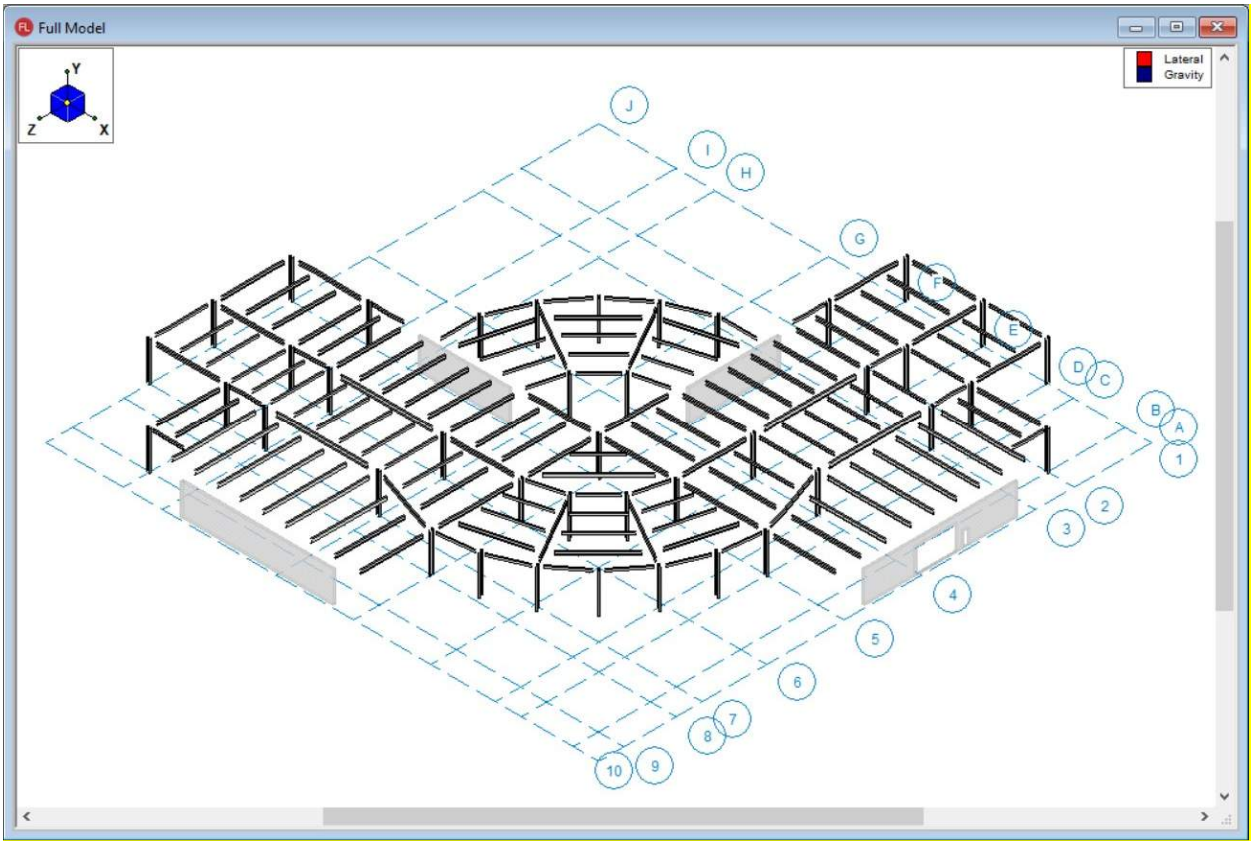
Click any cell. Notice the status bar displays an explanation of the current column.

Part A: Beam Supported Floors

This first part of this book (Part A) will focus on building beam supported floors within RISAFloor. With the guidance of the following seven tutorials, you will build, solve, and modify a typical building comprised of several different types of members and materials.

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 program folder under **Tutorials**, and are named **Tutorial A2 starter.rfl**, **Tutorial A3 starter.rfl**, etc.

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



To complete all seven 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 – Columns and Walls

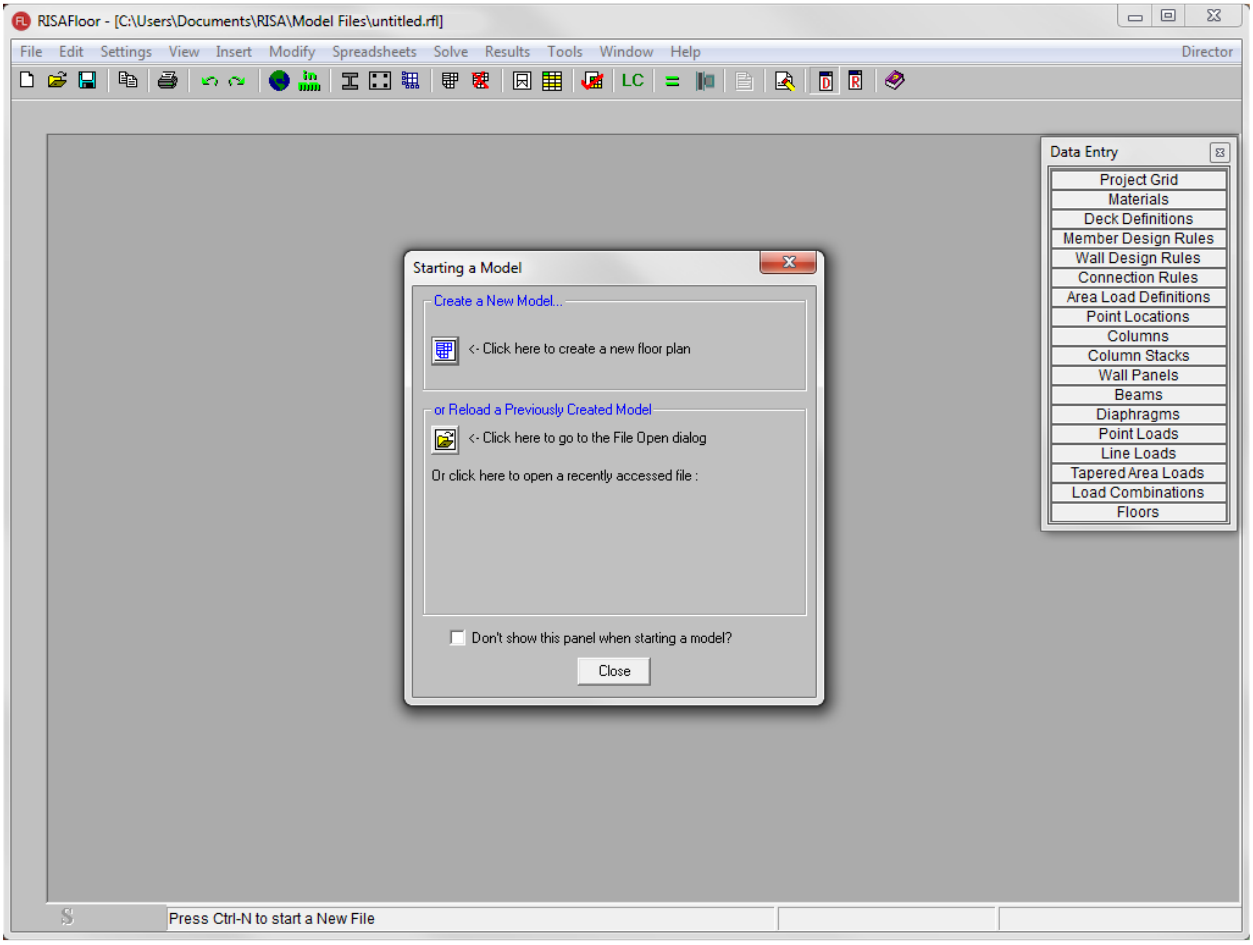
Overview

This first tutorial will introduce the various drawing features that RISAFloor has to offer. You will model a Project Grid, a Slab, several Footings and Grade Beams, and explore the Model Settings of the model.

Starting a New File

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

- On your desktop home screen, double click on the **RISAFloor** icon .



Note: The view shown above is from the Demonstration version of RISAFloor. If you are running the full version of the program, this dialog will look slightly different.

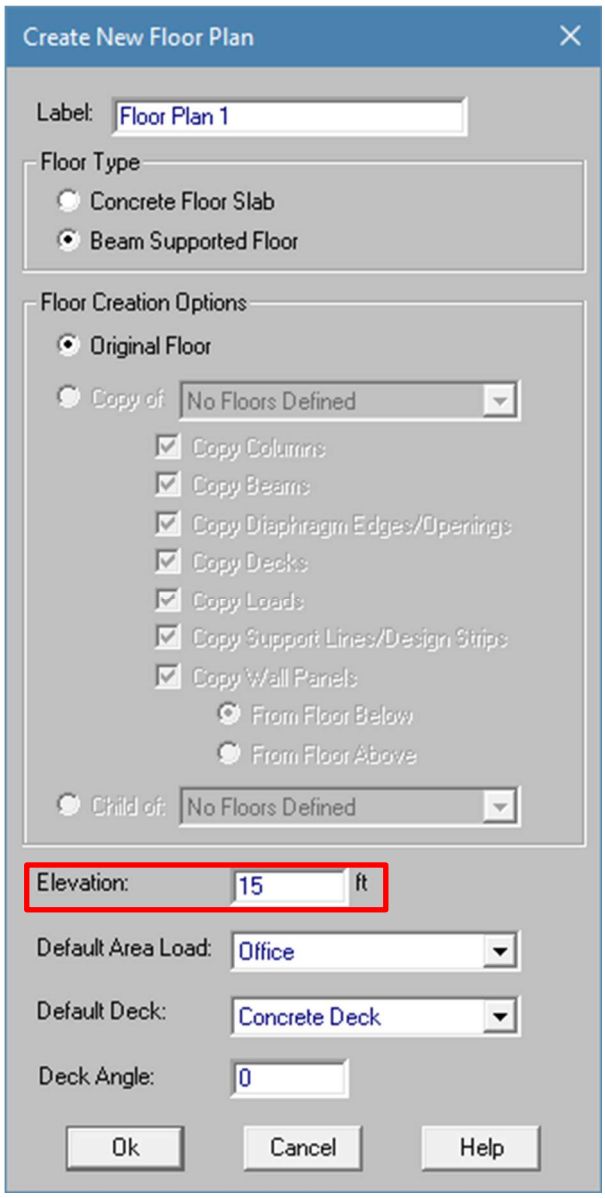
To create a new model, begin by creating a floor plan, or click **Close** and work on your own.

- Click Create a New Floor Plan  to begin.

Since this is the first floor in the model, the only option is **Original Floor**. After creating one or more floors, you may then use them as the starting point for additional floors, either by creating a **Copy** or a **Child** of an existing floor.

The **Create New Floor Plan** dialog box also allows you to set the **Elevation**, the **Default Area Load, and** the **Default Deck** for the floor. These settings are discussed in more detail on page 58.

- In the **Elevation** box, enter **15** ft.



- Click **Ok**.

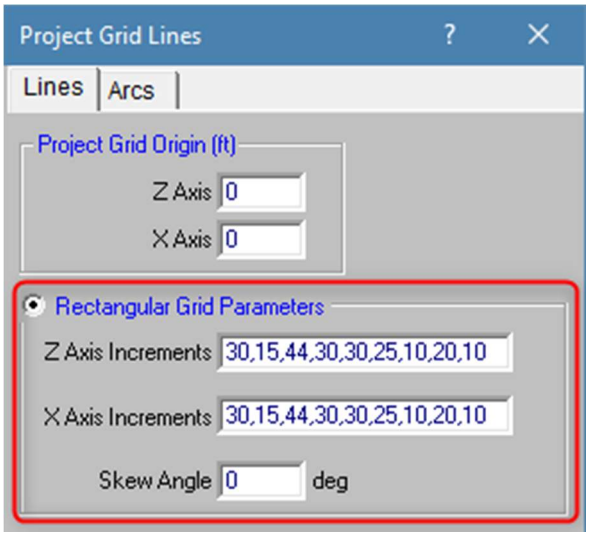
A new, blank floor will display, along with an empty **Project Grid** spreadsheet.

Project Grid

You can generate grids or type them in one at a time. To create grids:

- Under the Rectangular Grid Parameters in the Z Axis Increments box, type 30,15,44,30,30,25,10,20,10.

Since the grid spacing is the same for the **Z Axis** and the **X Axis** grid, copy and paste the **Z Axis Increments** into the box for the **X Axis Increments.**



Next, let's look at the labeling for the grids.

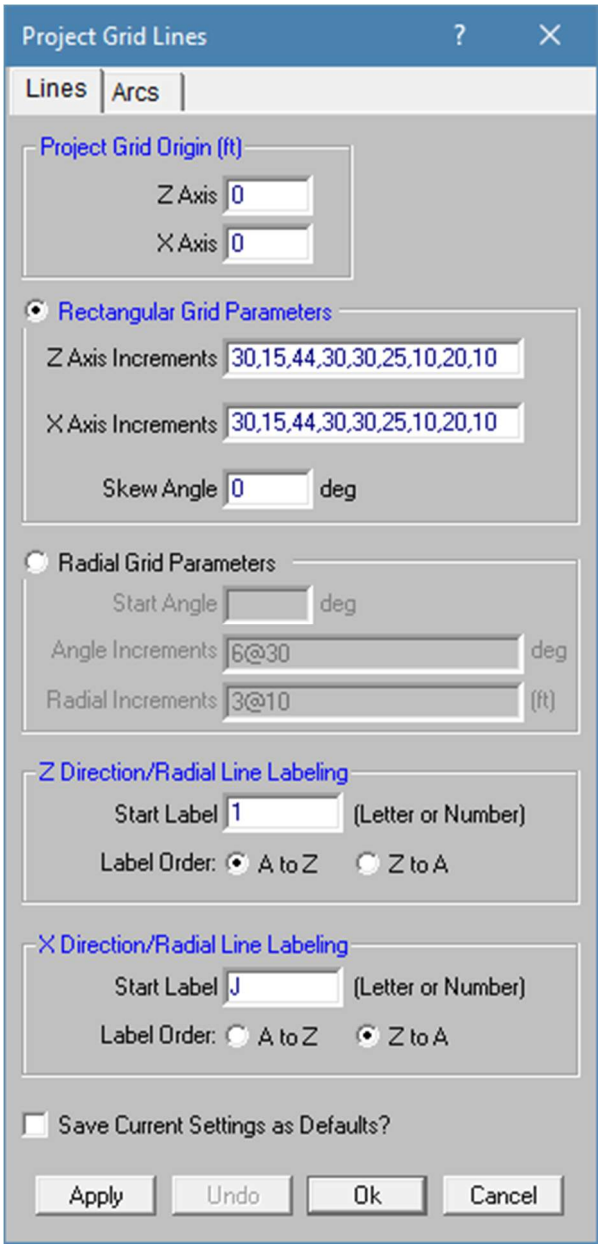
For Z Direction/Radial Line Labeling:

- 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 **J**.
  For the **Label Order**, click **Z to A**.

The **Project Grid Lines** dialog box should look similar to below:



- Click **Apply**. Then click **OK** to close the dialog box.

You can verify the values in the **Project Grid Lines** spreadsheet and modify the grid lines if needed.

- When complete, your **Project Grid Lines** spreadsheet should look similar to the image below:

Project Grid Lines

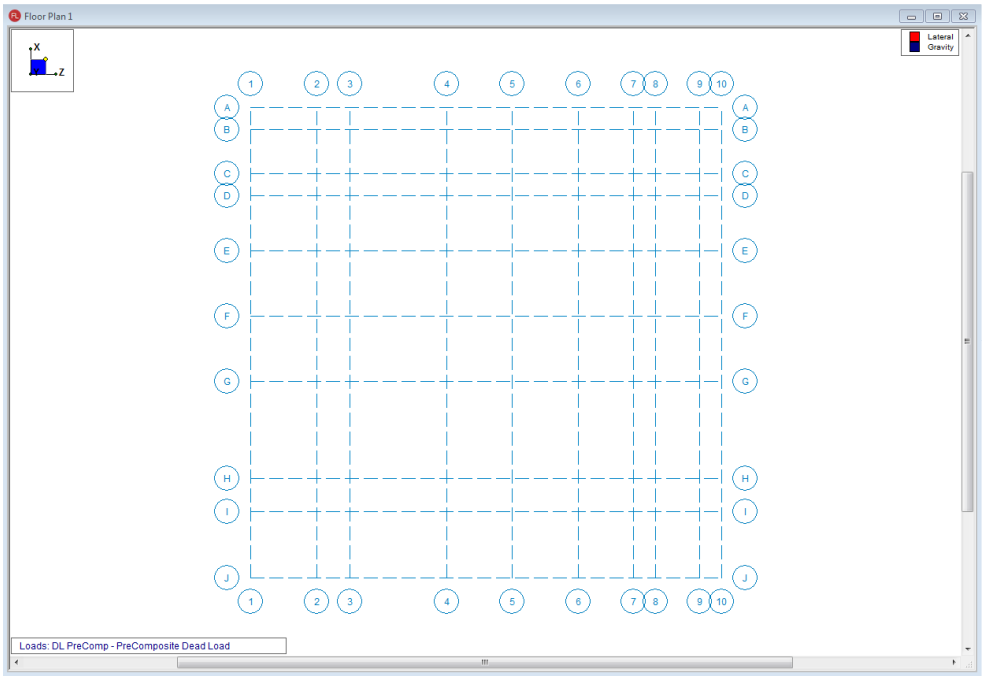
Lines | Arcs

Generate Project Grid Lines

	Label	Start Z [ft]	End Z [ft]	Start X [ft]	End X [ft]	Start Bubble	End Bubble
1	1	0	0	0	214	☑	☑
2	2	30	30	0	214	☑	☑
3	3	45	45	0	214	☑	☑
4	4	89	89	0	214	☑	☑
5	5	119	119	0	214	☑	☑
6	6	149	149	0	214	☑	☑
7	7	174	174	0	214	☑	☑
8	8	184	184	0	214	☑	☑
9	9	204	204	0	214	☑	☑
10	10	214	214	0	214	☑	☑
11	J	0	214	0	0	☑	☑
12	I	0	214	30	30	☑	☑
13	H	0	214	45	45	☑	☑
14	G	0	214	89	89	☑	☑
15	F	0	214	119	119	☑	☑
16	E	0	214	149	149	☑	☑
17	D	0	214	174	174	☑	☑
18	C	0	214	184	184	☑	☑
19	B	0	214	204	204	☑	☑
20	A	0	214	214	214	☑	☑

Note: Pasting is not available in the Demonstration version, so you must re-type these values if you are using the Demonstration version.

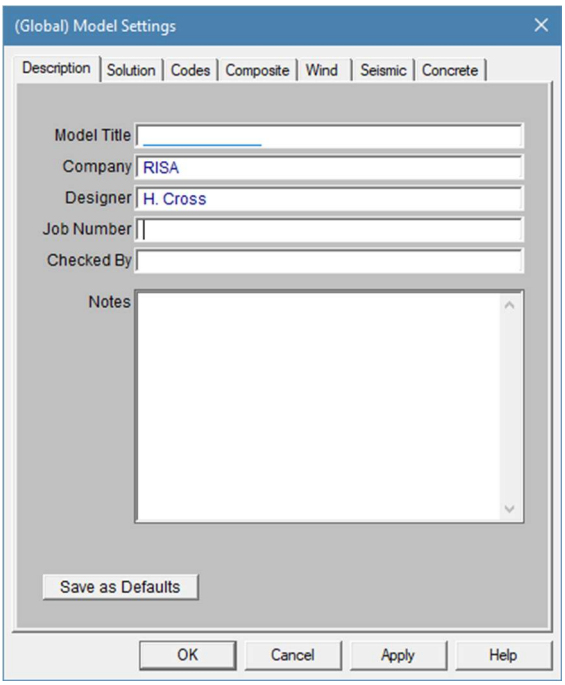
- Click **Redraw**  (on the Window toolbar) to get a better view. Your workspace should like this:



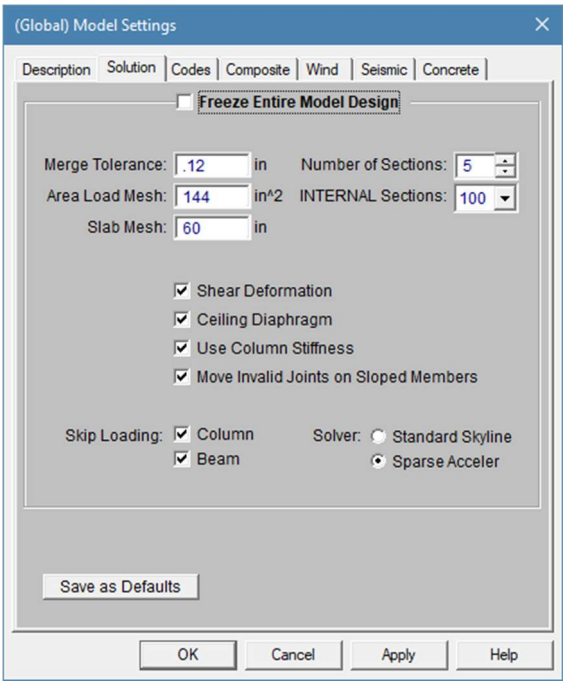
Model Settings

Model Settings are settings that apply to the entire model. To view these settings:

- On the RISA toolbar, click the **Model Settings** button  .
- On the **Description** tab, type in your **Company** and **Designer** name.

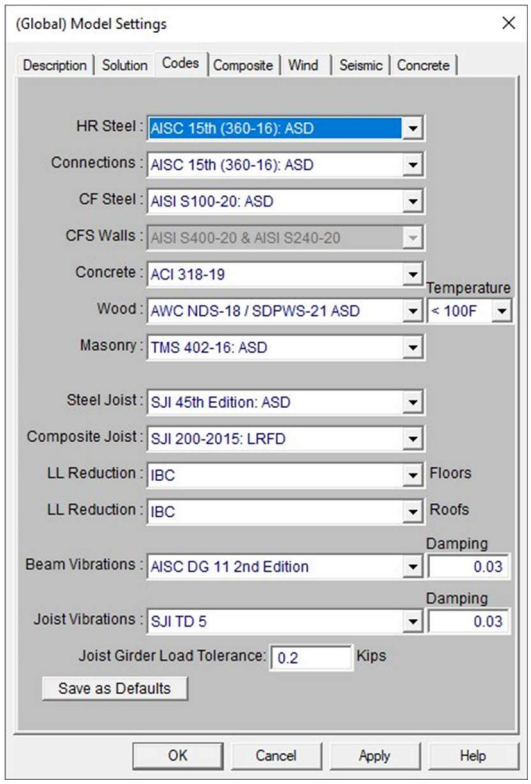


- Click on the **Solution** tab.



The entries under this tab are used to control settings that affect the general solution of the full model. For more information on any of these entries, click on the **Help** button to open the Help File.

- Click on the **Codes** tab. Make sure the inputs match the image below:



The entries under this tab present the available design codes to control the design of each material, live load reduction, and vibrations. For more information on any of these entries, click on the **Help** button to open the Help File.

- Click on the remaining tabs to review the settings for **Composite** design, **Wind** and **Seismic** loads, and **Concrete** design parameters.

When finished, close the **Model Settings** dialog box:

- Click **OK**.

Defining your Model

When building your model, you will define the elements in the order that they will be built. For example, supporting elements (columns & walls) must be in place first before you can define elements that use those supports (beams). You may enter model elements through the spreadsheets or with the drawing tools. The use of drawing tools will be the focus of the next few sections of this tutorial.

The Drawing toolbar is located at the top of the workspace and looks like this:

The toolbar buttons are arranged in the order you would use them to define your model (left to right).

- To turn the Drawing toolbar on or off, go to the Window toolbar, and click the **Graphic Editing** button  .

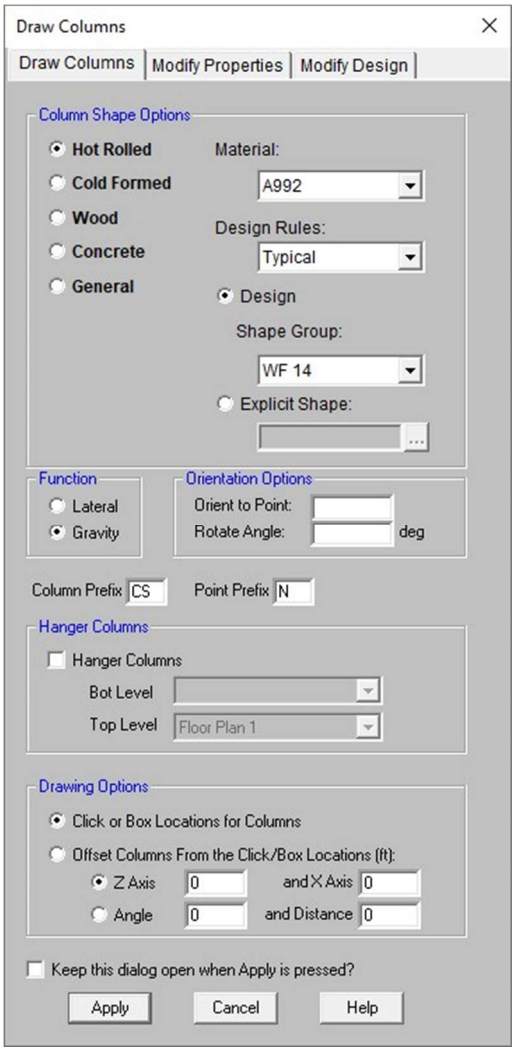
Columns

When creating columns, you may choose the material and shape of the column, as well as designate its orientation. You may orient the column towards a specific point, give it a specific angle, or combine the two. For this model, we will use several of these options.

- On the Drawing toolbar, click **Draw or Modify Columns** to view the **Draw Columns** dialog box.

Define the W14 gravity columns, as shown below.

- Under **Material**, select **A992**.
- Under **Shape Group**, select **WF14**.

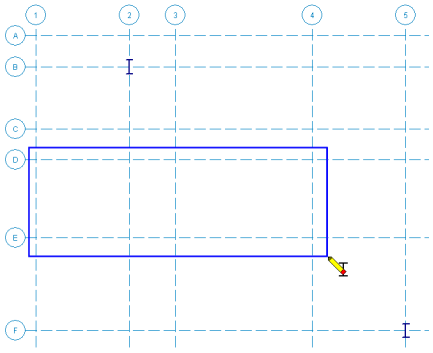


- Click **Apply** to start drawing the columns.

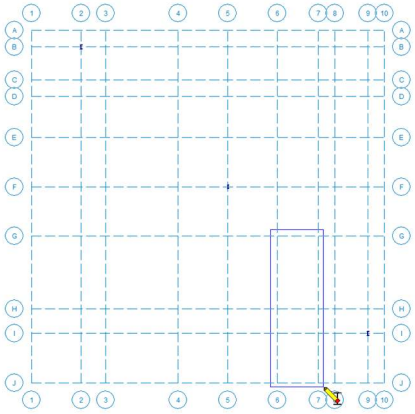
Note: When you enter the drawing mode, your cursor changes to .

You can now start defining the columns by clicking or drawing, as shown below:

- Enter columns by clicking on the **F5**, **B2**, and **I9** grid locations.
- Next, enter multiple columns by drawing a box around the grid intersections between **D1** and **E4**, as shown in the diagram below.



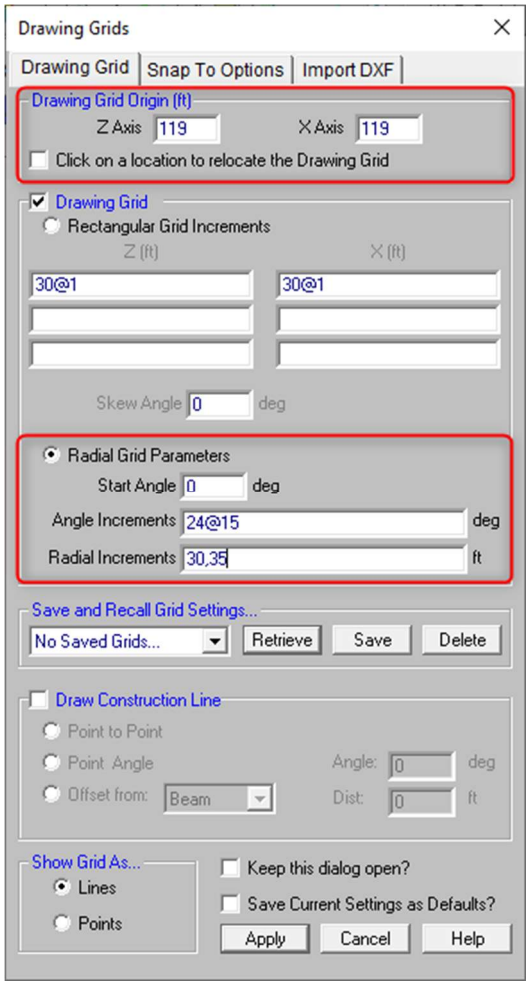
- Repeat for the grid intersections between **G6** and **J7**.



To further define the gravity columns:

- On the Drawing toolbar, click **Modify Drawing Grid**  to create a local drawing grid.
- Under **Drawing Grid Origin**, in the **Z Axis** and the **X Axis** boxes, type **119**.
- Click Radial Grid Parameters. In the Angle Increments box type 24@15, and in the Radial Increments box type 30,35.

The dialog box should look like this:

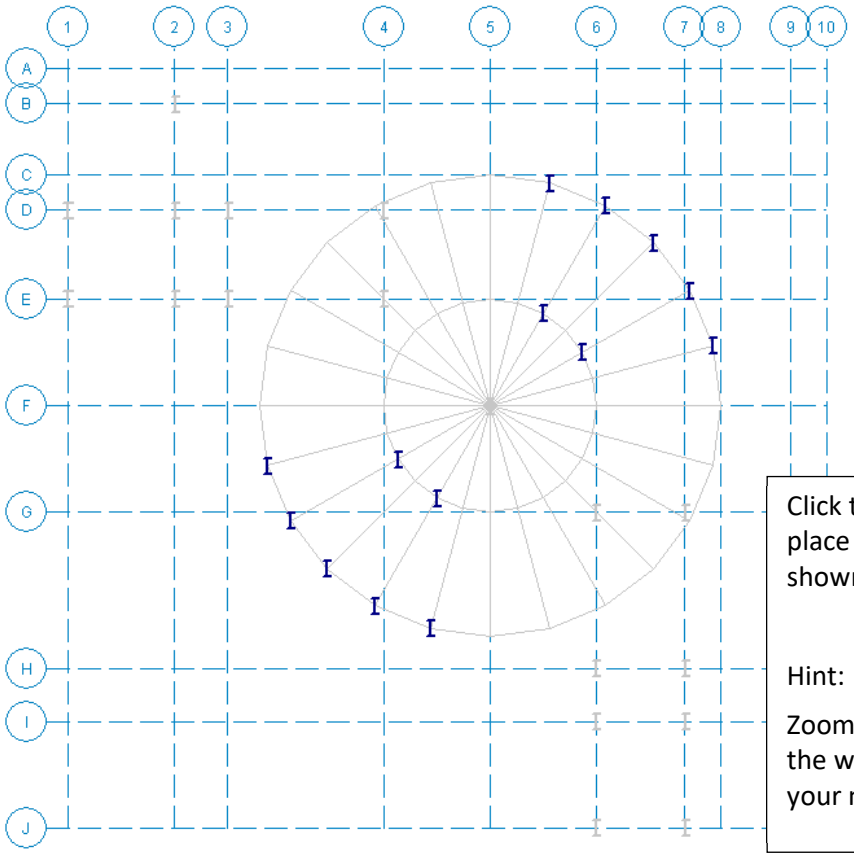


- Click **Apply** to close the dialog box.

Now use the newly created radial drawing grid to define the gravity columns.

- Create the gravity columns by clicking the radial grid points shown in the image below.

Note: If you have trouble clicking on the exact location, enlarge the model using the wheel button on your mouse to zoom in and out.



Click the grid points to place the columns shown.

Hint:

Zoom in and out using the wheel button on your mouse.

Next, add the lateral columns:

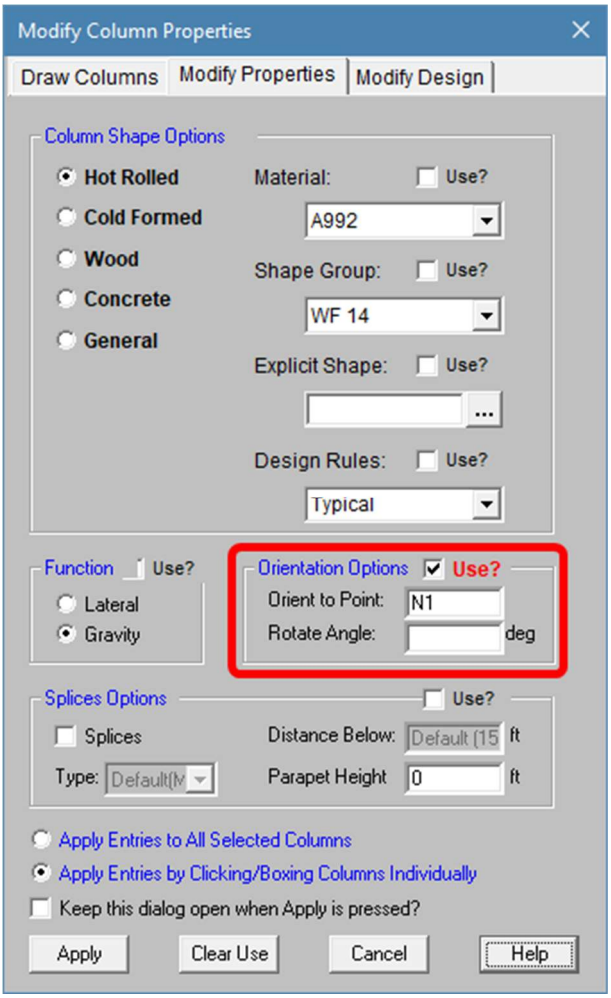
- On the Drawing toolbar, click **Draw or Modify Columns** .
- Under **Function**, click **Lateral**, and then click **Apply**.
- Add lateral columns at **C5**, **E5**, **F1**, **F2**, **F6**, **F8**, **I5**, and **J5**.

Note: Lateral columns are displayed in red; gravity columns are displayed in blue.

- When you finish adding these columns, use the right mouse button (right-click) or ESC to terminate the column drawing tool.

Notice that by default all the columns are oriented with their webs parallel to the vertical axis. To change the orientation of the columns toward the center column:

- On the Window toolbar, click the **Joint Labels** button  two times to turn on the labels of the point locations.
- Make note of the point label at the center of the radial grid. That point label will be used below to orient the columns in your radial grid.
- On the Drawing toolbar, click **Draw or Modify Columns**  again and select the **Modify Properties** tab.
- In the **Orientation Options** area, select the **Use?** check box (once it is selected, it will display in red). In the **Orient to Point** box, enter the point label for the center of your grid (this is the point label you made note of above).

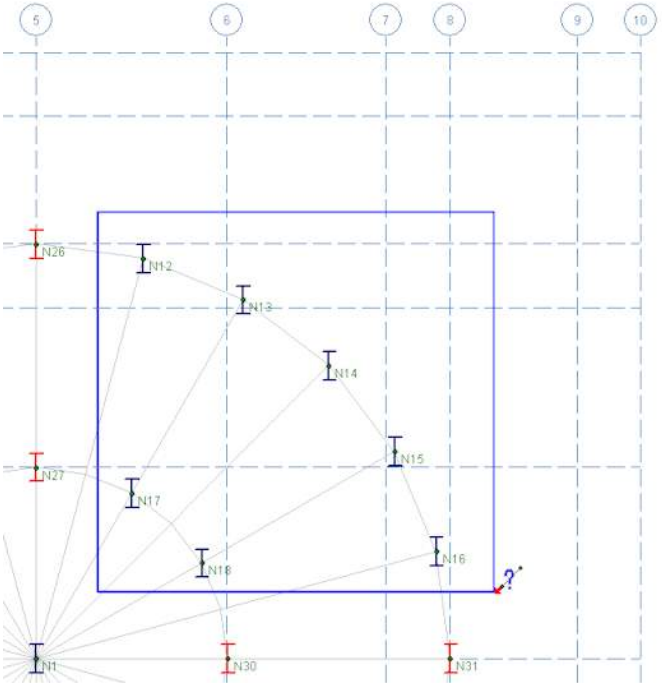


- Click Apply Entries by Clicking/Boxing Columns Individually, and then click Apply.

Note: The cursor will change to  .

Now, you can select the columns as follows:

- Click on or draw a box around the columns in the upper right portion of the radial grid, as shown in the figure below. Note that the orientation of these columns changes toward the center column.



- Continue clicking or drawing boxes around the remaining columns in the lower lefthand portions of the radial grid, so that all columns should now be oriented toward the center column.

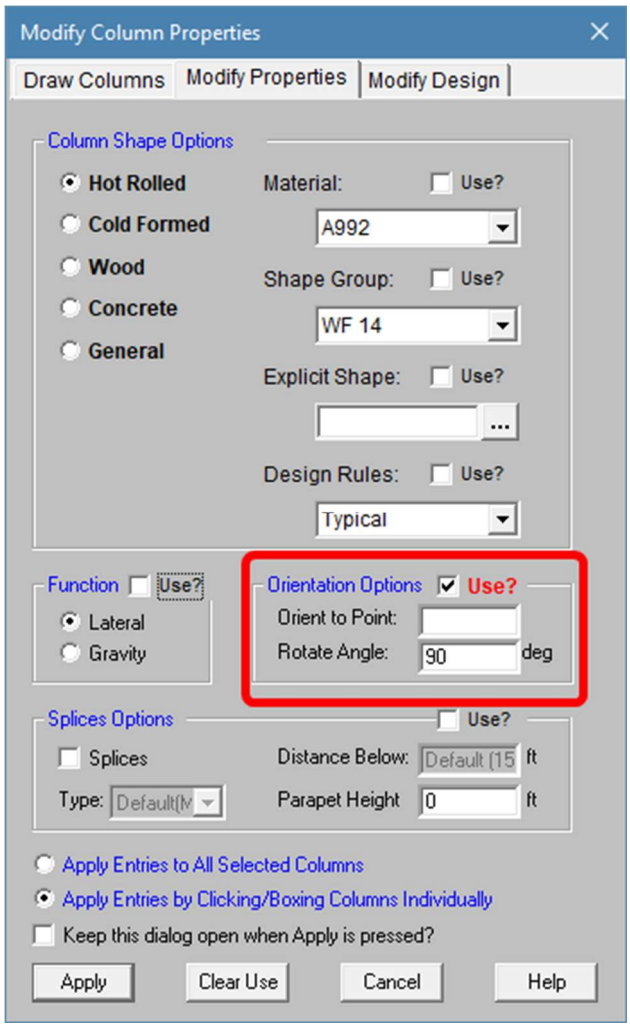
Note: Use care not to select the center column, as it already has the correct orientation.

When you are finished with the radial columns, you can now turn the labels and the drawing grid off.

- On the Window toolbar, click the **Joint Labels** button to turn off the labels.
- On the Drawing toolbar, click the **Drawing Grid** button to turn off the **Drawing Grid**.

Next, modify the column orientation for the columns that need to be rotated 90 degrees:

- On the Drawing toolbar, click **Draw or Modify Columns** again. Ensure that the **Modify Properties** tab is still selected.
- In the **Orientation Options** area, click the **Use?** check box (once it is selected, it will display in red). In the **Orient to Point** box, clear the point label and leave it blank. In the **Rotate Angle** box, enter **90**.

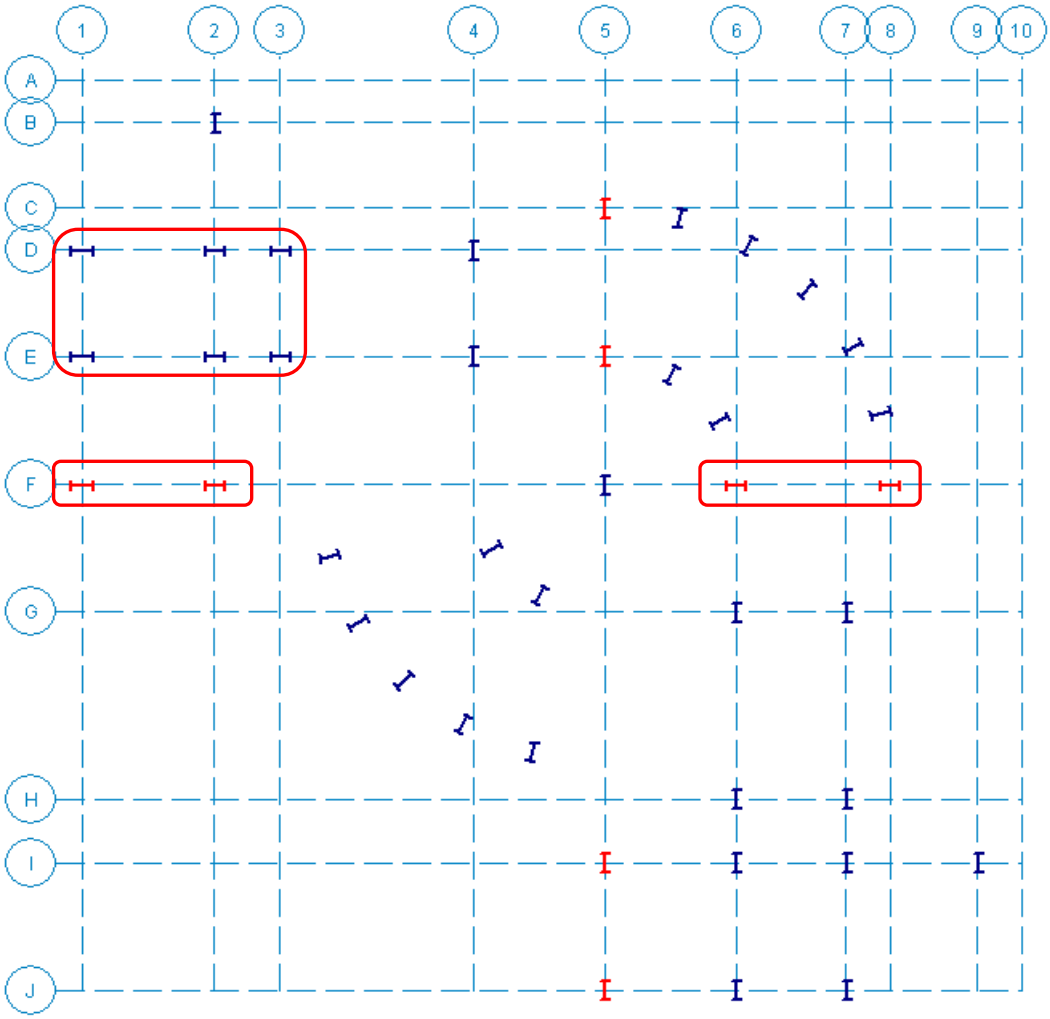


- Click **Apply**.

Note: The cursor will once again change to .

- Click on or draw a box around the columns circled in the next image.

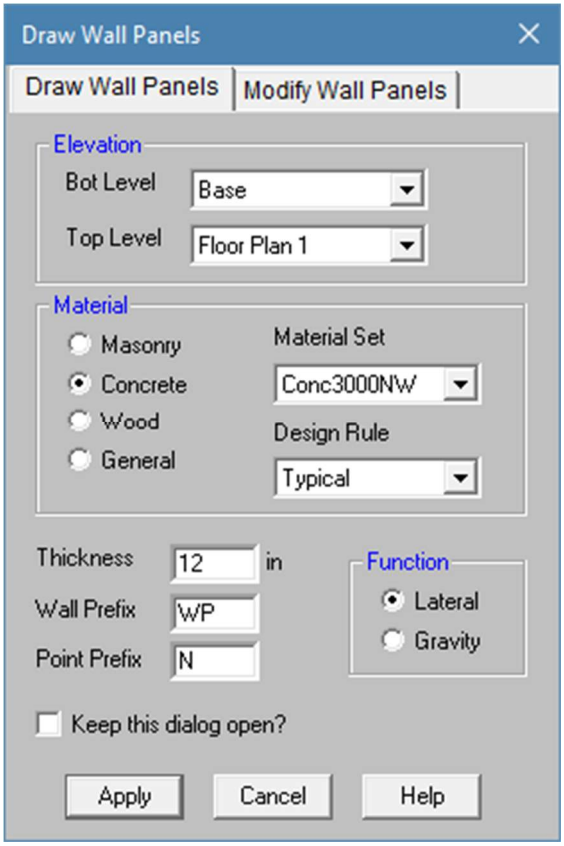
When you are done, your columns should have rotated 90 degrees, and oriented as shown below.



Wall Panels

Prepare to draw your wall panels:

- On the Drawing toolbar, click **Draw or Modify Walls** .
- Under **Material**, select **Concrete**.
- Under **Function**, click **Lateral**. The wall panel settings should match those in the image below.



- Click **Apply**.

Note: Your cursor will change to .

Start drawing your wall panels, as follows:

- Draw in a wall panel between **A3** and **A5**. To do this, click on **A3**, move the mouse to **A5** and click again.

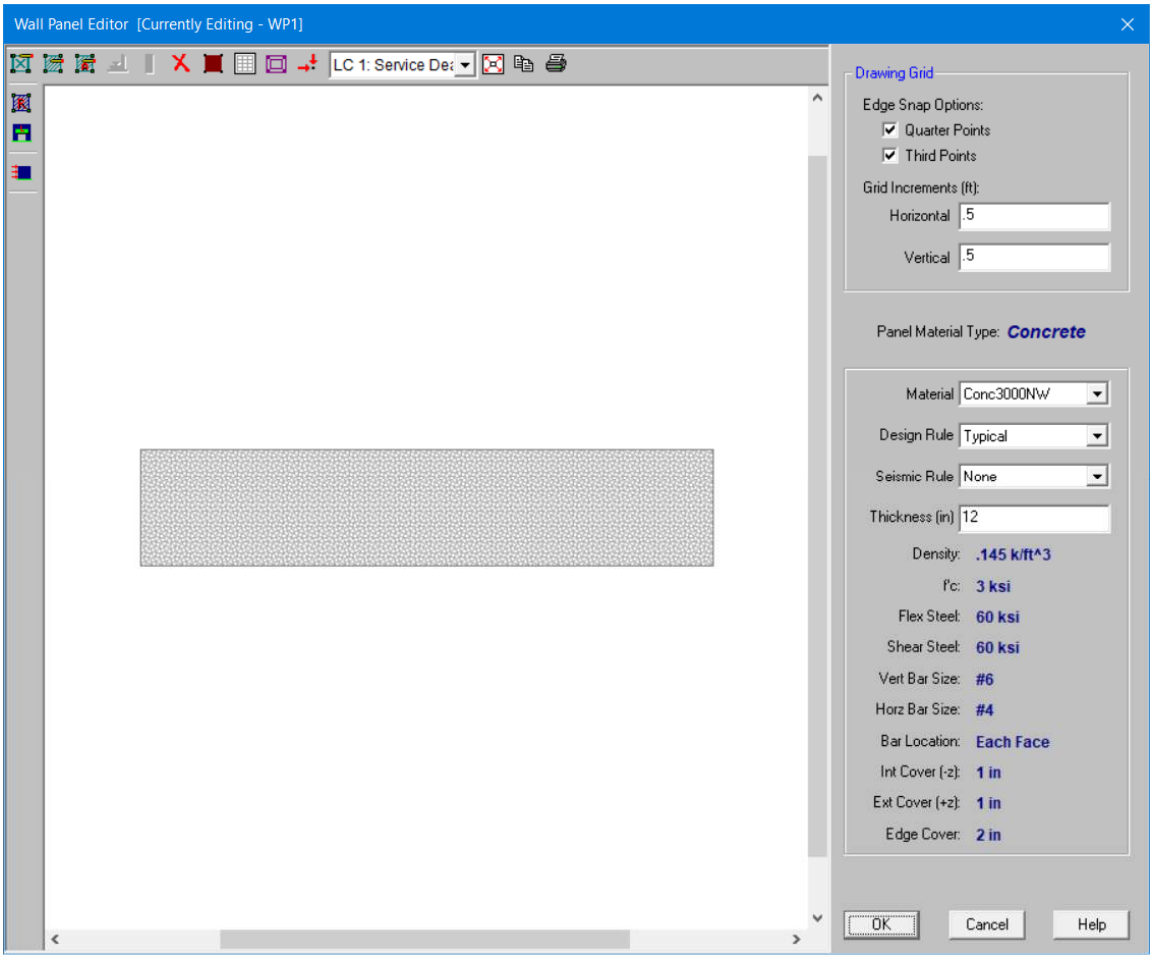
Note: After you draw a wall, your cursor is "anchored" to the end point of the wall. To release the drawing tool, use your right mouse button (right-click) or ESC.

- Draw the remaining wall panels between **F3** and **F4**, **G5** and **H5**, and **F10** and **H10**.
- When you finish adding the walls, use the right mouse button (right-click) or ESC to terminate the wall panel drawing tool.

If you make a mistake or if you want to delete a wall, click CTRL+Z to undo, or delete the wall as follows: on the **Modify** menu, click **Delete**, then click **Delete Based on This Criteria** and check the **Delete Selected Walls** check box.

Next, you will add an opening to the wall panel on gridline A.

- Double-click on the wall panel between **A3** and **A5** to open the **Wall Panel Editor**.



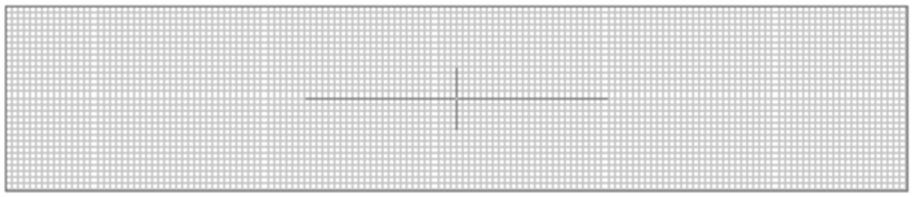
This **Editor** allows you to add openings and design regions to the wall panels. You also can edit the **Material**, **Design Rule**, and **Thickness** in this view.

Wall Panel Openings

You will add in an opening next. Before you do this, turn off the rendered view of the wall.

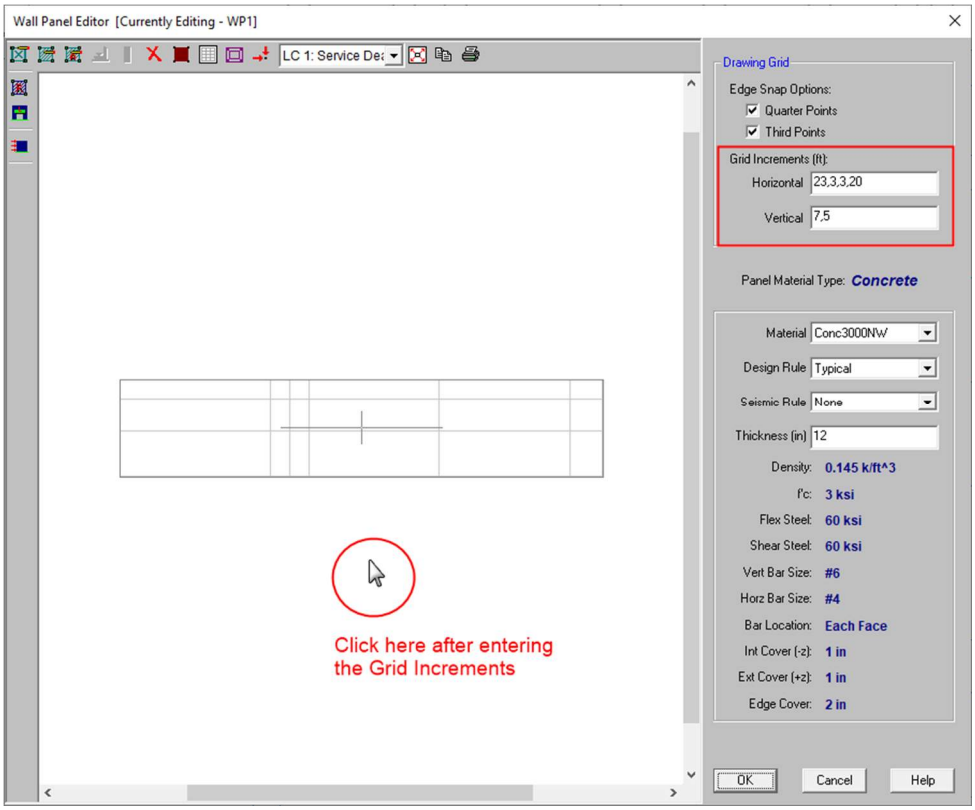
- Click the **Toggle Rendered View** button  at the top center of the **Editor** Drawing toolbar.

Now only the drawing grid is visible in the plane of the wall.



Next, set up a **Drawing Grid** to help define the opening.

- Under **Horizontal Grid** Increments, enter **23,3,3,20**.
- Under **Vertical Grid** Increments, enter **7,5**.
- Click into the **Editor** model space to update the grid.



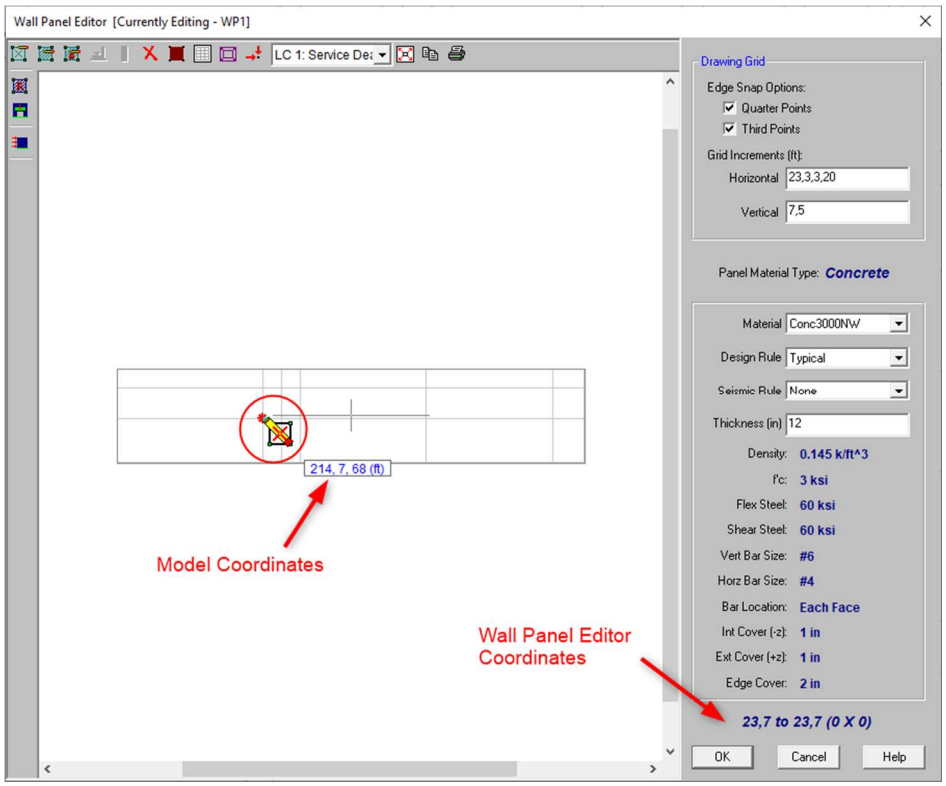
Now draw in the opening.

- Click on the **Create New Openings** button  (in the upper left corner of the Editor).

Note: Your cursor will change to  .

Use the coordinate information in the lower right Wall Panel Editor to help select the door opening coordinates.

- First click on the top left of the door opening (**23,7**). The coordinates in the **Wall Panel Editor** are based on the drawing grid in the **Wall Panel Editor** and do not match the Model Coordinates or Status Bar.



Note: Any existing region will need to be deleted before drawing the opening.

- Next click the lower right corner of the door opening (**26,0**).

When complete, the door opening will show with the X line and lintel label.

Continue to draw in the large door opening.

- Click on (**29,12**) and then on (**49,0**).
- Right-click or click ESC to cancel the opening drawing tool.

Wall Panel Design Regions

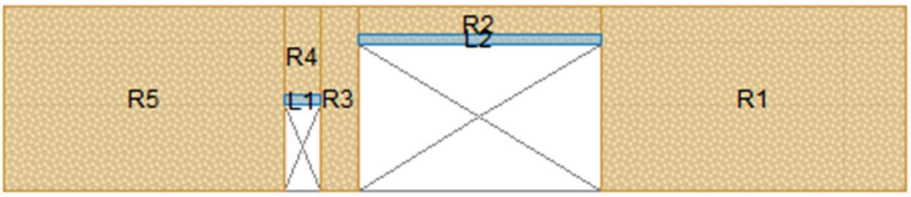
When finished, you can add in the design regions around the openings. The **Wall Panel Editor** allows you to create design regions in two ways: manually draw in each region, or use the automated generation tool. You will use the automated generation for this model.

- Click on the **Generate Wall Regions Automatically** button  (in the upper left corner of the Editor).

Turn the rendered view back on for a better view.

- Click the **Toggle Rendered View** button  at the top center of the **Editor** Drawing toolbar.

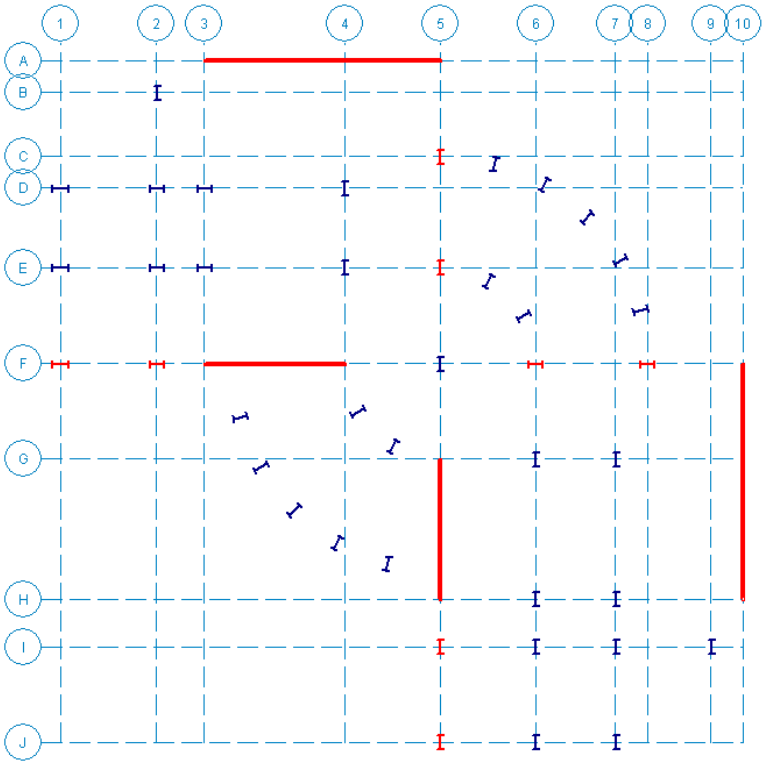
Now you can see the two openings, and five regions that were generated around the openings.



For more details on how to model wall panels, including information on masonry design and wood design, refer to https://risa.com/documentation.

- Click **OK** to close the **Wall Panel Editor**.

Your model should look similar to the following image:



This is the end of Tutorial A1.

You may save your model to be used as the starting point for the next tutorial, or begin the next tutorial using the .rfl starter file in the RISAFloor **Tutorials** folder. To save the model:

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

Part A: Tutorial 2 – Modeling Beams

This tutorial will guide you through drawing individual beams and explore the different time-saving options for adding infill beams. This tutorial continues from where the previous tutorial ended, so follow these steps to get your model up and running:

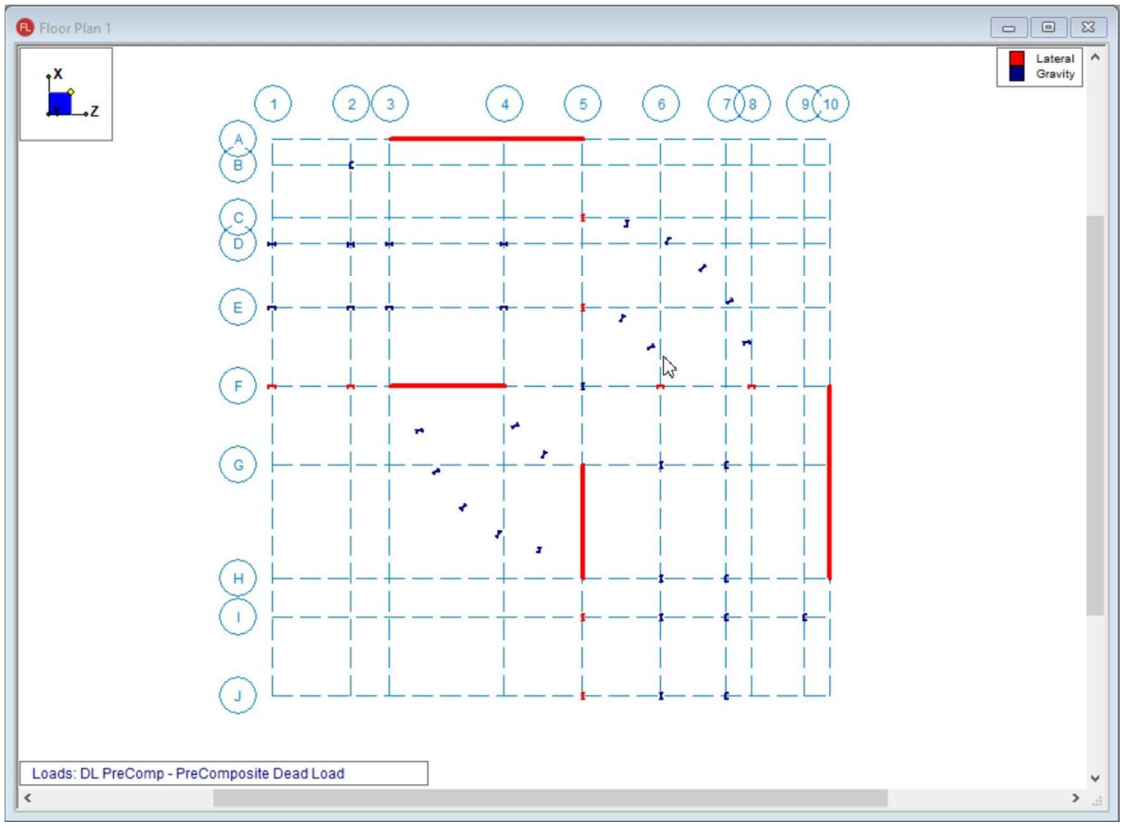
If you are continuing from the previous tutorial:

- From the **Window** menu, select **Single View**.
- On the Window toolbar, click the **Graphic Editing** button to activate the Drawing toolbar.
- On the **Data Entry** toolbar, click **Close** to close it. Skip ahead to the next page.

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

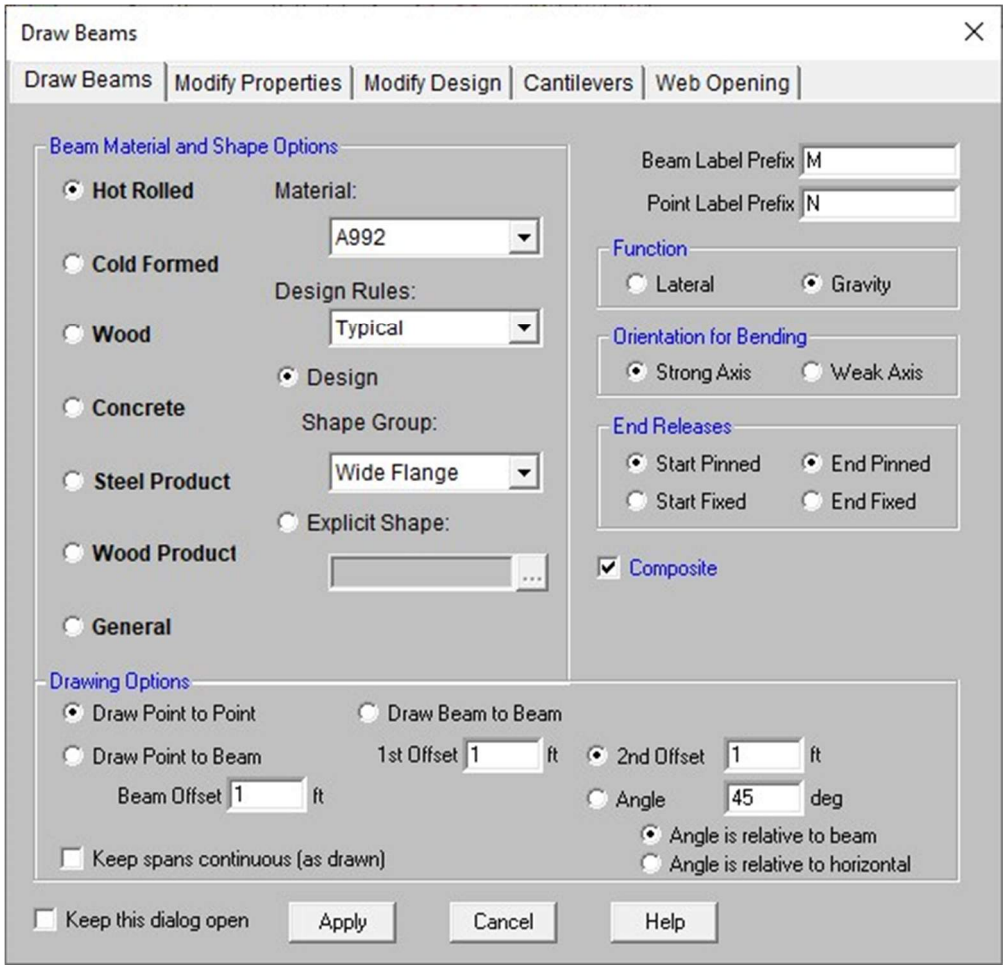
- Double-click on the **RISAFloor** icon to start the program.
- Click **Open Model**. Double-click the **Tutorials** folder, select **Tutorial A2 Starter.rfl** and click **Open**. Click **Close** (or **Cancel**) to exit the **Model Settings** dialog box.
- On the Window toolbar, click the **Graphic Editing** button to activate the Drawing toolbar.

Your screen should now look like this:



Primary Framing

- On the Drawing toolbar, click **Draw Beams** to open the **Draw Beams** dialog box.

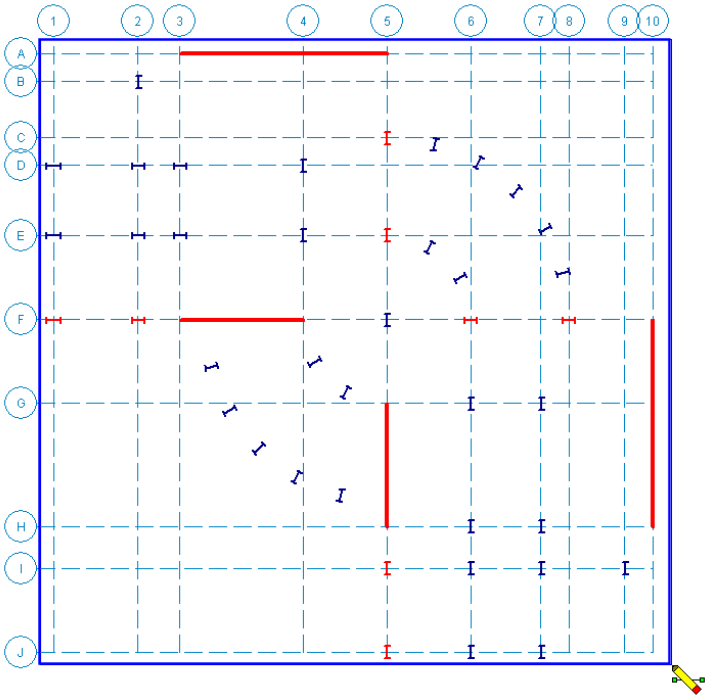


In this dialog box, materials and shapes may be chosen along with orientation options. You may also assign the beams to a set of design rules. You may draw single span members or continuous beams. For this model, we will select a wide flange, hot rolled steel member drawn point to point.

- Under **Material**, select **A992.**
- Under **Shape Group**, select **Wide Flange**.
- Under **Design Rules**, select **Typical**.
- Click **Apply**.

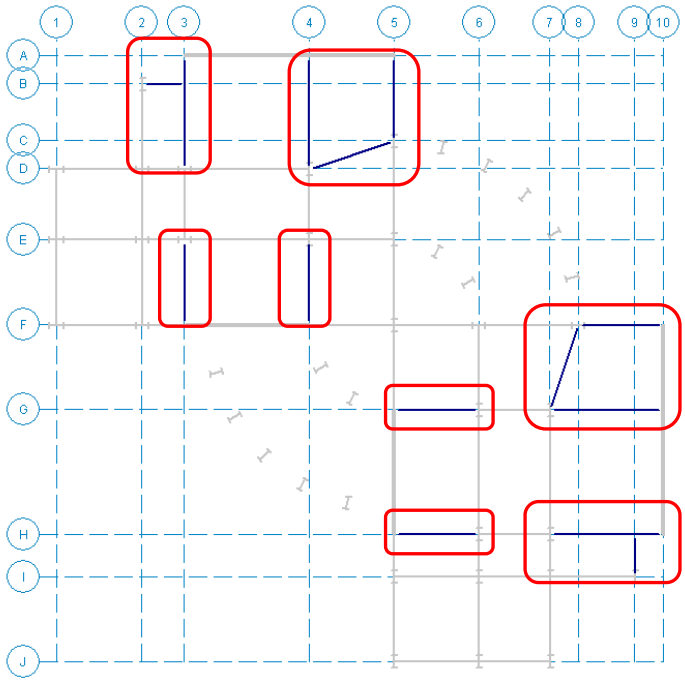
Note: The cursor will change to to signify that you are now in drawing mode.

- Draw a box around the entire model, as shown in the figure below:



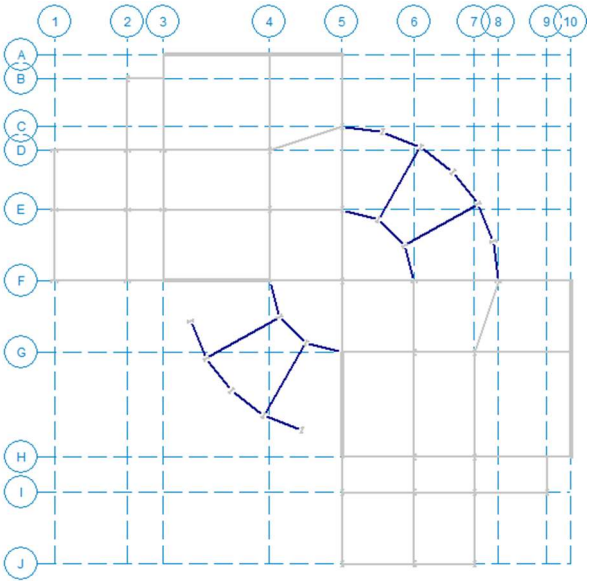
Notice that some beams were not automatically drawn in, such as those supported by wall panels or other beams. These members can be drawn individually, without going back to the dialog box.

- Draw in the 14 beams circled below by clicking from point to point. Start by clicking **A3**, then **D3**. Use the right mouse button (right-click) or press ESC to release the mouse between beams. Then click **B2** and **B3**. Continue clicking to draw the remaining beams.



The beams in the radial portion of the structure may also be drawn from point to point.

- Continue clicking from point to point to draw in the beams shown below.

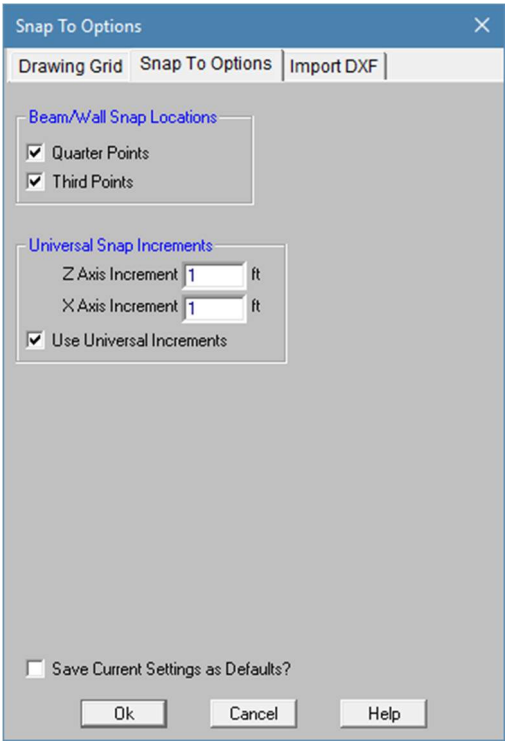


Hints:

Use the wheel button on your mouse to zoom in and out.

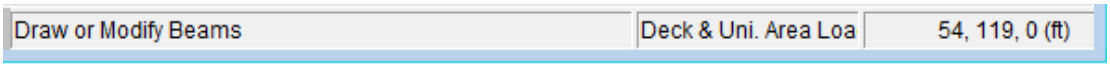
If you terminate the drawing beams tool before you are finished, you may recall it by clicking CTRL+D.

- On the Drawing toolbar, click **Modify Drawing Grid**  .
- Click the **Snap To Options** tab.
- Under **Universal Snap Increments**, make sure the **Z Axis** and **X Axis Increments** are both set to **1 ft**. Check the **Use Universal Increments** check box (this will allow you to draw freehand without being limited to a project or drawing grid).

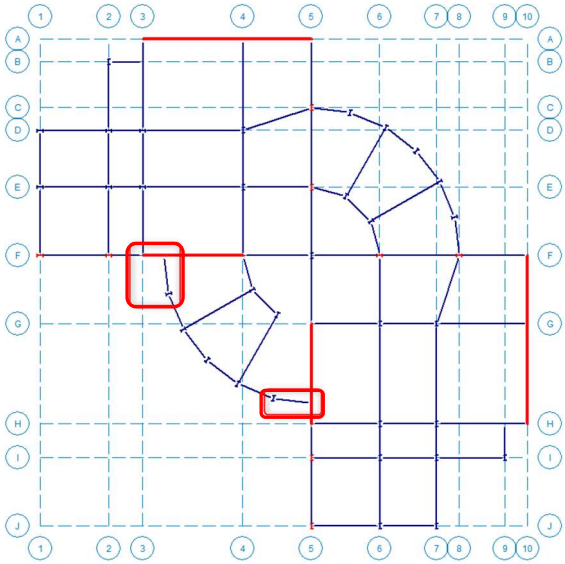


- Click **Ok** to return to the model view.

- Now, draw in two beams beginning at (**54, 119**) and (**119, 54**), as shown circled in the image below. To help you locate the coordinates, use the Status bar at the bottom of your screen (on the right).

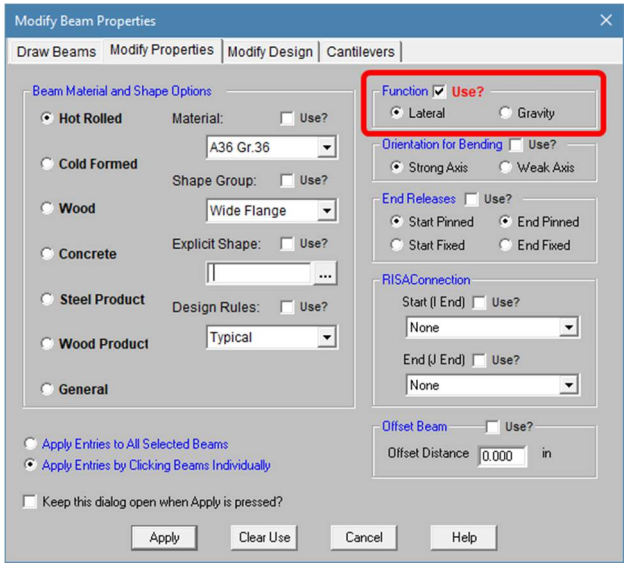


Continue drawing the beams until your model looks like this:



Next, change the function of the beams that span between the lateral columns to make them lateral elements.

- On the Drawing toolbar, click **Draw Beams** and select the **Modify Properties** tab.
- In the **Function** area, select the **Use?** checkbox (once it is selected, it will display in red). Under **Function**, click **Lateral**.

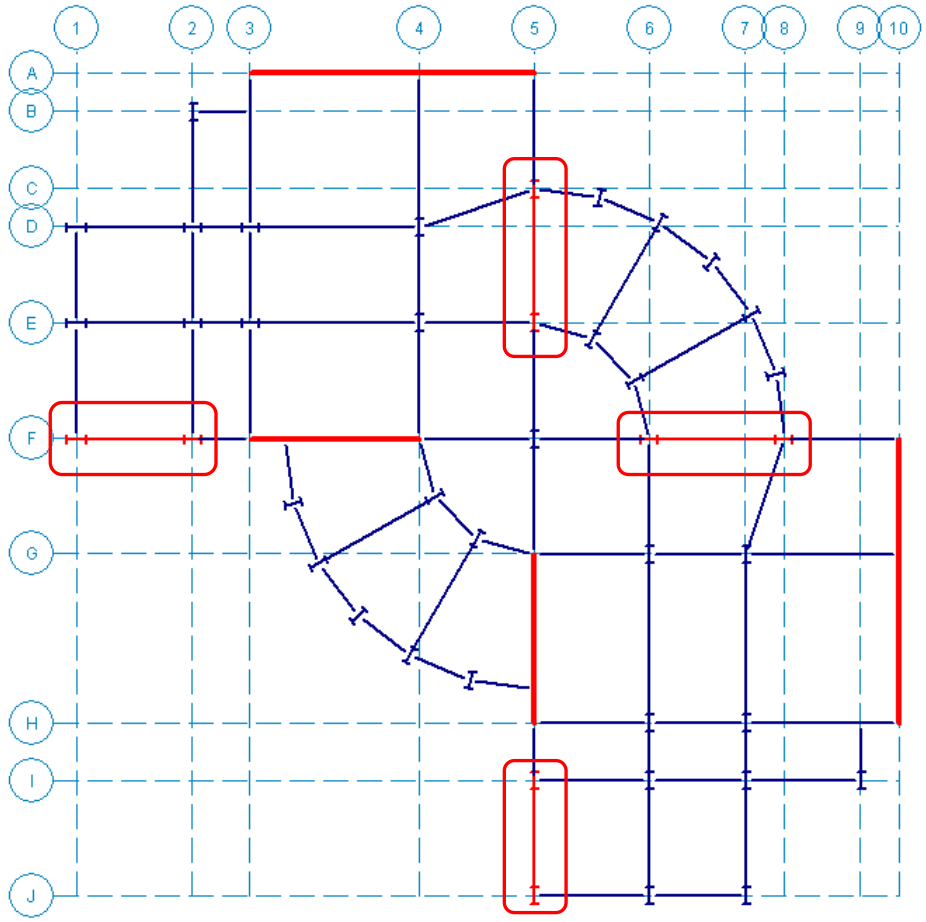


- Click Apply Entries by Clicking Beams Individually, and then click Apply.

- Now, click on the four beams that are shown circled in the image below (you can click anywhere on the beams).

Note: The color for these beams will change from blue to red because you are changing their function.

Your model should now look like this:



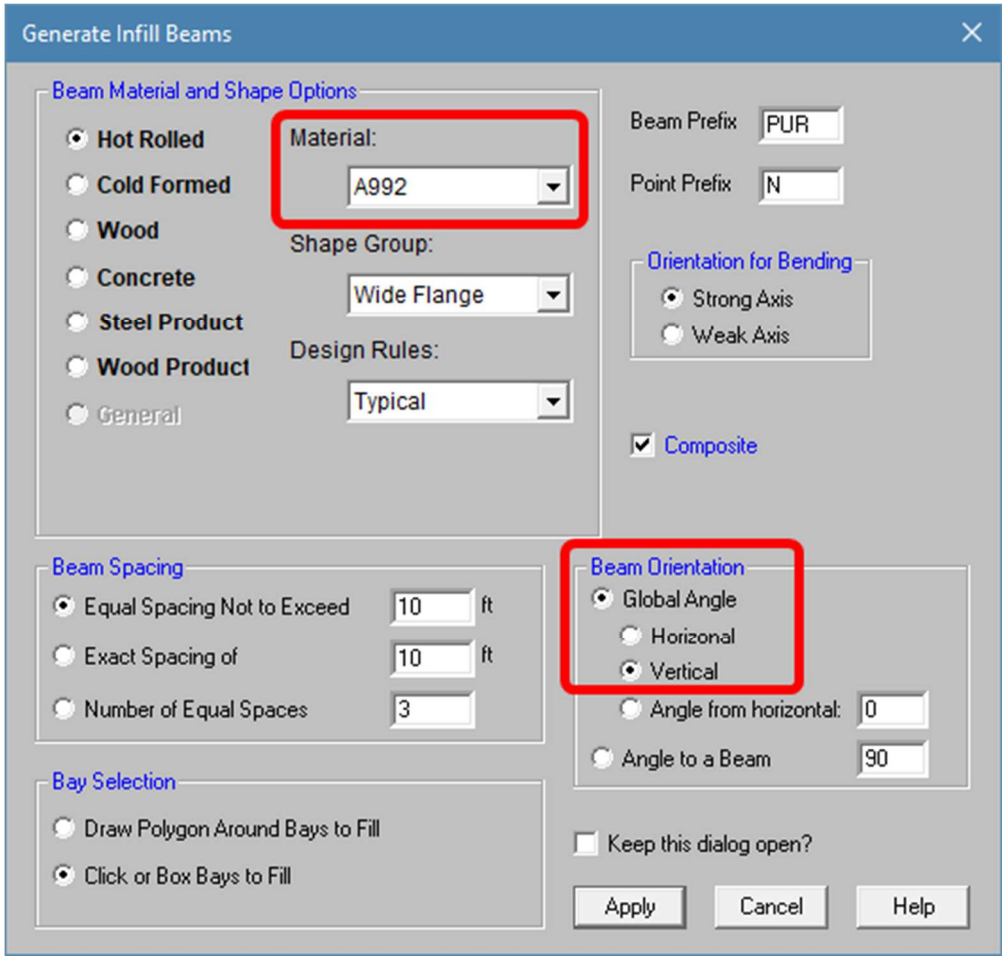
Lastly, turn off the **Universal Snap Points**.

- On the Drawing toolbar, click **Modify Drawing Grid**  .
- Click the **Snap To Options** tab.
- Uncheck the **Use Universal Increments** checkbox, then click **Ok** to go back to the model view.

Infill Framing

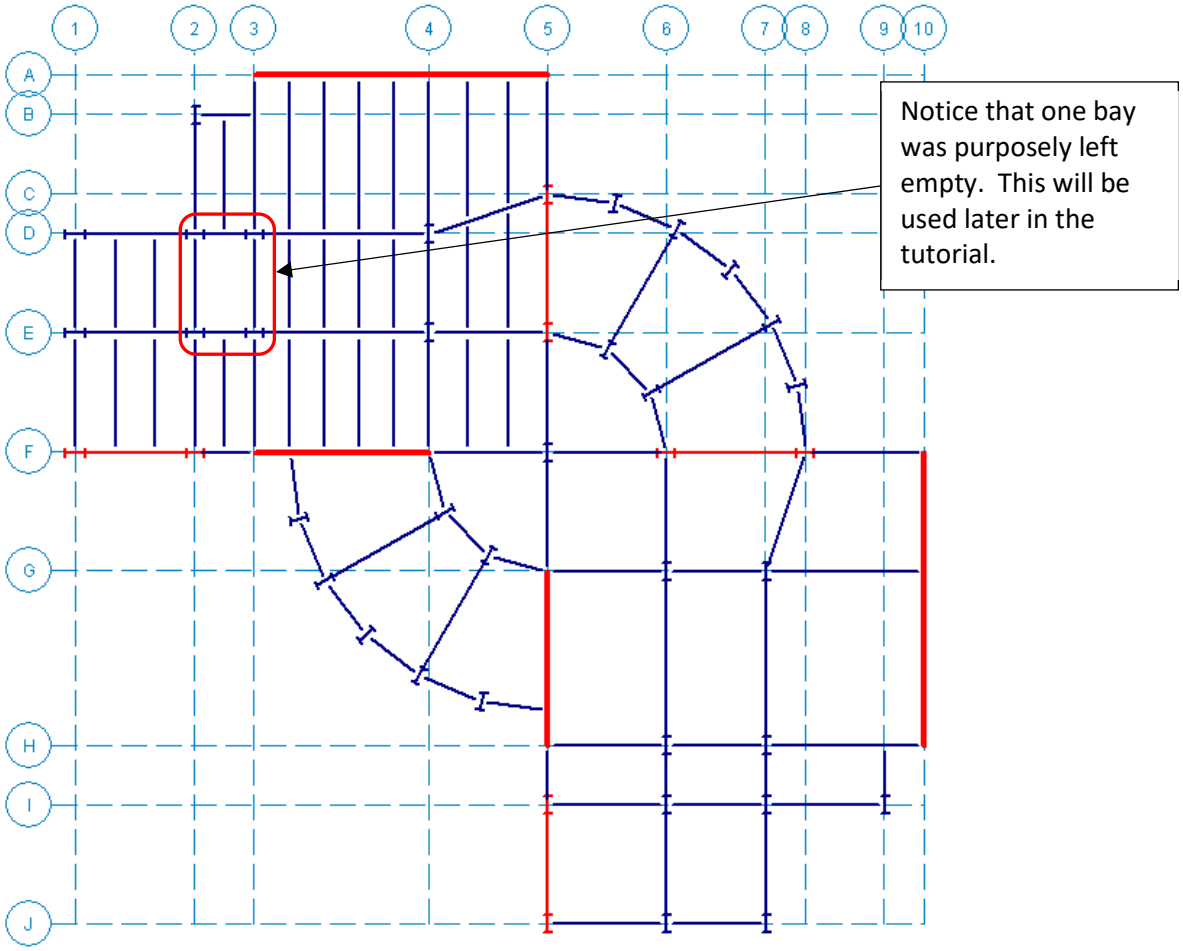
RISAFloor's Infill Framing tool assists your design process by quickly generating beams within existing bays. This tool will help you quickly create all of your secondary members:

- On the Drawing toolbar, click **Generate Beams**  .
- Under **Material**, select **A992**.
- Under Beam Orientation, click Vertical.



- Click **Apply**.

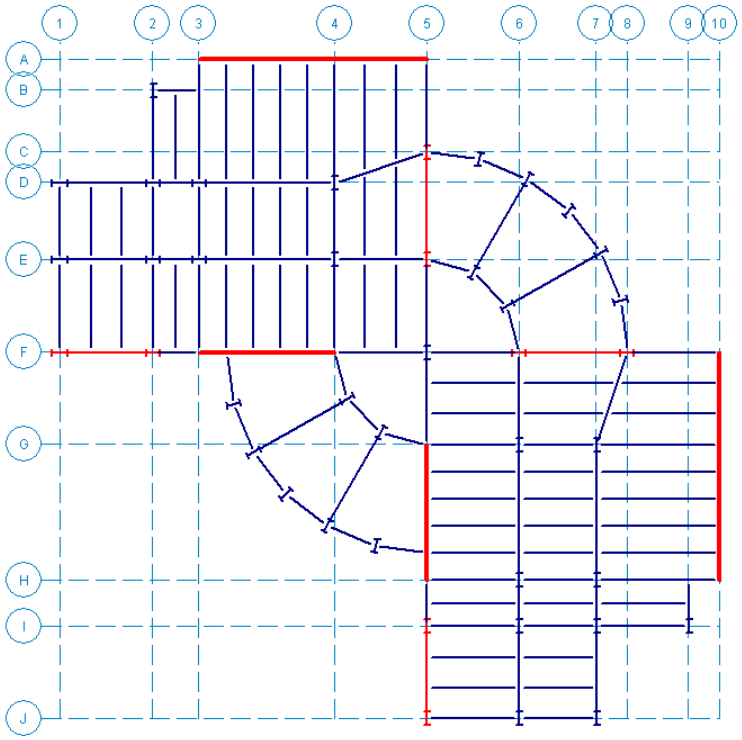
- Click once inside each open bay to fill in (as shown in the image below) until the upper left side of model looks like this:



If you zoom out to view a larger portion of your model, you can generate secondary beams in multiple bays at once.

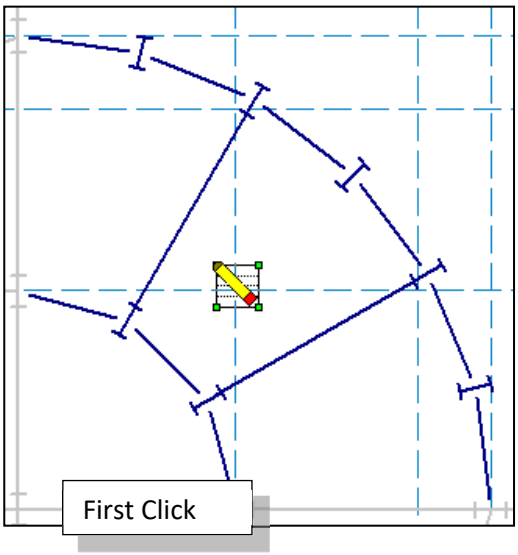
- On the Drawing toolbar, click **Generate Beams** again.
- Under **Beam Orientation**, click **Horizontal**, and then click **Apply**.
- Draw a box around the area between grid lines **F5** and **J10** to select the entire area.

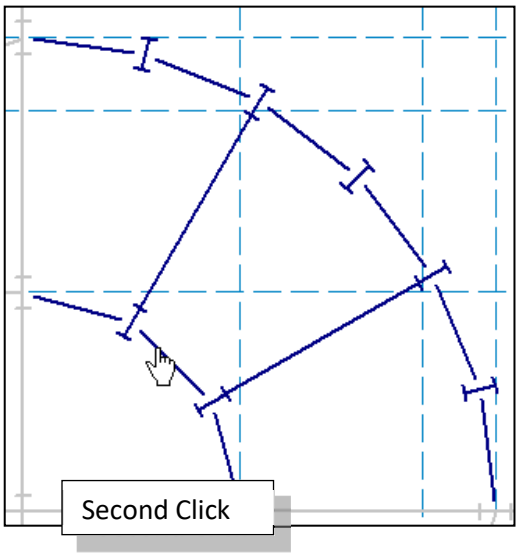
When finished, your model should look similar to the one shown below:



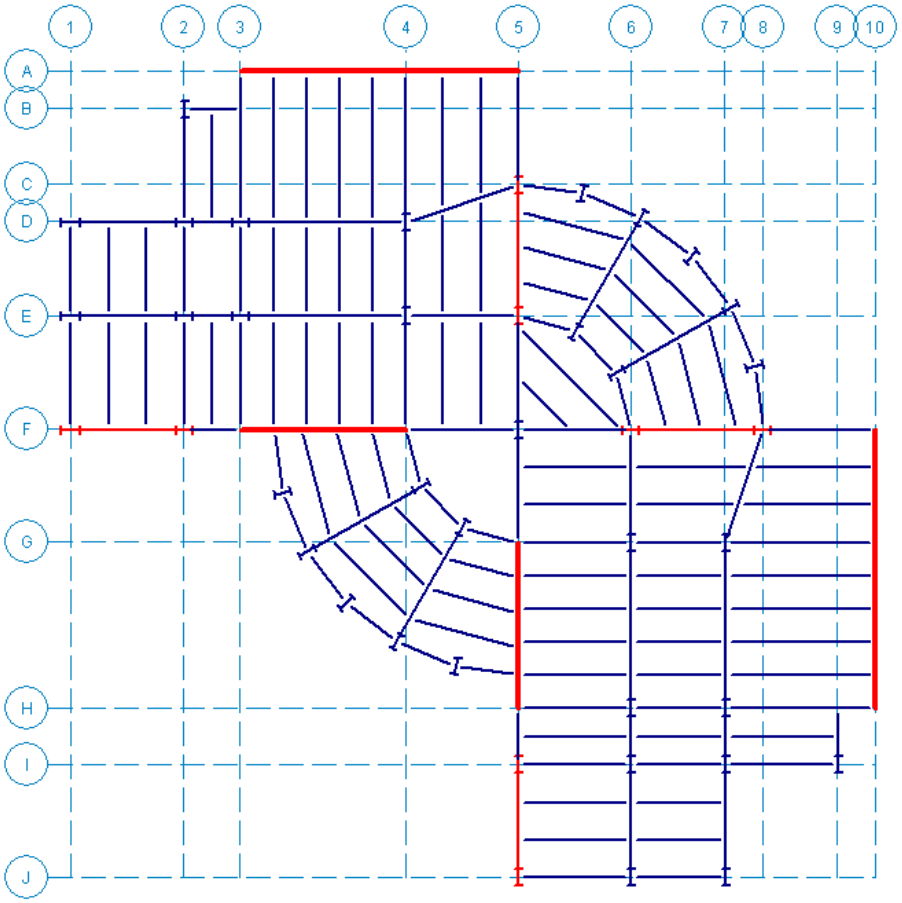
You can also use this tool to generate infill beams that are parallel to an adjacent member.

- On the Drawing toolbar, click **Generate Beams** .
- Under **Beam Orientation**, click **Angle to a Beam**, then enter an angle of **0**.
- Click **Apply**.
- First click on the bay you want to fill in and then click on the member that the new beams should be parallel to, as shown in the diagrams below.





- Continue with other bays until the radial portion of your model is similar to that shown below:



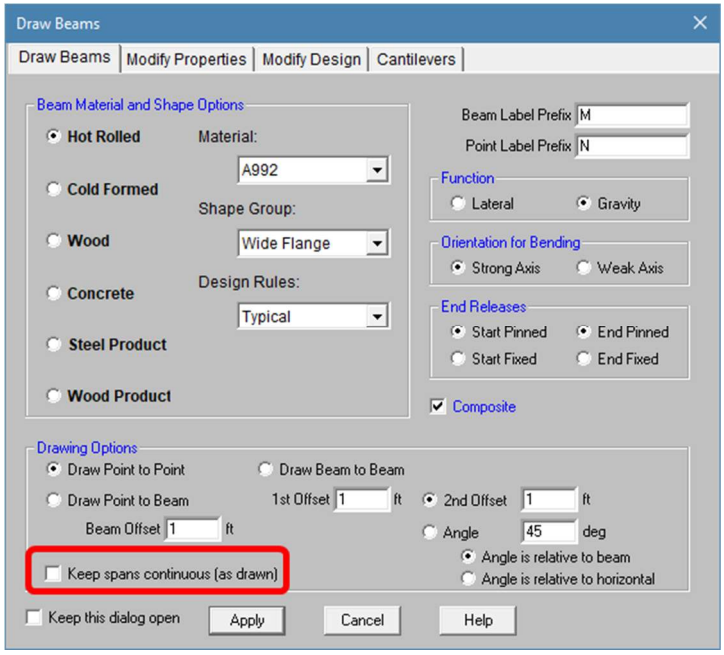
- Terminate the drawing tool using the right mouse button (right-click) or press ESC.

Additional Beam Drawing Tips (optional)

Before moving on, we recommend exploring some of the other beam modeling tools provided in RISAFloor. The steps in the remainder of this tutorial are optional, but you may find them very helpful to master.

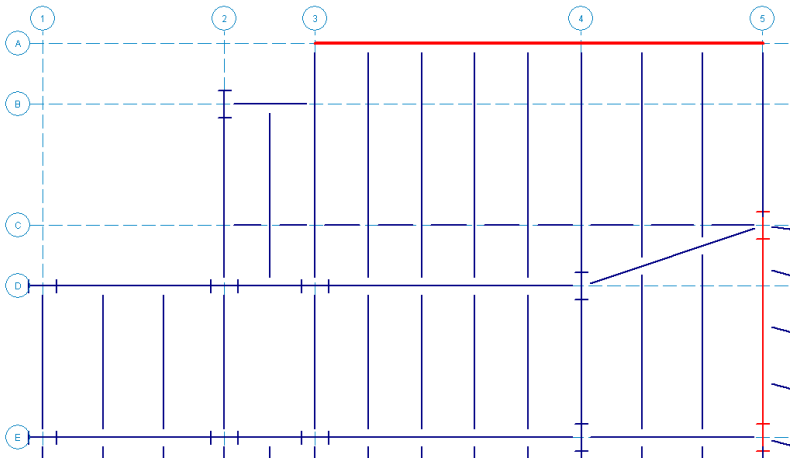
The first tool will demonstrate how to add ten members by simply clicking on two end points.

- On the Drawing toolbar, click the **Draw Beams** button again.
- Click the **Draw Beams** tab, make sure that the **Keep spans continuous (as drawn)** check box is NOT checked, then click **Apply**.



- Click at the start of the first beam (**C2**) and then again at the end (**C5**). Right-click to terminate the drawing tool.

Your model should now look like the image below. The single span beams will be automatically broken at the supports for you.

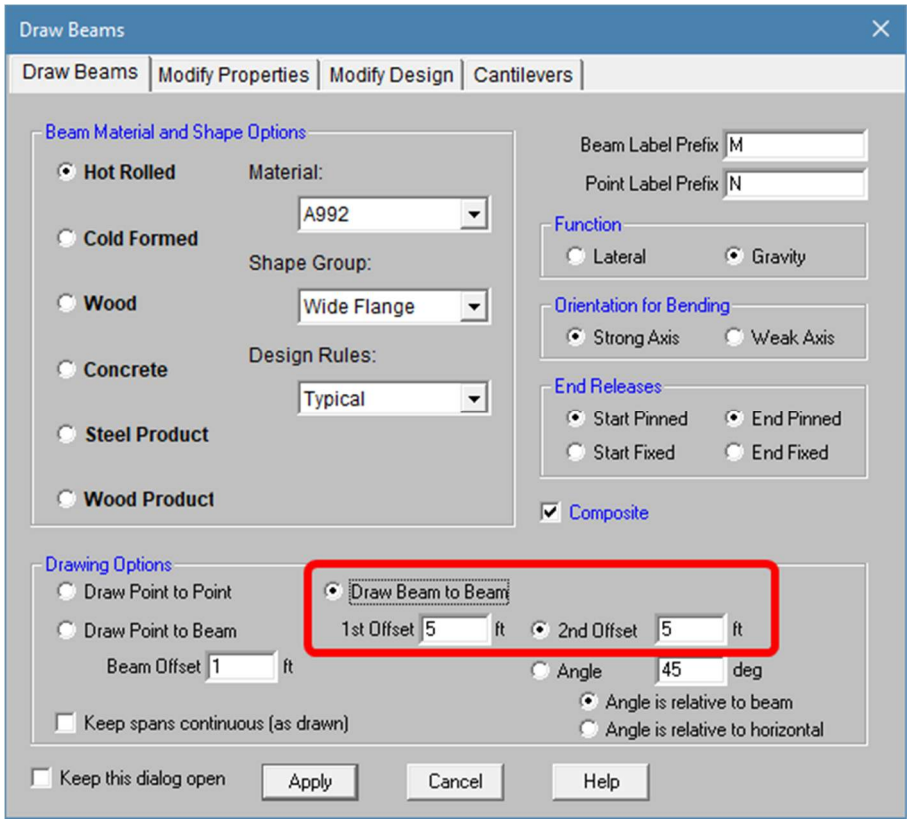


- Since these members should not be a part of your completed model, click the **Undo** button to undo the creation of the beams.

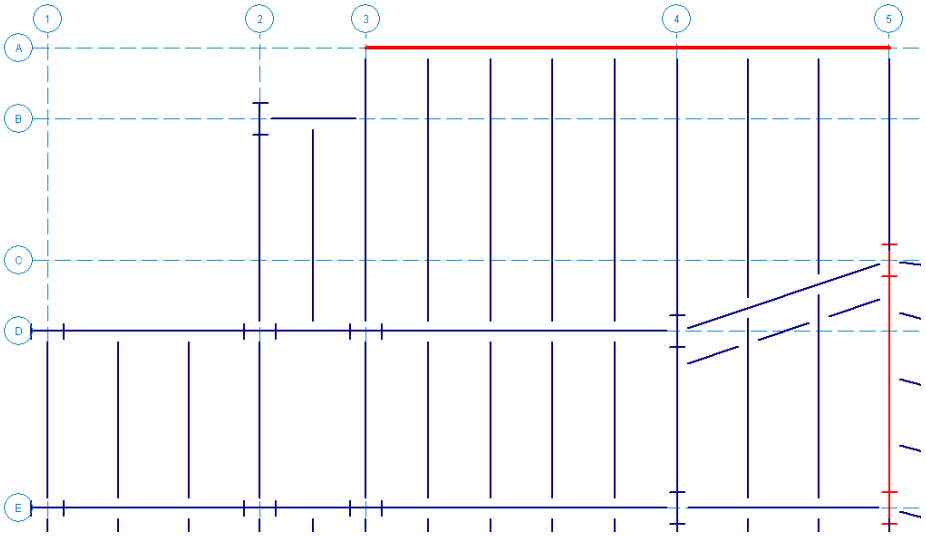
Note: When drawing new members, you may create either single or continuous span members. If the **Keep spans continuous** check box had been selected, RISAFloor would have created a continuous beam that spanned from **C2** to **C5**.

Another useful tool is the ability to offset beams from the end of another parallel beam:

- On the Drawing toolbar, click the **Draw Beams** button to again open the **Draw Beams** dialog box.
- Under **Drawing Options**, click **Draw Beam to Beam**. In both the **1st Offset** and **2nd Offset** boxes, enter **5** ft, and then click **Apply**.



- Experiment with this tool by clicking to create members parallel to the diagonal beam between **D4** and **C5**, as shown in the image below.



These offsets are automatically taken from the end of the beam that is closest to where you clicked.

- Click the **Undo** button to undo the creation of the beams.

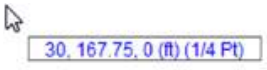
Lastly, we'll explore the quarter and third snap points on members, which can be a very powerful tool.

- On the Drawing toolbar, click the **Modify Drawing Grid** button and select the **Snap To Options** tab.
- Under **Beam/Wall Snap Locations**, make sure that the **Quarter Points** and **Third Points** check boxes are checked.
- Under **Universal Snap Increments**, make sure that the **Use Universal Increments** check box is cleared.
- Click **Ok**.

This allows you to draw to the quarter or third points of any beam or wall as a drawing point in your model. Use this tool to add the diagonal beams.

- On the Drawing toolbar, click the **Draw Beams** button again to open the **Draw Beams** dialog box.
- Under Drawing Options, select Draw Point to Point, and click Apply.
- Draw by clicking in the member from the quarter point of one beam, and dragging to the half point of another. Right-click to release the mouse.

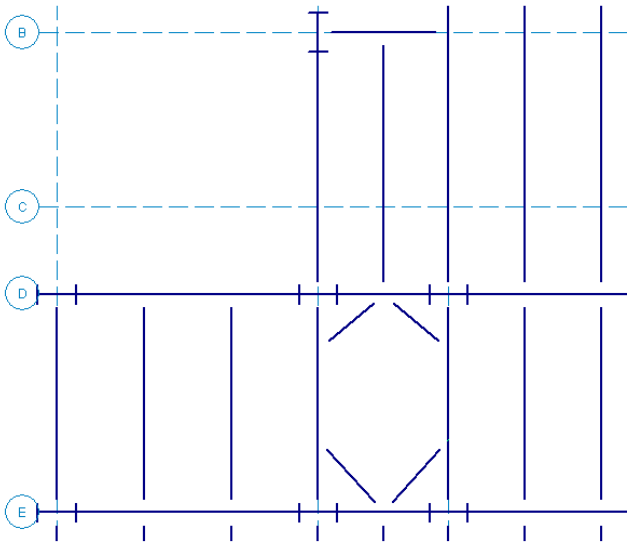
Note: The cursor provides additional information regarding the cursor coordinates. As you move your cursor around on your screen, a box adjacent to your cursor will appear and populate with the coordinates of the cursor.



The mouse coordinates that are displayed are the coordinates of the grid point or joint that is nearest to the cursor. These are the same coordinates displayed in the Status Bar in the third box when a model view is active.

Note: The Status Bar (in the lower right hand corner of the screen) reports not only the coordinates of your cursor, but also the half, third, and quarter point locations.

This tool eliminates the need to manually manipulate your **Project** or **Drawing Grids**.



- Click the **Undo** button several times to undo the creation of the beams.

This is the end of Tutorial A2.

You may save your model to be used as the starting point for the next tutorial, or begin the next tutorial using the .rfl starter file in the RISAFloor **Tutorials** folder. To save the model:

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

Note: If you are running the Demonstration version, the file is now too large to save. You will need to directly continue on with the remaining tutorials or use the Starter Files for the following tutorials.

Part A: Tutorial 3 – Decks & Diaphragms

This tutorial will guide you through adding a diaphragm edge, editing deck properties, and drawing in deck loads.

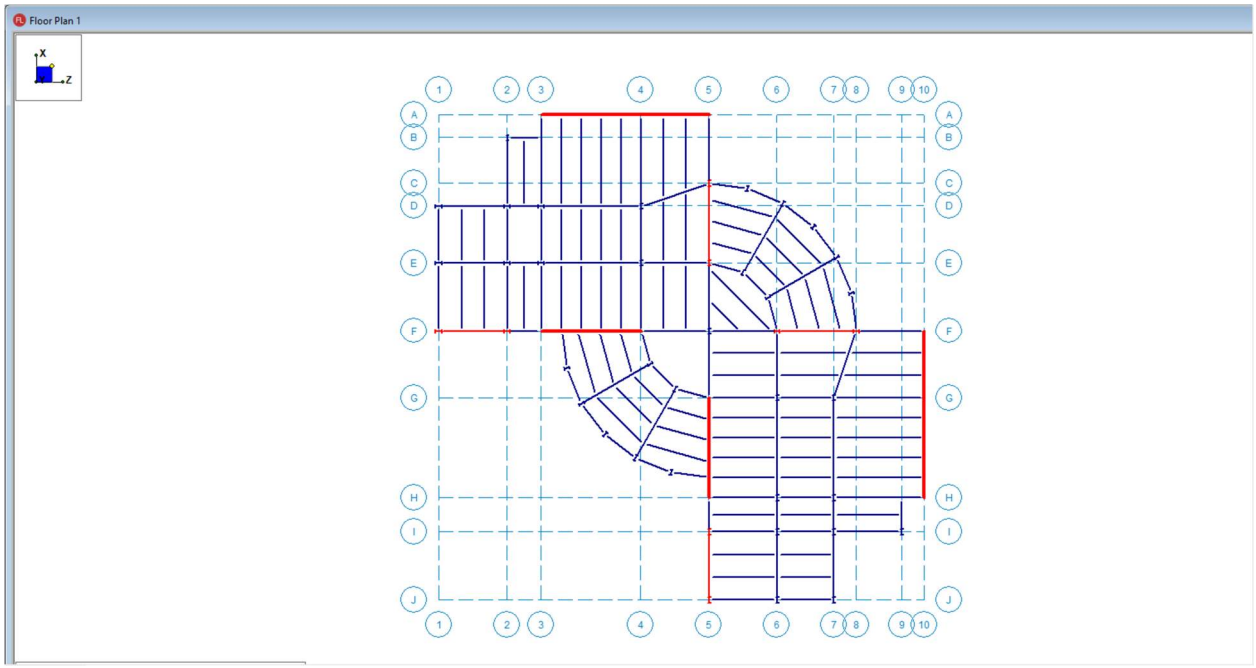
If you are continuing from the previous tutorial:

- From the **Window** menu, select **Single View**.
- On the Window toolbar, click the **Graphic Editing** button to activate the Drawing toolbar.
- On the **Data Entry** toolbar, click **Close** to close it. Skip ahead to the next page.

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

- Double-click on the **RISAFloor** icon to start the program.
- Click **Open Model**. Double-click the **Tutorials** folder, select **Tutorial A3 Starter.rfl** and click **Open**. Click **Close** (or **Cancel**) to exit the **Model Settings** dialog box.
- On the Window toolbar, click the **Graphic Editing** button to activate the Drawing toolbar.

Your screen should now look like this:

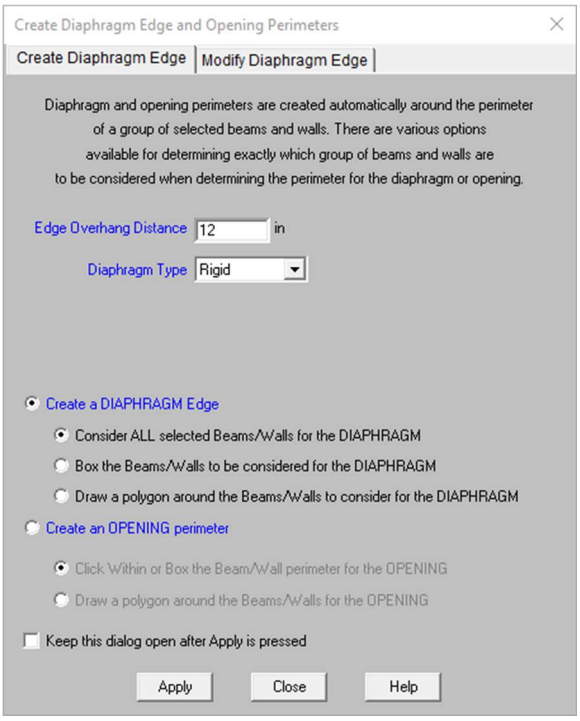


Creating Diaphragm Edges

RISAFloor provides tools to create or manipulate your diaphragm edges. To select the entire model and create a new diaphragm edge:

- On the Selection toolbar, click the **Select All** button to make sure the entire model is selected.

- On the Drawing toolbar, click the **Create Diaphragm Perimeters** button to open the following dialog box.



Hint:

When creating diaphragm edges, you can also use the selection tools to unselect any elements that you do not want inside the diaphragm edge.

Since the entire model is selected, your new diaphragm edge will encompass all the defined beams and walls.

- Click Create a DIAPHRAGM Edge.
- Click **Apply** to automatically create edges using the default settings.

Also, when defining diaphragm edges, you may box the elements or draw the edges yourself. This feature can help you select opening areas in your diaphragm.

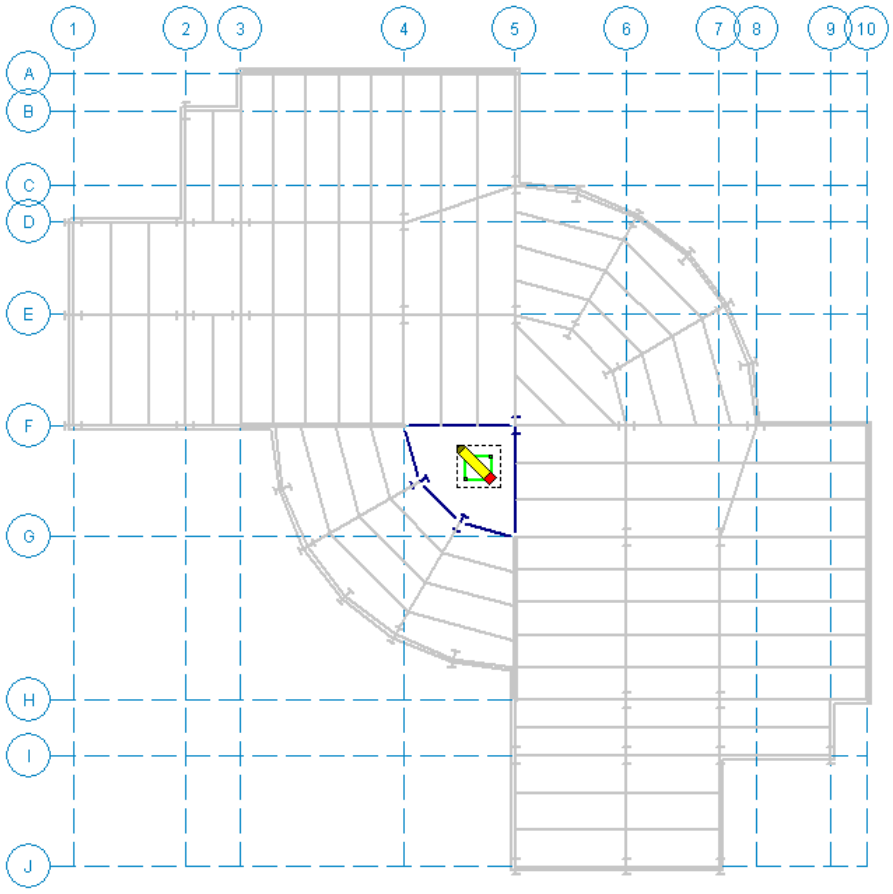
- On the Drawing toolbar, click the **Create Diaphragm Perimeters** button to open the dialog box again.
- Select **Create an OPENING perimeter**. When creating openings, you will primarily use the first option to click within the bays for the opening. The second option may be used when there are members that frame through the opening.

- Select Click Within or Box the Beam/Wall perimeter for the OPENING.
- Select the Keep this dialog open after Apply is pressed check box.
- Click **Apply**.

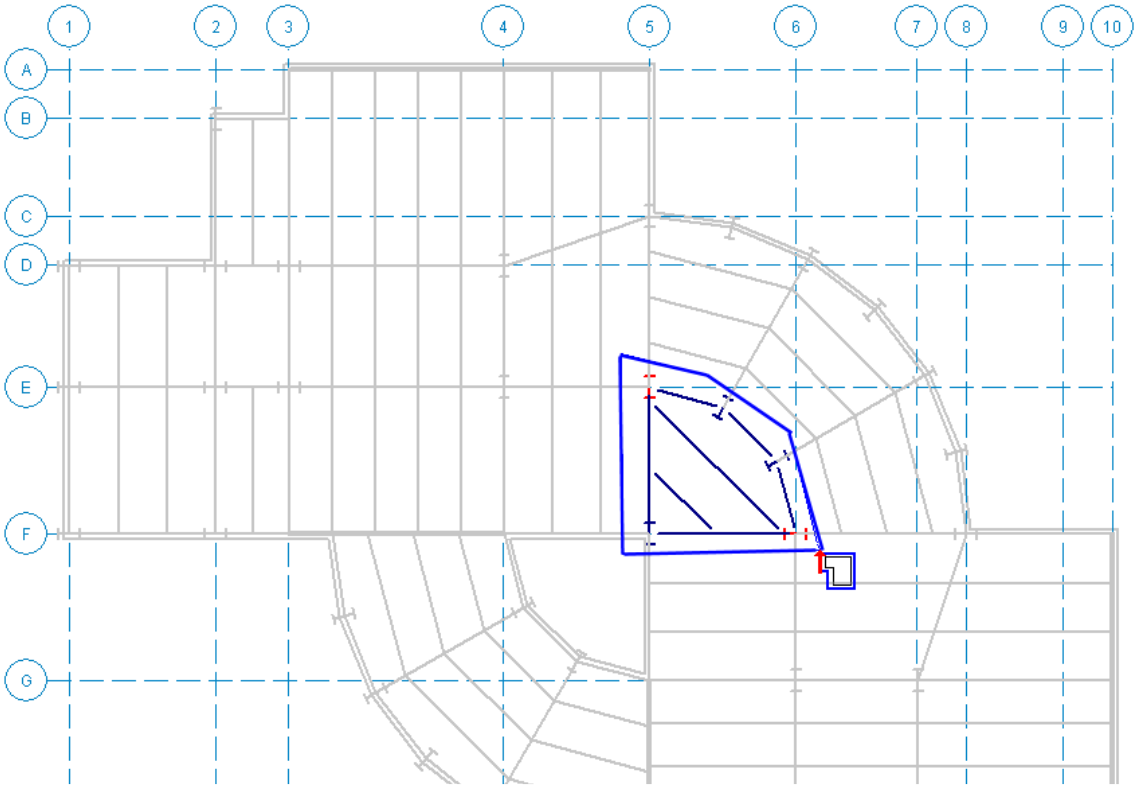
The Create Diaphragm Edge and Opening Perimeters dialog box should remain open.

Note: If the dialog box is covering any portion of your model, you can move it by clicking on the title bar and dragging it out of the way.

- Click in the bay, as shown below.



- Go back to the dialog box and select Draw a Polygon Around the Beams/Walls for the OPENING, then click Apply.
- Draw a polygon around the opening, as shown image below. When your polygon is complete, double-click to close it off.



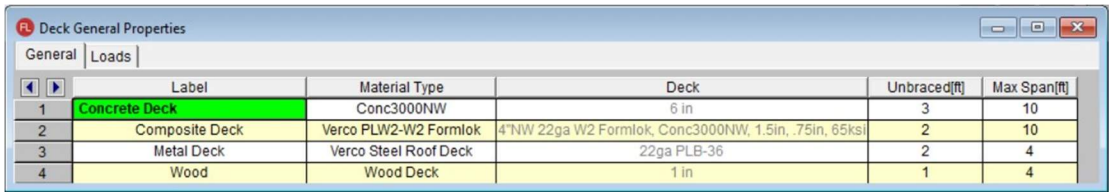
- When finished, use the right mouse button (right-click) or press ESC to terminate the drawing tool.
- Click Close to exit the Create Diaphragm Edge and Opening Perimeters dialog box.

You have now created your diaphragm edge. The next step is to review and/or modify the deck properties that are assigned to it.

Deck Properties

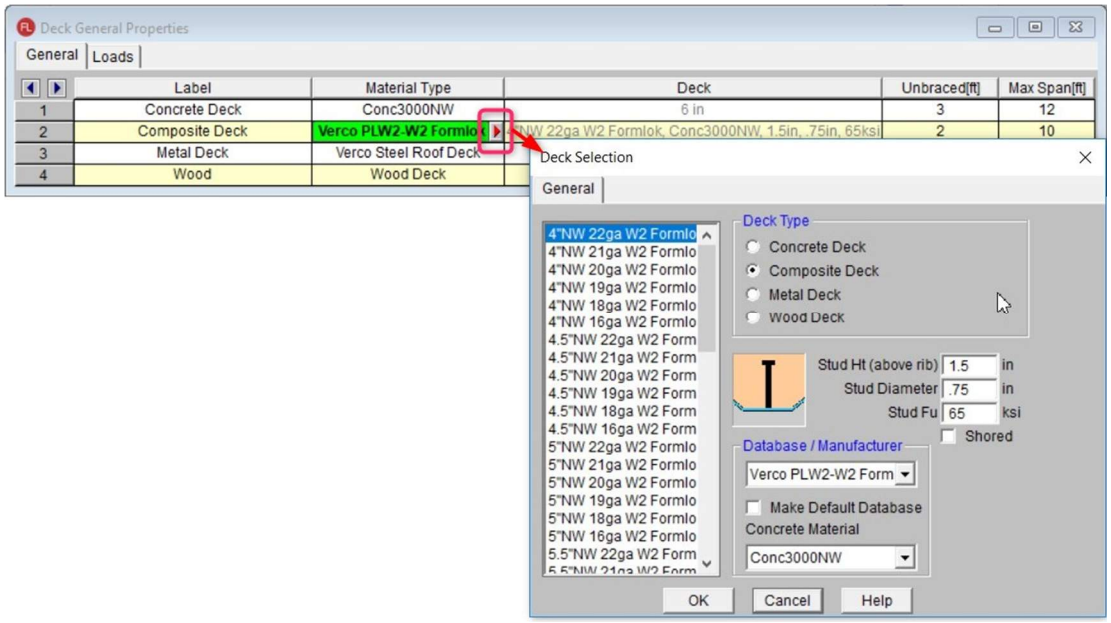
To specify the properties of the deck, it is important to understand a little about how the deck spreadsheet works:

- On the Main menu, click Spreadsheets, select Deck Properties.



	Label	Material Type	Deck	Unbraced[ft]	Max Span[ft]
1	Concrete Deck	Conc3000NW	6 in	3	10
2	Composite Deck	Verco PLW2-W2 FormIok	4"NW 22ga W2 FormIok, Conc3000NW, 1.5in, .75in, 65ksi	2	10
3	Metal Deck	Verco Steel Roof Deck	22ga PLB-36	2	4
4	Wood	Wood Deck	1 in	1	4

- For the **Concrete Deck**, change **Max Span** to **12** ft.
- Next, click within the **Material Type** cell of the second row (**Composite Deck**). Click on the red arrow to open the **Deck Selection** dialog.



- Click through the various **Deck Types** to see the available options. Click **Cancel** when you are finished.
- Next, click the **Loads** tab and review the loading input options.

The self-weight of the deck and the construction dead and construction live loads may all be specified in the **Loads** tab.

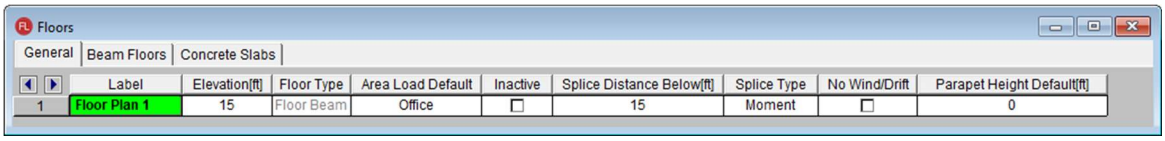
Note: The loads in the **Loads** tab of the **Deck** spreadsheet are different from the area loads defined in the **Area Loads** spreadsheet because these loads are specifically linked to the type of deck construction.

- Close the spreadsheet by clicking **Close** .

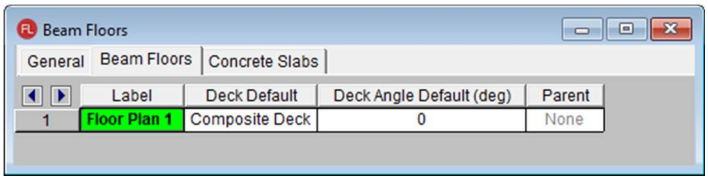
Next, you will learn how to apply these deck properties to your floor.

Each floor has a **Default Deck** and **Deck Angle** that will be used for the entire floor if no other deck is defined. To review and modify these defaults:

- On the **Main** menu, select **Spreadsheet** and click **Floors**.
- Under Area Load Default, select Office.



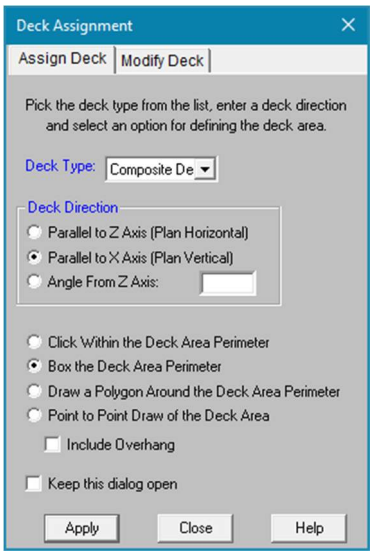
- Next, click the **Beam Floors** tab.
- Under Deck Default, select Composite Deck.
- Under Deck Angle Default, enter 0.



Unless you specifically assign a new area load or deck to a region of the floor, these default properties will be assumed to be applied over the whole floor (within the diaphragm edge).

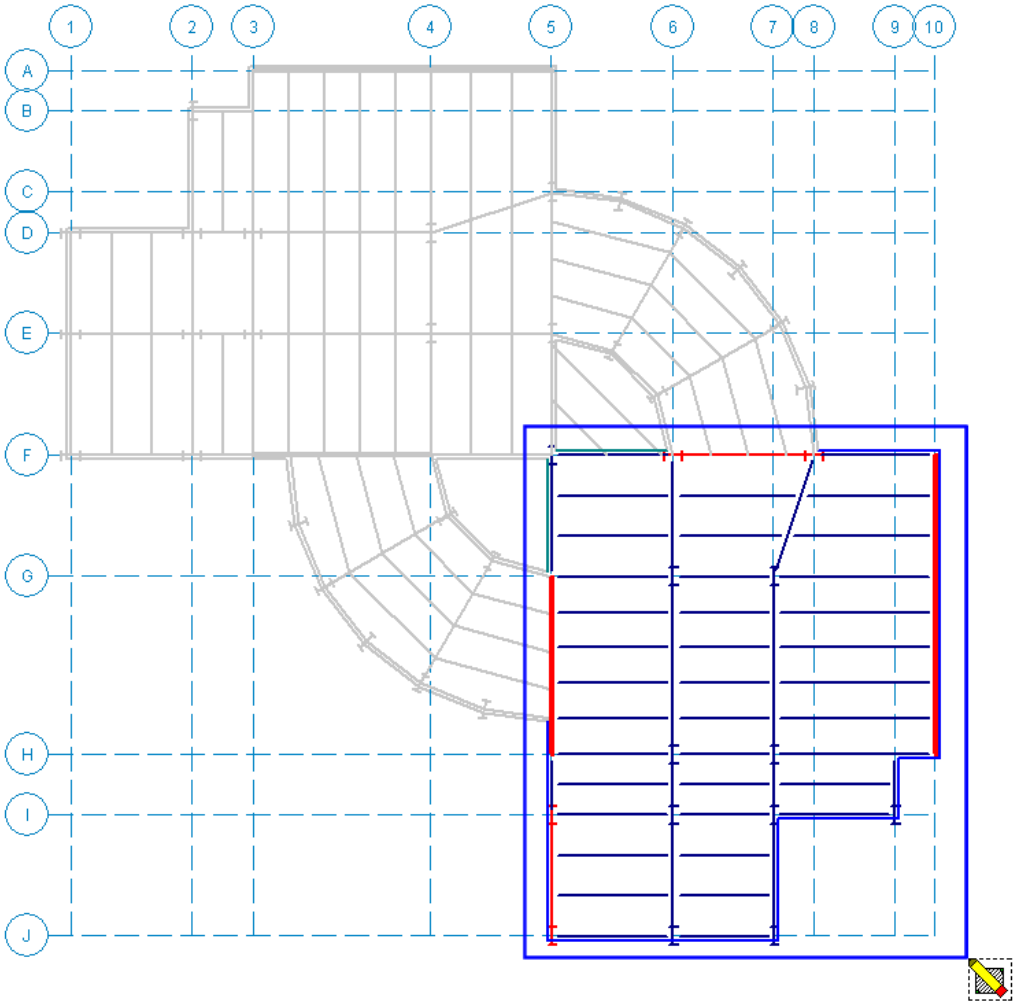
Since your model has some areas where the default slab and orientation will be different, assign a new deck angle to these areas:

- Click **Close** to exit the spreadsheet and return to the model view.
- On the Drawing toolbar, select the **Assign Diaphragm Edge and Deck Properties** button.
- In the Deck Type box, select Composite Deck.
- Under Deck Direction, click Parallel to X Axis.
- Select Box the Deck Area Perimeter.



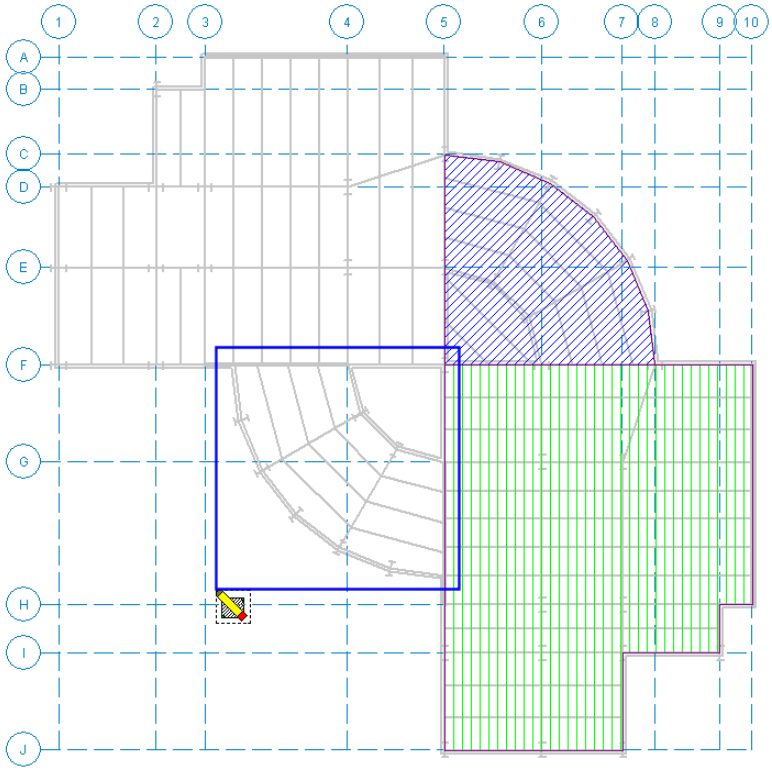
- Click **Apply**.

- Draw a box around the area between **F5** and **J10** as shown in the following image:



- Press CTRL+D to reopen the **Deck Assignment** dialog box.
- In the Deck Type box, select Concrete Deck.
- Under **Deck Direction**, click **Angle from Z Axis**, and enter an angle of **45** degrees.
- Click **Apply**.

- Assign this data to the two radial portions of your model by boxing them as shown in the image below.

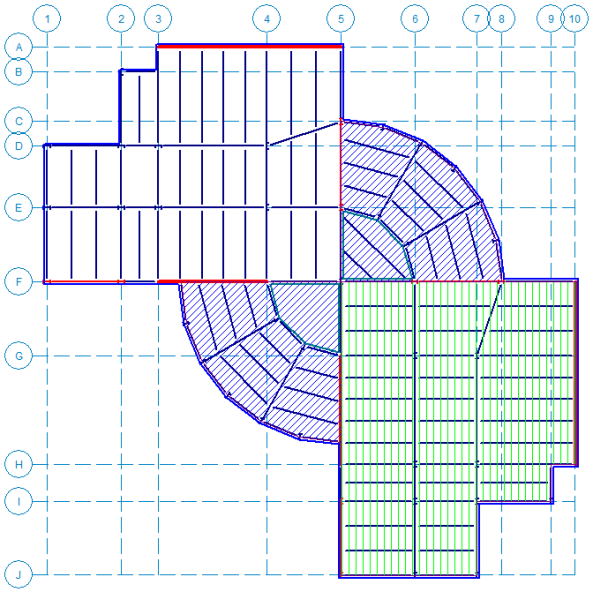


Note:

The color of the hatch identifies what type of deck is assigned and the direction of the hatching coincides with the general span direction of the deck. This hatching will show angles at intervals of 45 degrees, the closest multiple will be used.

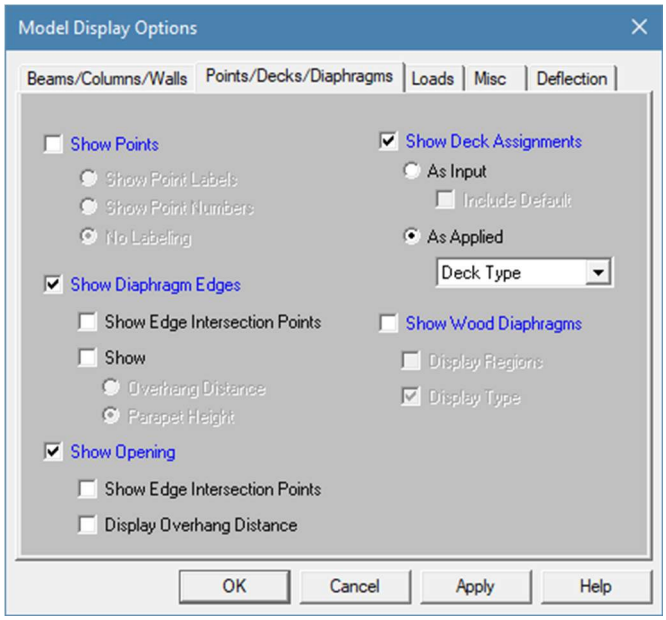
This is a good visual confirmation of the decks that you have applied to your floor. In the area where no local deck is applied, the default deck will be used. In areas where more than one local deck is applied, the top deck (the last one drawn) will be used.

When you are finished drawing in your deck, your screen should look like the image shown below.



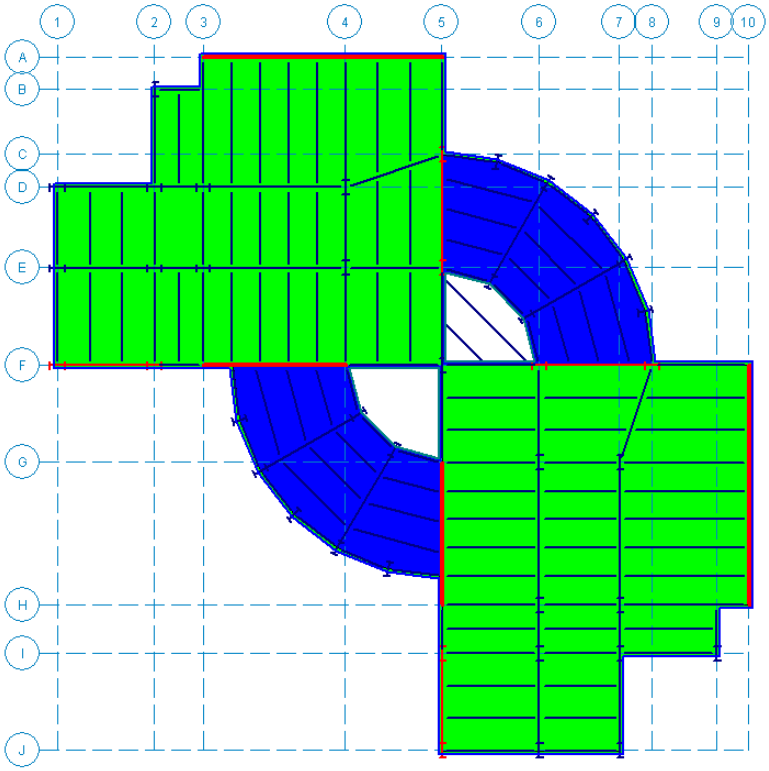
RISAFloor gives you two ways to plot the deck graphically. The first is to plot the deck properties as they are defined. The second is to view them as they are resolved by the solver and applied to the members. We are going to view the deck as applied:

- On the **Main** menu, click **View**, select **Model Display Options**.
- Select the Points/Decks/Diaphragms tab.
- Make sure the **Show Deck Assignments** check box is selected, and select **As Applied**.



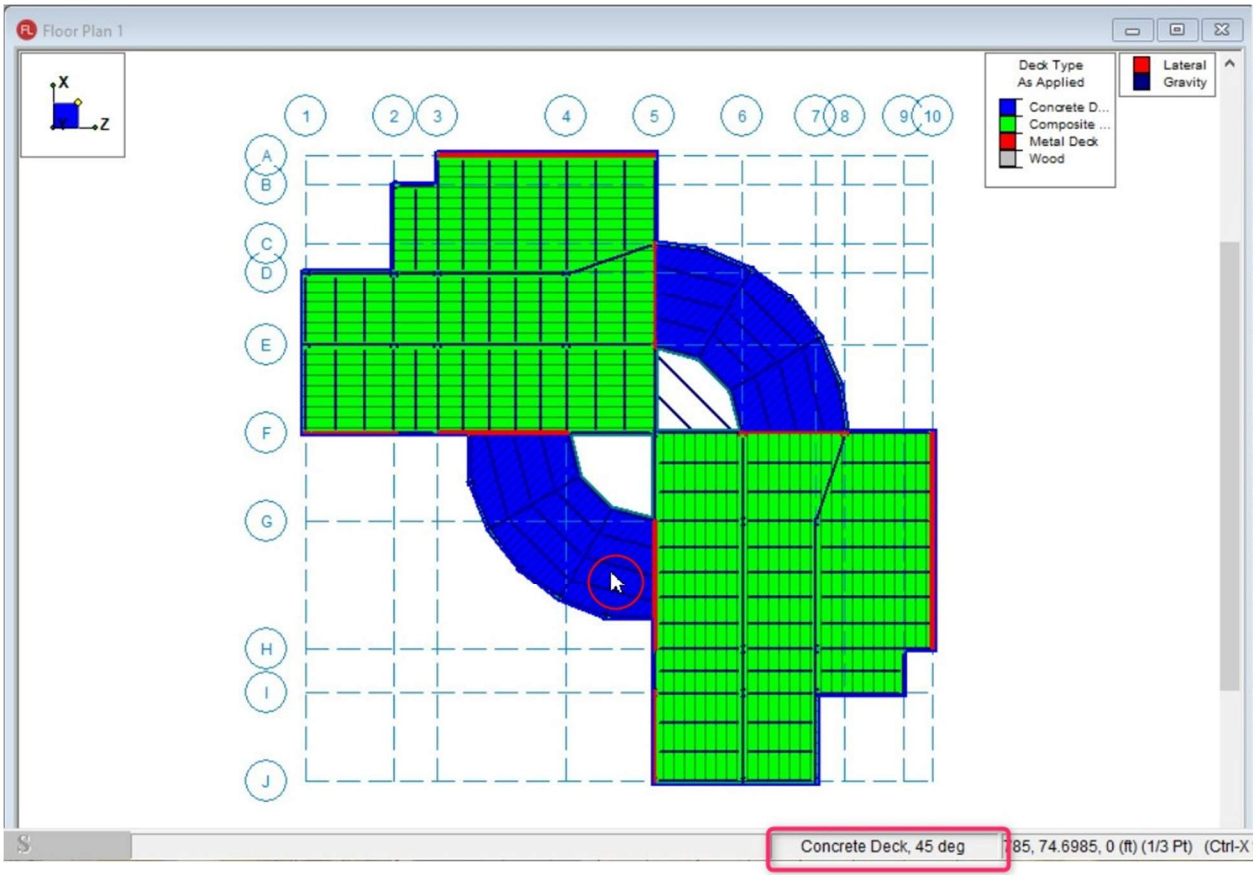
- Click **OK**.

Your display should change to look like the following image:



Notice the differences between this plot and the previous plot. In this plot, you can verify the location of openings.

- Move your mouse over the various decks. Notice the deck type and orientation information are displayed in the Status bar as you move the cursor.



Note: If your mouse cursor is still shown as the deck drawing cursor, use the right mouse (right-click) or ESC to terminate this drawing tool.

Since the next section deals with area loads, you can turn off the deck display:

- On the **Main** menu, click **View**, select **Model Display Options**.
- Select the Points/Decks/Diaphragms tab.
- Clear the **Show Deck Assignments** check box, and then click **OK** to close the dialog box.

This is the end of Tutorial A3.

You may save your model to be used as the starting point for the next tutorial, or begin the next tutorial using the .rfl starter file in the RISAFloor **Tutorials** folder. To save the model:

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

Note: If you are running the Demonstration version, the file is now too large to save. You will need to directly continue on with the remaining tutorials or use the Starter Files for the following tutorials.

Part A: Tutorial 4 – Loading

This tutorial will guide you through editing and adding the various types of loads available in RISAFloor.

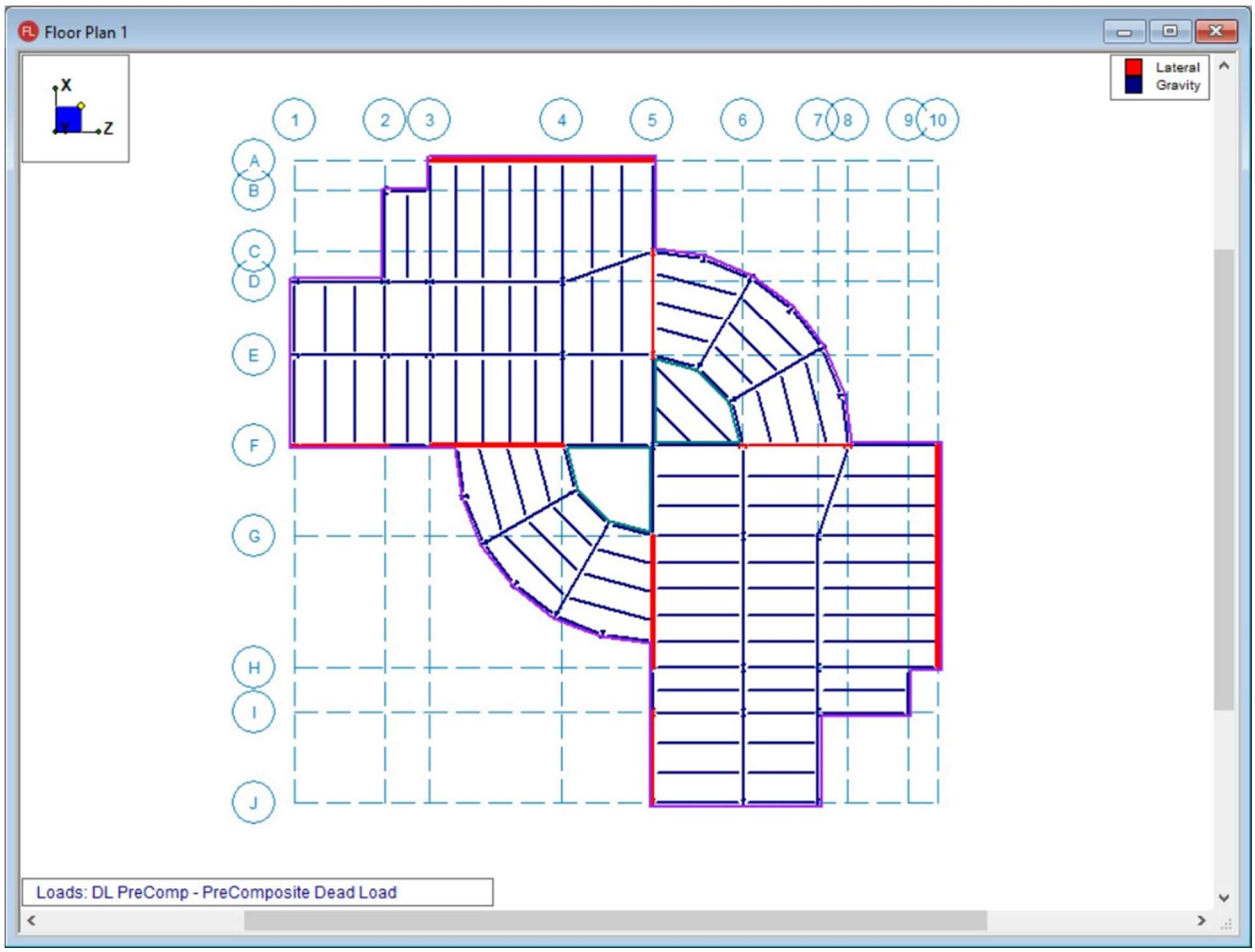
If you are continuing from the previous tutorial:

- From the **Window** menu, select **Single View**.
- On the Window toolbar, click the **Graphic Editing** button to activate the Drawing toolbar.
- On the **Data Entry** toolbar, click **Close** to close it. Skip ahead to the next page.

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

- Double-click on the **RISAFloor** icon to start the program.
- Click **Open Model**. Double-click the **Tutorials** folder, select **Tutorial A4 Starter.rfl** and click **Open**. Click **Close** (or **Cancel**) to exit the **Model Settings** dialog box.
- On the Window toolbar, click the **Graphic Editing** button to activate the Drawing toolbar.

Your screen should now look like this:

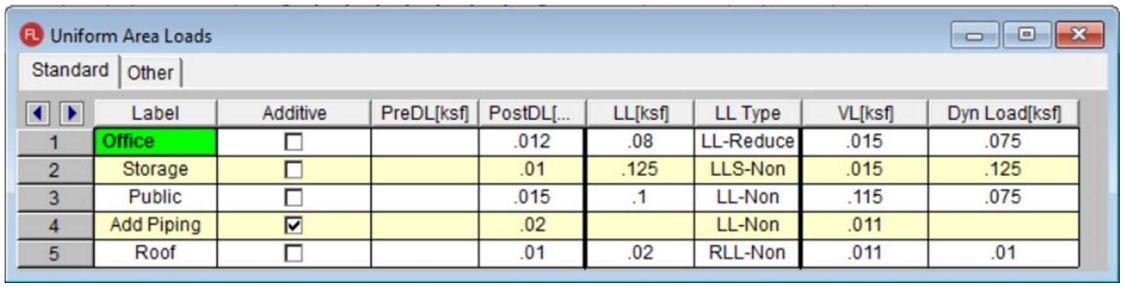


Area Loads

Area load magnitudes are defined in a spreadsheet and then applied to the floor as polygons, much like the deck properties in the previous section. Area loads that are within the diaphragm edges (but not inside an opening) are automatically attributed in the deck direction to the beams and walls and then reduced for member design according to the live load reduction methods chosen in **Model Settings**.

Edit the **Area Loads** spreadsheet, as shown below:

- On the **Main** menu, click **Spreadsheets**, select **Loads**, and then select **Area Load Definitions**.
- Edit the load information to match what is shown in the image below:



	Label	Additive	PreDL[ksf]	PostDL[...	LL[ksf]	LL Type	VL[ksf]	Dyn Load[ksf]
1	Office	☐		.012	.08	LL-Reduce	.015	.075
2	Storage	☐		.01	.125	LLS-Non	.015	.125
3	Public	☐		.015	.1	LL-Non	.115	.075
4	Add Piping	☑		.02		LL-Non	.011	
5	Roof	☐		.01	.02	RLL-Non	.011	.01

The entries in the **Area Loads** spreadsheet are described below:

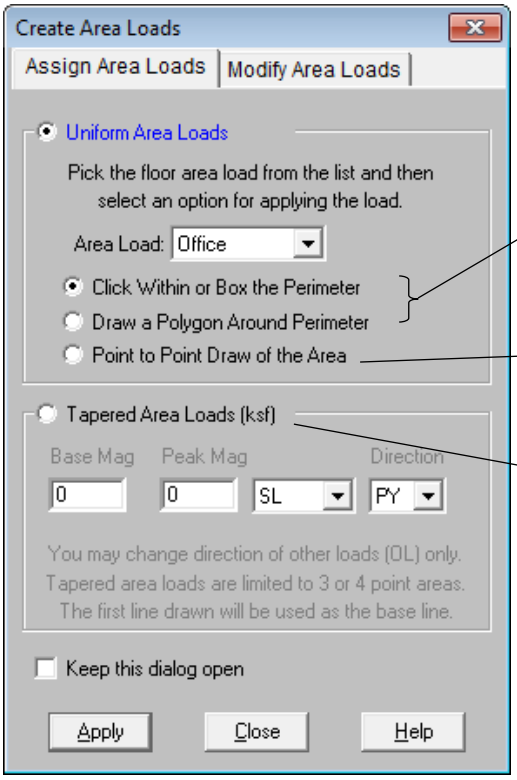
Label	The name you will later use to refer to the load when you apply or view it.
Additive	If this is checked, the load applied will add to the load that is drawn beneath it in the order of how they were drawn in the model.
PreDL PostDL LL LL Type	These entries allow you to specify load magnitudes for different categories that will later be used in the load combinations that you solve. ▪ PreDL and PostDL are for dead loads that are pre- and post-composite. Remember that other dead loads (deck self-weight and construction dead load) were already defined within the deck definitions spreadsheet. ▪ LL and LL Type load columns are for the live load magnitude and the type of live load which you may choose from a list.
VL	Specifies the vibration load. The vibration load (along with the self-weight of the beam and deck) will be used for calculating the mass for a vibration check, per *AISC Design Guide #11*.
Dyn Load	The dynamic load will be used to determine the diaphragm mass and mass moment of inertia for a RISA-3D seismic analysis.

Each floor has a default area load that will be used within the entire floor slab if no other load is specified. You can review this information as follows: on the **Main** menu, click **Spreadsheet**, then select **Floors**. The **Area Load Default** is set to **Office**. To use different loads in an area, you can graphically define local area loads.

- Click **Close** to exit the spreadsheets and return to the model view.

To apply local area loads:

- On the Drawing toolbar, click the **Draw Area Loads** button to display the following dialog box:



These two options allow you to apply the load automatically by clicking or drawing a polygon around the members which define the area.

-OR-

This option allows you to draw the polygon manually by clicking on points.

Notice you also have the option of drawing in a tapered load. This can be useful for modeling certain types of snow loads, but will not be used in this tutorial. For more information on tapered area loads please refer to the *RISAFloor General Reference*.

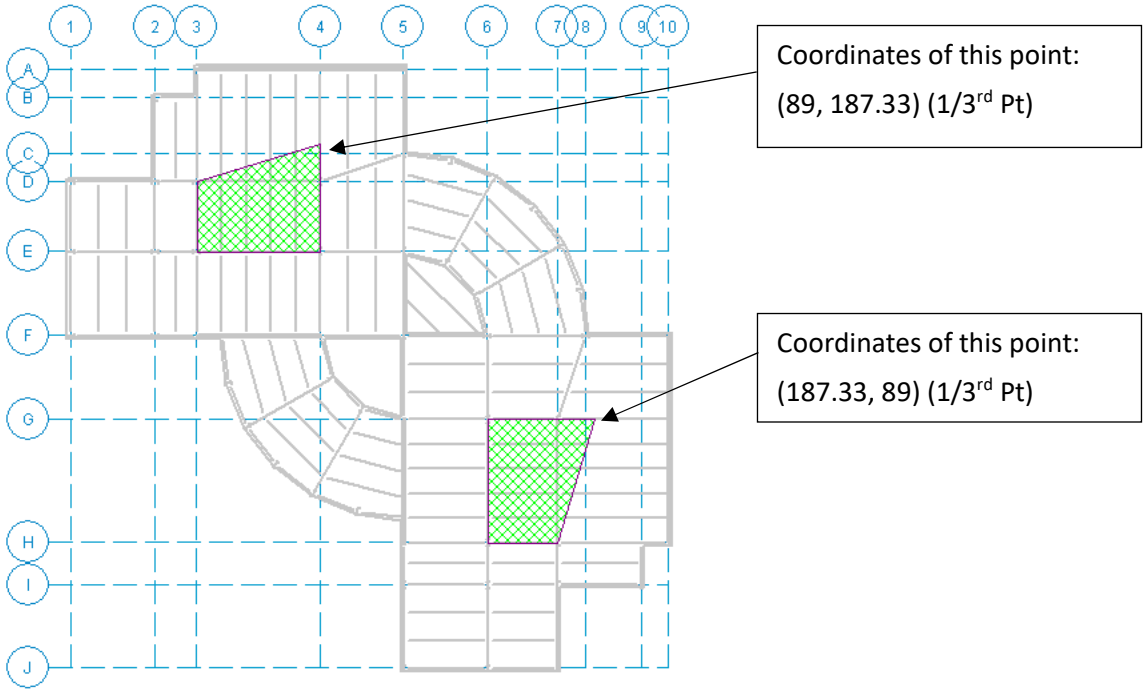
Draw the area loads:

- Under Uniform Area Loads, in the Area Load box, select Storage, then select the Point to Point Draw of the Area drawing option.
- Click **Apply**.

Note: The cursor will change to .

- Draw the polygons for the two area loads by clicking on points, as shown in the following figure. Double click on the last point to finish drawing the polygon.

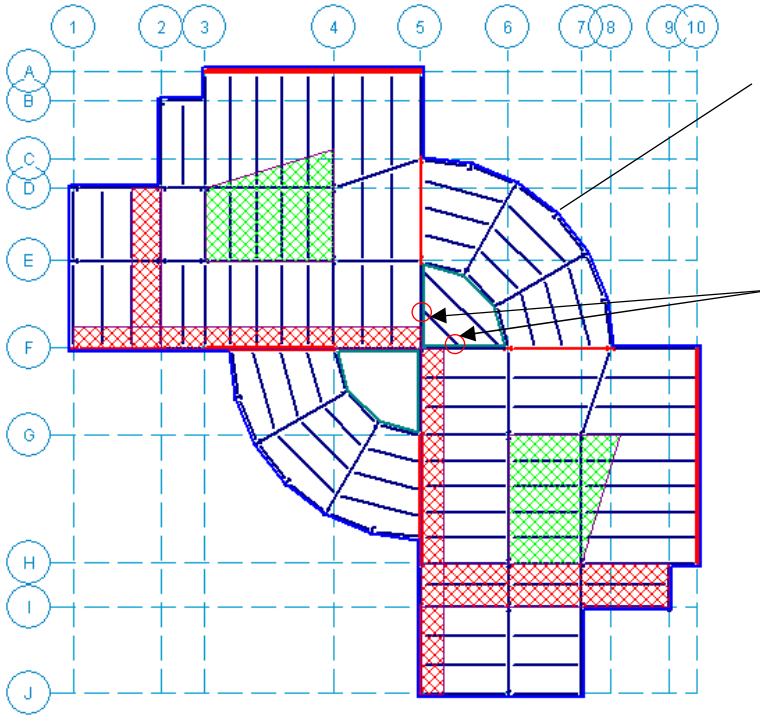
Tip: When you are in drawing mode, you can automatically snap to the quarter and third points of any beam (as demonstrated in the previous tutorial).



- Press CTRL+D to reopen the **Create Area Loads** dialog box.
- Under Uniform Area Loads, in the Area Load box, select Public, then select the Point to Point Draw of the Area drawing option.
- Click **Apply**.

- Draw in the public area loads, shown in the next image.

When you are finished drawing area loads, your screen should look like the image shown below.



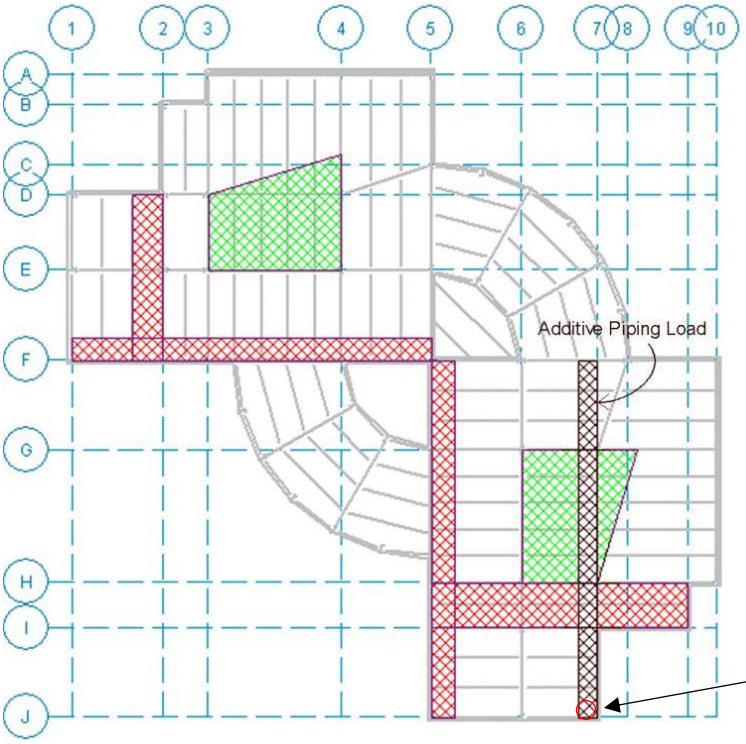
The blank areas where you have not drawn in a load will receive the default **Office** load automatically.

Area Load edges are at the quarter point locations.

Note: Loads do not have to start or end on project grid lines. In areas where more than one load is applied, the top load (the last one drawn) will be used by RISAFloor. The exception to this is that any additive load will be applied along with the load beneath it.

- Press CTRL+D to reopen the **Create Area Loads** dialog box.
- Under Uniform Area Loads, in the Area Load box, select Add Piping.
- Click **Apply**.

- Draw in a piping load that overlaps the other loads, as shown in the image below, by clicking on points.



This load is bounded either by the quarter point of the beam or by the project grid points. Since the piping was specified as additive, this new load will be automatically combined with the other existing loads.

Coordinate location: **(167.75, 0)**

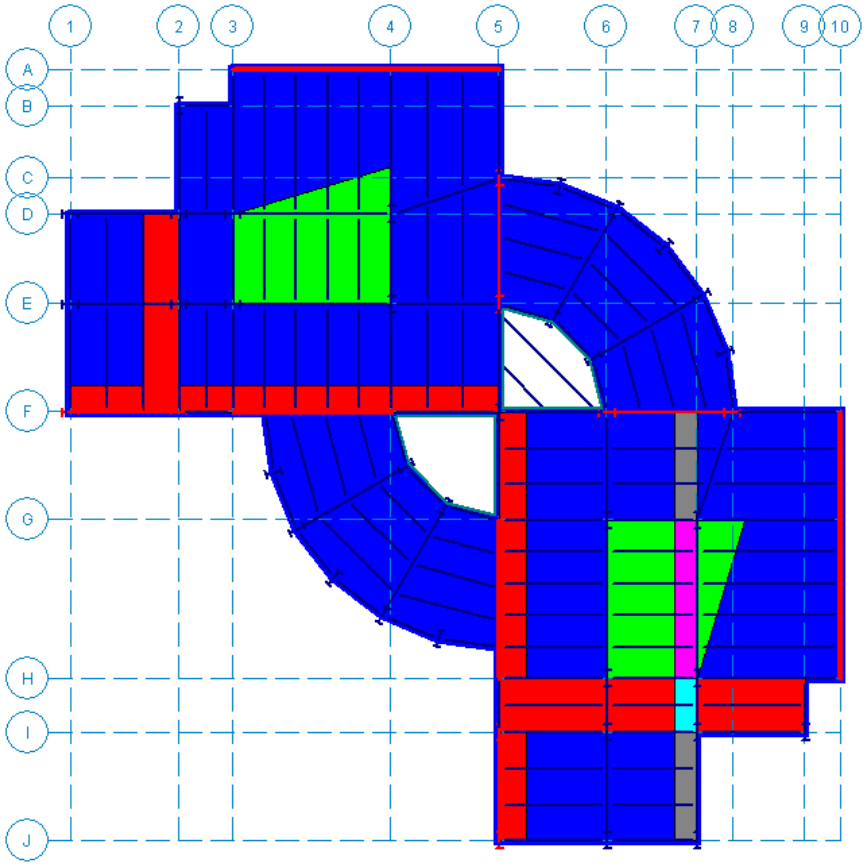
- Press ESC to cancel the drawing tool and return to your regular cursor.

RISAFloor gives you two ways to plot the area loads. The first is to view them as they were input. The second is to view them as they are resolved and applied by RISAFloor. You are going to view them as applied.

To plot the area loads:

- On the **Main** menu, click **View**, select **Model Display Options**.
- Select the **Loads** tab.
- Make sure the **Uniform Area Loads** check box is selected, click **As Applied**, and then click **Apply**.
- In the **Show Loads for Category** box, select **DL PostComp**. Click **OK** to exit.

Your screen should look similar to the figure shown below:



Notice that the default **Office** load was applied automatically, and that no area load is applied to the two slab openings. Also, note that the additive piping load was automatically combined with the **Office**, **Public**, and **Storage** loads that it crossed.

On the Window Toolbar, you can use these three controls to manipulate the display of your loads:

Toggle between load categories and combinations.

Toggle the graphical display of the loads.

Select the various load categories or combinations.

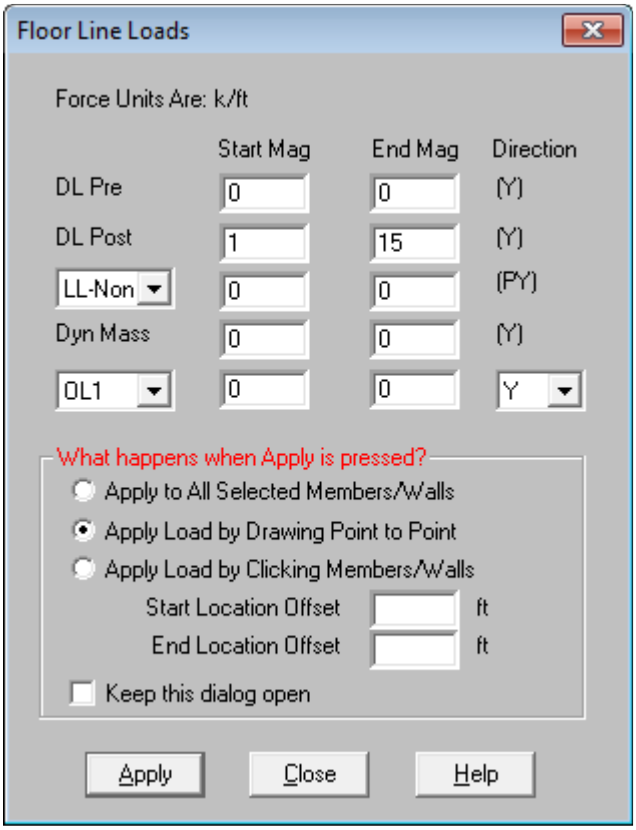
Turn off the display of the area loads, as follows:

- On the **Main** menu, click **View**, select **Model Display Options**.
- Click the **Loads** tab, and then click to clear the **Uniform Area Loads** check box.
- Click **OK**.

Line Loads

You can use line loads to model the weight of cladding, partitions, architectural components, etc. Prepare to draw in line loads as follows:

- On the Drawing toolbar, select the **Line Loads** button  .
- Enter the information as it appears in the following figure:

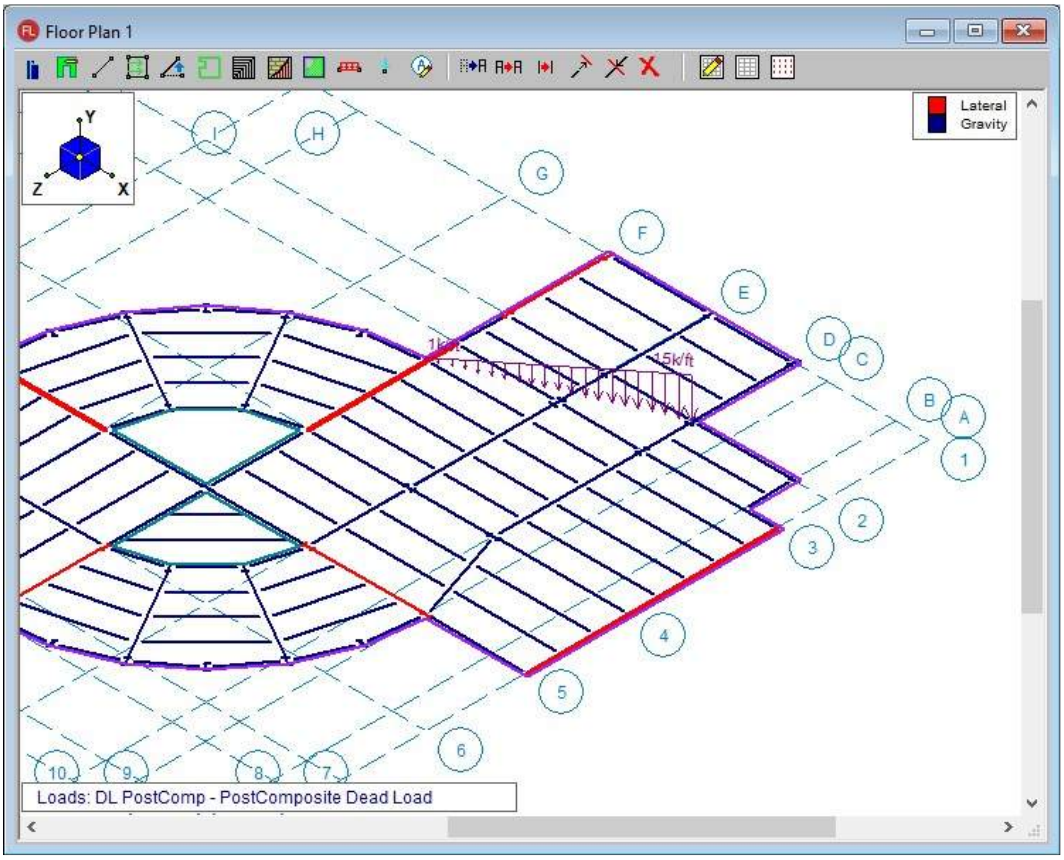


- Click **Apply**.

Note: The cursor will change to  .

To see the model better while you are drawing in the load, change to an isometric view and zoom in:

- On the Window toolbar, click on the **Isometric** button Iso.
- Click the **Zoom** buttons to zoom in on the right-hand portion of the floor as shown in the next image.
- Draw in the load by clicking on the two points, as shown in the image below. When finished, right-click to release the mouse.



When you are finished, notice that you have specified a tapered line load to represent a non-structural partition. The line load spans diagonally across the deck. This is important because it will actually result in parabolic loading of the supporting beams.

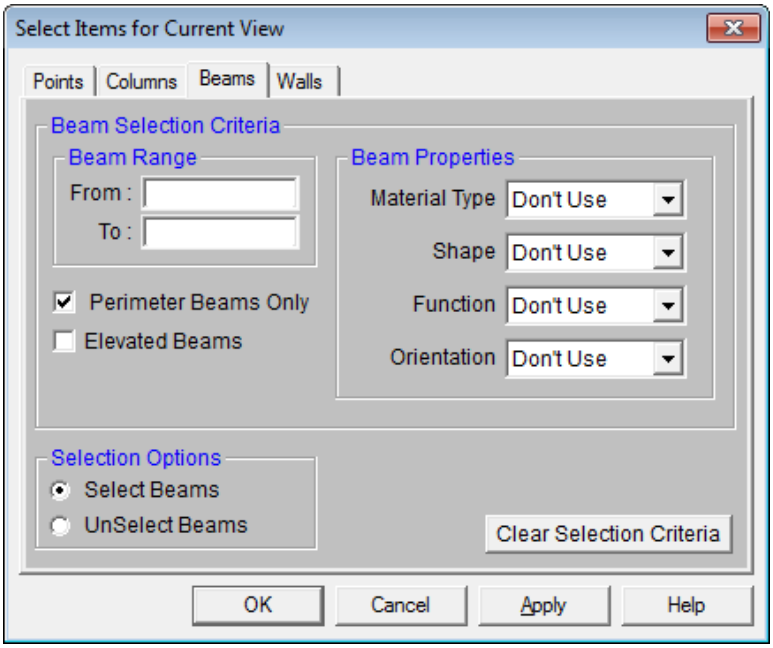
To return to a planar view and continue:

- Use the right mouse button (right-click) or press ESC to exit the drawing mode.
- On the Window toolbar, click the **Plan** button Pln.
- On the Window toolbar, click **Redraw** to return to a full view.

Next, the loads due to the exterior cladding need to be modeled. To do this, apply a uniform load of 2.5 kips/ft to all beams on the perimeter of the building. RISAFloor provides a selection tool that makes this easy.

First, unselect your entire model and make the following selection changes:

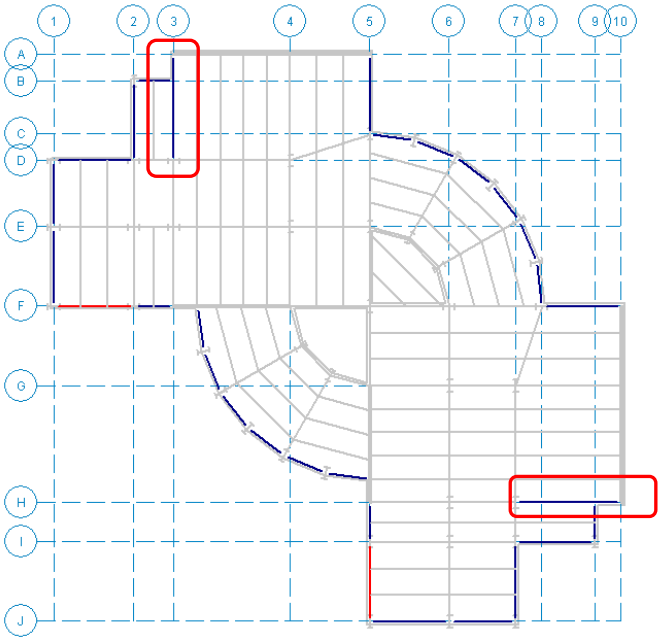
- On the Selection toolbar, click **UnSelect All** to unselect your entire model.
- Also on the Selection toolbar, click on the **Selection Criteria** button. The **Select Items for Current View** dialog box will open.
- Click the **Beams** tab and select the **Perimeter Beams Only** check box.
- Under Selection Options, click Select Beams.



- Click **OK** to apply the selection.

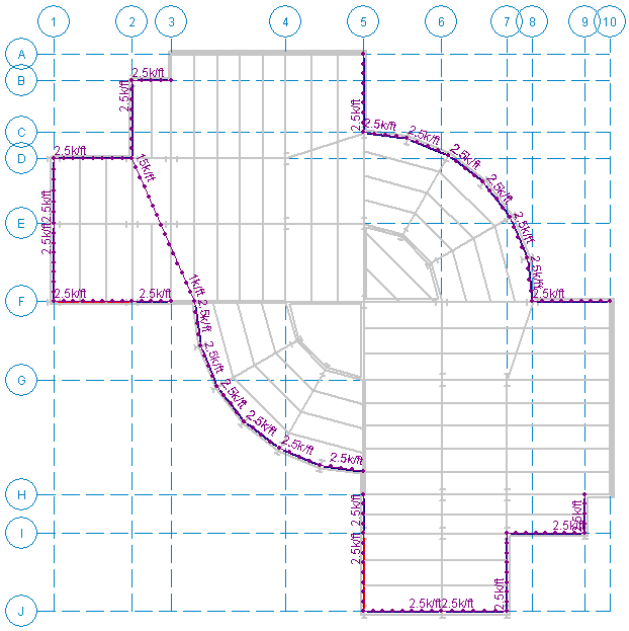
Notice that only the exterior beams of the building are selected. Also notice that you have selected two beams that only run along the perimeter for a portion of their length. You must apply your cladding to these beams separately.

- Unselect each of the beams circled below by clicking on them.



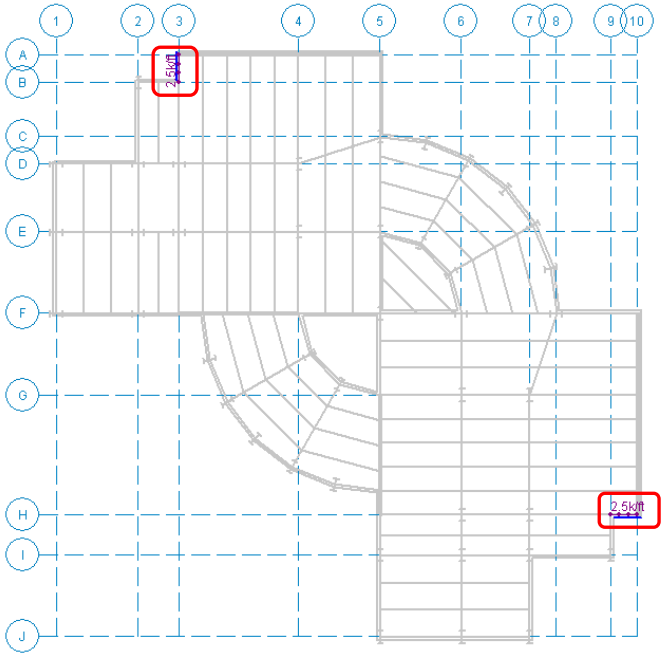
- On the Drawing toolbar, select the **Line Load** button  again to open the **Floor Line Loads** dialog box.
- For the **DL Post load**, enter a starting and ending magnitude of **2.5 kip/ft**.
- Under What happens when Apply is pressed?, click Apply to All Selected Members/Walls.
- Click **Apply**.

Your structure should look like this:



Now, add the last two cladding loads.

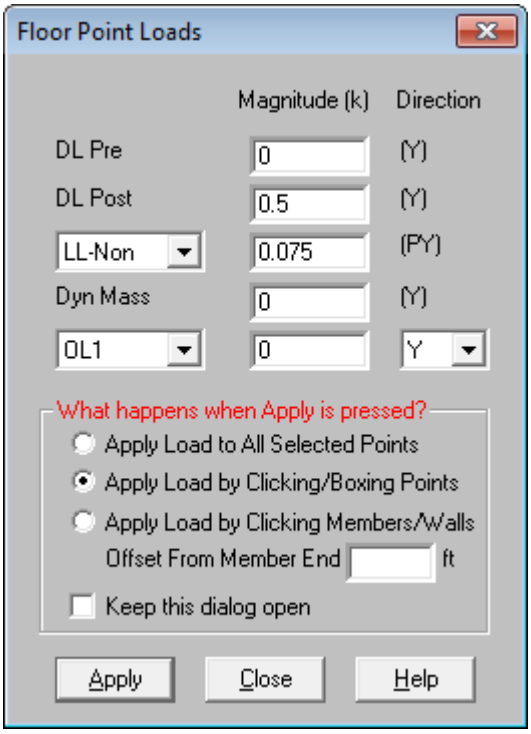
- Press CTRL+D to re-open the **Floor Line Loads** dialog box.
- Click Apply Load by Drawing Point to Point.
- Click **Apply**.
- Draw in the line loads between **A3** and **B3** by clicking each point. Use your right mouse button (right-click) or press ESC to release your mouse after drawing each load.
- Draw in the line loads between **H9** and **H10** by clicking each point.



Point Loads

Prepare to apply point loads:

- On the Selection toolbar, click **Select All** to select the model.
- On the Drawing toolbar, click **Assign point loads** .
- Enter the information show below:



- Click **Apply**.

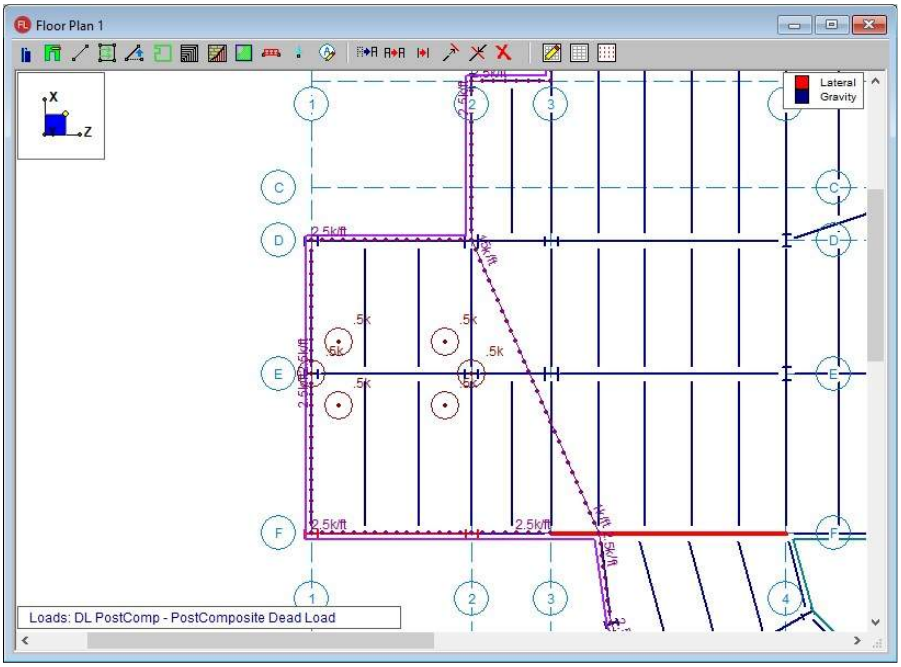
Since some of your loads are located in the middle of a deck span, it will be helpful to turn on the universal drawing snap points and zoom in on your model:

- On the Drawing toolbar, click the **Modify Drawing Grid** button .
- Click on the **Snap To Options** tab.
- Select the **Use Universal Increments** checkbox and click **Ok**.

Draw in the point loads:

- Zoom in significantly on the left side portion of the floor using the mouse roller, as shown below.
- Use the coordinates (notice they are displayed in the Status bar) and the snap points to draw in a point load at each of the following coordinates:
  (**0**,**149**)
  (**5**,**143**)
  (**5**,**155**)
  (**25**,**143**)
  (**25**,**155**)
  (**30**, **149**)

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



Note: Point loads that are applied inside the deck edges may be applied anywhere on the deck and will be attributed automatically to the beams, walls, and columns. However, if they are applied outside of deck edges or within deck openings, they must be applied directly on beams, walls or columns.

This is the end of Tutorial 4.

You may save your model to be used as the starting point for the next tutorial, or begin the next tutorial using the .rfl starter file in the RISAFloor **Tutorials** folder. To save the model:

On the **File** menu, click **Save As** and enter a file name.

Note: If you are running the Demonstration version, the file is now too large to save. You will need to directly continue on with the remaining tutorials or use the Starter Files for the following tutorials.

Part A: Tutorial 5 – Solution & Results

This tutorial will guide you through generating load combinations, running a solution, design optimization, and reviewing the results.

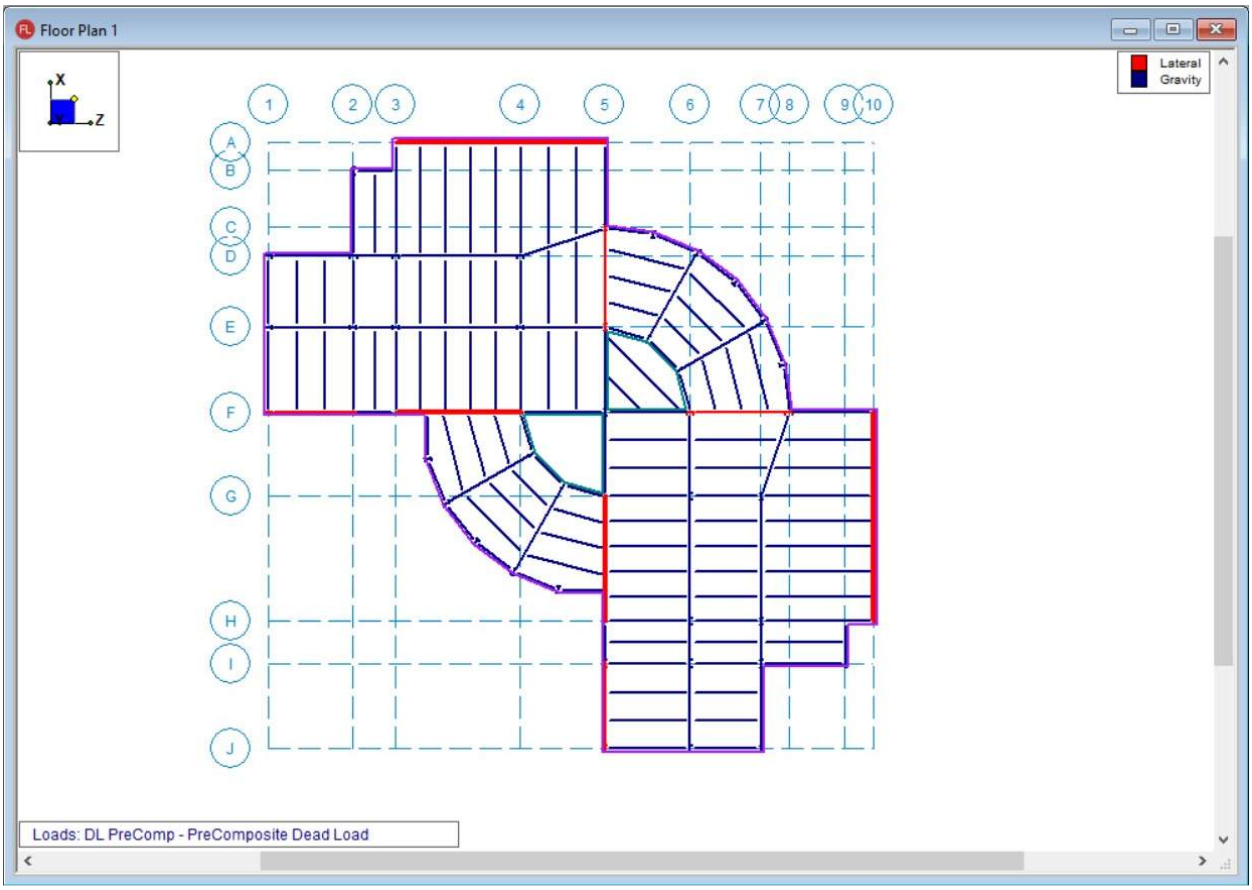
If you are continuing from the previous tutorial:

- From the **Window** menu, select **Single View**.
- On the Window toolbar, click the **Graphic Editing** button to activate the Drawing toolbar.
- On the **Data Entry** toolbar, click **Close** to close it. Skip ahead to the next page.

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

- Double-click on the **RISAFloor** icon to start the program.
- Click **Open Model**. Double-click the **Tutorials** folder, select **Tutorial A5 Starter.rfl** and click **Open**. Click **Close** (or **Cancel**) to exit the **Model Settings** dialog box.
- On the Window toolbar, click the **Graphic Editing** button to activate the Drawing toolbar.

Your screen should now look like this:

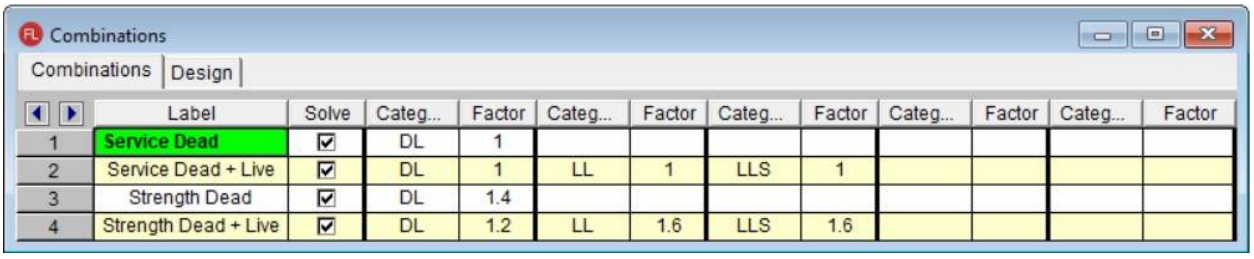


RISAFloor solves load combinations that are defined in the **Load Combinations** spreadsheet. As RISAFloor designs members, it chooses members that satisfy the codes specified in the **Model Settings** and the rules that are defined in the **Design Rules** spreadsheet.

Load Combinations

To view the load combinations:

- On the Main menu, click Spreadsheets, select Load Combinations.



On the **Combinations** tab, you will combine and factor the various loads that you have already applied to your model.

- Press the **F4** key four times to delete all of the default load combinations.

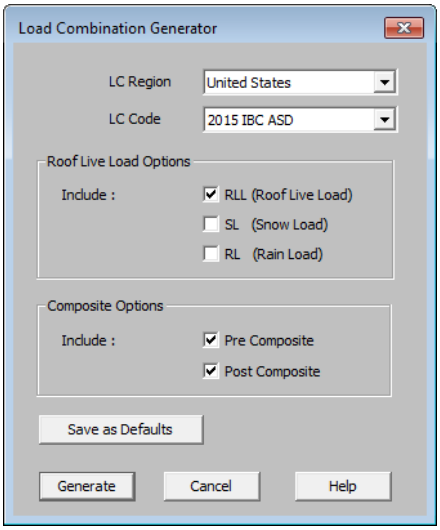
Now that you have a spreadsheet open, notice the Window toolbar looks different.

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

. The **Load Combination Generator** dialog box will be used to generate the load combinations needed to design the model.

For this tutorial, assume that in the regions where this project will be built, the governing roof loads are usually roof live load, rather than snow or rain.

- In the **LC Region** list, select **United States**.
- In the **LC Code** list, select **2015 IBC ASD**.
- Click to clear the **SL** and **RL** check boxes so that you will not be generating more load combinations than are needed.



- Click Generate.

The following load combinations will be generated by RISAFloor:

Combinations

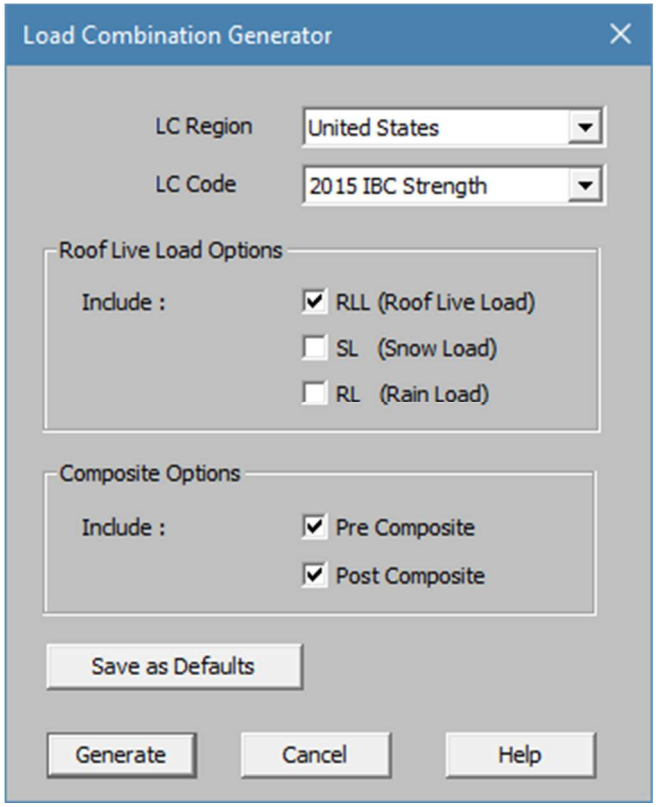
Combinations | Design

	Label	Solve	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor	C
1	IBC 16-8 Pre	☑	DLPre	1											
2	IBC 16-9 Pre	☑	DLPre	1	LLConst	1									
3	IBC 16-8 Post	☑	DL	1											
4	IBC 16-9 Post	☑	DL	1	LL	1	LLS	1							
5	IBC 16-10 (a) Post	☑	DL	1	RLL	1									
6	IBC 16-11 (a) Post	☑	DL	1	LL	.75	LLS	.75	RLL	.75					
7	IBC 16-11 (b) Post	☑	DL	1	LL	.75	LLS	.75							

Notice there are really two sets (categories) of load combinations. The first two (Load Combinations 1 and 2, above) use pre-composite loads only. The other five (Load Combinations 3-7) combine the total dead load with the various live loads that may be applied to your model.

Next, you will generate LRFD load combinations for your concrete wall panel design.

- Click the **LC Generator** button LC Generator.
- In the LC Code list, select 2015 IBC Strength.



- Click Generate.

This finalizes your load combinations as shown below:

Combinations

Combinations | Design

	Label	Solve	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor	C
1	IBC 16-8 Pre	☑	DLPre	1											
2	IBC 16-9 Pre	☑	DLPre	1	LLConst	1									
3	IBC 16-8 Post	☑	DL	1											
4	IBC 16-9 Post	☑	DL	1	LL	1	LLS	1							
5	IBC 16-10 (a) Post	☑	DL	1	RLL	1									
6	IBC 16-11 (a) Post	☑	DL	1	LL	.75	LLS	.75	RLL	.75					
7	IBC 16-11 (b) Post	☑	DL	1	LL	.75	LLS	.75							
8	IBC 16-1 Pre	☑	DLPre	1.4											
9	IBC 16-2 Pre	☑	DLPre	1.2	LLConst	1.6									
10	IBC 16-1 Post	☑	DL	1.4											
11	IBC 16-2 (a) Post	☑	DL	1.2	LL	1.6	LLS	1.6	RLL	.5					
12	IBC 16-2 (b) Post	☑	DL	1.2	LL	1.6	LLS	1.6							
13	IBC 16-3 (a) Post	☑	DL	1.2	RLL	1.6	LL	.5	LLS	1					
14	IBC 16-3 (c) Post	☑	DL	1.2	LL	.5	LLS	1							

Next, you will mark which load combinations to use for each material.

- Click the **Design** tab to review the remaining information associated with these generated load combinations.

Combination Design

Combinations | Design

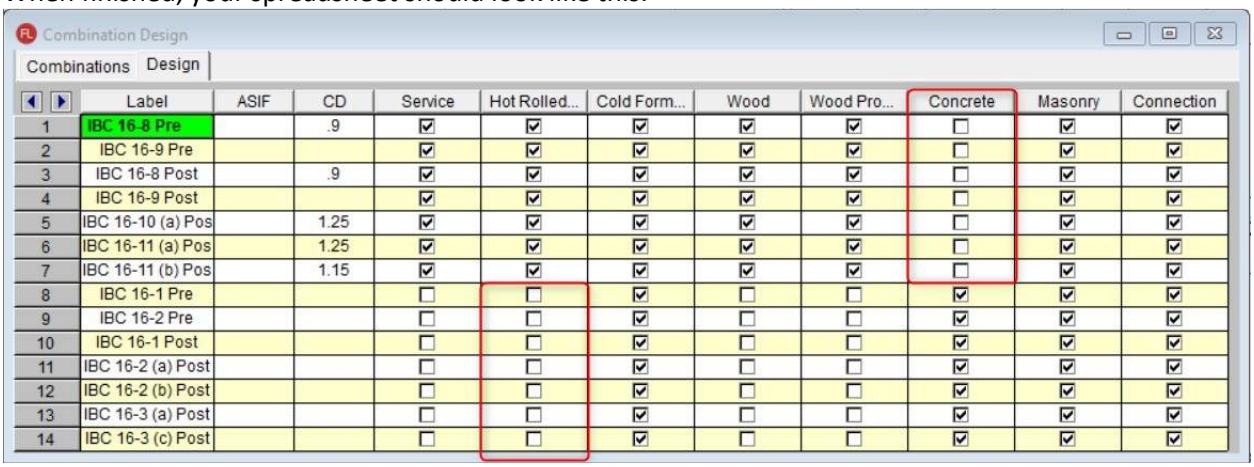
	Label	ASIF	CD	Service	Hot Rolled...	Cold Form...	Wood	Wood Pro...	Concrete	Masonry	Connection
1	IBC 16-8 Pre		.9	☑	☑	☑	☑	☑	☑	☑	☑
2	IBC 16-9 Pre			☑	☑	☑	☑	☑	☑	☑	☑
3	IBC 16-8 Post		.9	☑	☑	☑	☑	☑	☑	☑	☑
4	IBC 16-9 Post			☑	☑	☑	☑	☑	☑	☑	☑
5	IBC 16-10 (a) Pos		1.25	☑	☑	☑	☑	☑	☑	☑	☑
6	IBC 16-11 (a) Pos		1.25	☑	☑	☑	☑	☑	☑	☑	☑
7	IBC 16-11 (b) Pos		1.15	☑	☑	☑	☑	☑	☑	☑	☑
8	IBC 16-1 Pre			☐	☑	☑	☐	☐	☑	☑	☑
9	IBC 16-2 Pre			☐	☑	☑	☐	☐	☑	☑	☑
10	IBC 16-1 Post			☐	☑	☑	☐	☐	☑	☑	☑
11	IBC 16-2 (a) Post			☐	☑	☑	☐	☐	☑	☑	☑
12	IBC 16-2 (b) Post			☐	☑	☑	☐	☐	☑	☑	☑
13	IBC 16-3 (a) Post			☐	☑	☑	☐	☐	☑	☑	☑
14	IBC 16-3 (c) Post			☐	☑	☑	☐	☐	☑	☑	☑

This spreadsheet allows you to determine which load combinations shall apply to each material type. You also have the ability to enter the **ASIF** (Allowable Stress Increase Factor for the AISC 9th Edition Hot Rolled Steel Code) and the **CD** (Load Duration Factor) for NDS wood design.

Since you have generated both strength and service load combinations, you will need to use the material checkboxes to designate which combinations apply to the Hot Rolled Steel members and which apply to the Concrete walls.

- Uncheck the **Hot Rolled Steel** checkboxes for the strength level combinations (lines **8-14**).
- Uncheck the **Concrete** checkboxes for the service level combinations (lines **1-7**).

When finished, your spreadsheet should look like this:



	Label	ASIF	CD	Service	Hot Rolled...	Cold Form...	Wood	Wood Pro...	Concrete	Masonry	Connection
1	IBC 16-8 Pre		.9	☑	☑	☑	☑	☑	☐	☑	☑
2	IBC 16-9 Pre			☑	☑	☑	☑	☑	☐	☑	☑
3	IBC 16-8 Post		.9	☑	☑	☑	☑	☑	☐	☑	☑
4	IBC 16-9 Post			☑	☑	☑	☑	☑	☐	☑	☑
5	IBC 16-10 (a) Pos		1.25	☑	☑	☑	☑	☑	☐	☑	☑
6	IBC 16-11 (a) Pos		1.25	☑	☑	☑	☑	☑	☐	☑	☑
7	IBC 16-11 (b) Pos		1.15	☑	☑	☑	☑	☑	☐	☑	☑
8	IBC 16-1 Pre			☐	☐	☑	☐	☐	☑	☑	☑
9	IBC 16-2 Pre			☐	☐	☑	☐	☐	☑	☑	☑
10	IBC 16-1 Post			☐	☐	☑	☐	☐	☑	☑	☑
11	IBC 16-2 (a) Post			☐	☐	☑	☐	☐	☑	☑	☑
12	IBC 16-2 (b) Post			☐	☐	☑	☐	☐	☑	☑	☑
13	IBC 16-3 (a) Post			☐	☐	☑	☐	☐	☑	☑	☑
14	IBC 16-3 (c) Post			☐	☐	☑	☐	☐	☑	☑	☑

- Close the spreadsheet by clicking the **Close** button  .

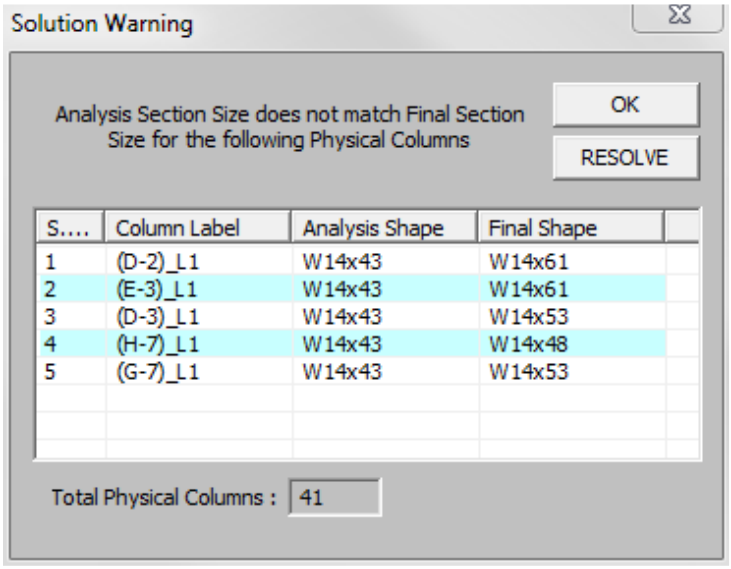
Solving and Reviewing Results

Note: Results produced by your model may not exactly match the results shown in this tutorial. This may be caused by a variety of factors, including how the model was created and which version of the program is being used.

Now that you have applied your loads and generated your load combinations, you can solve for results. To solve the model:

- On the Main menu, click Solve – Design Members/Walls/Slabs.

When the solution finishes, we are presented with the following dialog:

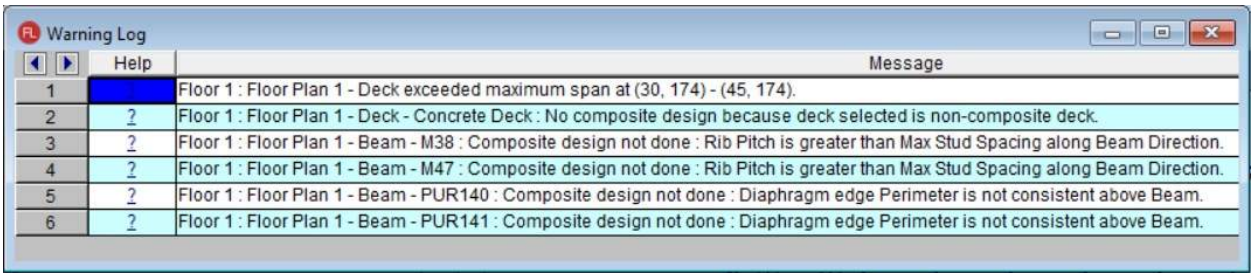


S....	Column Label	Analysis Shape	Final Shape
1	(D-2)_L1	W14x43	W14x61
2	(E-3)_L1	W14x43	W14x61
3	(D-3)_L1	W14x43	W14x53
4	(H-7)_L1	W14x43	W14x48
5	(G-7)_L1	W14x43	W14x53

This Warning is common in RISAFloor. When the model is solved, it is common for the original "required" size of a column to be insufficient at the end of the solution when the other members (beams and walls) have been optimized and re-sized.

- Click **RESOLVE** to replace the upper columns and resolve with the larger sizes.

When the solution is finished, the chosen member sizes are displayed on the plot (or, if you are in rendered mode, the model is rendered with the sizes) and the solution **Warning Log** is displayed.

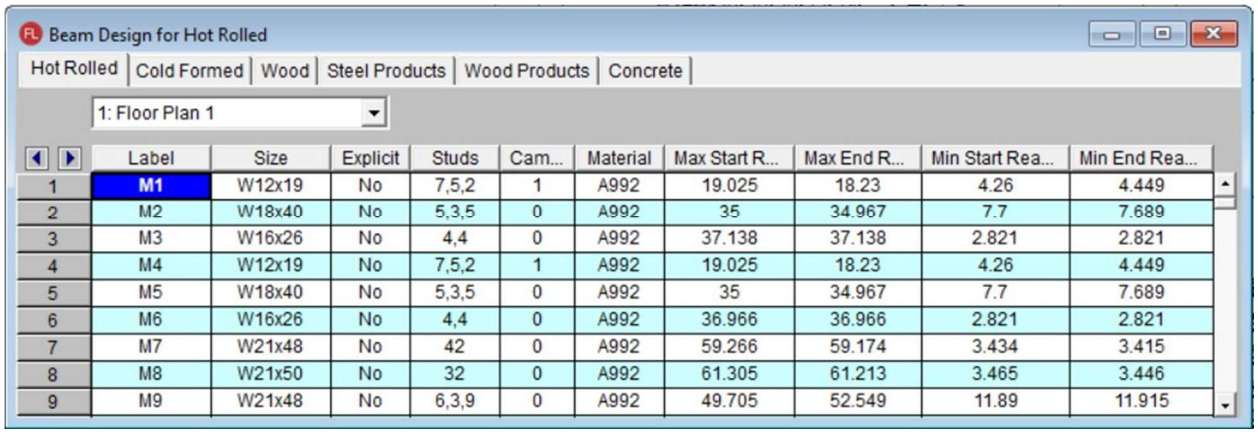


	Help	Message
1		Floor 1 : Floor Plan 1 - Deck exceeded maximum span at (30, 174) - (45, 174).
2	?	Floor 1 : Floor Plan 1 - Deck - Concrete Deck : No composite design because deck selected is non-composite deck.
3	?	Floor 1 : Floor Plan 1 - Beam - M38 : Composite design not done : Rib Pitch is greater than Max Stud Spacing along Beam Direction.
4	?	Floor 1 : Floor Plan 1 - Beam - M47 : Composite design not done : Rib Pitch is greater than Max Stud Spacing along Beam Direction.
5	?	Floor 1 : Floor Plan 1 - Beam - PUR140 : Composite design not done : Diaphragm edge Perimeter is not consistent above Beam.
6	?	Floor 1 : Floor Plan 1 - Beam - PUR141 : Composite design not done : Diaphragm edge Perimeter is not consistent above Beam.

Note: The purpose of the **Warning Log** is to inform you whenever a modeling issue or error arises which affects the design of the floor. For example, the first entry is informing you that you have exceeded the maximum deck span, including the actual coordinates where the deck span violation occurred. This will be helpful later when optimizing your design.

For now, open the **Design Results** spreadsheet:

- On the **Main** menu, click **Results** and select **Designs**.



	Label	Size	Explicit	Studs	Cam...	Material	Max Start R...	Max End R...	Min Start Rea...	Min End Rea...
1	M1	W12x19	No	7,5,2	1	A992	19.025	18.23	4.26	4.449
2	M2	W18x40	No	5,3,5	0	A992	35	34.967	7.7	7.689
3	M3	W16x26	No	4,4	0	A992	37.138	37.138	2.821	2.821
4	M4	W12x19	No	7,5,2	1	A992	19.025	18.23	4.26	4.449
5	M5	W18x40	No	5,3,5	0	A992	35	34.967	7.7	7.689
6	M6	W16x26	No	4,4	0	A992	36.966	36.966	2.821	2.821
7	M7	W21x48	No	42	0	A992	59.266	59.174	3.434	3.415
8	M8	W21x50	No	32	0	A992	61.305	61.213	3.465	3.446
9	M9	W21x48	No	6,3,9	0	A992	49.705	52.549	11.89	11.915

The **Design Results** spreadsheet summarizes all members across all material types (each material type is contained in tabs at the top). The **Hot Rolled** tab lists a summary of the member design complete with the number of studs, camber, and end reactions. Similar information would be found under the **Cold Formed** , **Wood**, or **Concrete** tabs if any such members were contained in this structure. Similarly, design information for Steel and Wood Joists, if used in the model, would be found under the **Steel Products** or **Wood Products** tabs.

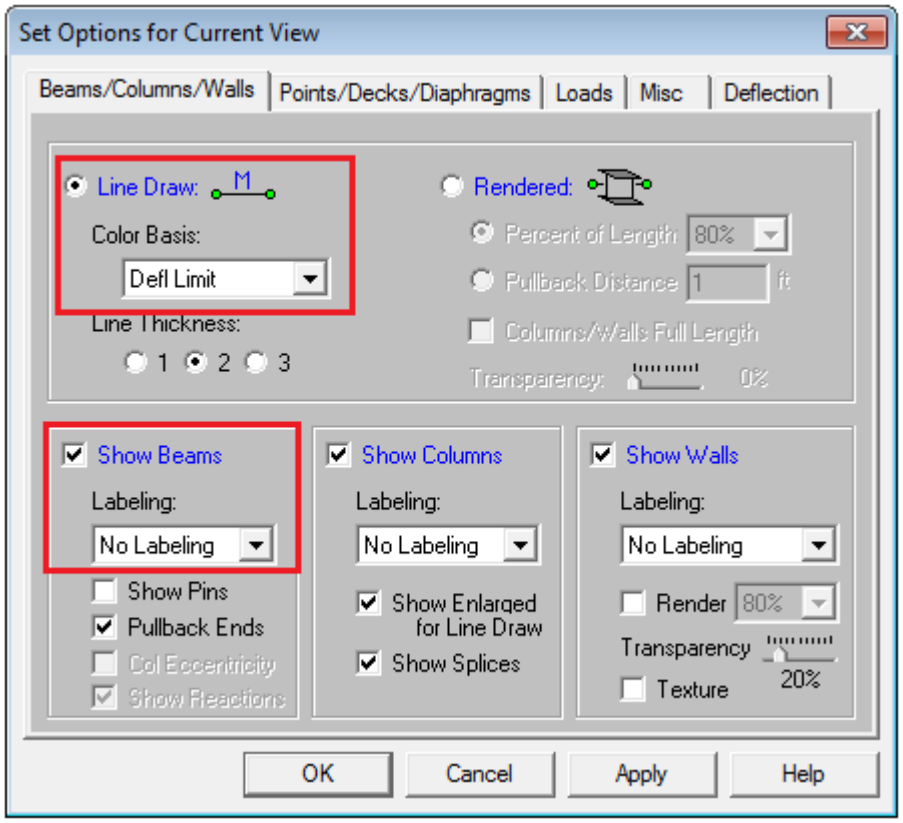
You may open the other spreadsheets to review the other types of results.

- On the **Main** menu, click **Results**, select **Code Checks**, **Shear**, **Bending**, etc.

The **Code Check** spreadsheet gives you a quick overview of the checks for bending, deflection, and shear. There are also individual results spreadsheets that give detailed information for the **Bending**, **Deflection**, **Shear**, and **Vibration** checks. The **Reactions** spreadsheet gives the member end reactions for the governing load categories.

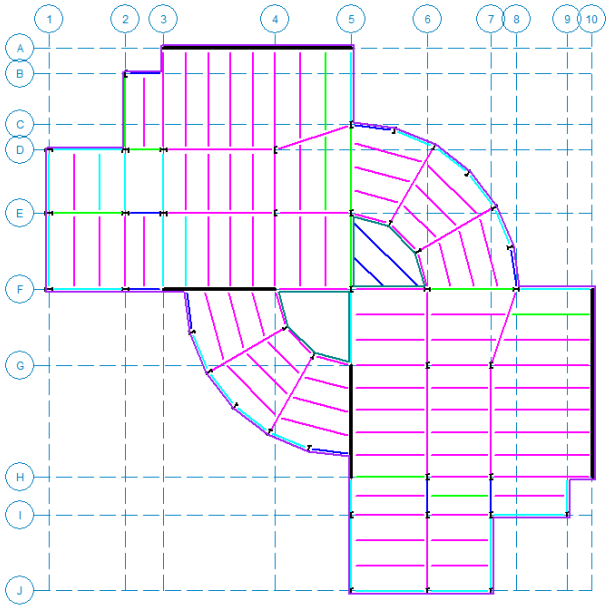
Return to your model view to explore the graphic result viewing options:

- Close any open spreadsheets.
- On the **Main** menu, click **View**, and select **Model Display Options**.
- Click the Beam/Columns/Walls tab. Under Line Draw, in the Color Basis list, select Defl Limit.
- Under **Show Beams** (the check box should already be selected), in the **Labeling** list, select **No Labeling**.



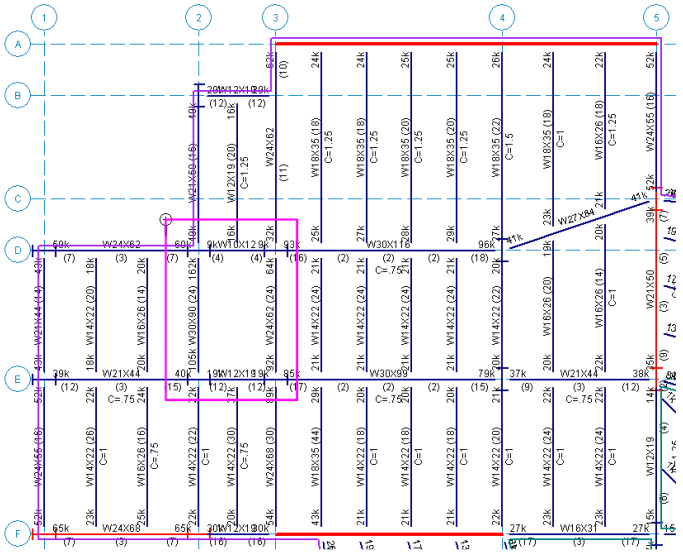
- Click **Apply**.

Your plot should now look like this:



This image gives a quick graphical representation of the adequacy of each beam for the deflection limits specified in the **Design Rules** spreadsheet. You can create a similar plot to display the Bending or Shear Checks. You can also plot the Vibration Frequencies and Accelerations to check your entire floor system for the AISC Vibration criteria.

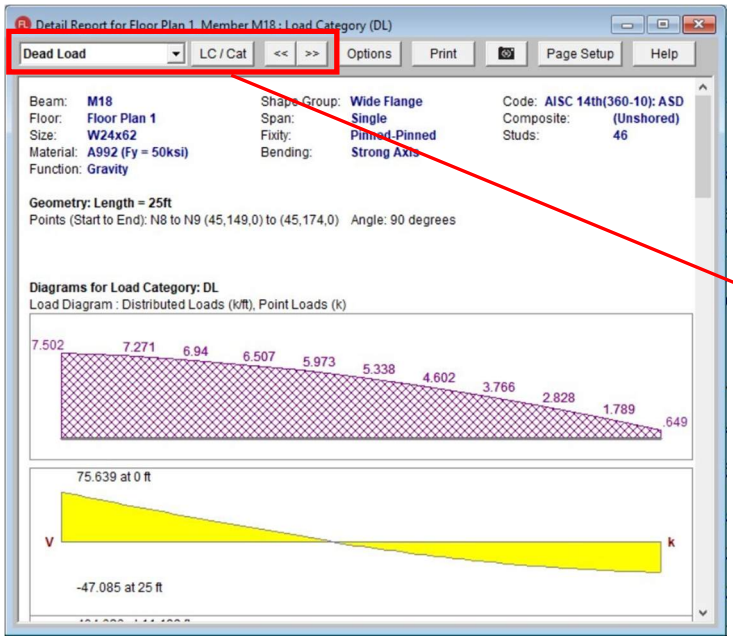
- Click **Cancel** to exit the dialog box and return to your model view.
- Click the **Box Zoom** button  and select the portion of the model where the most obvious deck span violation occurred, as shown in the figure below:



- On the Selection toolbar (left side), click the **Detail** button  .

Note: Your cursor will change to  .

- Click any one of the members that surround the gap in your framing (shown boxed in the previous image). This will display the member detail report for that member.
- Click on a few other members and review the detail reports for those members.



This detail report is shown for Dead Load results. To change categories, use the list at the upper left.

Use the **LC/Cat** button to switch between load categories and load combinations.

Use the arrow buttons to view the next or previous member.

The detail report displays loading information, and shear, moment, and deflection diagrams. It also gives design data, composite member properties, and the calculated allowable stresses. Since the model had a triangular dead load that was angled to the deck, it produced a parabolic load on the beam M18 (shown above). RISAFloor can accurately approximate this load pattern by modeling a series of partial length trapezoidal loads.

This type of loading is common whenever you have partial length area loads, or tapered line loads that do not run orthogonal to the deck span or beam framing. Since it can result in significant increases in a member's maximum moment and end reactions, it is important to provide an accurate representation of this load.

- Click **Close** to return to the model view before proceeding to the next section.

Modifying the Model

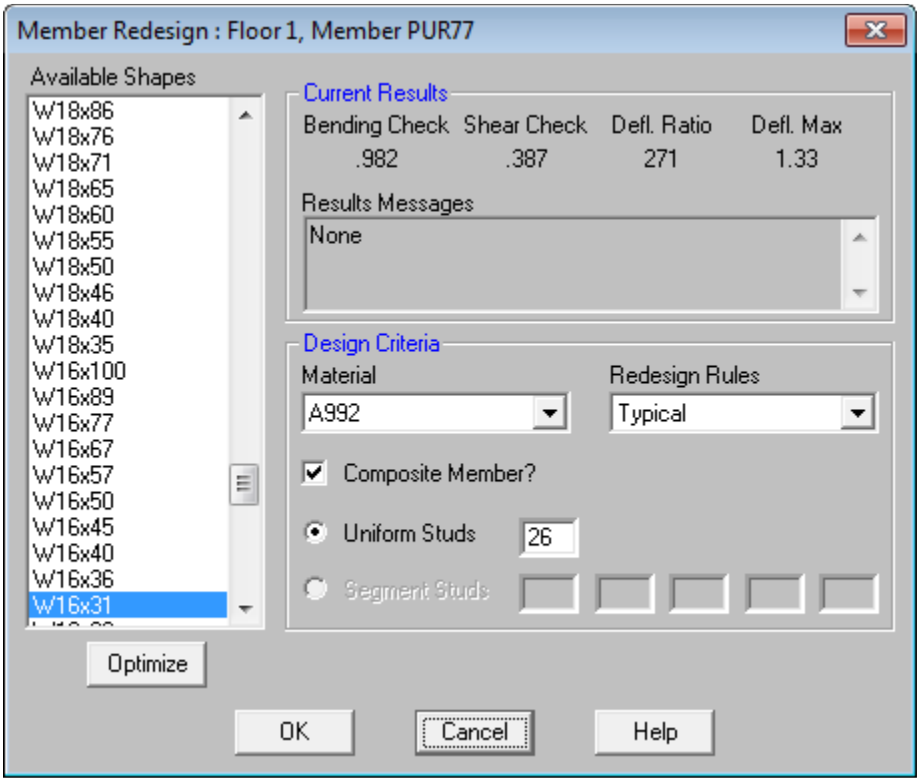
Now that you have reviewed all the data, it is time to make some changes to your model. Modifying the model may be done graphically or within the spreadsheets. Although this tutorial will concentrate on the graphical methods, remember that you may view and edit most data from within the spreadsheets also.

To perform an interactive redesign on any member:

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

, and then click on any member.

This will open the **Member Redesign** dialog box, as shown below. You can scroll through the available shapes and see how the bending, shear, and deflection are affected. To modify the selected beam, simply click **OK**. This interactive redesign is powerful because you are free to manipulate your model without erasing all of your design results. Keep in mind that when you are finished with all of your modifications, be sure to re-run your analysis to recalculate changes in self weight and relative floor stiffness.



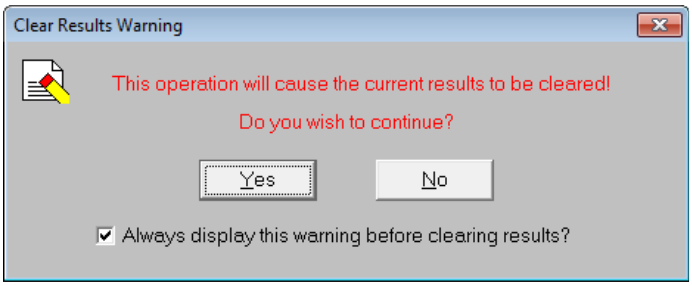
- Click **Cancel** to close the dialog box.

You can now make a framing change to your model.

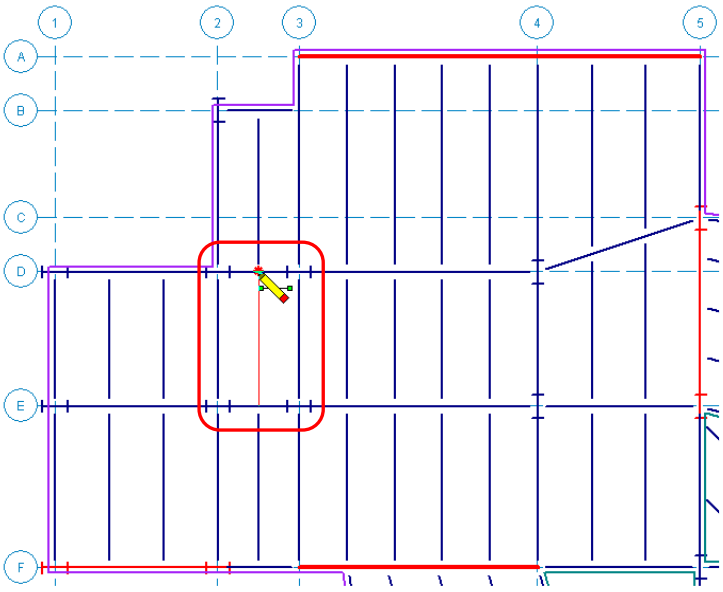
- On the Window toolbar, click the **Graphic Editing** button  to activate the Drawing toolbar if it is not already visible.
- On the Drawing toolbar, click **Draw Beams**  .

This is the same button that you initially used to draw the beams.

When you click on this button, a warning box will appear informing you that continuing with the operation will clear your design results.



- Since you are finished browsing the results, select **Yes** to continue.
- Click the **Draw Beams** tab. Under **Function** click **Gravity**, and then click **Apply**.
- Draw in the member, as shown circled below, by clicking on the two end points. Right-click to release the mouse.



Before you end, check the spreadsheet warning log:

- On the **Main** menu, click **Spreadsheets**, select **Warning Log**.

Notice that even though you have cleared your results, RISAFloor maintains the **Warning Log** to assist you when modifying your model.

This is the end of Tutorial 5.

You may save your model to be used as the starting point for the next tutorial, or begin the next tutorial using the .rfl starter file in the RISAFloor **Tutorials** folder. To save the model:

On the **File** menu, click **Save As** and enter a file name.

Note: If you are running the Demonstration version, the file is now too large to save. You will need to continue directly with the remaining tutorials or use the Starter Files for the following tutorials.

Part A: Tutorial 6 – RISA-3D Integration

This tutorial is intended to introduce you to the features of RISA-3D that directly relate to its interaction with RISAFloor. For more detailed information on RISA-3D, refer to the *RISA-3D User's Guide* and *RISA-3D General Reference*.

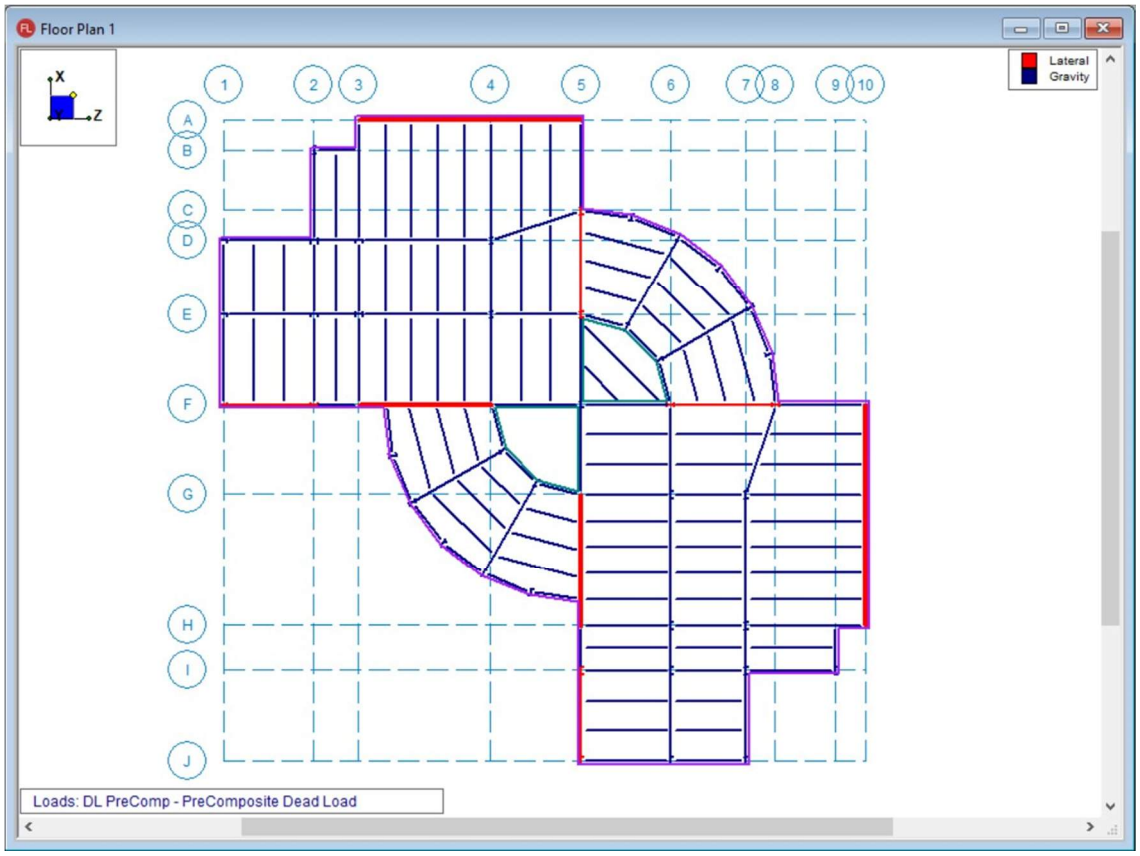
If you are continuing from the previous tutorial:

- From the **Window** menu, select **Single View**.
- On the Window toolbar, click the **Graphic Editing** button to activate the Drawing toolbar.
- On the **Data Entry** toolbar, click **Close** to close it. Skip ahead to below.

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

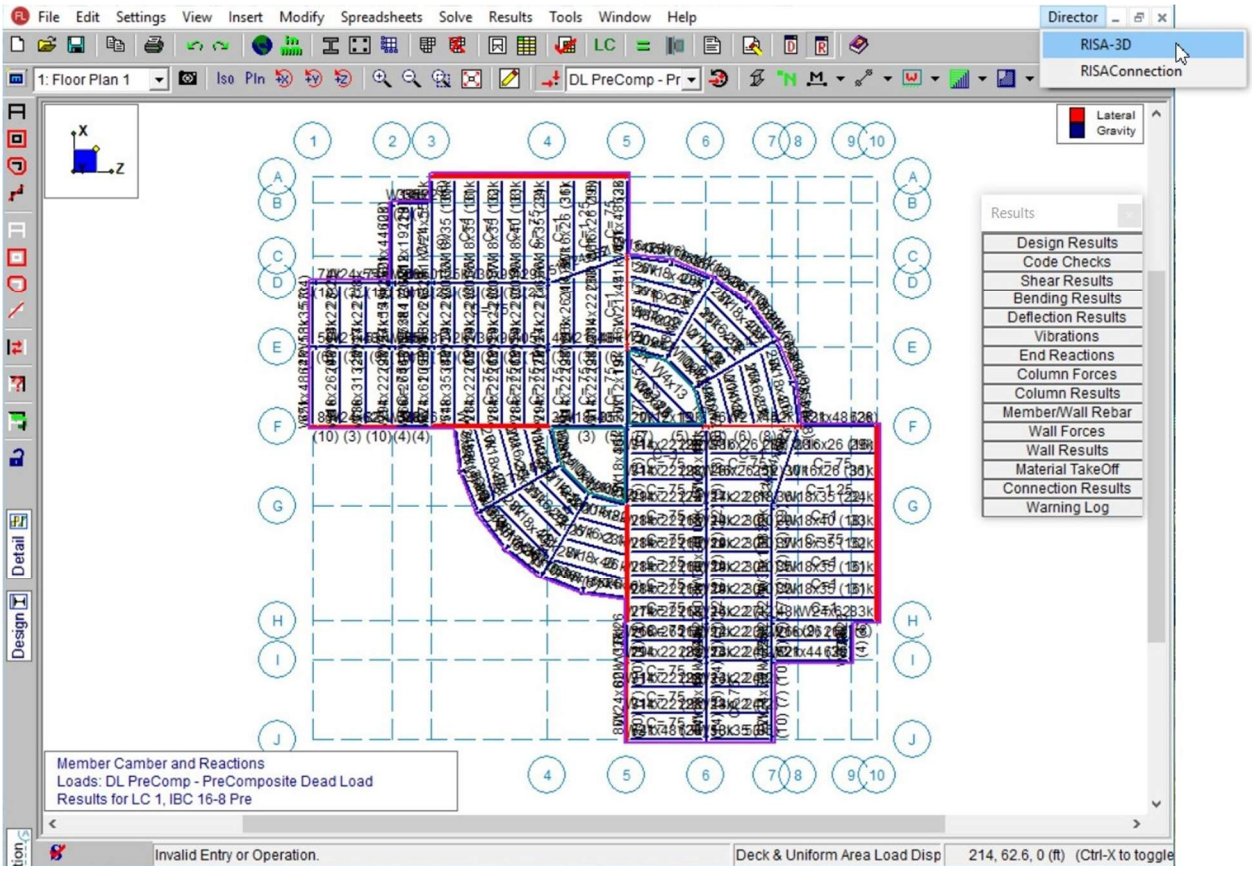
- Double-click on the **RISAFloor** icon to start the program.
- Click **Open Model**. Double-click the **Tutorials** folder, select **Tutorial A6 Starter.rfl** and click **Open**. Click **Close** (or **Cancel**) to exit the **Model Settings** dialog box.
- On the Window toolbar, click the **Graphic Editing** button to activate the Drawing toolbar.

Your model will look like the following:



To see the interaction between RISAFloor and RISA-3D, solve your RISAFloor model, and then use RISA-3D to analyze and design your lateral system.

- On the RISA toolbar, solve the model by clicking **Solve**  .
- Click **OK** from the **Solution Warning** dialog.
- On the **Main** menu, click the **Director** button (top right) and select **RISA-3D**.

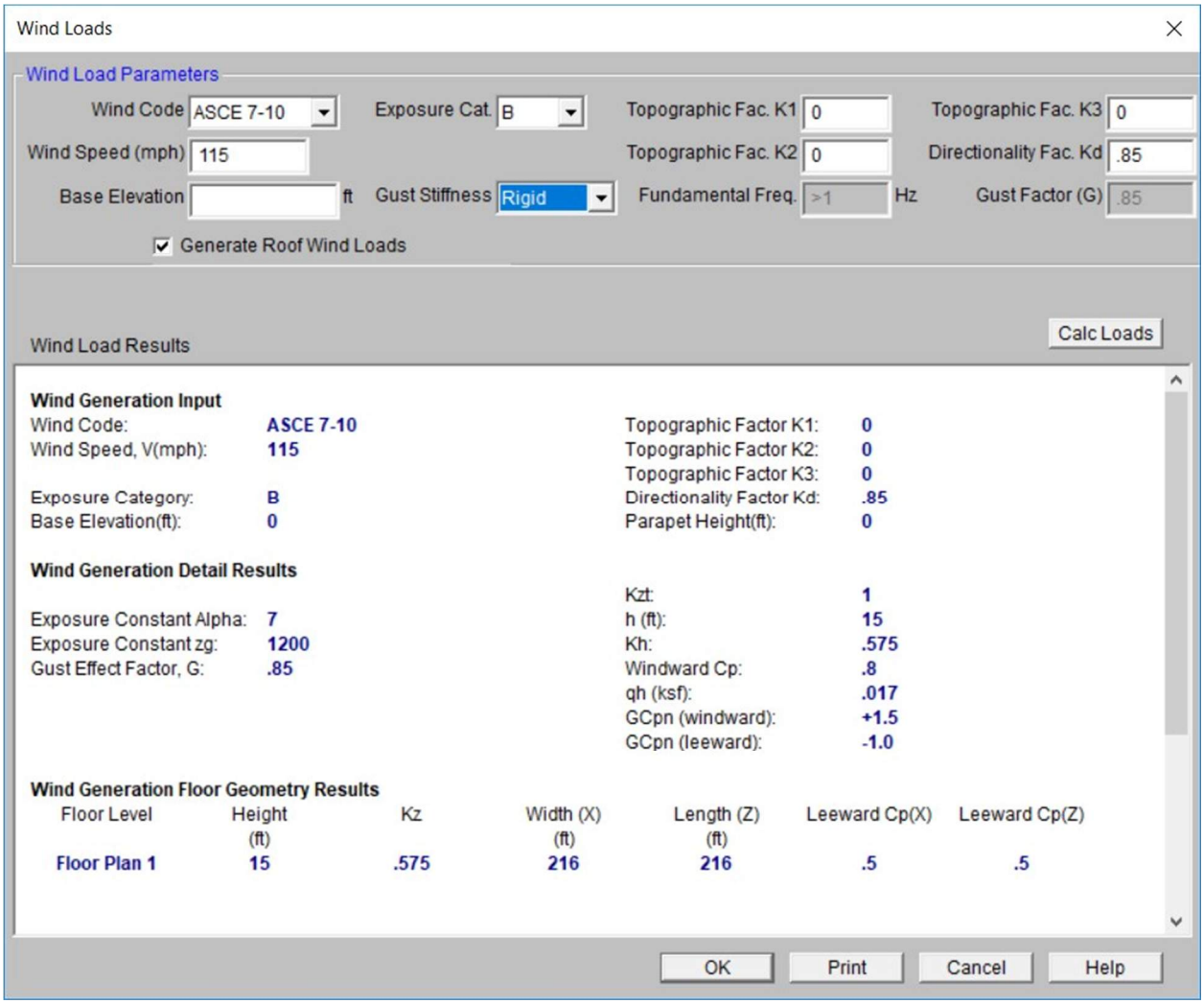


RISA-3D Lateral Load Generators

After you select RISA-3D, you will be prompted for information on the wind and seismic loads of your structure.

First, the **Wind Loads** dialog box will display.

- Make sure your parameters match those in the image below:



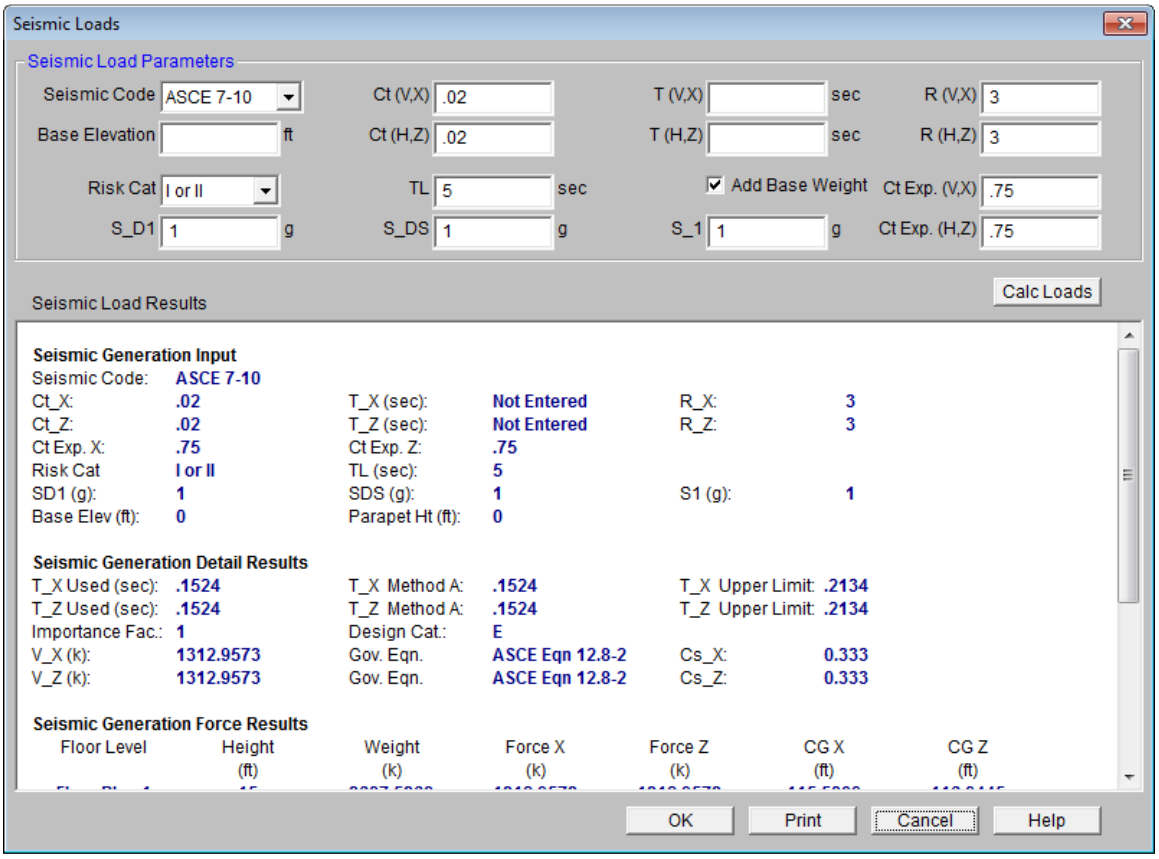
Review the information shown in the **Wind Loads** dialog box. In the future, if you make any changes in the **Wind Load Parameters** area, be sure to click **Calc Loads** to recalculate.

Be sure to also check the information generated in the **Wind Load Results** area. This is a summary of the wind loads that have been calculated for you.

- Click **OK** to accept the loads.

After the **Wind Loads** dialog box closes, the **Seismic Loads** dialog box will display.

- Make sure your parameters match those in the image below:



Review the information shown in the **Seismic Loads** dialog box. In the future, if you make any changes in the **Seismic Load Parameters** area, be sure to click **Calc Loads** to recalculate.

Be sure to also check the information generated in the **Seismic Load Results** area. This gives the equivalent static force distribution and base shear that have been calculated for you.

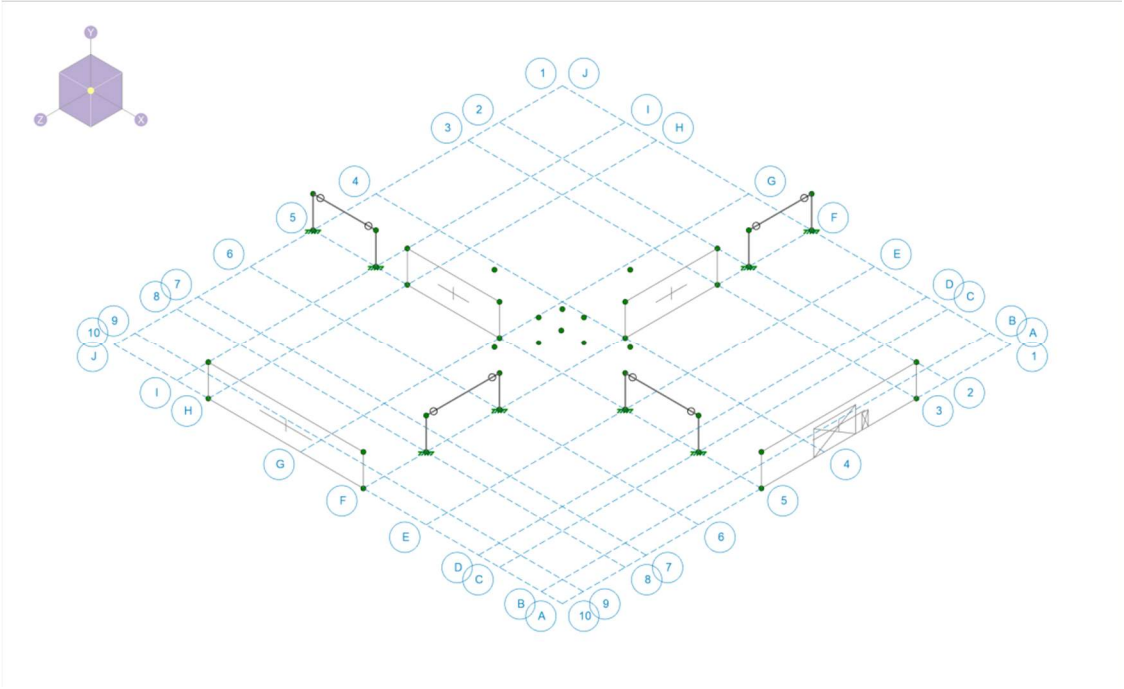
- Click **OK** to accept the loads.
- If the **Model Settings** dialog box displays, close it by clicking **OK** or **Cancel**.

RISA-3D will now open and display an isometric view of your model. You are now working completely in RISA-3D. The interface of RISA-3D is different from RISAFloor, but menus and buttons work the same way.

Before doing any graphical editing, turn off the display of the load:

- On the Window toolbar, click the **Show Joint Labels** button  to turn off the display of the Joint Labels.

Your screen should now look like this:



- On the **Data Entry** toolbar (at right), select **Basic Load Cases**. You can now review the load data that was generated.

As you review the data in this spreadsheet, notice that all your basic load cases have been created for you, including wind and seismic load cases for each direction.

Basic Load Cases

	BLC Description	Category	X Gravity	Y Gravity	Z Gravity	Nodal	Point	Distributed	Area(Member)	Surface(Plate/Wall)
1	Dead Load	DL		-1		18	70	53		
2	Live Load	LL				18	66	51		
3	Live Load Special (public assemb	LLS					12			
4	Roof Live Load	RLL								
5	Snow Load	SL								
6	Snow Load Nonshedding	SLN								
7	Rain Load	RL								
8	Wind Load X	WLX				1				
9	Partial X Wind Load 1	WLXP1				1				
10	Partial X Wind Load 2	WLXP2				1				
11	Wind Load Z	WLZ				1				
12	Partial Z Wind Load 1	WLZP1				1				
13	Partial Z Wind Load 2	WLZP2				1				
14	Earthquake Load X	ELX				1				
15	Earthquake Load X Plus Z Eccentr	ELX+Z				1				
16	Earthquake Load X Minus Z Eccent	ELX-Z				1				
17	Earthquake Load Z	ELZ				1				
18	Earthquake Load Z Plus X Eccentr	ELZ+X				1				
19	Earthquake Load Z Minus X Eccent	ELZ-X				1				
20	Other Load 1	OL1								
21	Other Load 2	OL2								
22	Other Load 3	OL3								
23	Other Load 4	OL4								
24	Wind Load Roof +X	WLX+R								
25	Wind Load Roof -X	WLX-R								
26	Wind Load Roof +Z	WLZ+R								
27	Wind Load Roof -Z	WLZ-R								
28		None								

- When you are finished reviewing the data, click **Close** to close the spreadsheet and return to your model view.

Diaphragms are also created in RISA-3D based on your RISAFloor geometry:

- On the **Data Entry** toolbar (at right), click **Diaphragms**.

Floor Diaphragms

General | Wood Diaphragms

	Elevation [ft]	Mass [k]	Mass MOI [k-ft²]	Center Of Mass [ft]	Plus X E...	Minus...	Plus Z E...	Minus...	Inactive	Diaphragm	Type	Region	Design Rule	SGAF	Material	Thickness [in]	No Wind/Drift
1	15	3676.823	2.853e+7	115.564, 116.07	%5	%5	%5	%5	☐	D1	Rigid		N/A	N/A	N/A	N/A	

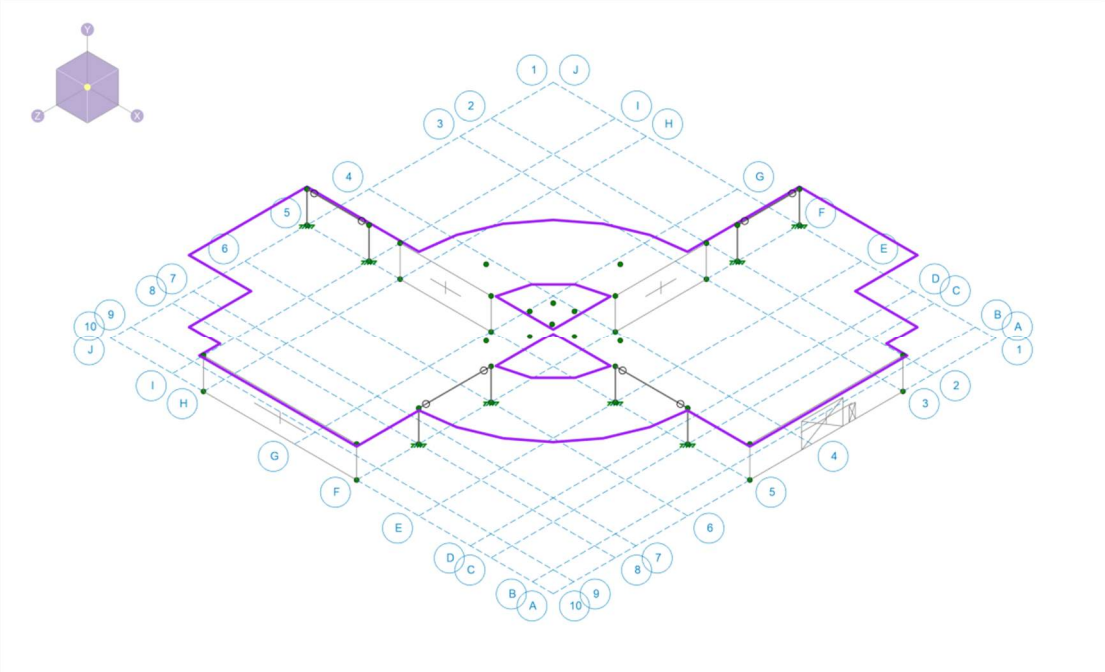
Notice that the diaphragms already contain the diaphragm mass and mass moment of inertia information that you would need for a dynamic analysis. In this spreadsheet, you can specify the percent of width you want RISA-3D to use to account for the accidental eccentricity in the X or Z directions.

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

- On the Window toolbar, click on the **Diaphragm Display** button  (it may already be displayed).

This will display the RISAFloor diaphragms which have been automatically created for you.

Notice that only the beams, columns, and wall panels that were specified as LATERAL members were generated in your RISA-3D model. You can always add vertical braces, modify your wall panels, or manipulate your model as you wish.

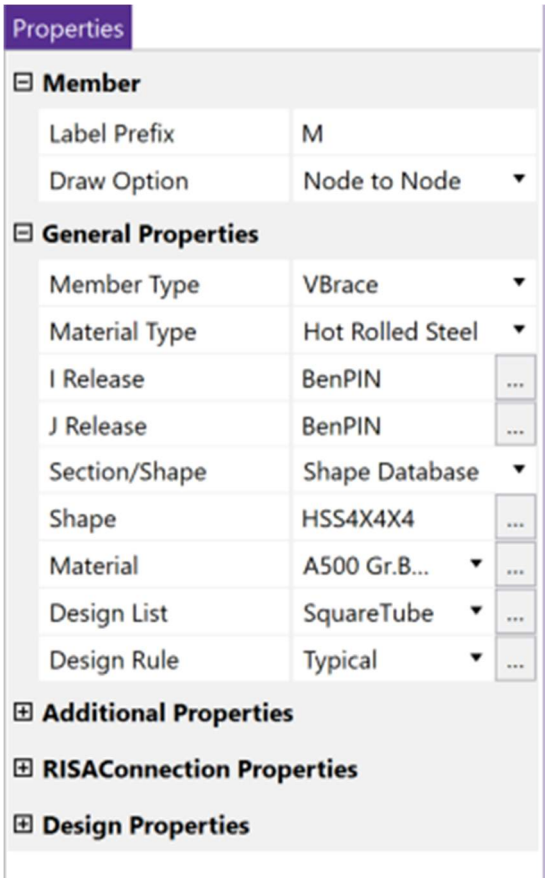


Next, add some X bracing, as follows:

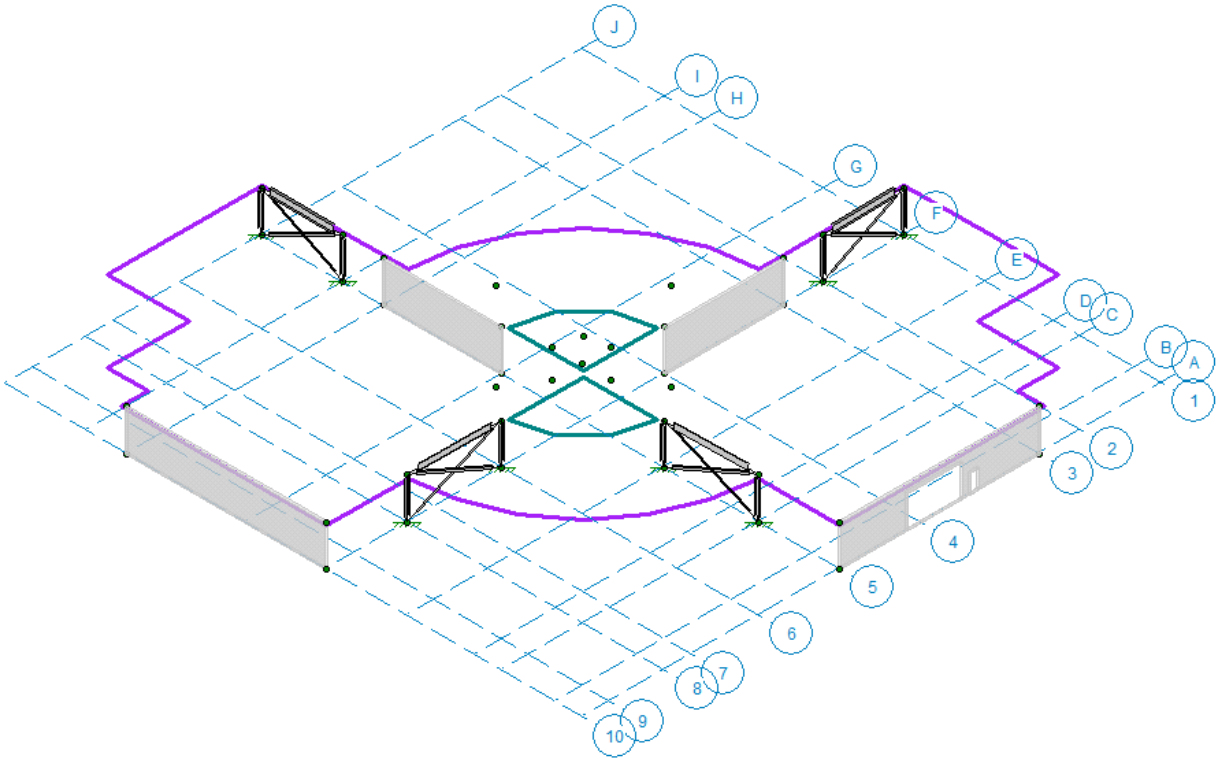
- On the **Home** menu, click the **Draw Members** button 

. Notice that the **Properties Panel** (at left) changes.

- Select **VBrace** for **Member Type**, **Hot Rolled Steel** for **Material Type**, **A500 Gr. B Rect**. for **Material**, and **SquareTube** for **Design List**.
- Select **Shape Database** for **Section/Shape**. Click the ellipsis button ··· next to **Shape**.
- Make sure you are on the **Hot Rolled** tab. Under **Shape Type**, select **Tube** and under **Shape Name**, type **HSS4x4x4**.
- Click **OK** once to close both dialogs.
- Click the ellipsis button ··· next to **I Release**. Select **Pinned (Torsion Fixed)** for both **I End Release Codes and J End Release Codes**. Click **OK**.



- Draw in the X braces (as shown in the image below) by clicking the nodes on each end of each member. Right-click to release the mouse when needed.



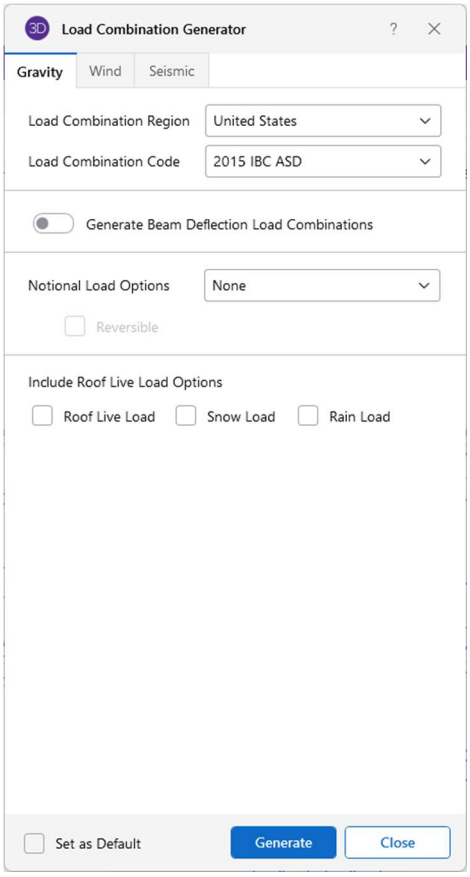
RISA-3D Load Combinations

Next, add the common load combinations used for this type of model, including all the code required eccentric loadings for wind and seismic loads. This will demonstrate how the program automatically generates these load combinations; saving you the time and energy normally spent entering them manually.

- On the **Data Entry** toolbar, click on **Load Combinations** to open the **Load Combinations** spreadsheet.

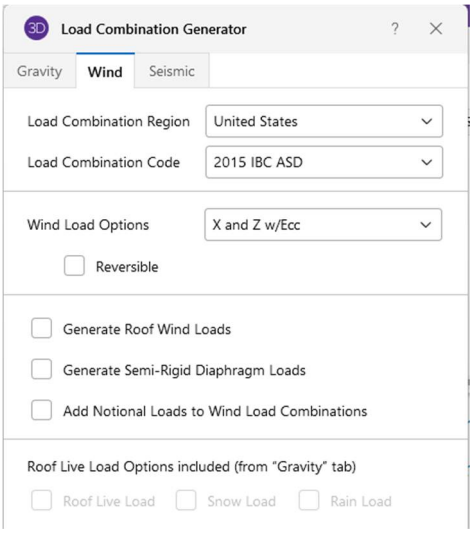
Although you previously generated load combinations for your RISAFloor model, you will now generate your lateral load combinations for use in the RISA-3D model.

- Click the LC Generator button within the spreadsheet to display the following dialog.

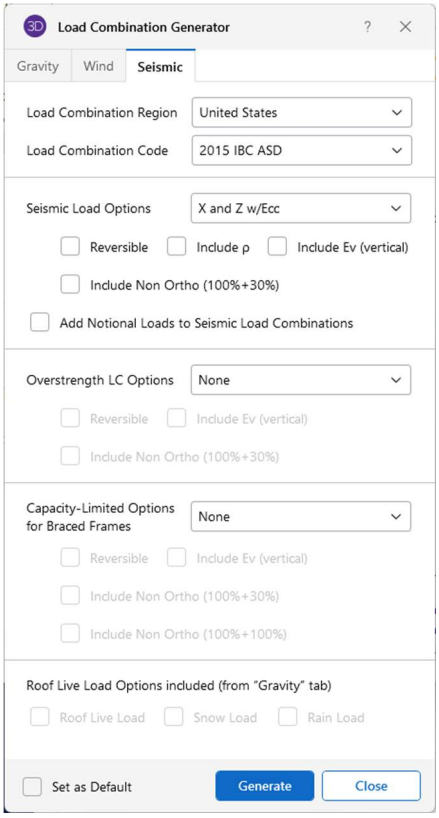


- On the first (**Gravity**) tab, in the **LC Region** list, select **United States**.
- On the first (**Gravity**) tab, in the **LC Code** list, select **2015 IBC ASD**.
- Toggle off the option to Generate Beam Deflection Load Combinations.
- Uncheck RLL (Roof Live Load), SL (Snow Load), and RL (Rain Load).
- Uncheck Generate Deflection LCs
- Click Generate.

- Click on the **Wind** tab.
- Under **Wind Load Options**, click **X and Z w/Ecc** (make sure the **Reversible** check box is cleared).

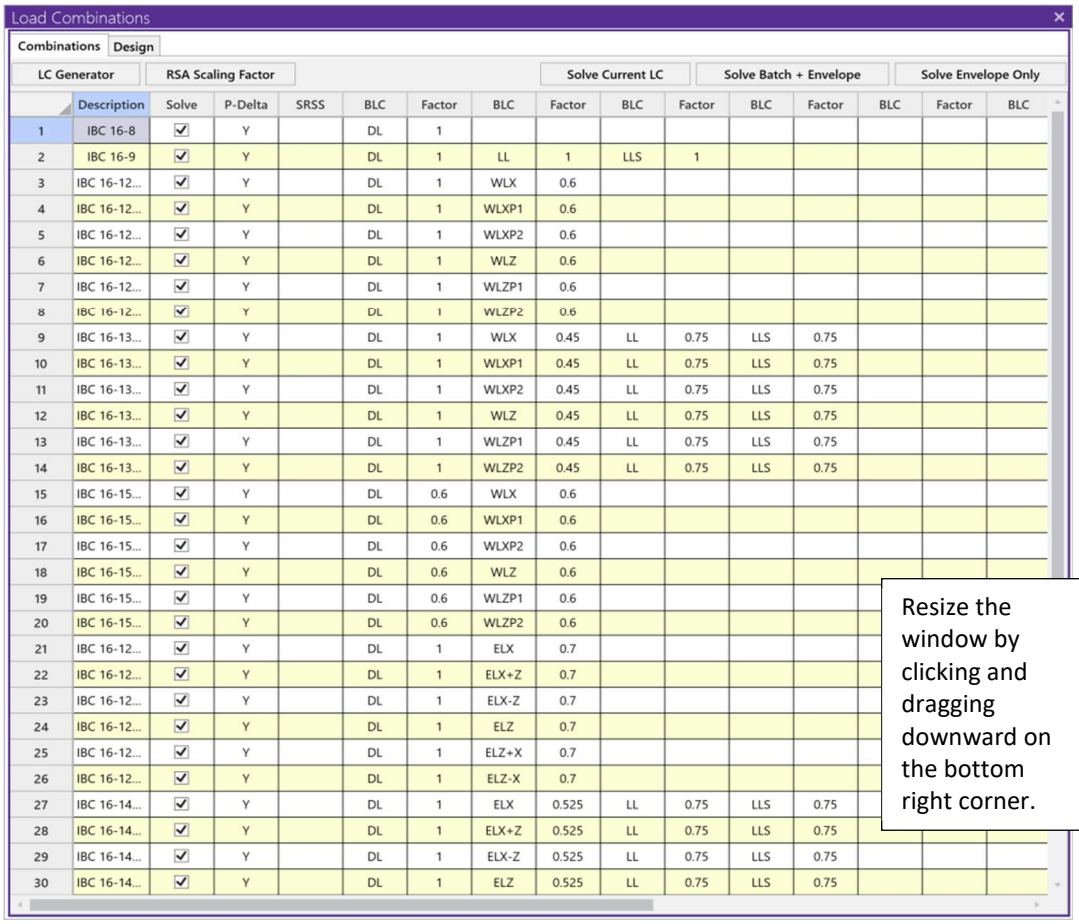


- Click Generate.
- Click on the **Seismic** tab.
- Under **Seismic Load Options**, click **X and Z w/Ecc** (make sure the **Reversible** check box is cleared).



- Click **Generate**. Then click **Close** to close the dialog.

This will generate the load combinations shown below.



	Description	Solve	P-Delta	SRSS	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC
1	IBC 16-8	☑	Y		DL	1									
2	IBC 16-9	☑	Y		DL	1	LL	1	LLS	1					
3	IBC 16-12...	☑	Y		DL	1	WLX	0.6							
4	IBC 16-12...	☑	Y		DL	1	WLXP1	0.6							
5	IBC 16-12...	☑	Y		DL	1	WLXP2	0.6							
6	IBC 16-12...	☑	Y		DL	1	WLZ	0.6							
7	IBC 16-12...	☑	Y		DL	1	WLZP1	0.6							
8	IBC 16-12...	☑	Y		DL	1	WLZP2	0.6							
9	IBC 16-13...	☑	Y		DL	1	WLX	0.45	LL	0.75	LLS	0.75			
10	IBC 16-13...	☑	Y		DL	1	WLXP1	0.45	LL	0.75	LLS	0.75			
11	IBC 16-13...	☑	Y		DL	1	WLXP2	0.45	LL	0.75	LLS	0.75			
12	IBC 16-13...	☑	Y		DL	1	WLZ	0.45	LL	0.75	LLS	0.75			
13	IBC 16-13...	☑	Y		DL	1	WLZP1	0.45	LL	0.75	LLS	0.75			
14	IBC 16-13...	☑	Y		DL	1	WLZP2	0.45	LL	0.75	LLS	0.75			
15	IBC 16-15...	☑	Y		DL	0.6	WLX	0.6							
16	IBC 16-15...	☑	Y		DL	0.6	WLXP1	0.6							
17	IBC 16-15...	☑	Y		DL	0.6	WLXP2	0.6							
18	IBC 16-15...	☑	Y		DL	0.6	WLZ	0.6							
19	IBC 16-15...	☑	Y		DL	0.6	WLZP1	0.6							
20	IBC 16-15...	☑	Y		DL	0.6	WLZP2	0.6							
21	IBC 16-12...	☑	Y		DL	1	ELX	0.7							
22	IBC 16-12...	☑	Y		DL	1	ELX+Z	0.7							
23	IBC 16-12...	☑	Y		DL	1	ELX-Z	0.7							
24	IBC 16-12...	☑	Y		DL	1	ELZ	0.7							
25	IBC 16-12...	☑	Y		DL	1	ELZ+X	0.7							
26	IBC 16-12...	☑	Y		DL	1	ELZ-X	0.7							
27	IBC 16-14...	☑	Y		DL	1	ELX	0.525	LL	0.75	LLS	0.75			
28	IBC 16-14...	☑	Y		DL	1	ELX+Z	0.525	LL	0.75	LLS	0.75			
29	IBC 16-14...	☑	Y		DL	1	ELX-Z	0.525	LL	0.75	LLS	0.75			
30	IBC 16-14...	☑	Y		DL	1	ELZ	0.525	LL	0.75	LLS	0.75			

Why so many load combinations? Each code load combination that includes an earthquake or wind load becomes six load combinations when you include all the possible eccentric cases (if you had selected the Reversible option, this would have expanded to 12 cases).

The **LC Generator** feature allows complete control over whether or not to include these more detailed load combinations. Load combinations or categories that are not relevant to your region can be completely eliminated by modifying the provided .xml spreadsheets.

For example, if you are in a year-round sunny climate like south Florida, do you really need to check your structure for all the combinations that include snow load? Therefore, to remove the snow loads from your generated load combinations, locate and modify the spreadsheet. The **United States** spreadsheet is located in **C:\RISA User Data\%Username%\Load Combinations** and is named **United States.xml**.

Note: Before editing this file, you may want to copy the original in case you want to restore all the settings back to the default.

As you can see from this tutorial, the interaction between RISAFloor and RISA-3D is completely seamless. However, this tutorial is only an introduction to the many features of RISA-3D. We recommend downloading the *RISA-3D User's Guide* for tutorials demonstrating the features of this powerful program, including solution and reviewing results.

Another option is that you can run your model in RISA-3D to resize your beams and columns. Then, return to RISAFloor to see how all those changes have been incorporated in the RISAFloor model for gravity load analysis.

This is the end of Tutorial 6.

You may save your model if you choose, but it is not necessary for the next tutorial. If you choose to save your model:

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

Note: You will notice that this file saves as a RISAFloor (.rfl) model, even though we have now been working in RISA-3D. This is because the file contains RISAFloor data that would be lost if the RISA-3D file were detached. To reopen the file in the future, you will need to open it in RISAFloor, solve, and use the **Director** tool to bring it back into RISA-3D.

Part A: Tutorial 7 – RISA Interoperability

RISAFloor has several options for importing or exporting projects to and from other industry software. This tutorial will introduce you to several of those options.

Note: This tutorial requires that you save your results in order to export them into other programs. Unfortunately it is not possible to save results with the Demonstration version of RISAFloor. However, if you would like to learn more about the export/import options, please see our website for a collection of detailed instructional videos and blogs which include tips and tricks for these features.

RISA-Revit Link

Because the RISA-Revit Link is a bi-directional exchange link, new models can be created in either RISAFloor or Revit and edited in either program.

Start by exploring the export options from RISAFloor to Revit. If you do not have the RISA-Revit Link, you may download it at no cost from our website. Here are instructions on how to install the link:

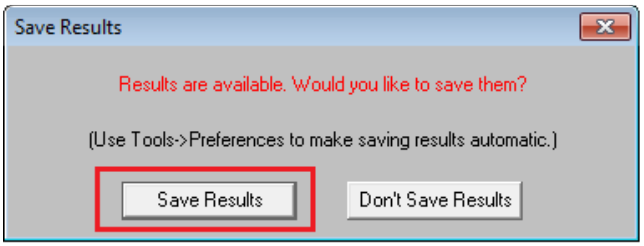
- Click **Close** to exit **RISAFloor** if you have the program open.
- Go to http://www.risa.com/partners/prt_revit.html, click **Downloads**, and then select the RISA-Revit Link version you want to download.
- Download and install the link.

Follow the steps below to load the RISAFloor starter file for this tutorial:

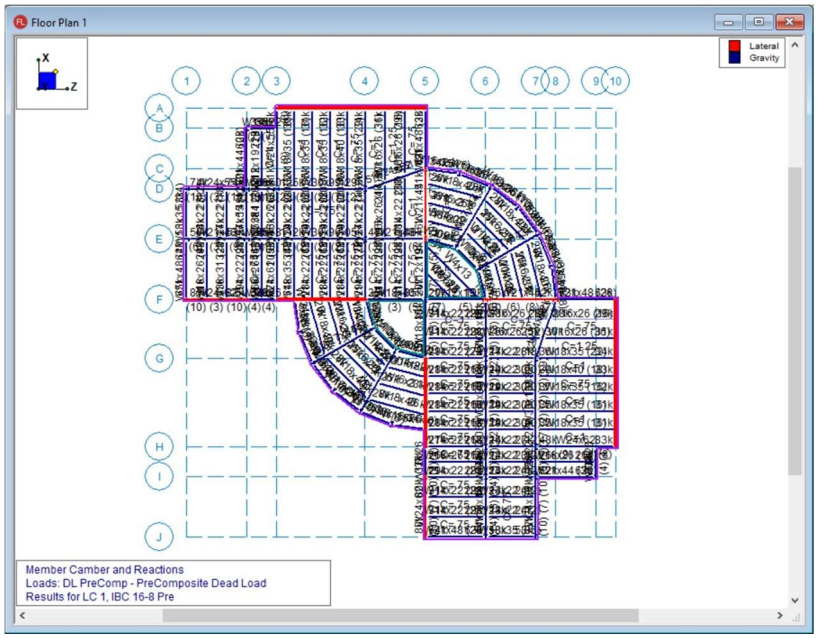
- Double-click on the **RISAFloor** icon to start the program.
- Click **Open Model**. Double-click the **Tutorials** folder, select **Tutorial A7 Starter.rfl** and click **Open**. Click **Close** (or **Cancel**) to exit the **Model Settings** dialog box.

Solve the model and save:

- Click **Solve** to solve the model. Click **OK** from the **Solution Warning** dialog.
- From the **File** menu, click **Save** and be sure to **Save Results**.



Your model should now display design data such as member sizes, end forces, and number of studs:



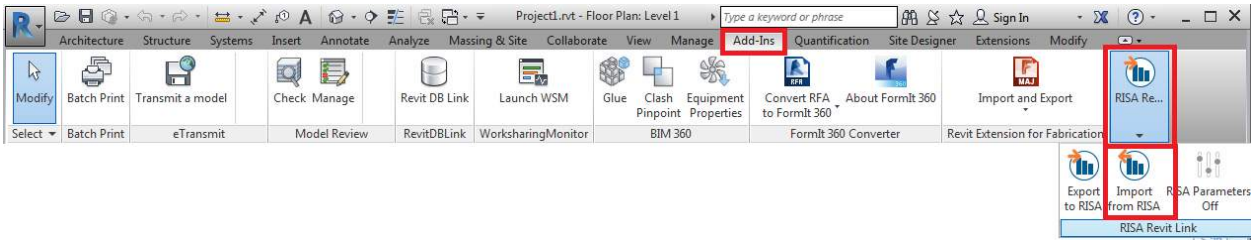
- There is no need to export anything. You simply need the RISAFloor input file (.rfl).
- Close RISAFloor by selecting **Exit** from the **File** menu on the Main menu.

You can now open Revit Structure and import this exchange file:

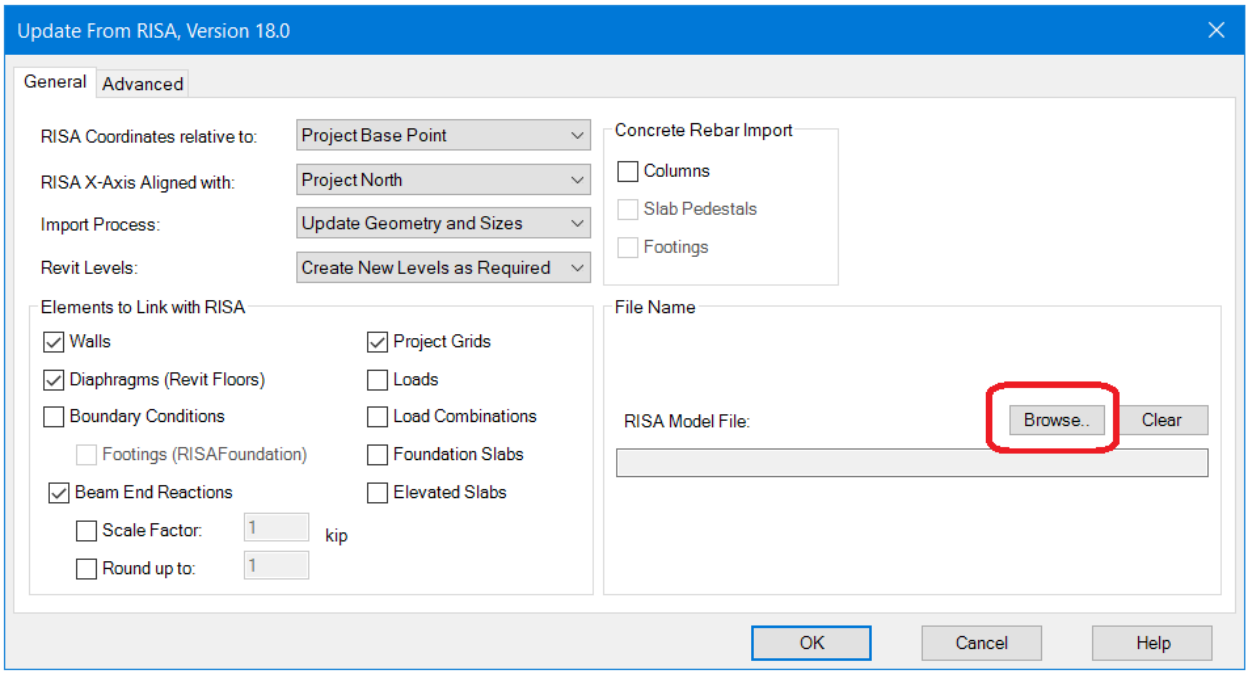
- Open a new file in **Revit Structure**.
- On the **Main** menu, **Save** the file as **TutorialA7.rvt** as the file name.

Note: This tutorial is based on the RISA interface with Revit Structure 2017. Please see our website for more information: http://www.risa.com/partners/prt_revit.html.

- Open a **3D Analytical** view.
- On the **Main** menu, click **Add-Ins**, and select **Import from RISA**.



- Click **Browse** to locate the Autodesk Revit (.exc) exchange file, as shown below.



Once in Revit, you can modify your model and then export back to RISAFloor. Because of its two-way functionality, you can modify the model in either program and the other will recognize the updates.

Note: This was just a basic description of the RISA-Revit Link interoperability. For more information, you may download documentation from the RISA website specific to the Link and how to use it.
Go to the RISA website: http://www.risa.com/partners/prt_revit.html.

CIS/2 Translator

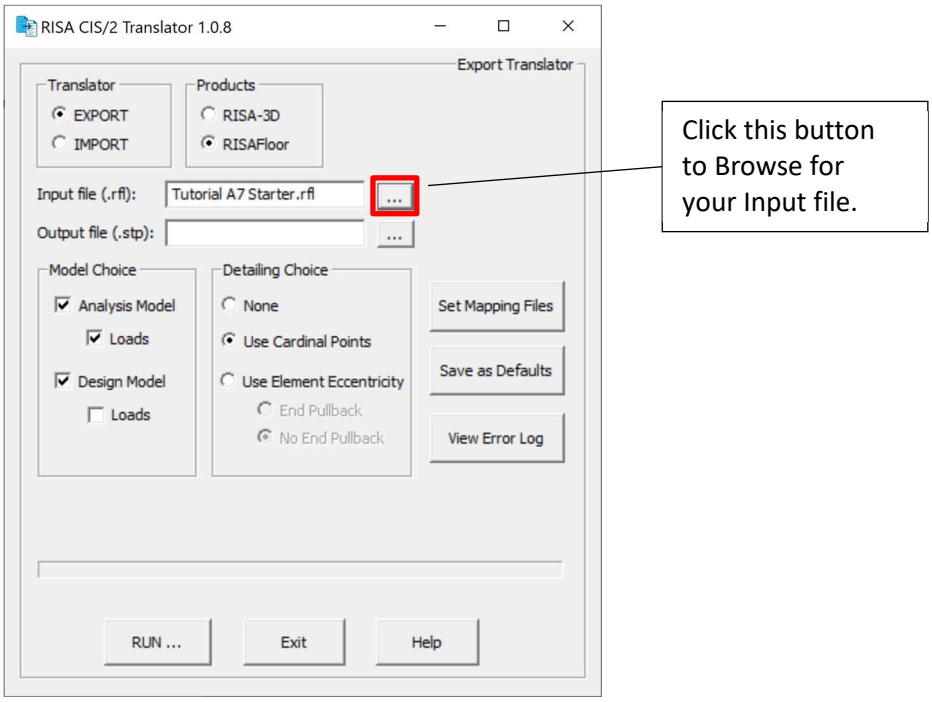
The RISA CIS/2 Translator is a tool for transferring data from RISA to the CIM steel part 21 file format. There are two basic modes for the CIS/2 Translator: export and import. Currently RISAFloor only exports data using the export mode. You may download this translator at no cost from the RISA website.

Here are instructions on how to install the translator:

- Click **Close** to exit **RISAFloor** if it is open.
- Go to https://portal.risa.com/login, to download the Translator.
- Install the Translator.

To open the RISA CIS/2 Translator:

- From the **Start Menu,** select **All Programs**, click **RISA**, and then click **RISA CIS2 Translator**. A dialog box will display.
- Select **RISAFloor** under **Products**. Then browse to select the **Input file**.



- Click **RUN**.
- The output **.stp** file is now available to import into your steel detailing software.

Note: This was a very basic description of the CIS/2 Translator. For more detailed information on the translator and how to use it, please see the *RISA CIS/2 Translator General Reference*, available for download from the RISA website: https://www.risa.com/documentation.

RISAFloor & CAD

RISAFloor offers a DXF import/export feature that provides two-way compatibility with any other program that can read and write DXF files--this includes most major CAD programs and many analysis programs. With this feature, RISAFloor is able to produce CAD-quality drawings that list your beam sizes, camber, stud layout, end reactions, etc.

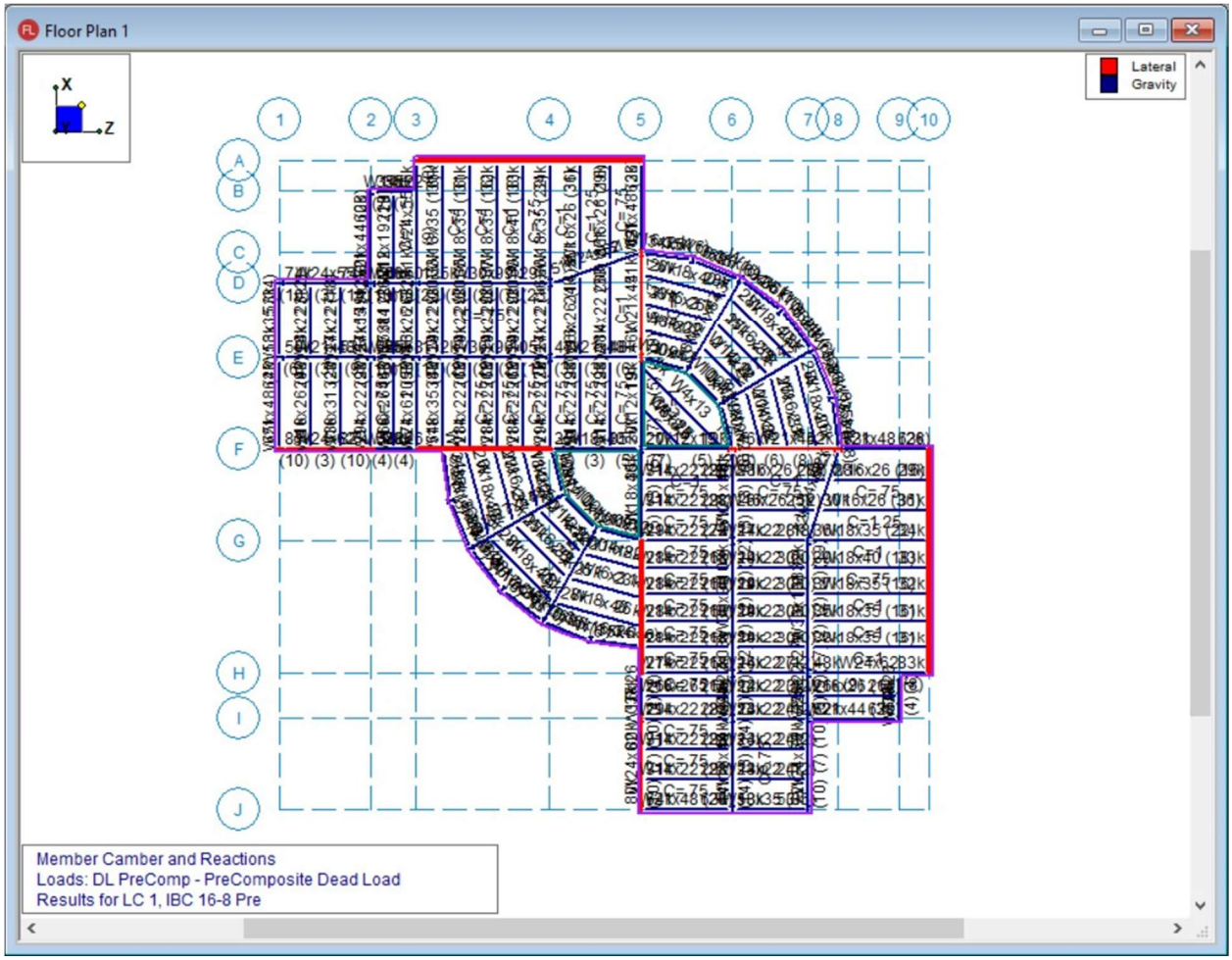
Follow the steps below to load the RISAFloor starter file for this portion of the tutorial:

- Double-click on the **RISAFloor** icon to start the program.
- Click **Open Model**  . Double-click the **Tutorials** folder, select **Tutorial A7 Starter.rfl** and click **Open**. Click **Close**  (or **Cancel**) to exit the **Model Settings** dialog box.

Solve the model and save:

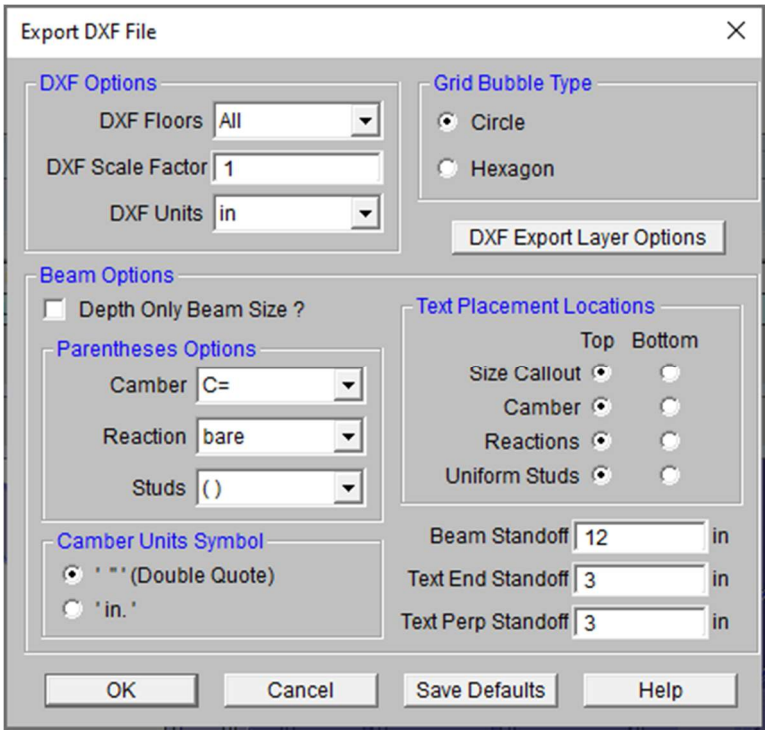
- Click **Solve**  to solve the model.
- On the **File** menu, click **Save As** and enter a new file name.

Your model should now display design data such as member sizes, end forces, and number of studs.

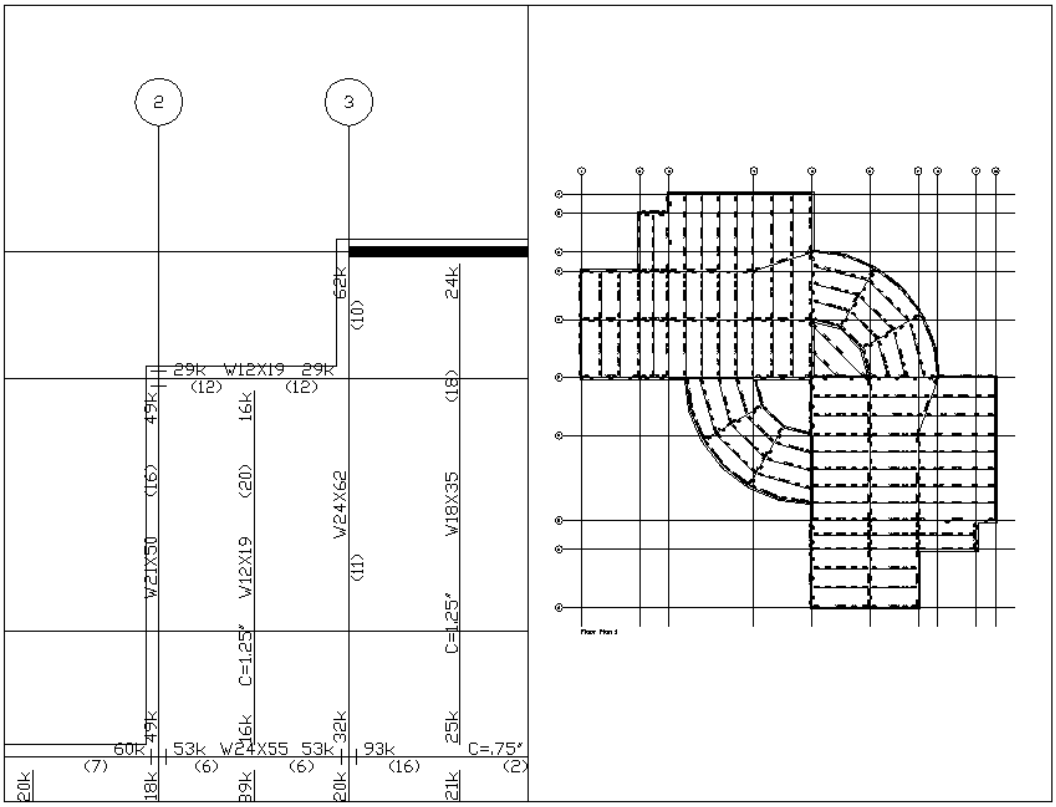


- On the **Main** menu, click **File**, click **Export**, and select **DXF File**.
- Enter the file name **Tutorial** and click **Save**. The **Export DXF File** dialog box will display.

- Enter the information shown below and click **OK**.



You should be able to open this DXF file with any standard drafting package. The following is an example of the information that can be written to the drawing file.

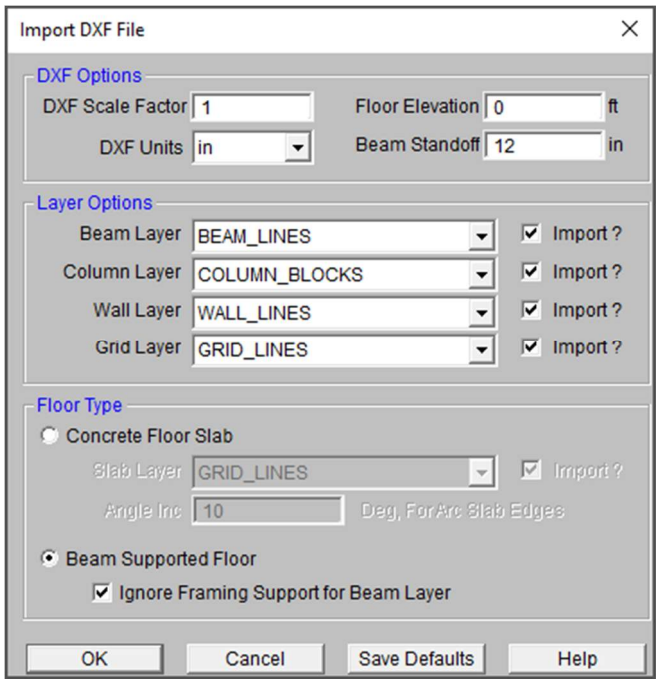


Similarly, you can import a DXF file into RISAFloor for analysis.

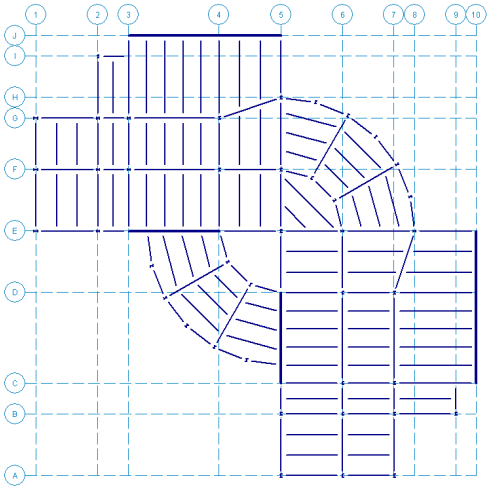
- On the RISA toolbar, click on **New Model** to start a fresh model.
- Because you want to import geometry rather than start from scratch, click **Close** to exit the **Starting a Model** dialog box.
- On the **Main** menu, click **File**, click **Import**, then select **DXF File**.
- Select the tutorial file you recently exported and saved. It should be named **Tutorial A7_FLOOR_1.dxf**.

The **Import DXF File** dialog box will open.

- Enter the information shown below:



- Click **OK**. Your model will import and open a new floor from the imported DXF file.



This is the end of Tutorial 7.

Conclusion

This completes Part A: Beam Supported Floors.

Congratulations on completing your introductory tour of RISAFloor! 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.

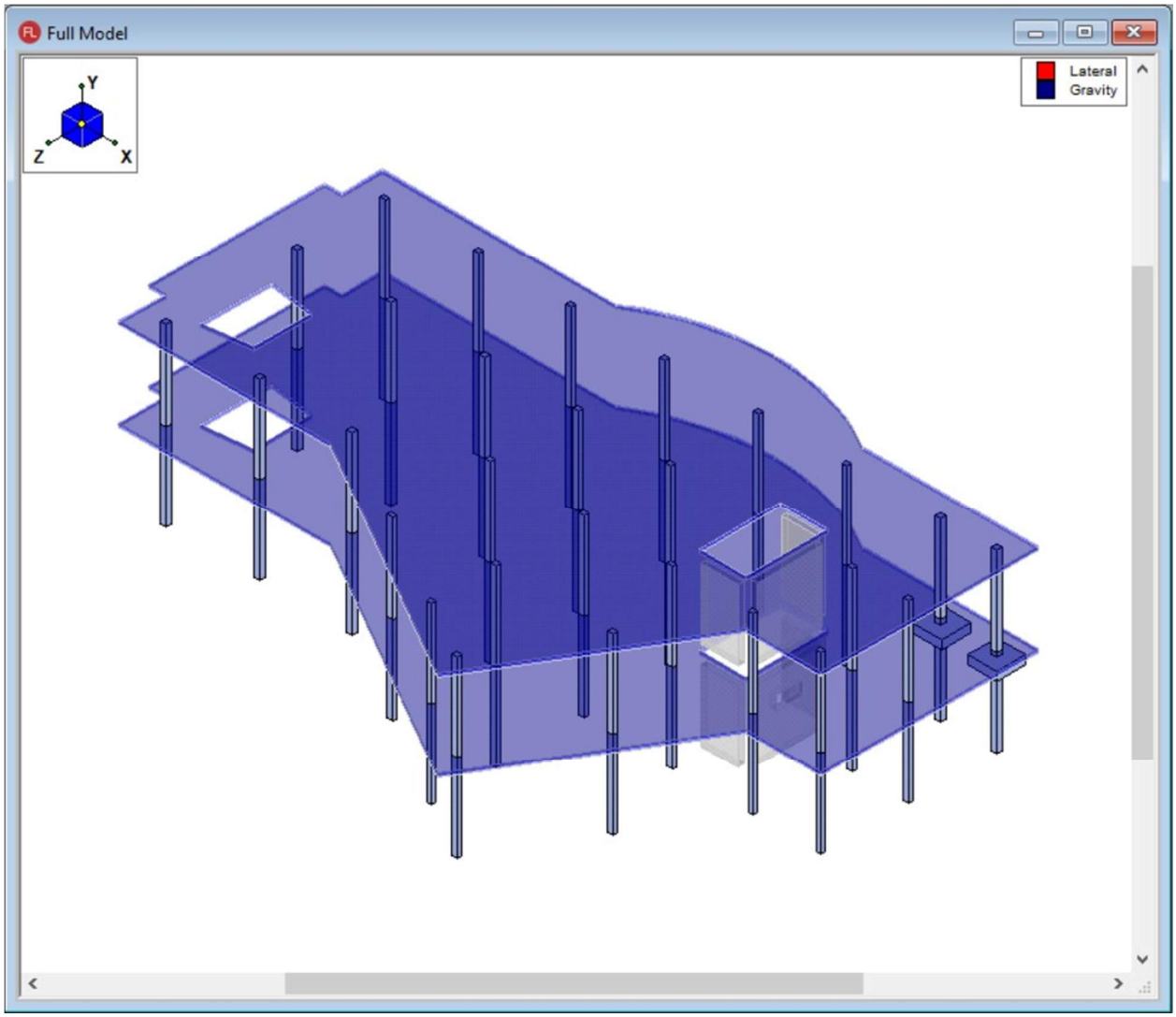
Part B: Elevated Slab Design

This second part of this book (Part B) will focus on building a model with elevated slab floors in RISAFloor ES. With the guidance of the following five tutorials, you will build, solve, and modify a typical building comprised of elevated concrete slabs, columns, and wall panels.

Note: In order to complete this model, you will be required to use RISAFloor ES. For more information on this new product and to find the demonstration version download, please see our website: www.risa.com.

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 program folder under **Tutorials**, and are named **Tutorial B2 starter.rfl**, **Tutorial B3 starter.rfl**, 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 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 B: Tutorial 1 – Columns & Walls

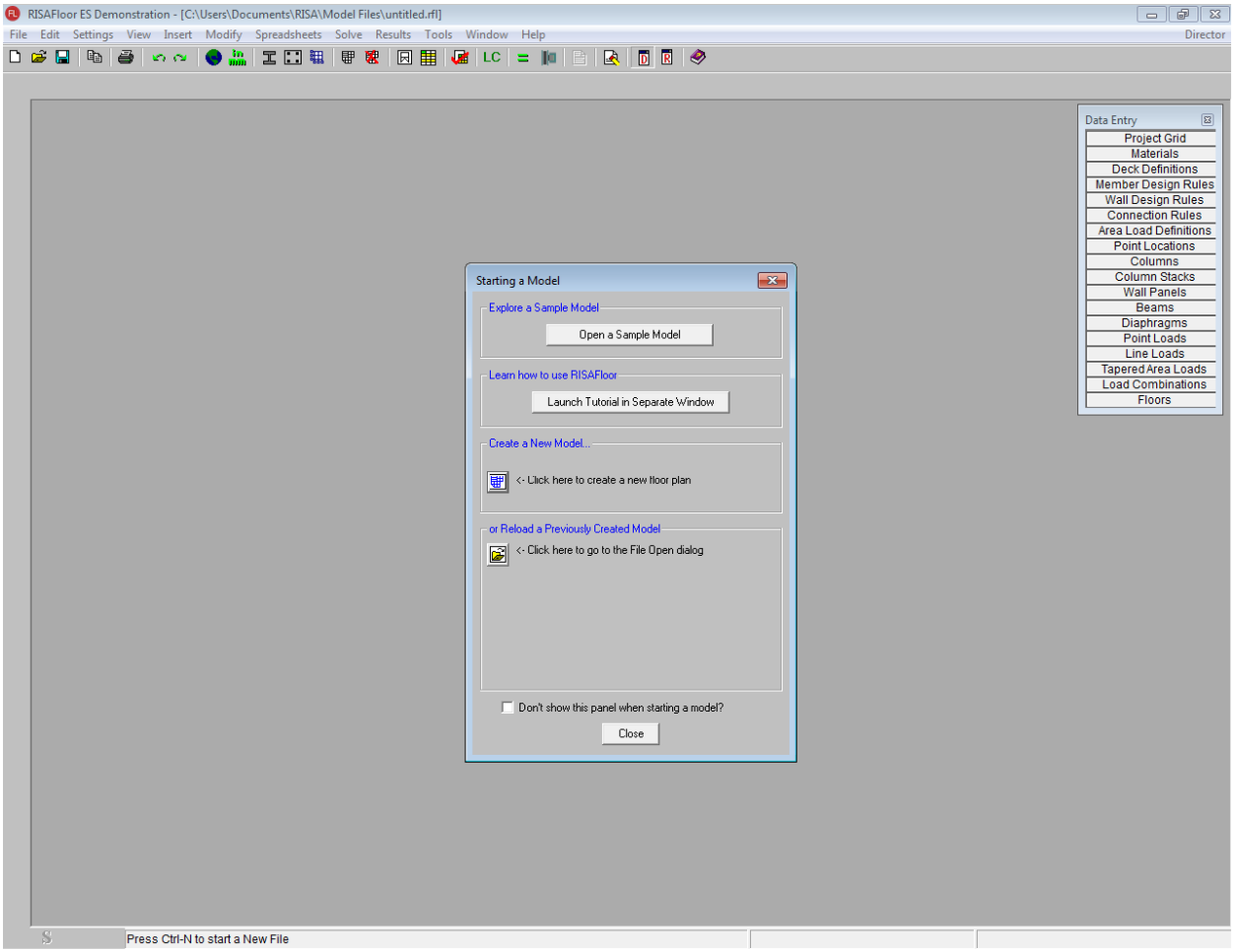
Overview

This first tutorial will introduce the various drawing features that RISAFloor ES has to offer. You will model several columns and walls and explore the Model Settings. Please note that in order to complete this model, you will be required to have RISAFloor ES installed. For more information on this new product and to find the demonstration version download, please see our website: www.risa.com.

Starting a New File

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

- On your desktop home screen, double click on the **RISAFloor** icon.



Note: The view shown above is from the Demonstration version of RISAFloor ES. If you are running the full version of the program, this dialog will look slightly different.

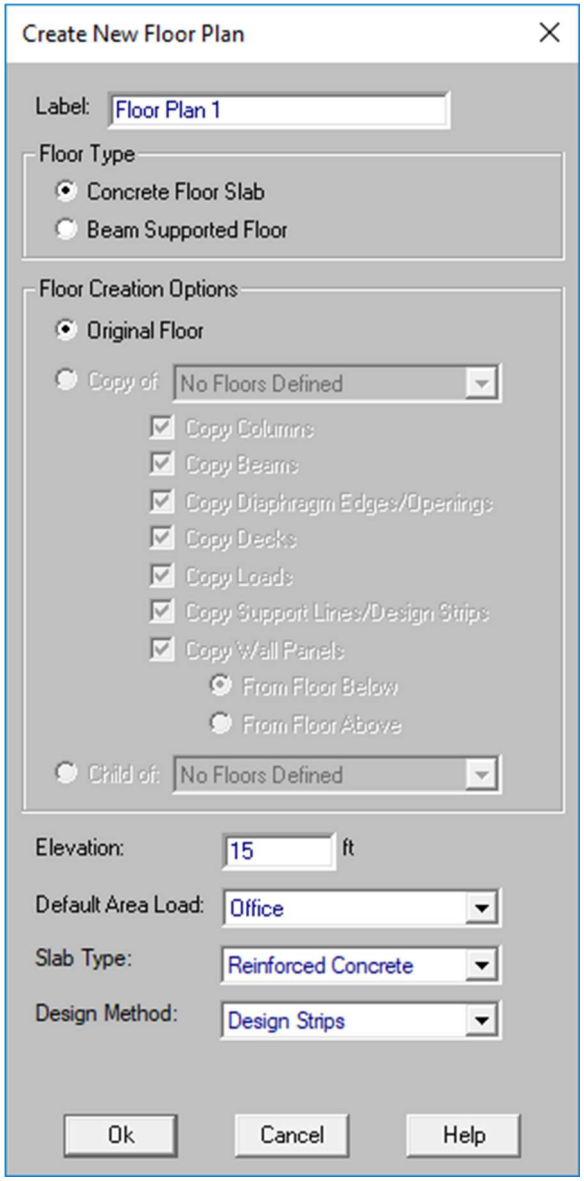
To create a new model, you may begin by creating a floor plan, open an existing file, or click **Close** and work on your own. With this tutorial we will create a new floor plan.

- Click the Create a New Floor Plan button  .

The **Create New Floor Plan** dialog box allows you to input your **Floor Type** as well as some general floor options. Since this is the first floor in the model, the only **Floor Creation Option** is **Original Floor**. After creating one or more floors, you may then use them as the starting point for additional floors, either by creating a **Copy** or a **Child** of an existing floor.

We will create a concrete elevated slab floor.

- Under **Floor Type**, select the **Concrete Floor Slab**.
- In the **Elevation** box, enter **15** ft.



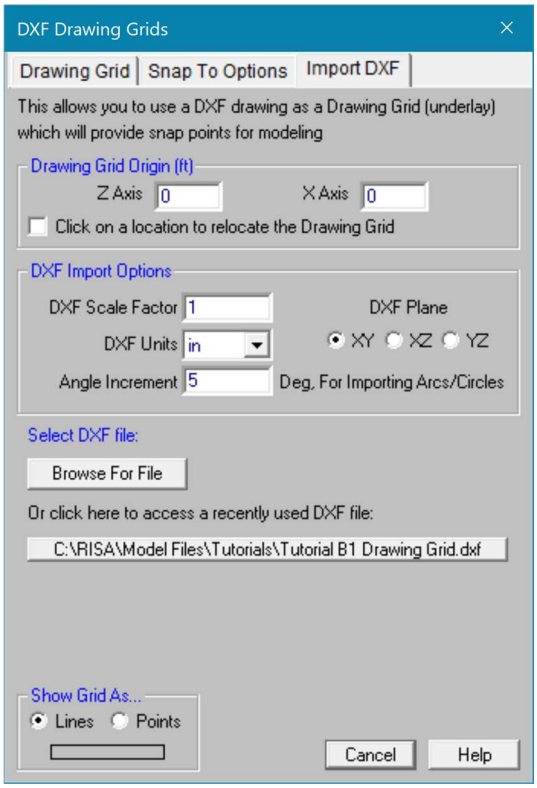
- Click **Ok** and a new blank floor will display.

Drawing Grid

RISAFloor ES offers both Project Grids and Drawing Grids. In our model we will use the DXF Import option to import in our Drawing Grid background.

Note: Please see the *RISAFloor General Reference Manual* to learn more about the difference between Project Grids and Drawing Grids.

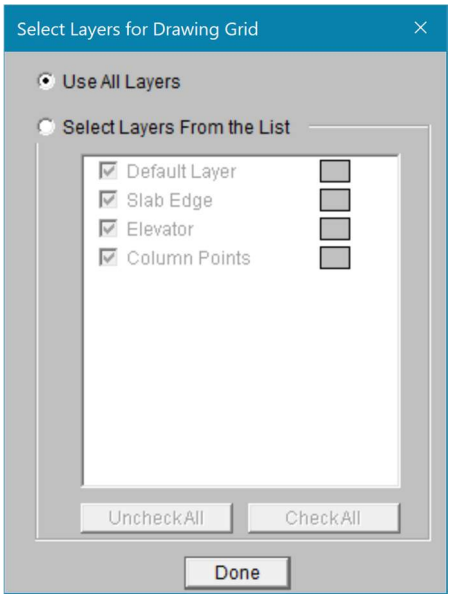
The DXF Drawing Grids dialog should have automatically opened for you. If not, select **Drawing Grids** from the **Modify** menu then click on the **Import DXF** tab.



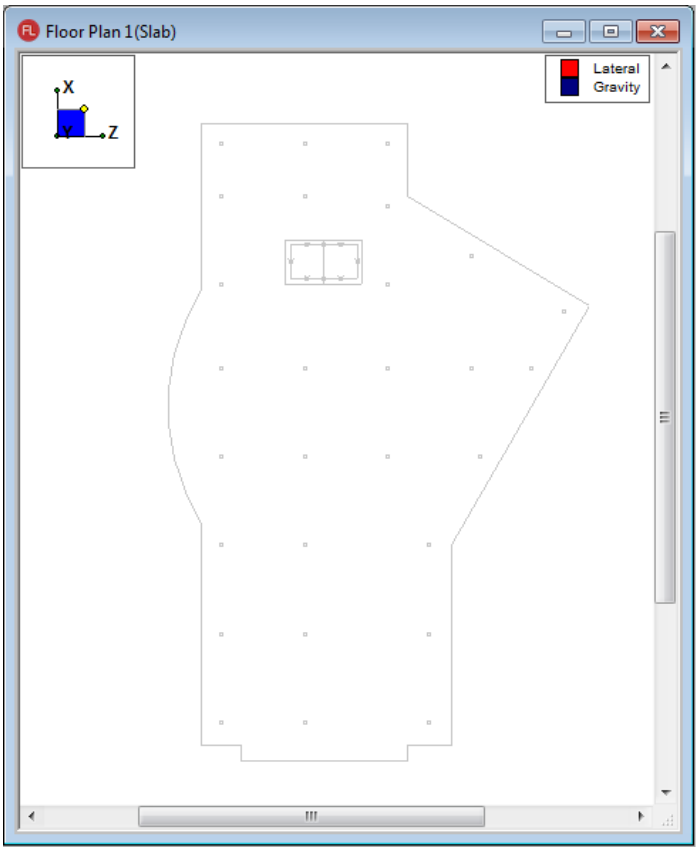
- Click on the **Browse for File** button and then browse to the **Tutorials** file under your **C:\RISA User Data\%Username%\Model Files\Tutorials** folder.
- Select the file called **Tutorial B1 Drawing Grid.dxf**.

Next you are given the option to select which layers in your DXF drawing to import. For this tutorial we will bring in all layers.

- Select the **Use All Layers** option and then click **Done**.



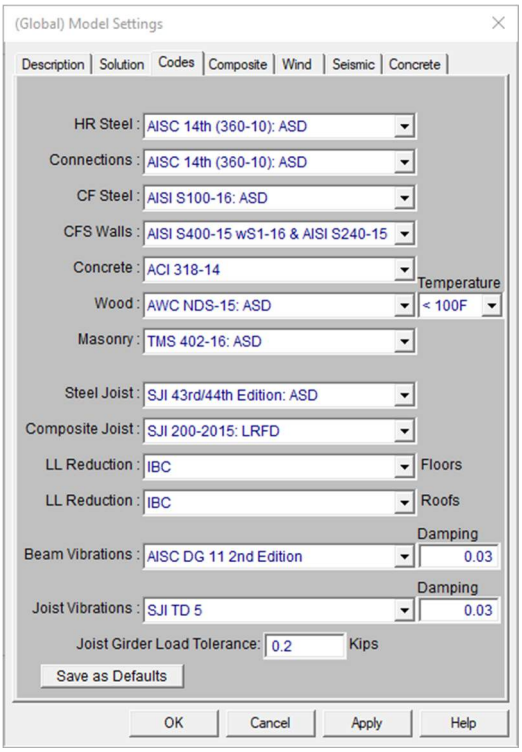
This will bring in the DXF drawing as a background drawing grid for us to use for drawing snap points.



Model Settings

Model Settings are settings that apply to the entire model. To view these settings:

- On the RISA toolbar, click the **Model Settings** button  .
- Click on each tab to review the settings (the default settings will be used for this tutorial, so there is no need to change any of this data).



In the **Model Settings** dialog box, you can enter model information and member design information.

Close the **Model Settings** dialog box:

- Click **Cancel**.

Drawing

When building your model, you will define the elements in the order that they will be built. For example, supporting elements (columns & walls) will be drawn in before the slab edge. You may enter model elements through the spreadsheets or with the drawing tools. The use of drawing tools will be the focus of the next few sections of this tutorial.

The Drawing Toolbar is located at the top of the workspace and looks like this:

The toolbar buttons are arranged in the order you would use them to define your model (left to right).

- To turn the Drawing toolbar on or off, go to the Window toolbar, and click the **Graphic Editing** button  .

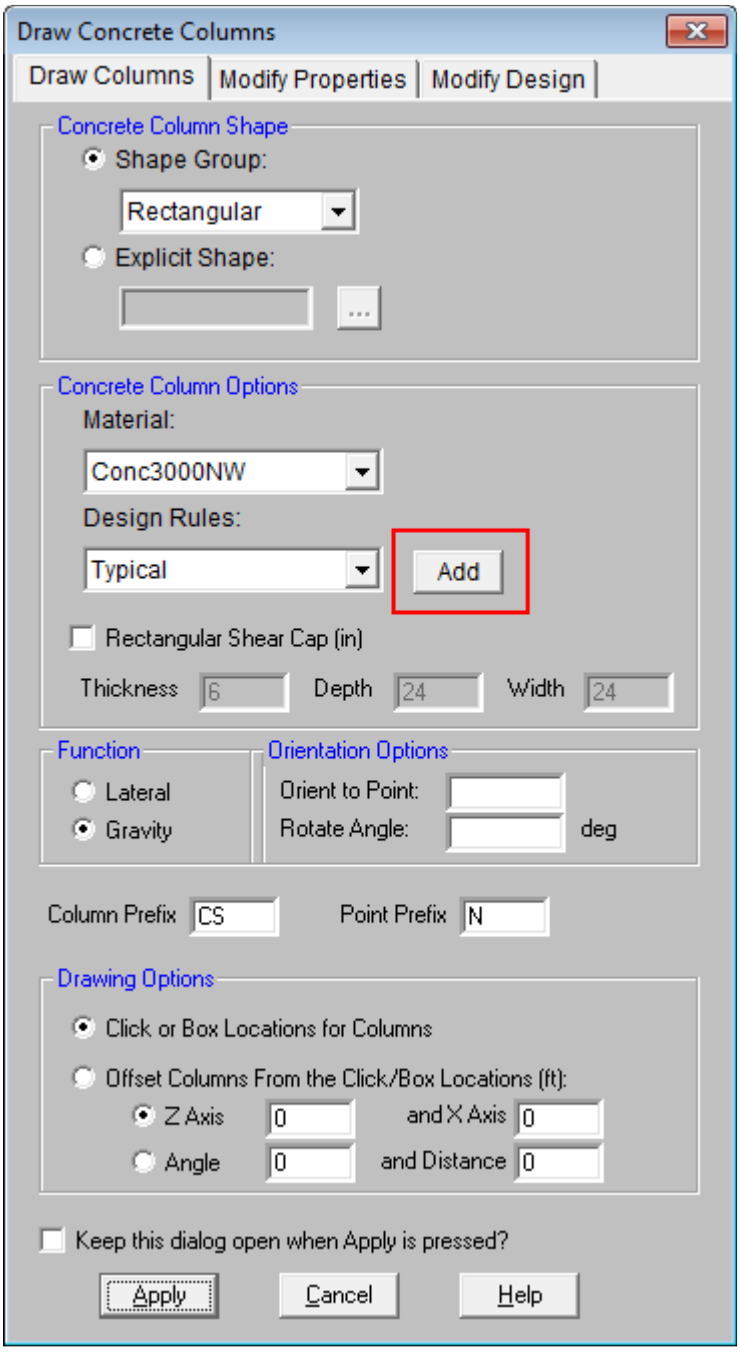
Columns

When creating columns, you may choose the material and shape of the column, as well as designate its orientation. You may orient the column towards a specific point, give it a specific angle, or combine the two. For this model, we will use several of these options.

- On the Drawing toolbar, click **Draw or Modify Columns** to view the **Draw Columns** dialog box.

Before drawing, let's set up a Design Rule to govern the optimization (design) of the columns.

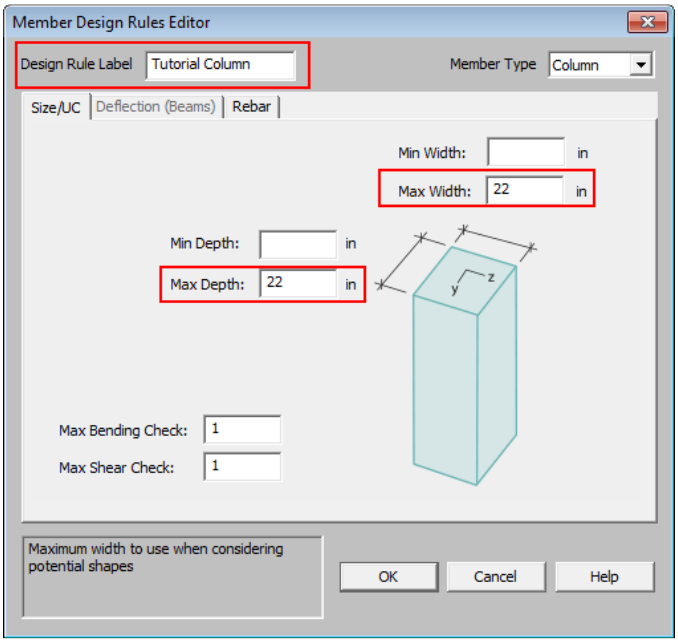
- Click on the **Add** button next to the **Design Rules** drop down menu.



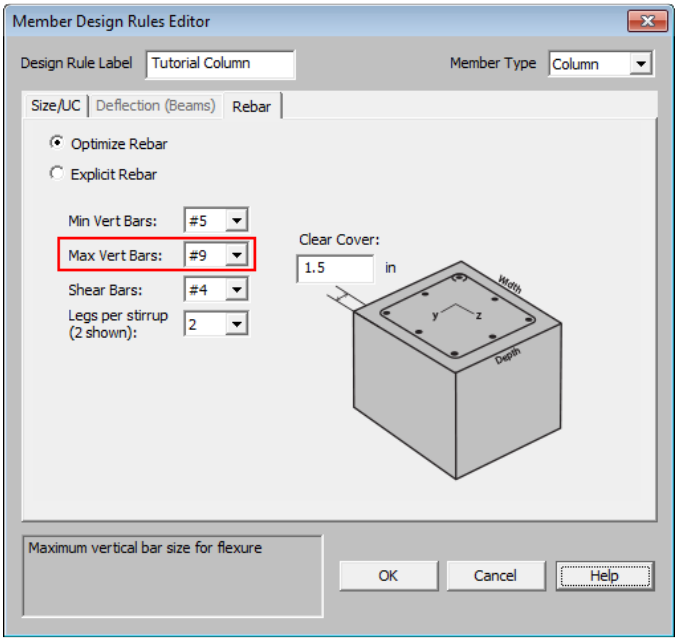
Column Design Rules

Create the design rule:

- Type in a **Design Rule Label** of "Tutorial Column".
- Type in a **Max Width** and **Depth** of **22**".



- Click onto the **Rebar** tab.
- Select **Optimize Rebar** and set the **Max Vert Bars** to a **#9** bar.

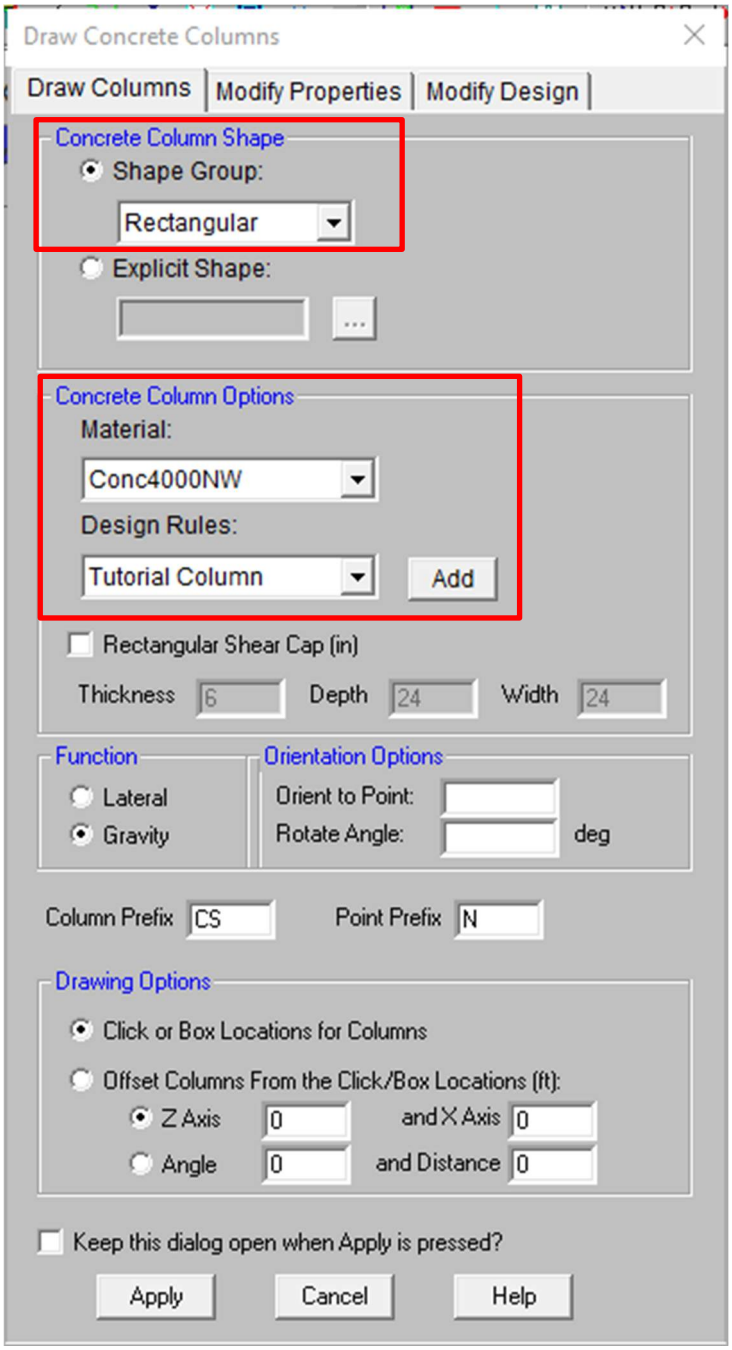


- Click **OK** to close the dialog and save your design rule.

Note: Design rules can be edited from the design rules spreadsheets.

Return to the **Draw Concrete Columns** dialog to finish defining the columns:

- Under Concrete Column Shape, select the Rectangular Shape Group.
- Under Concrete Column Options, select the Conc4000NW material.
- Make sure that your new **Design Rule** is still selected.

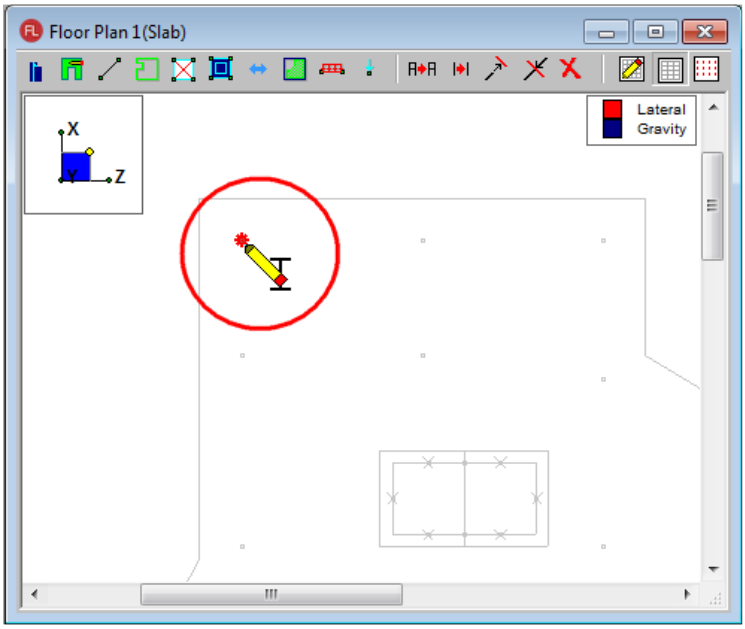


- Click **Apply** to start drawing the columns.

Note: When you enter the drawing mode, your cursor changes to .

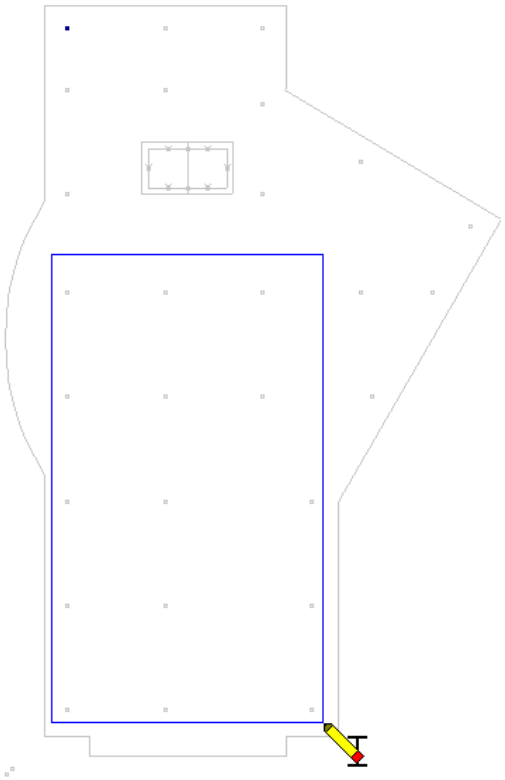
You can now start defining the columns by clicking on the drawing grid locations as shown below:

- Create a column at the top left corner by clicking on the column grid location.



You can also define columns by "box selecting" a group of drawing grid points.

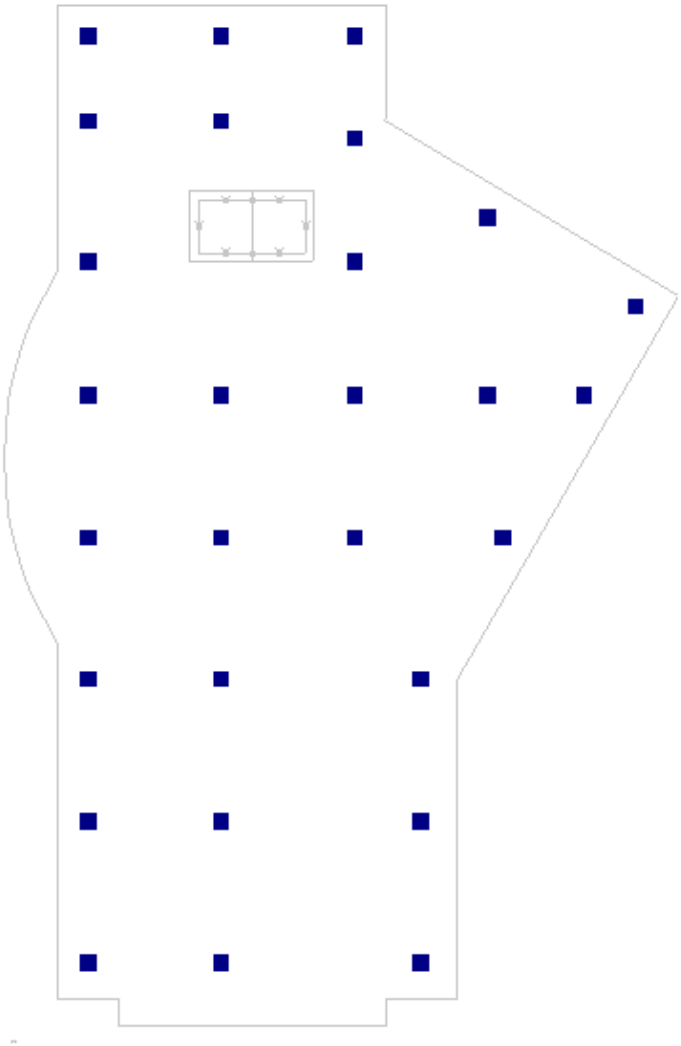
- Hold down the left mouse button and drag the cursor to select the box over your column grid locations.



- Use whichever method you prefer to select the remaining column grid locations.

- Right-click your mouse to cancel the drawing tool when done.

When finished, your model will look like this:

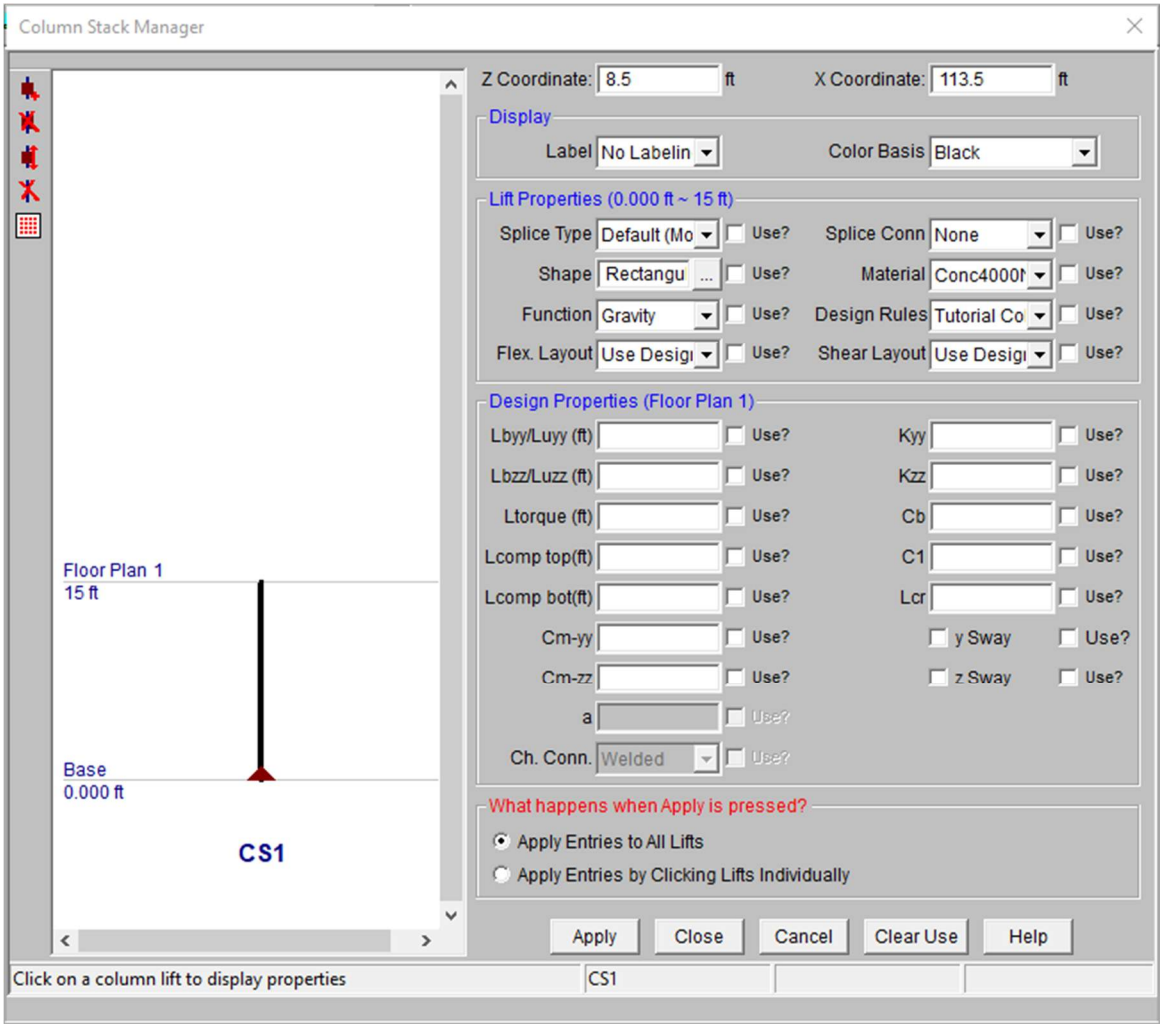


Modifying Your Columns

RISAFloor ES provides a quick and easy way to view and edit properties of any existing item in the model. You may do this through the input spreadsheets or using any of the **Modify** dialogs. For column stacks, you can also double-click the column in the model view and open the **Column Stack Manager**.

Column Stack Editor

- Double-click on any column in the model view. This will open the following dialog:



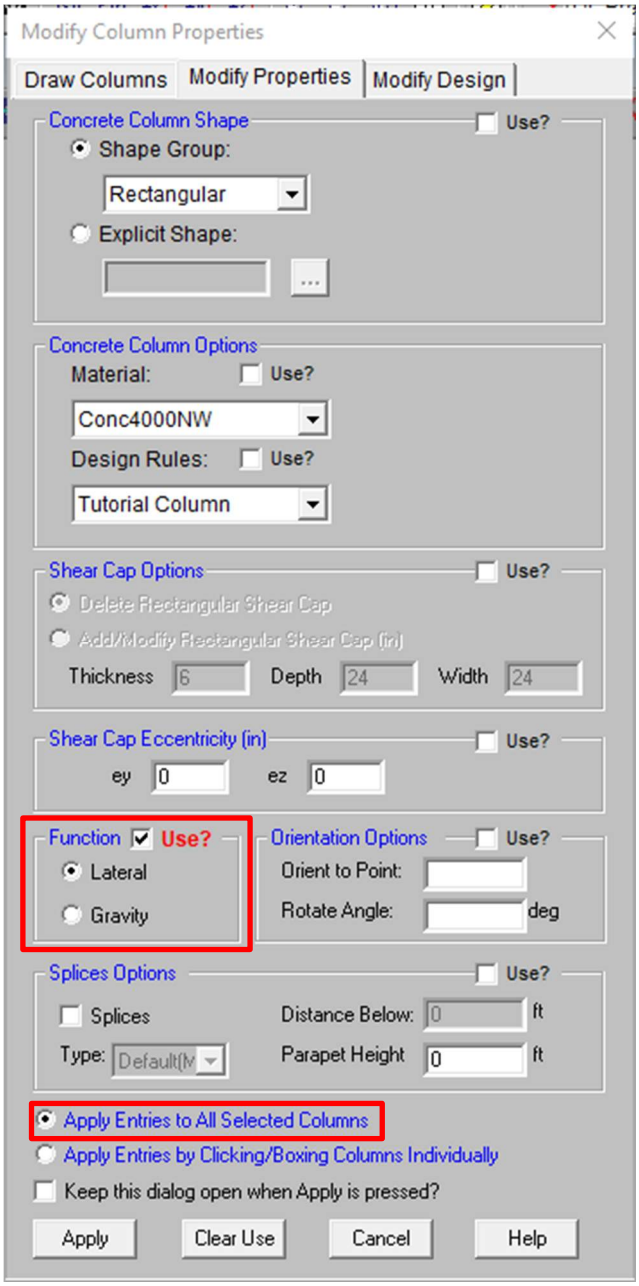
From this dialog you have the ability to edit the column properties. Simply make your changes and then click the "**Use?**" checkbox before clicking **Apply**.

For this tutorial, rather than change each column one by one through their editors, we will use the **Modify** dialog to change their **Function**.

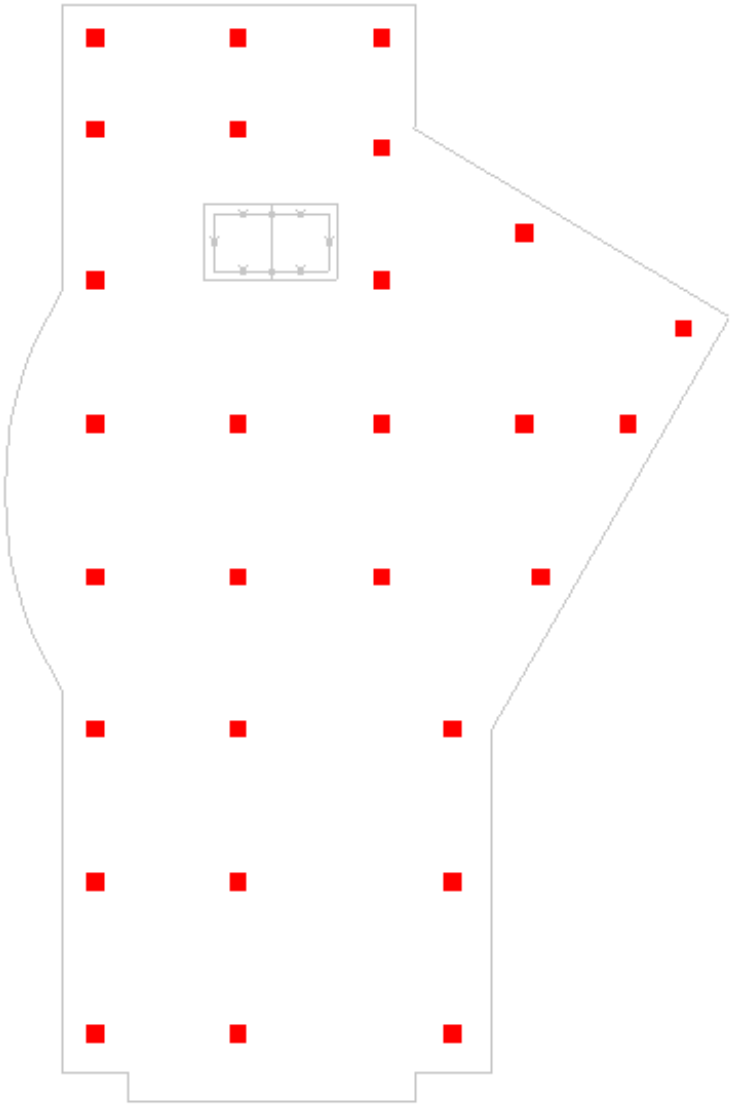
- Close the **Column Stack Manager** without making any changes by clicking **Cancel**.

Modifying Columns

- Select **Columns** from the **Modify** menu.
- Under **Function**, select **Lateral**. Check the "**Use?**" checkbox.
- At the bottom, select Apply Entries to All Selected Columns and then click Apply.



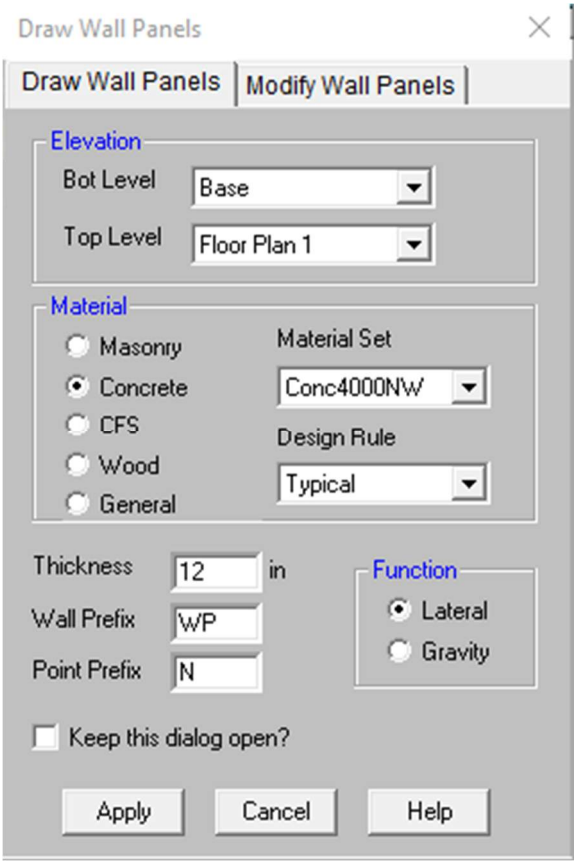
All of the columns will change to red, indicating that they are now all lateral members.



Wall Panels

Prepare to draw your wall panels:

- On the Drawing toolbar, click **Draw or Modify Walls** .
- Set the Material to Concrete and change the Material Set to Conc4000NW.
- Under **Function**, click **Lateral**. The wall panel settings should match those in the image below.

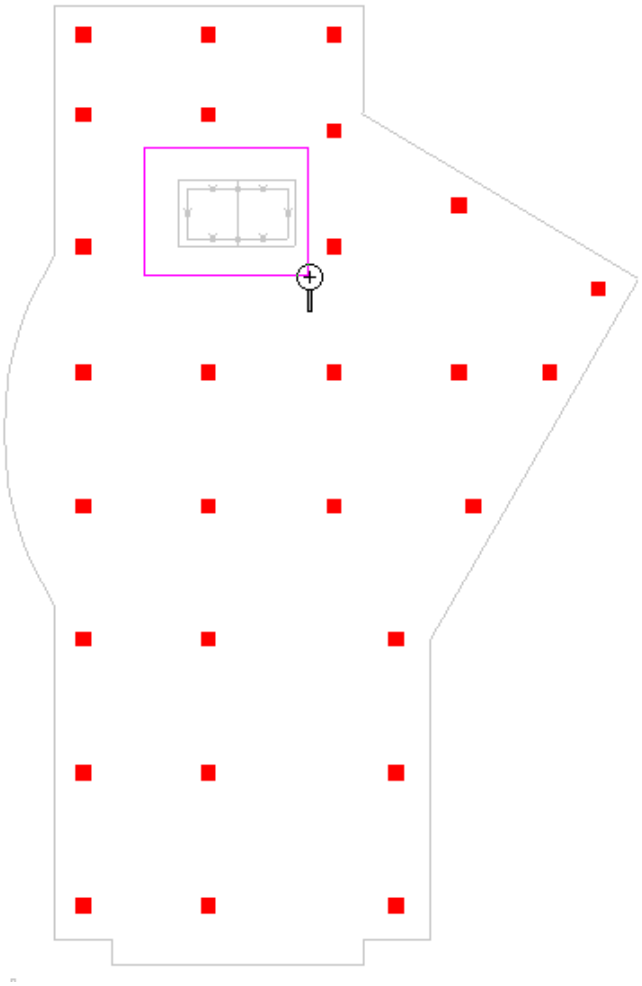


- Click **Apply**.

Note: Your cursor will change to to indicate that you are now ready to draw wall panels.

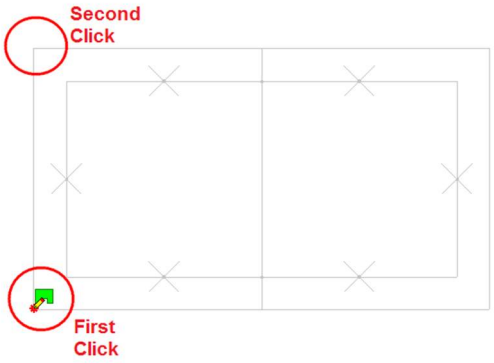
First zoom in on the area of the elevator opening using the Box Zoom tool.

- Click on **Box Zoom**  from the Drawing toolbar.
- Box select the elevator opening on the drawing grid.

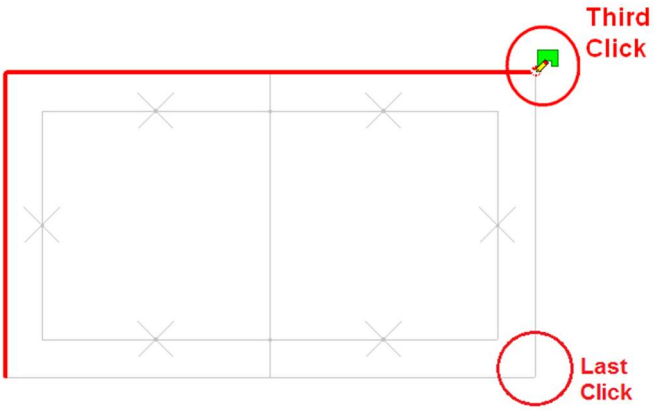


Now that you have a zoomed in view, start drawing your wall panels:

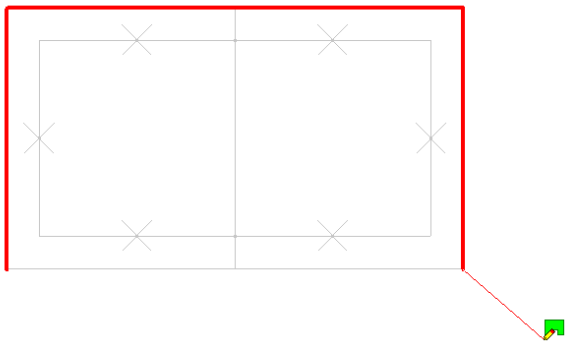
- Draw in the wall panels by first clicking on the outside lower left point of the elevator opening. Then click on the outside drawing grid corner directly above that.



- Continue around the outside of the elevator opening.

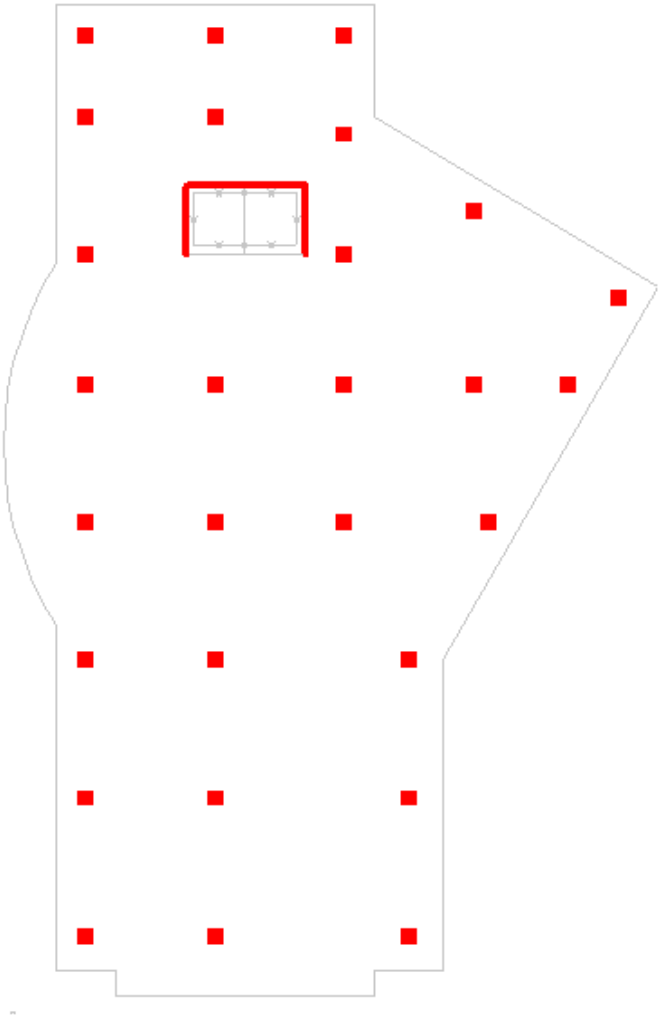


After you draw a wall, your cursor is “anchored” to the end point of the wall.



- To release the drawing tool, use your right mouse (right-click) or ESC.
- Return to the full model view by clicking on the **Full Model View** button  from the Drawing toolbar.

When finished, your model should look like the following.

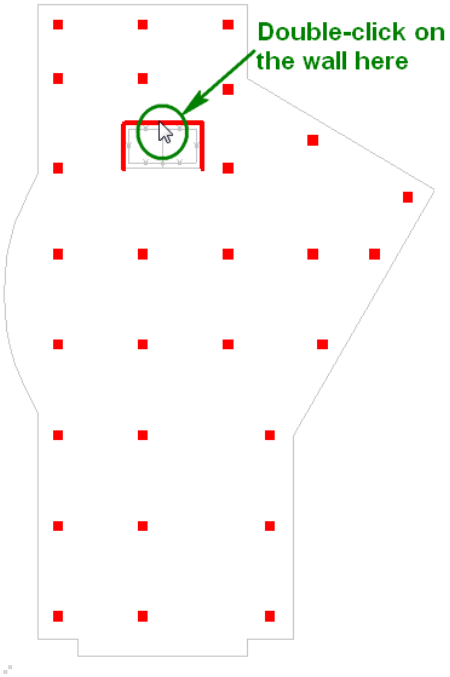


If you make a mistake or if you want to delete a wall, click CTRL+Z to undo, or delete the wall as follows: on the **Modify** menu, click **Delete**, then click **Delete Based on This Criteria** and check the **Delete Selected Walls** check box.

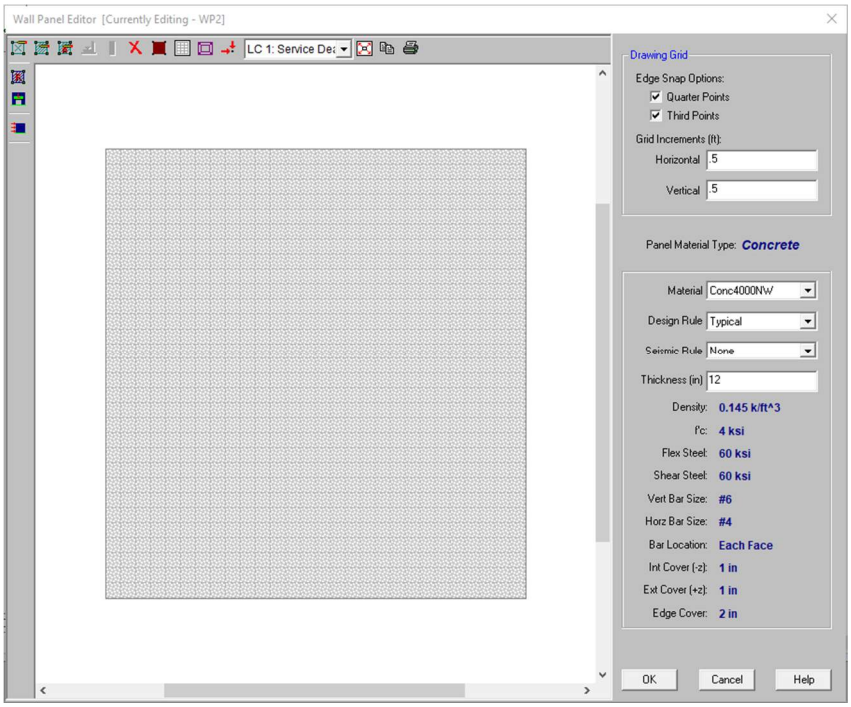
Modifying Wall Panels

Next, you will add in a wall panel opening, design region and continuous boundary condition.

- Double-click on the middle wall that is parallel to the global Z axis.



This will open the **Wall Panel Editor** dialog.



This **Editor** allows you to add openings and design regions to the wall panels. You also can edit the **Material**, **Design Rule**, and **Thickness** in this view.

Wall Panel Openings

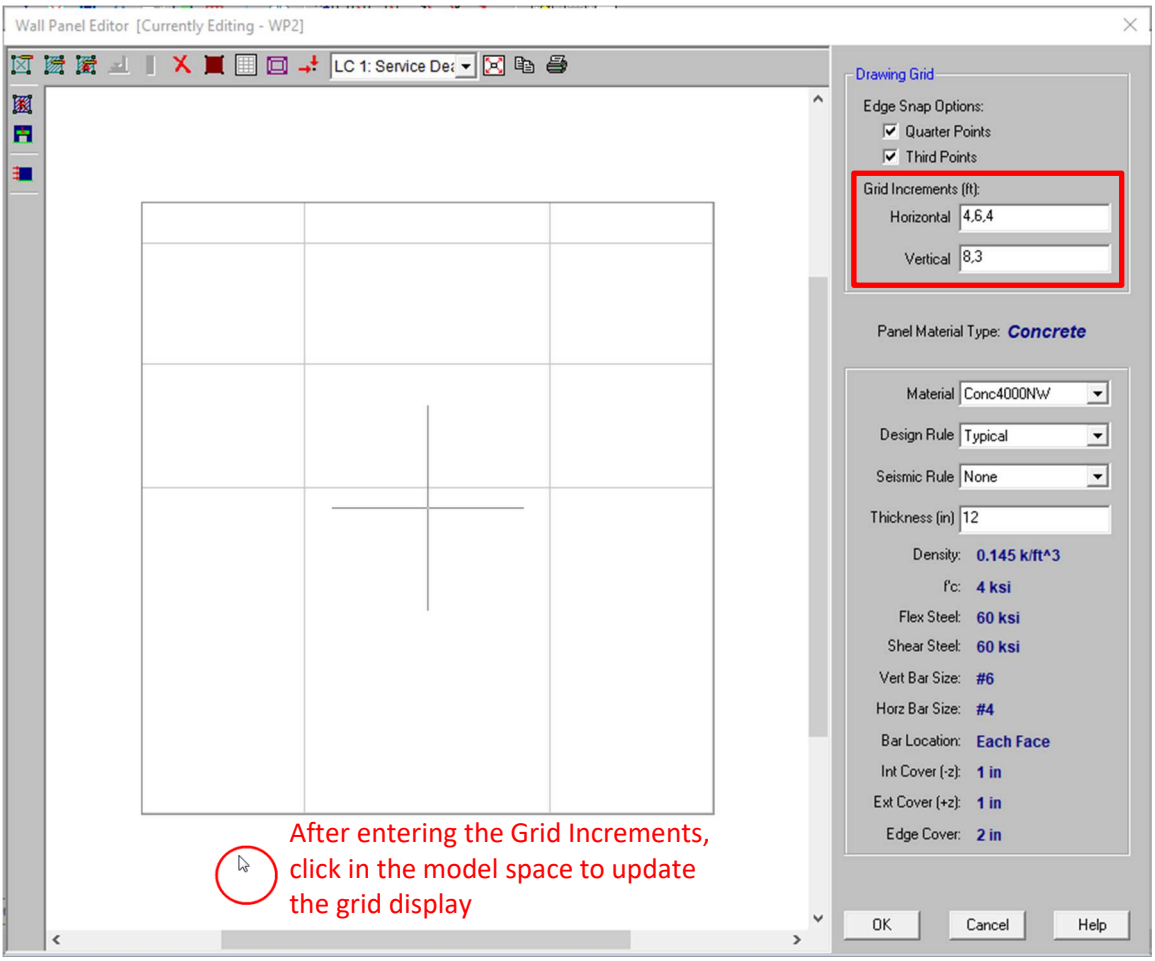
You will add in an opening next. Before you do this, turn off the rendered view of the wall.

- Click the **Toggle Rendered View** button  at the top center of the **Editor** Drawing toolbar.

Now only the drawing grid is visible in the plane of the wall.

Next, set up a **Drawing Grid** to help define the opening.

- Under Horizontal Grid Increments, enter 4,6,4.
- Under Vertical Grid Increments, enter 8,3.
- Click into the **Editor** model space to update the grid.



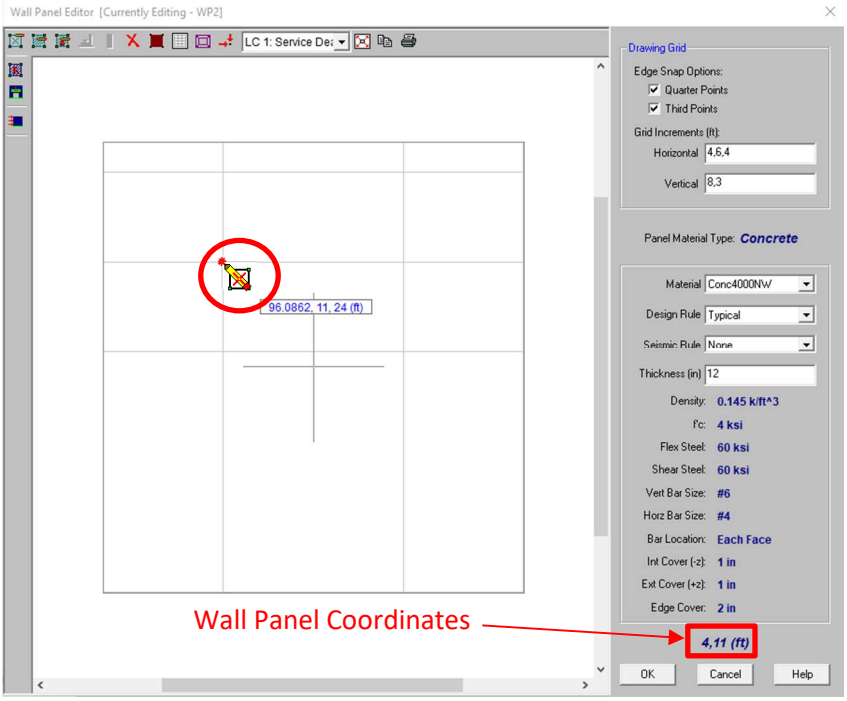
Now draw in the opening.

- Click on the **Create New Openings** button  (in the upper left corner of the Editor).

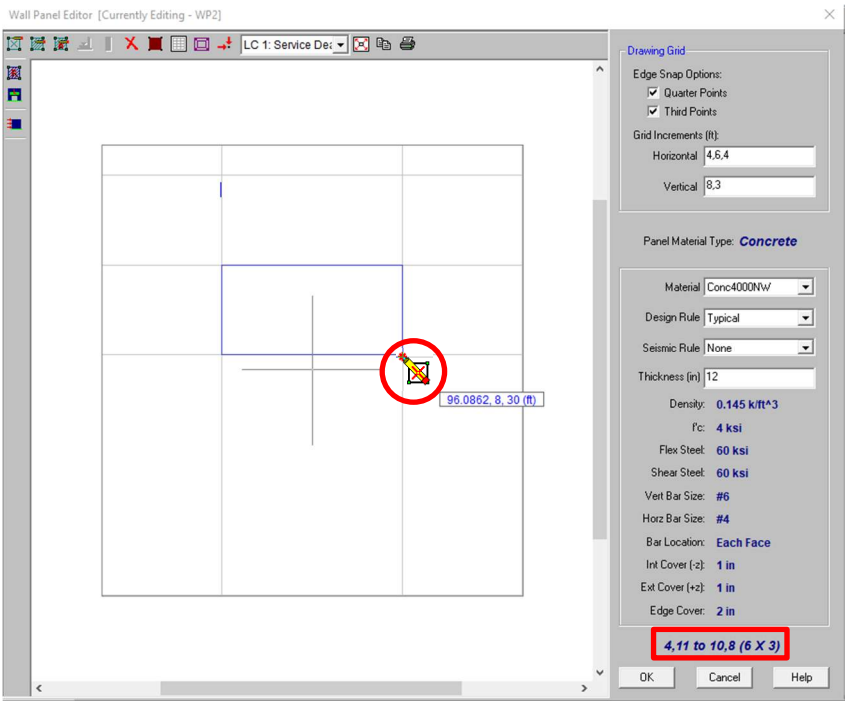
Note: Your cursor will change to  .

Use the coordinate information in the lower right of the Wall Panel Editor to select the opening coordinates.

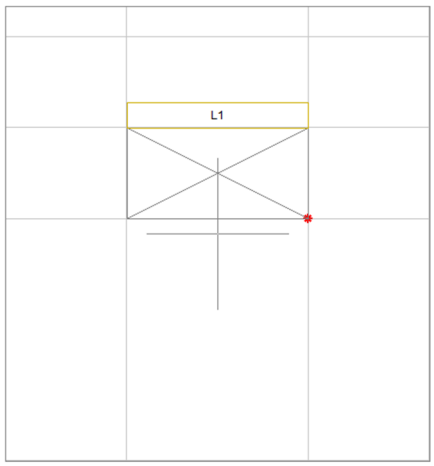
- First click on the top left of the opening (**4,11**).



- Next click the lower right corner of the opening (**10,8**).



When complete, the opening will show with the X lined and the lintel label.



- Right-click or click ESC to cancel the opening drawing tool.

Wall Panel Design Regions

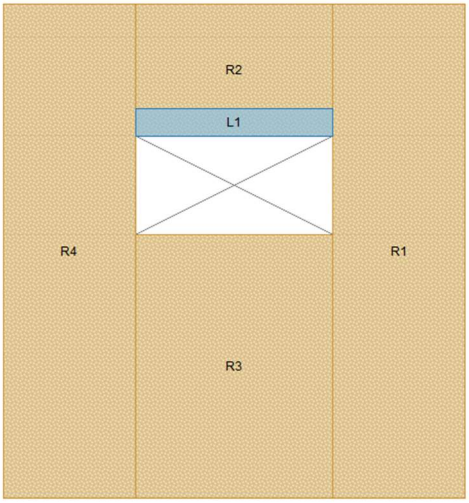
When finished, you can add in the design regions around the openings. The **Wall Panel Editor** allows you to create design regions in two ways: manually draw in each region, or by using the automated generation tool. You will use the automated generation for this model.

- Click on the **Generate Wall Regions Automatically** button  (in the upper left corner of the Editor).

Turn the rendered view back on for a better view.

- Click the **Toggle Rendered View** button  at the top center of the **Editor** Drawing toolbar.

Now you can see the opening and four regions that were generated around it.



For more details on how to model wall panels, including information on masonry design and wood design, refer to https://www.risa.com/documentation.

- Click **OK** to close the **Wall Panel Editor**.

This is the end of Tutorial B1.

If you are using the Demonstration version of the program, you will not be able to save this model, as it exceeds the Demonstration version limits. Please proceed to the next tutorial for instructions on how to open the Tutorial B2 Starter File provided by RISA.

If you are using the full version of the program, you may save by doing the following: On the **File** menu, click **Save As** and enter a file name.

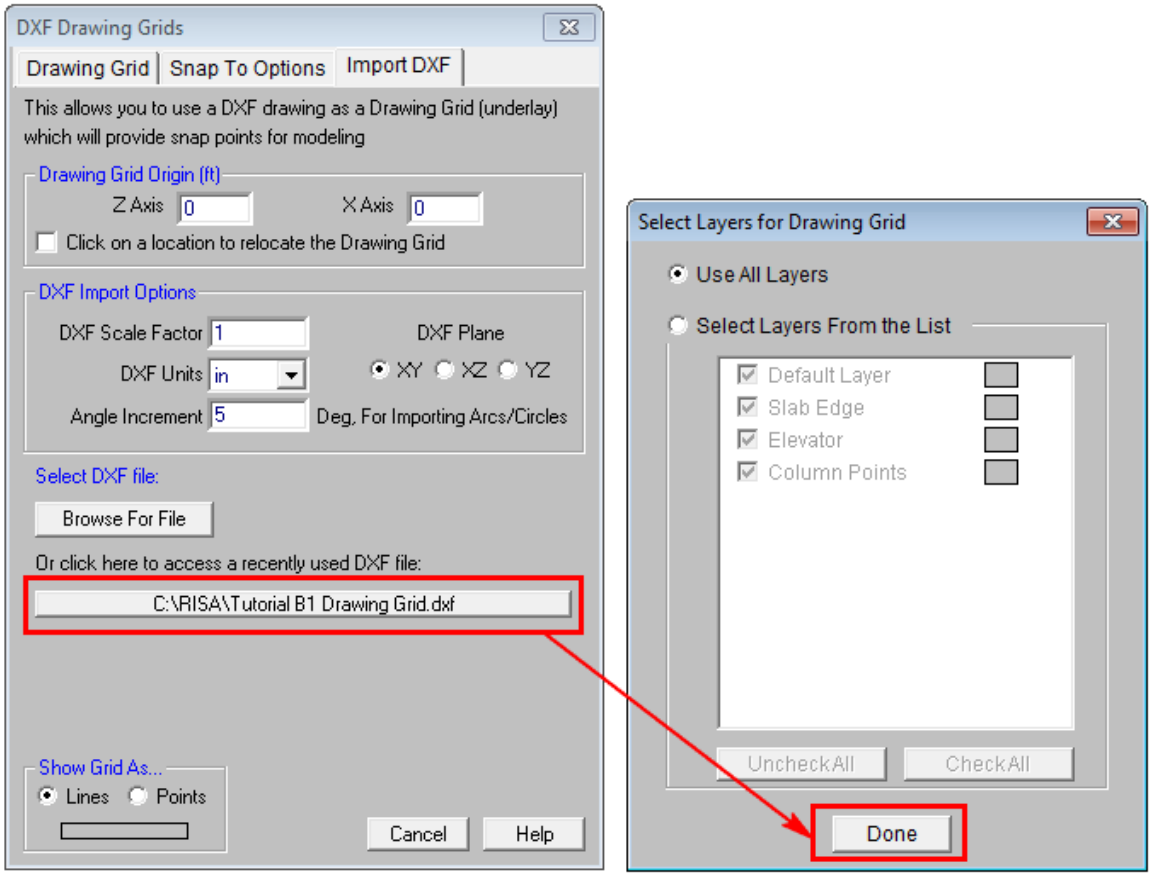
Part B: Tutorial 2 – Slab & Design Strips

This tutorial will guide you through the drawing and editing the slab edge, support lines, and design strips. This tutorial continues from where the previous tutorial ended, so follow these steps to open the Tutorial B2 Starter File provided by RISA:

- Double-click on the **RISAFloor** icon to start the program.
- Click **Open Model**  . Double-click the **Tutorials** folder, select **Tutorial B2 Starter.rfl** and click **Open**. Click **Close**  (or **Cancel**) to exit the **Model Settings** dialog box.
- On the Window toolbar, click the **Graphic Editing** button  to activate the Drawing toolbar.

Re-Import the DXF Drawing Grid

- Click on the **Modify Drawing Grid** button  to re-open the **Drawing Grid** dialog.
- Click on the **Import DXF** tab. Click to access the **recently used DXF file** then click **Done**.



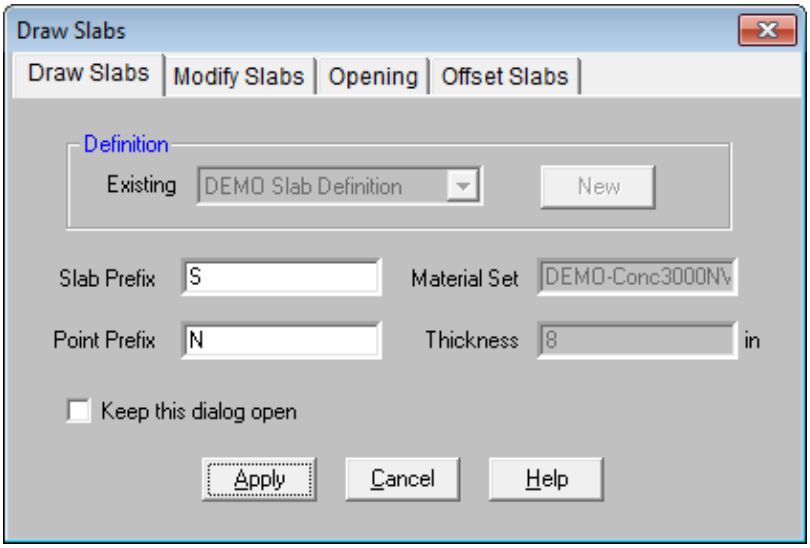
Draw the Slab

Start by drawing in the slab.

- On the Drawing toolbar, click **Create Slab Perimeter** to open the **Draw Slabs** dialog box.

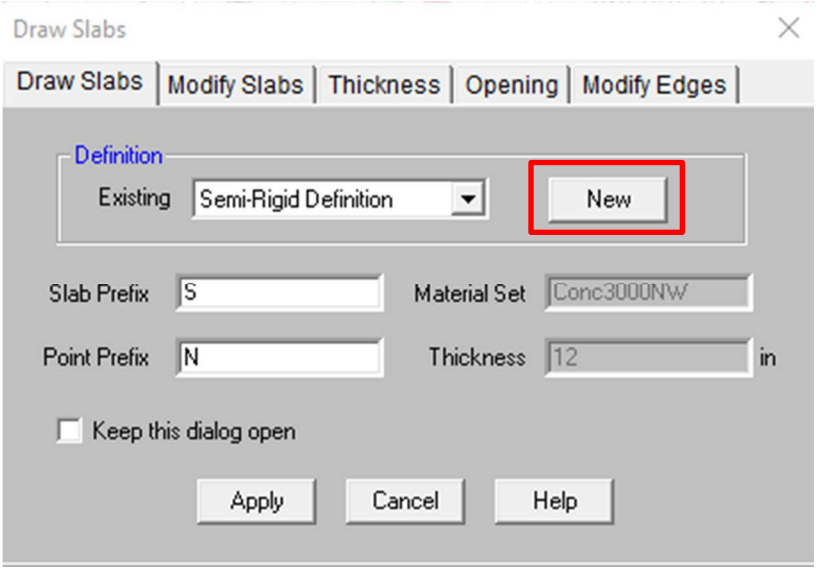
In this dialog box, select an existing slab definition or create a new one. For this tutorial, we will define a new slab.

Note: If you are using the Demonstration version, this option is not available, so you will need to skip down to the *Draw Slab* section on page 137 to accept the default slab definition and continue drawing.

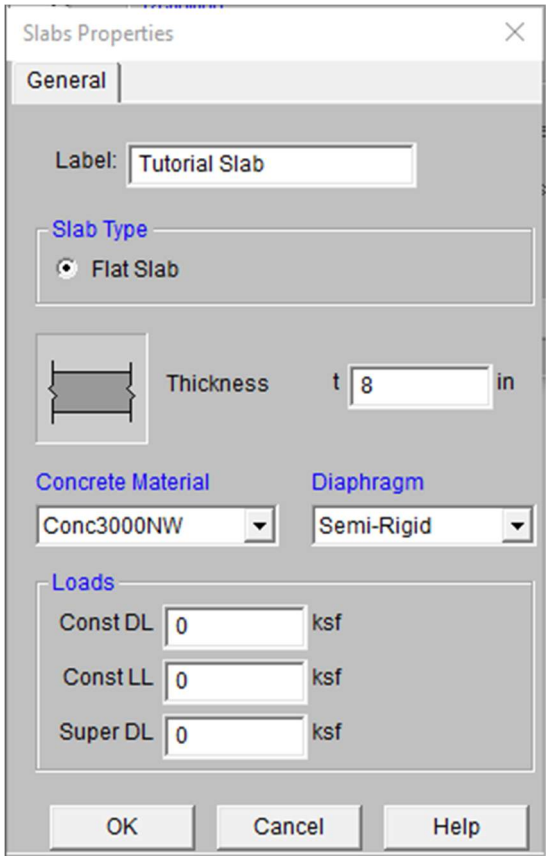


Slab Definition

- If you are not using the Demonstration version, click on the **New** button.

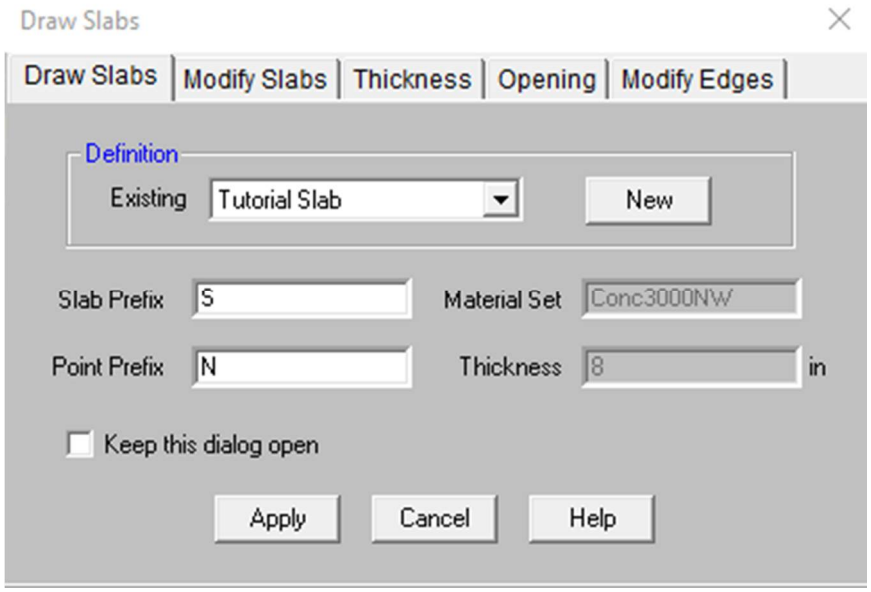


- Type in the label "**Tutorial Slab**", change the **Thickness** to **8"**, and set the **Diaphragm** type to **Semi-Rigid.**



- Click **OK** to exit the dialog and return to the **Draw Slabs** dialog.

The new slab definition should now be selected in the dialog.



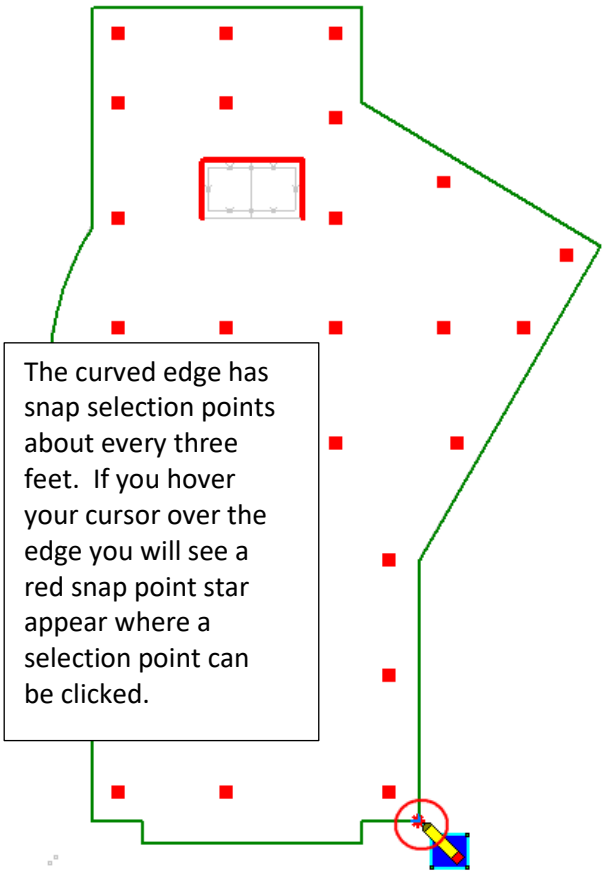
Draw Slab

- Click **Apply** to start drawing.

Note: The cursor will change to  to signify that you are now in drawing mode.

Draw your Slab edge:

- Using the snap points from our DXF drawing grid background to draw in the edge of the slab.



The curved edge has snap selection points about every three feet. If you hover your cursor over the edge you will see a red snap point star appear where a selection point can be clicked.

Draw in the slab by clicking around the edge at the corner points.

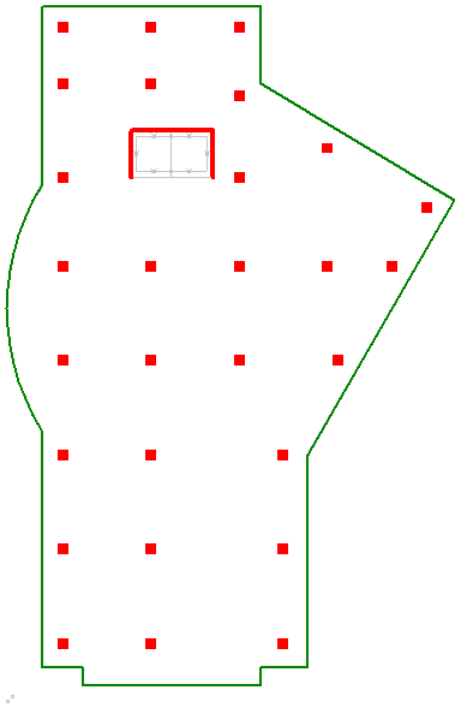
Notice that the slab edge is anchored as you select your points.

Note: the slab edge can be any polygon shape.

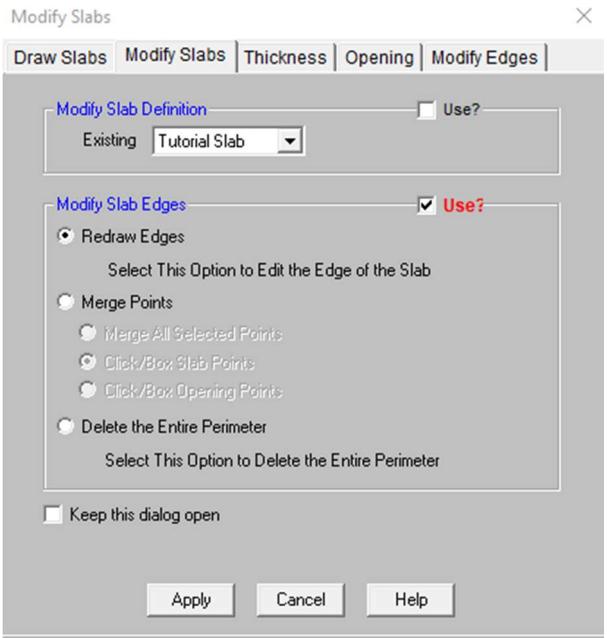
When you click back on the starting point a second time to finish, the tool will release and your slab edge will turn green as shown above.

- Right-click your mouse once more to cancel the drawing tool.

When finished, your model should look like the following.



If you have made any mistakes, you can use the **Redraw Edges** tool to correct the slab edge rather than having to re-draw it completely. To access this tool, click on the **Draw Slabs** button  from the Drawing toolbar and click on the **Modify Slabs** tab. Under **Modify Slabs**, select **Redraw Edges**. Click the **Use?** checkbox and then **Apply**. Now you can click on a slab edge and redraw just that edge of the slab.



For more direction on how to use this tool, please see the *Redraw Edges* section below on page 143.

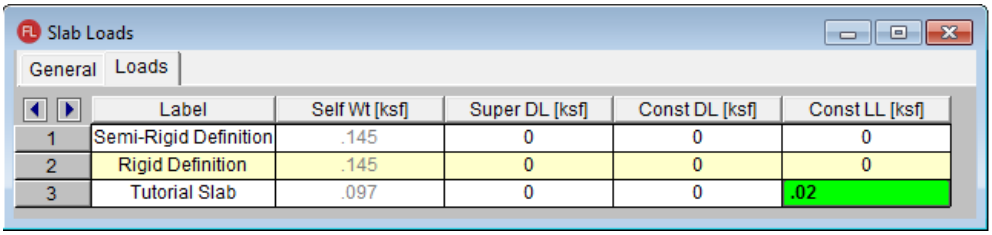
Modifying Your Slab

There are a number of ways in which you can modify your slab. We will start by editing the slab properties through the Slab Definitions spreadsheet.

- Click on **Slab Definitions** from the **Data Entry** toolbar. (If the **Data Entry** toolbar is not visible, click the **Data Entry** button on the RISA toolbar).

Here you can see your slab properties in spreadsheet form.

- Click on the **Loads** tab.
- Change the **Const LL** to **0.02** ksf for the **Tutorial Slab** on line 3.



	Label	Self Wt [ksf]	Super DL [ksf]	Const DL [ksf]	Const LL [ksf]
1	Semi-Rigid Definition	.145	0	0	0
2	Rigid Definition	.145	0	0	0
3	Tutorial Slab	.097	0	0	.02

- Close the spreadsheet by clicking on the **Close** button.

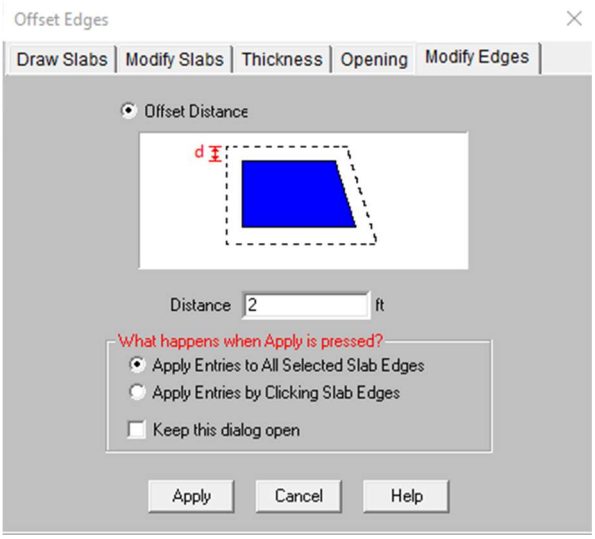
Next, you will modify the slab edges.

Offset Slabs

The first slab edge modification tool allows us to offset the slab edge overhang distance in one quick click.

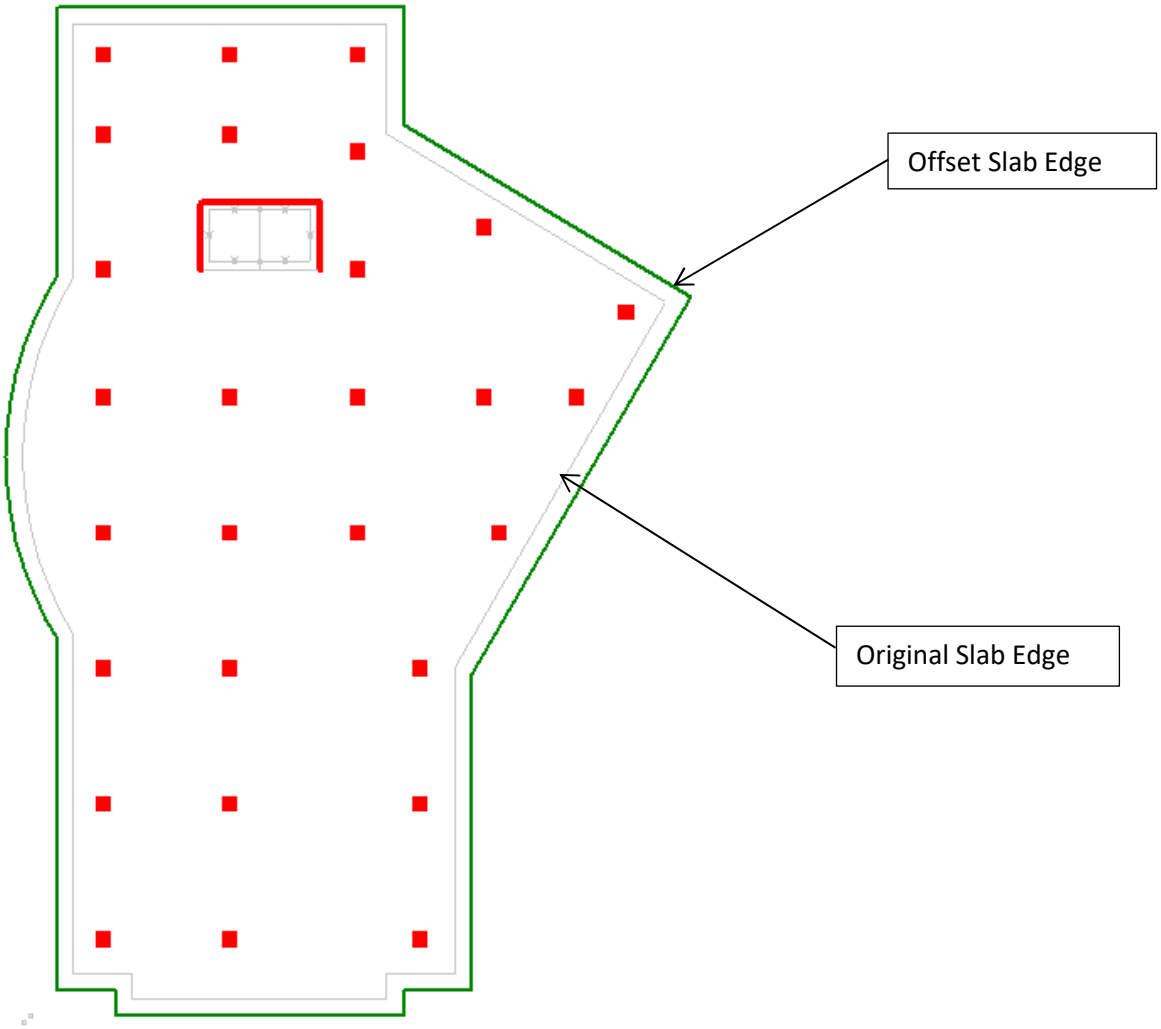
- Click on the **Modify Slab Edge** button from the Drawing toolbar.
- Enter an **Offset Distance** of **2** feet and select **Apply Entries to All Selected Slab Edges**.

Note: You can enter a negative value into the **Offset Distance** entry if you need to pull back your slab.



- Click **Apply**.

Once you click **Apply**, the entire slab edge will pop out by two feet. If you still have your DXF drawing grid in the background, you can clearly see the difference.



- Click **Undo**  once to return to the original slab edge.

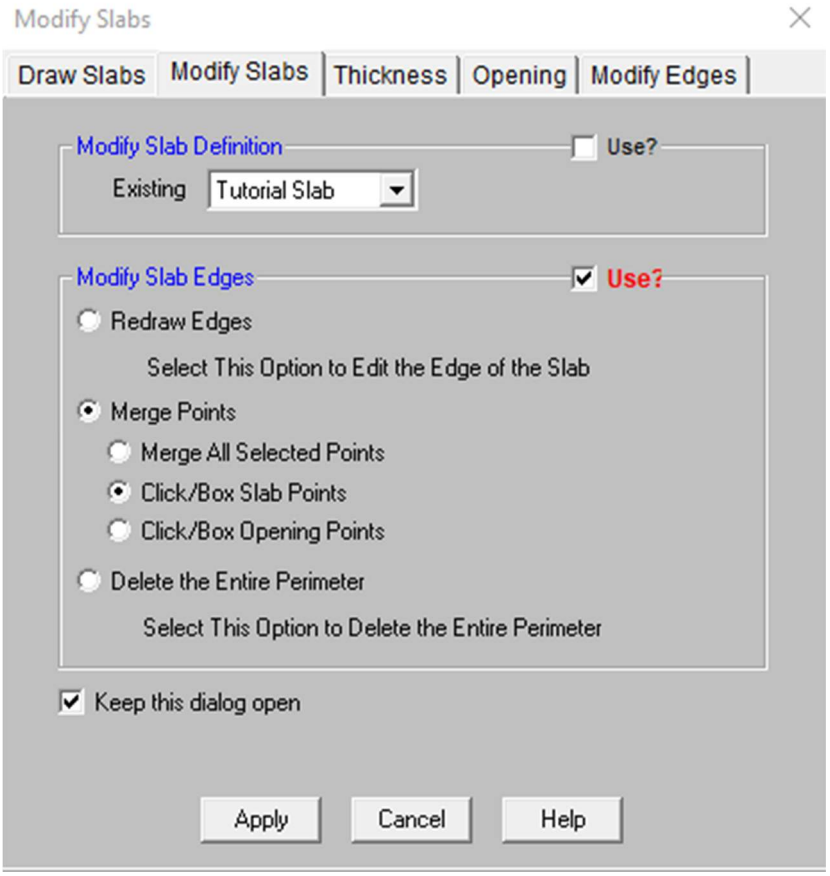
Modifying the Slab Edges

If you need to move a single side of your slab edge, or change them one at a time, you can use the **Modify Slab Edges** tool. We will test out a few of these options.

- Click on the **Create Slabs** button from the Drawing toolbar.
- Click on the **Modify Slabs** tab.

This feature allows you to select several points along a slab edge and "merge" them together onto one straight line. This is best understood by seeing it work, so you will test it out next.

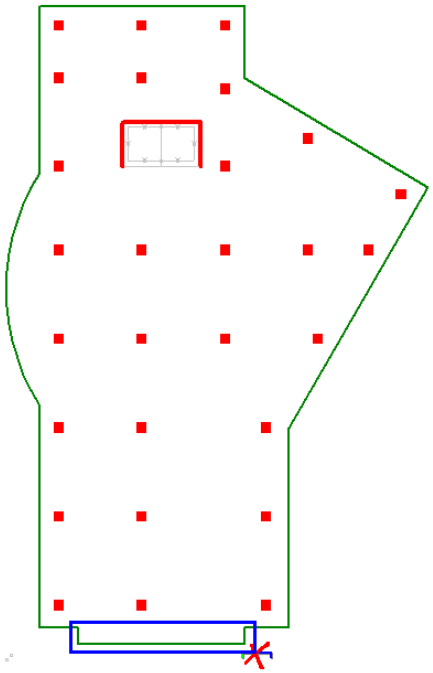
- Under Modify Slab Edges, select the Merge Points option.
- Under Merge Points, select Click/Box Slab Points.
- Select the **Use?** checkbox.
- Be sure to select the **Keep this dialog open** checkbox and then click **Apply**.



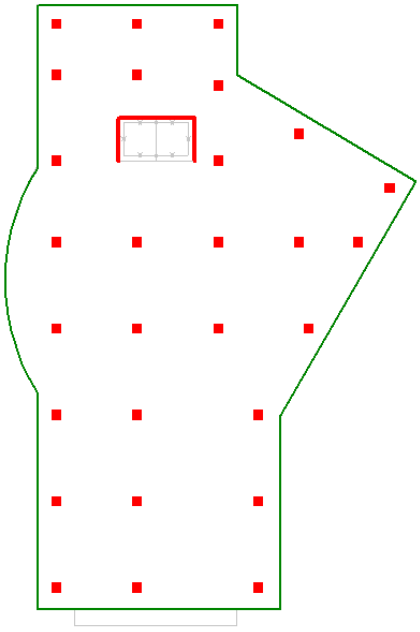
You may need to move the **Modify Slabs** dialog out of the way while working. You can simply "grab" it and move it by clicking on the title bar and holding your left mouse button.

Note: Your cursor in the graphical model view will now look like this to indicate that you are in modification mode.

- Box select the lower slab edge as shown here:

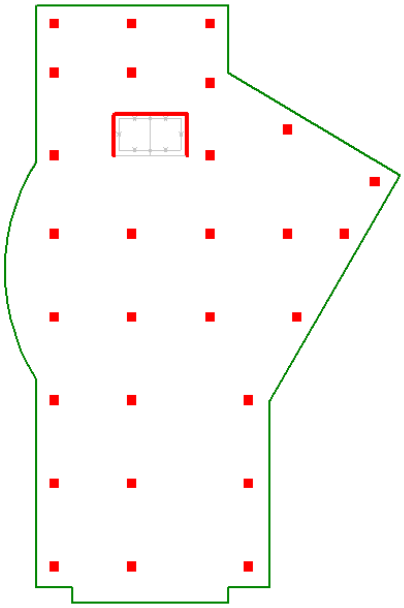


When you let go of your cursor, you will see the edge of the slab pops in and all the points align in a straight line:



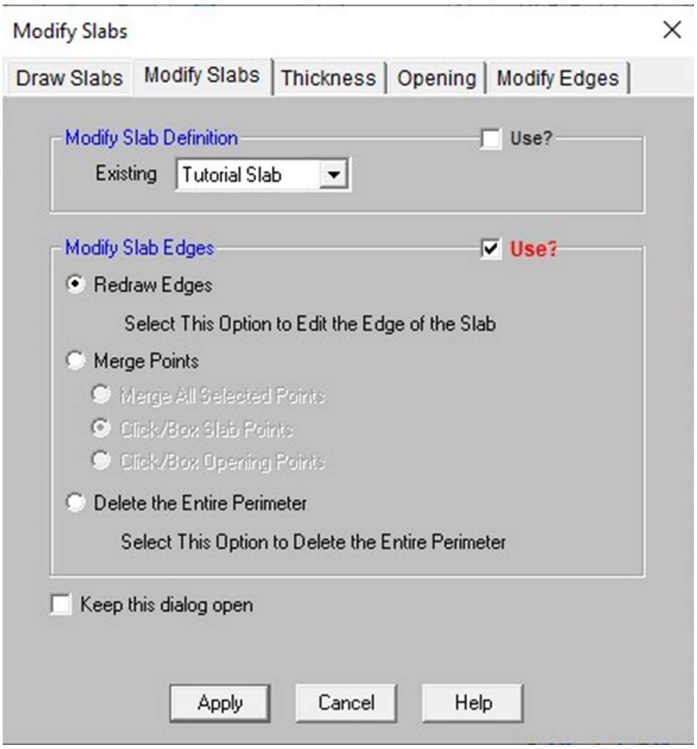
- Click ESC on your keyboard or right-click your mouse to escape the modification tool.
- Click **Undo** once to return to our original slab edge.

Your model should look like this again:



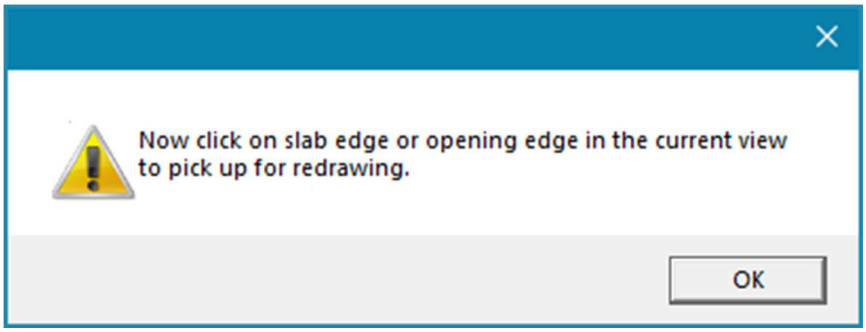
Next, test out the **Redraw Edges** option.

- Return to your **Modify Slabs** dialog, which should still be open. If not, press CTRL+D to reopen the last dialog.
- Under **Modify Slab Edges**, select **Redraw Edges**.
- Uncheck the **Keep this dialog open** checkbox.



- Click **Apply**.

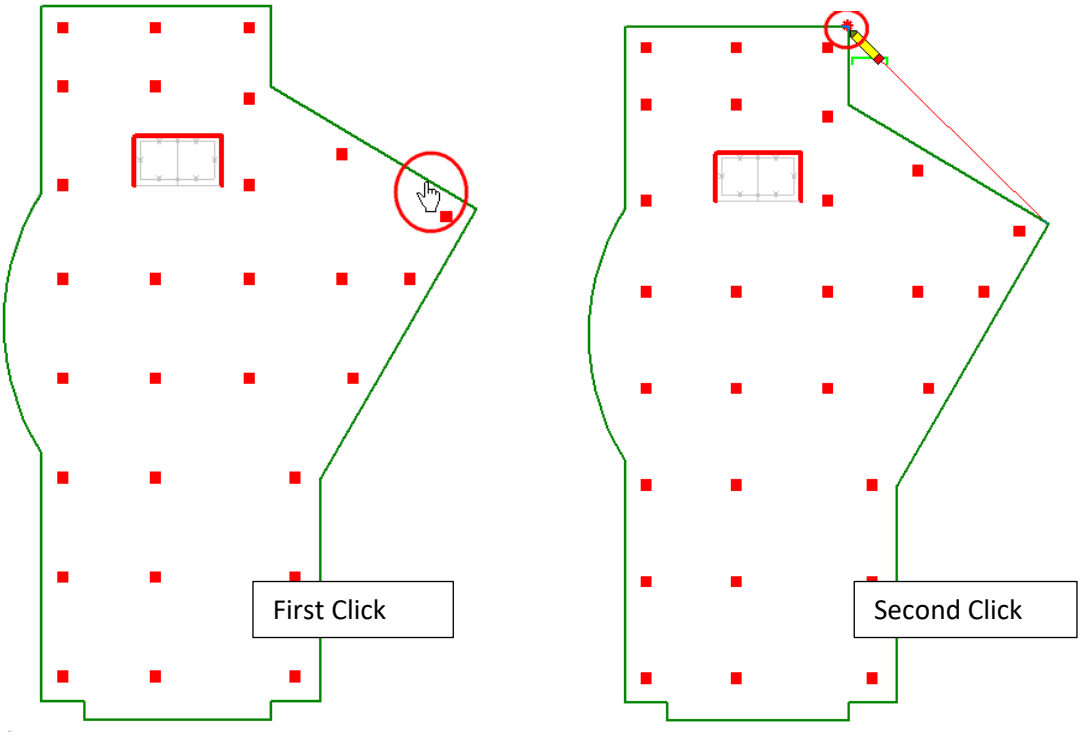
Once you click **Apply**, this instructional dialog will open:



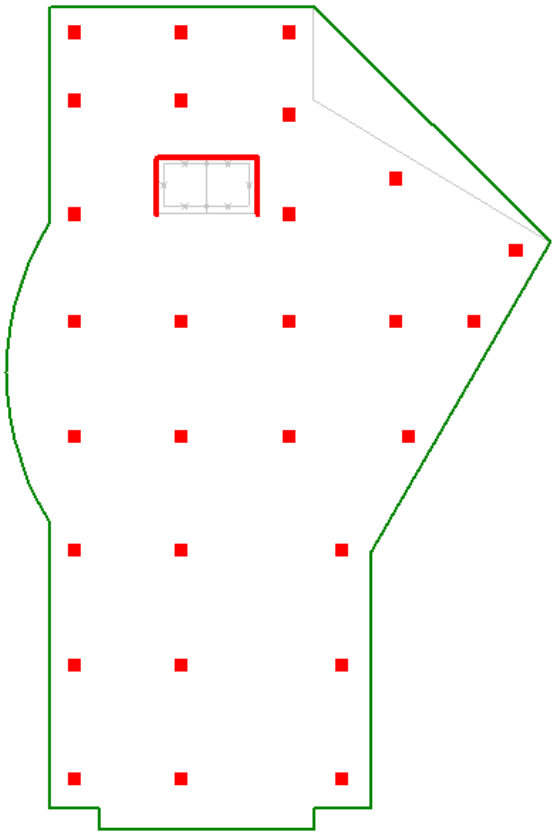
- Click **OK** to continue.

Note: Your cursor in the graphical model view will now look like this to indicate that you are in modification mode.

- Click first on the upper edge of the slab and then click on the point to re-draw the edge.



When finished, your slab edge will look like this:



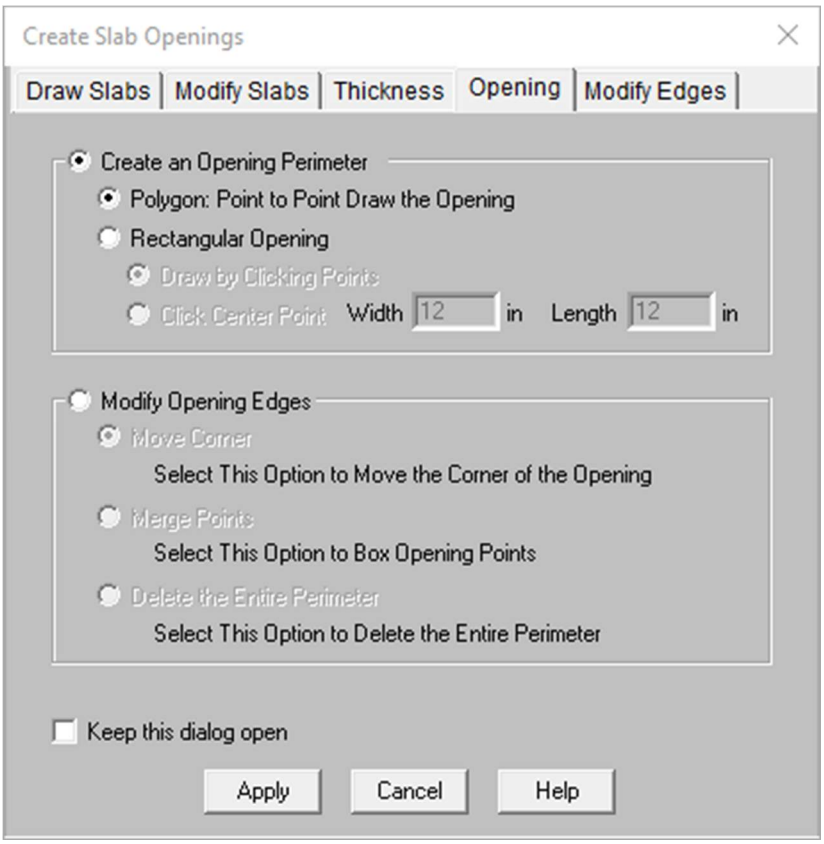
Undo this step to return to the original slab:

- Right-click your mouse to cancel the modification tool.
- Click **Undo** once to return to our original slab edge.

Creating a Slab Opening

Next, you will add an opening to the slab floor.

- Click on the **Create Slab Opening** button from the Drawing toolbar.
- Select Polygon: Point to Point Draw the Opening.



- Click **Apply**.

Note: Your cursor will change to to indicate that you are now in drawing mode.

Since the opening is not on the DXF drawing grid background, turn on the Universal Snap Points to help you draw in the opening.

- Click on the **Universal Snap Points** button from the Drawing toolbar.

Draw in the opening:

- Using the cursor coordinate information on the right-hand side of the Status Bar, draw in the opening by clicking on the following points:

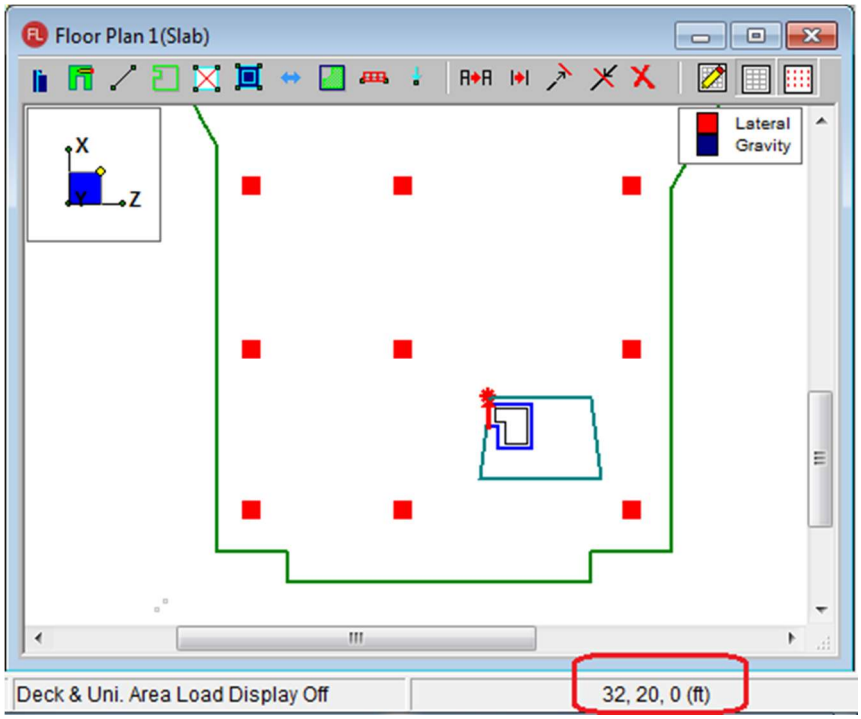
(32,20,0)

(42,20,0)

(43,12,0)

(31,12,0)

Once again on **(32,20,0)**



Note:

You can draw an opening polygon of any shape.

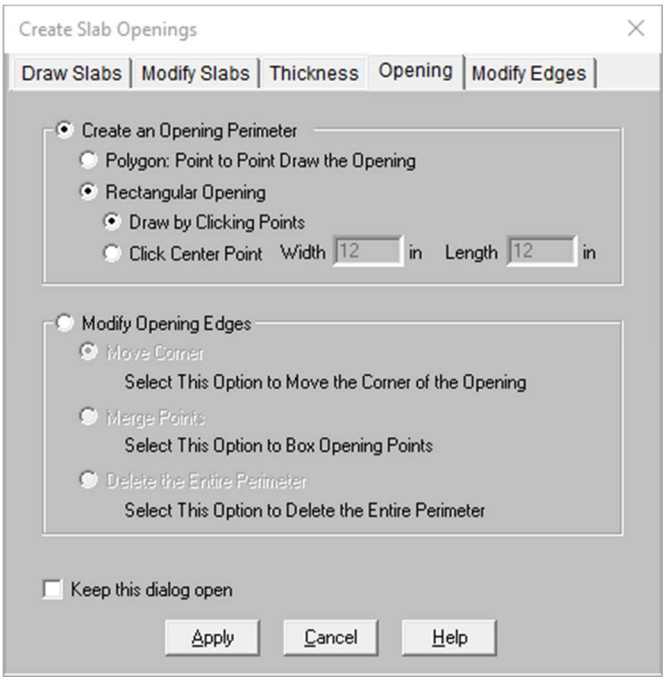
Zoom in on the lower portion of the model for an easier view.

Once you have drawn in the opening, be sure to turn off the Universal Snap Points and the Drawing Grid.

- Unselect the **Universal Snap Points** button from the Drawing toolbar.
- Unselect the **Drawing Grid** button from the Drawing toolbar.

Next, you will draw in the elevator shaft opening.

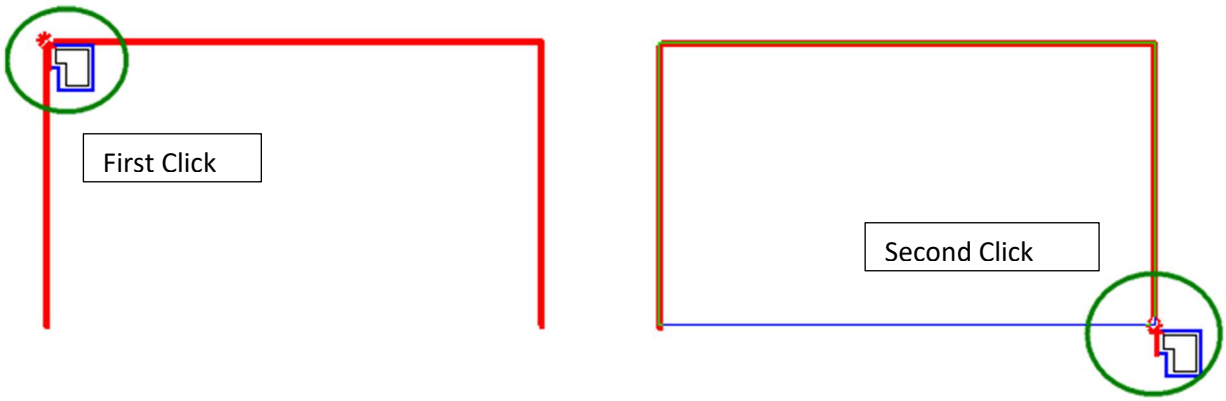
- Click CTRL+D to re-open the **Create Slab Openings** dialog box.
- Select Rectangular Opening and Draw by Clicking Points.



- Click **Apply**.

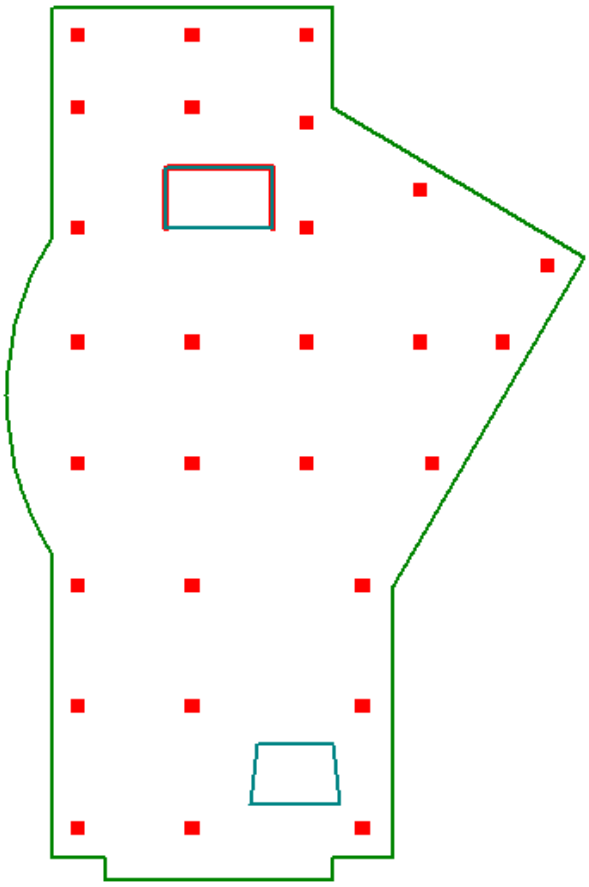
To draw in the rectangular opening, you only need to select two opposite corner points.

- First, zoom in on the upper portion of the model.
- Click on the upper left corner of the elevator walls.
- Pull the opening rectangle down to the lower right corner and click on the end of the wall at this location.



- Right-click to release the opening tool.

When complete, the model will look like the following image.



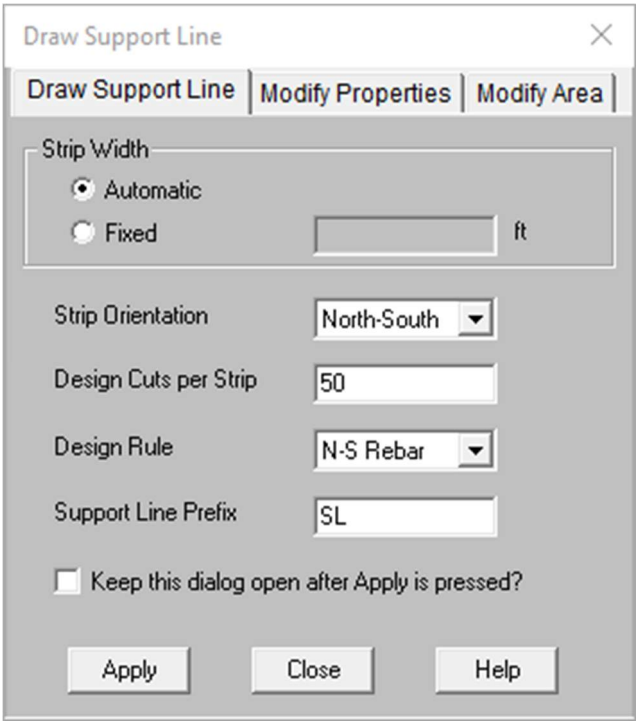
Support Lines and Design Strips

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 judgement is used* to determine an appropriate width for the design strip.

You define design strips with the use of support lines, and you can either manually enter a fixed width for each design strip, or have the program automatically generate strips based off of your support line locations. In this tutorial you will use the automated generation and then modify the design strips as necessary.

- Click on the **Support Lines** button 

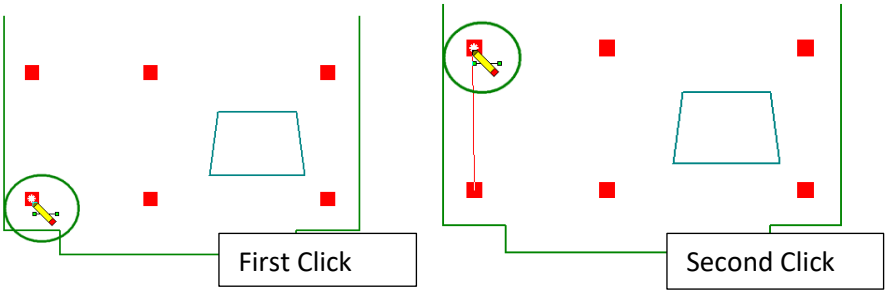
on the left side of your screen.
- Under **Strip Width**, select **Automatic**.
- Under **Strip Orientation**, select **North-South**.



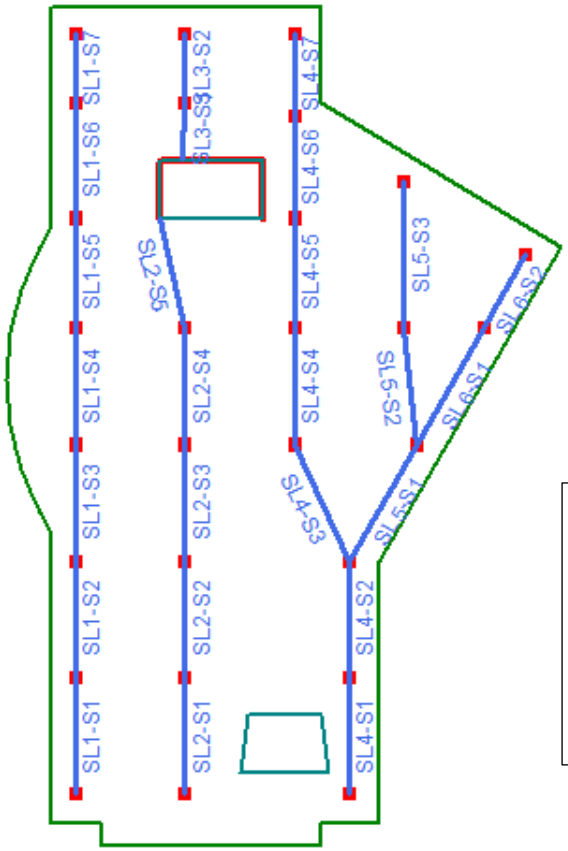
- Click **Apply**.

Note: Your cursor will change to  to indicate that you are now in drawing mode.

- Draw in your support lines by clicking column to column. Note that you can click over existing columns and the support line will break at the column as long as it's collinear with your line. Start by clicking on the red lateral column in the lower left corner.



- Continue to draw in all the **Support Lines** as shown below.



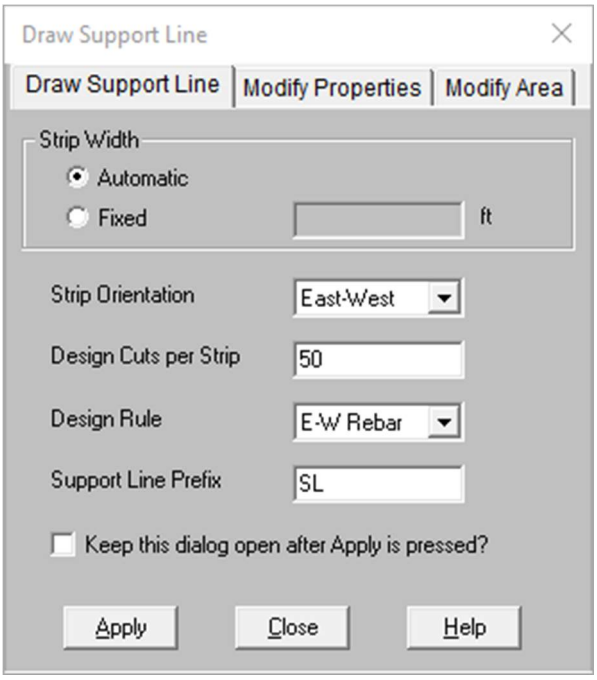
Note: After you draw a support line, your cursor is "anchored" to the last column you clicked on. To release the drawing tool, use your right mouse (right-click) or ESC.

Before drawing the East-West Support Lines, toggle off the view of the North-South Lines for a simpler view.

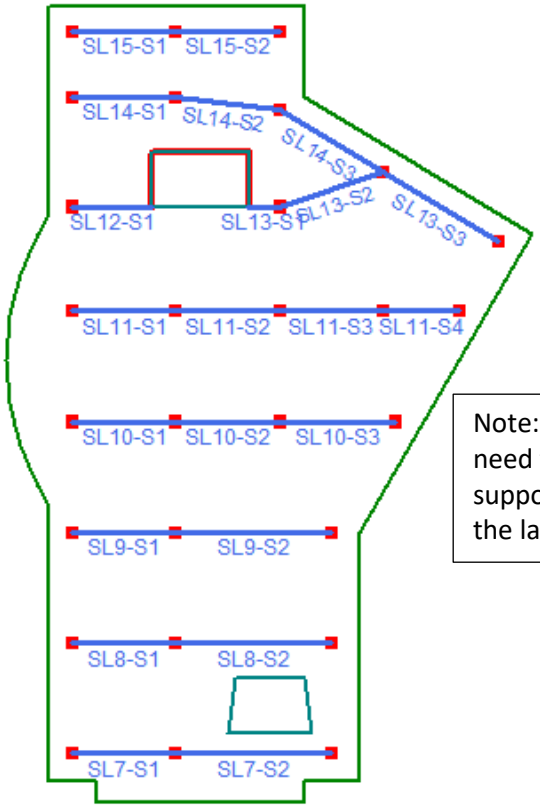
- On the Window toolbar, click on the black arrow next to the **Design Strip** button  (upper right corner).
- Select E-W Support Line.

The North-South Lines will be toggled off.

- Return to the **Draw Support Line** dialog by clicking CTRL+D.
- Change the Strip Orientation to East-West and the Design Rule to E-W Rebar.



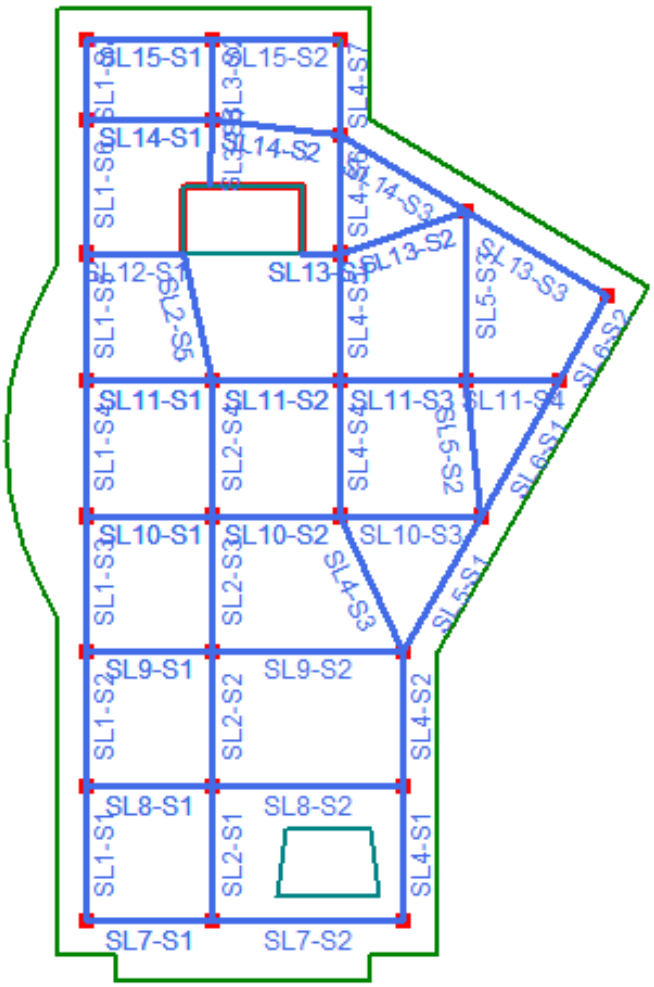
- Click **Apply**.
- Draw in all the **Support Lines** as shown below.



Note: There is no need to draw support lines over the lateral beams.

Toggle back on the display of all Support Lines.

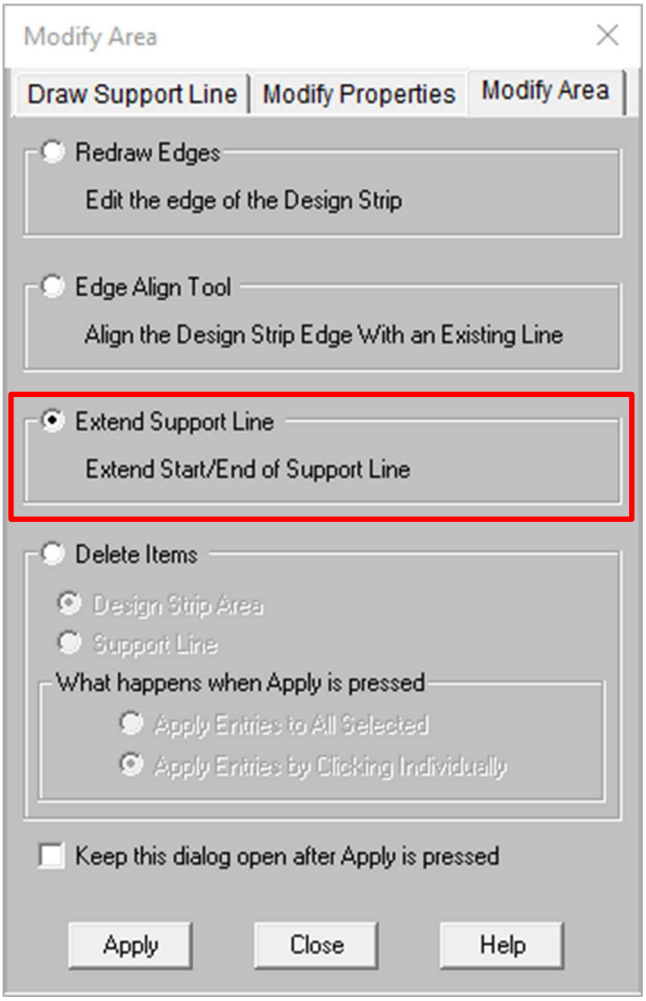
- On the Window toolbar, click on the black arrow next to the **Design Strip** button  .
- Select **All**.



Note: The Support Line labels may vary depending on the order in which they were input, so it is okay if your lines do not exactly match those shown above.

Since there are some significant cantilevered portions of the slab floor, you will next extend some of the Support Lines out to the edge of slab.

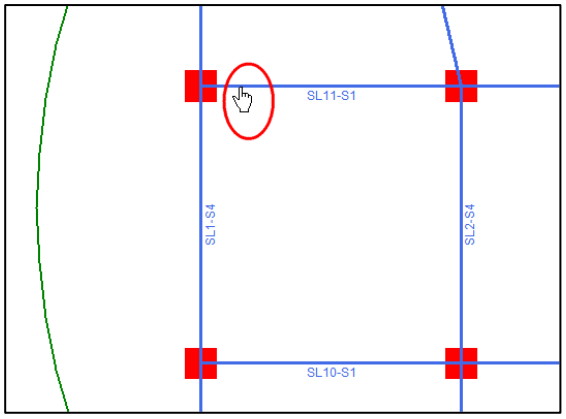
- Return to the **Draw Support Line** dialog by clicking CTRL+D.
- Click on the **Modify Area** tab.
- Select Extend Support Line.



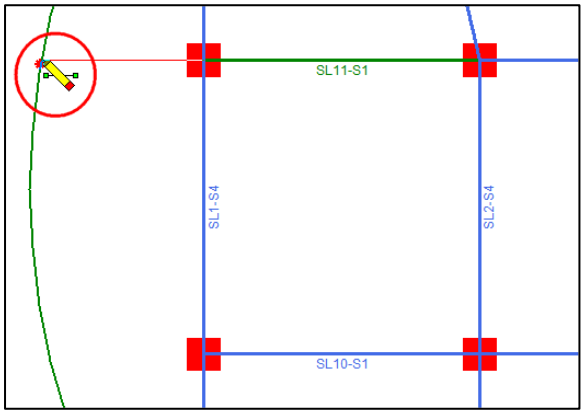
- Click **Apply**.

To extend a Support Line, you just click on the end of the Support Line that you want extended, and then select the edge location that you want to extend it out to. You will do this next.

- First click on the left side of the **E-W (horizontal) Support Line** near the curved cantilevered portion of the slab.
- Next, click on the edge of the slab.

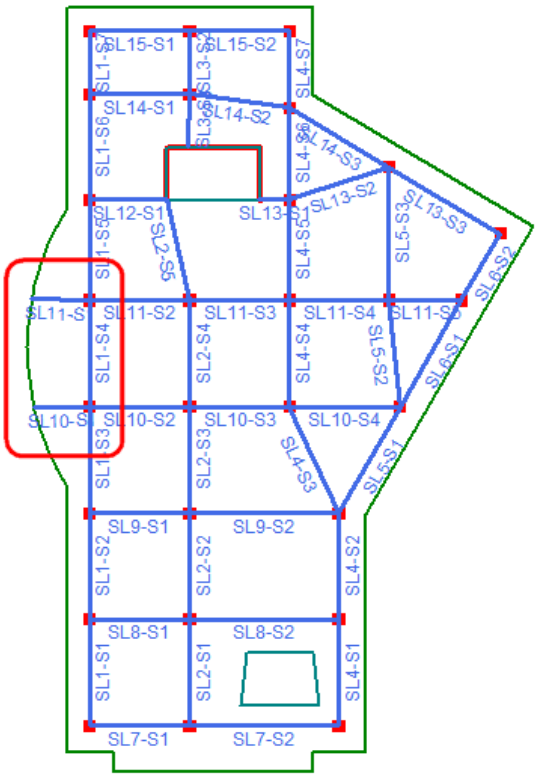


First Click



Second Click

- Repeat for the other **E-W Support Line** that extends over the curved cantilevered portion of the slab.



Generate Design Strips

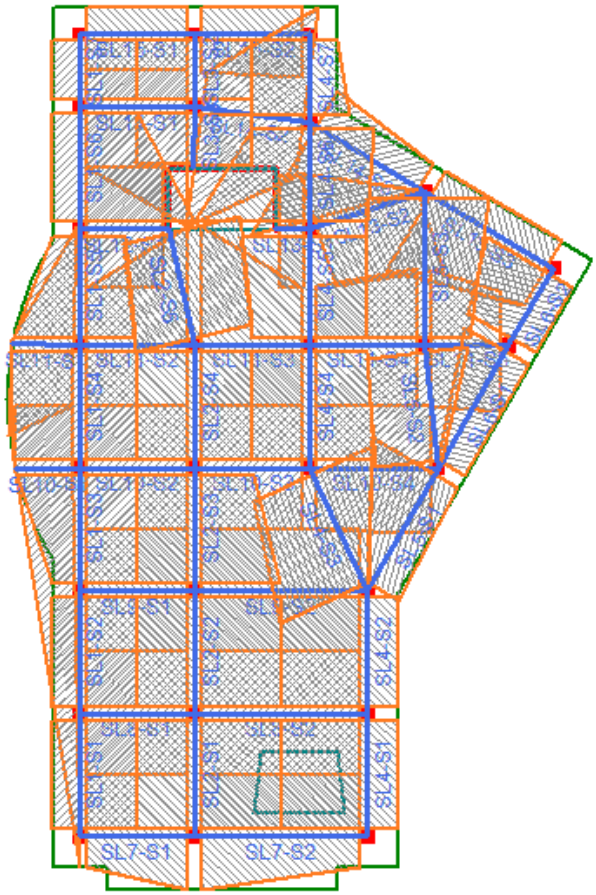
Now that you have laid out Support Lines between your columns and walls, you will use the automated **Design Strip Generator** to quickly define your design strips.

- Click on the **Generate Strips** button 

on the left side of your screen.

Note: if you had used the Fixed Support Line width option, you would not need this step.

You should now see the automated **Design Strips** displayed in orange:

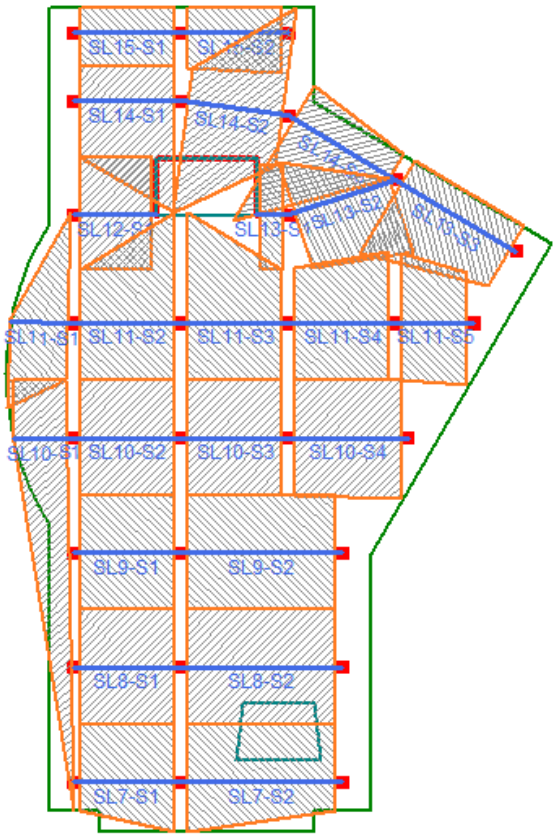


Note: Your design strips may look slightly different depending on how the support lines were drawn.

As seen above, there are some locations where the auto-generated Strips are overlapping or askew, therefore you will use the **Modify** tool to manually adjust their edges.

To make this easier to see, select just one set of Design Strips to view at a time.

- Click on the black arrow next to the **Design Strips** button  on the Window toolbar.
- Select **E-W Design Strip** and you will now see just the **E-W Support Lines** and **Design Strips** displayed.



You will now manually edit the **Design Strip** edges.

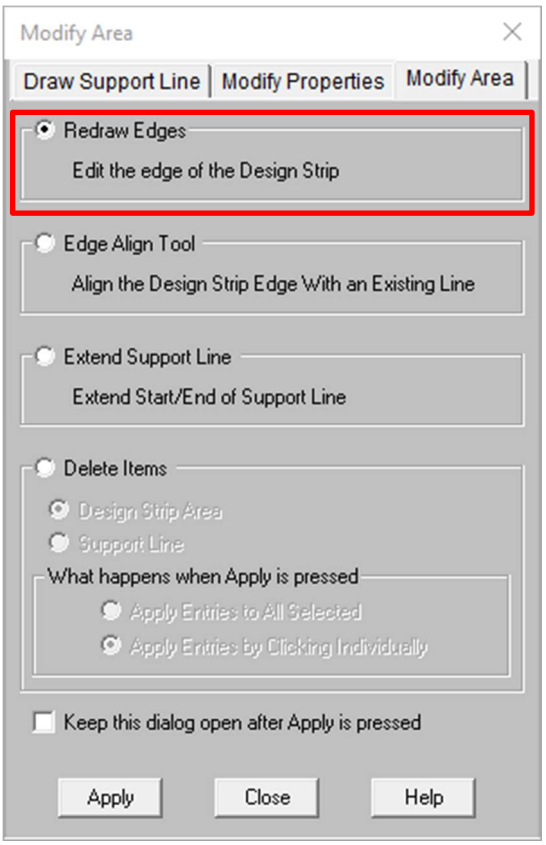
- Click on the **Support Line** button 

.
- Click on the **Modify Area** tab.

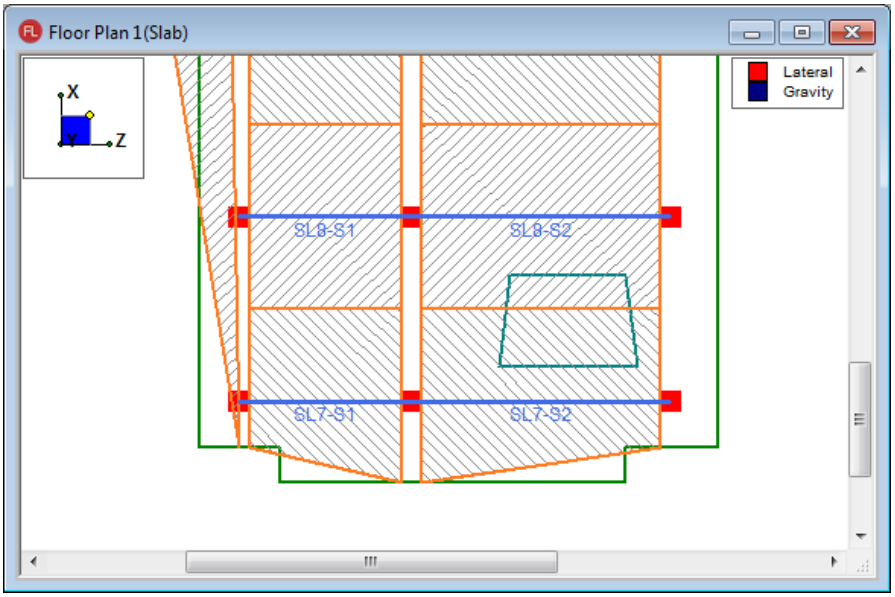
Redraw Edges

The first option within this dialog is the **Redraw Edges** option. Using this will allow you to select the edge of one of the automatically generated **Design Strips** and redraw it.

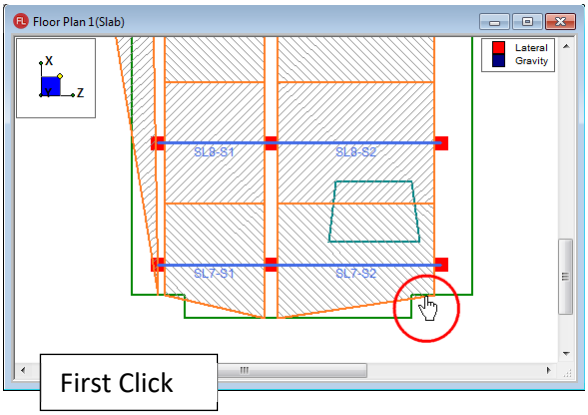
- Select Redraw Edges.

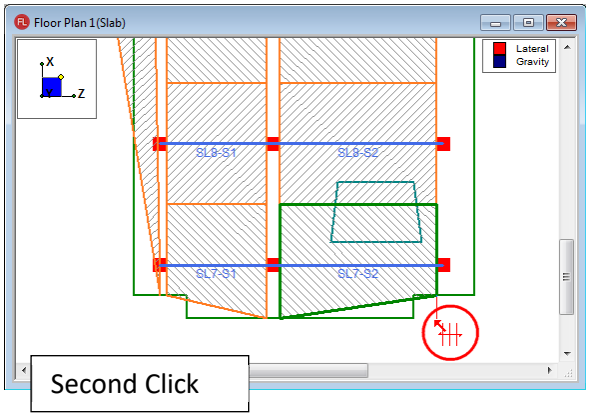


- Click **Apply**.
- Zoom in on the bottom of the building, as shown below.

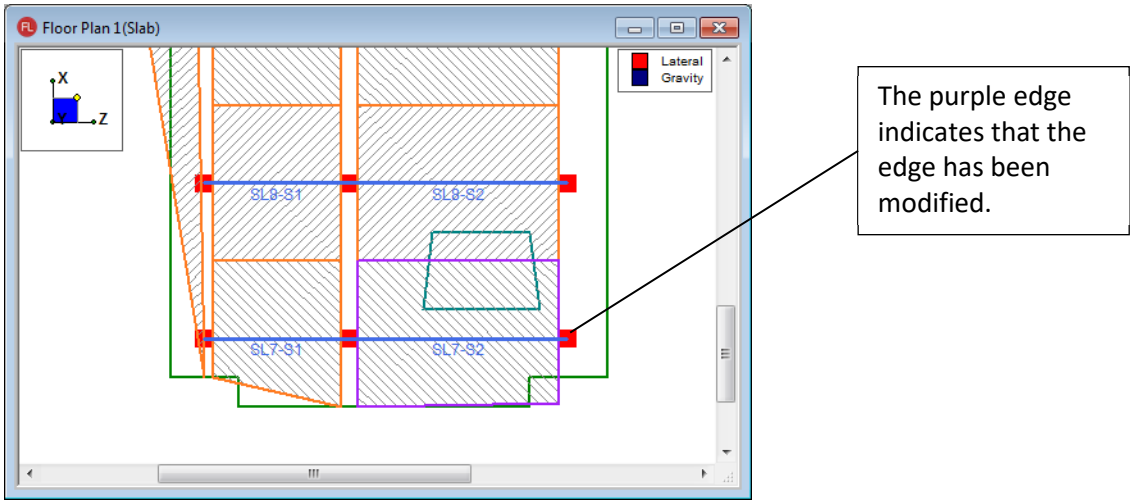


- First click on the **Design Strip** edge and then select the new location to "pull" the edge line out to:





When finished the **Design Strip** edge will turn purple to indicate that this design strip was manually modified.

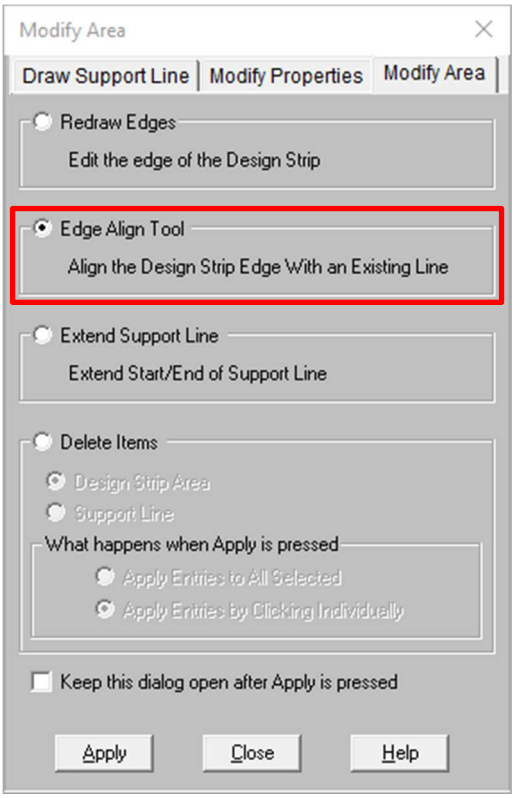


Note: As shown in the example above, it is okay to extend your design strip over the edge of your slab. Only the portion within the slab edge will be considered for design.

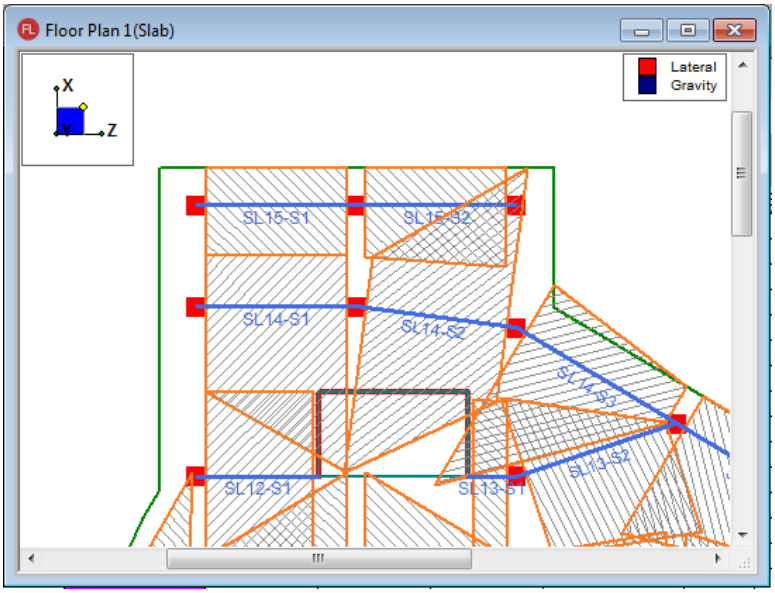
Edge Align Tool

Next, you will use the **Edge Align Tool** to modify some of the other Strips. This tool will allow you to redefine a **Design Strip** edge by simply snapping it to a nearby line.

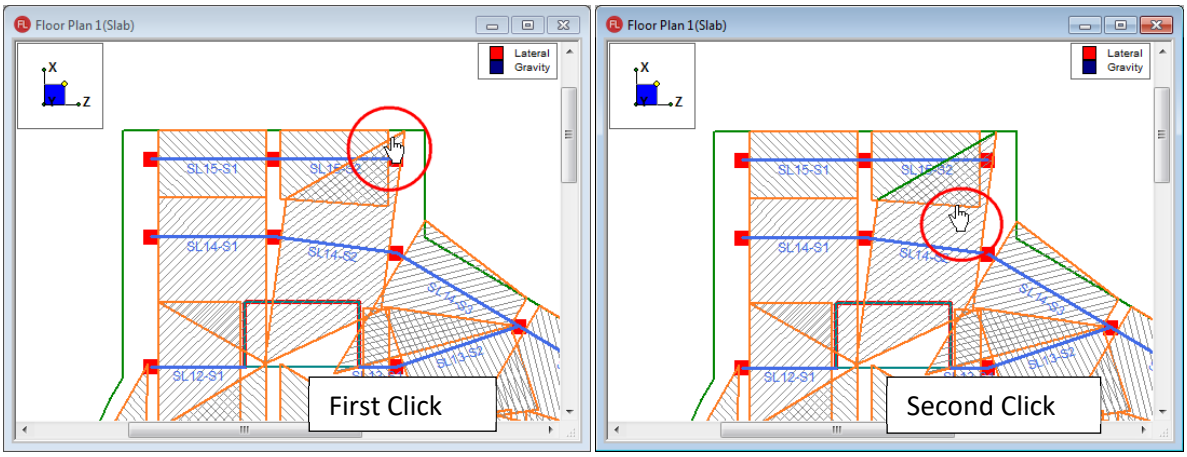
- Return to the **Modify Area** dialog by clicking CTRL+D.
- Select the Edge Align Tool.



- Click **Apply**.
- This time zoom in on the upper edge of the model, as shown below:

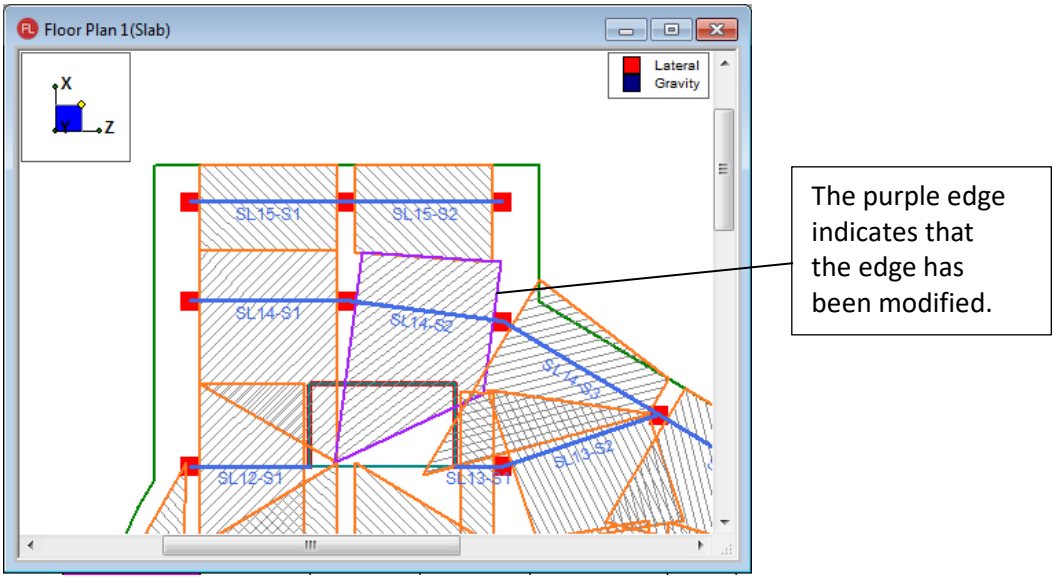


- First click on the **Design Strip** edge and then the line that you want to align with.



Note: Notice that after your first click, the selected edge highlights in green to show what is selected.

When finished your model should look like this:



Delete Items

You will not use this in the tutorial. However, should you ever need to delete a support line or a design strip you can use this tool to do so.

Note: To go through the modifications for each Design Strip may be a little time consuming. Feel free to go through them yourself (remember to do both the North-South and the East-West strips) or if you would like to save time, you can jump ahead to the **Tutorial B3 Starter.rfl** starter file in which these have been done for you.

Design Cuts

Design Cuts may be used to examine an area of interest and may also be used to double check Design Strip results. While Design Strips automatically contain Design Cuts, you may also draw Design Cuts individually.

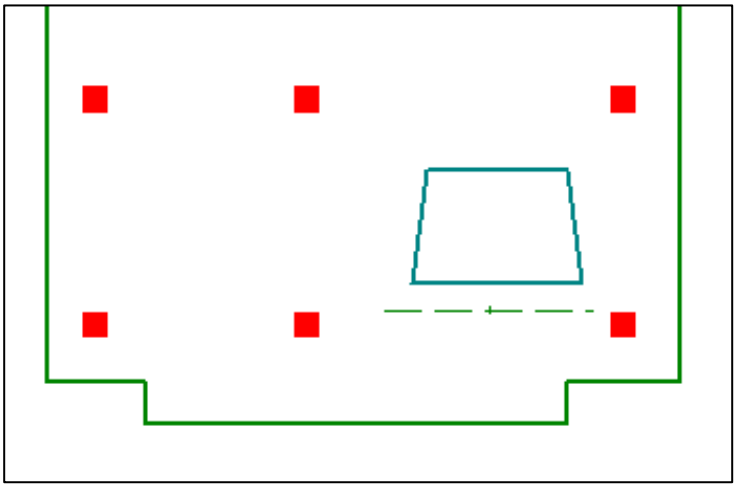
Before drawing the **Design Cut**, turn off the display of the **Design Strips**.

- Click on the black arrow next to the **Design Strips** button  on the Window toolbar.
- Select **None.**

Draw a design cut near the opening to investigate that particular area of interest.

- On the Selection toolbar, click **Design Cut**  . Your cursor will change to the saw tool  .
- Click on the **Universal Snap Points** button  from the Drawing toolbar to help draw. It will show as indented in when turned on.
- Draw in a **Design Cut** near the opening (the exact location does not matter).

Notice a green line appears showing the location of the design cut (you may have to zoom in to see it).



Note: If you skipped ahead to the **Tutorial B3 Starter.rfl** starter file, this step was already completed for you.

This is the end of Tutorial B2.

If you are using the Demonstration version of the program, you will not be able to save this model, as it exceeds the Demonstration version limits. Please proceed to the next tutorial for instructions on how to open the Tutorial B3 Starter File provided by RISA.

If you are using the full version of the program, you may save by doing the following: On the **File** menu, click **Save As** and enter a file name.

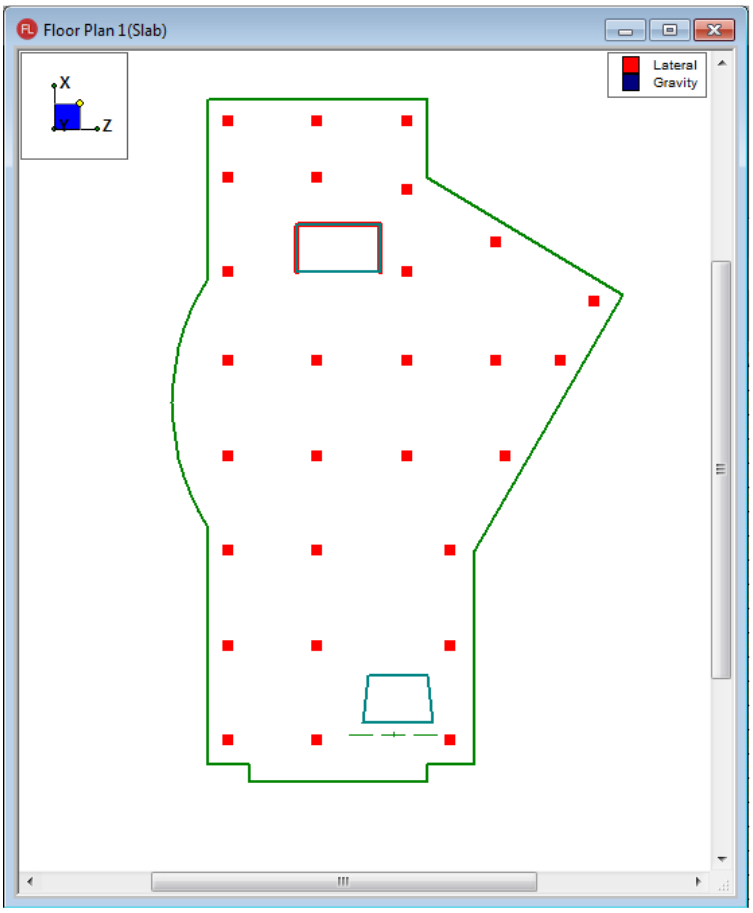
Part B: Tutorial 3 – Loading

This tutorial will guide you through adding and editing the various types of loads available in RISAFloor ES.

This tutorial continues from where the previous tutorial ended, so follow these steps to open the Tutorial B3 Starter File provided by RISA:

- Double-click on the **RISAFloor** icon to start the program.
- Click **Open Model**  . Double-click the **Tutorials** folder, select **Tutorial B3 Starter.rfl** and click **Open**. Click **Close**  (or **Cancel**) to exit the **Model Settings** dialog box.
- On the Window toolbar, click the **Graphic Editing** button  to activate the Drawing toolbar.
- Turn off the view of the **Design Strips** by clicking on the **Design Strip** button  on the Drawing toolbar. Click on the black menu arrow and select **None**.

Your screen should now look like this:

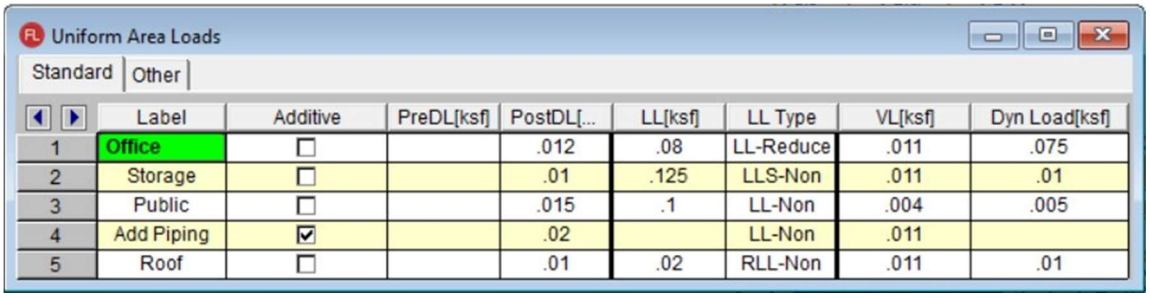


Area Loads

Area load magnitudes are defined in a spreadsheet and then applied to the floor as polygons. Area loads that are within the slab edges (but not inside an opening) are automatically attributed to the supporting elements based on the load Direction set in the Slab Definition.

Edit the **Area Loads** spreadsheet, as shown below:

- On the Data Entry toolbar, click **Area Load Definitions**.
- Edit the load information to match what is shown in the image below:



	Label	Additive	PreDL[ksf]	PostDL[...	LL[ksf]	LL Type	VL[ksf]	Dyn Load[ksf]
1	Office	☐		.012	.08	LL-Reduce	.011	.075
2	Storage	☐		.01	.125	LLS-Non	.011	.01
3	Public	☐		.015	.1	LL-Non	.004	.005
4	Add Piping	☑		.02		LL-Non	.011	
5	Roof	☐		.01	.02	RLL-Non	.011	.01

The entries in the **Area Loads** spreadsheet are described below:

Label	The name you will later use to refer to the load when you apply or view it.
Additive	If this is checked, the load applied will add to the load that is drawn beneath it in the order of how they were drawn in the model.
PreDL PostDL LL LL Type	These entries allow you to specify load magnitudes for different categories that will later be used in the load combinations that you solve. ▪ PreDL and PostDL are for dead loads that are pre- and post-composite. Remember that other dead loads (deck self-weight and construction dead load) were already defined within the deck definitions spreadsheet. ▪ LL and LL Type load columns are for the live load magnitude and the type of live load which you may choose from a list.
VL	Specifies the vibration load. The vibration load (along with the self-weight of the beam and deck) will be used for calculating the mass for a vibration check, per *AISC Design Guide #11*.
Dyn Load	The dynamic load will be used to determine the diaphragm mass and mass moment of inertia for a RISA-3D seismic analysis.

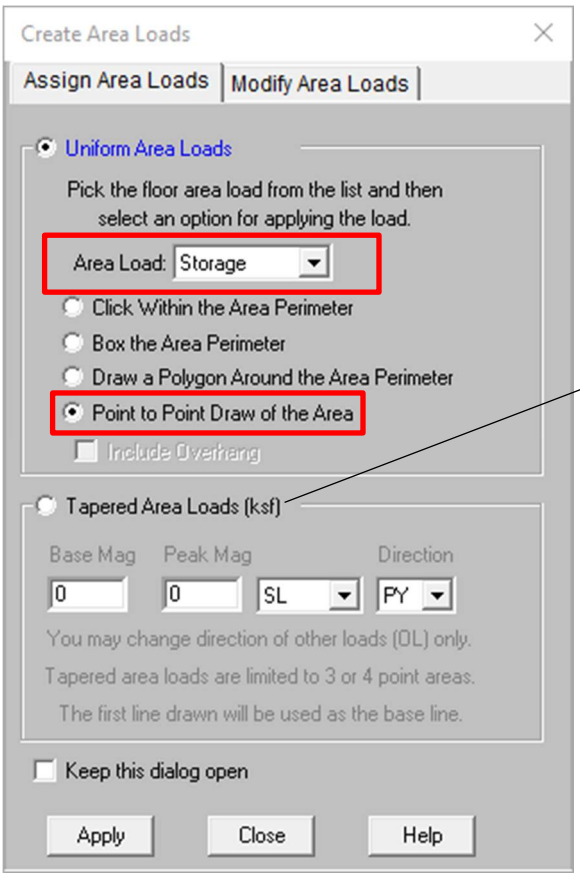
Each floor has a default area load that will be used within the entire floor slab if no other load is specified. On the **Data Entry** toolbar, click on **Floors** to see what defaults are applied to each floor. To use different loads in an area, you can graphically draw on local area loads.

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

Drawing Area Loads

To apply local area loads:

- On the Drawing toolbar, click the **Area Loads** button .
- Under Uniform Area Loads, in the Area Load box, select Storage, then select the Point to Point Draw of the Area drawing option.



Notice you also have the option of drawing in a tapered load. This can be useful for modeling certain types of snow loads but will not be used in this tutorial. For more information on tapered area loads please refer to the *RISAFloor General Reference*.

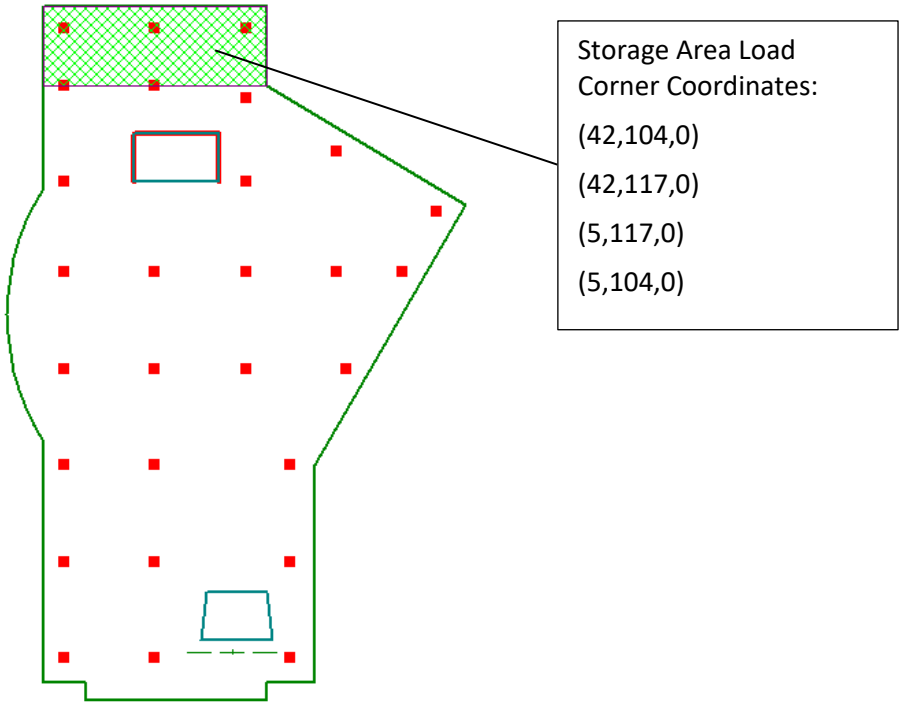
- Click **Apply**.

Note: The cursor will change to .

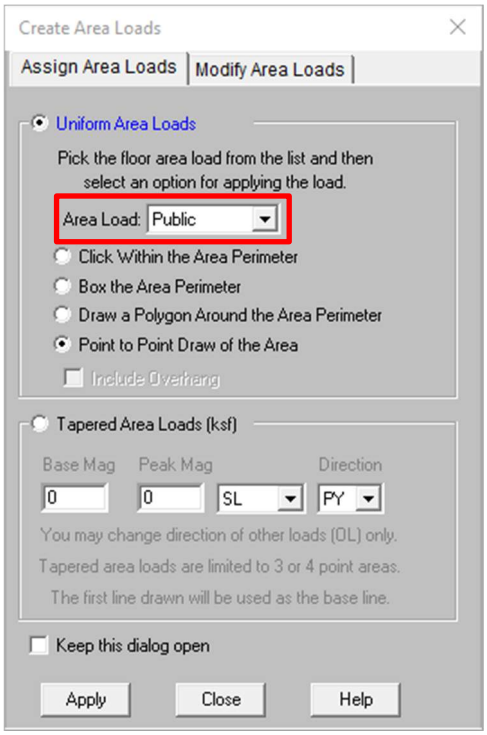
Before drawing, turn on the **Universal Snap Points**.

- Click the **Universal Snap Points** button from the Drawing toolbar.

- Draw the polygons for the area load by clicking on the drawing grid snap points, as shown in the following figure.

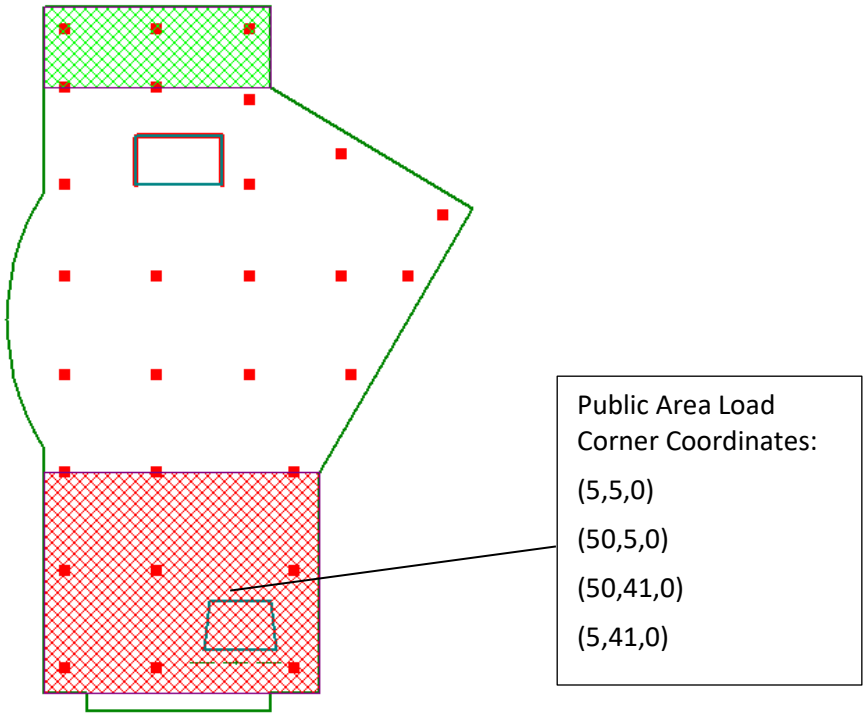


- Click on **Area Loads** button  to reopen the **Create Area Loads** dialog box.
- Under Uniform Area Loads, in the Area Load box, select Public.

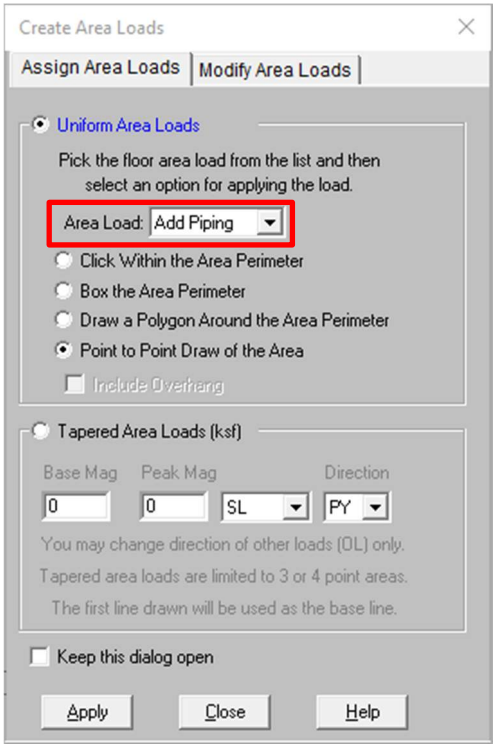


- Click **Apply**.

- Draw in the **Public** area load by clicking on the drawing grid snap points, as shown in the following figure.

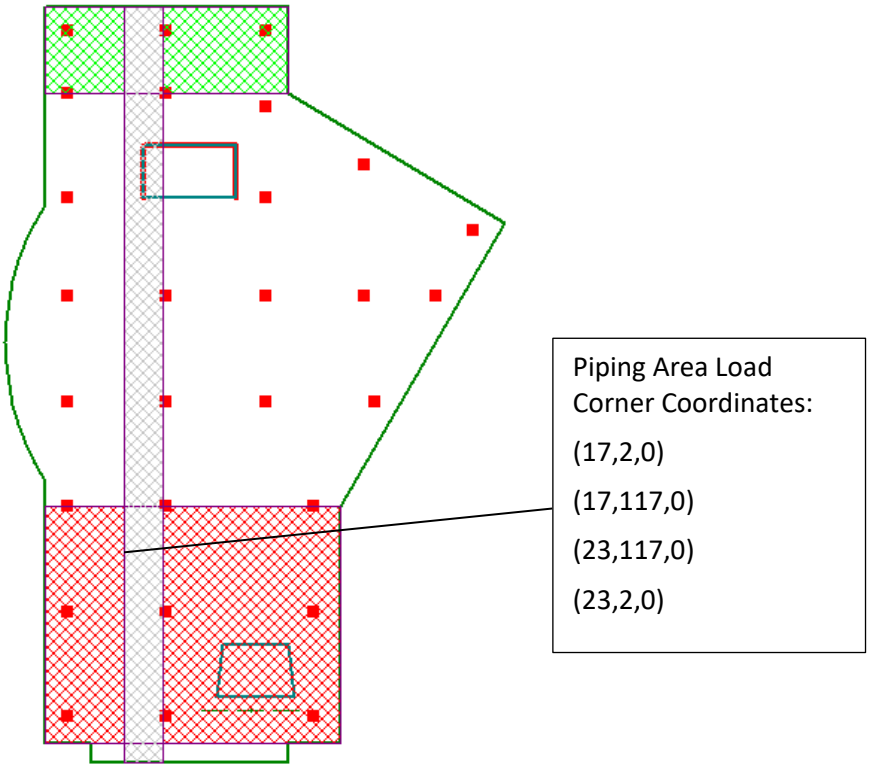


- Press CTRL+D to reopen the **Create Area Loads** dialog box.
- Under Uniform Area Loads, in the Area Load box, select Add Piping.



- Click **Apply**.

- Draw in a **Piping** load by clicking on the drawing grid snap points, as shown in the following figure.



Note: In areas where more than one load is applied on top of another, the top load (the last one drawn) will be used by RISAFloor ES. The exception to this is that any additive load will be applied along *with* the load beneath it.

- Right-click to release the drawing tool.

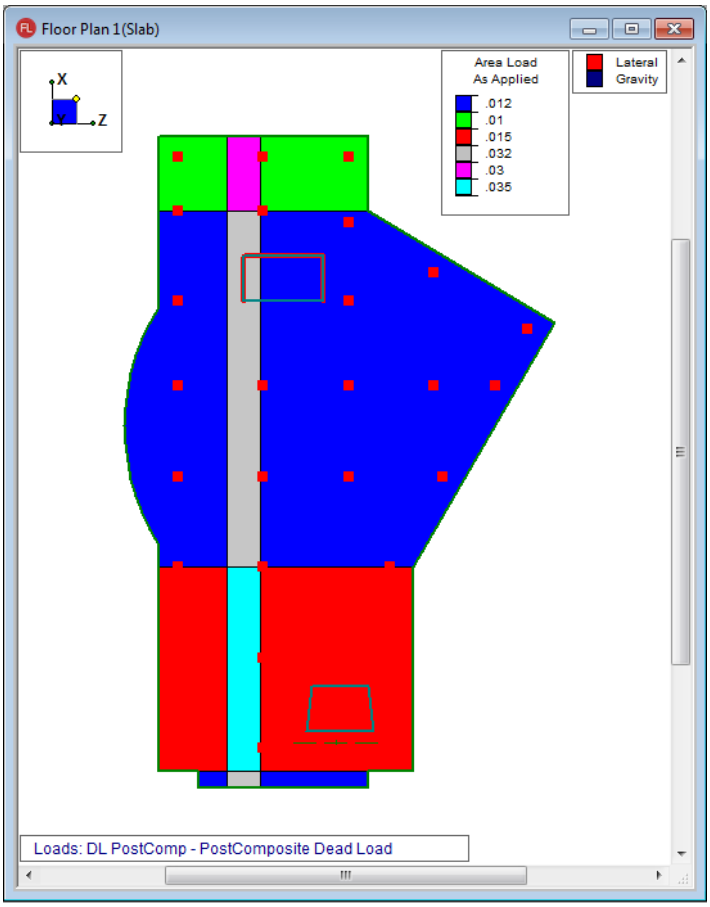
Plotting Area Loads

RISAFloor ES gives you two ways to plot the area loads. The first is to view them as they were input. The second is to view them as they are resolved and applied by RISAFloor ES. You are going to view them "as applied" for this model.

To plot the area loads:

- On the Window toolbar click the black arrow next to the **Area Loads** button  .
- Select the **As Applied**.
- Select the DL PostComp category display from the Load Display option  .

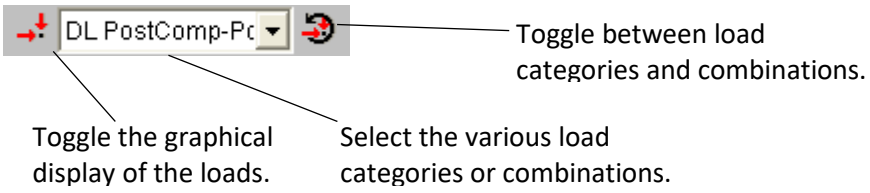
Your screen should look similar to the figure shown below:



Notice that the default **Office** load was applied automatically. Also, note that the additive piping load was automatically combined with the **Office**, **Public**, and **Storage** loads that it crossed.

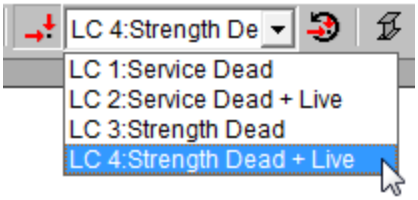
Toggle Load Display

On the Window toolbar, you can use these three controls to manipulate the display of your loads:



Use these buttons to change the display of loads.

- Click the **Switch Load Display** button  on the Window toolbar to change from load category display to load combination display.
- Use the Load menu to select LC 4: Strength Dead + Live.



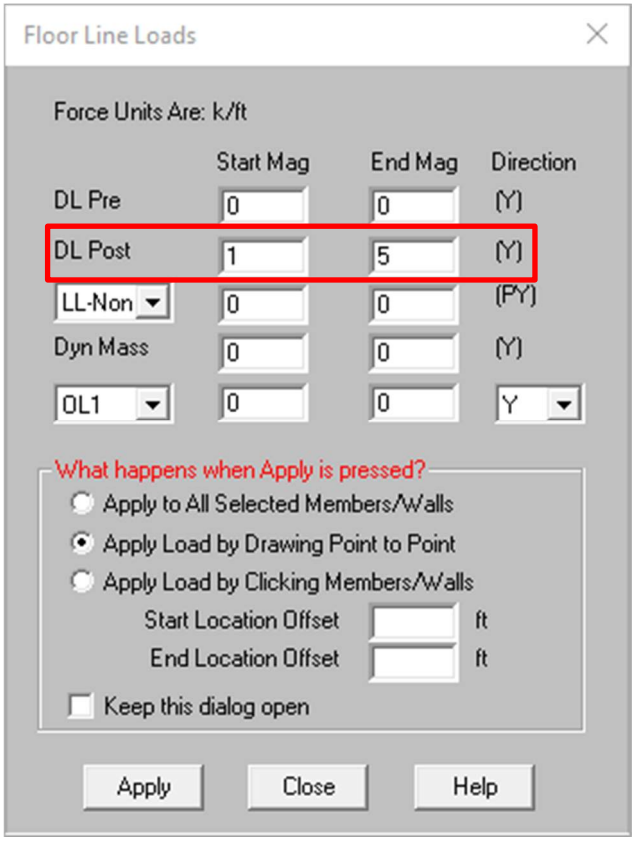
You will see the "As Applied" load magnitudes change to reflect the selected load combination. Scroll through the other load display options. When finished, turn off the display of the area loads, as follows:

- On the Window toolbar click the black arrow next to the **Area Loads** button  .
- Select the **None**.

Line Loads

You can use line loads to model the weight of cladding, partitions, architectural components, etc. Prepare to draw in line loads as follows:

- On the Drawing toolbar, select the **Line Loads** button  .
- Enter the information as it appears in the following figure:

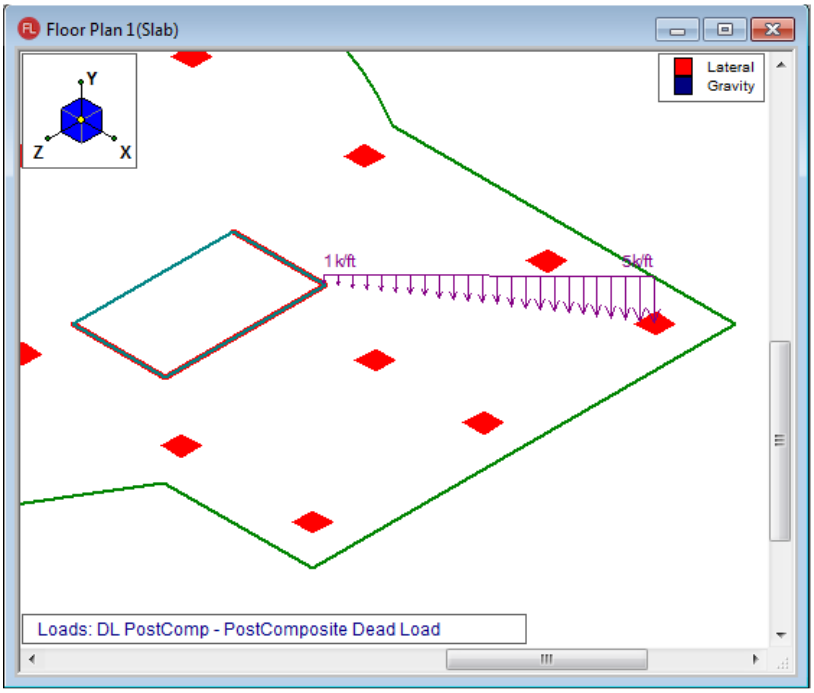


- Click **Apply**.

Note: The cursor will change to  to designate that you are now in drawing mode.

To see the model better while you are drawing in the load, change to an isometric view and zoom in:

- On the Window toolbar, click on the **Isometric** button ISO.
- Click the **Zoom** buttons to zoom in on the right-hand portion of the floor as shown in the next image.
- Draw in the load by clicking on the two points, as shown in the image below (left to right). When finished, right-click to release the mouse.



When you are finished, notice that you have specified a tapered line load to represent a non-structural partition. The line load spans diagonally across the slab.

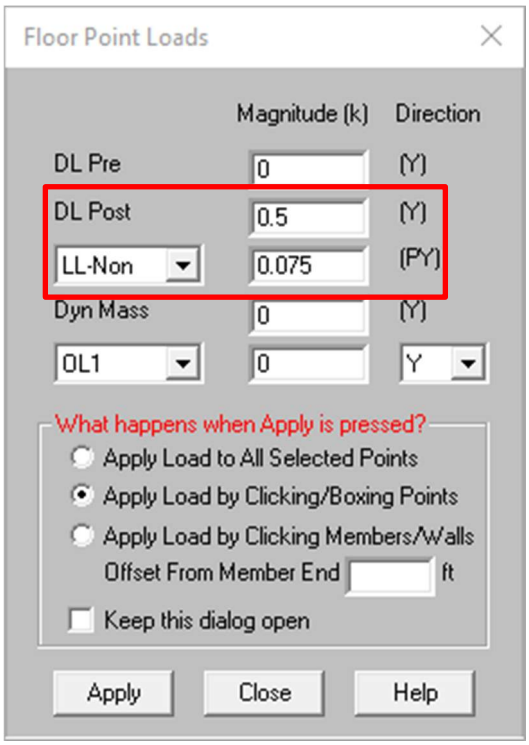
To return to a planar view and continue:

- Use the right mouse button (right-click) or press ESC to exit the drawing mode.
- On the Window toolbar, click the **Plan** button PIn.
- On the Window toolbar, click **Redraw** to return to a full view.

Point Loads

Prepare to apply point loads:

- On the Drawing toolbar, click **Assign point loads** .
- Enter the information show below:



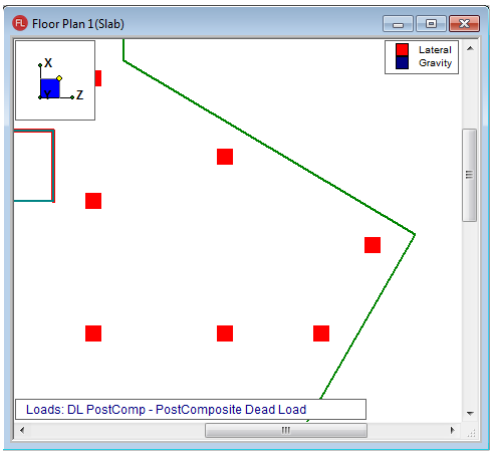
- Click **Apply**.

Note: The cursor will change to to designate that you are now in drawing mode.

Toggle on the **Universal Snap Points** to help you draw the point loads.

- Make sure the **Universal Snap Points** button on the Drawing toolbar is turned on (the button will be depressed).

- Zoom in significantly on the triangular portion of the floor using the mouse roller, as shown below.

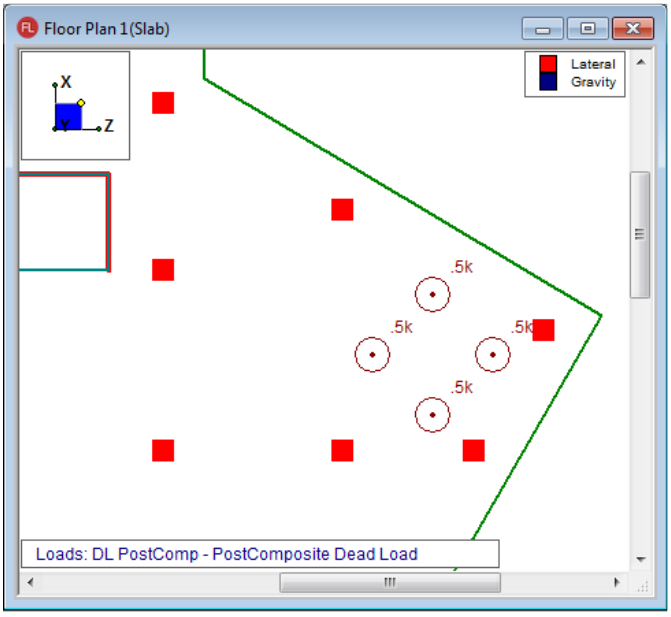


Draw in the point loads:

- Use the coordinates (notice they are displayed in the Status bar) and the snap points to draw in a point load at each of the following coordinates:

  (61,86,0)
  (61,76,0)
  (56,81,0)
  (66,81,0)

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

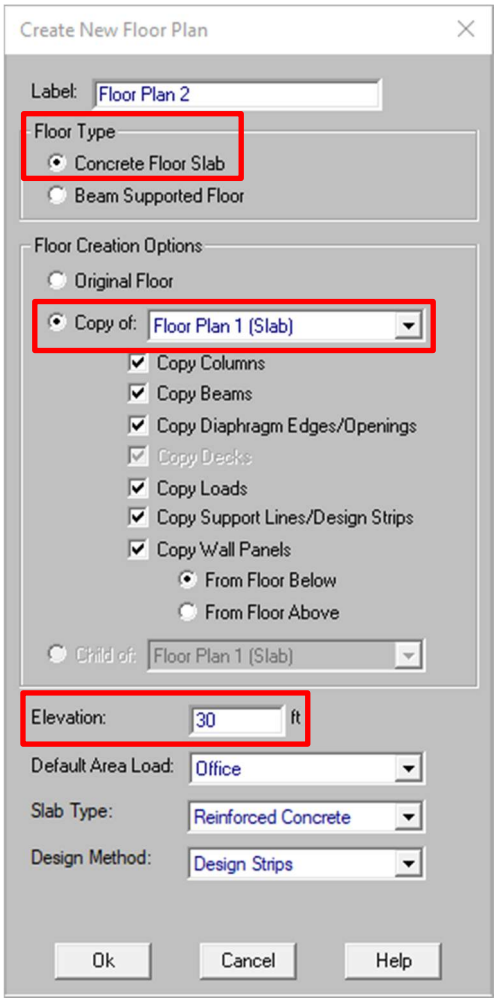


Note: Point loads that are applied inside the deck edges may be applied anywhere on the deck and will be attributed automatically to the beams, walls, and columns.

- Right-click to release the drawing tool.

The last modification step is to create a second floor.

- To create a new floor level, click the **New Floor** button from the RISA toolbar.
- Under **Floor Type**, select **Concrete Floor Slab**.
- Under **Floor Creation Options**, select **Copy of Floor Plan 1**.
- Enter an **Elevation** of **30** feet.



Note: Notice that you also have the option in this dialog to add a new **Beam Supported Floor**. It is possible to have different floor types within one model.

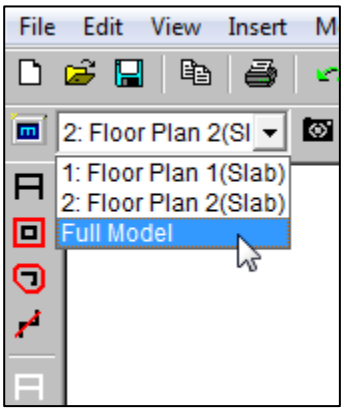
- Click **Ok**.

This will open a second model view window that shows the new second floor. Return to the default view.

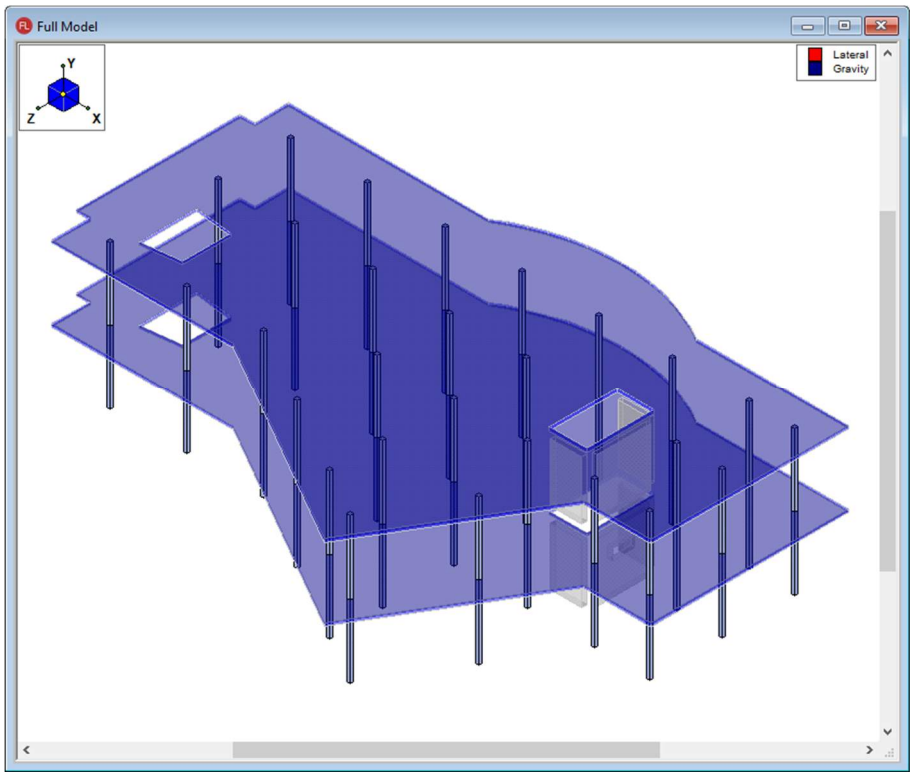
- From the **Window** menu, select **Single View**.

Next, view the full isometric model view.

- Click on the **Floors** menu on the Window toolbar and select **Full Model**.



This will show the full model (both floors) in a rendered isometric view.



Note: Notice that the opening in the elevator shaft wall created in the first floor was not copied into the second floor. Wall openings must be created individually.

This is the end of Tutorial B3.

If you are using the Demonstration version of the program, you will not be able to save this model, as it exceeds the Demonstration version limits. Please proceed to the next tutorial for instructions on how to open the Tutorial B4 Starter File provided by RISA.

If you are using the full version of the program, you may save by doing the following: On the **File** menu, click **Save As** and enter a file name.

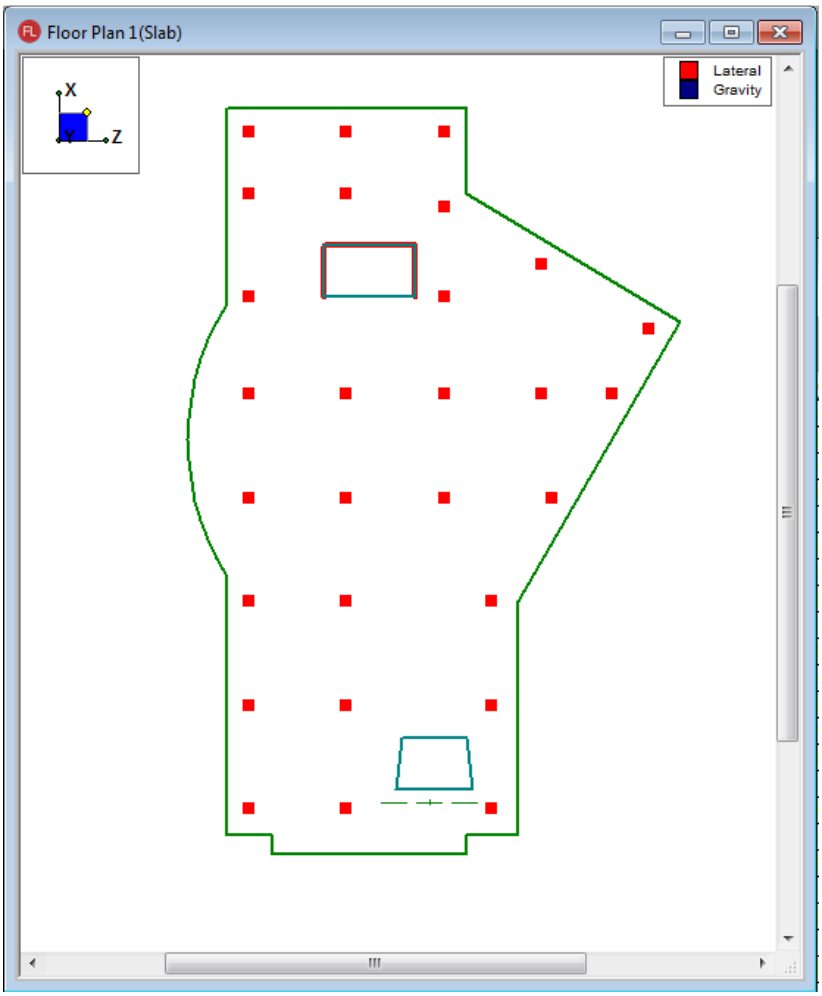
Part B: Tutorial 4 – Solution & Results

This tutorial will guide you through generating load combinations, running a solution, design optimization, and reviewing the results.

This tutorial continues from where the previous tutorial ended, so follow these steps to open the Tutorial B4 Starter File provided by RISA:

- Double-click on the **RISAFloor** icon to start the program.
- Click **Open Model**  . Double-click the **Tutorials** folder, select **Tutorial B4 Starter.rfl** and click **Open**. Click **Close**  (or **Cancel**) to exit the **Model Settings** dialog box.
- On the Window toolbar, click the **Graphic Editing** button  to activate the Drawing toolbar.
- Turn off the view of the **Design Strips** by clicking on the **Design Strip** button  on the Drawing toolbar. Click on the black menu arrow and select **None**.

Your screen should now look like this:

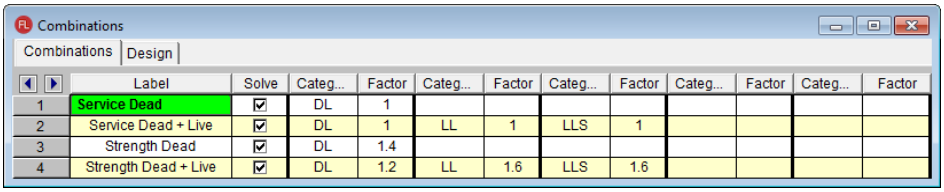


Load Combinations

RISAFloor ES solves load combinations that are defined in the **Load Combinations** spreadsheet. As RISAFloor ES designs members, it chooses members that satisfy the codes specified in the **Model Settings** and the rules that are defined in the **Design Rules** spreadsheets.

To view the load combinations:

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



	Label	Solve	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor
1	Service Dead	☑	DL	1								
2	Service Dead + Live	☑	DL	1	LL	1	LLS	1				
3	Strength Dead	☑	DL	1.4								
4	Strength Dead + Live	☑	DL	1.2	LL	1.6	LLS	1.6				

On the **Combinations** tab, you will combine and factor the various load categories that you have already applied to your model.

- Press the **F4** key four times to delete the default load combinations.

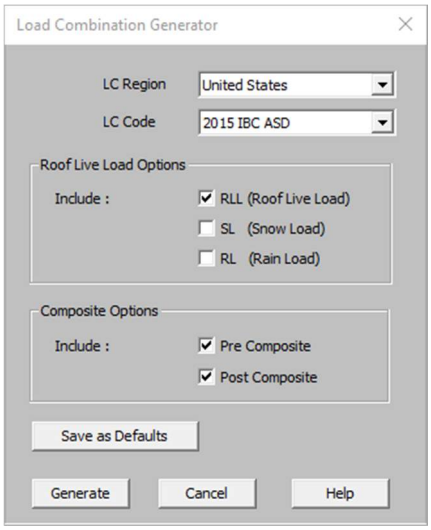
Load Combination Generator

Now that you have a spreadsheet open, notice the Window toolbar looks different.

- On the Window toolbar, click the **LC Generator** button. The **Load Combination Generator** dialog box will be used to generate the load combinations needed to design the model.

For this tutorial, assume that in the regions where this project will be built, the governing roof loads are usually roof live load, rather than snow or rain.

- In the **LC Region** list, select **United States**.
- In the **LC Code** list, select **2015 IBC ASD**.
- Click to clear the **SL** and **RL** check boxes so that you will not be generating more load combinations than are needed.



- Click **Generate**.

The following load combinations will be generated by RISAFloor:

Combinations

Combinations | Design

	Label	Solve	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor
1	IBC 16-8 Pre	☑	DLPre	1										
2	IBC 16-9 Pre	☑	DLPre	1	LLConst	1								
3	IBC 16-8 Post	☑	DL	1										
4	IBC 16-9 Post	☑	DL	1	LL	1	LLS	1						
5	IBC 16-10 (a) Pos	☑	DL	1	RLL	1								
6	IBC 16-11 (a) Pos	☑	DL	1	LL	.75	LLS	.75	RLL	.75				
7	IBC 16-11 (b) Pos	☑	DL	1	LL	.75	LLS	.75						

- Click the **Design** tab to review the remaining information associated with these pre-generated load combinations.

Combination Design

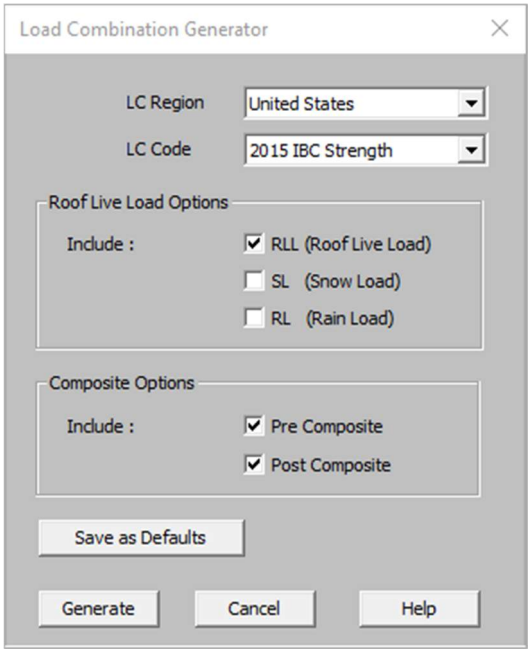
Combinations | Design

	Label	ASIF	CD	Service	Hot Rolled...	Cold Form...	Wood	Wood Pro...	Concrete	Masonry	Connection
1	IBC 16-8 Pre		.9	☑	☑	☑	☑	☑	☑	☑	☑
2	IBC 16-9 Pre			☑	☑	☑	☑	☑	☑	☑	☑
3	IBC 16-8 Post		.9	☑	☑	☑	☑	☑	☑	☑	☑
4	IBC 16-9 Post			☑	☑	☑	☑	☑	☑	☑	☑
5	IBC 16-10 (a) Pos		1.25	☑	☑	☑	☑	☑	☑	☑	☑
6	IBC 16-11 (a) Pos		1.25	☑	☑	☑	☑	☑	☑	☑	☑
7	IBC 16-11 (b) Pos		1.15	☑	☑	☑	☑	☑	☑	☑	☑

On this tab you can specify which load combination can be applied to a specified material. The Service checkbox can also be used to designate which combinations will be used for Slab Point Deflection results (on concrete slab floors only).

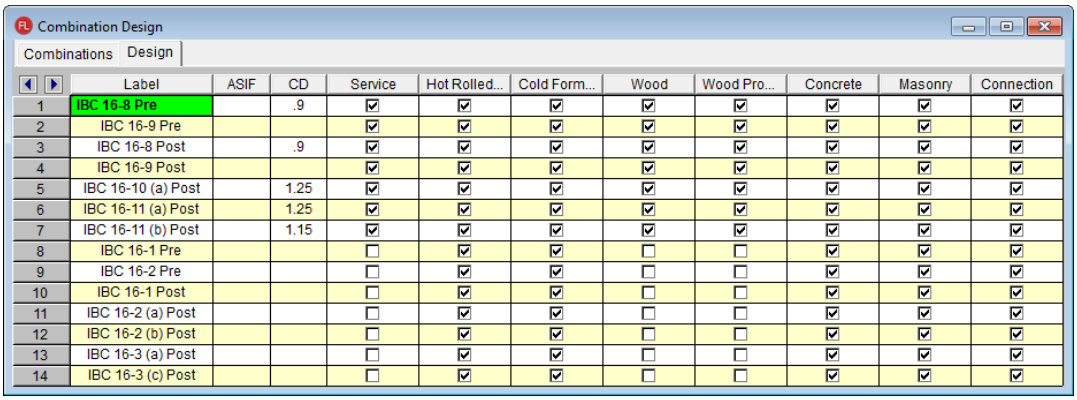
Finish by generating strength level combinations.

- Click the **LC Generator** button LC Generator.
- In the **LC Code** list, select **2015 IBC Strength**.



- Click Generate.

This finalizes your load combinations as shown below:



	Label	ASIF	CD	Service	Hot Rolled...	Cold Form...	Wood	Wood Pro...	Concrete	Masonry	Connection
1	IBC 16-8 Pre		.9	☑	☑	☑	☑	☑	☑	☑	☑
2	IBC 16-9 Pre			☑	☑	☑	☑	☑	☑	☑	☑
3	IBC 16-8 Post		.9	☑	☑	☑	☑	☑	☑	☑	☑
4	IBC 16-9 Post			☑	☑	☑	☑	☑	☑	☑	☑
5	IBC 16-10 (a) Post		1.25	☑	☑	☑	☑	☑	☑	☑	☑
6	IBC 16-11 (a) Post		1.25	☑	☑	☑	☑	☑	☑	☑	☑
7	IBC 16-11 (b) Post		1.15	☑	☑	☑	☑	☑	☑	☑	☑
8	IBC 16-1 Pre			☐	☑	☑	☐	☐	☑	☑	☑
9	IBC 16-2 Pre			☐	☑	☑	☐	☐	☑	☑	☑
10	IBC 16-1 Post			☐	☑	☑	☐	☐	☑	☑	☑
11	IBC 16-2 (a) Post			☐	☑	☑	☐	☐	☑	☑	☑
12	IBC 16-2 (b) Post			☐	☑	☑	☐	☐	☑	☑	☑
13	IBC 16-3 (a) Post			☐	☑	☑	☐	☐	☑	☑	☑
14	IBC 16-3 (c) Post			☐	☑	☑	☐	☐	☑	☑	☑

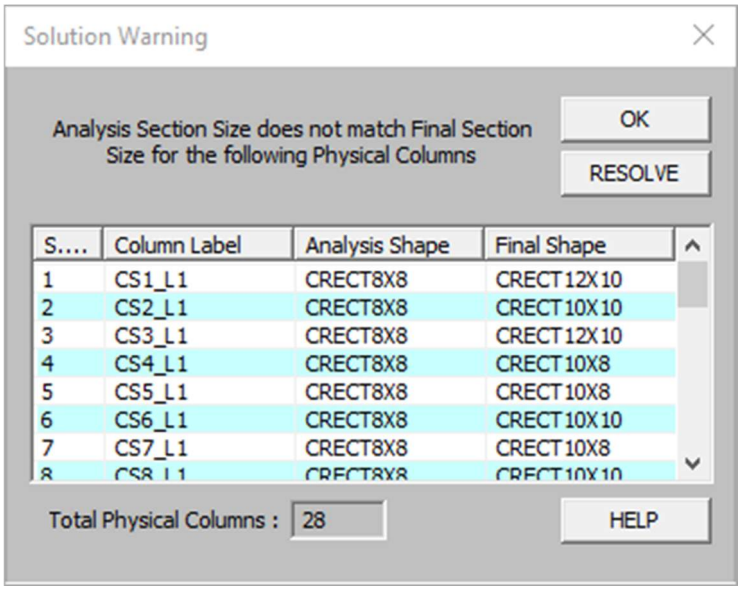
Notice you have total control of the design of all member types using the check boxes. If you had any wood members in your model, they would only be designed for the ASD load combinations.

Solving the Model

To solve the model:

- On the Main menu, click **Solve – Design Members/Walls/Slabs**.

The first solution results in a **Solution Warning**.



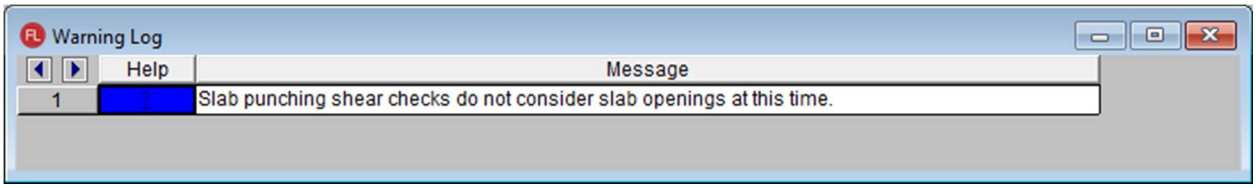
This Warning comes up because the program performs a solution for the upper floor first. When the model is solved "top-down" it is common for the original "required" size of a column at the upper level to be insufficient at the lower level. This dialog allows you to update the upper level column sizes and solve again.

- Click **RESOLVE** to replace the upper columns and resolve with the larger sizes.

The Solution Warning will come up a second time. This time just accept the current column sizes so you can move on and review your results.

- Click **OK** when the next **Solution Warning** dialog comes up.

When the solution is finished, the chosen member sizes are displayed on the plot (or, if you are in rendered mode, the model is rendered with the sizes) and the solution **Warning Log** is displayed.



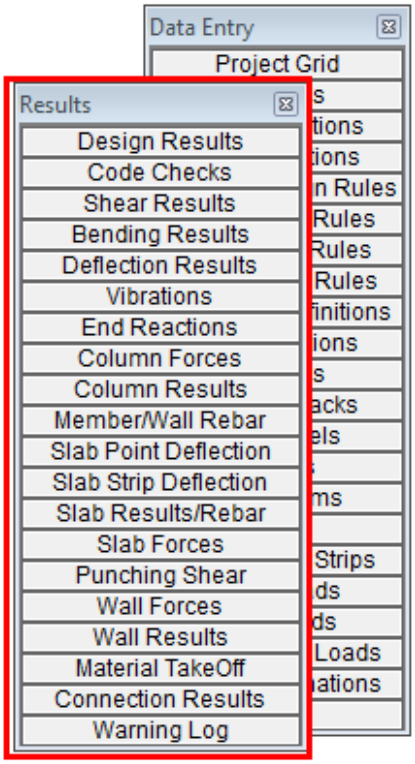
The purpose of the **Warning Log** is to inform you whenever a modeling issue or error arises which affects the design of the floor. For example, in the image above, you are being warned that the punching shear checks do not consider slab openings. This is a program limitation, not an input error.

Note: Results produced by your model may not exactly match the results shown in this tutorial. This may be caused by a variety of factors, including how the model was created and which version of the program is being used.

Reviewing the Results

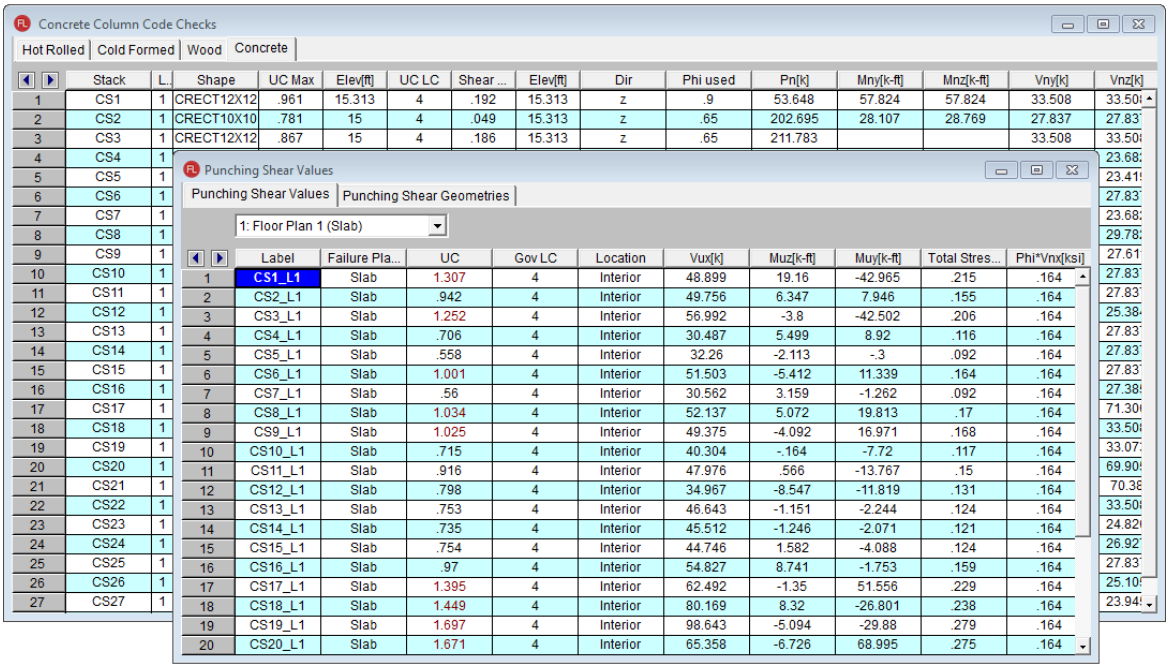
After the solution, the Results toolbar automatically opens on the right side of the screen.

The **Results** toolbar should now open on the upper right of your model space on top of the **Data Entry** toolbar.



Take a look at some of the results.

- From the **Results** toolbar, click on **Column Results**.
- Select the **Concrete** tab.
- From the **Results** toolbar, click on **Punching Shear**.



	Stack	L..	Shape	UC Max	Elev[ft]	UC LC	Shear ...	Elev[ft]	Dir	Phi used	Pn[k]	Mny[k-ft]	Mnz[k-ft]	Vny[k]	Vnz[k]
1	CS1	1	CRECT12X12	.961	15.313	4	.192	15.313	z	.9	53.648	57.824	57.824	33.508	33.50
2	CS2	1	CRECT10X10	.781	15	4	.049	15.313	z	.65	202.695	28.107	28.769	27.837	27.83
3	CS3	1	CRECT12X12	.867	15	4	.186	15.313	z	.65	211.783			33.508	33.50
4	CS4	1													23.68
5	CS5	1													23.41
6	CS6	1													27.83
7	CS7	1													23.68
8	CS8	1													29.78
9	CS9	1													27.61
10	CS10	1													27.83
11	CS11	1													27.83
12	CS12	1													25.38
13	CS13	1													27.83
14	CS14	1													27.83
15	CS15	1													27.83
16	CS16	1													27.38
17	CS17	1													71.30
18	CS18	1													33.50
19	CS19	1													33.07
20	CS20	1													69.90
21	CS21	1													70.38
22	CS22	1													33.50
23	CS23	1													24.82
24	CS24	1													26.92
25	CS25	1													27.83
26	CS26	1													25.10
27	CS27	1													23.94

	Label	Failure Pla...	UC	Gov LC	Location	Vux[k]	Muz[k-ft]	Muy[k-ft]	Total Stres...	Phi*Vnx[ksi]
1	CS1_L1	Slab	1.307	4	Interior	48.899	19.16	-42.965	.215	.164
2	CS2_L1	Slab	.942	4	Interior	49.756	6.347	7.946	.155	.164
3	CS3_L1	Slab	1.252	4	Interior	56.992	-3.8	-42.502	.206	.164
4	CS4_L1	Slab	.706	4	Interior	30.487	5.499	8.92	.116	.164
5	CS5_L1	Slab	.558	4	Interior	32.26	-2.113	-.3	.092	.164
6	CS6_L1	Slab	1.001	4	Interior	51.503	-5.412	11.339	.164	.164
7	CS7_L1	Slab	.56	4	Interior	30.562	3.159	-1.262	.092	.164
8	CS8_L1	Slab	1.034	4	Interior	52.137	5.072	19.813	.17	.164
9	CS9_L1	Slab	1.025	4	Interior	49.375	-4.092	16.971	.168	.164
10	CS10_L1	Slab	.715	4	Interior	40.304	-.164	-7.72	.117	.164
11	CS11_L1	Slab	.916	4	Interior	47.976	.566	-13.767	.15	.164
12	CS12_L1	Slab	.798	4	Interior	34.967	-8.547	-11.819	.131	.164
13	CS13_L1	Slab	.753	4	Interior	46.643	-1.151	-2.244	.124	.164
14	CS14_L1	Slab	.735	4	Interior	45.512	-1.246	-2.071	.121	.164
15	CS15_L1	Slab	.754	4	Interior	44.746	1.582	-4.088	.124	.164
16	CS16_L1	Slab	.97	4	Interior	54.827	8.741	-1.753	.159	.164
17	CS17_L1	Slab	1.395	4	Interior	62.492	-1.35	51.556	.229	.164
18	CS18_L1	Slab	1.449	4	Interior	80.169	8.32	-26.801	.238	.164
19	CS19_L1	Slab	1.697	4	Interior	98.643	-5.094	-29.88	.279	.164
20	CS20_L1	Slab	1.671	4	Interior	65.358	-6.726	68.995	.275	.164

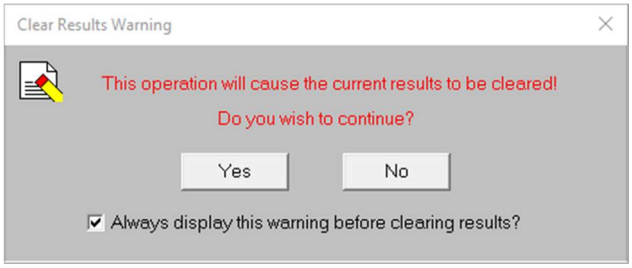
Notice that there are several failures where the punching shear UC exceeds 1.0. These are displayed in red text. The **Punching Shear** spreadsheet reports the governing load combination (LC) as well as the values that came up with the failure. For more information you can also see the Detail Report.

These failures are most likely due to the slab only being eight inches thick. Therefore, you can thicken up the section of slab around the columns with shear caps.

Editing the Model

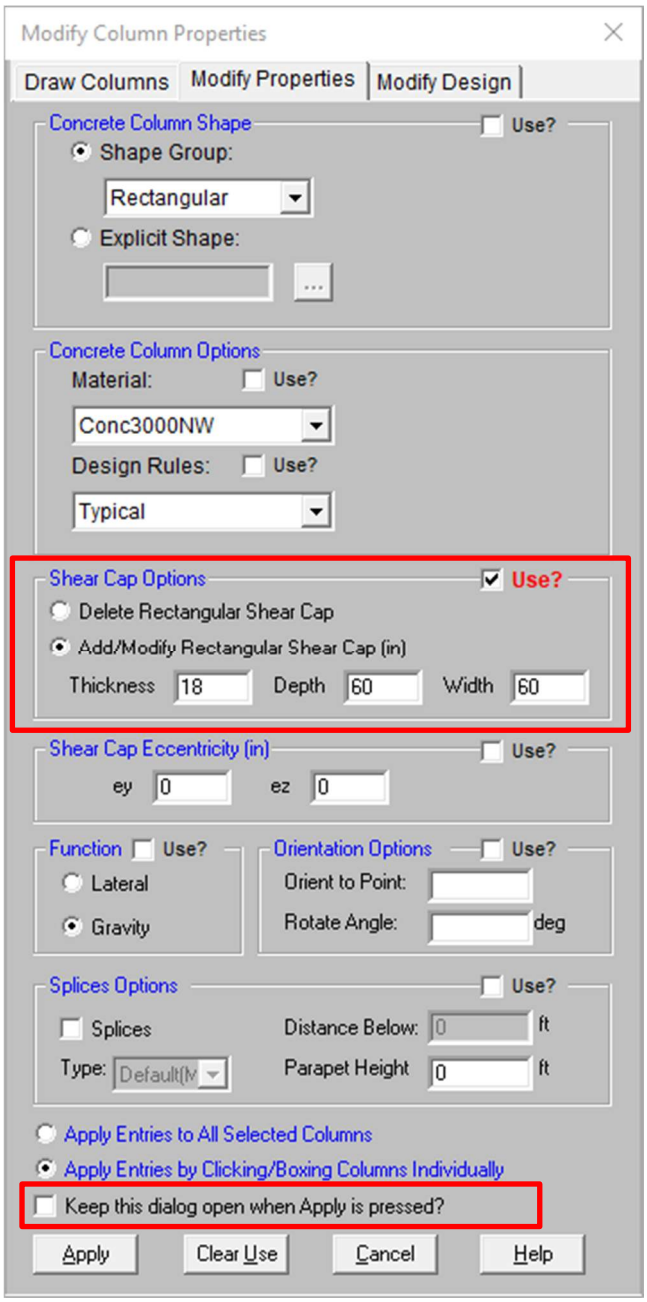
Add on a couple shear caps to investigate how to correct a failing model. From the Punching Shear spreadsheet above, you know columns CS1 and CS3 are failing on the first floor level. You will add shear caps to these columns.

- Close all open spreadsheets.
- Click on the **Floors** menu on the Window toolbar and select **1: Floor Plan 1(Slab)**
- From the **Modify** menu, select **Columns.**
- Click Yes to close the Clear Results Warning.



Use the **Modify Column Properties** dialog to add in a shear cap.

- Under **Shear Cap Options**, select the **Use?** checkbox.
- Select **Add/Modify Rectangular Shear Cap (in)**.
- Enter a **Thickness** of **18”**, a **Depth** of **60”** and a **Width** of **60”**.
- Select **Apply Entries by Clicking/Boxing Columns Individually**

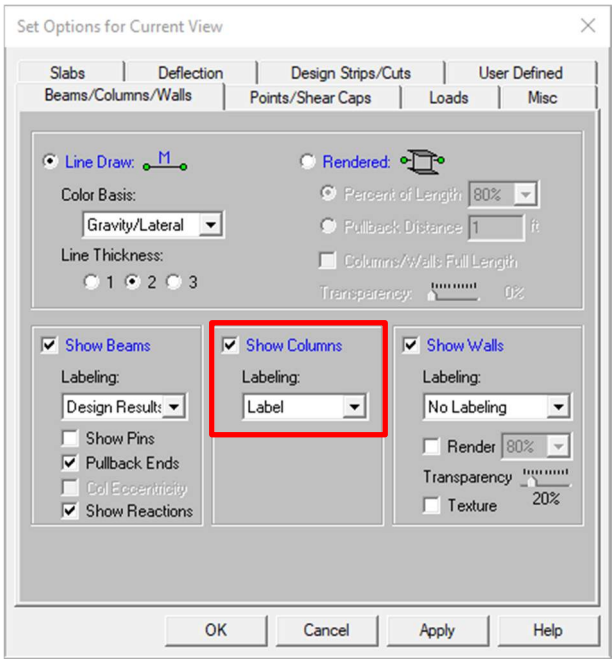


- Click **Apply**.

Note: Your cursor will change to  to designate that you are in modification mode.

To find the location of the columns, turn on the column labels.

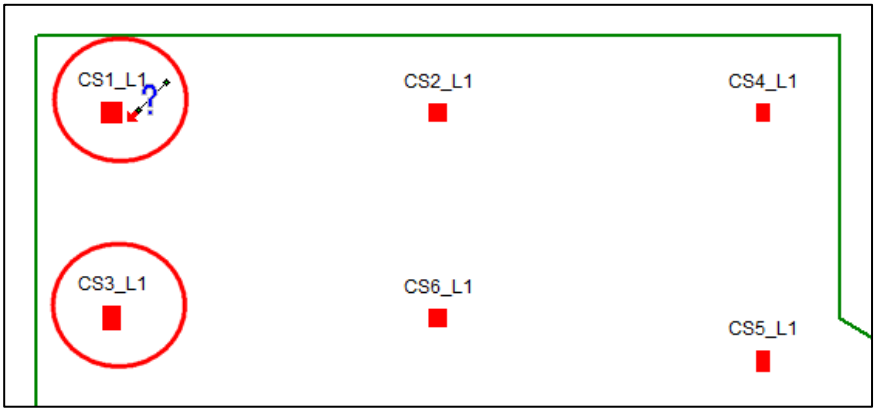
- From the Window toolbar, click on the **Model Display Options** button  to open the **Set Options for Current View** dialog.
- Click on the **Beams/Columns/Walls** tab.
- Under **Show Columns**, set the **Labeling** option to **Label**.



- Click **OK** and you can see that columns **CS1** and **CS3** are in the upper left corner of the first floor.

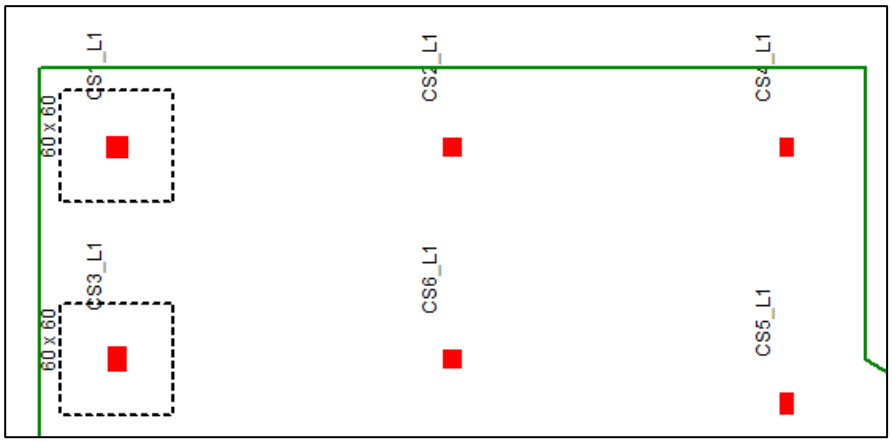
Now add a **Shear Cap** to these columns.

- Click on the two columns with your drawing tool cursor.



- Right-click or click ESC to escape the modification tool.

When complete, the model should show the shear cap perimeters.



Now that you have made the modification to the model, solve again.

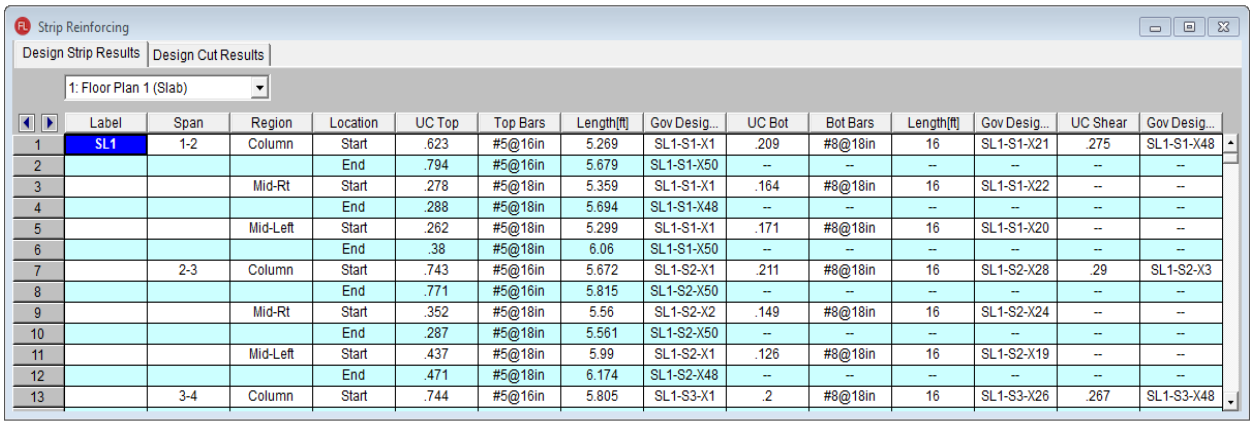
- Solve the model by clicking on the **Solve** button from the RISA toolbar.
- Click OK from the Solution Warning.
- Close all open spreadsheets by clicking on the **Close** button.

Now you can take a look at the rest of the results.

Spreadsheet Results

Open the **Strip Reinforcing** spreadsheet:

- On the **Results** toolbar, select **Slab Results/Rebar**.

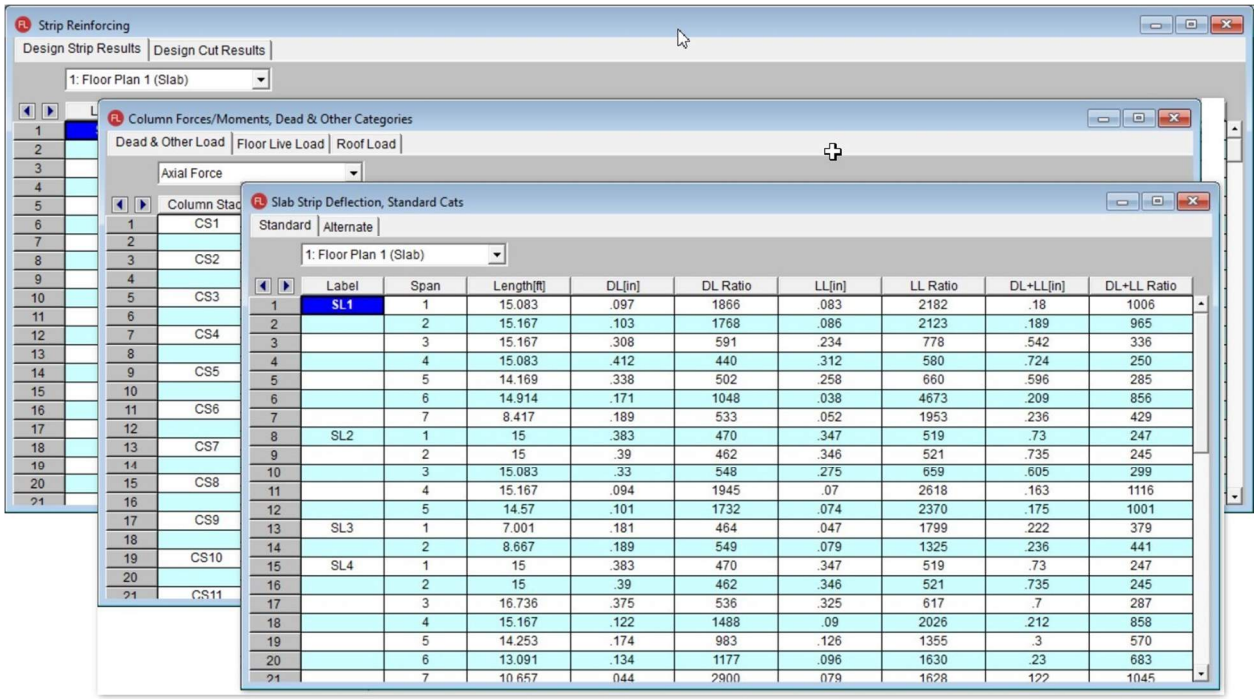


	Label	Span	Region	Location	UC Top	Top Bars	Length[ft]	Gov Desig...	UC Bot	Bot Bars	Length[ft]	Gov Desig...	UC Shear	Gov Desig...
1	SL1	1-2	Column	Start	.623	#5@16in	5.269	SL1-S1-X1	.209	#8@18in	16	SL1-S1-X21	.275	SL1-S1-X48
2				End	.794	#5@16in	5.679	SL1-S1-X50	--	--	--	--	--	--
3			Mid-Rt	Start	.278	#5@18in	5.359	SL1-S1-X1	.164	#8@18in	16	SL1-S1-X22	--	--
4				End	.288	#5@18in	5.694	SL1-S1-X48	--	--	--	--	--	--
5			Mid-Left	Start	.262	#5@18in	5.299	SL1-S1-X1	.171	#8@18in	16	SL1-S1-X20	--	--
6				End	.38	#5@18in	6.06	SL1-S1-X50	--	--	--	--	--	--
7		2-3	Column	Start	.743	#5@16in	5.672	SL1-S2-X1	.211	#8@18in	16	SL1-S2-X28	.29	SL1-S2-X3
8				End	.771	#5@16in	5.815	SL1-S2-X50	--	--	--	--	--	--
9			Mid-Rt	Start	.352	#5@18in	5.56	SL1-S2-X2	.149	#8@18in	16	SL1-S2-X24	--	--
10				End	.287	#5@18in	5.561	SL1-S2-X50	--	--	--	--	--	--
11			Mid-Left	Start	.437	#5@18in	5.99	SL1-S2-X1	.126	#8@18in	16	SL1-S2-X19	--	--
12				End	.471	#5@18in	6.174	SL1-S2-X48	--	--	--	--	--	--
13		3-4	Column	Start	.744	#5@16in	5.805	SL1-S3-X1	.2	#8@18in	16	SL1-S3-X26	.267	SL1-S3-X48

The **Strip Reinforcing** spreadsheet summarizes all design strip and cut results. The **Design Strip Results** tab lists a summary of the reinforcement design, including the **UC Top** (code check value at top), **UC Bot** (code check at the bottom), and the **UC Shear** (shear code check). Similar information can be found under the **Design Cut Results** tab.

You may open the other spreadsheets to review the other types of results.

- From the Results toolbar select **Column Forces**, **Slab Strip Deflection**, etc.



	Label	Span	Length[ft]	DL[in]	DL Ratio	LL[in]	LL Ratio	DL+LL[in]	DL+LL Ratio
1	SL1	1	15.083	.097	1866	.083	2182	.18	1006
2		2	15.167	.103	1768	.086	2123	.189	965
3		3	15.167	.308	591	.234	778	.542	336
4		4	15.083	.412	440	.312	580	.724	250
5		5	14.169	.338	502	.258	660	.596	285
6		6	14.914	.171	1048	.038	4673	.209	856
7		7	8.417	.189	533	.052	1953	.236	429
8	SL2	1	15	.383	470	.347	519	.73	247
9		2	15	.39	462	.346	521	.735	245
10		3	15.083	.33	548	.275	659	.605	299
11		4	15.167	.094	1945	.07	2618	.163	1116
12		5	14.57	.101	1732	.074	2370	.175	1001
13	SL3	1	7.001	.181	464	.047	1799	.222	379
14		2	8.667	.189	549	.079	1325	.236	441
15	SL4	1	15	.383	470	.347	519	.73	247
16		2	15	.39	462	.346	521	.735	245
17		3	16.736	.375	536	.325	617	.7	287
18		4	15.167	.122	1488	.09	2026	.212	858
19		5	14.253	.174	983	.126	1355	.3	570
20		6	13.091	.134	1177	.096	1630	.23	683
21		7	10.657	.044	2900	.079	1628	.122	1045

Return to your model view:

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

Viewing Graphical Results

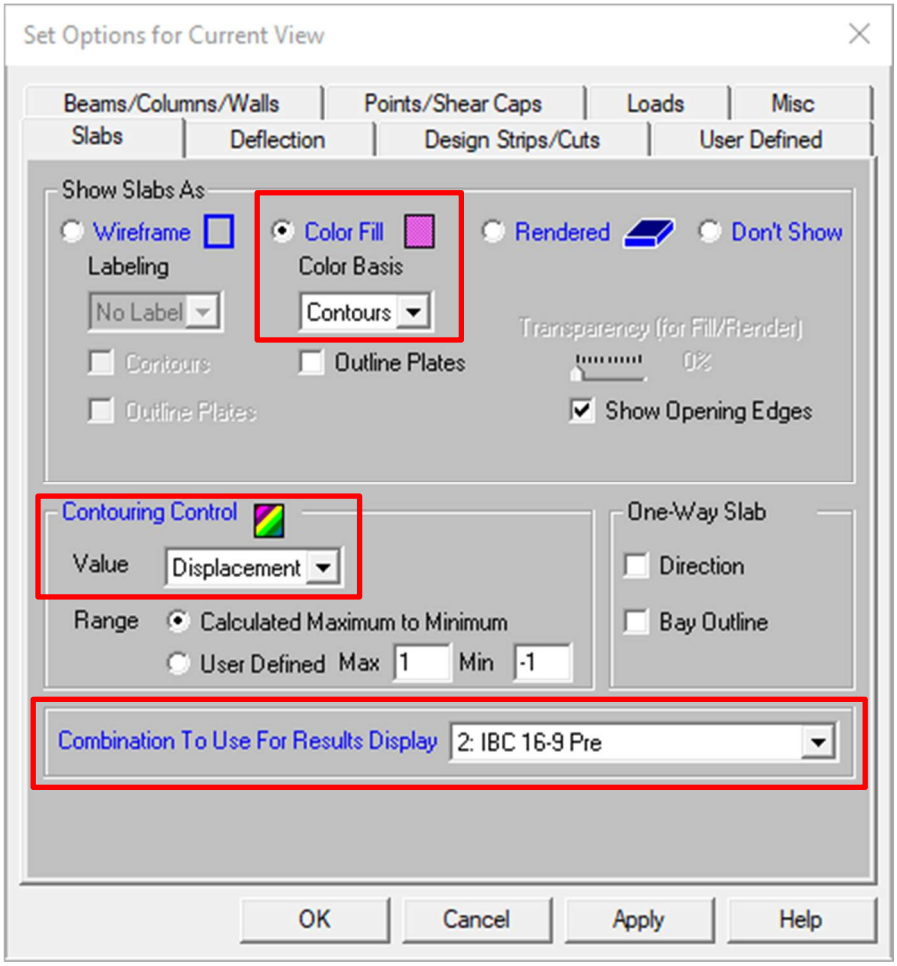
Explore the graphic result viewing options:

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

The **Set Options for Current View** (i.e. Model Display Options) dialog contains all the options for graphic display in your model. This includes input and results information. If you have not solved your model, the results options will be unavailable.

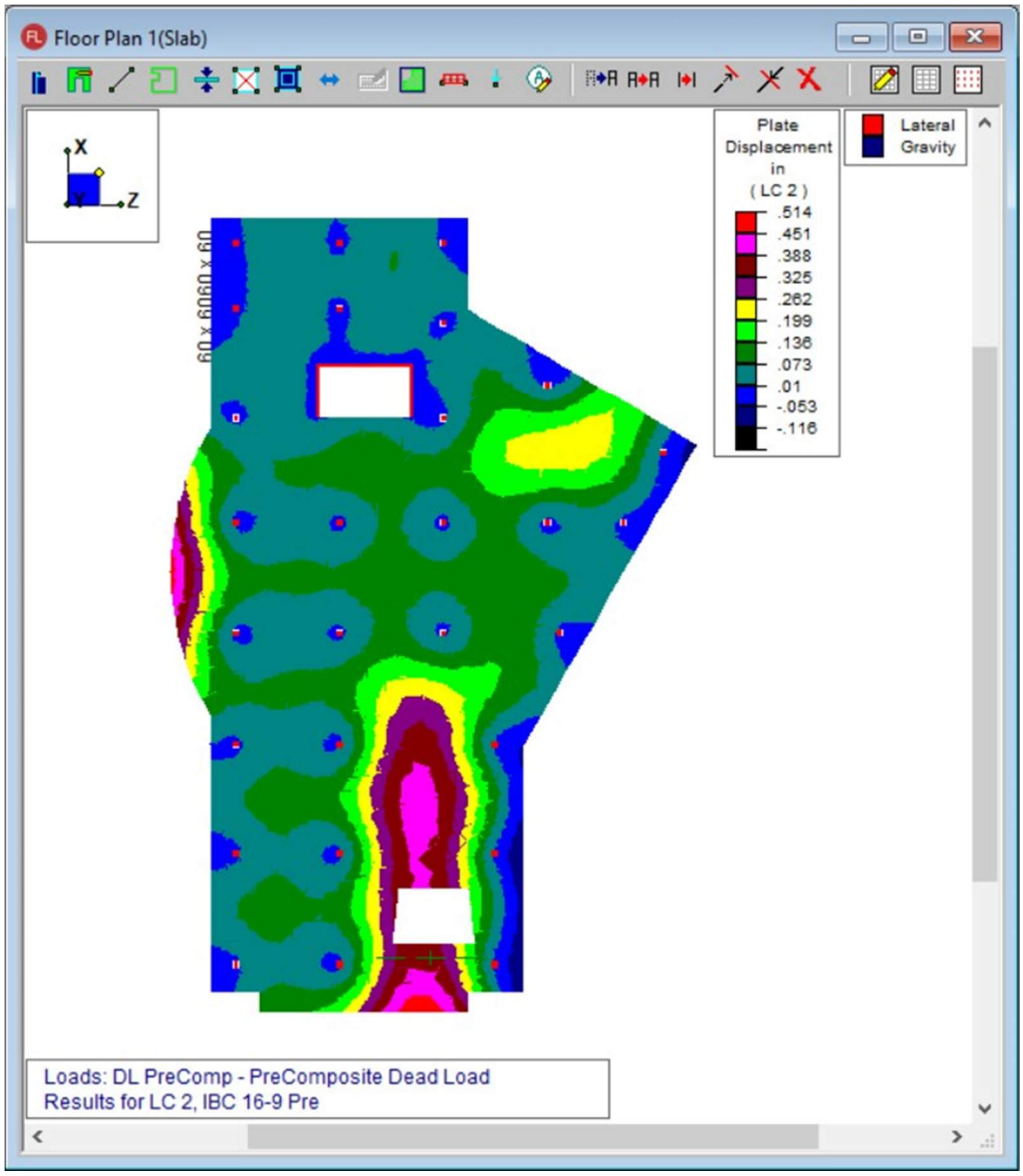
Turn on color coding contours of the displacements from Load Combination 2.

- Click the **Slabs** tab. Under **Show Slabs As**, select **Color Fill**.
- Under **Color Basis**, select **Contours**. Then, under the **Contouring Control** section, set the **Value** to **Displacement**.
- Set the Combination To Use for Results Display to LC2: IBC 16-9 Pre.



- Click **Apply**.

Your plot should now look like this:



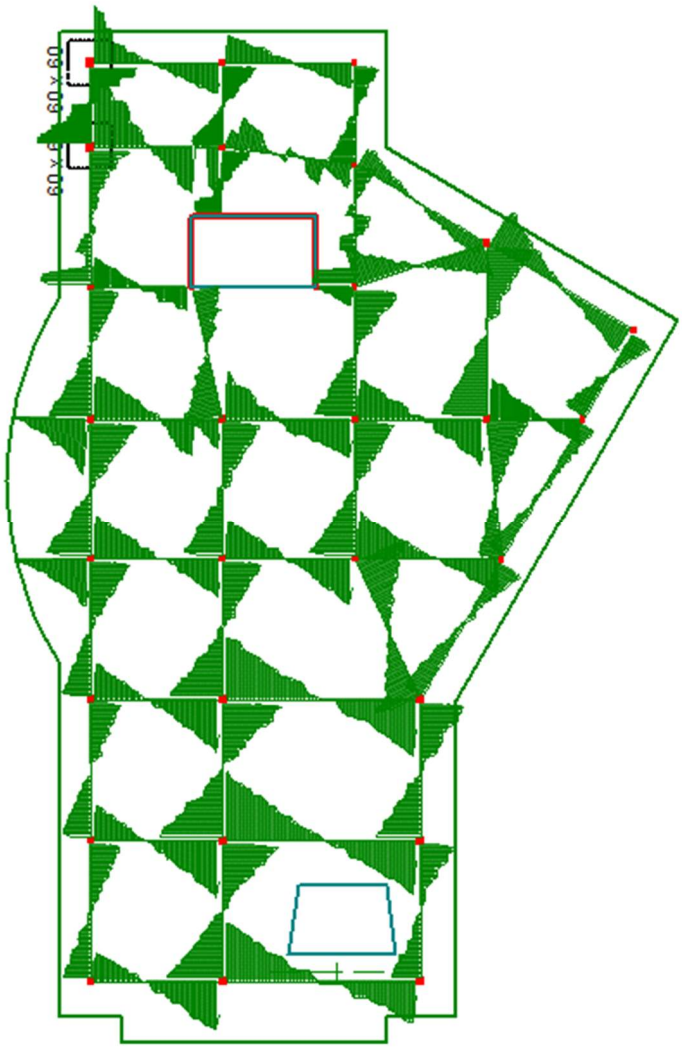
This image gives a quick graphical representation of the displacements per Load Combination 2. The color-coding legend is displayed in the upper right corner (you may need to move your **Results** or **Data Entry** toolbars to see this). You can create a similar plot to display the Bending or Shear forces.

- Click **Cancel** to exit the dialog box and return to your model view.

Next, take a look at the design strip graphical results but this time you will use the Window toolbar buttons. These give you quicker access to the graphical plot options.

- Click on the black arrow next to the **Design Strip Diagram** button  from the Window toolbar.
- Select **Out of Plane Shear**.

Your model view should now look like this:



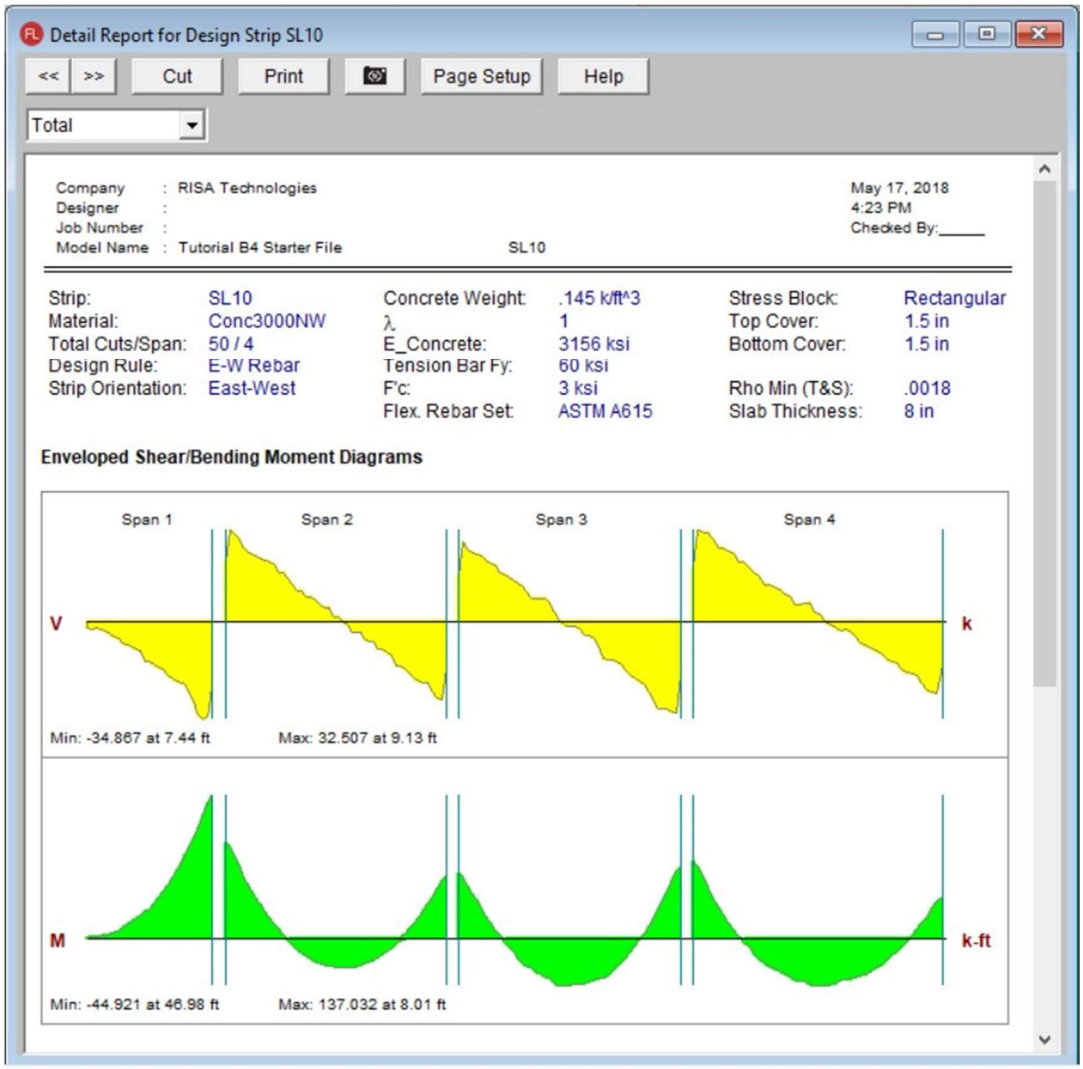
Detail Reports

Take a look at the details of the design for the Support Lines and Design Strips.

- On the Selection toolbar (left side), click the **Detail** button  .

Note: Your cursor will change to  .

- Click on any of the **Support Lines** to review the additional information in the Strip **Detail Report.**

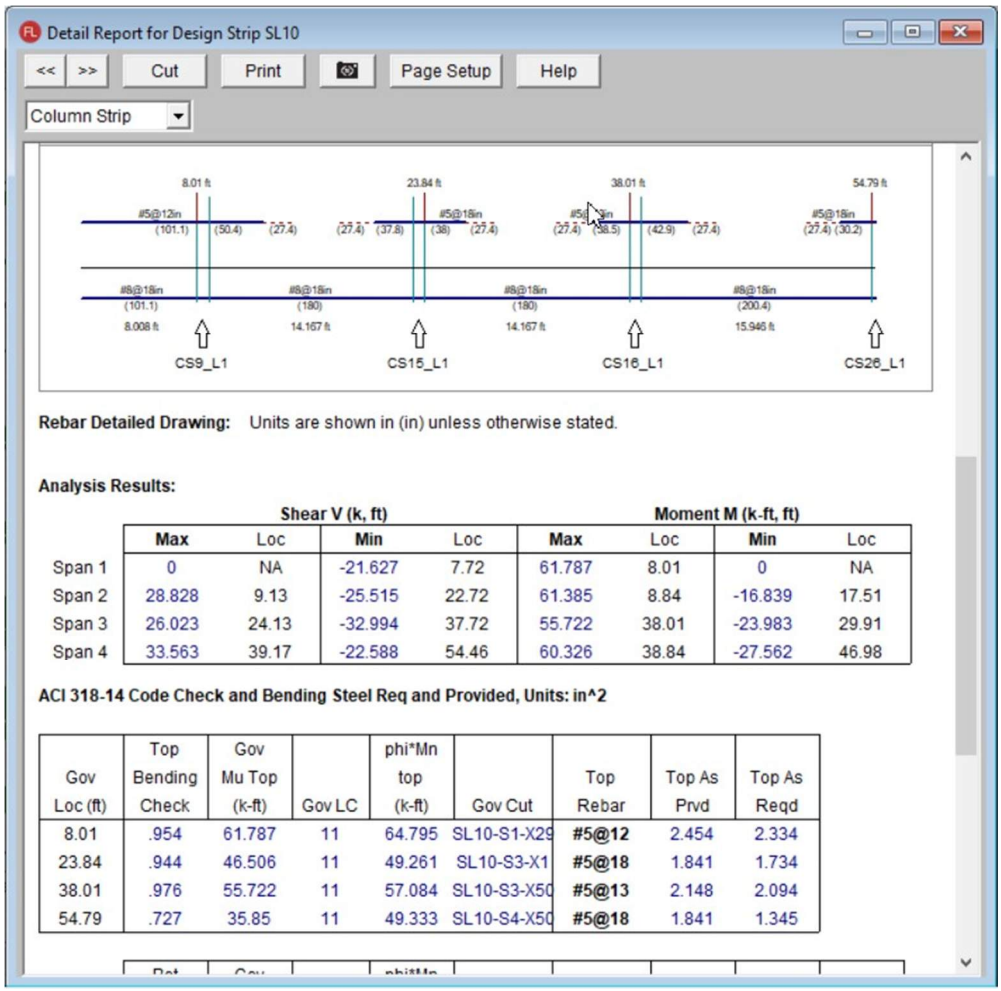


The **Detail Report** displays enveloped shear and moment diagrams for each span of the **Support Line**. It also gives **Analysis Results** (**Max** and **Min Shear** and **Moment** values used for reinforcement design) and **Deflection Results** for each span.

You can use the drop-down menu in the upper left to see the **Total** report or the more detailed reports for the **Column Strip**, **Mid-Strip Right**, or **Mid-Strip Left**.

- Select **Column Strip** from the drop-down menu on the upper left.

- Scroll down to the bottom of the detail report and you can review the **Rebar Details**, **Analysis Results**, **Code Checks**, and **Deflection Results**.

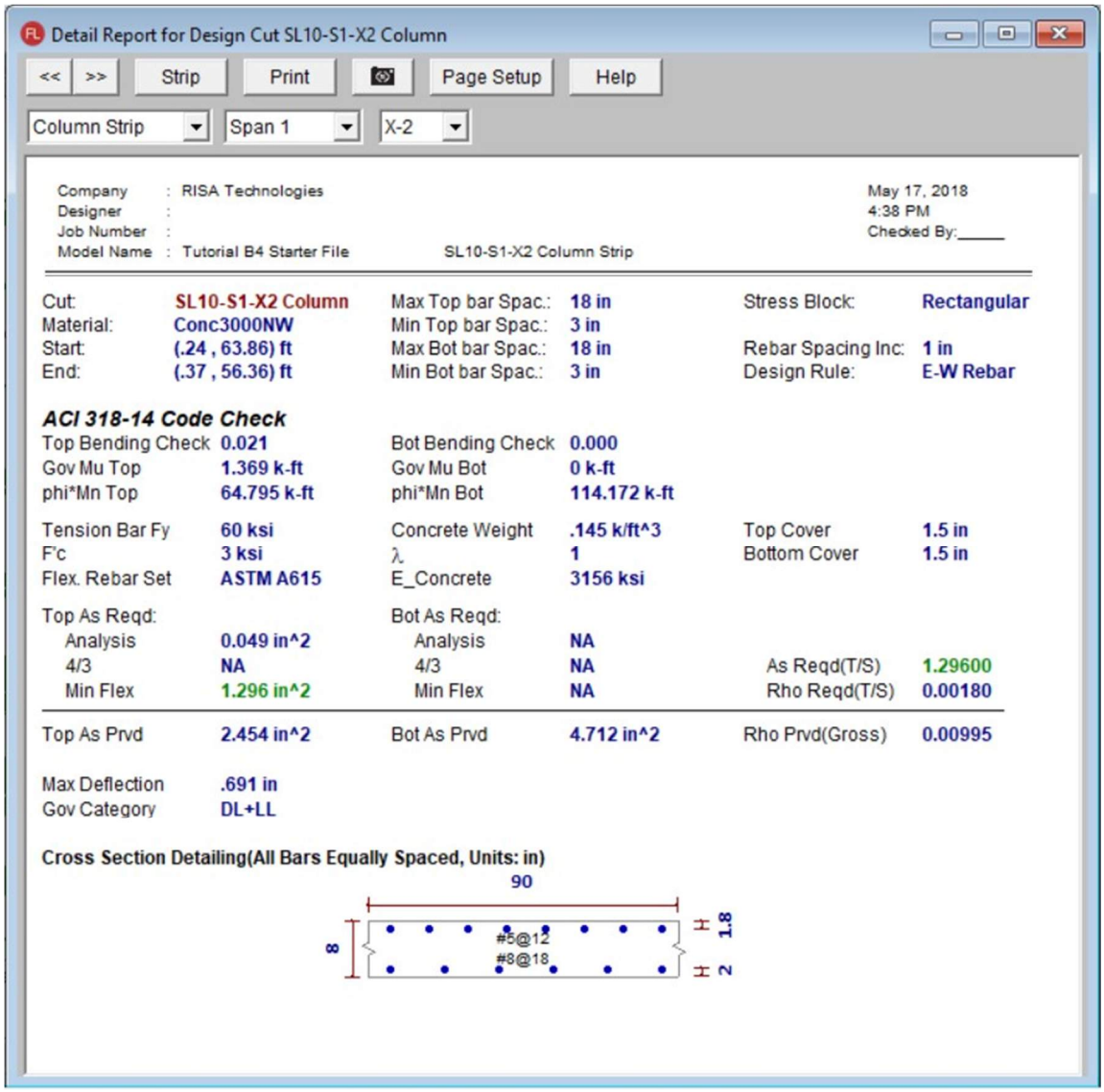


Next, you will take a look at the governing **Cut** results.

The reinforcement for each **Design Strip** is governed by an internal **Design Cut**.

- Click on the **Cut** button  from the **Design Strip Detail Report**.

This will open the **Design Cut Detail Report** for the governing cut for **Column Strip Span 1**.



The **Design Cut Detail Report** shows the detailed information and values that governed the design of each **Cut** within the **Design Strip**.

Note: You can click on the forward and backward arrows  at the top left to scroll quickly through the **Design Cut Detail Reports**.

You can also review the design results for the single **Design Cut** drawn in by the opening.

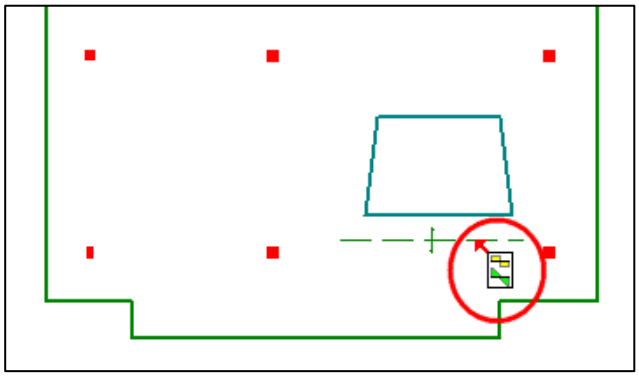
- First, click **Close** to close the **Detail Report** and return to the model view.

Turn off the display of the support lines.

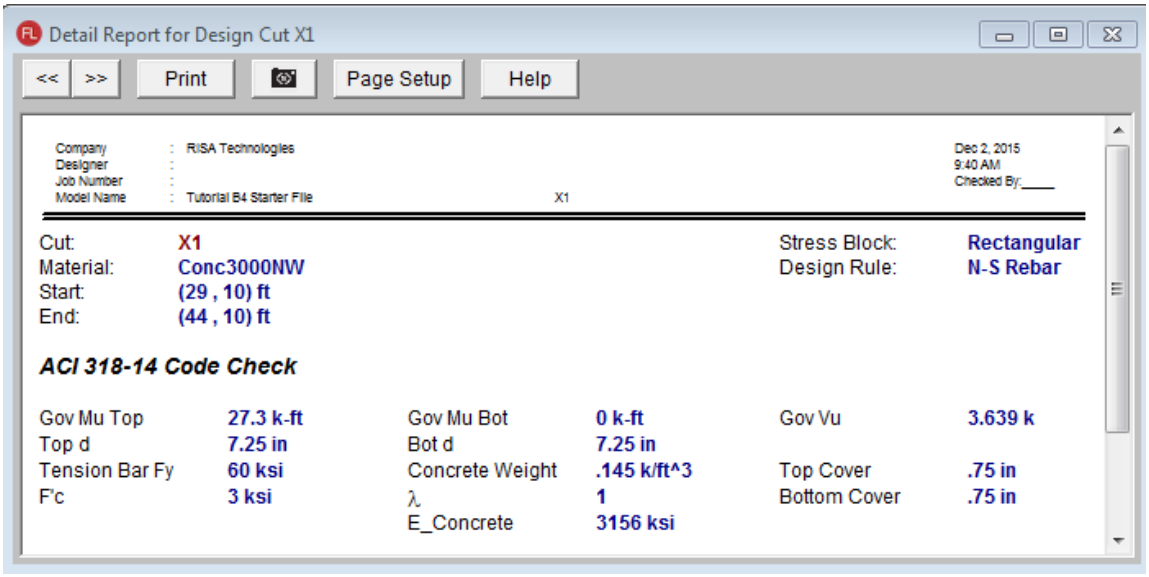
- Click on the black arrow next to the **Design Strip Diagram** button from the Window toolbar.
- Select **None**.

Now, take a look at the **Design Cut Results**:

- With your Detail tool cursor still active, zoom in next to the opening and click on the green **Design Cut** that you drew on in Tutorial B2.



This will open the **Detail Report** for the single **Design Cut**.

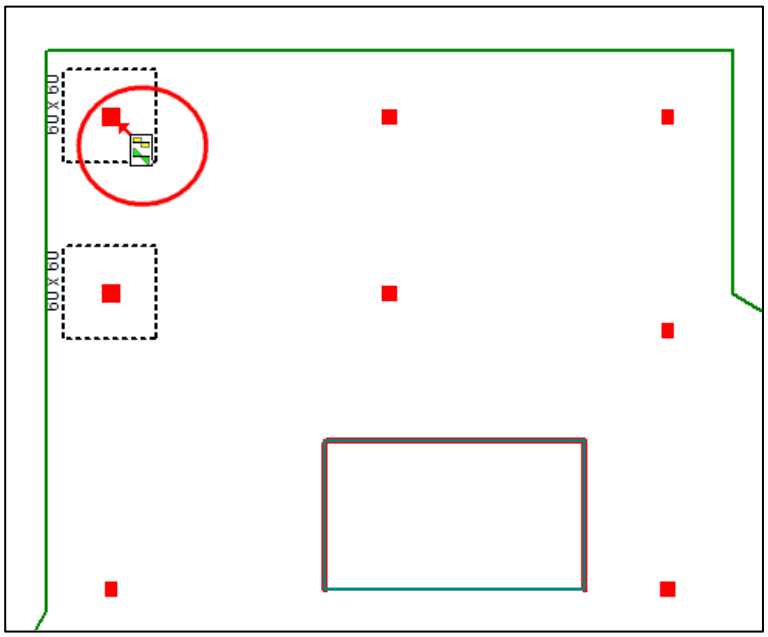


When finished reviewing these results, close the **Detail Report**.

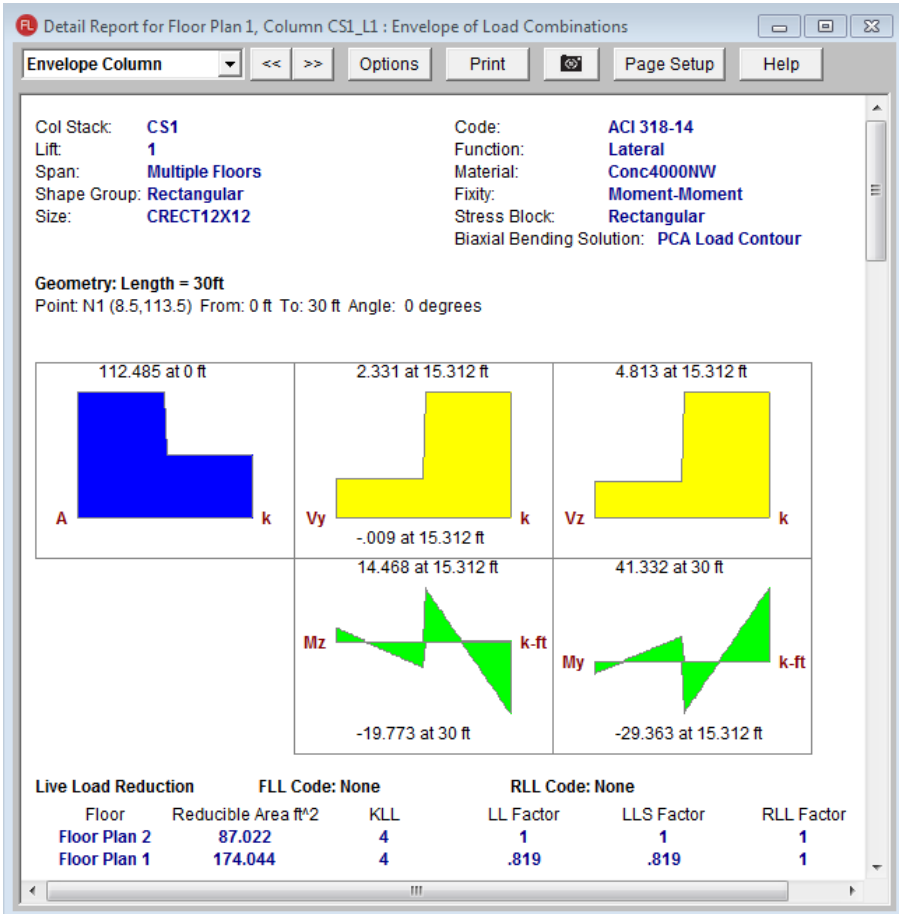
- Click **Close** to close the **Detail Report** and return to the model view.

Now you can review the column design results in the **Detail Report**.

- With your **Detail** cursor  still active, click on the top left column (with the shear cap).



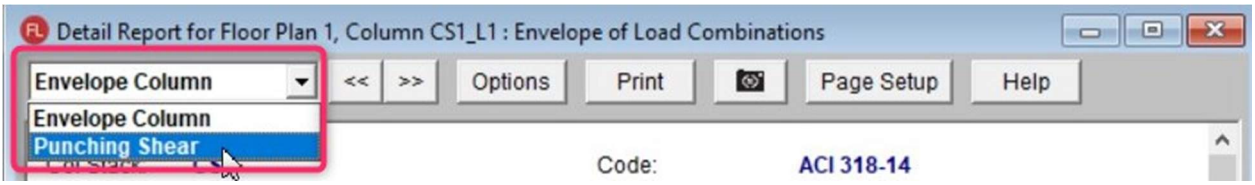
This will open the **Envelope Column Detail Report**.



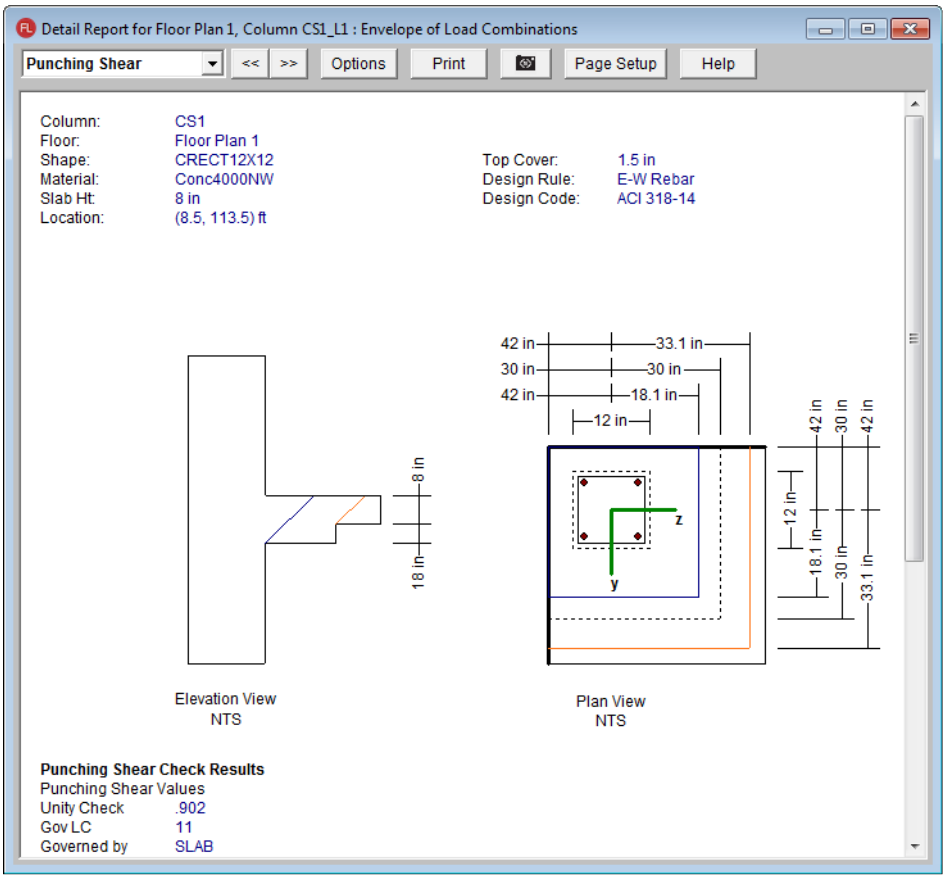
- Scroll down through the report to review the enveloped force diagrams, **Code Check**, **Interaction Diagrams,** and reinforcement design details.

Next, take a look at the **Punching Shear** results.

- From the drop-down menu in the upper left, select **Punching Shear**.



This will open the **Punching Shear Detail Report** for the column which now shows the passing code check.



- Scroll down through the report to review detail views, **Check Results**, and **Punching Shear Geometries**.
- When finished, close the **Detail Report** by clicking on the **Close** button  .

This is the end of Tutorial B4.

If you are using the Demonstration version of the program, you will not be able to save this model, as it exceeds the Demonstration version limits. Please proceed to the next tutorial for instructions on how to open the Tutorial B5 Starter File provided by RISA.

If you are using the full version of the program, you may save by doing the following: On the **File** menu, click **Save As** and enter a file name.

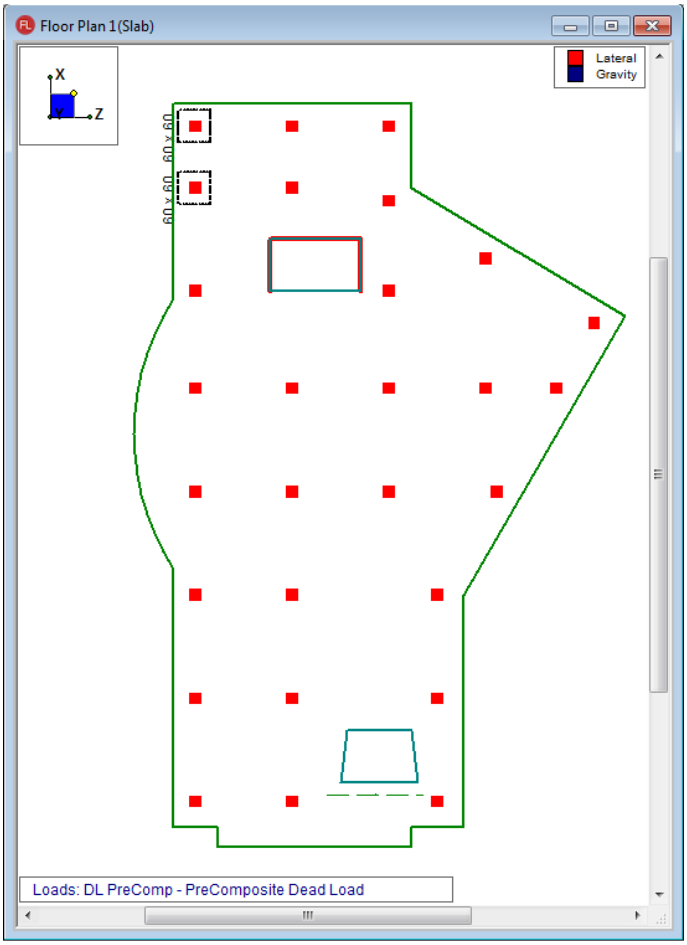
Part B: Tutorial 5 – RISA-3D Integration

This tutorial is intended to introduce you to the features of RISA-3D that directly relate to its interaction with RISAFloor ES. For more detailed information on RISA-3D, refer to the *RISA-3D User's Guide* and *RISA-3D General Reference*.

This tutorial continues from where the previous tutorial ended, so follow these steps to open the Tutorial B5 Starter File provided by RISA:

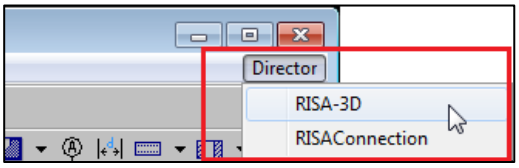
- Double-click on the **RISAFloor** icon to start the program.
- Click **Open Model**  . Double-click the **Tutorials** folder, select **Tutorial B5 Starter.rfl** and click **Open**. Click **Close**  (or **Cancel**) to exit the **Model Settings** dialog box.
- On the Window toolbar, click the **Graphic Editing** toolbar button  to activate the Drawing toolbar.
- Turn off the view of the design strips by clicking on the **Design Strip** button  on the Drawing toolbar. Click on the black menu arrow and select **None**.

Your screen should now look like this:



To see the interaction between RISAFloor ES and RISA-3D, solve your RISAFloor ES model, and then use RISA-3D to analyze and design your lateral system.

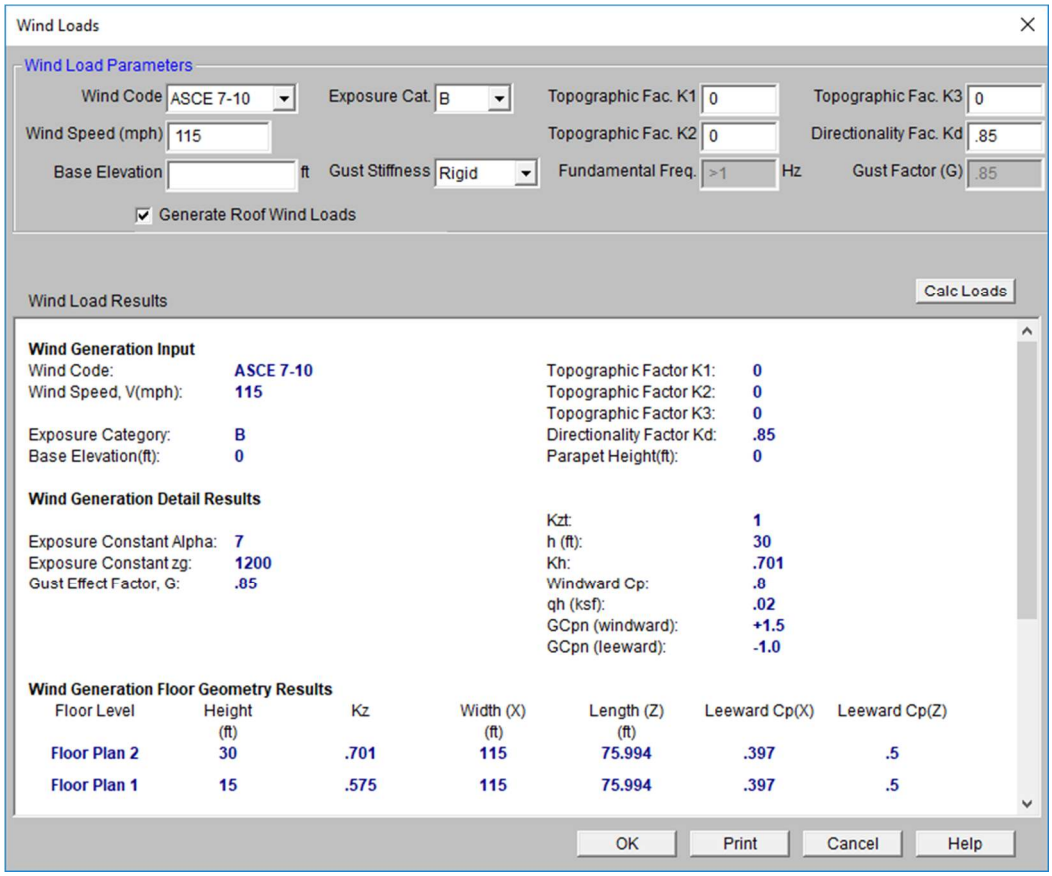
- On the RISA toolbar, solve the model by clicking **Solve** .
- Click **OK** from the **Solution Warning** dialog (if you are continuing from a previous solution).
- On the **Main** menu, click the **Director** button (top right) and select **RISA-3D**.



Lateral Load Generators

When you select RISA-3D you will be prompted for information on the wind and seismic loads of your structure.

First, the **Wind Loads** dialog box will display:

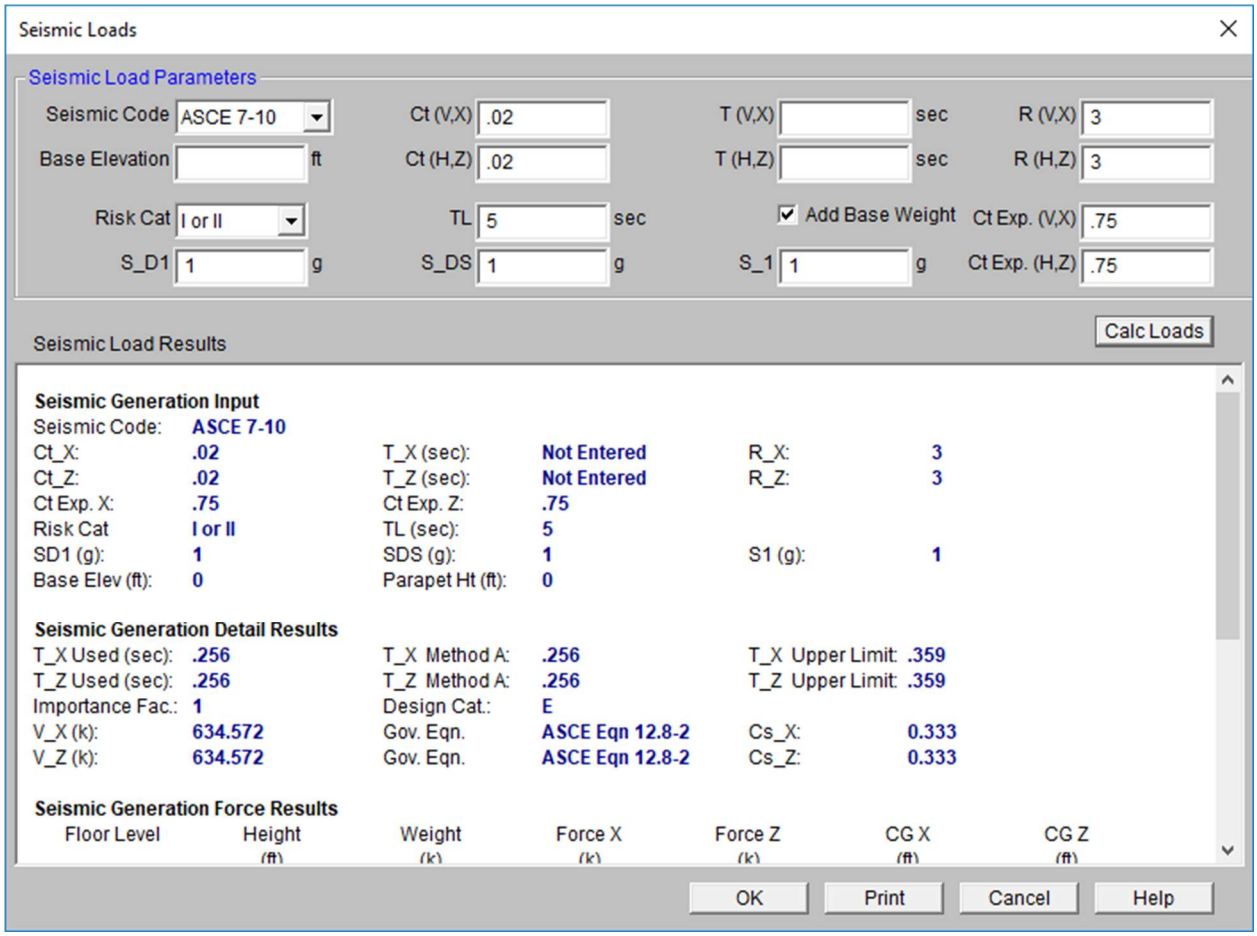


Review the information shown in the **Wind Loads** dialog box. In the future, if you make any changes in the **Wind Load Parameters** area, be sure to click **Calc Loads** to recalculate.

Be sure to also check the information generated in the **Wind Load Results** area. This is a summary of the wind loads that have been calculated for you.

- Click **OK** to accept the loads.

After the **Wind Loads** dialog box closes, the **Seismic Loads** dialog box will display:

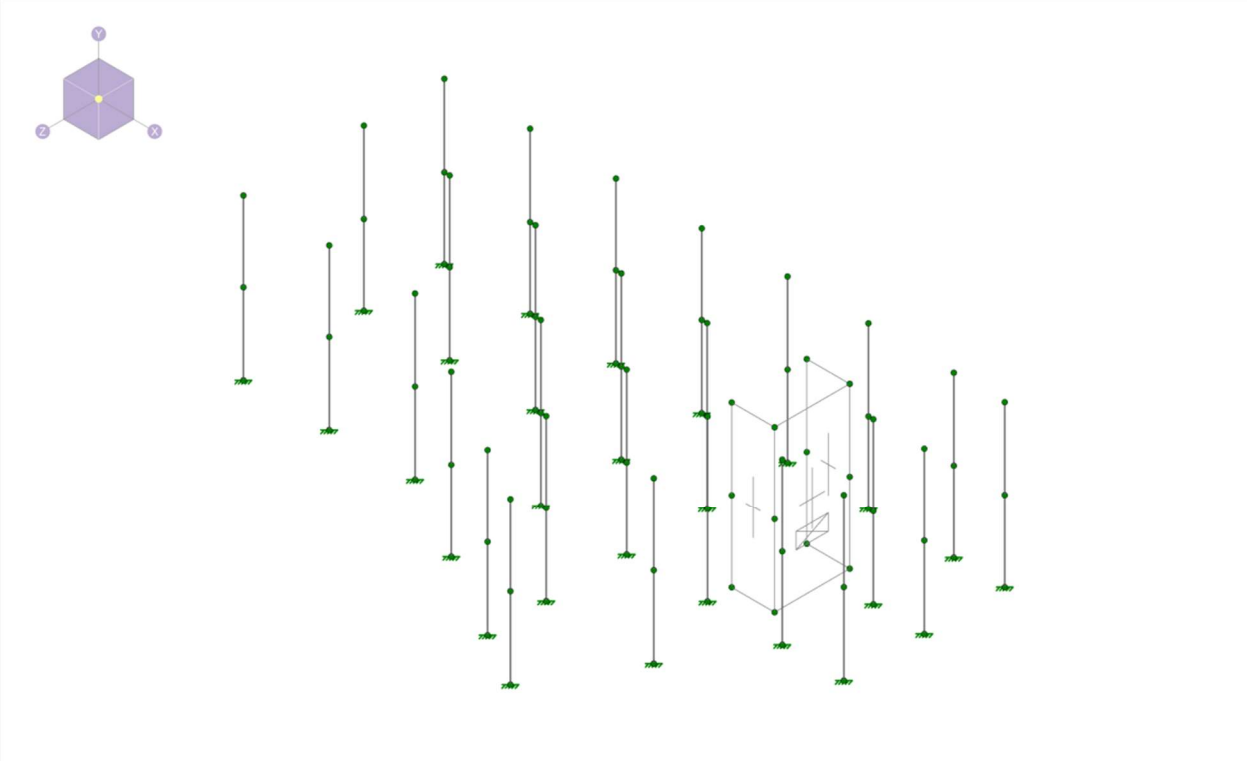


Review the information shown in the **Seismic Loads** dialog box. In the future, if you make any changes in the **Seismic Load Parameters** area, be sure to click **Calc Loads** to recalculate.

Be sure to also check the information generated in the **Seismic Load Results** area. This gives the equivalent static force distribution and base shear values that have been calculated for you.

- Click **OK** to accept the loads.
- If the **Model Settings** dialog box displays, close it by clicking **OK** or **Cancel**.

RISA-3D will now open and show an isometric view of your model. The interface of RISA-3D looks different from RISAFloor but the buttons work the same way.



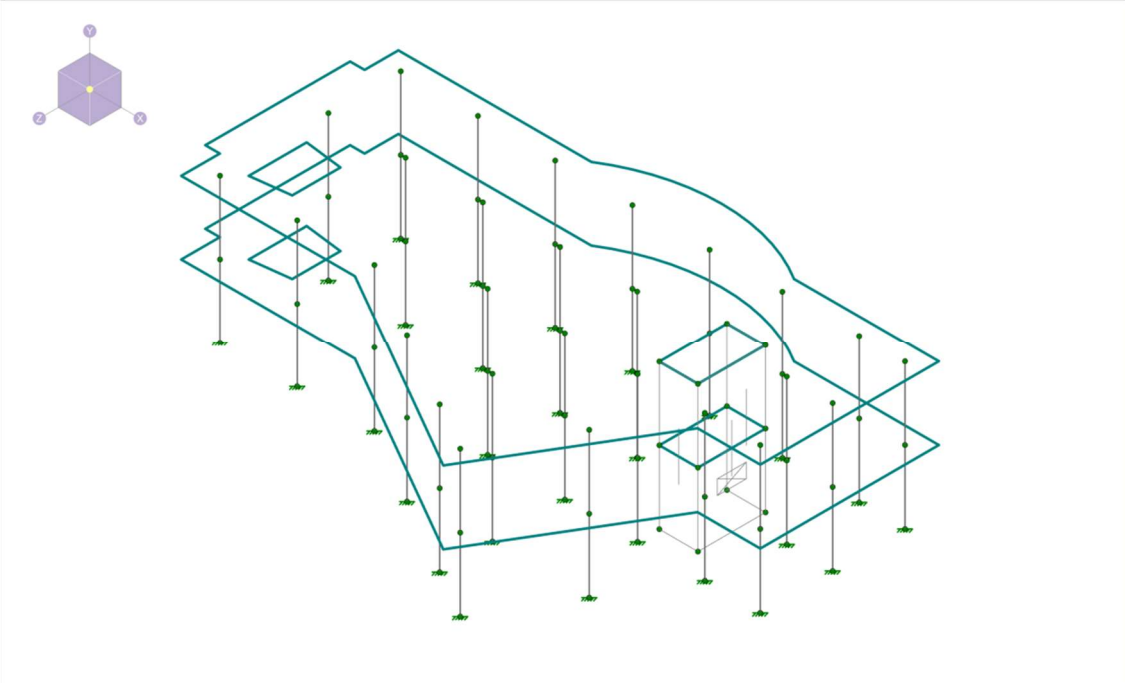
Notice that only the columns and wall panels that were specified as LATERAL members were generated in your RISA-3D model. You can always add vertical braces, modify your wall panels, or manipulate your model as you wish. For this tutorial, you will leave the model as it is.

RISA-3D Diaphragms

RISA-3D does not import the slab element, but you will have a rigid diaphragm in the exact shape and size of the slab that RISA-3D will use to distribute lateral loads to the columns and walls.

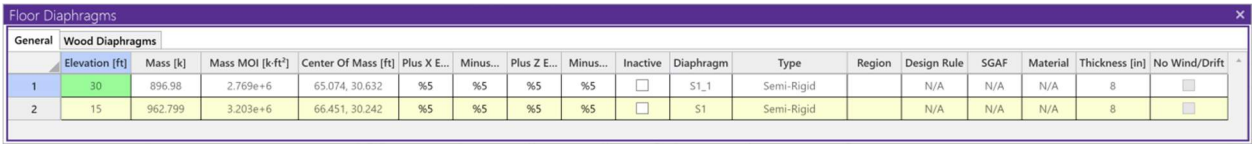
Before doing any graphical editing, turn on the display of the diaphragms:

- On the Window toolbar, click the **Toggle Diaphragm** button to turn on/off the display of the diaphragm edges and openings.



Next, look at the diaphragm information in the **Diaphragms** spreadsheet:

- On the **Data Entry** toolbar (at right), click **Diaphragms**.



	Elevation [ft]	Mass [k]	Mass MOI [k·ft²]	Center Of Mass [ft]	Plus X E...	Minus...	Plus Z E...	Minus...	Inactive	Diaphragm	Type	Region	Design Rule	SGAF	Material	Thickness [in]	No Wind/Drift
1	30	896.98	2.769e+6	65.074, 30.632	%5	%5	%5	%5	☐	S1_1	Semi-Rigid		N/A	N/A	N/A	8	☐
2	15	962.799	3.203e+6	66.451, 30.242	%5	%5	%5	%5	☐	S1	Semi-Rigid		N/A	N/A	N/A	8	☐

Notice that the diaphragms already contain the diaphragm mass and mass moment of inertia information that you would need for a dynamic analysis. In this spreadsheet, you can specify the percent of width you want RISA-3D to use to account for the accidental eccentricity in the X or Z directions.

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

RISA-3D Loading

Next, look at the loading information that was brought over from RISAFloor ES.

- On the **Data Entry** toolbar (at right), click **Basic Load Cases**. You can now review the load data that was generated.

As you review the data in this spreadsheet, notice that all your **Basic Load Cases** have been created for you, including wind and seismic load cases for each direction.

Basic Load Cases

	BLC Description	Category	X Gravity	Y Gravity	Z Gravity	Nodal	Point	Distributed	Area(Member)	Surface(Plate/Wall)
1	Dead Load	DL		-1			8	2		24
2	Live Load	LL					8			18
3	Live Load Special (public assemb	LLS								6
4	Roof Live Load	RLL								
5	Snow Load	SL								
6	Snow Load Nonshedding	SLN								
7	Rain Load	RL								
8	Wind Load X	WLX								
9	Partial X Wind Load 1	WLXP1								
10	Partial X Wind Load 2	WLXP2								
11	Wind Load Z	WLZ								
12	Partial Z Wind Load 1	WLZP1								
13	Partial Z Wind Load 2	WLZP2								
14	Earthquake Load X	ELX					56	9		18
15	Earthquake Load X Plus Z Eccentr	ELX+Z					56	11		26
16	Earthquake Load X Minus Z Eccent	ELX-Z					56	11		26
17	Earthquake Load Z	ELZ					56	9		18
18	Earthquake Load Z Plus X Eccentr	ELZ+X					56	9		20
19	Earthquake Load Z Minus X Eccent	ELZ-X					56	9		20
20	Other Load 1	OL1								
21	Other Load 2	OL2								
22	Other Load 3	OL3								
23	Other Load 4	OL4								
24	Wind Load Roof +X	WLX+R								
25	Wind Load Roof -X	WLX-R								
26	Wind Load Roof +Z	WLZ+R								
27	Wind Load Roof -Z	WLZ-R								
28	Semi-Rigid Wind Load +X	WL+X						40		
29	Semi-Rigid Wind Load -X	WL-X						40		
30	Semi-Rigid Wind Load +Z	WL+Z						44		
31	Semi-Rigid Wind Load -Z	WL-Z						44		

- When you are finished reviewing the data, click **Close** to close the spreadsheet and return to your model view.

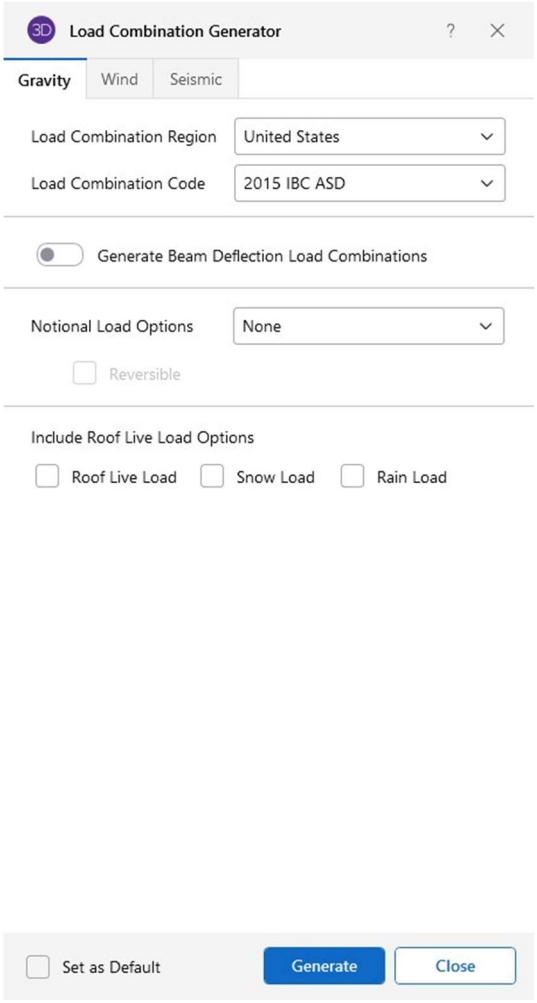
RISA-3D Load Combination Generator

Next, add the common load combinations used for this type of model, including all the code required eccentric loadings for wind and seismic loads. This will demonstrate how the program automatically generates these load combinations; saving you the time and energy normally spent entering them manually.

- On the **Data Entry** toolbar, click on **Load Combinations** to open the **Load Combinations** spreadsheet.

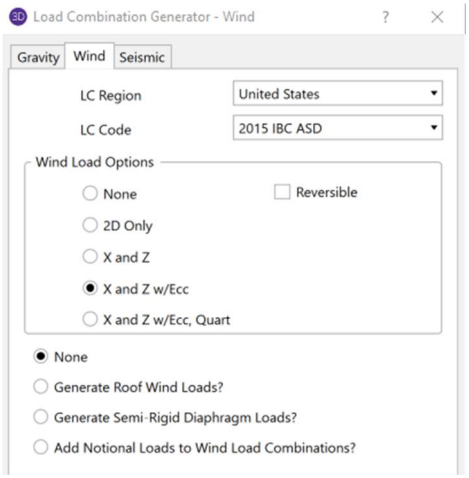
Although you previously generated load combinations for your RISAFloor model, you will now generate your lateral load combinations separately for use in the RISA-3D model.

- Click the **LC Generator** button within the **Load Combinations** spreadsheet.

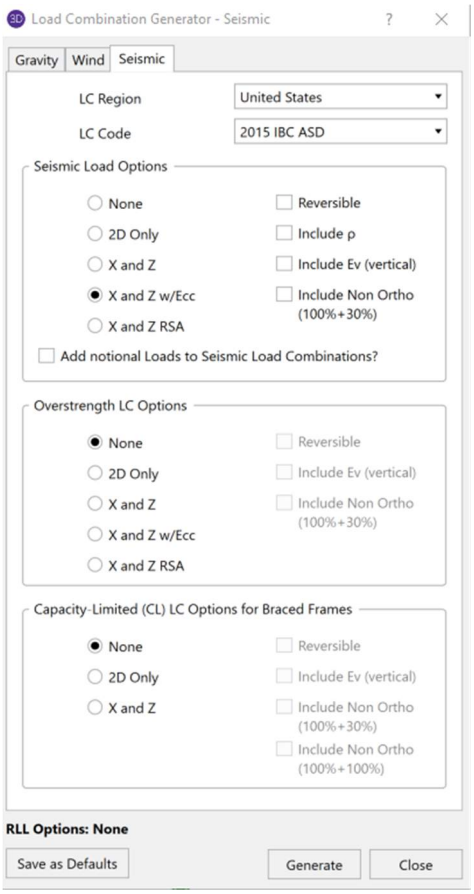


- On the first (**Gravity**) tab, in the **LC Region** list, select **United States**.
- On the first (**Gravity**) tab, in the **LC Code** list, select **2015 IBC ASD**.
- Toggle off the **Generate Deflection LCs** box and uncheck the **Roof Live Load Options**.
- Click **Generate**.

- Click on the **Wind** tab.
- Under **Wind Load Options**, click **X and Z w/Ecc** (make sure the **Reversible** check box is cleared).



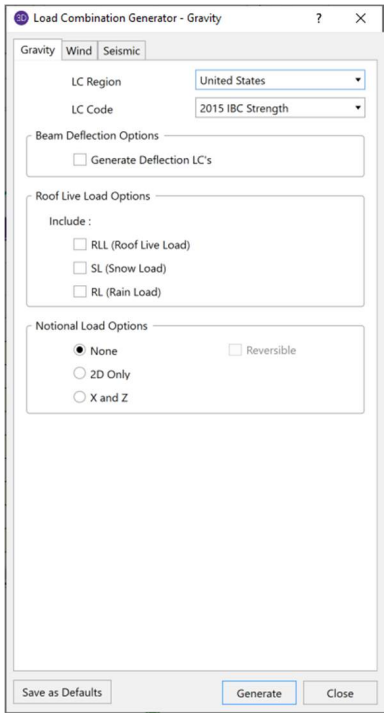
- Click **Generate**.
- Click on the **Seismic** tab.
- Under **Seismic Load Options**, click **X and Z w/Ecc** (make sure the **Reversible** check box is cleared).



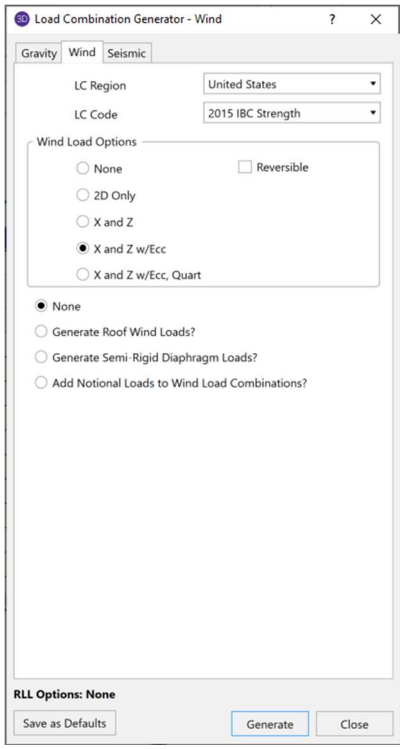
- Click **Generate**.

Go back and generate the strength level combinations.

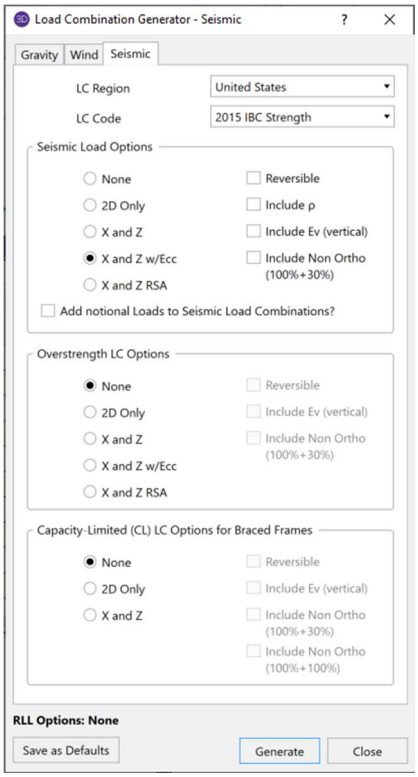
- Click back to the **Gravity** tab and change the **LC Code** list to **2015 IBC Strength**.



- Click Generate.
- Click on the **Wind** tab. Make sure that the **LC Code** also updated on this tab.

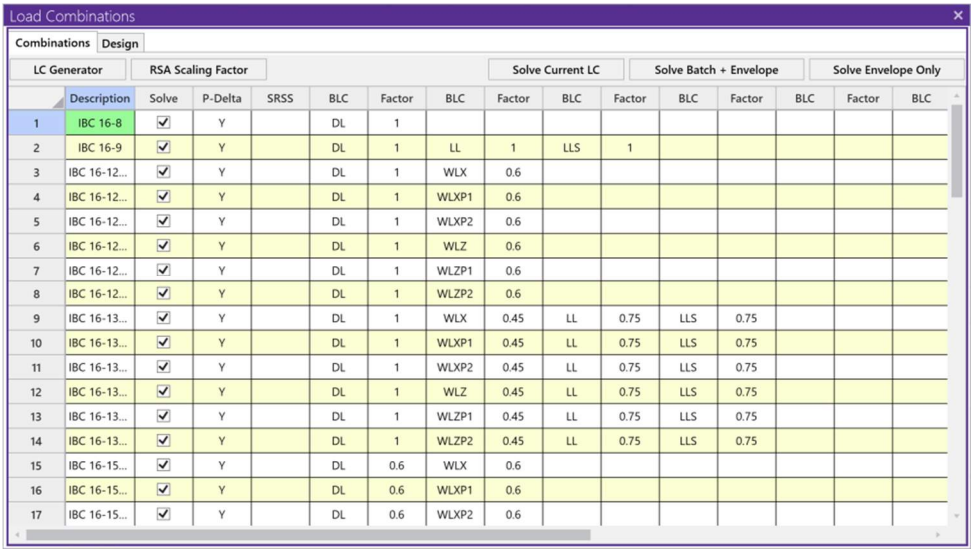


- Click Generate.
- Click on the **Seismic** tab. Make sure that the **LC Code** also updated on this tab.



- Click **Generate**. Then click **Close** to close the dialog.

This will generate 70 load combinations of which 17 are shown below.



	Description	Solve	P-Delta	SRSS	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC
1	IBC 16-8	☑	Y		DL	1									
2	IBC 16-9	☑	Y		DL	1	LL	1	LLS	1					
3	IBC 16-12...	☑	Y		DL	1	WLX	0.6							
4	IBC 16-12...	☑	Y		DL	1	WLXP1	0.6							
5	IBC 16-12...	☑	Y		DL	1	WLXP2	0.6							
6	IBC 16-12...	☑	Y		DL	1	WLZ	0.6							
7	IBC 16-12...	☑	Y		DL	1	WLZP1	0.6							
8	IBC 16-12...	☑	Y		DL	1	WLZP2	0.6							
9	IBC 16-13...	☑	Y		DL	1	WLX	0.45	LL	0.75	LLS	0.75			
10	IBC 16-13...	☑	Y		DL	1	WLXP1	0.45	LL	0.75	LLS	0.75			
11	IBC 16-13...	☑	Y		DL	1	WLXP2	0.45	LL	0.75	LLS	0.75			
12	IBC 16-13...	☑	Y		DL	1	WLZ	0.45	LL	0.75	LLS	0.75			
13	IBC 16-13...	☑	Y		DL	1	WLZP1	0.45	LL	0.75	LLS	0.75			
14	IBC 16-13...	☑	Y		DL	1	WLZP2	0.45	LL	0.75	LLS	0.75			
15	IBC 16-15...	☑	Y		DL	0.6	WLX	0.6							
16	IBC 16-15...	☑	Y		DL	0.6	WLXP1	0.6							
17	IBC 16-15...	☑	Y		DL	0.6	WLXP2	0.6							

Why so many load combinations? Each code load combination that includes an earthquake or wind load becomes six load combinations when you include all the possible eccentric cases (if you had selected the Reversible option, this would have expanded to 12 cases).

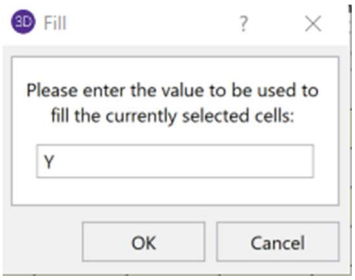
The **LC Generator** feature allows complete control over whether or not to include these more detailed load combinations. Load combinations or categories that are not relevant to your region can be completely eliminated by modifying the provided .xml spreadsheets.

For example, if you are in a year-round sunny climate like south Florida, do you really need to check your structure for all the combinations that include snow load? Therefore, to remove the snow loads from your generated load combinations, locate and modify the spreadsheet. The **United States** spreadsheet is located in **C:\users\Documents\RISA\Load Combination** (or C:\RISA\Load Combinations) and is named **United States.xml**.

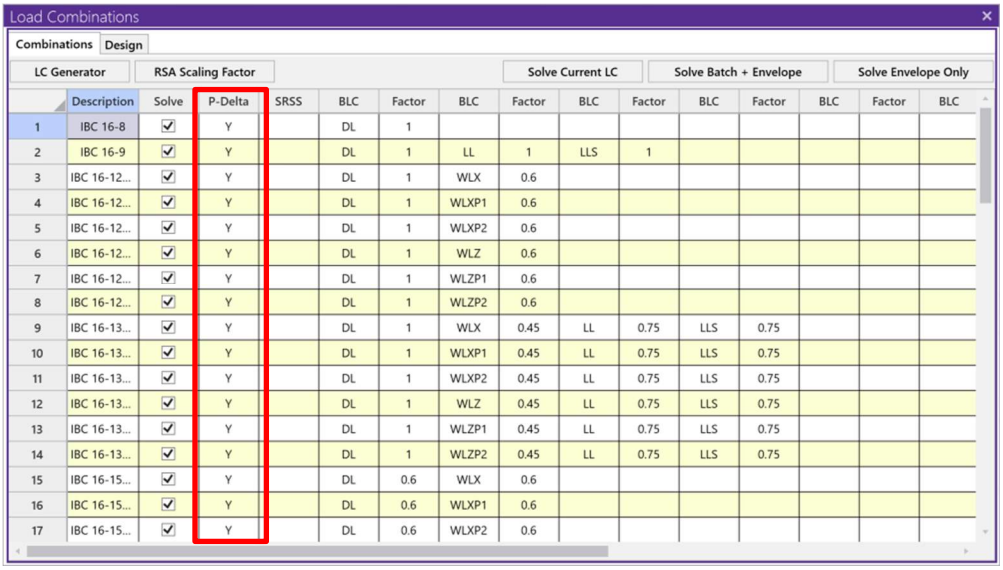
Note: Before editing the .xml spreadsheet, you may want to copy the original in case you want to restore all the settings back to the default.

With the requirements of the *AISC 14th Edition Steel Code* and the *ACI 318-14 Concrete Code*, you will need to include P-Delta effects in your results. This is not included by default, but you can include it in the **Load Combinations** spreadsheet.

- Click the heading of the **P-Delta** column to highlight the whole column.
- Click CTRL + F to open the **Block Fill** dialog box.
- Type "**Y**" into the dialog box and click **Ok**.



This will fill in the **P-Delta** column with **Y**, representing a "Yes, include P-Delta effects" for all the load combinations.



	Description	Solve	P-Delta	SRSS	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC
1	IBC 16-8	☑	Y		DL	1									
2	IBC 16-9	☑	Y		DL	1	LL	1	LLS	1					
3	IBC 16-12...	☑	Y		DL	1	WLX	0.6							
4	IBC 16-12...	☑	Y		DL	1	WLXP1	0.6							
5	IBC 16-12...	☑	Y		DL	1	WLXP2	0.6							
6	IBC 16-12...	☑	Y		DL	1	WLZ	0.6							
7	IBC 16-12...	☑	Y		DL	1	WLZP1	0.6							
8	IBC 16-12...	☑	Y		DL	1	WLZP2	0.6							
9	IBC 16-13...	☑	Y		DL	1	WLX	0.45	LL	0.75	LLS	0.75			
10	IBC 16-13...	☑	Y		DL	1	WLXP1	0.45	LL	0.75	LLS	0.75			
11	IBC 16-13...	☑	Y		DL	1	WLXP2	0.45	LL	0.75	LLS	0.75			
12	IBC 16-13...	☑	Y		DL	1	WLZ	0.45	LL	0.75	LLS	0.75			
13	IBC 16-13...	☑	Y		DL	1	WLZP1	0.45	LL	0.75	LLS	0.75			
14	IBC 16-13...	☑	Y		DL	1	WLZP2	0.45	LL	0.75	LLS	0.75			
15	IBC 16-15...	☑	Y		DL	0.6	WLX	0.6							
16	IBC 16-15...	☑	Y		DL	0.6	WLXP1	0.6							
17	IBC 16-15...	☑	Y		DL	0.6	WLXP2	0.6							

As you can see from this tutorial, the interaction between RISAFloor ES and RISA-3D is completely seamless. However, this tutorial is only an introduction to the many features of RISA-3D. We recommend downloading the RISA-3D User's Guide for further tutorials on this program's powerful features including solution, results, modifications, etc.

This is the end of Tutorial B5.

Note: If you save the model, you will notice that this file saves as a RISAFloor (.rfl) model, even though we have now been working in RISA-3D. This is because the file contains RISAFloor data that would be lost if the RISA-3D file were detached. To reopen the file in the future, you will need to open it in RISAFloor, solve, and use the **Director** tool to bring it back into RISA-3D.

Part B: Tutorial 6 – DXF Export

RISAFloor ES offers a DXF import/export feature that provides two-way compatibility with any other program that can read and write DXF files--this includes most major CAD programs and many analysis programs. With this feature, RISAFloor ES is able to produce CAD-quality drawings that list your beam sizes, camber, stud layout, end reactions, etc.

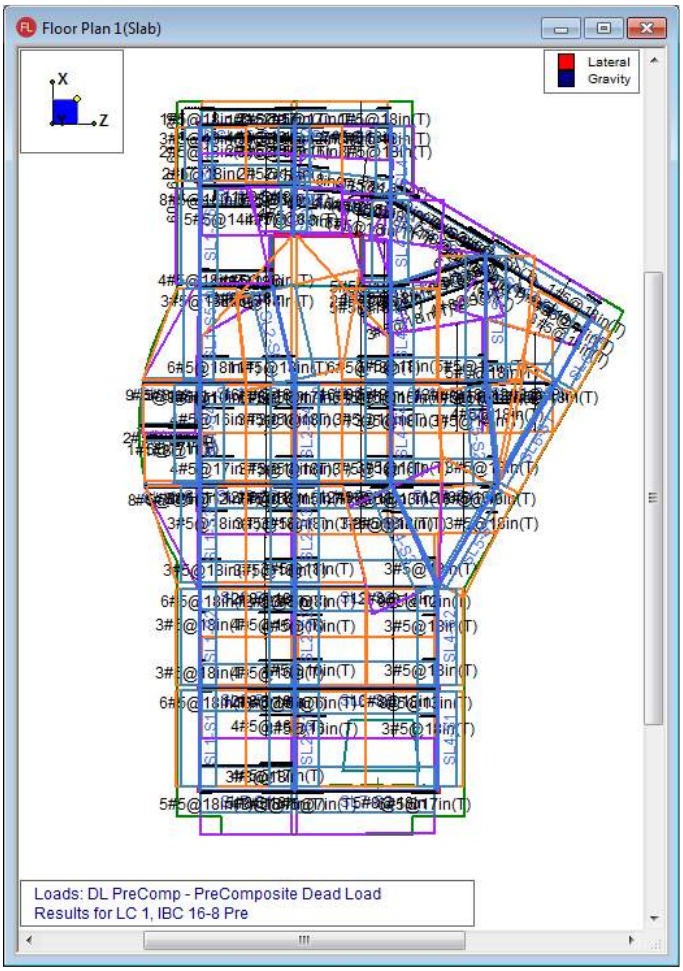
Follow the steps below to load the RISAFloor ES starter file for this portion of the tutorial:

- Double-click on the **RISAFloor** icon to start the program.
- Click **Open Model**  . Double-click the **Tutorials** folder, select **Tutorial B6 Starter.rfl** and click **Open**. Click **Close**  (or **Cancel**) to exit the **Model Settings** dialog box.

Solve the model and save:

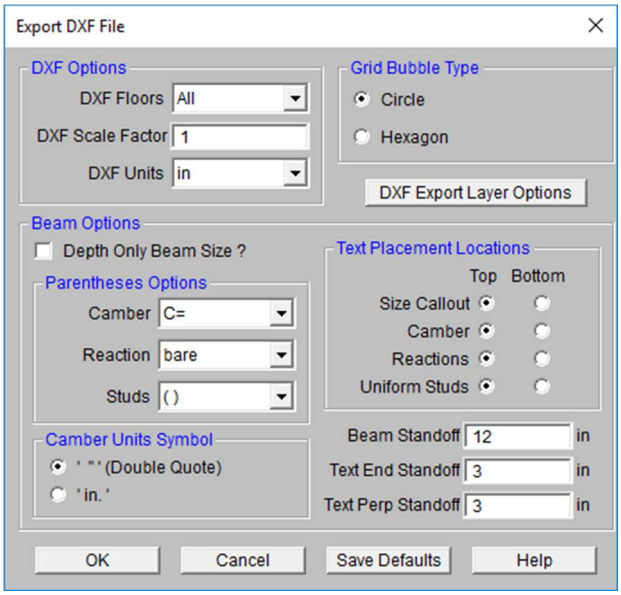
- Click **Solve**  to solve the model.
- On the **File** menu, click **Save As** and enter a new file name. Be sure to **Save Results**.

Your model should now display design data.

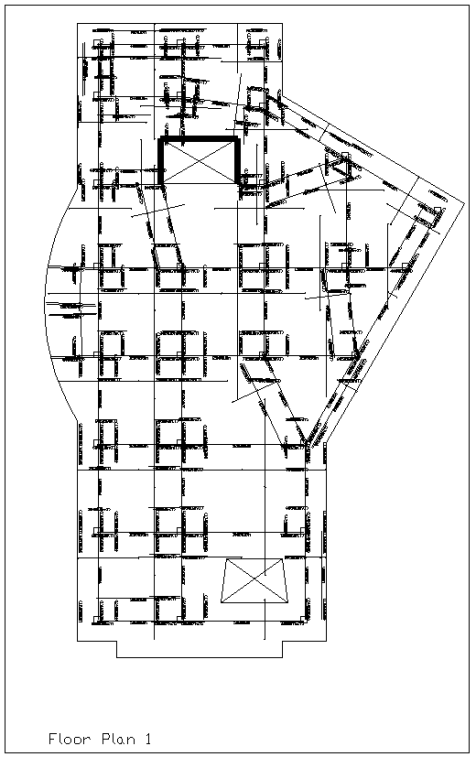


DXF Export

- On the **Main** menu, click **File**, click **Export**, and select **DXF File**.
- Enter the file name **Tutorial** and click **Save**. The **Export DXF File** dialog box will display.
- Enter the information shown below and click **OK**.



You should be able to open this DXF file with any standard drafting package. The following is an example of the information that can be written to the drawing file.



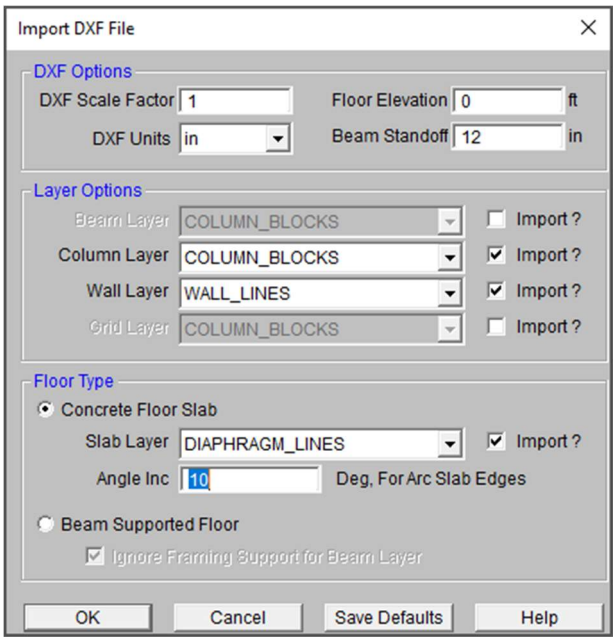
DXF Import

Similarly, you can import a DXF file into RISAFloor ES for analysis.

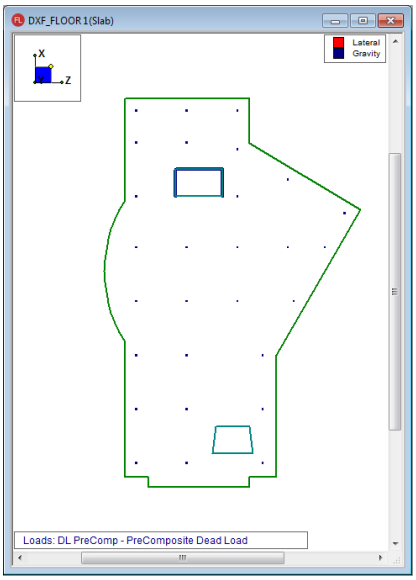
- On the RISA toolbar, click on **New Model** to start a fresh model.
- Click **Close** to exit the **Starting a Model** dialog box.
- On the **Main** menu, click **File**, click **Import**, then select **DXF File**. Select the first floor tutorial file you recently exported and saved. It should be named **Tutorial FLOOR_1.dxf**.

The **Import DXF File** dialog box will open.

- Enter the information shown below:



- Click **OK**. Your model will import and open a new floor from the imported DXF file.



This is the end of Tutorial 7.

Conclusion

This completes Part B: Elevated Slab Design.

Congratulations on completing your introductory tour of RISAFloor ES! 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 – RISAFloor Toolbar Button Quick Reference

RISA Toolbar

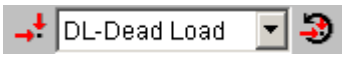
Button	Title	Label in RISAFoundation	Shortcut
	New Model Start a new model	Start a new model	(CTRL+N)
	Open Model Open an existing model	Reload from disk a previous model	(CTRL + O)
	Save Save current model to disk	Save the current model data to disk	(CTRL+S)
	Copy Copy to the clipboard	Copy to the clipboard	(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)
	(Global) Model Settings Set Model Settings	Set Model Settings	
	Define Units Define units to be used	Define units to be used	
	Shape Database	Edit or view the shape database	
	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	
	Create New Floor Plan	Create a new floor plan	

	Delete Floor Plans 	Delete existing floor plans	
	**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 plot options	Bring up the Model Display Options dialog	F2
	Floor Menu 	Select which floor you want to view	
	Snapshot	Save an image of the current model view for later use in the printed report	
Iso	**Isometric** Display an isometric view	Snap to an Isometric View	
Pln	**Plan** Display an XZ planar view	Snap to an XZ Planar View	
	Rotate buttons	Rotate the view counter-clockwise about the X axis	
		Rotate the view counter-clockwise about the Y axis	
		Rotate the view counter-clockwise about the Z axis	
	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
	3 buttons below collectively refer to **Loads Display** 		
	**Display Loads**  Toggle display of the loads (LC or CAT) **Loads List** – lists the available loads 	Toggle display of the loads (LC or CAT)	
	**Switch Loads**  Switch loads display (combinations or categories)	Switch loads display between combinations and categories	
	**Rendering**  Toggle between wireframe and rendering of members	Toggle between wireframe and rendering of members	
	**Joint Labels**  Joint labels toggle	Toggle the joint labels	
	**Beam Labels**  Beam labels toggle	Toggle the beam labels	
	Member End Options  Member end display options	Member end display options	
	**Show Wall Properties**  Wall Panel toggle	Toggles on the display of Wall Panel properties	
	Show Deck Assignment  Show Deck Assignment	Toggles on the display of the deck for beam supported floors only	
	Show Area Load Types  Show Area Load	Toggles on the display of the area loads	
	Project Grid Display  Show Project Grid toggle	Toggles on the display of the Project Grid	

	Distance Tool Distance Tool	Measures the distance between two points	F5
	Design Strip Rebar Design Strip rebar toggle	Toggles on the display of the Design Strip rebar	
	Design Strip Design Strip toggle	Toggles on the display of the Design Strips and/or the Support Lines	
	Design Strip Diagram Design Strip force diagram toggle	Toggles on the display of the force diagram on the Design Strip	

…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
	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 Columns Draw or modify columns	Draw or modify columns	
	Draw or Modify Walls Draw or modify walls	Draw or modify walls	
	Draw or Modify Beams Draw or modify beams	Draw or modify beams	
	Generate Infill Beams Generate infill beams	Generate infill beams within bays	
	Elevate Points Change the elevation of selected points	Change the elevation of selected points in the model	
	Draw Slabs/Diaphragms Draw slabs or diaphragms	Draw or modify slabs or diaphragms	
	Assign Deck Assign deck properties	Assign deck properties	
	Add Wood Diaphragm Regions Add wood diaphragm regions	Add wood diaphragm regions	
	Draw Area Loads Draw area loads	Draw area loads	
	Draw Line Loads Draw line loads point to point	Draw line loads point to point	
	Assign Point Loads Draw point loads	Draw point loads	

	 Project Grid Draw Project Grid Lines	Draw Project Grid Lines
	 Move	Move selected parts of the model
	 Copy	Copy selected parts of the model
	 Offset Copy	Copy selected parts of the model using an offset distance
	 Extend Extend the member	Extend the member
	 Trim Trim the member	Trim the member
	 **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	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