# **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) <u>risa.com</u>

Copyright © 2024, RISA Tech, Inc. All Rights Reserved. RISA is part of the Nemetschek Group.

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

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

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

## **Table of Contents**

| Introduction1                                     |  |
|---|--|
| How to Use this Book1                             |  |
| Using the Online Help3                            |  |
| Technical Support Information4                    |  |
| RISA Online4                                      |  |
| Before You Begin5                                 |  |
| RISAFloor Overview5                               |  |
| Hardware Requirements5                            |  |
| Program Limits                                    |  |
| License Agreement7                                |  |
| Installation7                                     |  |
| First Look at RISAFloor8                          |  |
| Starting RISAFloor8                               |  |
| Windows and Dialog Boxes8                         |  |
| Menus and Toolbars10                              |  |
| Managing Windows, Model Views, and Spreadsheets14 |  |
| Working in Spreadsheets15                         |  |
| Part A: Beam Supported Floors16                   |  |
| Part A: Tutorial 1 – Columns and Walls17          |  |
| Starting a New File                               |  |
| Project Grid19                                    |  |
| Model Settings                                    |  |
| Columns24   |  |
| Wall Panels32                                     |  |
| Part A: Tutorial 2 – Modeling Beams               |  |
| Primary Framing40                                 |  |
| Infill Framing45                                  |  |
| Additional Beam Drawing Tips (optional)49         |  |

| Part A: Tutorial 3 – Decks & Diaphragms53  |
|--|
| Creating Diaphragm Edges53   |
| Deck Properties57  |
| Part A: Tutorial 4 – Loading64   |
| Area Loads65   |
| Line Loads71   |
| Point Loads76  |
| Part A: Tutorial 5 – Solution & Results78  |
| Load Combinations79  |
| Solving and Reviewing Results82  |
| Modifying the Model87  |
| Part A: Tutorial 6 – RISA-3D Integration89   |
| RISA-3D Lateral Load Generators91  |
| RISA-3D Load Combinations98  |
| Part A: Tutorial 7 – RISA Interoperability102  |
| Fart A. Tutohar 7 – Kisa Interoperability  |
| RISA-Revit Link  |
|  |
| RISA-Revit Link  |
| 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  |
| 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  |
| 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  |
| 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                               |
| 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 |
| RISA-Revit Link102CIS/2 Translator105RISAFloor & CAD106Conclusion109Part B: Elevated Slab Design110Part B: Tutorial 1 – Columns & Walls112Starting a New File112Drawing Grid114Columns117Wall Panels125Wall Panel Openings130  |
| RISA-Revit Link102CIS/2 Translator105RISAFloor & CAD106Conclusion109Part B: Elevated Slab Design110Part B: Tutorial 1 – Columns & Walls112Starting a New File112Drawing Grid114Columns117Wall Panels125Wall Panel Openings130Part B: Tutorial 2 – Slab & Design Strips134  |

| Support Lines and Design Strips                       |     |
|---|-----|
| Part B: Tutorial 3 – Loading                          |     |
| Area Loads  |     |
| Line Loads  |     |
| Point Loads   |     |
| Part B: Tutorial 4 – Solution & Results               |     |
| Load Combinations                                     |     |
| Solving the Model                                     |     |
| Spreadsheet Results                                   |     |
| Viewing Graphical Results                             |     |
| Detail Reports  |     |
| Part B: Tutorial 5 – RISA-3D Integration              |     |
| Lateral Load Generators                               |     |
| RISA-3D Diaphragms                                    | 201 |
| RISA-3D Loading                                       | 202 |
| RISA-3D Load Combination Generator                    | 203 |
| Part B: Tutorial 6 – DXF Export                       |     |
| 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                                     |     |

## 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 <u>First Look at RISAFloor</u> on page 8 to familiarize yourself with the RISAFloor menus, toolbars, and shortcuts. <u>Appendix A – RISAFloor Toolbar Button Quick Reference</u> 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: <u>Part A</u> and <u>Part B</u>, 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.

<u>Part A – Beam Supported Floors</u> will guide you step-by-step through the RISAFloor modeling process to build and analyze a model using beam supported floors; <u>Part B – Elevated Slab Design</u> 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 <a href="https://www.risa.com/documentation">https://www.risa.com/documentation</a>.

### **Document Conventions**

The following conventions are used throughout this book:

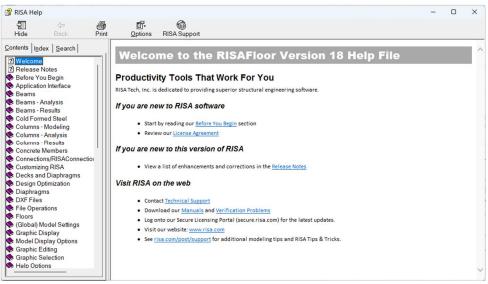
| This convention: | Indicates:  |
|------------------|---|
| CAPITAL LETTERS  | Names of keys on the keyboard – for example, SHIFT, CTRL, or ALT.                         |
| KEY+KEY          | One key should be held down and then another key pressed – for example, CTRL+P or ALT+F4. |
| Bold text        | User interface options – for example, <b>File</b> menu.                                   |
| Boxed text       | Notes or modeling tip information.  |
| Bulleted text    | Action item for building the tutorial model.  |
| ۲                | 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 S. 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<br>specific<br>feature<br>(sontoxt | As you work, notice the <b>Help</b> buttons at the bottom of many of the dialog boxes.<br>These provide direct access to the Help information related to the task you are<br>performing. |
|--|--|
| (context-<br>sensitive help)                 | This context-sensitive help may be accessed by pressing the <b>Help</b> button on the dialog box or by pressing the F1 key.  |
| Help on<br>toolbar<br>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.                  |
|  | Ison   |

Snap to an Isometric View

## **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: <u>www.risa.com/support</u>.

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 <a href="http://www.risa.com">http://www.risa.com</a>. 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: <a href="mailto:support@risa.com">support@risa.com</a>

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 <u>http://www.risa.com</u> 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 (<u>license@risa.com</u>) 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.

23

8

Start RISAFloor as follows:

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

#### Title bar Main menu **RISA** toolbar Minimize Maximize Close RISAFloor - [C:\Users\Documents\RISA\Model Files\untitled.rfl] View Insert Modify Spreadsheets Solve Result File Edit Settings Tools Window Help Director D 🛎 🖬 🖶 🗠 🗠 🌒 🏭 I 🛄 🏭 😻 🕺 🗒 🖬 🕼 LC = 📗 🖹 🛃 🗍 🖗 Data Entry Project Grid Materials Deck Definition: Member Design Rules Starting a Model X Wall Design Rules Connection Rules Create a New Model. Area Load Definitions Point Locations Click here to create a new floor plan Columns Column Stacks Wall Panels or Reload a Previously Created Model Beams Diaphragms Click here to go to the File Open dialog Point Loads Or click here to open a recently accessed file Line Loads Tapered Area Loads Load Combinations Floors Don't show this panel when starting a model? Close S Press Ctrl-N to start a New File Status bar Starting a Model dialog box Data Entry Workspace

## 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.  |  |  |  |  |  |  |  |
|------------------------|--|--|--|--|--|--|--|--|
|                        | RISAFloor - [C:\Users\Documents\RISA\Model Files\untitled.rfl]   |  |  |  |  |  |  |  |
|                        | To move the window, press and hold the title bar with your mouse, then drag to the desired location.   |  |  |  |  |  |  |  |
| Minimize,<br>Maximize, | The three buttons  |  |  |  |  |  |  |  |
| Close                  | • Click <b>Minimize</b> lo minimize the window to a button on the taskbar.   |  |  |  |  |  |  |  |
|                        | <ul> <li>Click Maximize I to maximize the window to full screen. Once it is full screen, click Restore Down I to restore the window down to its original size.</li> </ul>  |  |  |  |  |  |  |  |
|                        | Click <b>Close</b> ito 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 <b>Starting a New Model</b> 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 <b>Member Redesign</b> dialog box. |  |  |  |  |  |  |  |
|                        | Flat Slab, 0 deg         0, 28, 0 (ft)   |  |  |  |  |  |  |  |
|                        | To the right of the "S" are 3 status boxes:  |  |  |  |  |  |  |  |
|                        | <ul> <li>The first status box displays general information relative to the task you are performing.</li> </ul>   |  |  |  |  |  |  |  |
|                        | • The second (middle) status box reports information about your deck or area load.   |  |  |  |  |  |  |  |
|                        | <ul> <li>The third status box (on the far right) reports the cursor coordinates as you work<br/>in the model view. This will be demonstrated throughout the tutorial.</li> </ul>   |  |  |  |  |  |  |  |
| Dialog<br>boxes        | Dialog boxes are windows that help you perform a specific function within RISAFloor. For example, the <b>Starting a Model</b> dialog box is presented when you first open RISAFloor, which helps you find the file you wish to open.   |  |  |  |  |  |  |  |

## **Menus and Toolbars**

#### Main Menu

<u>File Edit Settings View Insert Modify Spreadsheets Solve Results Tools Window Help</u> Director

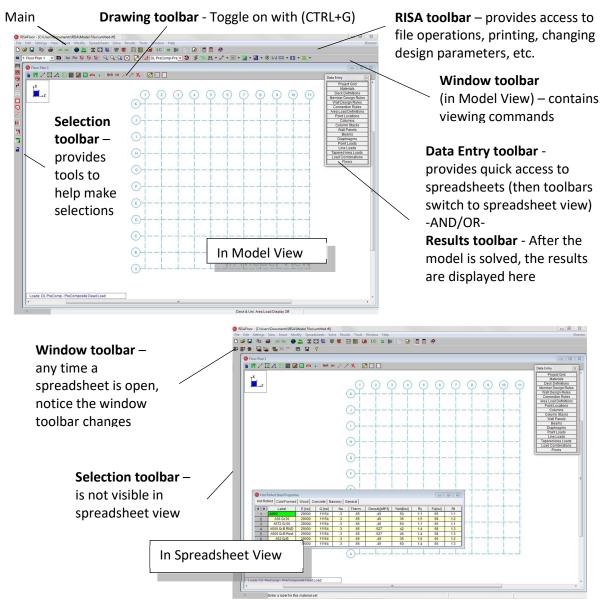
The Main menu and its submenus provide access to all features RISAFloor has to offer, as summarized below:

| 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.<br>You may use this menu to add or remove information from the spreadsheets or<br>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 <b>Spreadsheets</b> 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 <b>Application Settings</b> 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 <u>Using the Online Help</u> 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.



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   |
|---|
| 📼 1:Floor Plan 1 🔄 🔯   Iso Pln 🗞 🦃 🤤 🔍 🔍 🔍 🔯 💋 斗 DL PreComp-Pre 🚽 😓 🗗 🐂 🥂 🛪 🖉 🖌 🔛 🛪 📓 🖉 👘 🕌 🖛 🖓 🚧 📼 🛪 🕅 🛪 🚈 |

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 | Spre | eadsh | leet Viev | v |   |   |   |        |
|----|------|-------|-----------|---|---|---|---|--------|
| ₽  | Ľ    | •     | 12.3 to-  | ∎ | X | 瓃 | Ê | ,<br>v |

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

| i. | 17    | /1    | 3 🟒   | 2 🗐   | 1 | ш | ł | <b>%</b> | ₿◆R R◆R | + | 1 | × | X |   |  |
|----|-------|-------|-------|-------|---|---|---|----------|---------|---|---|---|---|---|--|
| 0  | n Cor | crete | Floor | Slabs |   |   |   |          |         |   |   |   |   |   |  |
| li | 17    | / 8   | - +   | X 🗖   | + | щ | ł | ٥        | ₽+R R+R | + | 1 | × | Х | 2 |  |

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.

r

#### 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 **D** or the **Results** button **R**.

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.

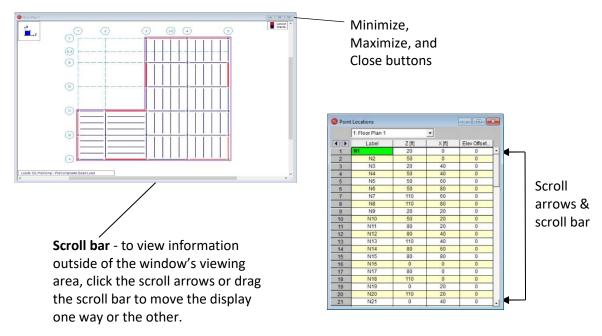
| Data Entry            | 8          |
|-----------------------|------------|
| Project               | t Grid     |
| Mater                 | ials       |
| Deck Def              | initions   |
| Slab Def              | initions   |
| Results 🛛 🛛           | ign Rules  |
| Design Results        | n Rules    |
| Code Checks           | h Rules    |
| Shear Results         | n Rules    |
| Bending Results       | efinitions |
| Deflection Results    | ations     |
| Vibrations            | nns        |
| End Reactions         | Stacks     |
| Column Forces         | nels       |
| Column Results        | ns         |
| Member/Wall Rebar     | gms        |
| Slab Point Deflection | S          |
| Slab Strip Deflection | n Strips   |
| Slab Results/Rebar    | bads       |
| Slab Forces           | ads        |
| Punching Shear        | a Loads    |
| Wall Forces           | inations   |
| Wall Results          | 'S         |
| Material TakeOff      |            |
| Connection Results    |            |
| Warning Log           |            |
|                       | -          |

**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.



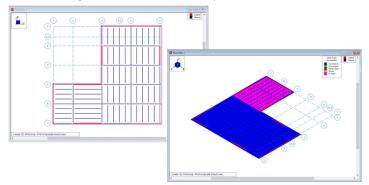
#### Managing Model Views

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.

explanation of the current column.

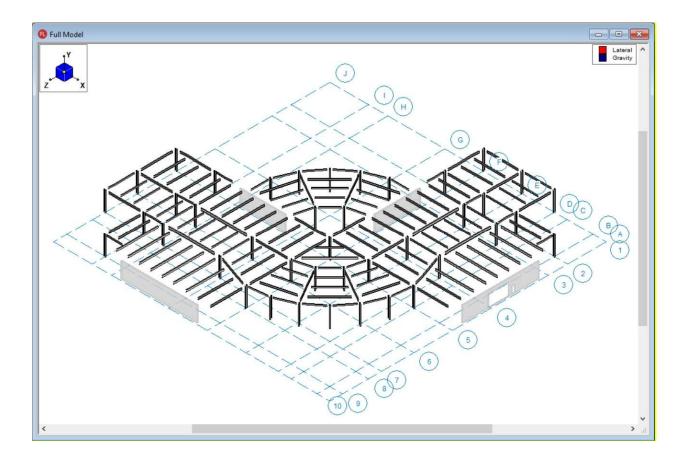
|    | 1: Floor Plan 1 |        | *      |             | column label to select                                  |
|----|-----------------|--------|--------|-------------|---|
|    | Label           | Z [ft] | X [ft] | Elev Offset | the entire column                                       |
| 1  | N1              | 119    | 119    | 0           |   |
| 2  | N2              | 30     | 204    | 0           |   |
| 3  | N3              | 204    | 30     | 0           | N   |
| 4  | N4              | 0      | 149    | 0           |   |
| 5  | N5              | 0      | 174    | 0           | Scroll bar  |
| 6  | N6              | 30     | 149    | 0           |   |
| 7  | N7              | 30     | 174    | 0           |   |
| 8  | N8              | 45     | 149    | 0           |   |
| 9  | N9              | 45     | 174    | 0           |   |
| 10 | N10             | 89     | 149    | 0           | Row of cells – Click                                    |
| 11 | N11             | 89     | 174    | 0           | the row label (at left)                                 |
| 12 | N12             | 149    | 0      | 0           | to select the entire                                    |
| 13 | N13             | 149    | 30     | 0           |   |
| 14 | N14             | 149    | 45     | 9           | row   |
| 15 | N15             | 149    | 89     | 0           |   |
| 16 | N16             | 174    | 0      | 0           |   |
| 17 | N17             | 174    | 30     | 0           | Column of cells   |
| 18 | N18             | 174    | 45     | 0           |   |
| 19 | N19             | 174    | 89     | 0           | -   |
|    |                 |        |        |             | Click any cell.<br>Notice the status<br>bar displays an |

## **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:

| ۲ | On your desktop home screen | , double click on the <b>RISAFloor</b> icon 拜 | Ļ |
|---|-----------------------------|---|---|
|---|-----------------------------|---|---|

| RISAFloor - [C:\Users\Documents\RISA\Model Files\untitled.rfi]  |   |
|---|---|
| File Edit Settings View Insert Modify Spreadsheets Solve Results Tools Window Help  | Director  |
| File Edit Settings View Insert Modify Spreadsheets Solve Results Tools Window Help         Image: I | Data Entry 3<br>Project Grid<br>Materials<br>Deck Definitions<br>Member Design Rules<br>Wall Design Rules<br>Connection Rules<br>Area Load Definitions<br>Point Locations<br>Columns<br>Column Stacks<br>Wall Panels<br>Beams<br>Diaphragms |
| <ul> <li>Click here to go to the File Open dialog</li> <li>Or click here to open a recently accessed file :</li> <li>Don't show this panel when starting a model?</li> </ul>  | Point Loads<br>Line Loads<br>Tapered Area Loads<br>Load Combinations<br>Floors  |
| S Press Ctrl-N to start a New File  |   |

**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.

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.

| Create New Floor                | Plan                            | × |  |  |  |
|---------------------------------|---------------------------------|---|--|--|--|
| Label: Floor Plan               | 1                               |   |  |  |  |
| Floor Type                      |                                 |   |  |  |  |
| C Concrete Flo                  | or Slab                         |   |  |  |  |
| Beam Suppo                      | rted Floor                      |   |  |  |  |
| -Floor Creation Opt             | ions                            |   |  |  |  |
| Original Floor                  | r                               |   |  |  |  |
| Copy of No                      | Floors Defined                  |   |  |  |  |
|                                 | opy Columns                     |   |  |  |  |
|                                 | opy Beams                       |   |  |  |  |
|                                 | opy Diaphragm Edges/Openings    |   |  |  |  |
| Copy Diaprilagin Edges/openings |                                 |   |  |  |  |
|                                 | opy Loads                       |   |  |  |  |
| <b>V</b> 0                      | opy Support Lines/Design Strips |   |  |  |  |
| <b>N</b> 0                      | opy Wall Panels                 |   |  |  |  |
|                                 | S From Floor Below              |   |  |  |  |
|                                 | From Floor Above                |   |  |  |  |
| Child of: No                    | Floors Defined                  |   |  |  |  |
| Elevation:                      | 15 ft                           |   |  |  |  |
| Default Area Load:              | Office                          |   |  |  |  |
|                                 |                                 |   |  |  |  |
| Default Deck:                   | Concrete Deck 🗨                 |   |  |  |  |
| Default Deck:<br>Deck Angle:    | Concrete Deck                   |   |  |  |  |

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.

| Lines Arcs   |
|--|
| Project Grid Origin (ft)<br>Z Axis 0                             |
| × Axis 0     Rectangular Grid Parameters                         |
| Z Axis Increments 30,15,44,30,30,25,10,20,10                     |
| X Axis Increments 30,15,44,30,30,25,10,20,10<br>Skew Angle 0 deg |

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.

| Z Direction/Radial Line Labeling  | g                  |
|-----------------------------------|--------------------|
| Start Label 1                     | (Letter or Number) |
| Label Order: ① A to Z             | C Z to A           |
| -X Direction/Radial Line Labeling | g                  |
| Start Label J                     | (Letter or Number) |
| Label Order: O A to Z             | ⓒ Z to A           |

| Project Grid Lines ? X                                  | < |
|---|---|
| Lines Arcs  |   |
| Project Grid Origin (ft)                                |   |
| Z Axis 0  |   |
| X Axis 0  |   |
| Rectangular Grid Parameters                             |   |
| Z Axis Increments 30,15,44,30,30,25,10,20,10            |   |
| X Axis Increments 30,15,44,30,30,25,10,20,10            |   |
| Skew Angle 0 deg  |   |
| Radial Grid Parameters                                  |   |
| Start Angle deg   |   |
| Angle Increments 6@30 de                                | g |
| Radial Increments 3@10 (ft)                             |   |
| Z Direction/Radial Line Labeling                        |   |
| Start Label 1 (Letter or Number)                        |   |
| Label Order: <ul> <li>A to Z</li> <li>Z to A</li> </ul> |   |
| × Direction/Radial Line Labeling                        | _ |
| Start Label J (Letter or Number)                        |   |
| Label Order: O A to Z O Z to A                          |   |
| Save Current Settings as Defaults?                      |   |
| Apply Undo Ok Cancel                                    |   |

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

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

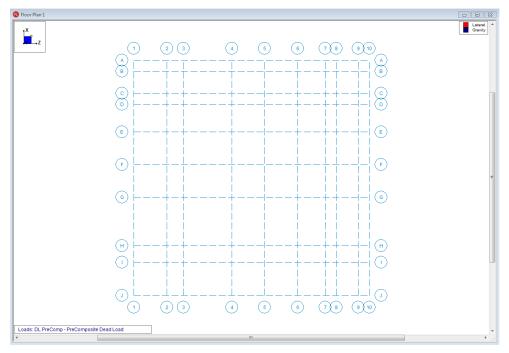
You can verify the values in the **Project Grid Lines** spreadsheet and modify the grid lines if needed.

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

| 📵 Project Grid Lines |                           |              |            |              |            |              |            |
|----------------------|---------------------------|--------------|------------|--------------|------------|--------------|------------|
| Lines Arcs           |                           |              |            |              |            |              |            |
|                      | · ·                       |              |            |              |            |              |            |
| Ge                   | nerate 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        |              | Z          |
| 3                    | 3                         | 45           | 45         | 0            | 214        | V            | Z          |
| 4                    | 4                         | 89           | 89         | 0            | 214        |              | Z          |
| 5                    | 5                         | 119          | 119        | 0            | 214        | V            | Z          |
| 6                    | 6                         | 149          | 149        | 0            | 214        |              | V          |
| 7                    | 7                         | 174          | 174        | 0            | 214        | V            | Z          |
| 8                    | 8                         | 184          | 184        | 0            | 214        |              | V          |
| 9                    | 9                         | 204          | 204        | 0            | 214        |              | 2          |
| 10                   | 10                        | 214          | 214        | 0            | 214        |              | Z          |
| 11                   | J                         | 0            | 214        | 0            | 0          | V            | Z          |
| 12                   |                           | 0            | 214        | 30           | 30         |              | ×          |
| 13                   | Н                         | 0            | 214        | 45           | 45         | V            | Z          |
| 14                   | G                         | 0            | 214        | 89           | 89         |              | V          |
| 15                   | F                         | 0            | 214        | 119          | 119        |              | 2          |
| 16                   | E                         | 0            | 214        | 149          | 149        |              | V          |
| 17                   | D                         | 0            | 214        | 174          | 174        | V            | Z          |
| 18                   | С                         | 0            | 214        | 184          | 184        |              | V          |
| 19                   | В                         | 0            | 214        | 204          | 204        | V            | Z          |
| 20                   | A                         | 0            | 214        | 214          | 214        |              | V          |

**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:



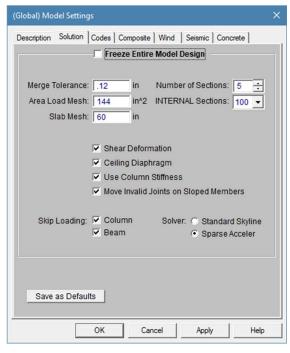
## **Model Settings**

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

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

| (Global) Model Settings  | × |
|--|---|
| Description   Solution   Codes   Composite   Wind   Seismic   Concrete |   |
|  |   |
| Model Title  |   |
| Company RISA   |   |
| Designer H. Cross  |   |
| Job Number   |   |
| Checked By   |   |
| Notes  |   |
| Save as Defaults   |   |
| OK Cancel Apply Help   |   |

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:

| Global) Model Settings                                    |                 |
|---|-----------------|
| Description   Solution Codes   Composite   Wind   Seismic | Concrete        |
| HR Steel : AISC 15th (360-16); ASD                        | <b>•</b>        |
| Connections : AISC 15th (360-16): ASD                     |                 |
| CF Steel : AISI S100-20: ASD                              | -               |
| CFS Walls : AISI S400-20 & AISI S240-20                   | ~               |
| Concrete : ACI 318-19                                     | -<br>Temperatur |
| Wood : AWC NDS-18 / SDPWS-21 ASD                          | < 100F          |
| Masonry : TMS 402-16: ASD                                 | •               |
| Steel Joist : SJI 45th Edition: ASD                       | -               |
| Composite Joist : SJI 200-2015: LRFD                      |                 |
| LL Reduction : IBC  | - Floors        |
| LL Reduction : IBC  | - Roofs         |
|   | Damping         |
| Beam Vibrations : AISC DG 11 2nd Edition                  | • 0.03          |
| Joist Vibrations : SJI TD 5                               | Damping<br>0.03 |
| Joist Girder Load Tolerance: 0.2 Kips                     | s               |
| Save as Defaults  |                 |
|   |                 |
| OK Cancel App   | ly Help         |

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 <a>[2]</a>.

#### 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.

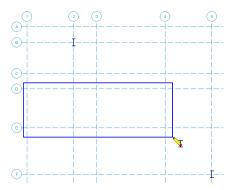
| Draw Columns                |                                    | X |
|-----------------------------|------------------------------------|---|
| Draw Columns Mo             | odify Properties   Modify Design   |   |
| Column Shape Optio          | ons                                |   |
| Hot Rolled                  | Material:                          |   |
| Cold Formed                 | A992 💌                             |   |
| ○ Wood                      | Design Rules:                      |   |
| C Concrete                  | Typical 🗨                          |   |
| General                     | Design                             |   |
|                             | Shape Group:                       |   |
|                             | WF 14 👻                            |   |
|                             | C Explicit Shape:                  |   |
|                             |                                    |   |
| Function                    | Orientation Options                |   |
| C Lateral                   | Orient to Point:                   |   |
| <ul> <li>Gravity</li> </ul> | Rotate Angle: deg                  |   |
| Column Prefix CS            | Point Prefix N                     |   |
| - Hanger Columns            | . out to an int                    |   |
| Hanger Column               | 20                                 |   |
| Bot Level                   |                                    |   |
| Top Level                   | Floor Plan 1                       |   |
| Drawing Options             |                                    |   |
| Click or Box Lo             | cations for Columns                |   |
| Offset Columns              | From the Click/Box Locations (ft): |   |
| C Axis                      | 0 and×Axis 0                       |   |
| C Angle                     | 0 and Distance 0                   | 0 |
|                             | pen when Apply is pressed?         |   |
| Apply                       | Cancel Help                        |   |

Click Apply to start drawing the columns.

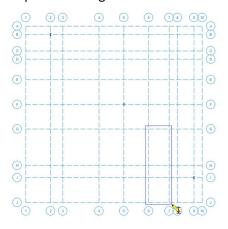
Note: When you enter the drawing mode, your cursor changes to  $\mathbf{Y}$ .

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 2 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:

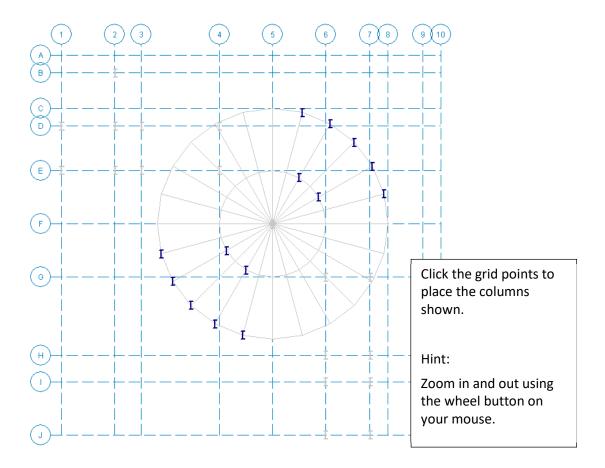
| Drawing Grids X  |
|--|
| Drawing Grid Snap To Options   Import DXF  |
| Drawing Grid Origin (ft)<br>Z Axis 119 X Axis 119<br>Click on a location to relocate the Drawing Grid              |
| ✓ Drawing Grid           ○ Rectangular Grid Increments           ∠ (it)         × (it)           30@1         30@1 |
|  |
| Skew Angle 0 deg   |
| Radial Grid Parameters<br>Start Angle 1 deg  |
| Angle Increments 24@15 deg   |
| Radial Increments 30,35 ft   |
| Save and Recall Grid Settings<br>No Saved Grids   Retrieve Save Delete   |
| Praw Construction Line     Point to Point  |
| C Point Angle Angle: 0 deg   |
| Offset from: Beam V Dist: 0 ft   |
| Show Grid As<br>C Lines Save Current Settings as Defaults?   |
| C Points Apply Cancel Help   |

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.



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).

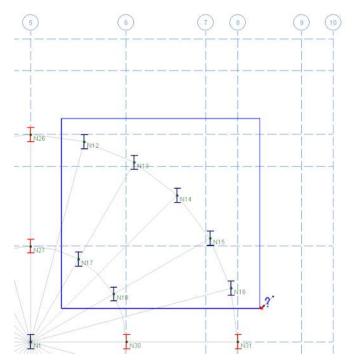
| Modify Column Properties                              |                                |
|---|--------------------------------|
| Draw Columns Modify Properties Modify Design          |                                |
| Column Shape Options                                  |                                |
| Hot Rolled  | Material: 🗌 Use?               |
| Cold Formed   | A992 👻                         |
| ◯ Wood  | Shape Group: Use?              |
| C Concrete  | WF 14                          |
| C General   |                                |
|   | Explicit Shape: Use?           |
|   |                                |
|   | Design Rules: 🔲 Use?           |
|   | Typical                        |
| -Function _ Use?                                      | Orientation Options 🔽 Use?     |
| C Lateral   | Orient to Point: N1            |
| Gravity   | Rotate Angle: deg              |
| Splices Options                                       | Use?                           |
| Splices   | Distance Below: Default (15 ft |
| Type: Default(M 💌                                     | Parapet Height 0 ft            |
| C Apply Entries to All Selected Columns               |                                |
| Apply Entries by Clicking/Boxing Columns Individually |                                |
| Keep this dialog open when Apply is pressed?          |                                |
| Apply Clear   | Use Cancel Help                |

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

**Note:** The cursor will change to  $\cancel{1}$ .

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 IN to turn off the labels.
- On the Drawing toolbar, click the Drawing Grid button III 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.

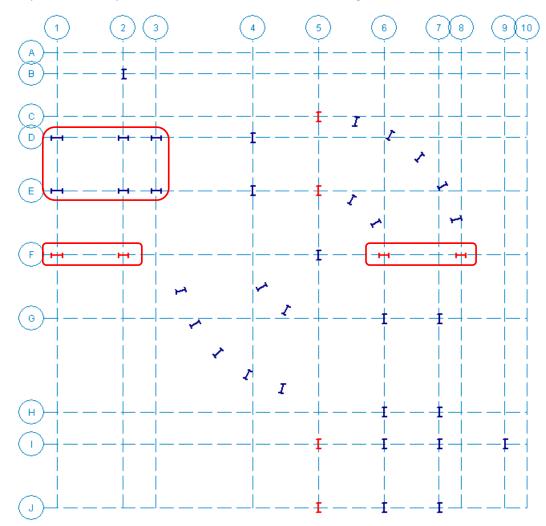
In the Orientation Options area, click the Use? check box (once it is selected, it will display in red). In the Orient to Point box, clear the point label and leave it blank. In the Rotate Angle box, enter 90.

| Modify Column Propert                   | ies X   |
|---|---|
| Draw Columns Modify                     | y Properties   Modify Design  |
| Column Shape Options                    |   |
| Hot Rolled                              | Material: 🗌 Use?  |
| Cold Formed                             | A992 🗸  |
| O Wood                                  | Shape Group: 🔲 Use?   |
| C Concrete                              | WF 14 -   |
| C General                               | Explicit Shape: Use?  |
|   |   |
|   | <u> </u>  |
|   | Design Rules: 🔽 Use?  |
|   | Typical   |
| Function Use?<br>• Lateral<br>• Gravity | Orientation Options Vuse?<br>Orient to Point:<br>Rotate Angle: 90 deg                           |
| Splices Options                         | Use?  |
| Splices                                 | Distance Below: Default (15 ft  |
| Type: Default(M -                       | Parapet Height 0 ft   |
| Keep this dialog open                   | ected Columns<br>ing/Boxing Columns Individually<br>when Apply is pressed?<br>r Use Cancel Help |
| lick 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 1.
- Under Material, select Concrete.
- Under Function, click Lateral. The wall panel settings should match those in the image below.

| Draw Wall Panels  | × |
|---|---|
| Draw Wall Panels   Modify Wall Panels   |   |
| Elevation       Bot Level       Base       Top Level       Floor Plan 1   |   |
| Material     Material Set       ○ Masonry     Material Set       ○ Concrete     Conc3000NW       ○ Wood     Design Rule       ○ General     Typical |   |
| Thickness     12     in     Function       Wall Prefix     WP <ul> <li>Lateral</li> <li>Gravity</li> </ul>  |   |
| Keep this dialog open?  |   |
| Apply Cancel Help   |   |
| 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.

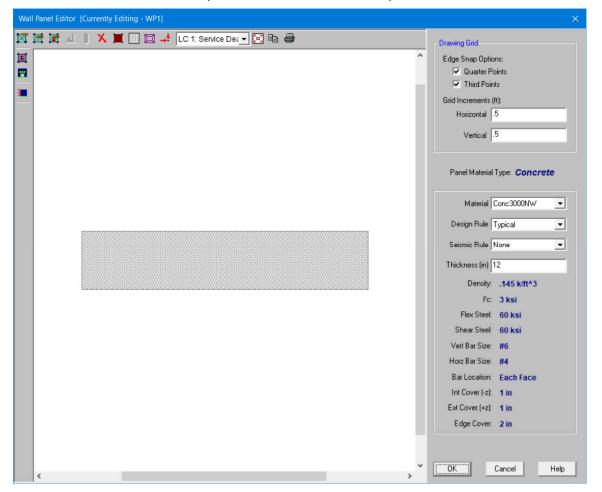
**Note:** After you draw a wall, your cursor is "anchored" to the end point of the wall. To release the drawing tool, use your right mouse button (right-click) or ESC.

- 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.



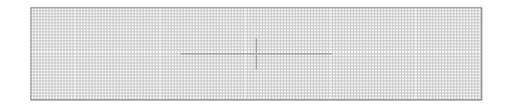
This **Editor** allows you to add openings and design regions to the wall panels. You also can edit the **Material**, **Design Rule**, and **Thickness** in this view.

#### 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.

| 😹 😹 💷 📗 🗶 📕 🛄 🛄 斗 🛛 LC 1: Service De: 🔽 🔀 🖶 🎒    | Drawing Grid                          |
|--|---------------------------------------|
|  | Edge Snap Options:                    |
|  | Quarter Points                        |
|  | Third Points                          |
|  | Grid Increments (ft):                 |
|  | Horizontal 23,3,3,20                  |
|  | Vertical 7.5                          |
|  | Panel Material Type: Concrete         |
|  | Material Conc3000NW                   |
|  | Design Rule Typical                   |
|  | Seismic Rule None                     |
|  | Thickness (in) 12                     |
|  |                                       |
|  | Density: 0.145 k/ft^3                 |
|  | fc: 3 ksi                             |
|  | Flex Steel: 60 ksi                    |
|  | Shear Steel: 60 ksi                   |
|  | Vert Bar Size: #6                     |
|  | Horz Bar Size: #4                     |
| Olish have after extension                       | Bar Location: Each Face               |
| Click here after entering<br>the Grid Increments | Int Cover (-z): 1 in                  |
| the Grid inclements                              | Ext Cover (+z): 1 in                  |
|  | Edge Cover: 2 in                      |
|  |                                       |
|  | · · · · · · · · · · · · · · · · · · · |

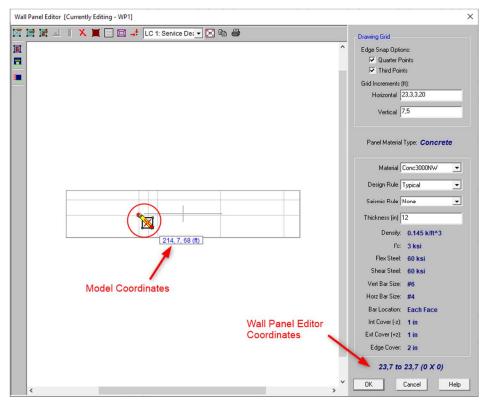
Now draw in the opening.

◆ Click on the **Create New Openings** button III (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

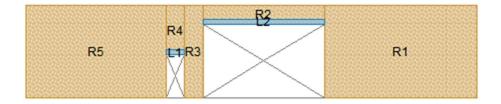
When finished, you can add in the design regions around the openings. The **Wall Panel Editor** allows you to create design regions in two ways: manually draw in each region, or use the automated generation tool. You will use the automated generation for this model.

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

Turn the rendered view back on for a better view.

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

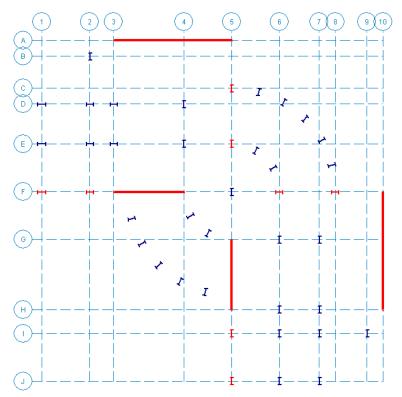
Now you can see the two openings, and five regions that were generated around the openings.



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

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:

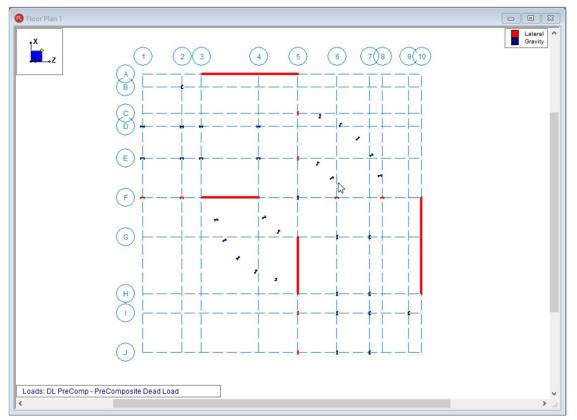
- From the Window menu, select Single View.
- On the Window toolbar, click the Graphic Editing button 2 to activate the Drawing toolbar.
- On the Data Entry toolbar, click Close E to close it. Skip ahead to the next page.

-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 <sup>C</sup>. Double-click the Tutorials folder, select Tutorial A2 Starter.rfl and click Open. Click Close <sup>C</sup> (or Cancel) to exit the Model Settings dialog box.

• On the Window toolbar, click the **Graphic Editing** button 2 to activate the Drawing toolbar.

Your screen should now look like this:



### **Primary Framing**

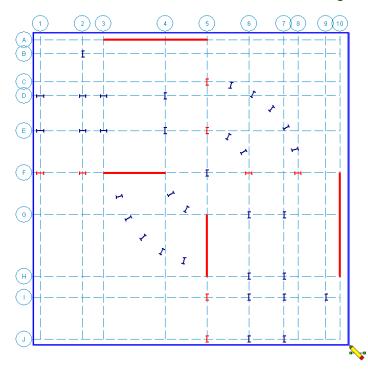
• On the Drawing toolbar, click **Draw Beams** it to open the **Draw Beams** dialog box.

| eam Material and Sha | pe Options            | Beam Label Pref        | ix M                        |
|----------------------|-----------------------|------------------------|-----------------------------|
| Hot Rolled           | Material:             | Point Label Pref       |                             |
|                      | A992 🔻                | - Function             |                             |
| Cold Formed          | Design Rules:         | C Lateral              | <ul> <li>Gravity</li> </ul> |
| O Wood               | Typical               | - Orientation for Bend | ina                         |
|                      | Design                | Strong Axis            | 🔿 Weak Axis                 |
| Concrete             | Shape Group:          | End Releases           |                             |
| C Steel Product      | Wide Flange 👻         | Start Pinned           | End Pinned                  |
| Steer Product        | C Explicit Shape:     | Start Fixed            | C End Fixed                 |
| O Wood Product       |                       | Composite              |                             |
| C General            | · · · · · ·           |                        |                             |
| rawing Options       |                       |                        |                             |
| Draw Point to Point  | t 📀 Draw Beam to Beam |                        |                             |
| O Draw Point to Bear | m 1st Offset 1 ft     | 2nd Offset             | ft                          |
| Beam Offset 1        | ft                    | C Angle 45             | deg                         |
| Keep spans contin    |                       | Angle is relati        | ve to beam                  |

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.

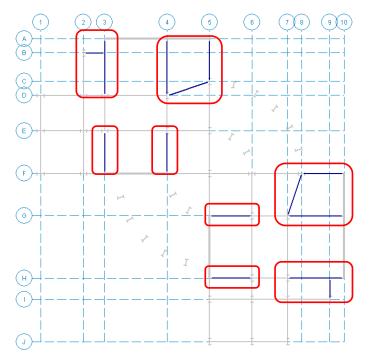
**Note:** The cursor will change to **b** to signify that you are now in drawing mode.



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

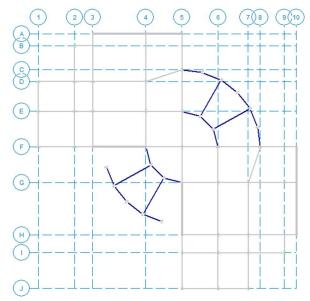
Notice that some beams were not automatically drawn in, such as those supported by wall panels or other beams. These members can be drawn individually, without going back to the dialog box.

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 2.
- 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).

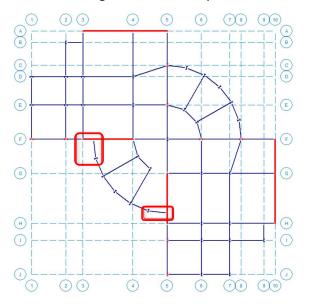
| Snap To Options   | × |
|---|---|
| Drawing Grid Snap To Options Import DXF   |   |
| Beam/Wall Snap Locations<br>✓ Quarter Points<br>✓ Third Points  |   |
| Universal Snap Increments<br>Z Axis Increment 1 ft<br>X Axis Increment 1 ft<br>Use Universal Increments |   |
| Save Current Settings as Defaults?  |   |

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).

| Draw or Modify Beams | Deck & Uni, Area Loa  | 54, 119, 0 (ft) |
|----------------------|-----------------------|-----------------|
| Draw or Modify Beams | Deck & Offi. Area Lua | 54, 119, 0 (ii) |

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.

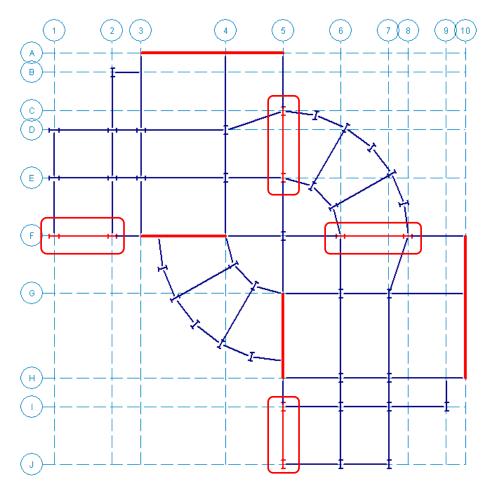
| Beam Material and Shap Hot Rolled Hot Rolled Kood Kood Kood Kood Kood Kood Kood Ko | Alterial: Use?<br>A36 Gr.36<br>Shape Group: Use?<br>Wide Flange<br>Explicit Shape: Use?<br>Design Rules: Use?<br>Typical | Function       ✓ Use?         ● Lateral       ● Gravity         Orientation for Bending       Use?         ● Strong Axis       ● Weak Axis         End Releases       Use?         ● Start Pinned       ● End Pinned         ● Start Fixed       ● End Pinned         ■ RISAConnection       Start [LEnd]       Use?         None       ▼         ■ End (J End)       Use? |
|--|--|--|
| O General  |  | None   |
| Apply Entries to All Sel     Apply Entries by Clicki     Keep this dialog open     |  | Offset Beam Use?<br>Offset Distance 0.000 in   |

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 2.
- Click the Snap To Options tab.
- Uncheck the **Use Universal Increments** checkbox, then click **Ok** to go back to the model view.

## **Infill Framing**

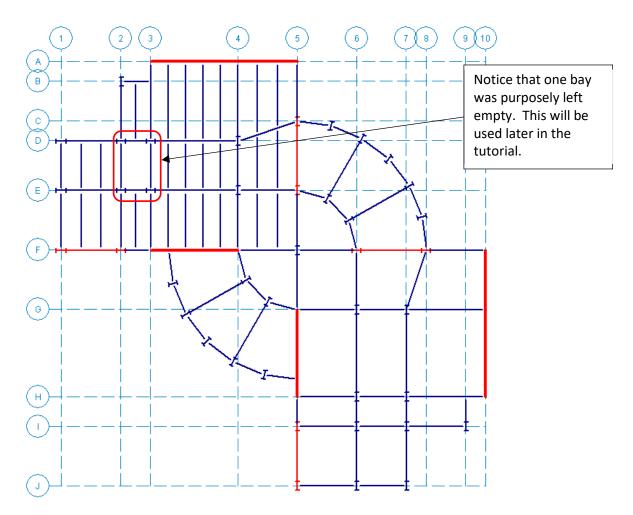
RISAFloor's Infill Framing tool assists your design process by quickly generating beams within existing bays. This tool will help you quickly create all of your secondary members:

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

| Generate Infill Beams  | ×   |
|--|---|
| Beam Material and Shape Options         Hot Rolled         Cold Formed         Wood         Vood         Shape Group:         Concrete         Wide Flange         Wood Product         General  | Beam Prefix PUR<br>Point Prefix N<br>Orientation for Bending<br>• Strong Axis<br>• Weak Axis<br>Veak Axis |
| Beam Spacing            • Equal Spacing Not to Exceed         10         ft         • Exact Spacing of         10         ft         • Number of Equal Spaces         3             Bay Selection         • Draw Polygon Around Bays to Fill         • Click or Box Bays to Fill | Beam Orientation            Global Angle  |

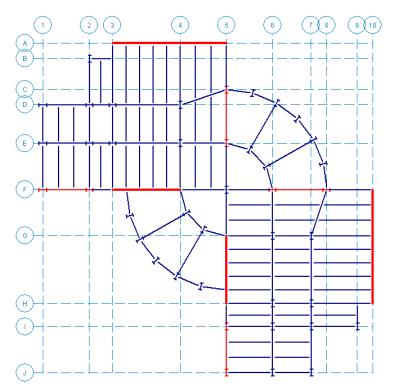
• Click Apply.

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.

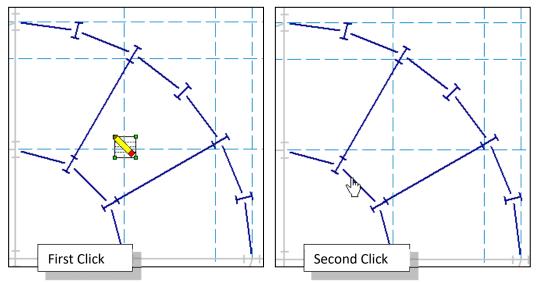
- On the Drawing toolbar, click Generate Beams I again.
- Under **Beam Orientation**, click **Horizontal**, and then click **Apply**.
- Traw a box around the area between grid lines **F5** and **J10** to select the entire area.



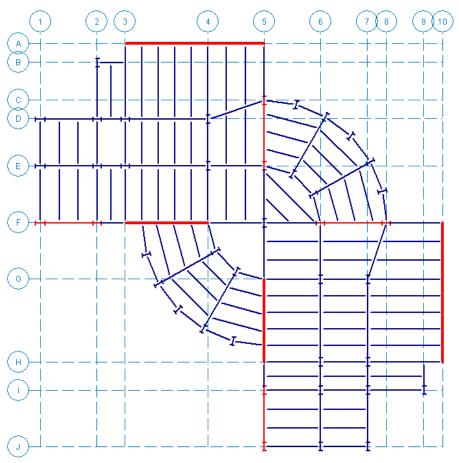
When finished, your model should look similar to the one shown below:

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.

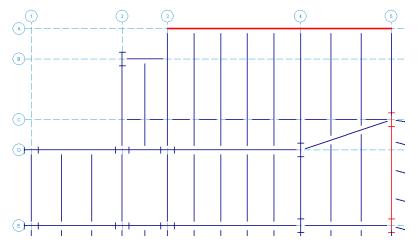
The first tool will demonstrate how to add ten members by simply clicking on two end points.

- 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.

| Draw Beams  | × |
|---|---|
| Draw Beams   Modify Properties   Modify Design   Cantilevers  |   |
| Beam Material and Shape Options         Beam Label Prefix         M           © Hot Rolled         Material:         Point Label Prefix         N   |   |
| C Cold Formed Shape Group:  |   |
| Wood     Wide Flange     Orientation for Bending     Strong Axis     Veak Axis  |   |
| C Concrete<br>Typical<br>C Steel Product<br>C Steel Product<br>C Steel Product<br>C Steel Product<br>C Start Fixed<br>C Start Fixed |   |
| C Wood Product  |   |
| Drawing Options     Oraw Point to Point     O Draw Beam to Beam   |   |
| C Draw Point to Beam 1st Offset 1 ft © 2nd Offset 1 ft<br>Beam Offset 1 ft C Angle 45 deg   |   |
| Keep spans continuous (as drawn)     C Angle is relative to beam     Angle is relative to horizontal  |   |
| Keep this dialog open Apply Cancel Help   |   |

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.

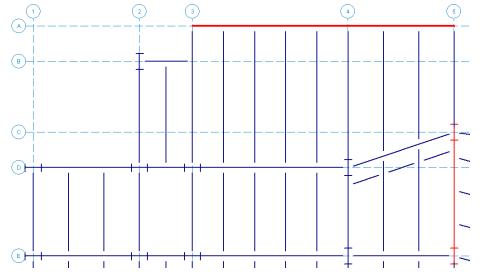
**Note**: When drawing new members, you may create either single or continuous span members. If the **Keep spans continuous** check box had been selected, RISAFloor would have created a continuous beam that spanned from **C2** to **C5**.

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 1<sup>st</sup> Offset and 2<sup>nd</sup> Offset boxes, enter 5 ft, and then click Apply.

| Draw Beams   | ×   |
|--|---|
| Draw Beams   Modify Properties   Modify Design   Ca  | ntilevers   |
| Beam Material and Shape Options         ● Hot Rolled       Material:         A992       ▼         Cold Formed       Shape Group:         Wood       Wide Flange         Concrete       Design Rules:         Typical       ▼         Steel Product       ▼ | Beam Label Prefix M<br>Point Label Prefix N<br>Function<br>C Lateral  C Gravity<br>Drientation for Bending<br>Strong Axis C Weak Axis<br>End Releases<br>Start Pinned C End Pinned<br>Start Fixed C End Fixed |
| wood Product   | Composite   |
| Drawing Options<br>Draw Point to Point<br>Draw Point to Beam<br>Beam Offset 1 ft<br>Keep spans continuous (as drawn)   |   |
| Keep this dialog open     Apply     Cancel   | Help  |

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 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.

30, 167.75, 0 (ft) (1/4 Pt)

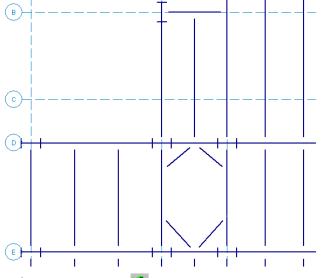
2

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.

30, 167.75, 0 (ft) (1/4 Pt)

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



Click the Undo 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.

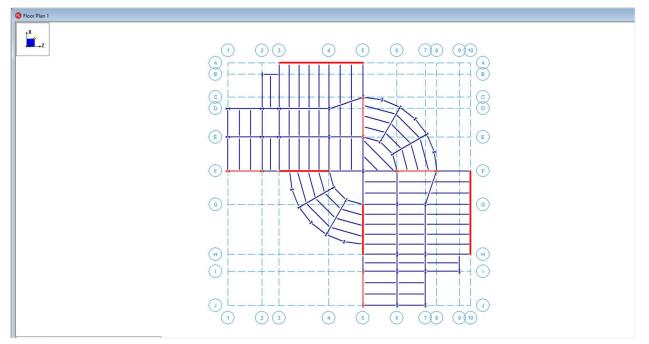
If you are continuing from the previous tutorial:

- From the Window menu, select Single View.
- On the Window toolbar, click the Graphic Editing button 2 to activate the Drawing toolbar.
- On the Data Entry toolbar, click 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 <sup>C</sup>. Double-click the Tutorials folder, select Tutorial A3 Starter.rfl and click Open. Click Close <sup>C</sup> (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 Drawing toolbar, click the Create Diaphragm Perimeters button <sup>2</sup> to open the following dialog box.

| Create Diaphragm Edge and Opening Perimeters   | × |
|--|---|
| Create Diaphragm Edge   Modify Diaphragm Edge  |   |
| Diaphragm and opening perimeters are created automatically around the perimeter  |   |
| of a group of selected beams and walls. There are various options  |   |
| available for determining exactly which group of beams and walls are   |   |
| to be considered when determining the perimeter for the diaphragm or opening.  |   |
| Edge Overhang Distance 12 in   |   |
| Diaphragm Type Rigid 💌   |   |
|  |   |
|  |   |
|  |   |
| Create a DIAPHRAGM Edge  |   |
| Create a DIAPHRAGM Edge     Consider ALL selected Beams/Walls for the DIAPHRAGM  |   |
|  |   |
| Consider ALL selected Beams/Walls for the DIAPHRAGM  |   |
| Consider ALL selected Beams/Walls for the DIAPHRAGM     Box the Beams/Walls to be considered for the DIAPHRAGM     Draw a polygon around the Beams/Walls to consider for the DIAPHRAGM   |   |
| Consider ALL selected Beams/Walls for the DIAPHRAGM     Box the Beams/Walls to be considered for the DIAPHRAGM   |   |
| Consider ALL selected Beams/Walls for the DIAPHRAGM     Box the Beams/Walls to be considered for the DIAPHRAGM     Draw a polygon around the Beams/Walls to consider for the DIAPHRAGM     Create an OPENING perimeter   |   |
| Consider ALL selected Beams/Walls for the DIAPHRAGM     Box the Beams/Walls to be considered for the DIAPHRAGM     Draw a polygon around the Beams/Walls to consider for the DIAPHRAGM     Create an OPENING perimeter     Click Within or Box the Beam/Wall perimeter for the OPENING   |   |
| Consider ALL selected Beams/Walls for the DIAPHRAGM     Box the Beams/Walls to be considered for the DIAPHRAGM     Draw a polygon around the Beams/Walls to consider for the DIAPHRAGM     Create an OPENING perimeter     Click Within or Box the Beam/Wall perimeter for the OPENING     Draw a polygon around the Beams/Walls for the OPENING |   |



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.

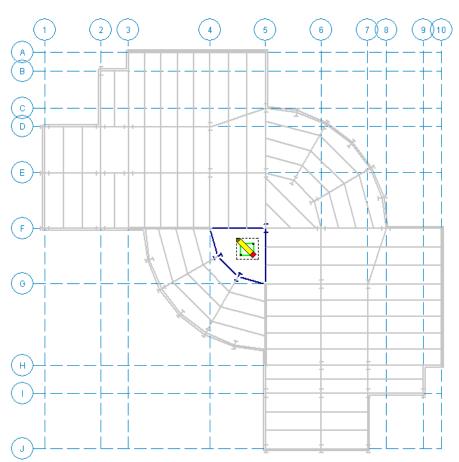
- On the Drawing toolbar, click the Create Diaphragm Perimeters button <sup>2</sup> to open the dialog box again.
- Select Create an OPENING perimeter. When creating openings, you will primarily use the first option to click within the bays for the opening. The second option may be used when there are 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.

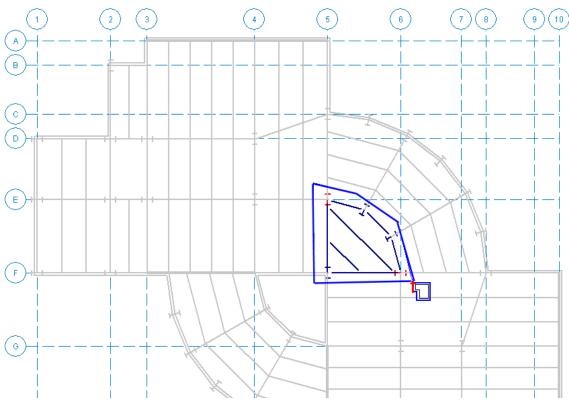
The Create Diaphragm Edge and Opening Perimeters dialog box should remain open.

**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.

| Deck   | General Properties |                       |   |              |              |
|--------|--------------------|-----------------------|---|--------------|--------------|
| Genera | I Loads            |                       |   |              |              |
|        | Label              | Material Type         | Deck  | Unbraced[ft] | Max Span[ft] |
| 1      | Concrete Deck      | Conc3000NW            | 6 in  | 3            | 10           |
| 2      | Composite Deck     | Verco PLW2-W2 Formlok | 4"NW 22ga W2 Formlok, 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.

| General | Loads          |                       |   |  |                        |
|---------|----------------|-----------------------|---|--|------------------------|
|         | Label          | Material Type         | Deck  | Unbraced[ft]   | Max Span[f             |
| 1       | Concrete Deck  | Conc3000NW            | 6 in  | 3  | 12                     |
| 2       | Composite Deck |                       |   | si 2   | 10                     |
| 3       | Metal Deck     | Verco Steel Roof Deck | Deck Selection  |  |                        |
| 4       | Wood           | Wood Deck             | General   |  |                        |
|         |                |                       | 4.57NW 19ga W2 Form<br>4.57NW 19ga W2 Form<br>4.57NW 18ga W2 Formlo<br>57NW 21ga W2 Formlo<br>57NW 21ga W2 Formlo<br>57NW 19ga W2 Formlo<br>57NW 19ga W2 Formlo<br>57NW 19ga W2 Formlo<br>57NW 19ga W2 Formlo<br>57NW 21ga W2 Formlo | (above rib) 1.5<br>d Diameter .75<br>Stud Fu 65<br>cturer Shor | in<br>in<br>ksi<br>red |

- 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 EXE.

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.

| 8 Floors            |                |            |                   |          |                           |             |               |                            |
|---------------------|----------------|------------|-------------------|----------|---------------------------|-------------|---------------|----------------------------|
| General Beam Floors | Concrete Slabs | 5          |                   |          |                           |             |               |                            |
| Label               | Elevation[ft]  | Floor Type | Area Load Default | Inactive | Splice Distance Below[ft] | Splice Type | No Wind/Drift | Parapet Height Default[ft] |
| 1 Floor Plan 1      | 15             | Floor Beam | Office            |          | 15                        | Moment      |               | 0                          |

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

| 📵 Beam | 🖲 Beam Floors |                  |                          |        |  |  |  |
|--------|---------------|------------------|--------------------------|--------|--|--|--|
| Genera | Beam Floor    | S Concrete Slabs |                          |        |  |  |  |
|        | Label         | Deck Default     | Deck Angle Default (deg) | Parent |  |  |  |
| 1      | Floor Plan 1  | Composite Deck   | 0                        | None   |  |  |  |
|        |               |                  |                          |        |  |  |  |

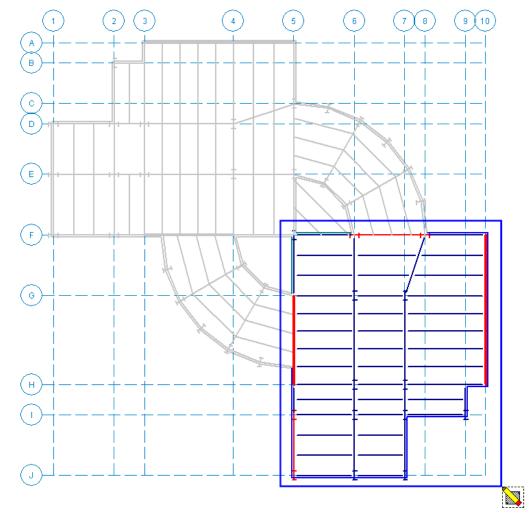
Unless you specifically assign a new area load or deck to a region of the floor, these default properties will be assumed to be applied over the whole floor (within the diaphragm edge).

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 I to exit the spreadsheet and return to the model view.
- On the Drawing toolbar, select the Assign Diaphragm Edge and Deck Properties button 1.
- In the Deck Type box, select Composite Deck.
- Under Deck Direction, click Parallel to X Axis.
- Select Box the Deck Area Perimeter.

| Deck Assignment X   |
|---|
| Assign Deck Modify Deck   |
| Pick the deck type from the list, enter a deck direction<br>and select an option for defining the deck area.  |
| Deck Type: Composite De 💌   |
| Deck Direction  |
| O Parallel to Z Axis (Plan Horizontal)  |
| <ul> <li>Parallel to X Axis (Plan Vertical)</li> </ul>  |
| C Angle From Z Axis:  |
| <ul> <li>Click Within the Deck Area Perimeter</li> <li>Box the Deck Area Perimeter</li> <li>Draw a Polygon Around the Deck Area Perimeter</li> <li>Point to Point Draw of the Deck Area</li> <li>Include Overhang</li> <li>Keep this dialog open</li> </ul> |
| Apply Close Help  |
| Click <b>Apply</b> .  |

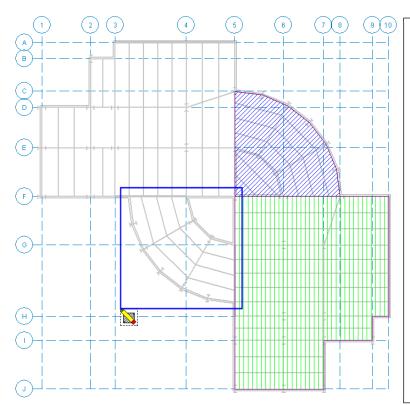
58



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.

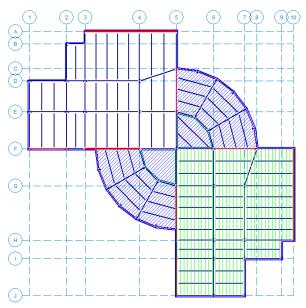
Assign this data to the two radial portions of your model by boxing them as shown in the image below.



### 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.



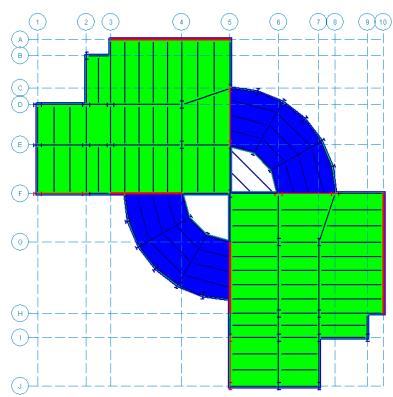
When you are finished drawing in your deck, your screen should look like the image shown below.

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.

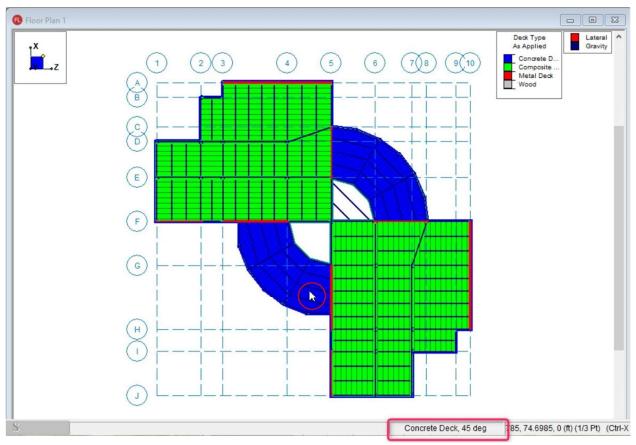
| Model Display Options                                 |                  |                      |   | ×          |  |  |
|---|------------------|----------------------|---|------------|--|--|
| Beams/Columns/Walls                                   | Points/Decks/    | Diaphragms           | Loads Misc  | Deflection |  |  |
| Show Points Show Point !                              | abels            | V                    | Show Deck Ass                                     | signments  |  |  |
| <ul> <li>Show Point i</li> <li>No Labeling</li> </ul> |                  |                      | <ul> <li>As Applied</li> <li>Deck Type</li> </ul> | 1          |  |  |
| Show Diaphragm  |                  | Show Wood Diaphragms |   |            |  |  |
| Overhang<br>Parapet H                                 |                  |                      | 🔲 Display Flegi<br>🗹 Display Type                 |            |  |  |
| Show Opening  | ntersection Poin | ts                   |   |            |  |  |
| 🗖 Display Over  | hang Distance    |                      | 1   |            |  |  |
|   | ОК               | Cancel               | Apply   | Help       |  |  |

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.

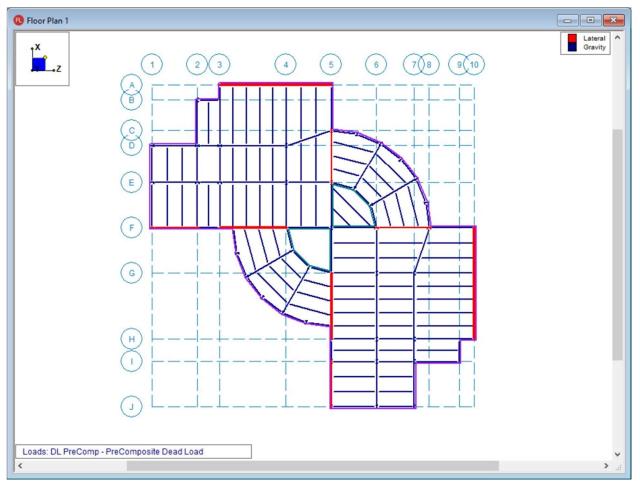
If you are continuing from the previous tutorial:

- From the Window menu, select Single View.
- On the Window toolbar, click the Graphic Editing button 2 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 <sup>C</sup>. Double-click the Tutorials folder, select Tutorial A4 Starter.rfl and click Open. Click Close <sup>C</sup> (or Cancel) to exit the Model Settings dialog box.
- On the Window toolbar, click the **Graphic Editing** button 2 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:

| 📵 Unifo | rm Area Loads |          |            |         |         |           |         | - • ×         |
|---------|---------------|----------|------------|---------|---------|-----------|---------|---------------|
| Standa  | rd Other      |          |            |         |         |           |         |               |
|         | 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    | N        |            | .02     |         | LL-Non    | .011    |               |
| 5       | Roof          |          |            | .01     | .02     | RLL-Non   | .011    | .01           |

| 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<br>in the order of how they were drawn in the model.  |  |  |  |  |
| PreDL<br>PostDL | These entries allow you to specify load magnitudes for different categories that will later be used in the load combinations that you solve.   |  |  |  |  |
| LL<br>LL Type   | <ul> <li>PreDL and PostDL are for dead loads that are pre- and post-composite.<br/>Remember that other dead loads (deck self-weight and construction<br/>dead load) were already defined within the deck definitions<br/>spreadsheet.</li> </ul> |  |  |  |  |
|                 | <ul> <li>LL and LL Type load columns are for the live load magnitude and the<br/>type of live load which you may choose from a list.</li> </ul>  |  |  |  |  |
| 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.   |  |  |  |  |
|                 | en el de la construction d'altre de la construction de la construction de la construction de la construction de  |  |  |  |  |

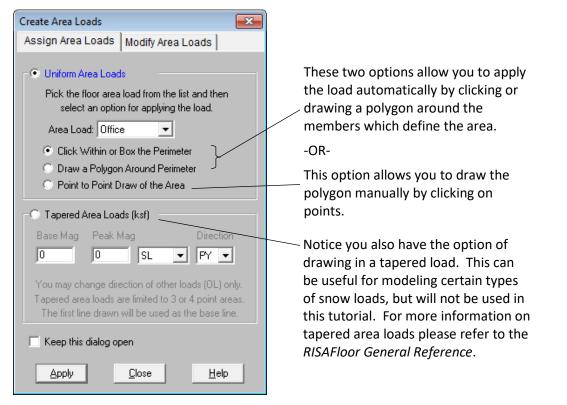
The entries in the **Area Loads** spreadsheet are described below:

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 E 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 at to display the following dialog box:



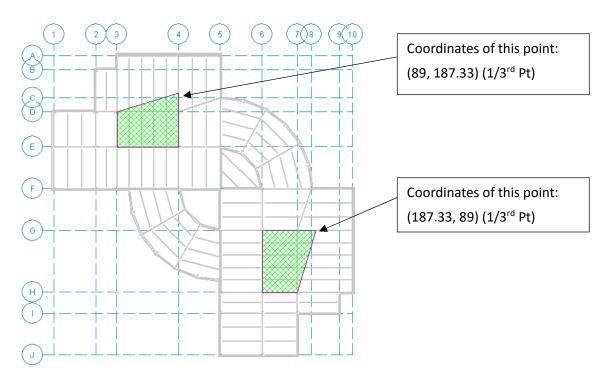
Draw the area loads:

- 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 **Note**.

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.

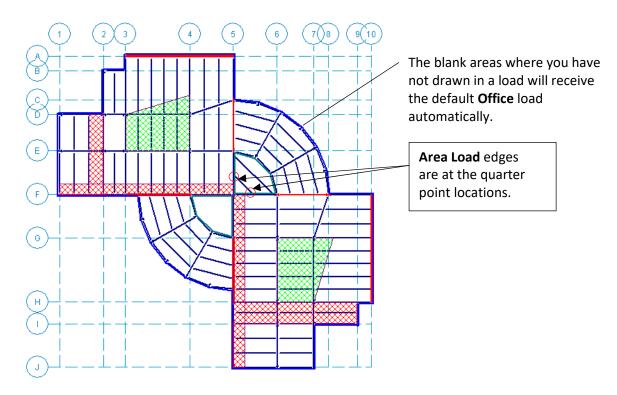
**Tip**: When you are in drawing mode, you can automatically snap to the quarter and third points of any beam (as demonstrated in the previous tutorial).



- 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.

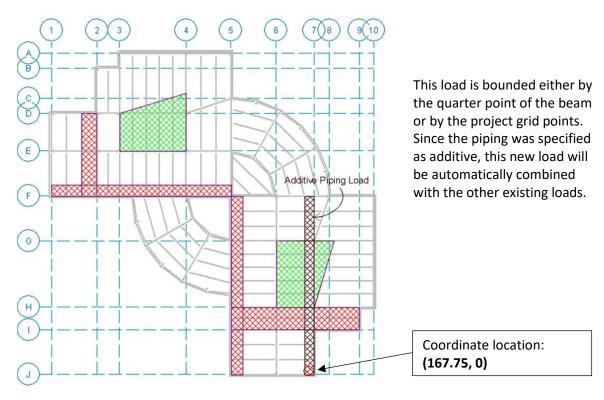
When you are finished drawing area loads, your screen should look like the image shown below.



**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.

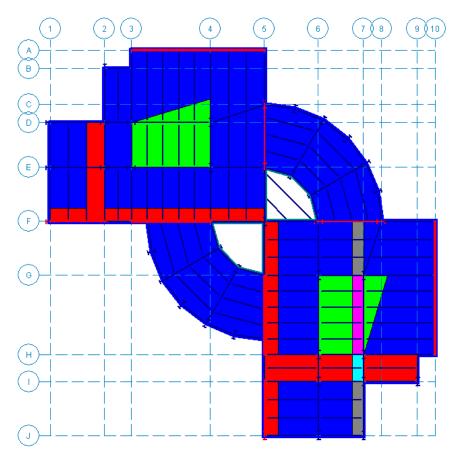


Press ESC to cancel the drawing tool and return to your regular cursor.

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 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:

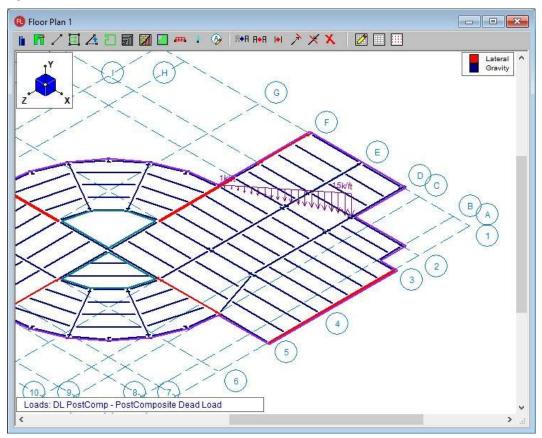
- On the Drawing toolbar, select the Line Loads button <sup>m</sup>.
- Enter the information as it appears in the following figure:

| Floor Line Loads   |   |                 | ×         |  |  |  |  |  |  |  |
|--|---|-----------------|-----------|--|--|--|--|--|--|--|
| Force Units Are  | e: k∕ft   |                 |           |  |  |  |  |  |  |  |
|  | Start Mag   | End Mag         | Direction |  |  |  |  |  |  |  |
| DL Pre   | 0   | 0               | (Y)       |  |  |  |  |  |  |  |
| DL Post  | 1   | 15              | (Y)       |  |  |  |  |  |  |  |
| LL-Non 🔻   | 0   | 0               | (PY)      |  |  |  |  |  |  |  |
| Dyn Mass   | 0   | 0               | M         |  |  |  |  |  |  |  |
| 0L1 💌  | 0   | 0               | Υ·        |  |  |  |  |  |  |  |
| <ul> <li>Apply to A</li> <li>Apply Loa</li> <li>Apply Loa</li> <li>Start I</li> <li>End I</li> </ul> | What happens when Apply is pressed?         Apply to All Selected Members/Walls         Apply Load by Drawing Point to Point         Apply Load by Clicking Members/Walls         Start Location Offset         End Location Offset         Keep this dialog open |                 |           |  |  |  |  |  |  |  |
|  | <u>C</u> lose   | <u><u> </u></u> | elp       |  |  |  |  |  |  |  |
| Click <b>Apply</b> .   |   |                 |           |  |  |  |  |  |  |  |

| - <u></u>                              | <u>n</u> , |
|--|------------|
| Note: The cursor will change to $\Phi$ |            |

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 lso.
- 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 <sup>Pln</sup>.
- On the Window toolbar, click **Redraw** it o 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.

| Select Items for Current View   | <b>—</b>   |
|---|--|
| Points Columns Beams Walls  | 1  |
| Beam Selection Criteria<br>Beam Range<br>From :<br>To :<br>Perimeter Beams Only<br>Elevated Beams | Beam Properties<br>Material Type Don't Use<br>Shape Don't Use<br>Function Don't Use<br>Orientation Don't Use |
| Selection Options<br>Select Beams<br>UnSelect Beams   | Clear Selection Criteria   |

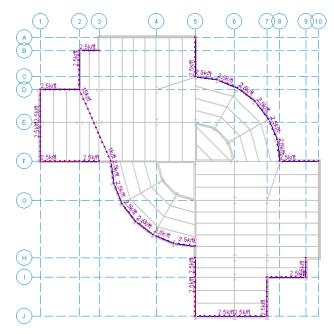
Click OK to apply the selection.

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.

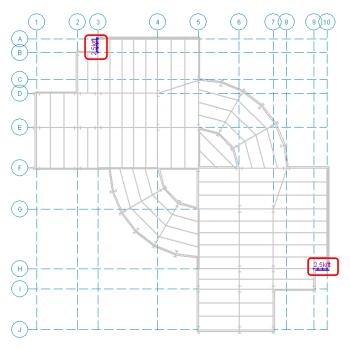
- On the Drawing toolbar, select the Line Load button again to open the Floor Line Loads dialog box.
- For the **DL Post load**, enter a starting and ending magnitude of **2.5 kip/ft**.
- Under What happens when Apply is pressed?, click Apply to All Selected Members/Walls.
- Click Apply.

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 H to select the model.
- On the Drawing toolbar, click Assign point loads
- Enter the information show below:

| Floor Point Loads  |  | <b>-X</b> -       |
|--|--|-------------------|
|  | Magnitude (k)  | Direction         |
| DL Pre   | 0  | M                 |
| DL Post  | 0.5  | M                 |
| LL-Non 💌   | 0.075  | (PY)              |
| Dyn Mass   | 0  | (Y)               |
| OL1 🔻  | 0  | Y 💌               |
| <ul> <li>Apply Load I</li> <li>Apply Load I</li> <li>Offset From I</li> <li>Keep this dia</li> </ul> | o All Selected Po<br>by Clicking/Boxin<br>by Clicking Memb<br>Member End | oints<br>g Points |
|  | <u>C</u> lose  | <u>H</u> elp      |

#### • 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.

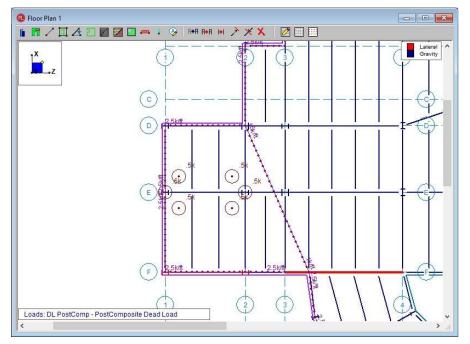
Draw in the point loads:

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.

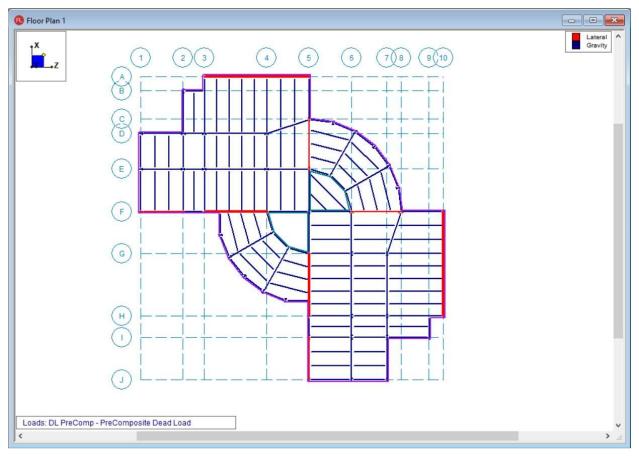
If you are continuing from the previous tutorial:

- From the Window menu, select Single View.
- On the Window toolbar, click the Graphic Editing button 2 to activate the Drawing toolbar.
- On the Data Entry toolbar, click Close E 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.

| Com    | binations            |       |       |        |       |        |       |        |         |        |       | • •   |
|--------|----------------------|-------|-------|--------|-------|--------|-------|--------|---------|--------|-------|-------|
| Combii | nations Design       |       |       |        |       |        |       |        |         |        |       |       |
|        | Label                | Solve | Categ | Factor | Categ | Factor | Categ | Factor | Categ   | Factor | Categ | Facto |
| 1      | Service Dead         |       | DL    | 1      |       |        |       |        |         |        |       |       |
| 2      | Service Dead + Live  |       | DL    | 1      | LL    | 1      | LLS   | 1      |         |        |       |       |
| 3      | Strength Dead        |       | DL    | 1.4    |       |        |       |        | а.<br>Л |        |       |       |
| 4      | Strength Dead + Live |       | DL    | 1.2    | LL    | 1.6    | LLS   | 1.6    |         |        |       |       |

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 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.

| Load Combination Gener  | ator 💌               |
|-------------------------|----------------------|
| LC Region               | United States        |
| LC Code                 | 2015 IBC ASD         |
| -Roof Live Load Options | ;                    |
| Include :               | RLL (Roof Live Load) |
|                         | SL (Snow Load)       |
|                         | 🗌 RL (Rain Load)     |
| Composite Options       |                      |
| Include :               | ✓ Pre Composite      |
|                         | Post Composite       |
| Save as Defaults        |                      |
| Generate                | Cancel Help          |

Click Generate.

| 📵 Comi | binations          |       |       |        |         |        |       | - 474  |          |        |       | ÷.     |       |        | ۲. |
|--------|--------------------|-------|-------|--------|---------|--------|-------|--------|----------|--------|-------|--------|-------|--------|----|
| Combin | nations Design     |       |       |        |         |        |       |        |          |        |       |        |       |        |    |
|        | Label              | Solve | Categ | Factor | Categ   | Factor | Categ | Factor | Categ    | Factor | Categ | Factor | Categ | Factor |    |
| 1      | IBC 16-8 Pre       | V     | DLPre | 1      |         |        |       |        |          |        |       |        |       |        | Ŀ  |
| 2      | IBC 16-9 Pre       | V     | DLPre | 1      | LLConst | 1      |       |        |          |        |       | 1      |       |        | 1  |
| 3      | IBC 16-8 Post      | V     | DL    | 1      |         |        |       |        | 21<br>16 |        |       |        |       |        | 1  |
| 4      | IBC 16-9 Post      |       | DL    | 1      | LL      | 1      | LLS   | 1      |          |        |       |        |       |        | 1- |
| 5      | IBC 16-10 (a) Post | ٤     | DL    | 1      | RLL     | 1      |       |        |          |        |       |        |       |        | ]  |
| 6      | IBC 16-11 (a) Post |       | DL    | 1      | LL      | .75    | LLS   | .75    | RLL      | .75    |       |        |       |        | 1  |
| 7      | IBC 16-11 (b) Post | V     | DL    | 1      | LL      | .75    | LLS   | .75    | -        |        |       |        |       | -      | ], |

The following load combinations will be generated by RISAFloor:

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
- In the LC Code list, select 2015 IBC Strength.

| Load Combination Gener                         | ator ×   |
|--|--|
| LC Region<br>LC Code<br>Roof Live Load Options | United States<br>2015 IBC Strength   |
| Include :                                      | <ul> <li>RLL (Roof Live Load)</li> <li>SL (Snow Load)</li> <li>RL (Rain Load)</li> </ul> |
| Composite Options                              |  |
| Include :                                      | Pre Composite     Post Composite   |
| Save as Defaults                               | ]  |
| Generate                                       | Cancel Help  |

Click Generate.

| Combi   | nations Design     |       |       |        |         |        |       |        |       |        |       |        |       |        |
|---------|--------------------|-------|-------|--------|---------|--------|-------|--------|-------|--------|-------|--------|-------|--------|
| 2011101 | Design             |       |       |        |         |        |       |        |       |        |       |        |       |        |
|         | Label              | Solve | Categ | Factor | Categ   | Factor | Categ | Factor | Categ | Factor | Categ | Factor | Categ | Factor |
| 1       | IBC 16-8 Pre       | V     | DLPre | 1      |         |        |       |        |       |        |       |        |       |        |
| 2       | IBC 16-9 Pre       |       | DLPre | 1      | LLConst | 1      |       |        |       |        |       |        |       |        |
| 3       | IBC 16-8 Post      | V     | DL    | 1      | 1       |        |       |        |       |        |       |        |       |        |
| 4       | IBC 16-9 Post      |       | DL    | 1      | LL      | 1      | LLS   | 1      |       |        |       |        |       |        |
| 5       | IBC 16-10 (a) Post | V     | DL    | 1      | RLL     | 1      |       |        |       |        |       |        |       |        |
| 6       | IBC 16-11 (a) Post |       | DL    | 1      | LL      | .75    | LLS   | .75    | RLL   | .75    |       |        |       |        |
| 7       | IBC 16-11 (b) Post | V     | DL    | 1      | LL      | .75    | LLS   | .75    |       |        |       |        |       | -      |
| 8       | IBC 16-1 Pre       |       | DLPre | 1.4    |         |        |       |        | 1     |        |       |        |       |        |
| 9       | IBC 16-2 Pre       | V     | DLPre | 1.2    | LLConst | 1.6    |       |        |       |        |       |        |       |        |
| 10      | IBC 16-1 Post      |       | DL    | 1.4    |         |        |       |        |       |        |       |        |       |        |
| 11      | IBC 16-2 (a) Post  | V     | DL    | 1.2    | LL      | 1.6    | LLS   | 1.6    | RLL   | .5     |       |        |       |        |
| 12      | IBC 16-2 (b) Post  | V     | DL    | 1.2    | LL      | 1.6    | LLS   | 1.6    |       |        |       |        |       |        |
| 13      | IBC 16-3 (a) Post  | V     | DL    | 1.2    | RLL     | 1.6    | LL    | .5     | LLS   | 1      |       | -      |       | -      |
| 14      | IBC 16-3 (c) Post  |       | DL    | 1.2    | LL      | .5     | LLS   | 1      | 2     |        | ·     | -      |       | 6      |

This finalizes your load combinations as shown below:

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.

| iompir | nations Design    |      |      |         |            |           |      |          |            |         |            |
|--------|-------------------|------|------|---------|------------|-----------|------|----------|------------|---------|------------|
|        | Label             | ASIF | CD   | Service | Hot Rolled | Cold Form | Wood | Wood Pro | Concrete   | Masonry | Connection |
| 1      | IBC 16-8 Pre      |      | .9   | V       |            |           | V    |          |            | V       |            |
| 2      | IBC 16-9 Pre      |      |      |         |            |           |      |          |            | V       |            |
| 3      | IBC 16-8 Post     |      | .9   | V       |            |           | V    |          |            |         |            |
| 4      | IBC 16-9 Post     |      |      |         |            |           |      |          |            |         |            |
| 5      | IBC 16-10 (a) Pos |      | 1.25 | V       | V          |           | V    |          | V          |         |            |
| 6      | IBC 16-11 (a) Pos |      | 1.25 |         |            |           |      |          |            |         |            |
| 7      | IBC 16-11 (b) Pos |      | 1.15 | V       | V          |           | V    | V        | V          |         |            |
| 8      | IBC 16-1 Pre      |      |      |         |            |           |      |          | - <u>-</u> |         |            |
| 9      | IBC 16-2 Pre      |      |      |         |            |           |      |          |            |         |            |
| 10     | IBC 16-1 Post     |      |      |         |            |           |      |          |            |         |            |
| 11     | IBC 16-2 (a) Post |      |      |         |            |           |      |          | V          |         |            |
| 12     | IBC 16-2 (b) Post |      |      |         |            |           |      |          |            |         |            |
| 13     | IBC 16-3 (a) Post |      |      |         |            |           |      |          | V          |         |            |
| 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 9<sup>th</sup> 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).

|       | national Decign   |      |      |         |            |           |      |          |          |         |           |
|-------|-------------------|------|------|---------|------------|-----------|------|----------|----------|---------|-----------|
| compi | nations Design    |      |      |         |            |           |      |          |          |         |           |
|       | Label             | ASIF | CD   | Service | Hot Rolled | Cold Form | Wood | Wood Pro | Concrete | Masonry | Connectio |
| 1     | IBC 16-8 Pre      |      | .9   |         | V          |           | V    |          |          | V       |           |
| 2     | IBC 16-9 Pre      |      |      |         |            |           |      |          |          | N       |           |
| 3     | IBC 16-8 Post     |      | .9   | V       | V          |           | V    | V        |          | V       |           |
| 4     | IBC 16-9 Post     |      |      |         |            |           |      |          |          | N       |           |
| 5     | IBC 16-10 (a) Pos |      | 1.25 | V       | V          |           | 2    | V        |          | V       | •         |
| 6     | IBC 16-11 (a) Pos |      | 1.25 |         |            |           |      | V        |          | N       |           |
| 7     | IBC 16-11 (b) Pos |      | 1.15 |         | V          |           | V    | V        |          | V       |           |
| 8     | IBC 16-1 Pre      |      |      |         |            |           |      |          |          | V       |           |
| 9     | IBC 16-2 Pre      |      |      |         |            |           |      |          | V        | V       |           |
| 10    | IBC 16-1 Post     |      |      |         |            |           |      |          |          | V       |           |
| 11    | IBC 16-2 (a) Post |      |      |         |            |           |      |          | V        | V       |           |
| 12    | IBC 16-2 (b) Post |      |      |         |            |           |      |          |          | V       |           |
| 13    | IBC 16-3 (a) Post |      |      |         |            |           |      |          |          | V       |           |
| 14    | IBC 16-3 (c) Post |      |      |         |            |           |      |          |          | V       |           |

#### When finished, your spreadsheet should look like this:

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:

|       |                    | es not match Final S<br>ing Physical Columns |             |
|-------|--------------------|--|-------------|
| 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      |
| Total | Physical Columns : | 41   |             |

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 | 2    | Floor 1 : Floor Plan 1 - Deck - Concrete Deck : No composite design because deck selected is non-composite deck.                  |
| 3 | 2    | Floor 1 : Floor Plan 1 - Beam - M38 : Composite design not done : Rib Pitch is greater than Max Stud Spacing along Beam Direction |
| 4 | 2    | Floor 1 : Floor Plan 1 - Beam - M47 : Composite design not done : Rib Pitch is greater than Max Stud Spacing along Beam Direction |
| 5 | 2    | Floor 1 : Floor Plan 1 - Beam - PUR140 : Composite design not done : Diaphragm edge Perimeter is not consistent above Beam.       |
| 5 | ?    | 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.

| Beam     | Design for Ho | t Rolled |            |            |          |           |             |           |               |             |
|----------|---------------|----------|------------|------------|----------|-----------|-------------|-----------|---------------|-------------|
| lot Roll | ed Cold For   | med Wood | Steel Prod | ucts   Woo | d Produc | ts Concre | ete         |           |               |             |
| Γ        | 1: Floor Plan | 1        | •          |            |          |           |             |           |               |             |
|          | 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.

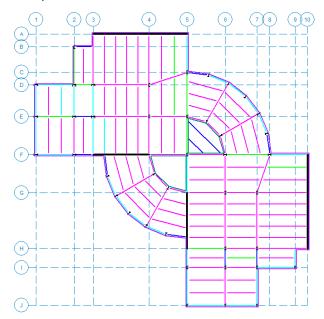
The **Code Check** spreadsheet gives you a quick overview of the checks for bending, deflection, and shear. There are also individual results spreadsheets that give detailed information for the **Bending**, **Deflection**, **Shear**, and **Vibration** checks. The **Reactions** spreadsheet gives the member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

| Set Options for Current Vie   | w   | <b>-X</b> -  |
|---|---|--|
| Beams/Columns/Walls Po  | ints/Decks/Diaphragms   L   | oads Misc Deflection   |
| <ul> <li>Line Draw:</li></ul>   | Pulba     Colum   | t of Length 80% 🔽<br>sk. Distance 1 ît<br>ns/Walls Full Length<br>ney: 👢 0%  |
| <ul> <li>Show Beams</li> <li>Labeling:</li> <li>No Labeling</li> <li>Show Pins</li> <li>Show Pins</li> <li>Pullback Ends</li> <li>Col Eccentricity</li> <li>Show Reactions</li> </ul> | <ul> <li>Show Columns</li> <li>Labeling:</li> <li>No Labeling</li> <li>Show Enlarged<br/>for Line Draw</li> <li>Show Splices</li> </ul> | <ul> <li>Show Walls</li> <li>Labeling:</li> <li>No Labeling</li> <li>Render 80%</li> <li>Transparency 20%</li> </ul> |
|   | OK Cancel   | Apply Help   |

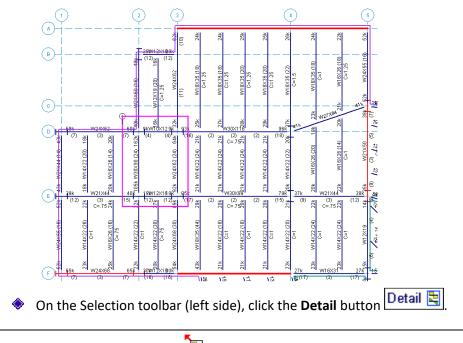
Click Apply.



Your plot should now look like this:

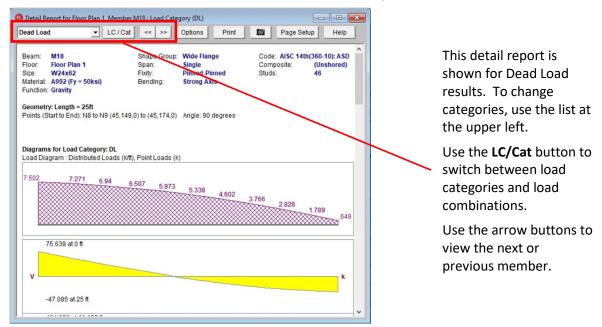
This image gives a quick graphical representation of the adequacy of each beam for the deflection limits specified in the **Design Rules** spreadsheet. You can create a similar plot to display the Bending or Shear Checks. You can also plot the Vibration Frequencies and Accelerations to check your entire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



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.



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 E 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 Design, 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.

| Member Redesign :                              | Floor | 1, Member PUR77   |
|--|-------|---|
| Available Shapes                               |       | - Current Besults   |
| W18x86<br>W18x76<br>W18x71<br>W18x65<br>W18x60 | ^     | Bending Check Shear Check Defl. Ratio Defl. Max<br>.982 .387 271 1.33<br>Results Messages |
| W18x55<br>W18x50<br>W18x46<br>W18x40<br>W18x35 |       | Design Criteria   |
| W16x100<br>W16x89<br>W16x77<br>W16x67          |       | Material Redesign Rules<br>A992  Typical  |
| W16x57<br>W16x50<br>W16x45<br>W16x40           | Ш     | Composite Member?     Uniform Studs 26  |
| W16x36<br>W16x31<br>Optimize                   | Ŧ     | Segment Study   |
|  | (     | DK Cancel Help  |

Click Cancel to close the dialog box.

You can now make a framing change to your model.

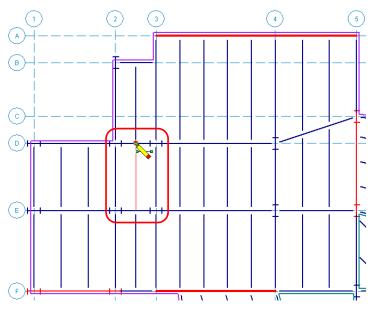
- On the Window toolbar, click the Graphic Editing button 2 to activate the Drawing toolbar if it is not already visible.
- On the Drawing toolbar, click Draw Beams

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.

| Clear Res | ults Warning   | × |
|-----------|--|---|
|           | This operation will cause the current results to be cleared!<br>Do you wish to continue? |   |
|           | Yes No   |   |
|           | Always display this warning before clearing 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:

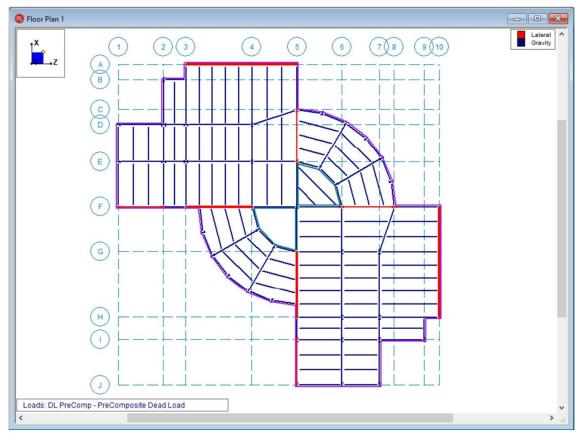
- From the Window menu, select Single View.
- On the Window toolbar, click the Graphic Editing button 2 to activate the Drawing toolbar.
- On the **Data Entry** toolbar, click **Close I** to close it. Skip ahead to below.

-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.

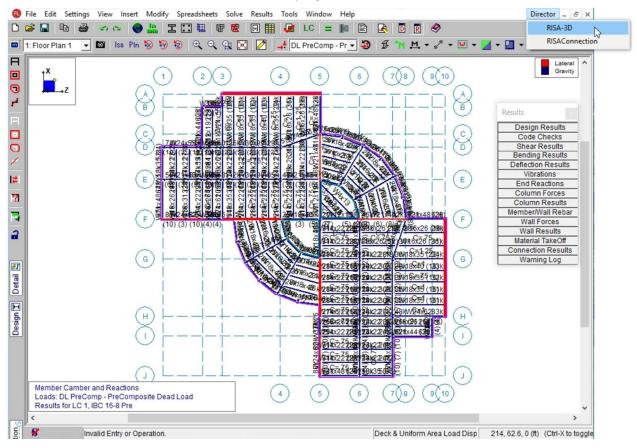
• On the Window toolbar, click the **Graphic Editing** button 2 to activate the Drawing toolbar.

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:

| Wind Code ASCE<br>Wind Speed (mph) 115 | 7-10 🔻          | 5             |           |                       |              |                       |          |
|--|-----------------|---------------|-----------|-----------------------|--------------|-----------------------|----------|
| Wind Speed (mph) 115                   |                 | Exposure Ca   | t. B 👻    | Topographic Fac. K1   | 0            | Topographic Fac. I    | K3 0     |
| · · · · · · · · · · · · · · · · · · ·  |                 |               |           | Topographic Fac. K2   | 20           | Directionality Fac. I | Kd .85   |
| Base Elevation                         |                 | Gust Stiffnes | S Rigid - | Fundamental Freq      |              | Gust Factor (         |          |
| Dase Lievalion                         |                 | Gustounies    |           | 1 undamentari req     |              | 00011 0001 (          | 0/[.05   |
| Generation Generation                  | ate Roof Wind L | oads          |           |                       |              |                       |          |
|  |                 |               |           |                       |              |                       |          |
|  |                 |               |           |                       |              |                       | (        |
| Wind Load Results                      |                 |               |           |                       |              | Ca                    | Ic Loads |
|  |                 |               |           |                       |              |                       |          |
| Wind Generation Input                  |                 |               |           |                       |              |                       |          |
| Wind Code:                             | ASCE 7-10       |               |           | Topographic Factor    |              |                       |          |
| Wind Speed, V(mph):                    | 115             |               |           | Topographic Factor    |              |                       |          |
|  | -               |               |           | Topographic Factor    |              |                       |          |
| Exposure Category:                     | в               |               |           | Directionality Factor |              |                       |          |
| Base Elevation(ft):                    | 0               |               |           | Parapet Height(ft):   | 0            |                       |          |
| Wind Generation Detail                 | Results         |               |           |                       |              |                       |          |
|  |                 |               |           | Kzt:                  | 1            |                       |          |
| Exposure Constant Alpha                | a: 7            |               |           | h (ft):               | 15           |                       |          |
| Exposure Constant zg:                  | 1200            |               |           | Kh:                   | .575         |                       |          |
| Gust Effect Factor, G:                 | .85             |               |           | Windward Cp:          | .8           |                       |          |
|  |                 |               |           | gh (ksf):             | .017         |                       |          |
|  |                 |               |           | GCpn (windward):      | +1.5         |                       |          |
|  |                 |               |           | GCpn (leeward):       | -1.0         |                       |          |
| Wind Generation Floor G                | eometry Result  | s             |           |                       |              |                       |          |
|  | Height          | Kz            | Width (X) | Length (Z)            | Leeward Cp(X | ) Leeward Cp(Z)       |          |
|  | (ft)            |               | (ft)      | (ft)                  |              | ,p(L)                 |          |
| Floor Plan 1                           | 15              | .575          | 216       | 216                   | .5           | .5                    |          |
|  |                 |               |           | 2.00                  |              |                       |          |
|  |                 |               |           |                       |              |                       |          |

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 par       | ameters mat                  | tch those in tl | ne image b | elow:           |                       |            |
|-------|---------------------------|---------------------|------------------------------|-----------------|------------|-----------------|-----------------------|------------|
| Seisr | nic Loads                 |                     |                              |                 |            |                 |                       |            |
| – Se  | ismic Load Pa             | rameters            |                              |                 |            |                 |                       |            |
| 9     | Seismic Code              | ASCE 7-10 🔻         | Ct (V,X) .02                 |                 | T (V,X)    | sec             | R (V,X) 3             |            |
|       |                           |                     | · · · · · ·                  |                 | 1 (1,11)   |                 | 1 (0,00)              |            |
| B     | ase Elevation             | ft                  | Ct (H,Z) .02                 |                 | T (H,Z)    | sec             | R (H,Z) 3             |            |
|       | Risk Cat                  | l or II 👻           | TL 5                         | sec             | Add        | d Base Weight ( | Ct Exp. (V,X) .7      | /5         |
|       | S_D1                      | 1 9                 | S_DS 1                       | g               | S_1 1      | g (             | ,<br>Ct Exp. (H,Z) .7 | ′5         |
| Se    | eismic Load Re            | sults               |                              |                 |            |                 |                       | Calc Loads |
| Se    | eismic Generat            | tion Input          |                              |                 |            |                 |                       |            |
|       | eismic Code:              | ASCE 7-10           |                              |                 |            |                 |                       |            |
|       | _X:                       | .02                 | T_X (sec):                   | Not Entered     | R_X:       | 3               |                       |            |
|       | _Z:                       | .02                 | T_Z (sec):                   | Not Entered     | R_Z:       | 3               |                       |            |
|       | t Exp. X:                 | .75                 | Ct Exp. Z:                   | .75             |            |                 |                       |            |
|       | isk Cat                   | lorll               | TL (sec):                    | 5<br>1          | 04 (-);    | 1               |                       |            |
|       | D1 (g):<br>ase Elev (ft): | 1<br>0              | SDS (g):<br>Parapet Ht (ft): | 1               | S1 (g):    | 1               |                       |            |
|       | ase Elev (II).            | 0                   | Falapet Fit (it).            | U               |            |                 |                       |            |
| Se    | eismic Generat            | tion Detail Results |                              |                 |            |                 |                       |            |
| Т     | X Used (sec):             | .1524               | T X Method A:                | .1524           | T X Uppe   | er Limit: .2134 |                       |            |
| T     | Z Used (sec):             | .1524               | T Z Method A:                | .1524           | T Z Uppe   | er Limit: .2134 |                       |            |
| Im    | portance Fac.:            | 1                   | Design Cat.:                 | E               |            |                 |                       |            |
| V     | X (k):                    | 1312.9573           | Gov. Eqn.                    | ASCE Eqn 12.8-2 | Cs_X:      | 0.333           |                       |            |
| V_    | _Z (k):                   | 1312.9573           | Gov. Eqn.                    | ASCE Eqn 12.8-2 | Cs_Z:      | 0.333           |                       |            |
| Se    | eismic Generat            | tion Force Results  |                              |                 |            |                 |                       |            |
|       | Floor Level               | Height              | Weight                       | Force X         | Force Z    | CGX             | CG Z                  |            |
|       | 51 DI 4                   | (ft)                | (k)                          | (k)             | (k)        | (ft)            | (ft)                  |            |
|       |                           |                     |                              |                 | ок         | Print [         | Cancel                | Help       |

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.

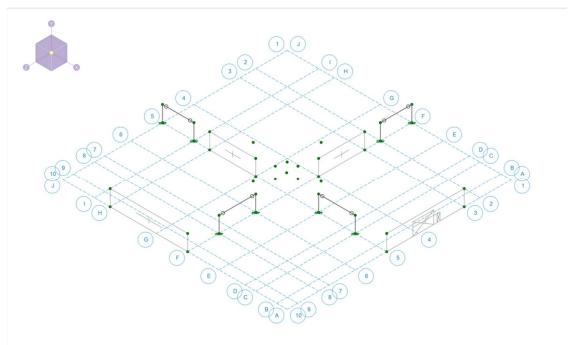
- Click **OK** to accept the loads.
- If the **Model Settings** dialog box displays, close it by clicking **OK** or **Cancel**.

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 Not 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.

| asic Loa | ad 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 is 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 D | iaphragms      |          |                               |                     |          |       |          |       |          |           |       |        |             |      |          |                | 1             |
|---------|----------------|----------|-------------------------------|---------------------|----------|-------|----------|-------|----------|-----------|-------|--------|-------------|------|----------|----------------|---------------|
| Genera  | Wood Diaphra   | igms     |                               |                     |          |       |          |       |          |           |       |        |             |      |          |                |               |
|         | Elevation [ft] | Mass [k] | Mass MOI [k·ft <sup>2</sup> ] | Center Of Mass [ft] | Plus X E | Minus | Plus Z E | Minus | Inactive | Diaphragm | Туре  | 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            |               |
| -       |                |          |                               |                     |          |       |          |       |          |           |       |        |             |      |          |                |               |

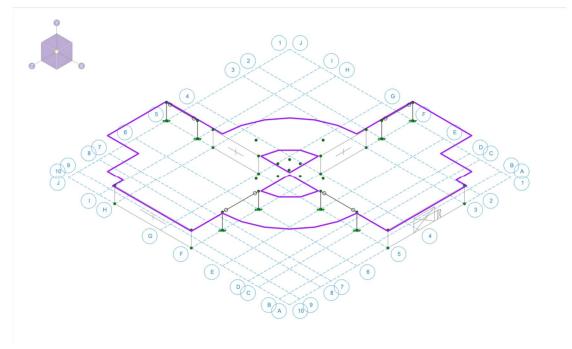
Notice that the diaphragms already contain the diaphragm mass and mass moment of inertia information that you would need for a dynamic analysis. In this spreadsheet, you can specify the percent of width you want RISA-3D to use to account for the accidental eccentricity in the X or Z directions.

Click Close k 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.



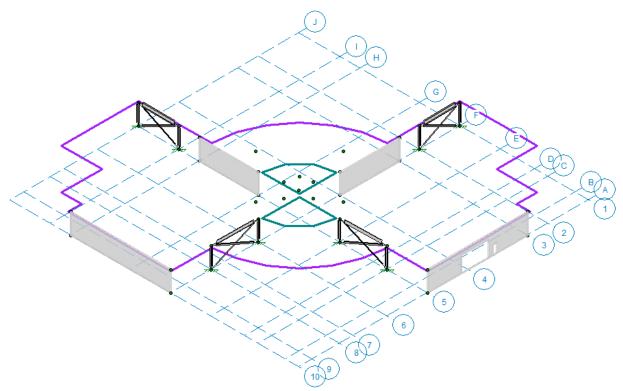
Next, add some X bracing, as follows:

On the Home menu, click the Draw Members button Members. 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.

| Properties           |                    |
|----------------------|--------------------|
| Member               |                    |
| Label Prefix         | м                  |
| Draw Option          | Node to Node 🔹     |
| General Properties   |                    |
| Member Type          | VBrace •           |
| Material Type        | Hot Rolled Steel 🔻 |
| I Release            | BenPIN             |
| J Release            | BenPIN             |
| Section/Shape        | Shape Database 🔻   |
| Shape                | HSS4X4X4           |
| Material             | A500 Gr.B 🔻        |
| Design List          | SquareTube 💌       |
| Design Rule          | Typical 🔹          |
| Additional Propertie | 25                 |
| RISAConnection Pro   | perties            |
| Design Properties    |                    |

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.

| 3D Lo   | ad Combi   | nation Ge | nerator                 | ?     | ? × |  |  |
|---------|------------|-----------|-------------------------|-------|-----|--|--|
| Gravity | Wind       | Seismic   |                         |       |     |  |  |
| Load Co | ombinatio  | n Region  | United States           |       | ~   |  |  |
| Load Co | ombinatio  | n Code    | 2015 IBC ASD            |       | ~   |  |  |
|         | Generat    | e Beam De | flection Load Combinati | ions  |     |  |  |
| Notiona | al Load Op | tions     | None                    |       | ~   |  |  |
|         | Revers     | ible      |                         |       |     |  |  |
|         |            |           |                         |       |     |  |  |
| Set     | as Defaul  | t         | Generate                | Close | e   |  |  |

- 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).

| Gravity | Wind       | Seismic     |                 |   |
|---------|------------|-------------|-----------------|---|
| Load C  | ombinatio  | n Region    | United States   | ~ |
| Load C  | ombinatio  | n Code      | 2015 IBC ASD    | ~ |
| Wind L  | oad Optic  | ins         | X and Z w/Ecc   | ~ |
|         | Rever      | sible       |                 |   |
| G       | ienerate R | oof Wind L  | oads            |   |
| G       | ienerate S | emi-Rigid D | Diaphragm Loads |   |
|         |            |             |                 |   |

- 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).

| 3D L    | oad Combi                | ination Ge  | nerator ? >                                      |
|---------|--------------------------|-------------|--|
| Gravity | Wind                     | Seismic     |  |
| Load C  | ombinatio                | n Region    | United States ~                                  |
| Load C  | ombinatio                | n Code      | 2015 IBC ASD V                                   |
| Seismic | : Load Opt               | ions        | X and Z w/Ecc 🗸                                  |
| (       | Revers                   | ible        | Include p Include Ev (vertical)                  |
| (       | Include                  | e Non Orth  | no (100%+30%)                                    |
| _ A     | dd Notion                | al Loads to | Seismic Load Combinations                        |
| 0       |                          | 0           |  |
| Overst  | rength LC                |             | None   |
|         | Revers                   | ible 🔄      | Include Ev (vertical)                            |
|         |                          | e Non Orth  | no (100%+30%)                                    |
|         | ty-Limited<br>ced Frame: |             | None ~   |
|         | Revers                   | ible 🗌      | Include Ev (vertical)                            |
|         |                          | e Non Orth  | no (100%+30%)                                    |
|         | Include                  | e Non Orth  | no (100%+100%)                                   |
|         |                          |             |  |
| 0       |                          |             | and the set of the set of                        |
|         |                          |             | uded (from "Gravity" tab)                        |
|         | ve Load O                |             | uded (from "Gravity" tab)<br>Snow Load Rain Load |
| R       |                          | bad         |  |

Click Generate. Then click Close to close the dialog.

|             | ations Design<br>enerator | RSA Scaling Factor |   |  |     |        |       |            | Solve Current LC Solve Batch + Envelope |        |     |        |     | Solve Envelope Only       |       |  |
|-------------|---------------------------|--------------------|---|--|-----|--------|-------|------------|---|--------|-----|--------|-----|---------------------------|-------|--|
| Description |                           | Solve              |   |  | 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      | ш     | 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                 | <b>v</b>           | 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                 | $\checkmark$       | 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        |   |        |     |        | R   | esize t                   | he    |  |
| 20          | IBC 16-15                 | ✓                  | Y |  | DL  | 0.6    | WLZP2 | 0.6        |   |        |     |        |     | window by<br>clicking and |       |  |
| 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                 | $\checkmark$       | Y |  | DL  | 1      | ELX-Z | 0.7        |   |        |     |        | d   | raggin                    | 3     |  |
| 24          | IBC 16-12                 | ✓                  | Y |  | DL  | 1      | ELZ   | 0.7        |   |        |     |        |     | downward on<br>the bottom |       |  |
| 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   | r   | ight co                   | rner. |  |
| 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   |     |                           |       |  |

This will generate the load combinations shown below.

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 E to exit RISAFloor if you have the program open.
- Go to <u>http://www.risa.com/partners/prt\_revit.html</u>, click **Downloads**, and then select the RISA-Revit Link version you want to download.
- Download and install the link.

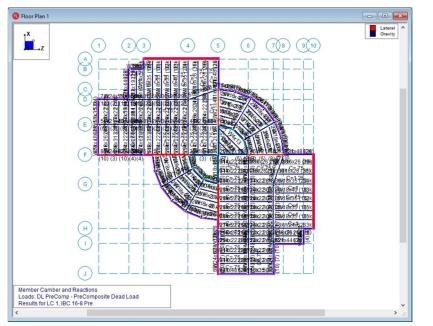
Follow the steps below to load the RISAFloor starter file for this tutorial:

- 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.

| Save Results                 | <b>—</b>                        |
|------------------------------|---------------------------------|
| Results are available. W     | ould you like to save them?     |
| (Use Tools->Preferences to r | make saving results automatic.) |
| Save Results                 | Don't 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: <u>http://www.risa.com/partners/prt\_revit.html</u>.

- 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.

| Update From RISA, Version 18.0  | ×  |
|---|--|
| General Advanced  |  |
| RISA Coordinates relative to:       Project Base Point         RISA X-Axis Aligned with:       Project North         Import Process:       Update Geometry and Sizes         Revit Levels:       Create New Levels as Required         Elements to Link with RISA         Walls       Project Grids         Diaphragms (Revit Floors)       Loads         Boundary Conditions       Load Combinations         Footings (RISAFoundation)       Foundation Slabs         Scale Factor:       1         kip       Round up to: | Concrete Rebar Import Columns Slab Pedestals Footings File Name RISA Model File: Browse. Clear |
|   | OK Cancel Help   |

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: <a href="http://www.risa.com/partners/prt">http://www.risa.com/partners/prt</a> 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 <u>https://portal.risa.com/login</u>, 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.

| C IMPORT            | C RISA-3D<br>RISAFloor<br>prial A7 Starter.rfl |                   | Click this button<br>to Browse for<br>your Input file. |
|---------------------|--|-------------------|--|
| Output file (.stp): | Detailing Choice                               |                   | you mput me.   |
| Analysis Model      | C None   | Set Mapping Files |  |
| ✓ Loads             | Use Cardinal Points                            |                   |  |
| Design Model        | C Use Element Eccentricity                     | Save as Defaults  |  |
| ☐ Loads             | C End Pullback<br>No End Pullback              | View Error Log    |  |
|                     |  |                   |  |
|                     |  |                   |  |

- 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.

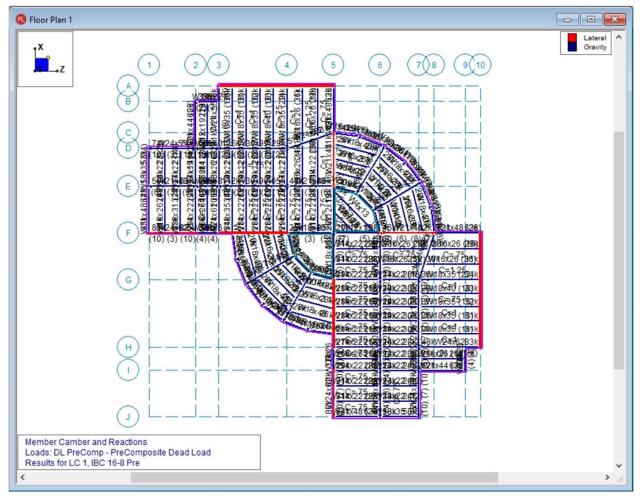
Follow the steps below to load the RISAFloor starter file for this portion of the tutorial:

- Double-click on the **RISAFloor** icon to start the program.
- Click Open Model <sup>C</sup>. Double-click the Tutorials folder, select Tutorial A7 Starter.rfl and click Open. Click Close <sup>C</sup> (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.

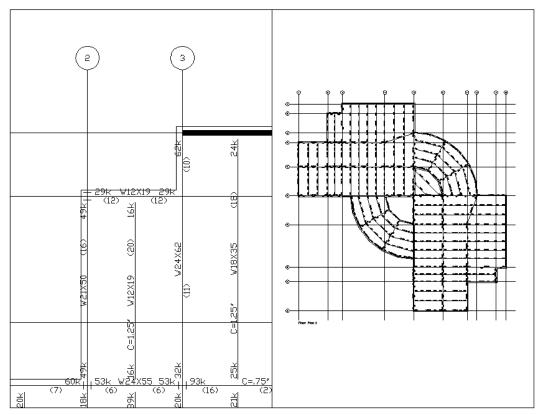


- On the Main menu, click File, click Export, and select DXF File.
- Enter the file name Tutorial and click Save. The Export DXF File dialog box will display.

Export DXF File × DXF Options Grid Bubble Type DXF Floors All - Circle DXF Scale Factor 1 Hexagon DXF Units in • DXF Export Layer Options Beam Options Depth Only Beam Size ? Text Placement Locations Top Bottom Parentheses Options Size Callout 💿 C Camber C= • Camber .  $\mathbf{C}$ Reaction bare • Reactions . C Uniform Studs 🖲 C Studs () • Beam Standoff 12 Camber Units Symbol in • ' "' (Double Quote) Text End Standoff 3 in O 'in.' Text Perp Standoff 3 in OK Cancel Save Defaults Help

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.

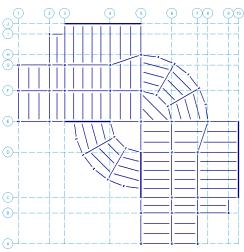
- On the RISA toolbar, click on New Model <sup>D</sup> to start a fresh model.
- Because you want to import geometry rather than start from scratch, click Close to exit the Starting a Model dialog box.
- On the Main menu, click File, click Import, then select DXF File.
- Select the tutorial file you 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:

| Import DXF File                                    |                               |                                      | ×          |
|--|-------------------------------|--------------------------------------|------------|
| DXF Options<br>DXF Scale Factor<br>DXF Units       |                               | Floor Elevation 0<br>Beam Standoff 1 |            |
| Layer Options                                      |                               |                                      |            |
| Beam Layer   | BEAM_LINES                    | -                                    | Import ?   |
| Column Layer                                       | COLUMN_BLOC                   | KS 👻                                 | Import ?   |
| Wall Layer   | WALL_LINES                    | -                                    | ✓ Import ? |
| Grid Layer   | GRID_LINES                    | •                                    | Import?    |
| Floor Type   |                               |                                      |            |
| C Concrete Floor                                   | Slab                          |                                      |            |
| Slab Layer   | GRID_LINES                    | ~                                    | Import ?   |
| Angle Inc  | 10                            | Deg, For Arc Slab                    |            |
| <ul> <li>Beam Support</li> <li>Ignore F</li> </ul> | ed Floor<br>raming Support fo | r Beam Layer                         |            |
| ОК   | Cancel                        | Save Defaults                        | Help       |

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 <u>info@risa.com</u>.

# 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: <u>www.risa.com</u>.

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.

Full Model Lateral Gravity <

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: <u>www.risa.com</u>.

## **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.

| 📵 RISAFloor ES Demonstration - [C:\Users\Documents\RISA\Model Files\untitled.rfl]  |   |
|--|---|
| File Edit Settings View Insert Modify Spreadsheets Solve Results Tools Window Help | Director                                |
| 🗅 😅 🖬 🖶 🗢 🐢 🌑 🏭 IIII 👯 🖩 🗱 🖃 🏢 🕼 LC = 🔟 😑 🐼 👩 🗹 🧇                                  |   |
|  |   |
|  |   |
|  | Data Entry 🖾                            |
|  | Project Grid                            |
|  | Materials                               |
|  | Deck Definitions<br>Member Design Rules |
|  | Wall Design Rules                       |
|  | Connection Rules                        |
|  | Area Load Definitions                   |
|  | Point Locations<br>Columns              |
| Starting a Model   | Column Stacks                           |
|  | Wall Panels                             |
| Explore a Sample Model   | Beams                                   |
| Open a Sample Model  | Diaphragms<br>Point Loads               |
|  | Line Loads                              |
| Learn how to use RISAFloor   | Tapered Area Loads                      |
|  | Load Combinations<br>Floors             |
| Launch Tutorial in Separate Window   |   |
|  |   |
| Create a New Model   |   |
| - Uick here to create a new floor plan   |   |
| C - Lick nere to create a new indui plan   |   |
|  |   |
| or Reload a Previously Created Model   |   |
| Click here to go to the File Open dialog   |   |
|  |   |
|  |   |
|  |   |
|  |   |
|  |   |
|  |   |
|  |   |
| Don't show this panel when starting a model?                                       |   |
| Close  |   |
|  |   |
|  |   |
|  |   |
|  |   |
|  |   |
|  |   |
|  |   |
|  |   |
|  |   |
| Press Ctrl-N to start a New File   |   |
| Press Ctrl-N to start a New File   |   |

**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.

| reate No                         | ew Floor      | Plan         |                            |           | × |
|----------------------------------|---------------|--------------|----------------------------|-----------|---|
| .abel:                           | Floor Plan    | 1            |                            |           |   |
| Floor Ty                         | pe            |              |                            |           |   |
| • Co                             | ncrete Flo    | or Slab      |                            |           |   |
| O Be                             | am Suppo      | rted Floor   |                            |           |   |
| Floor Cre                        | eation Opti   | ions         |                            |           |   |
| 🖲 Ori                            | ginal Floor   |              |                            |           |   |
| • Co                             | py of No      | Floors De    | fined                      | -         |   |
|                                  |               | opy Colum    |                            |           |   |
|                                  |               | opy Beams    |                            |           |   |
|                                  |               |              | agm Edges                  | /Openings |   |
|                                  | 🖂 🖂           | opy Decks    |                            |           |   |
| 🔽 Copy Loads                     |               |              |                            |           |   |
| Copy Support Lines/Design Strips |               |              |                            |           |   |
|                                  | M 0:          | opy Wall P   |                            |           |   |
|                                  |               |              | Floor Below<br>Floor Above |           |   |
| <b>C</b> (1)                     | 11 / <b>N</b> |              |                            |           |   |
| U Un                             | id of: INC    | o Floors Die | hined                      | ~         |   |
| Elevation                        | r:            | 15           | ft                         |           |   |
| Default A                        | Area Load:    | Office       |                            | •         |   |
| Slab Typ                         | e:            | Reinford     | ced Concret                | e 💌       |   |
| Design N                         | lethod:       | Design       | Strips                     | •         |   |
|                                  |               |              |                            |           |   |
|                                  |               |              |                            |           |   |
| 1                                | 0k            | Can          | aal                        | Help      |   |

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.

| DXF Drawing Grids ×   |
|---|
| Drawing Grid   Snap To Options Import DXF   |
| This allows you to use a DXF drawing as a Drawing Grid (underlay) which will provide snap points for modeling |
| Drawing Grid Origin (ft)<br>Z Axis 0 X Axis 0   |
| Click on a location to relocate the Drawing Grid  |
| DXF Import Options  |
| DXF Scale Factor 1 DXF Plane  |
| DXF Units in 💌 💿 XY 🔿 XZ 🔿 YZ   |
| Angle Increment 5 Deg, For Importing Arcs/Circles   |
| Select DXF file:  |
| Browse For File   |
| Or click here to access a recently used DXF file:   |
| C:\RISA\Model Files\Tutorials\Tutorial B1 Drawing Grid.dxf  |
| Show Grid As  |
| Lines O Points  |
| Cancel Help   |

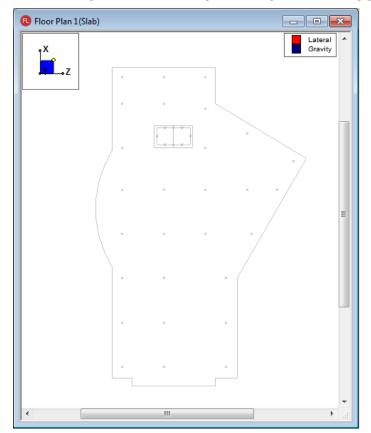
- Click on the Browse for File button and then browse to the Tutorials file under your C:\RISA User Data\%Username%\Model Files\Tutorials folder.
- Select the file called **Tutorial B1 Drawing Grid.dxf**.

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.

| Select Layers for Drawing Grid   | × |
|--|---|
| Use All Layers   |   |
| Select Layers From the List  |   |
| Image: Market of the second secon |   |
| UncheckAll CheckAll  |   |
| Done   |   |

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

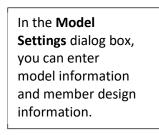


### **Model Settings**

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

- On the RISA toolbar, click the Model Settings button S.
- 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).

| (Global) Model Settings $\qquad \qquad \qquad$   |
|--|
| Description   Solution Codes   Composite   Wind   Seismic   Concrete   |
|  |
| HR Steel : AISC 14th (360-10): ASD   |
| Connections : AISC 14th (360-10): ASD  |
| CF Steel : AISI S100-16: ASD   |
| CFS Walls : AISI S400-15 wS1-16 & AISI S240-15 💌   |
| Concrete : ACI 318-14 Temperature  |
| Wood : AWC NDS-15: ASD <   |
| Masonry : TMS 402-16: ASD  |
| Steel Joist : SJI 43rd/44th Edition: ASD   |
| Composite Joist : SJI 200-2015: LRFD   |
| LL Reduction : IBC   |
| LL Reduction : IBC   |
| Damping  |
| Beam Vibrations : AISC DG 11 2nd Edition   Output  Damping  Dampin |
| Damping  |
| Joist Vibrations : SJI TD 5 0.03   |
| Joist Girder Load Tolerance: 0.2 Kips  |
| Save as Defaults   |
| OK Cancel Apply Help   |



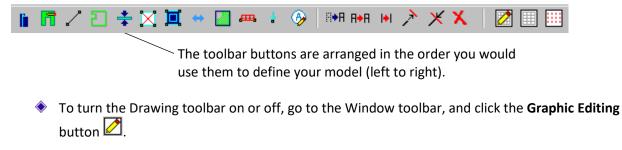
Close the Model Settings dialog box:

• 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:



## 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 is 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.

| Draw Concrete Columns   | × |
|---|---|
| Draw Columns   Modify Properties   Modify Design  |   |
| Concrete Column Shape   |   |
| C Explicit Shape:   |   |
| Concrete Column Options<br>Material:  |   |
| Conc3000NW  |   |
| Design Rules:   |   |
| Typical Add   |   |
| Rectangular Shear Cap (in)<br>Thickness 6 Depth 24 Width 24   | 1 |
| Function     Orientation Options       O Lateral     Orient to Point:       O Gravity     Rotate Angle:                                   |   |
| Column Prefix CS Point Prefix N   |   |
| Drawing Options   |   |
| Click or Box Locations for Columns  |   |
| <ul> <li>Offset Columns From the Click/Box Locations (ft):</li> <li>Z Axis</li> <li>Angle</li> <li>Angle</li> <li>and Distance</li> </ul> |   |
| Keep this dialog open when Apply is pressed?  |   |

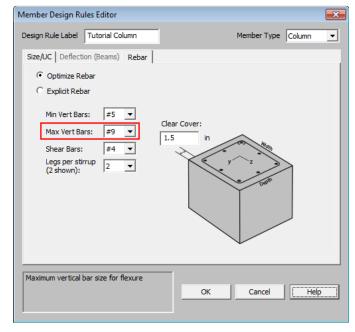
#### **Column Design Rules**

Create the design rule:

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

| Member Design Rules Editor  | ×  |
|---|--|
| Design Rule Label Tutorial Column   | Member Type Column   |
| Size/UC Deflection (Beams) Rebar  |  |
|   | Min Width: in<br>Max Width: 22 in  |
| Min Depth:<br>Max Depth: 22<br>Max Bending Check: 1<br>Max Shear Check: 1 | in the second se |
| Maximum width to use when considering<br>potential shapes                 | OK Cancel Help   |

- 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.

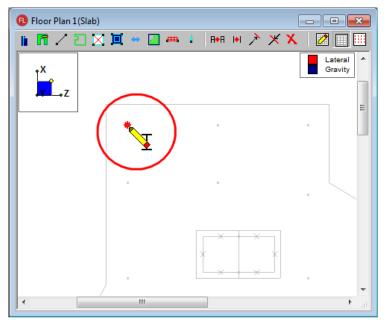
| Draw Concrete Columns X   |
|---|
| Draw Columns   Modify Properties   Modify Design  |
| Concrete Column Shape     Shape Group:     Rectangular  |
| C Explicit Shape:   |
| - Concrete Column Options<br>Material:  |
| Conc4000NW  |
| Design Rules:   |
| Tutorial Column   Add   |
| Thickness     6     Depth     24     Width     24       Function     Orientation Options       © Lateral     Orient to Point:   |
| Column Prefix CS Point Prefix N   |
| Drawing Options   |
| <ul> <li>Click or Box Locations for Columns</li> </ul>  |
| Offset Columns From the Click/Box Locations (ft):     Z Axis     Axis     Axis     Axis     Angle     Angle     Angle     Axis     Ax |
| Keep this dialog open when Apply is pressed?  |

Click Apply to start drawing the columns.

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

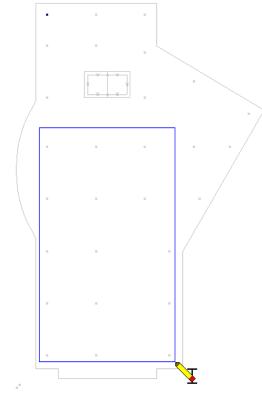
You can now start defining the columns by clicking on the drawing grid locations as shown below:

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.

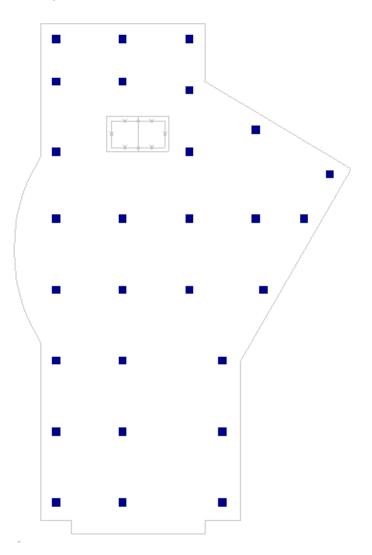
Hold down the left mouse button and drag the cursor to select the box over your column grid locations.



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:

| Column Stack Manager                         |  | $\times$ |
|--|--|----------|
| •  |  | ft       |
| <b>米</b><br>                                 | Display<br>Label No Labelin 💌 Color Basis Black            | •        |
| *  | Lift Properties (0.000 ft ~ 15 ft)                         |          |
|  | Splice Type Default (Mo 🗸 🗆 Use? Splice Conn None 👻        | Use?     |
|  | Shape Rectangu 🗆 Use? Material Conc40001 🗸                 | Use?     |
|  | Function Gravity VIC Use? Design Rules Tutorial Co V       | Use?     |
|  | Flex. Layout Use Desigi 🗸 🗖 Use? Shear Layout Use Desigi 🗙 | Use?     |
|  | Design Properties (Floor Plan 1)                           |          |
|  | Lbyy/Luyy (ft) 🔽 Use? Kyy                                  | Use?     |
|  | Lbzz/Luzz (ft) Use? Kzz                                    | Use?     |
|  | Ltorque (ft) Use? Cb                                       | Use?     |
| Floor Plan 1                                 | Lcomp top(ft) Use? C1                                      | Use?     |
| 15 ft  | Lcomp bot(ft) Use? Lcr                                     | Use?     |
|  | Cm-yy Use? y Sway  | Use?     |
|  |  | Use?     |
|  | a User   |          |
|  | Ch. Conn. Welded   |          |
| Base   |  |          |
|  | What happens when Apply is pressed?                        |          |
| CS1  | Apply Entries to All Lifts                                 |          |
|  | C Apply Entries by Clicking Lifts Individually             |          |
| <  | Apply Close Cancel Clear Use Help                          |          |
| Click on a column lift to display properties | CS1  |          |

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**.

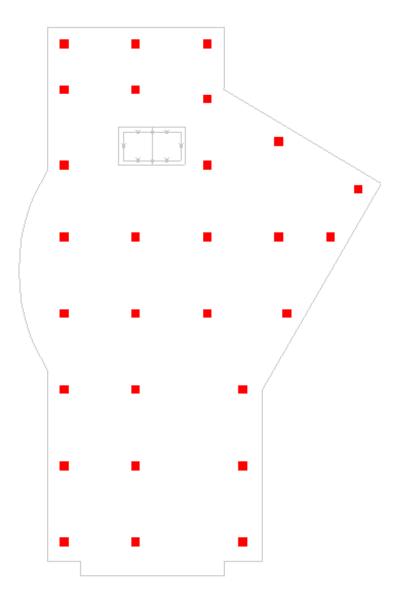
Close the **Column Stack Manager** without making any changes by clicking **Cancel**.

## 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.

| Modify Column Proper   | ties  | × |
|--|---|---|
| Draw Columns Modi  | fy Properties Modify Design   |   |
| Concrete Column Shap<br>Shape Group:<br>Rectangular                          | Use?  |   |
| C Explicit Shape   |   |   |
| Concrete Column Optic<br>Material:   | ns<br>Use?  |   |
| Conc4000NW   | <b>_</b>  |   |
| Design Rules:  | Use?  |   |
| Tutorial Column  | <b>•</b>  |   |
| Shear Cap Options<br>C Delete Rectangula<br>Add/Modify Rectar<br>Thickness 6 |   |   |
| Shear Cap Eccentricity<br>ey 0   | (in) Use?   |   |
| Function 🔽 Use? -  | Orientation Options — 🔲 Use? —  |   |
| <ul> <li>Lateral</li> </ul>  | Orient to Point:  |   |
| C Gravity  | Rotate Angle: deg   |   |
| Splices Options  | Use?  |   |
| Splices  | Distance Below: 0 ft  |   |
| Type: Default(M -  | Parapet Height 0 ft   |   |
|  | elected Columns<br>ting/Boxing Columns Individually<br>h when Apply is pressed? |   |
| Apply  | Use Cancel Help   |   |

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



## Wall Panels

Prepare to draw your wall panels:

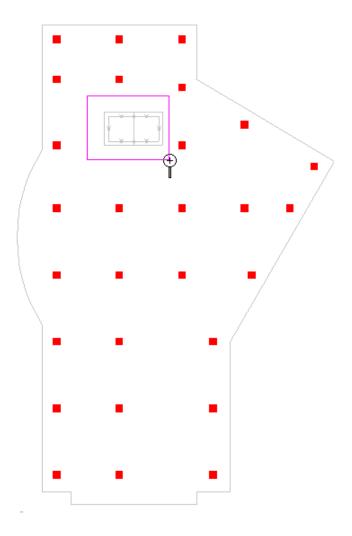
- On the Drawing toolbar, click Draw or Modify Walls
- Set the Material to Concrete and change the Material Set to Conc4000NW.
- Under **Function**, click **Lateral**. The wall panel settings should match those in the image below.

| Draw Wall Panel   | s<br>els Modify Wall Panels                              | ~ |
|---|--|---|
| Elevation<br>Bot Level                                    | Base 💌<br>Floor Plan 1 💌                                 |   |
| Material<br>Masonry<br>Concrete<br>CFS<br>Wood<br>Ceneral | Material Set<br>Conc4000NW 💌<br>Design Rule<br>Typical 💌 |   |
| Wall Prefix W<br>Point Prefix N                           |  |   |
| Click Apply.  | Cancel Help  |   |

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

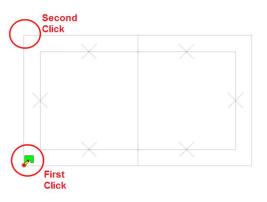
First zoom in on the area of the elevator opening using the Box Zoom tool.

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

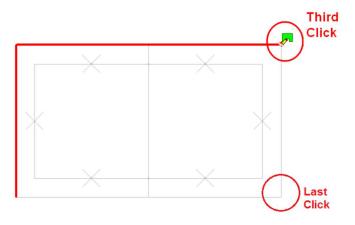


Now that you have a zoomed in view, start drawing your wall panels:

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.



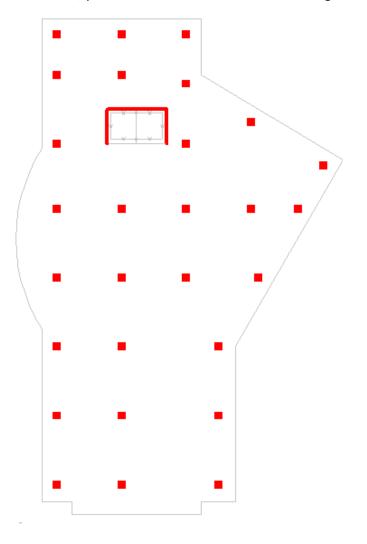
Continue around the outside of the elevator opening.



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.



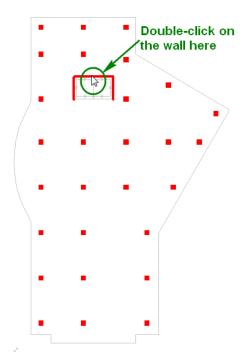
When finished, your model should look like the following.

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.

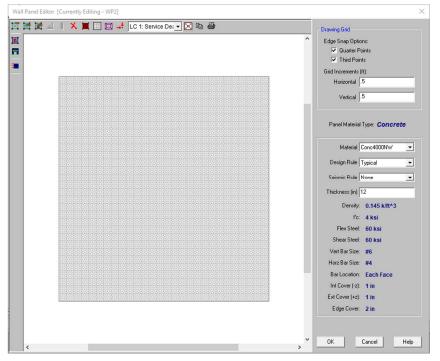
### **Modifying Wall Panels**

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 A 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.

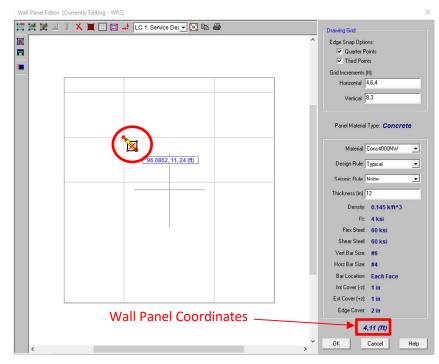
| Vall Panel Editor [Currently Editing - WP2] | >                             |
|---|-------------------------------|
| 🏹 🈹 😹 🔳 📗 🗶 📕 🗐 🚅 LC 1: Service De: 🔽 🖻 🖨   | Drawing Grid                  |
|   | Edge Snap Options:            |
| 9   | Quarter Points                |
|   | Third Points                  |
|   | Grid Increments (ft):         |
|   | Horizontal 4,6,4              |
|   | Vertical 8,3                  |
|   |                               |
|   | Panel Material Type: Concrete |
|   | Material Conc4000NW           |
|   | Design Rule Typical 💌         |
|   | Seismic Rule None             |
|   | Thickness (in) 12             |
|   | Density: 0.145 k/ft^3         |
|   | fc: 4 ksi                     |
|   | Flex Steel: 60 ksi            |
|   | Shear Steel: 60 ksi           |
|   | Vert Bar Size: #6             |
|   | Horz Bar Size: #4             |
|   | Bar Location: Each Face       |
|   | Int Cover (-z): 1 in          |
|   | Ext Cover (+z): 1 in          |
| After entering the Grid Increments,         | Edge Cover: 2 in              |
| click in the model space to update          |                               |
| the grid display                            |                               |
|   | OK Cancel Help                |

Now draw in the opening.

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

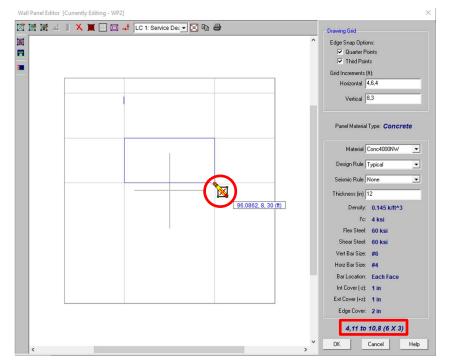
Note: Your cursor will change to 🔯 .

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

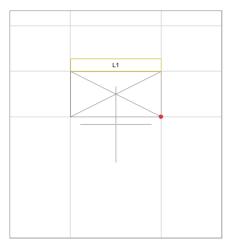


First click on the top left of the opening (4,11).

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

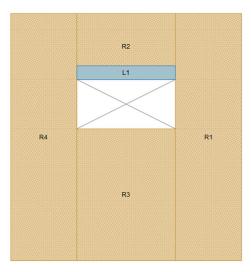
When finished, you can add in the design regions around the openings. The **Wall Panel Editor** allows you to create design regions in two ways: manually draw in each region, or by using the automated generation tool. You will use the automated generation for this model.

Click on the Generate Wall Regions Automatically button 2 (in the upper left corner of the Editor).

Turn the rendered view back on for a better view.

Click the Toggle Rendered View button A 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 <u>https://www.risa.com/documentation</u>.

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 <sup>C</sup>. Double-click the Tutorials folder, select Tutorial B2 Starter.rfl and click Open. Click Close <sup>C</sup> (or Cancel) to exit the Model Settings dialog box.
- On the Window toolbar, click the Graphic Editing button 2 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.

| DXF Drawing Grids   |                                |
|---|--------------------------------|
| Drawing Grid Snap To Options Import DXF   |                                |
| This allows you to use a DXF drawing as a Drawing Grid (underlay) which will provide snap points for modeling                   |                                |
| Drawing Grid Origin (It)         Z Axis       0         X Axis       0         Click on a location to relocate the Drawing Grid | Select Layers for Drawing Grid |
| DXF Import Options  | Use All Layers                 |
| DXF Scale Factor 1 DXF Plane  | Select Layers From the List    |
| DXF Units 📊 💽 XY 🔍 XZ 🔍 YZ  | 🗹 Default Layer                |
| Angle Increment 5 Deg, For Importing Arcs/Circles   | I Slab Edge                    |
| Select DXF file:  | Column Points                  |
| Browse For File   |                                |
| Or click here to access a recently used DXF file:   |                                |
| C:\RISA\Tutorial B1 Drawing Grid.dxf  |                                |
|   |                                |
|   |                                |
| Show Grid As  | UncheckAll CheckAll            |
| ⊙ Lines ○ Points  |                                |
| Cancel Help   | Done                           |

## **Draw the Slab**

Start by drawing in the slab.

On the Drawing toolbar, click Create Slab Perimeter 2 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.

| Draw Slabs   |  |
|--|--|
| Draw Slabs   Modify Slabs   Opening   Offset Slabs |  |
| Definition<br>Existing DEMO Slab Definition Vew    |  |
| Slab Prefix S Material Set DEMO-Conc3000NV         |  |
| Point Prefix N Thickness 8 in                      |  |
| Keep this dialog open                              |  |
| Apply Cancel Help                                  |  |

## Slab Definition

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

| Draw Slabs   | $\times$ |
|--|----------|
| Draw Slabs   Modify Slabs   Thickness   Opening   Modify Edges |          |
| Definition<br>Existing Semi-Rigid Definition  New              |          |
| Slab Prefix S Material Set Conc3000NW                          |          |
| Point Prefix N Thickness 12                                    | in       |
| Keep this dialog open  |          |
| Apply Cancel Help  |          |

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

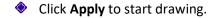
| Slabs Properties            | $\times$ |
|-----------------------------|----------|
| General                     |          |
| Label: Tutorial Slab        |          |
| Slab Type<br>Flat Slab      |          |
| Thickness t 8               | in       |
| Concrete Material Diaphragm |          |
| Conc3000NW   Semi-Rigid     | •        |
| Loads                       |          |
| Const DL 0 ksf              |          |
| Const LL 0 ksf              |          |
| Super DL 0 ksf              |          |
| OK Cancel Help              |          |

• 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 Slabs                                  |                      |           |                 |                   | $\times$ |
|---|----------------------|-----------|-----------------|-------------------|----------|
| Draw Slabs M                                | lodify Slabs         | Thickness | Openi           | ng Modify Edges   |          |
| Definition       Existing       Slab Prefix | Tutorial Slab        |           | ▼<br>terial Set | New<br>Conc3000NW |          |
| Point Prefix                                | N                    | Т         | hickness        | 8                 | in       |
| 🗖 Keep this                                 | dialog open<br>Apply | Cancel    | 1               | Help              |          |
|   |                      |           |                 |                   |          |

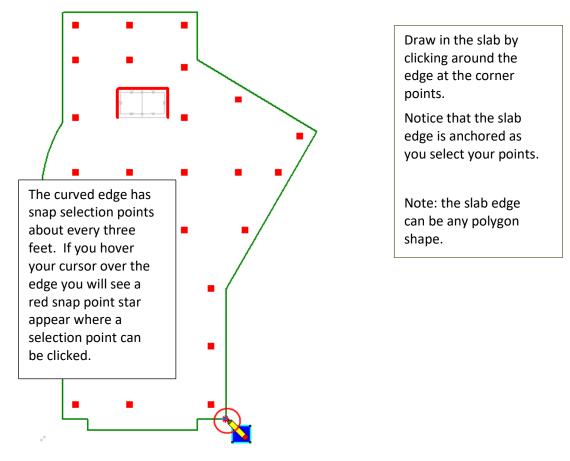
### Draw Slab



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

Draw your Slab edge:

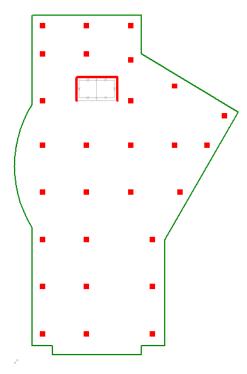
• Using the snap points from our DXF drawing grid background to draw in the edge of the slab.



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.

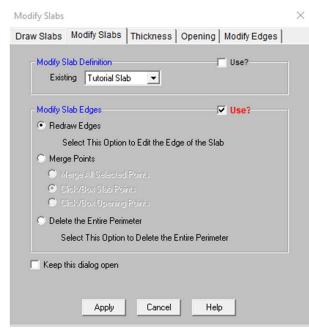
Right-click your mouse once more to cancel the drawing tool.

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 the **D**rawing 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 **D** 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.

| 📵 Slab L | Loads                 |               |                |                |                |
|----------|-----------------------|---------------|----------------|----------------|----------------|
| Genera   | l Loads               |               |                |                |                |
|          | 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

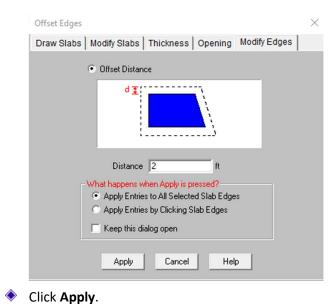
Next, you will modify the slab edges.

#### **Offset Slabs**

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

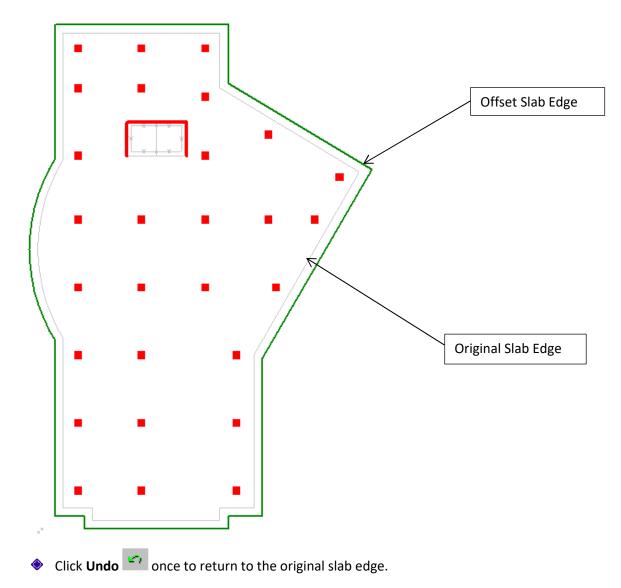
- Click on the Modify Slab Edge button I from the Drawing toolbar.
- Enter an Offset Distance of 2 feet and select Apply Entries to All Selected Slab Edges.

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



• • • • • • •

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.



#### 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 1 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.

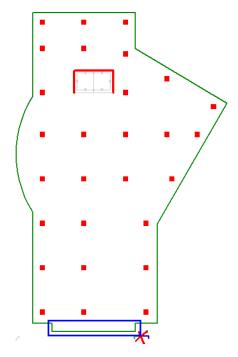
| Modify Slabs   | $\times$ |
|--|----------|
| Draw Slabs Modify Slabs Thickness Opening Modify F                                     | Edges    |
| Modify Slab Definition Use? -<br>Existing Tutorial Slab                                |          |
| Modify Slab Edges Vise?  |          |
| Sedraw Edges   |          |
| Select This Option to Edit the Edge of the Slab  |          |
| Merge Points   |          |
| Merge All Selected Points  |          |
| Click/Box Slab Points  |          |
| C Click/Box Opening Points   |          |
| O Delete the Entire Perimeter  |          |
| Select This Option to Delete the Entire Perimeter                                      |          |
| <ul> <li>Keep this dialog open</li> <li>Apply</li> <li>Cancel</li> <li>Help</li> </ul> |          |

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.

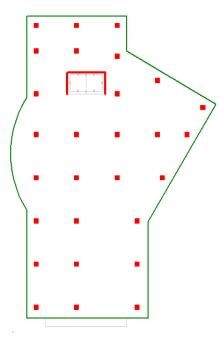
. .

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

Box select the lower slab edge as shown here:

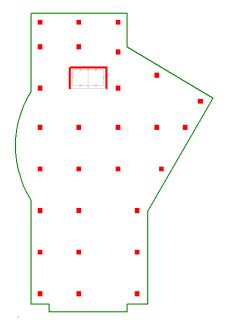


When you let go of your cursor, you will see the edge of the slab pops in and all the points align in a straight line:



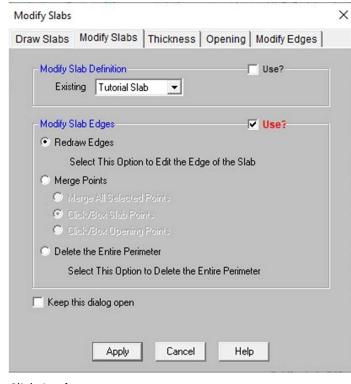
- Click ESC on your keyboard or right-click your mouse to escape the modification tool.
- Click Undo source to return to our original slab edge.

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.

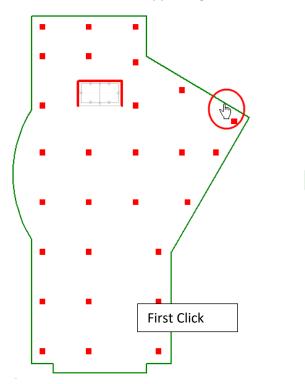
Once you click **Apply**, this instructional dialog will open:

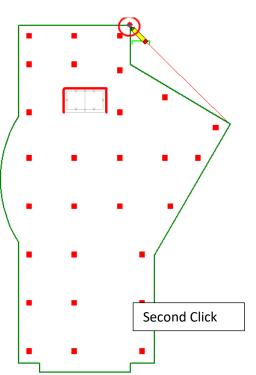


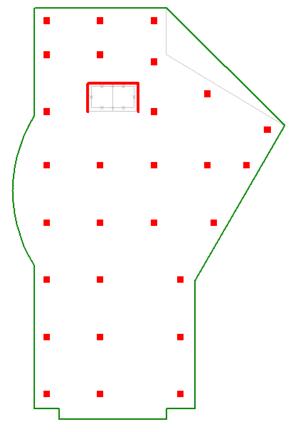
Click **OK** to continue.

**Note:** Your cursor in the graphical model view will now look like this  $\sqrt[n]{}$  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:

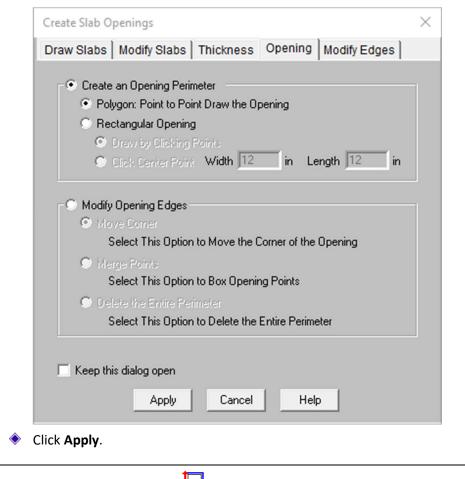
Undo this step to return to the original slab:

- 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 K from the Drawing toolbar.
- Select Polygon: Point to Point Draw the Opening.



**Note:** Your cursor will change to <sup>1</sup> 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 is 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:

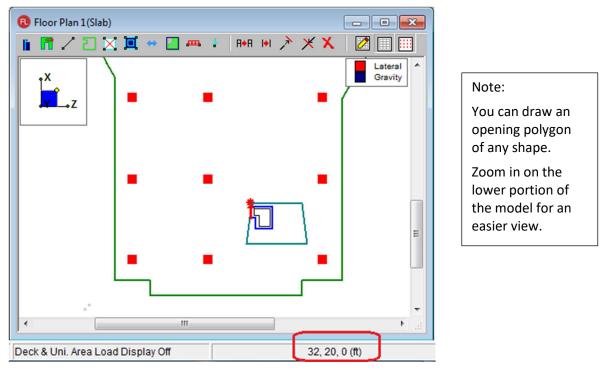
(32,20,0)

(42,20,0)

(43,12,0)

(31,12,0)

Once again on (32,20,0)

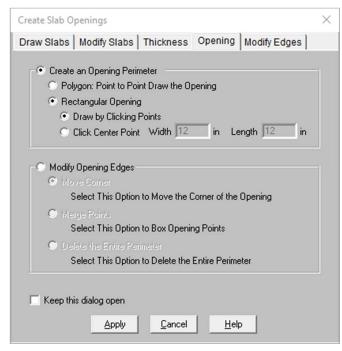


Once you have drawn in the opening, be sure to turn off the Universal Snap Points and the Drawing Grid.

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



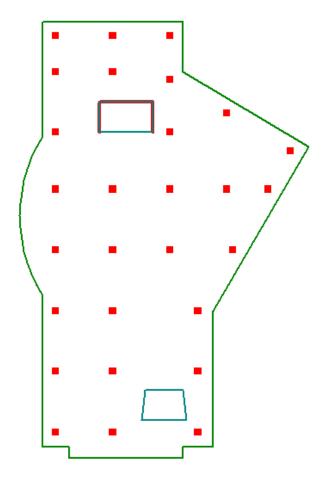
#### Click Apply.

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.

| First Click |              |
|-------------|--------------|
|             | Second Click |
| •           |              |

Right-click to release the opening tool.



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

## **Support Lines and Design Strips**

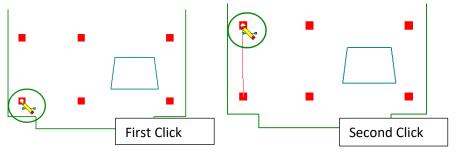
Design strips are used to create design regions within a slab. Each design strip will contain automatically defined design cuts which will control the reinforcement design for that design strip. The results for the entire design strip will be determined by the maximum moment demand of the governing design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

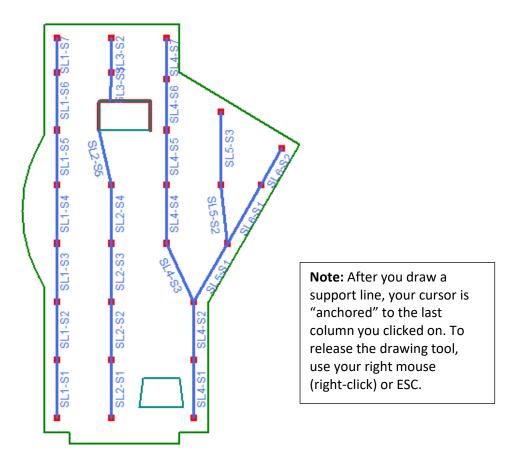
- Click on the Support Lines button Support Line on the left side of your screen.
- Under Strip Width, select Automatic.
- Under Strip Orientation, select North-South.

|   | Draw Support Line     |                             | $\times$  |  |
|---|-----------------------|-----------------------------|-----------|--|
|   | Draw Support Line     | Modify Properties   Mod     | lify Area |  |
|   | Strip Width           |                             |           |  |
|   | Automatic             |                             |           |  |
|   | ◯ Fixed               | f                           | t         |  |
|   | Strip Orientation     | North-South 💌               |           |  |
|   | Design Cuts per Strip | 50                          |           |  |
|   | Design Rule           | N-S Rebar 💌                 |           |  |
|   | Support Line Prefix   | SL                          |           |  |
|   | 🔲 Keep this dialog op | pen after Apply is pressed? | ,         |  |
|   | Apply                 | Close Help                  |           |  |
| ۲ | Click Apply.          |                             |           |  |

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.

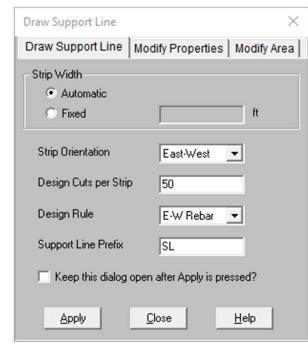


Before drawing the East-West Support Lines, toggle off the view of the North-South Lines for a simpler view.

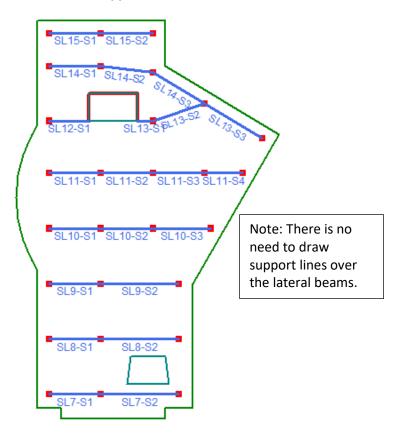
- On the Window toolbar, click on the black arrow next to the **Design Strip** button (upper right corner).
- Select E-W Support Line.

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.

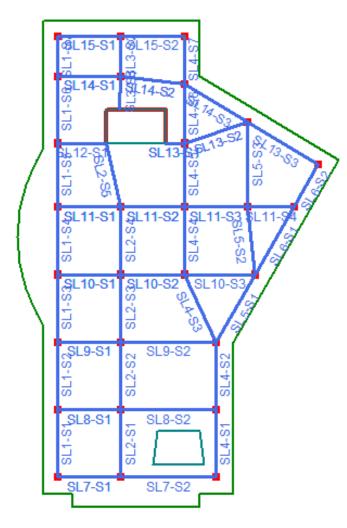


- Click Apply.
- Draw in all the Support Lines as shown below.



Toggle back on the display of all Support Lines.

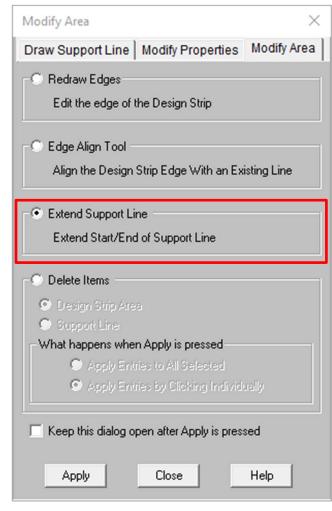
- On the Window toolbar, click on the black arrow next to the Design Strip button
- Select All.



**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.

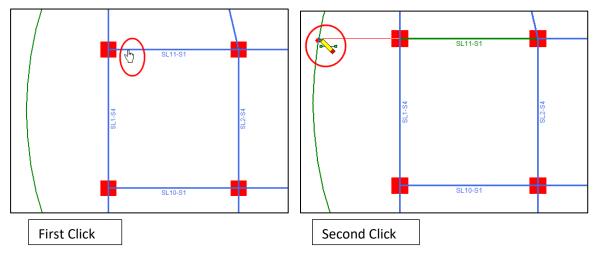
- Return to the **Draw Support Line** dialog by clicking CTRL+D.
- Click on the Modify Area tab.
- Select Extend Support Line.



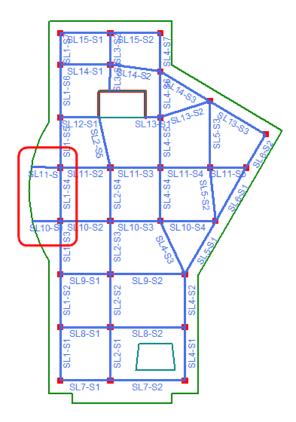
Click Apply.

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.



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



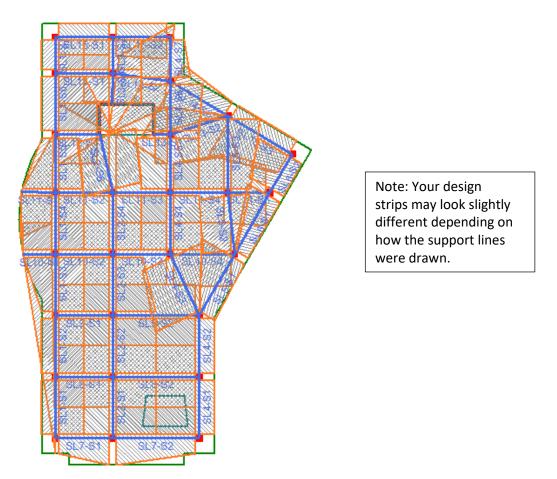
#### **Generate Design Strips**

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 Click 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:



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.



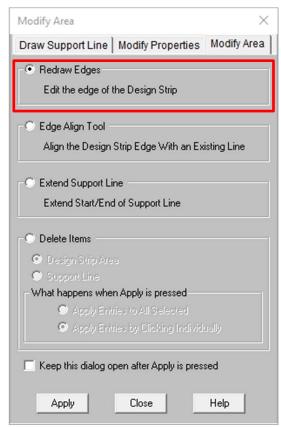
You will now manually edit the **Design Strip** edges.

- Click on the Support Line button
- Click on the Modify Area tab.

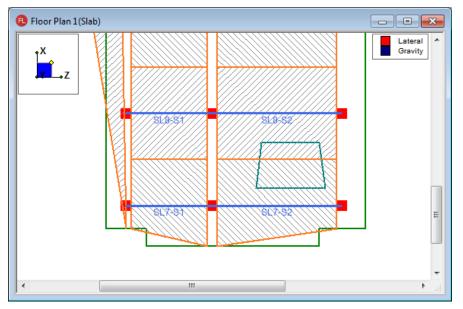
#### **Redraw Edges**

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.

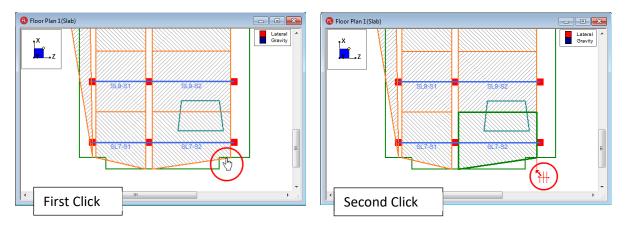
Select Redraw Edges.



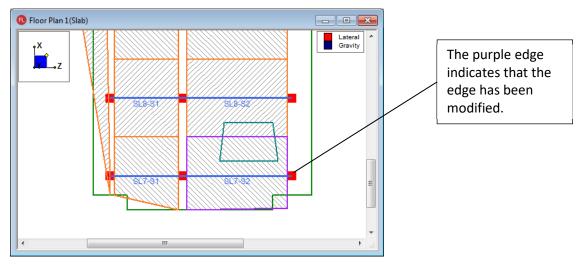
- Click Apply.
- Zoom in on the bottom of the building, as shown below.



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.

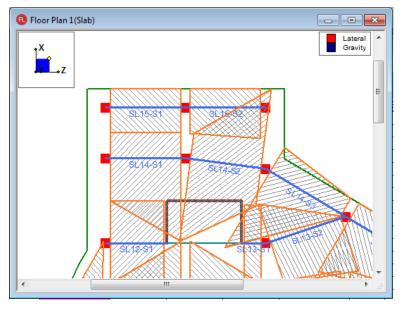
## Edge Align Tool

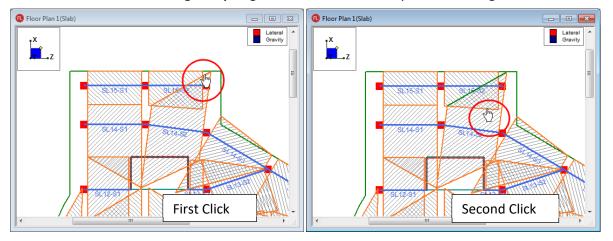
Next, you will use the **Edge Align Tool** to modify some of the other Strips. This tool will allow you to redefine a **Design Strip** edge by simply snapping it to a nearby line.

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

| Modify Area                                | $\times$    |
|--|-------------|
| Draw Support Line   Modify Properties      | Modify Area |
| C Redraw Edges                             |             |
| Edit the edge of the Design Strip          |             |
| 🕘 Edge Align Tool —————                    |             |
| Align the Design Strip Edge With an Exi    | isting Line |
| C Extend Support Line                      |             |
| Extend Start/End of Support Line           |             |
| C Delete Items                             |             |
| 🗴 Design Strip Area                        |             |
| Support Line                               |             |
| What happens when Apply is pressed         |             |
| Apply Entries to All Selected              |             |
| Apply Entries by Clicking Individe         | ally        |
| Keep this dialog open after Apply is press | sed         |
|  |             |

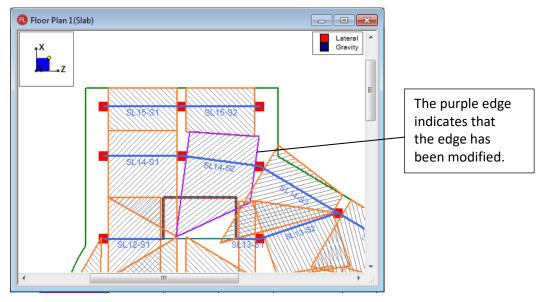
- Click Apply.
- This time zoom in on the upper edge of the model, as shown below:





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.

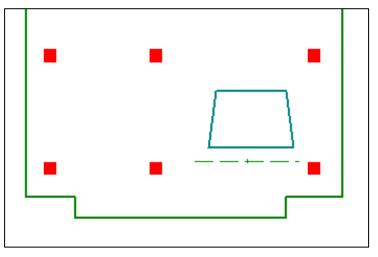
Before drawing the **Design Cut**, turn off the display of the **Design Strips**.

- Click on the black arrow next to the Design Strips button III 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 if 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.

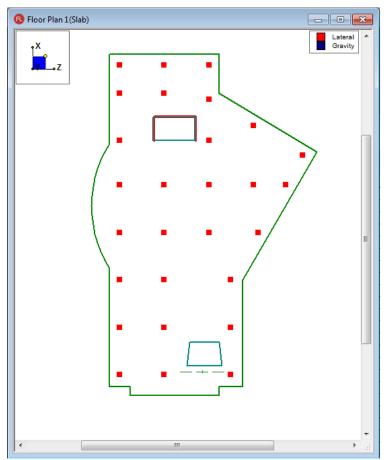
# Part B: Tutorial 3 – Loading

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 *C* 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:

| 📵 Unifo | orm Area Loads |          |            |         |         |           |         | - • ×         |
|---------|----------------|----------|------------|---------|---------|-----------|---------|---------------|
| Standa  | rd Other       |          |            |         |         |           |         |               |
|         | 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     | N        |            | .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<br>PostDL | These entries allow you to specify load magnitudes for different categories that will later be used in the load combinations that you solve.   |
| LL<br>LL Type   | <ul> <li>PreDL and PostDL are for dead loads that are pre- and post-composite.<br/>Remember that other dead loads (deck self-weight and construction<br/>dead load) were already defined within the deck definitions spreadsheet.</li> </ul> |
|                 | <ul> <li>LL and LL Type load columns are for the live load magnitude and the type<br/>of live load which you may choose from a list.</li> </ul>  |
| 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 is to exit the spreadsheet and return to the model view.

### Drawing Area Loads

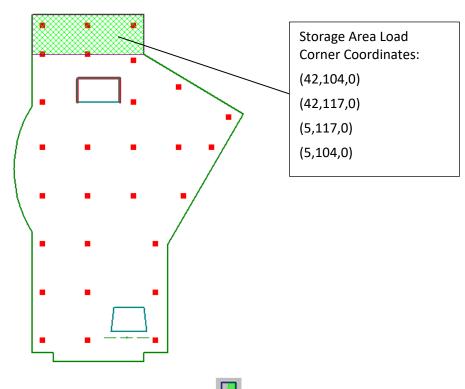
To apply local area loads:

- On the Drawing toolbar, click the Area Loads button <a>[</a>]
- Under Uniform Area Loads, in the Area Load box, select Storage, then select the Point to Point Draw of the Area drawing option.

Before drawing, turn on the Universal Snap Points.

Click the Universal Snap Points button is 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.

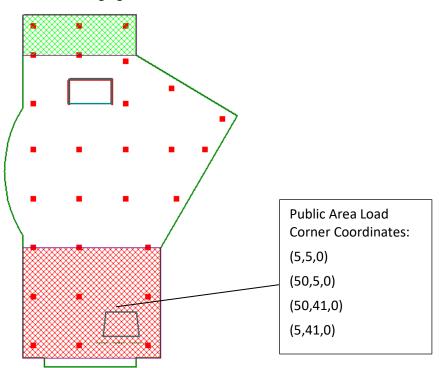


- Click on Area Loads button loss to reopen the Create Area Loads dialog box.
- Under Uniform Area Loads, in the Area Load box, select Public.

| Create Area L | .oads                                   |              |                | ) |
|---------------|---|--------------|----------------|---|
| Assign Area   | Loads Modi                              | ify Area Lo  | ads            |   |
| ⊂ ● Uniform A | vrea Loads                              |              |                |   |
|               | floor area load fi<br>t an option for a |              |                |   |
|               | ad: Public                              |              |                |   |
|               | √ithin the Area<br>ne Area Perimete     |              |                |   |
|               | a Polygon Arour                         |              | Perimeter      |   |
|               | to Point Draw of                        |              |                |   |
|               | ude Overhang                            |              |                |   |
| - C Tapered.  | Area Loads (ksf                         | )            |                |   |
| Base Mag      | Peak Mag                                | L 🔻          | Direction      |   |
| You may cl    | nange direction                         | of other loa | ds (OL) only.  |   |
| Tapered are   | ea loads are limit                      | ed to 3 or 4 | 4 point areas. |   |
| The first lin | ne drawn will be                        | used as th   | e base line.   |   |
| Keep this     | dialog open                             |              |                |   |
|               | Clos                                    |              |                |   |

166

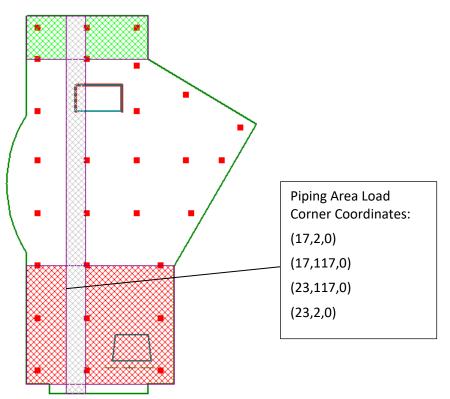
Draw in the **Public** area load by clicking on the drawing grid snap points, as shown in the following figure.



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

| Create Area Loads                      |               |                     | > |
|--|---------------|---------------------|---|
| Assign Area Loads                      | Modify Are    | a Loads             |   |
| 🕞 Uniform Area Load                    | ls —          |                     |   |
| Pick the floor area<br>select an optio |               |                     |   |
| Area Load: Add                         | Piping 💌      |                     |   |
| Click Within the                       | e Area Perim  | eter                |   |
| O Box the Area P                       | erimeter      |                     |   |
| O Draw a Polygor                       | h Around the  | Area Perimeter      |   |
| Point to Point D                       | raw of the A  | rea                 |   |
| 📃 🔲 Include Over                       |               |                     |   |
| C Tapered Area Loa                     | ds (ksf) —    |                     |   |
| Base Mag Peak M                        | lag           | Direction           |   |
| 0 0                                    | SL            | ▼ PY ▼              |   |
| You may change dire                    | ection of oth | er loads (OL) only. |   |
| Tapered area loads a                   |               |                     |   |
| The first line drawn                   |               |                     |   |
| The first and didnit                   | 1111 20 0000  | do 010 2000 mile.   |   |
| Keep this dialog op                    | ben           |                     |   |
| Apply                                  | Close         | Help                | 1 |
|  | 1             | <u></u> op          | ] |

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.

→ DL PostComp - F 🔻

-30

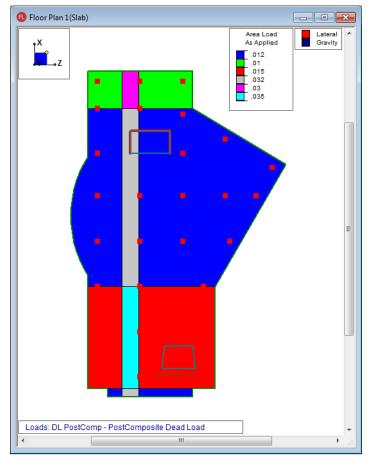
#### **Plotting Area Loads**

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.

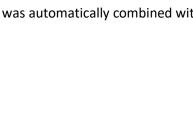
To plot the area loads:

- 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:

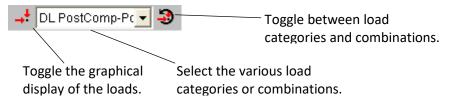


Notice that the default **Office** load was applied automatically. Also, note that the additive piping load was automatically combined with the **Office**, **Public**, and **Storage** loads that it crossed.



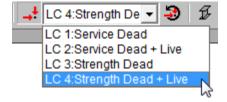
#### **Toggle Load Display**

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:

- ightarrow 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:

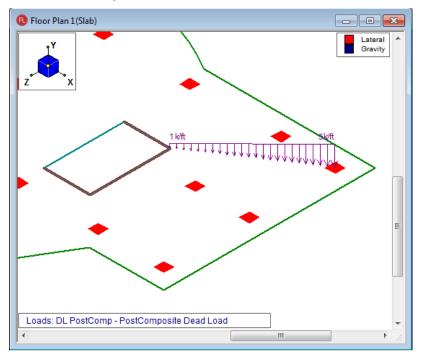
- On the Drawing toolbar, select the Line Loads button
- Enter the information as it appears in the following figure:

| Force Units Ar   | e: k/ft  |                                 |           |
|--|--|---------------------------------|-----------|
|  | Start Mag  | End Mag                         | Direction |
| DL Pre   | 0  | 0                               | (Y)       |
| DL Post  | 1  | 5                               | (Y)       |
| LL-Non 💌   | 0  | 0                               | (PY)      |
| Dyn Mass   | 0  | 0                               | (Y)       |
| OL1 🔻  | 0  | 0                               | Y v       |
| <ul> <li>Apply Lo</li> <li>Apply Lo</li> <li>Start</li> <li>End</li> </ul> | All Selected Mer<br>ad by Drawing F<br>ad by Clicking M<br>Location Offset<br>Location Offset<br>dialog open | Point to Point<br>lembers/Walls | ft        |
|  | 1  | 1                               |           |
| Apply  | Close  | He                              | elp 📗     |

| -                                      |  |  |
|--|--|--|
| <b>Note:</b> 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 lso.
- 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 <sup>Pln</sup>.
- On the Window toolbar, click **Redraw** K 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:

| Floor Point Loads                                  |   | $\times$           |
|--|---|--------------------|
|  | Magnitude (k)   | Direction          |
| DL Pre   | 0   | M                  |
| DL Post  | 0.5   | (Y)                |
| LL-Non 💌   | 0.075   | (PY)               |
| Dyn Mass   | 0   | (Y)                |
| OL1 💌  | 0   | Υ·                 |
| <ul> <li>Apply Load</li> <li>Apply Load</li> </ul> | to All Selected Po<br>by Clicking/Boxin<br>by Clicking Memb<br>Member End | oints<br>Ig Points |
| Apply  | Close   | Help               |

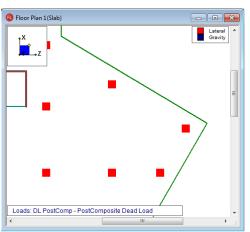
Click Apply.

| 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 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.

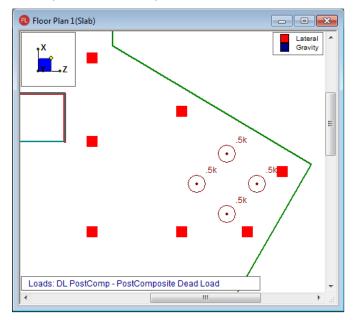


Draw in the point loads:

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:

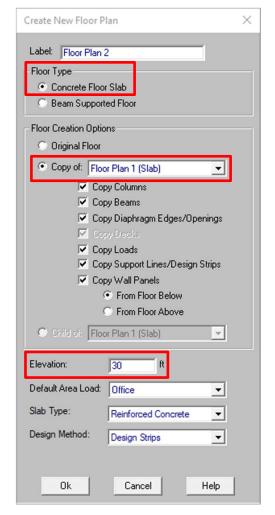


**Note:** Point loads that are applied inside the deck edges may be applied anywhere on the deck and will be attributed automatically to the beams, walls, and columns.

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 <sup> </sup> 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.

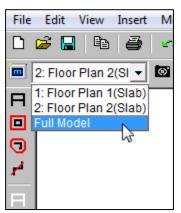
Click Ok.

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

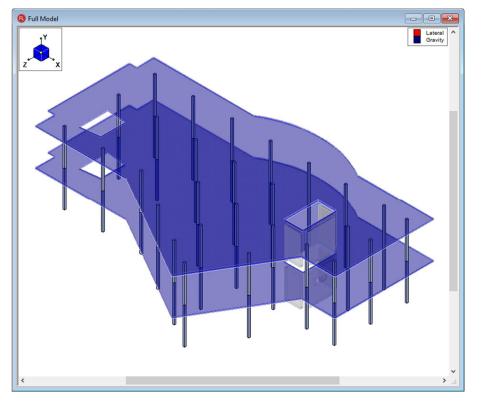
From the Window menu, select Single View.

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.

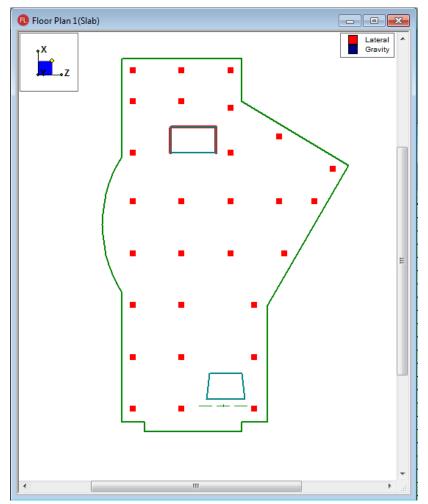
## Part B: Tutorial 4 – Solution & Results

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 Starter.rfl and 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 2 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.

| 📵 Com | binations            |       |       |        |       |        |       |        |       |        |       | • 🗙    |
|-------|----------------------|-------|-------|--------|-------|--------|-------|--------|-------|--------|-------|--------|
| Combi | inations Design      |       |       |        |       |        |       |        |       |        |       |        |
|       | Label                | Solve | Categ | Factor |
| 1     | Service Dead         | V     | 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.

| LC Region             | United States        |
|-----------------------|----------------------|
| LC Code               | 2015 IBC ASD         |
| Roof Live Load Option | ns                   |
| Include :             | RLL (Roof Live Load) |
|                       | SL (Snow Load)       |
|                       | 🔲 RL (Rain Load)     |
| Composite Options     |                      |
| Include :             | ✓ Pre Composite      |
|                       | Post Composite       |

Click Generate.

|   | binations<br>nations Design |          |       |        |         |        |       |        |       |        |       |        |       | • 🔀    |
|---|-----------------------------|----------|-------|--------|---------|--------|-------|--------|-------|--------|-------|--------|-------|--------|
|   | Label                       | Solve    | Categ | Factor | Categ   | Factor | Categ | Factor | Categ | Factor | Categ | Factor | Categ | Factor |
| 1 | IBC 16-8 Pre                | ٢        | DLPre | 1      |         |        |       |        |       |        |       |        |       |        |
| 2 | IBC 16-9 Pre                | ٤        | DLPre | 1      | LLConst | 1      |       |        |       |        |       |        |       |        |
| 3 | IBC 16-8 Post               | ব        | 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           | <b>N</b> | DL    | 1      | LL      | .75    | LLS   | .75    |       |        |       |        |       |        |

The following load combinations will be generated by RISAFloor:

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

| 📵 Coml | bination Design   |      |      |         |            |           |      |          |          | _       |            |
|--------|-------------------|------|------|---------|------------|-----------|------|----------|----------|---------|------------|
| Combir | nations Design    |      |      |         |            |           |      |          |          |         |            |
|        | Label             | ASIF | CD   | Service | Hot Rolled | Cold Form | Wood | Wood Pro | Concrete | Masonry | Connection |
| 1      | IBC 16-8 Pre      |      | .9   | V       | V          | V         | V    | V        | V        | V       | V          |
| 2      | IBC 16-9 Pre      |      |      | V       | V          | V         | V    |          |          | V       |            |
| 3      | IBC 16-8 Post     |      | .9   | V       | V          | V         | V    |          |          | V       |            |
| 4      | IBC 16-9 Post     |      |      | V       |            | V         | V    |          |          |         |            |
| 5      | IBC 16-10 (a) Pos |      | 1.25 | V       | V          | V         | V    | V        |          | V       |            |
| 6      | IBC 16-11 (a) Pos |      | 1.25 |         |            |           | V    |          |          |         |            |
| 7      | IBC 16-11 (b) Pos |      | 1.15 | V       | V          | V         | V    | V        | ~        | V       |            |

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
- In the LC Code list, select 2015 IBC Strength.

| Load Combination G | ienerator ×          |
|--------------------|----------------------|
|                    |                      |
| LC Regio           | n United States 💌    |
| LC Code            | 2015 IBC Strength 💌  |
| Roof Live Load Opt | tions                |
| Include :          | RLL (Roof Live Load) |
|                    | SL (Snow Load)       |
|                    | RL (Rain Load)       |
| Composite Options  |                      |
| Include :          | Pre Composite        |
|                    | Post Composite       |
|                    |                      |
| Save as Default    | ts                   |
| Generate           | Cancel Help          |
| Generate           | Cancer               |

Click Generate.

| 🖲 Com | bination Design    |      |      |         |            |           |      |          |          |          |            |
|-------|--------------------|------|------|---------|------------|-----------|------|----------|----------|----------|------------|
| Combi | nations Design     |      |      |         |            |           |      |          |          |          |            |
|       | Label              | ASIF | CD   | Service | Hot Rolled | Cold Form | Wood | Wood Pro | Concrete | Masonry  | Connection |
| 1     | IBC 16-8 Pre       |      | .9   | V       |            | V         |      |          |          | <b>V</b> |            |
| 2     | IBC 16-9 Pre       |      |      |         |            |           |      |          | •        |          |            |
| 3     | IBC 16-8 Post      |      | .9   | V       |            | V         | ~    | V        | V        | V        |            |
| 4     | IBC 16-9 Post      |      |      |         |            |           |      |          |          |          |            |
| 5     | IBC 16-10 (a) Post |      | 1.25 | V       |            | V         | •    | V        |          | V        | V          |
| 6     | IBC 16-11 (a) Post |      | 1.25 | V       |            |           |      |          |          |          |            |
| 7     | IBC 16-11 (b) Post |      | 1.15 | V       |            | V         | ~    | V        | 2        | V        |            |
| 8     | IBC 16-1 Pre       |      |      |         |            |           |      |          |          |          |            |
| 9     | IBC 16-2 Pre       |      |      |         |            | V         |      |          | •        | V        |            |
| 10    | IBC 16-1 Post      |      |      |         |            |           |      |          |          |          |            |
| 11    | IBC 16-2 (a) Post  |      |      |         |            | V         |      |          | 2        |          |            |
| 12    | IBC 16-2 (b) Post  |      |      |         |            |           |      |          |          |          |            |
| 13    | IBC 16-3 (a) Post  |      |      |         |            | V         |      |          | ~        | <b>V</b> |            |
| 14    | IBC 16-3 (c) Post  |      |      |         |            |           |      |          |          | V        |            |

This finalizes your load combinations as shown below:

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**.

|       |                    | es not match Final Se<br>ng Physical Columns | ection  | OK<br>RESOLVE |   |
|-------|--------------------|--|---------|---------------|---|
| S     | Column Label       | Analysis Shape                               | Final S | hape          | ^ |
| 1     | CS1 L1             | CRECT8X8                                     | CRECT   |               | 1 |
| 2     | CS2 L1             | CRECT8X8                                     | CRECT   | 10X10         |   |
| 3     | CS3 L1             | CRECT8X8                                     | CRECT   | 12X10         |   |
| 4     | CS4 L1             | CRECT8X8                                     | CRECT   | 10X8          |   |
| 5     | CS5 L1             | CRECT8X8                                     | CRECT   | 10X8          |   |
| 6     | CS6 L1             | CRECT8X8                                     | CRECT   | 10X10         |   |
| 7     | CS7 L1             | CRECT8X8                                     | CRECT   | 10X8          |   |
| 8     | CS8   1            | CRECTRYS                                     | CRECT   | 10X10         | ~ |
| Total | Physical Columns : | 28   |         | HELP          |   |

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.

| ſ | 📵 Warnir | ng Log |  |  |
|---|----------|--------|--|--|
|   |          | Help   | Message  |  |
|   | 1        |        | Slab punching shear checks do not consider slab openings at this time. |  |
|   |          |        |  |  |
|   |          |        |  |  |

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.

| Data Entry  | ×                          |
|---|----------------------------|
| Project C   | Grid                       |
| Results 🛛 Results   | s<br>tions                 |
| Code Checks<br>Shear Results  | ions<br>n Rules<br>Rules   |
| Bending Results<br>Deflection Results<br>Vibrations<br>End Reactions  | Rules<br>Rules<br>initions |
| Column Forces<br>Column Results<br>Member/Wall Rebar                  | ions<br>s<br>acks          |
| Slab Point Deflection<br>Slab Strip Deflection<br>Slab Results/Rebar  | els<br>;<br>ms             |
| Slab Forces<br>Punching Shear<br>Wall Forces                          | Strips<br>ds<br>ds         |
| Wall Results<br>Material TakeOff<br>Connection Results<br>Warning Log | Loads<br>Loads<br>lations  |

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**.

| ot Rolle | ed Cold Fo | rme | d Wood Cor | ncrete       |              |         |          |          |          |          |           |           |             |              |       |
|----------|------------|-----|------------|--------------|--------------|---------|----------|----------|----------|----------|-----------|-----------|-------------|--------------|-------|
|          | Stack      | L.  | Shape      | UC Max       | Elev[ft]     | UCLC    | 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   | Runching   |              |              |         |          |          |          |          |           |           |             |              | 23.41 |
| 6        | CS6        | 1   | Punching S | Shear Value  | es Punching  | Shear G | eometrie | s        |          |          |           |           |             |              | 27.83 |
| 7        | CS7        | 1   | 4.1        | Floor Plan * | 1 (Clob)     | _       |          |          |          |          |           |           |             |              | 23.68 |
| 3        | CS8        | 1   |            | FIOULFIAN    | r (Slab)     | •       |          |          |          |          |           |           |             |              | 29.78 |
| 9        | CS9        | 1   |            | Label        | Failure Pla. | . U     | c        | Gov LC   | Location | Vux[k]   | Muz[k-ft] | Muy[k-ft] | Total Stres | Phi*Vnx[ksi] | 27.6* |
| 0        | CS10       | 1   | 1          | CS1_L1       | Slab         | 1.3     | 07       | 4        | Interior | 48.899   | 19.16     | -42.965   | .215        | .164 🔺       | 27.83 |
| 1        | CS11       | 1   | 2          | CS2_L1       | Slab         | .94     | 42       | 4        | Interior | 49.756   | 6.347     | 7.946     | .155        | .164         | 27.83 |
| 2        | CS12       | 1   |            | CS3_L1       | Slab         | 1.2     | 52       | 4        | Interior | 56.992   | -3.8      | -42.502   | .206        | .164         | 25.38 |
| 3        | CS13       | 1   | 4          | CS4_L1       | Slab         | .70     | 06       | 4        | Interior | 30.487   | 5.499     | 8.92      | .116        | .164         | 27.83 |
| 4        | CS14       | 1   | 5          | CS5_L1       | Slab         | .58     | 58       | 4        | Interior | 32.26    | -2.113    | 3         | .092        | .164         | 27.83 |
| 5        | CS15       | 1   | 6          | CS6_L1       | Slab         | 1.0     | 01       | 4        | Interior | 51.503   | -5.412    | 11.339    | .164        | .164         | 27.83 |
| 6        | CS16       | 1   | 7          | CS7_L1       | Slab         | .5      | 6        | 4        | Interior | 30.562   | 3.159     | -1.262    | .092        | .164         | 27.38 |
| 7        | CS17       | 1   | 8          | CS8_L1       | Slab         | 1.0     | 34       | 4        | Interior | 52.137   | 5.072     | 19.813    | .17         | .164         | 71.30 |
| 8        | CS18       | 1   | 9          | CS9_L1       | Slab         | 1.0     | 25       | 4        | Interior | 49.375   | -4.092    | 16.971    | .168        | .164         | 33.50 |
| 9        | CS19       | 1   |            | CS10_L1      | Slab         | .71     | 15       | 4        | Interior | 40.304   | 164       | -7.72     | .117        | .164         | 33.07 |
| 0        | CS20       | 1   | 11 (       | CS11_L1      | Slab         | .91     | 16       | 4        | Interior | 47.976   | .566      | -13.767   | .15         | .164         | 69.90 |
| 1        | CS21       | 1   | 12 0       | CS12_L1      | Slab         | .79     | 98       | 4        | Interior | 34.967   | -8.547    | -11.819   | .131        | .164         | 70.3  |
| 2        | CS22       | 1   | 13 0       | CS13_L1      | Slab         | .75     | 53       | 4        | Interior | 46.643   | -1.151    | -2.244    | .124        | .164         | 33.50 |
| 3        | CS23       | 1   | 14 0       | CS14_L1      | Slab         | .73     | 35       | 4        | Interior | 45.512   | -1.246    | -2.071    | .121        | .164         | 24.82 |
| 4        | CS24       | 1   | 15 0       | CS15_L1      | Slab         | .75     | 54       | 4        | Interior | 44.746   | 1.582     | -4.088    | .124        | .164         | 26.92 |
| 5        | CS25       | 1   | 16 0       | CS16_L1      | Slab         | .9      | 7        | 4        | Interior | 54.827   | 8.741     | -1.753    | .159        | .164         | 27.83 |
| 6        | CS26       | 1   |            | CS17_L1      | Slab         | 1.3     | 95       | 4        | Interior | 62.492   | -1.35     | 51.556    | .229        | .164         | 25.10 |
| 7        | CS27       | 1   | 18 0       | CS18_L1      | Slab         | 1.4     | 49       | 4        | Interior | 80.169   | 8.32      | -26.801   | .238        | .164         | 23.94 |
|          |            |     | 19 0       | CS19_L1      | Slab         | 1.6     | 97       | 4        | Interior | 98.643   | -5.094    | -29.88    | .279        | .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.

| Clear Re | sults Warning  | $\times$ |
|----------|--|----------|
|          | This operation will cause the current results to be cleared!<br>Do you wish to continue? |          |
|          | Yes No   |          |
|          | Always display this warning before clearing results?                                     |          |

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

| Aodify Column Propertie  | 25 ×  |
|--|---|
| Draw Columns Modify  | Properties Modify Design  |
| Concrete Column Shape<br>Shape Group:<br>Rectangular<br>Explicit Shape:  | Use?  |
| Concrete Column Options<br>Material:   | Use?<br>Use?<br>Use?  |
| Shear Cap Options C Delete Rectangular S C Add/Modify Rectangular S Thickness 18 Shear Cap Eccentricity (in ey 0 | ular Shear Cap (in)<br>Depth 60 Width 60                          |
| Function Use?<br>C Lateral<br>Gravity  | Orientation Options Use?<br>Orient to Point:<br>Rotate Angle: deg |
| Splices Options<br>Splices<br>Type: Default(M 💌  | Distance Below: 0 ft<br>Parapet Height 0 ft                       |
| Apply Entries to All Sele  | ected Columns   |

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.

| Slabs Deflection<br>Beams/Columns/Walls   | Design Strips/C<br>Points/Shear Caps | uts User Defined  |
|---|--------------------------------------|---|
| Line Draw: <u>M</u>   | Rendered:                            | •]]•  |
| Color Basis:  | 🗴 Percen                             | t of Length 80% 💌   |
| Gravity/Lateral 💌   | 🗢 Pulbac                             | k Distance 1 it   |
| Line Thickness:   | Column                               | s/Walls Full Length   |
| ○1 ● 2 ○ 3  |                                      | ey: 📶 0%  |
| Labeling:<br>Design Result:<br>Show Pins<br>Pullback Ends<br>Col Eccentribity<br>Show Reactions | Labeling:<br>Label                   | Labeling:<br>No Labeling<br>Render 80%<br>Transparency<br>Texture 20% |
|   |                                      |   |

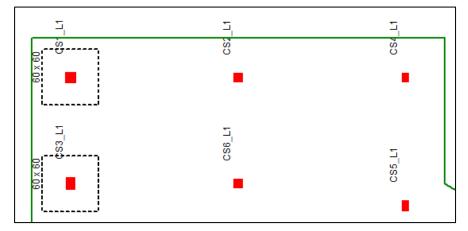
Click OK and you can see that columns CS1 and CS3 are in the upper left corner of the first floor.

Now add a Shear Cap to these columns.

Click on the two columns with your drawing tool cursor.

| CS1_L1 | CS2_L1 | CS4_L1 |
|--------|--------|--------|
| CS3_L1 | CS6_L1 | CS5_L1 |

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:

|        | Reinforcing     |              |          |          |        |          |            |            |        |          |            |            |          |            |
|--------|-----------------|--------------|----------|----------|--------|----------|------------|------------|--------|----------|------------|------------|----------|------------|
| esign) | Strip Results   | Design Cut R | tesults  |          |        |          |            |            |        |          |            |            |          |            |
|        | 1: Floor Plan 1 | 1 (Slab)     | •        |          |        |          |            |            |        |          |            |            |          |            |
|        | 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 |

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

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.

| ign Strip | Results  | Design Cut Res | sults                     |   |                 |            |        | 6          |              |            |           |             |    |
|-----------|--|----------------|---------------------------|---|-----------------|------------|--------|------------|--------------|------------|-----------|-------------|----|
| 1: Flo    | oor Plan 1 (                                   | Slab)          | •                         |   |                 |            |        |            |              |            |           |             |    |
|           | Column Forces/Moments, Dead & Other Categories |                |                           |   |                 |            |        |            |              |            |           | - • •       |    |
|           | Dead &   | Other Load F   | loor Live L               | oad Roof Lo   | ad              |            |        |            |              |            |           |             |    |
|           |  | Axial Force    |                           |   |                 |            |        |            | <b>'</b> 04' |            |           |             |    |
|           |  |                | 0                         |   | 0.1.101         |            |        |            |              |            |           |             | ~  |
|           |  | Column Stac    | Contraction of the second | a state of the second se | , Standard Cats |            |        |            |              |            |           |             | ~  |
| _         | 2  | CS1            | Standa                    | rd Alternate  |                 |            |        |            |              |            |           |             |    |
| -         | 3  | CS2            |                           | 1: Floor Plan   | 1 (Slab)        | •          |        |            |              |            |           |             |    |
|           | 4  |                |                           | Label   | Span            | Length[ft] | DL[in] | DL Ratio   | LL[in]       | LL Ratio   | DL+LL[in] | DL+LL Ratio | 1  |
|           | 5  | CS3            | 1                         | SL1   | 1               | 15.083     | .097   | 1866       | .083         | 2182       | .18       | 1006        | ٦. |
|           | 6  |                | 2                         | UL I  | 2               | 15.167     | .103   | 1768       | .086         | 2123       | .189      | 965         | -  |
|           | 7  | CS4            | 3                         |   | 3               | 15.167     | .308   | 591        | .234         | 778        | .542      | 336         |    |
|           | 8  |                | 4                         |   | 4               | 15.083     | .412   | 440        | .312         | 580        | .724      | 250         |    |
| _         | 9  | CS5            | 5                         |   | 5               | 14.169     | .338   | 502        | .258         | 660        | .596      | 285         |    |
| _         | 10   | 000            | 6                         |   | 6               | 14.914     | .171   | 1048       | .038         | 4673       | .209      | 856         |    |
| _         | 11   | CS6            | 7                         |   | 7               | 8.417      | .189   | 533        | .052         | 1953       | .236      | 429         |    |
|           | 12<br>13                                       | CS7            | 8                         | SL2   | 1               | 15         | .383   | 470        | .347         | 519        | .73       | 247         |    |
| -         | 13   | 037            | 9                         |   | 2               | 15         | .39    | 462        | .346         | 521        | .735      | 245         |    |
| -         | 15   | CS8            | 10                        |   | 3               | 15.083     | .33    | 548        | .275         | 659        | .605      | 299         |    |
|           | 16   | 000            | 11                        |   | 4               | 15.167     | .094   | 1945       | .07          | 2618       | .163      | 1116        |    |
|           | 17   | CS9            | 12                        |   | 5               | 14.57      | .101   | 1732       | .074         | 2370       | .175      | 1001        |    |
|           | 18   |                | 13                        | SL3   | 1               | 7.001      | .181   | 464        | .047         | 1799       | .222      | 379         | -  |
|           | 19   | CS10           | 14                        | 01.4  | 2               | 8.667      | .189   | 549        | .079         | 1325       | .236      | 441         |    |
|           | 20   |                | 15                        | SL4   | 1               | 15<br>15   | .383   | 470        | .347         | 519        | .73       | 247         | -  |
|           | 21   | CS11           | 16                        |   | 2               | 15         | .39    | 462<br>536 | .346         | 521<br>617 | .735      | 245<br>287  |    |
|           |  |                | 17                        |   | 4               | 15.167     | .375   | 1488       | .325         | 2026       | .212      | 858         | -  |
|           |  |                | 18<br>19                  |   | 4               | 15.167     | .122   | 983        | .126         | 1355       | .212      | 570         | -  |
|           |  |                | 20                        |   | 6               | 13.091     | .174   | 1177       | .096         | 1630       | .3        | 683         | -  |
|           |  |                | 20                        |   | 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.

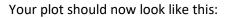
The **Set Options for Current View** (i.e. Model Display Options) dialog contains all the options for graphic display in your model. This includes input and results information. If you have not solved your model, the results options will be unavailable.

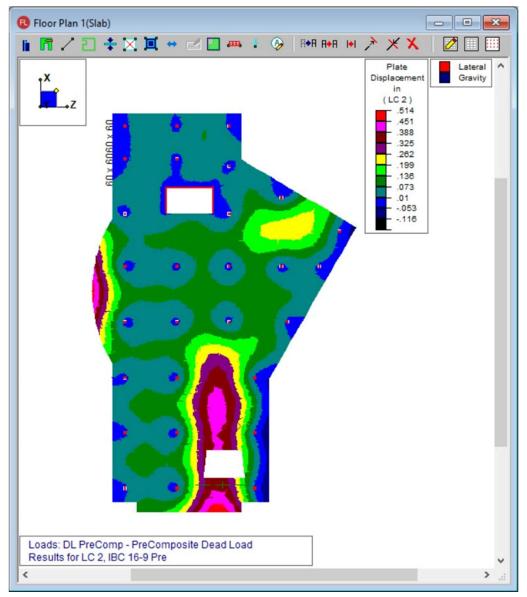
Turn on color coding contours of the displacements from Load Combination 2.

- 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.

| Show Slabs  | Color Fill<br>Color Bas                           | is<br>Tran        | lered 🛹 🔊  |   |
|-------------|---|-------------------|------------|---|
| Conto       | e Plates  | e Plates          | Show Openi |   |
| Value 🛛     | Displacement  Calculated Maximum User Defined Max |                   | 🗖 Directi  |   |
| Combination | To Use For Results Di                             | splay 2: IBC 16-9 | Pre        | • |

Click Apply.





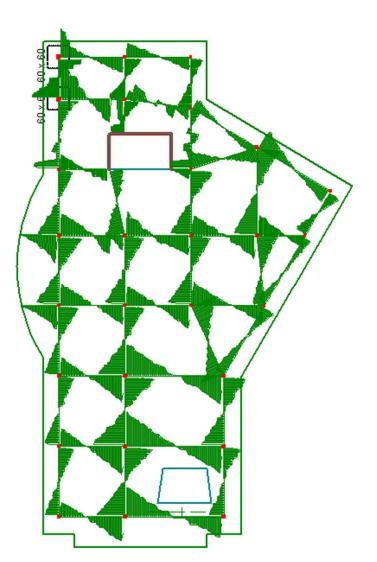
This image gives a quick graphical representation of the displacements per Load Combination 2. The color-coding legend is displayed in the upper right corner (you may need to move your **Results** or **Data Entry** toolbars to see this). You can create a similar plot to display the Bending or Shear forces.

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:



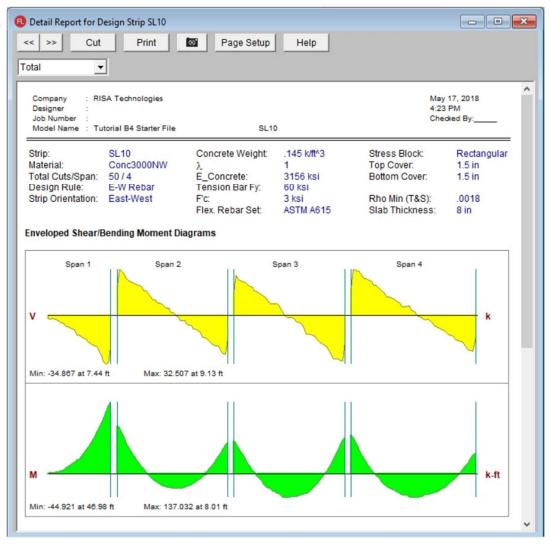
#### **Detail Reports**

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 Detail

| 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.

| Detail Report for Design Strip SL10                     |              |
|---|--------------|
| << >> Cut Print 🔯 Page Setup Help                       |              |
| Column Strip 🗸  |              |
| Total   | ^            |
| Column Strip<br>Mid Strip-Right                         | May 17, 2018 |
| and outpartight   | 4:25 PM      |
| Mid Strip-Left  | Checked By:  |
| Model Name : Tutorial B4 Starter File SL10 Column Strip |              |

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

| < >>  | Cut  | Print  | 100  | Page  | Setup   | Help  |  |  |                         |
|---|--|--|--|---|---|---|--|--|-------------------------|
| olumn Stri  | ip 👻   |  |  |   |   |   |  |  |                         |
|   |  |  |  |   |   |   |  |  |                         |
|   | 8.01 ft  |  |  | 23.84 ft  |   | 38.01   | ft.  |  | 54.79 ft                |
| _   | #5@12in<br>(101.1)   | (50.4) (27.4)  | (27.4)   |   | (27.4)  | #56 Sin<br>(27.4) (38.5)  | (42.9) (27.4)  | -  | #5@18in<br>(7.4) (30.2) |
|   | (101.1)  | (30.4) (27.4)  | ) (21.4)   | (37.8) (38)   | (21.4)  | (27.4) (38.5)   | (42.9) (21.4)  | (4   | (30.2)                  |
|   | #8@18in  |  | 8@18in   |   | #8@18   |   |  | #8@18in  |                         |
| _   | (101.1)  |  | (180)  |   | (180)   |   |  | (200.4)  |                         |
|   | 8.008 h  | 1  | 4.167 ft   | Û   | 14.167  | Û   | 1  | 15.946 ft  | Û                       |
|   | CS9_   | L1   |  | CS15_L1   |   | CS16  | _L1  |  | CS26_L1                 |
|   |  |  |  |   |   |   |  |  |                         |
| Rebar Det<br>Analysis R   | ailed Drawin<br>Results:   | ng: Units  | are showr  | n in (in) uni   | less otherwi  | se stateu.  |  |  |                         |
|   |  |  | are showr<br>hear V (k,  |   | less otherwi  | se stateu.  | Moment   | M (k-ft, ft)   |                         |
|   |  |  |  | ft)   | Loc   | Max   | Moment I   | <mark>M (k-ft, ft)</mark><br>Min                                     | Loc                     |
|   | esults:  | SI   | near V (k,   | ft)<br>in   | Loc   |   |  |  | Loc<br>NA               |
| analysis R  | esults:<br>Max   | SI<br>Loc  | hear V (k,<br>Mi   | ft)<br>in<br>627  | Loc<br>7.72   | Max   | Loc  | Min  |                         |
| Analysis R<br>Span 1  | esults:<br>Max<br>0  | St<br>Loc<br>NA  | hear V (k,<br>Mi<br>-21.<br>-25.   | ft)<br>in<br>627<br>515 2   | Loc<br>7.72<br>22.72  | <b>Max</b><br>61.787  | Loc<br>8.01  | Min<br>0   | NA                      |
| span 1<br>Span 2  | <b>Max</b><br>0<br>28.828  | SI<br>Loc<br>NA<br>9.13  | hear V (k,<br>Mi<br>-21.<br>-25.   | <b>ft)</b><br>in<br>627<br>515 2<br>994 3   | Loc<br>7.72<br>22.72<br>37.72   | Max<br>61.787<br>61.385   | Loc<br>8.01<br>8.84  | Min<br>0<br>-16.839  | NA<br>17.51             |
| Span 1<br>Span 2<br>Span 3<br>Span 4  | Max<br>0<br>28.828<br>26.023<br>33.563   | SI<br>Loc<br>NA<br>9.13<br>24.13<br>39.17  | near V (k,<br>Mi<br>-21.<br>-25.<br>-32.<br>-22.   | ft)<br>in<br>627<br>515 2<br>994 3<br>588 5   | Loc<br>7.72<br>22.72<br>37.72<br>54.46  | Max<br>61.787<br>61.385<br>55.722<br>60.326   | Loc<br>8.01<br>8.84<br>38.01                                     | Min<br>0<br>-16.839<br>-23.983                                       | NA<br>17.51<br>29.91    |
| Span 1<br>Span 2<br>Span 3<br>Span 4  | Max<br>0<br>28.828<br>26.023<br>33.563   | SI<br>Loc<br>NA<br>9.13<br>24.13<br>39.17  | near V (k,<br>Mi<br>-21.<br>-25.<br>-32.<br>-22.   | ft)<br>in<br>627<br>515 2<br>994 3<br>588 5   | Loc<br>7.72<br>22.72<br>37.72   | Max<br>61.787<br>61.385<br>55.722<br>60.326   | Loc<br>8.01<br>8.84<br>38.01                                     | Min<br>0<br>-16.839<br>-23.983                                       | NA<br>17.51<br>29.91    |
| Span 1<br>Span 2<br>Span 3<br>Span 4  | Max<br>0<br>28.828<br>26.023<br>33.563   | SI<br>Loc<br>NA<br>9.13<br>24.13<br>39.17  | near V (k,<br>Mi<br>-21.<br>-25.<br>-32.<br>-22.   | ft)<br>in<br>627<br>515 2<br>994 3<br>588 5   | Loc<br>7.72<br>22.72<br>37.72<br>54.46  | Max<br>61.787<br>61.385<br>55.722<br>60.326   | Loc<br>8.01<br>8.84<br>38.01                                     | Min<br>0<br>-16.839<br>-23.983                                       | NA<br>17.51<br>29.91    |
| Span 1<br>Span 2<br>Span 3<br>Span 4  | Max<br>0<br>28.828<br>26.023<br>33.563<br>4 Code Chec  | 51<br>Loc<br>NA<br>9.13<br>24.13<br>39.17<br>ck and Ben                                      | near V (k,<br>Mi<br>-21.<br>-25.<br>-32.<br>-22.   | ft)<br>in<br>627<br>515 2<br>994 3<br>588 5<br>I Req and F  | Loc<br>7.72<br>22.72<br>37.72<br>54.46  | Max<br>61.787<br>61.385<br>55.722<br>60.326   | Loc<br>8.01<br>8.84<br>38.01                                     | Min<br>0<br>-16.839<br>-23.983                                       | NA<br>17.51<br>29.91    |
| Span 1<br>Span 2<br>Span 3<br>Span 4<br>ACI 318-14                            | Max<br>0<br>28.828<br>26.023<br>33.563<br>Code Chec<br>Top   | 51<br>Loc<br>NA<br>9.13<br>24.13<br>39.17<br>Ck and Ben<br>Gov                               | near V (k,<br>Mi<br>-21.<br>-25.<br>-32.<br>-22.   | ft)<br>in<br>627<br>515 2<br>994 3<br>588 5<br>I Req and F<br>phi*Mn                                      | Loc<br>7.72<br>22.72<br>37.72<br>54.46  | Max<br>61.787<br>61.385<br>55.722<br>60.326<br>iits: in^2                                       | Loc<br>8.01<br>8.84<br>38.01<br>38.84                            | Min<br>0<br>-16.839<br>-23.983<br>-27.562                            | NA<br>17.51<br>29.91    |
| Span 1<br>Span 2<br>Span 3<br>Span 4<br>ACI 318-14<br>Gov                     | Max<br>0<br>28.828<br>26.023<br>33.563<br>Code Chec<br>Top<br>Bending  | 54<br>Loc<br>NA<br>9.13<br>24.13<br>39.17<br>Ck and Ben<br>Gov<br>Mu Top                     | hear V (k,<br>Mi<br>-21.<br>-25.<br>-32.<br>-22.<br>ding Steel   | ft)<br>in<br>627<br>515 2<br>994 3<br>588 5<br>I Req and F<br>phi*Mn<br>top<br>(K-ft)                     | Loc<br>7.72<br>22.72<br>37.72<br>54.46<br>Provided, Ur                          | Max<br>61.787<br>61.385<br>55.722<br>60.326<br>iits: in^2<br>Top<br>Rebar                       | Loc<br>8.01<br>8.84<br>38.01<br>38.84<br>Top As                  | Min<br>0<br>-16.839<br>-23.983<br>-27.562<br>Top As                  | NA<br>17.51<br>29.91    |
| Span 1<br>Span 2<br>Span 3<br>Span 4<br>ACI 318-14<br>Gov<br>Loc (ft)         | Max<br>0<br>28.828<br>26.023<br>33.563<br>Code Check   | St<br>Loc<br>NA<br>9.13<br>24.13<br>39.17<br>Ck and Ben<br>Gov<br>Mu Top<br>(k-ft)           | hear V (k,<br>Mi<br>-21.<br>-25.<br>-32.<br>-22.<br>ding Steel<br>Gov LC   | ft)<br>in<br>627<br>515 2<br>994 3<br>588 5<br>I Req and F<br>phi*Mn<br>top<br>(K-ft)                     | Loc<br>7.72<br>22.72<br>37.72<br>54.46<br>Provided, Ur<br>Gov Cut               | Max<br>61.787<br>61.385<br>55.722<br>60.326<br>iits: in^2<br>Top<br>Rebar<br>29 #5@12           | Loc<br>8.01<br>8.84<br>38.01<br>38.84<br>Top As<br>Prvd          | Min<br>0<br>-16.839<br>-23.983<br>-27.562<br>Top As<br>Reqd          | NA<br>17.51<br>29.91    |
| Span 1<br>Span 2<br>Span 3<br>Span 4<br>ACI 318-14<br>Gov<br>Loc (ft)<br>8.01 | Max         0           28.828         26.023           33.563         33.563           Code Check         Top           Bending         Check           .954         .954 | St<br>Loc<br>NA<br>9.13<br>24.13<br>39.17<br>Ck and Ben<br>Gov<br>Mu Top<br>(k-ft)<br>61.787 | hear V (k,           Mi           -21.           -25.           -32.           -22.           ding Steel           Gov LC           11 | ft)<br>in<br>627<br>515 2<br>994 3<br>588 5<br>I Req and F<br>phi*Mn<br>top<br>(k-ft)<br>64.795<br>49.261 | Loc<br>7.72<br>22.72<br>37.72<br>54.46<br>Provided, Ur<br>Gov Cut<br>SL10-S1-X2 | Max<br>61.787<br>61.385<br>55.722<br>60.326<br>iits: in^2<br>Top<br>Rebar<br>9 #5@12<br>1 #5@18 | Loc<br>8.01<br>8.84<br>38.01<br>38.84<br>Top As<br>Prvd<br>2.454 | Min<br>0<br>-16.839<br>-23.983<br>-27.562<br>Top As<br>Reqd<br>2.334 | NA<br>17.51<br>29.91    |

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**.

| 8                     | Detail Re  | eport                       | for Des | ign Cut SL10-S  | 51-X2   | Colur                   | nn  |   |              |                               |                        |
|-----------------------|--|-----------------------------|---------|---|---------|-------------------------|---|---|--------------|-------------------------------|------------------------|
| <                     | < >>   |                             | Strip   | Print   |         |                         | Page Setup  | Help  |              |                               |                        |
|                       | lune O   |                             |         | On an 4   |         | × 0                     |   |   |              |                               |                        |
|                       | olumn S  | uip                         | •       | Span 1  | -       | X-2                     | <b>•</b>  |   |              |                               |                        |
|                       | Compan<br>Designer<br>Job Num<br>Model N                     | nber                        | :       | Technologies<br>rial B4 Starter Fi  | le      |                         | SL10-S1-X2 Co   | umn Strip   |              | 4:38 F                        | 7, 2018<br>M<br>ed By: |
|                       | ut   |                             | \$1 10  | -S1-X2 Colum  | m       | Max                     | Top bar Spac.:  | 18 in   |              | Stress Block:                 | Rectangular            |
|                       | laterial:  |                             |         | 3000NW  |         |                         | Top bar Spac.:  | 3 in  |              | Suess Diock.                  | Nectangular            |
|                       | tart:  |                             |         | 63.86) ft   |         |                         | Bot bar Spac.:  | 18 in   |              | Rebar Spacing Inc:            | 1 in                   |
| E                     | ind:   |                             |         | 56.36) ft   |         |                         | Bot bar Spac.:  | 3 in  |              | Design Rule:                  | E-W Rebar              |
| T<br>G<br>P<br>T<br>F | op Bend<br>ov Mu T<br>hi*Mn To<br>ension I<br>'c<br>lex. Reb | ding C<br>op<br>op<br>Bar F | heck    | Check<br>0.021<br>1.369 k-ft<br>64.795 k-ft<br>60 ksi<br>3 ksi<br>ASTM A615 |         | Gov<br>phi*<br>Con<br>λ | Bending Check<br>Mu Bot<br>Mn Bot<br>crete Weight<br>concrete | 0.000<br>0 k-ft<br>114.172 k-ft<br>.145 k/ft^3<br>1<br>3156 ksi |              | Top Cover<br>Bottom Cover     | 1.5 in<br>1.5 in       |
| Т                     | op As R<br>Analys  |                             |         | 0.049 in^2  |         | A                       | As Reqd:<br>nalysis   | NA  |              |                               |                        |
|                       | 4/3<br>Min Fle   | ex                          |         | NA<br>1.296 in^2  |         |                         | /3<br>lin Flex  | NA  |              | As Reqd(T/S)<br>Rho Reqd(T/S) | 1.29600<br>0.00180     |
| Ŧ                     | op As P  |                             |         | 2.454 in^2  |         |                         | As Prvd   | 4.712 in^2  |              | Rho Prvd(Gross)               | 0.00995                |
|                       | lax Defle  | ection                      |         | .691 in   |         |                         |   |   |              |                               |                        |
|                       | ov Cate  |                             |         | DL+LL   |         |                         |   |   |              |                               |                        |
| с                     | ross Se  | ection                      | Detai   | ling(All Bars Ε   | iqually | y Spa                   | #5@12<br>#8@18  | •••   | H H<br>2 1.8 |                               |                        |

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 | s at the top left to scroll quickly |  |
|--|-------------------------------------|--|
| through the <b>Design Cut Detail Reports</b> .         |                                     |  |

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

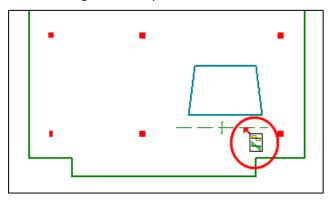
First, click **Close** I to close the **Detail Report** and return to the model view.

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**.

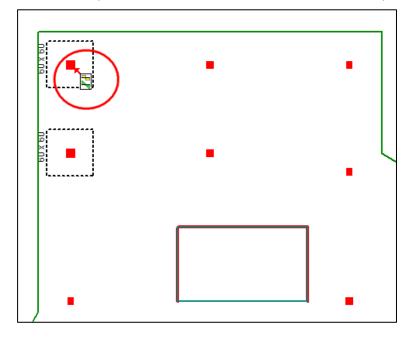
| 📵 Detail Report fo  | r Design Cut X1                                      |   |   |                                     |                                       | 23 |
|---|--|---|---|-------------------------------------|---------------------------------------|----|
| << >> Pr  | int 🔯 Pa   | age Setup Help  |   |                                     |                                       |    |
| Designer :<br>Job Number :                                    | RISA Technologies<br>Tutorial B4 Starter File        | X1  |   |                                     | Dec 2, 2015<br>9:40 AM<br>Checked By: |    |
| Material: C<br>Start: (<br>End: (                             | K1<br>Conc3000NW<br>29 , 10) ft<br>44 , 10) ft       |   |   | Stress Block:<br>Design Rule:       | Rectangular<br>N-S Rebar              | ш  |
| ACI 318-14 Co<br>Gov Mu Top<br>Top d<br>Tension Bar Fy<br>F'c | ode Check<br>27.3 k-ft<br>7.25 in<br>60 ksi<br>3 ksi | Gov Mu Bot<br>Bot d<br>Concrete Weight<br>λ<br>E_Concrete | 0 k-ft<br>7.25 in<br>.145 k/ft^3<br>1<br>3156 ksi | Gov Vu<br>Top Cover<br>Bottom Cover | 3.639 k<br>.75 in<br>.75 in           |    |

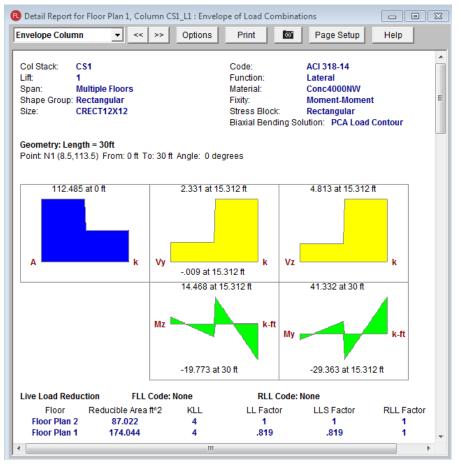
When finished reviewing these results, close the Detail Report.

Click Close I 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).



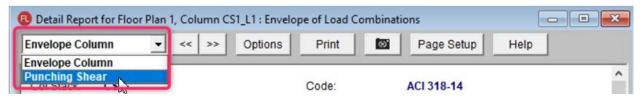


This will open the Envelope Column Detail Report.

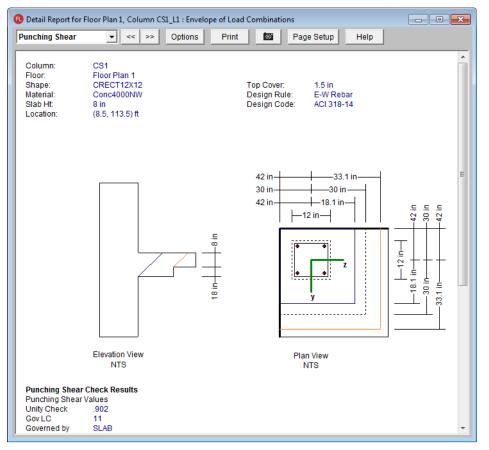
 Scroll down through the report to review the enveloped force diagrams, Code Check, Interaction Diagrams, and reinforcement design details.

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.

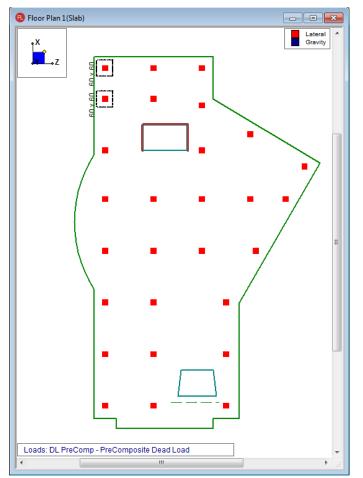
# Part B: Tutorial 5 – RISA-3D Integration

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 <sup>C</sup>. Double-click the Tutorials folder, select Tutorial B5 Starter.rfl and click Open. Click Close <sup>C</sup> (or Cancel) to exit the Model Settings dialog box.
- On the Window toolbar, click the Graphic Editing toolbar button 2 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.

|               |     | Director |            |   |  |  |  |  |
|---------------|-----|----------|------------|---|--|--|--|--|
|               |     | RISA     | -3D        |   |  |  |  |  |
| 🛛 🕶 🛞 👫 📼 🕶 🚺 | 10. | RISA     | Connection | 5 |  |  |  |  |

#### 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:

| Wind Loads               |                |                |           |                       |               |                   | ×          |
|--------------------------|----------------|----------------|-----------|-----------------------|---------------|-------------------|------------|
| Wind Load Parameters -   |                |                |           |                       |               |                   |            |
| Wind Code ASCE           | 7-10 🔻         | Exposure Cat   | t B 👻     | Topographic Fac. K1   | 0             | Topographic Fa    | c. K3 0    |
| Wind Speed (mph) 115     |                |                |           | Topographic Fac. K2   |               | Directionality Fa | . Kd 85    |
| Base Elevation           | ft             | Gust Stiffness | s Riaid 👻 | Fundamental Freq      |               | Gust Facto        |            |
| Dase Elevation           | n              | Gust Sunnes:   | s Rigia 💌 | Fundamentarried       | >1 HZ         | Gustradio         | (0) [.85   |
| Genera                   | te Roof Wind L | oads           |           |                       |               |                   |            |
|                          |                |                |           |                       |               |                   |            |
| Wind Load Results        |                |                |           |                       |               | (                 | Calc Loads |
| wind Load Results        |                |                |           |                       |               |                   |            |
| Wind Generation Input    |                |                |           |                       |               |                   |            |
| Wind Code:               | ASCE 7-10      |                |           | Topographic Factor    |               |                   |            |
| Wind Speed, V(mph):      | 115            |                |           | Topographic Factor    |               |                   |            |
|                          | -              |                |           | Topographic Factor    |               |                   |            |
| Exposure Category:       | В              |                |           | Directionality Factor |               |                   |            |
| Base Elevation(ft):      | 0              |                |           | Parapet Height(ft):   | 0             |                   |            |
| Wind Generation Detail R | Results        |                |           |                       |               |                   |            |
|                          |                |                |           | Kzt:                  | 1             |                   |            |
| Exposure Constant Alpha  |                |                |           | h (ft):               | 30            |                   |            |
| Exposure Constant zg:    | 1200           |                |           | Kh:                   | .701          |                   |            |
| Gust Effect Factor, G:   | .85            |                |           | Windward Cp:          | .8            |                   |            |
|                          |                |                |           | qh (ksf):             | .02           |                   |            |
|                          |                |                |           | GCpn (windward):      | +1.5          |                   |            |
|                          |                |                |           | GCpn (leeward):       | -1.0          |                   |            |
| Wind Generation Floor G  | eometry Resul  | ts             |           |                       |               |                   |            |
| Floor Level H            | leight         | Kz             | Width (X) | Length (Z)            | Leeward Cp(X) | Leeward Cp        | (Z)        |
|                          | (ft)           |                | (ft)      | (ft)                  |               |                   |            |
| Floor Plan 2             | 30             | .701           | 115       | 75.994                | .397          | .5                |            |
| Floor Plan 1             | 15             | .575           | 115       | 75.994                | .397          | .5                | ~          |
|                          |                |                |           | ОК                    | Print         | Cancel            | Help       |
|                          |                |                |           |                       |               |                   |            |

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.

| eismic Loads       |                   |                  |                 |          |               |                   |            |
|--------------------|-------------------|------------------|-----------------|----------|---------------|-------------------|------------|
| Seismic Load Para  | meters            |                  |                 |          |               |                   |            |
| Seismic Code A     | SCE 7-10 🔻        | Ct (V,X) .02     |                 | T (V,X)  | sec           | R (V,X) 3         |            |
| Base Elevation     | ft                | Ct (H,Z) .02     |                 | T (H,Z)  | sec           | R (H,Z) 3         |            |
| Risk Cat           | or II 👻           | TL 5             | sec             | Add      | Base Weight   | Ct Exp. (V,X) .75 | 5          |
| S_D1               | g                 | S_DS 1           | g               | S_1 1    | g             | Ct Exp. (H,Z) .75 | 5          |
| Seismic Load Res   | sults             |                  |                 |          |               | C                 | Calc Loads |
|                    |                   |                  |                 |          |               |                   |            |
| Seismic Generation |                   |                  |                 |          |               |                   |            |
|                    | ASCE 7-10         |                  |                 |          |               |                   |            |
|                    | .02               | T_X (sec):       | Not Entered     | R_X:     | 3             |                   |            |
| -                  | .02               | T_Z (sec):       | Not Entered     | R_Z:     | 3             |                   |            |
|                    | .75               | Ct Exp. Z:       | .75             |          |               |                   |            |
|                    | lorll             | TL (sec):        | 5               | 04 (-)   |               |                   |            |
|                    | 1                 | SDS (g):         | 1               | S1 (g):  | 1             |                   |            |
| Base Elev (ft):    | 0                 | Parapet Ht (ft): | 0               |          |               |                   |            |
| Seismic Generation | on Detail Results |                  |                 |          |               |                   |            |
| T_X Used (sec):    | .256              | T_X Method A:    | .256            | T_X Uppe | r Limit: .359 |                   |            |
| T_Z Used (sec):    |                   | T_Z Method A:    | .256            | T_Z Uppe | r Limit: .359 |                   |            |
| Importance Fac.:   |                   | Design Cat.:     | E               |          |               |                   |            |
| -                  | 634.572           | Gov. Eqn.        | ASCE Eqn 12.8-2 | _        | 0.333         |                   |            |
| V_Z (k):           | 634.572           | Gov. Eqn.        | ASCE Eqn 12.8-2 | Cs_Z:    | 0.333         |                   |            |
| Seismic Generation | on Force Results  |                  |                 |          |               |                   |            |
| Floor Level        | Height            | Weight           | Force X         | Force Z  | CGX           | CG Z              |            |
|                    | (冊)               | (k)              | (k)             | (k)      | (ff)          | (冊)               |            |
|                    |                   |                  |                 | ок       | Print         | Cancel            | Help       |

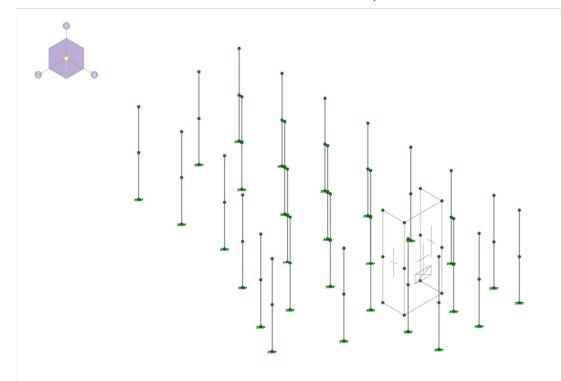
After the **Wind Loads** dialog box closes, the **Seismic Loads** dialog box will display:

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.



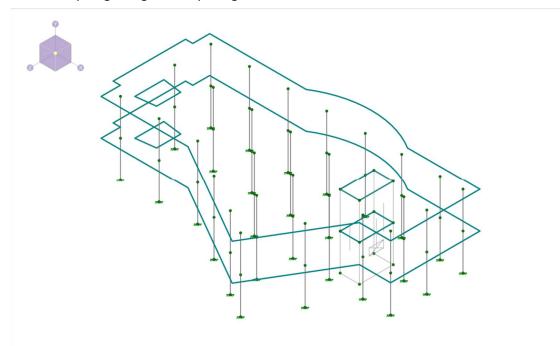
Notice that only the columns and wall panels that were specified as LATERAL members were generated in your RISA-3D model. You can always add vertical braces, modify your wall panels, or manipulate your model as you wish. For this tutorial, you will leave the model as it is.

### **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.

| Floor Di | aphragms       |          |                               |                     |          |       |          |       |          |           |            |        |             |      |          |                |               | × |
|----------|----------------|----------|-------------------------------|---------------------|----------|-------|----------|-------|----------|-----------|------------|--------|-------------|------|----------|----------------|---------------|---|
| General  | Wood Diaphra   | agms     |                               |                     |          |       |          |       |          |           |            |        |             |      |          |                |               |   |
|          | Elevation [ft] | Mass [k] | Mass MOI [k·ft <sup>2</sup> ] | Center Of Mass [ft] | Plus X E | Minus | Plus Z E | Minus | Inactive | Diaphragm | Туре       | 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 K 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 Loa | ad 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                  | 1 |
| 3         | Live Load Special (public assemb | LLS      |           |           |           |       |       |             |              | 6                   | 1 |
| 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    |           |           |           |       |       |             |              |                     | 1 |
| 28        | Semi-Rigid Wind Load +X          | WL+X     |           |           |           |       |       | 40          |              |                     | 1 |
| 29        | Semi-Rigid Wind Load -X          | WL-X     |           |           |           |       |       | 40          |              |                     | 1 |
| 30        | Semi-Rigid Wind Load +Z          | WL+Z     |           |           |           |       |       | 44          |              |                     | 1 |
| 31        | Semi-Rigid Wind Load -Z          | WL-Z     |           |           |           |       |       | 44          |              |                     | 1 |

When you are finished reviewing the data, click Close is 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.

| Gravity    | Wind Sei       | smic    |                       |        |
|------------|----------------|---------|-----------------------|--------|
| Load Co    | mbination Reg  | gion    | United States         | ~      |
| Load Co    | mbination Co   | de      | 2015 IBC ASD          | ~      |
|            | Generate Bea   | am Def  | lection Load Combinat | ions   |
| Notiona    | I Load Options | 5       | None                  | ~      |
|            | Reversible     |         |                       |        |
| Include    | Roof Live Load | l Optio | ns                    |        |
| <b>—</b> - | of Live Load   |         | Snow Load 🗌 Rai       | n Load |
| Ro         |                |         |                       |        |
| Ro         |                |         |                       |        |
| Ro         |                |         |                       |        |
|            |                |         |                       |        |

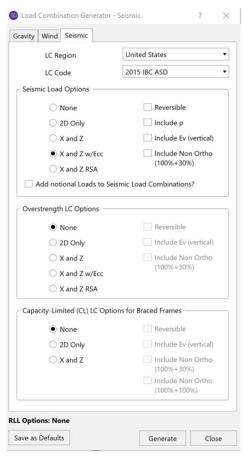


- On the first (Gravity) tab, in the LC Region list, select United States.
- On the first (Gravity) tab, in the LC Code list, select 2015 IBC ASD.
- Toggle off the Generate Deflection LCs box and uncheck the Roof Live Load Options.
- Click Generate.

- 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.

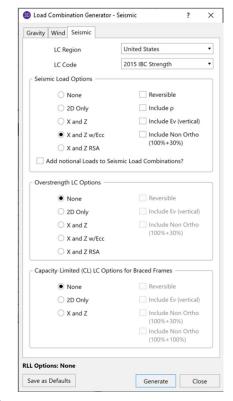
• Click back to the **Gravity** tab and change the **LC Code** list to **2015 IBC Strength**.

| Gravity | Wind     | Seismic         |                   |   |
|---------|----------|-----------------|-------------------|---|
| ununy   |          |                 | United States     |   |
|         | LC Re    | -               |                   | • |
|         | LC Co    | ode             | 2015 IBC Strength | • |
| Bean    | n Deflec | tion Options -  |                   |   |
|         | G        | ienerate Deflec | ction LC's        |   |
| Roof    | Live Lo  | ad Options —    |                   |   |
| In      | clude :  |                 |                   |   |
|         | R        | LL (Roof Live L | .oad)             |   |
|         | S        | L (Snow Load)   |                   |   |
|         | R        | L (Rain Load)   |                   |   |
| Notic   | onal Loa | d Options —     |                   |   |
|         | • N      | lone            | Reversible        |   |
|         | 0 2      | D Only          |                   |   |
|         | Ox       | and Z           |                   |   |
|         |          |                 |                   |   |
|         |          |                 |                   |   |
|         |          |                 |                   |   |
|         |          |                 |                   |   |
|         |          |                 |                   |   |
|         |          |                 |                   |   |
|         |          |                 |                   |   |
|         |          |                 |                   |   |
|         |          |                 |                   |   |
|         |          |                 |                   |   |

- 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.

| ombin | ations Design | r.           |              |      |     |        |       |        |            |        |            |            |     |              |          |
|-------|---------------|--------------|--------------|------|-----|--------|-------|--------|------------|--------|------------|------------|-----|--------------|----------|
| LC G  | enerator      | RSA Sca      | aling Factor |      |     |        |       | Solve  | Current LC | : 5    | olve Batch | + Envelope | ·   | Solve Envelo | ope Only |
|       | 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     | $\checkmark$ | 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     | $\checkmark$ | 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    |            |        |            |            |     |              |          |

This will generate 70 load combinations of which 17 are shown below.

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 14<sup>th</sup> 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.

| LC Generator |             | RSA Scaling Factor |         |      |     |        |       | Solve Current LC |     | : 5    | Solve Batch + Envelope |        |     | Solve Envelope Onl |     |
|--------------|-------------|--------------------|---------|------|-----|--------|-------|------------------|-----|--------|------------------------|--------|-----|--------------------|-----|
|              | 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              |     |        |                        |        |     |                    |     |

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.

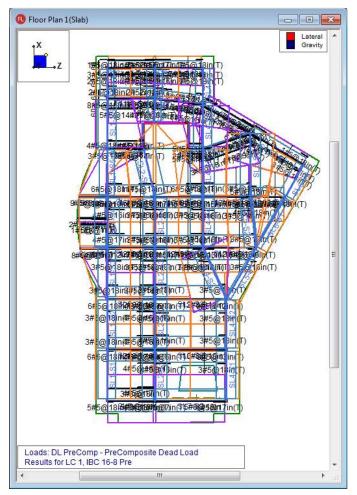
Follow the steps below to load the RISAFloor ES starter file for this portion of the tutorial:

- 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.

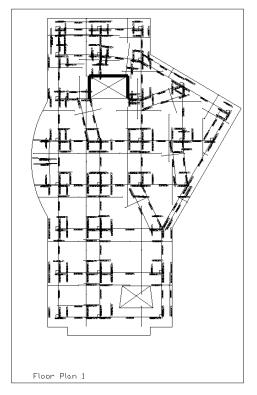


### **DXF Export**

- 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.

| Export DXF File                        | ×                        |
|--|--------------------------|
| DXF Options                            | Grid Bubble Type         |
| DXF Floors All                         | Circle                   |
| DXF Scale Factor 1                     | O Hexagon                |
| DXF Units in                           | DXF Export Layer Options |
| Beam Options                           |                          |
| Depth Only Beam Size ?                 | Text Placement Locations |
| Parentheses Options                    | Top Bottom               |
| Camber C= -                            | Size Callout 💿 🛛         |
|  | Camber 💿                 |
| Reaction bare -                        | Reactions                |
| Studs ()                               | Uniform Studs 💿 📀        |
| Camber Units Symbol                    | Beam Standoff 12 in      |
| <ul> <li>* " (Double Quote)</li> </ul> | Text End Standoff 3 in   |
| ⊙ 'in.'                                | Text Perp Standoff 3 in  |
| OK Cancel                              | Save Defaults Help       |

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 <sup>D</sup> 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:

| Import DXF File  |                        |                  | ×          |  |
|------------------|------------------------|------------------|------------|--|
| DXF Options      |                        |                  |            |  |
| DXF Scale Factor | 1                      | Floor Elevation  | D ft       |  |
| DXF Units        | in 💌                   | Beam Standoff    | 12 in      |  |
| Layer Options    |                        |                  |            |  |
| Beam Layer       | COLUMN_BLO             | DCKS 🚽           | Import ?   |  |
| Column Layer     | COLUMN_BLO             | OCKS -           | Import ?   |  |
| Wall Layer       | WALL_LINES             | •                | ✓ Import ? |  |
| Grid Layer       |                        | DCKS 🗾           | Import?    |  |
| Floor Type       |                        |                  |            |  |
| Concrete Floor   | Slab                   |                  |            |  |
| Slab Layer       | DIAPHRAGM_             | LINES            | Import ?   |  |
| Angle Inc        | 10                     | Deg, For Arc Sla | b Edges    |  |
| C Beam Support   | C Beam Supported Floor |                  |            |  |
| 🗹 Ignore F       | raming Suppor          | i for Beam Layer |            |  |
| ОК               | Cancel                 | Save Defaults    | Help       |  |

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

| DXF_FLOOR 1(Slab)                          | - • •                |
|--|----------------------|
| tx<br>↓ z                                  | Lateral A<br>Gravity |
|  |                      |
|  | $\overline{}$        |
|  |                      |
|  |                      |
| · · · _ ·                                  |                      |
|  |                      |
| Loads: DL PreComp - PreComposite Dead Load | •<br>ai              |

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**

| 1 Co Co 🙆 🤐                             | <b>〒</b> 🖪 🖽 🗱 | = 📗 🖹 🛃 🗊 🗷 🧇 |
|---|----------------|---------------|
| - · · · · · · · · · · · · · · · · · · · |                |               |

| Button    | Title  | Label in RISAFoundation                | Shortcut   |
|-----------|--|--|------------|
| Ľ         | New Model 🗋<br>Start a new model                           | Start a new model                      | (CTRL+N)   |
| <b>F</b>  | <b>Open Model</b><br>Open an existing model                | Reload from disk a previous model      | (CTRL + O) |
|           | <b>Save 🗔</b><br>Save current model to disk                | Save the current model data to disk    | (CTRL+S)   |
| Ē         | Copy 🗈<br>Copy to the clipboard                            | Copy to the clipboard                  | (Ctrl+C)   |
| ð         | <b>Print </b><br>Print a report or graphic image           | Print a report or graphic image        | (Ctrl+P)   |
| 5         | Undo 🗠<br>Undo the last operation                          | Undo the last operation                | (Ctrl+Z)   |
| 2         | <b>Redo P</b><br>Reverse the most recent undo<br>operation | Reverse the most recent undo operation | (Ctrl+Y)   |
| •         | (Global) Model Settings 🔍<br>Set Model Settings            | Set Model Settings                     |            |
| in<br>min | Define Units to be used                                    | Define units to be used                |            |
| I         | Shape Database I   | Edit or view the shape database        |            |
|           | <b>Rebar Layout Create and edit a custom rebar</b> layout  | Create and edit a custom rebar layout  |            |
| 飄         | Project Grid  Upen the Project Grid spreadsheet            | Opens the Project Grid spreadsheet     |            |
| ■         | Create New Floor Plan 🛡                                    | Create a new floor plan                |            |

| 8        | Delete Floor Plans 键   | Delete existing floor<br>plans                            |             |
|----------|--|---|-------------|
|          | New Model View 园<br>Create a new model view                              | Create a new model<br>view                                | (CTRL + F2) |
|          | <b>Open Spreadsheets H</b><br>Select spreadsheets to open                | Select spreadsheets to open                               |             |
|          | <b>Refresh                                    </b>                       | Refresh all open<br>windows with the most<br>current data |             |
| LC       | Load Combinations LC<br>Open the load combinations<br>spreadsheet        | Open the Load<br>Combinations<br>Spreadsheet              |             |
| =        | Solve =<br>Perform the analysis and<br>design calculations               | Perform the analysis and design calculations              | (F7)        |
|          | Browse Results 🖹<br>Select results to browse                             | Select results to browse                                  |             |
| R        | Erase Results 🛃<br>Erase all solution results                            | Erase all solution results                                |             |
| D        | Data Entry toolbar D<br>Turn the spreadsheet<br>shortcuts menu on or off | Turn the Spreadsheet<br>Shortcuts menu off or<br>on       |             |
| R        | <b>Results toolbar R</b><br>Turn the results shortcuts<br>menu on or off | Turn the Results<br>Shortcuts menu off or<br>on           |             |
| <i>.</i> | Help 🧇<br>View help topics   | View Help menu topics                                     | (CTRL + F1) |

## Window Toolbar

... in Model View

📼 1: Floor Plan 1(Sl 🚽 🔯 | Iso Pln 🧞 🤴 🧐 🔍 🔍 🕵 🔀 💋 🚅 DL PreComp-Pre 🚽 😓 💋 🛠 🦮 🙇 👻 🖉 🖉 🐨 🔛 🗸 🖓 🙌 📼 🕶 💷 👻 🔤

| Button      | Title   | Label in RISAFoundation   | Shortcut  |
|-------------|---|---|-----------|
|             | Model Display Options 🗐<br>Display plot options             | Bring up the Model Display<br>Options dialog                                      | F2        |
|             | Floor Menu<br>2: Floor Plan 2                               | Select which floor you want to view   |           |
| 0           | Snapshot 🖾  | Save an image of the current<br>model view for later use in<br>the printed report |           |
| lso         | Isometric Iso<br>Display an isometric view                  | Snap to an Isometric View   |           |
| PIn         | <b>Plan <sup>Pln</sup></b><br>Display an XZ planar view     | Snap to an XZ Planar View   |           |
| €           | Rotate buttons  | Rotate the view counter-<br>clockwise about the X axis                            |           |
| 9           |   | Rotate the view counter-<br>clockwise about the Y axis                            |           |
| Ð           |   | Rotate the view counter-<br>clockwise about the Z axis                            |           |
|             | 3 buttons below are collectively called <b>Zoom</b> buttons |   |           |
| Ð           | Zoom In 🔍   | Zoom IN (closer view) on the model  | PLUS (+)  |
| Θ           | Zoom Out 🔍  | Zoom OUT (view farther away) on the model   | MINUS (-) |
| <b>9</b>    | Box Zoom 强  | Draw a box around the part of the model to be zoomed                              |           |
| $\boxtimes$ | <b>Redraw 🗵</b><br>Redraw full model view                   | Redraw full model view  |           |

|            | <b>Graphic Editing</b> Activate the Graphic Editing (AKA Drawing) toolbar  | Activate the Graphic Editing<br>(Drawing) toolbar                 | Ctrl+G |
|------------|--|---|--------|
| <b>_</b> ŧ | 3 buttons below collectively<br>refer to Loads Display<br>JE-Dead Load   S<br>Display Loads<br>Toggle display of the loads (LC | Toggle display of the loads<br>(LC or CAT)                        |        |
|            | or CAT)<br>Loads List – lists the available<br>loads<br>DL-Dead Load   |   |        |
| 3          | Switch Loads 蒆<br>Switch loads display<br>(combinations or categories)   | Switch loads display<br>between combinations and<br>categories    |        |
| F          | <b>Rendering F</b><br>Toggle between wireframe and<br>rendering of members   | Toggle between wireframe and rendering of members                 |        |
| "N         | Joint Labels N<br>Joint labels toggle  | Toggle the joint labels   |        |
| M          | Beam Labels 🚨<br>Beam labels toggle  | Toggle the beam labels  |        |
| 8          | Member End Options   | Member end display options  |        |
| W          | Show Wall Properties W<br>Wall Panel toggle  | Toggles on the display of<br>Wall Panel properties                |        |
|            | Show Deck Assignment 🗐<br>Show Deck Assignment   | Toggles on the display of the deck for beam supported floors only |        |
|            | Show Area Load Types 🞑<br>Show Area Load   | Toggles on the display of the area loads                          |        |
| <b>()</b>  | Project Grid Display 🖗<br>Show Project Grid toggle   | Toggles on the display of the<br>Project Grid                     |        |

| <sub>€</sub> <sup>d</sup> → | Distance Tool  | Measures the distance between two points                                   | F5 |
|-----------------------------|--|--|----|
|                             | Design Strip Rebar 📟<br>Design Strip rebar toggle              | Toggles on the display of the<br>Design Strip rebar                        |    |
|                             | Design Strip   | Toggles on the display of the<br>Design Strips and/or the<br>Support Lines |    |
|                             | Design Strip Diagram 돝<br>Design Strip force diagram<br>toggle | Toggles on the display of the force diagram on the Design Strip            |    |

#### ... in Spreadsheet View



| Button     | Title  | Label in RISAFoundation                    | Shortcut |
|------------|--|--|----------|
| ₩          | New Line III<br>Insert a new line before<br>the current line | Insert a new line before the current line  | F3       |
| <b>t</b> ង | Delete Line <sup>11</sup><br>Delete the current line         | Delete the current line                    | F4       |
| <b>#</b>   | Repeat Line 🌐<br>Repeat the current line                     | Repeat the current line                    | F8       |
| 12.3       | Fill Fill Fill be currently marked block                     | Fill the currently marked block            | Ctrl+F   |
| ₩<br>₩     | Math 🖼<br>Perform math on the<br>currently marked block      | Perform math on the currently marked block | Ctrl+M   |
| ₩<br>₩     | Mark Lines 開<br>Mark all the lines                           | Mark all the lines                         |          |

|            | <b>Delete Lines</b><br>Delete the currently<br>marked lines   | Delete the currently marked lines    | Ctrl+D   |
|------------|---|--------------------------------------|----------|
| ₩<br>¥     | Unmark Lines 開<br>Unmark all the lines                        | UnMark all the lines                 | Ctrl+L   |
| Ê          | Paste 🛍<br>Paste from the clipboard                           | Paste from the clipboard             | Ctrl+V   |
|            | Save as Defaults 🖼<br>Save the current data as<br>the default | Save the current data as the default |          |
| <u>,</u> , | <b>Help</b><br>Help on the current<br>window                  | Help for the current window          | SHIFT+F1 |

| Button | Title  | Label in RISAFoundation            | Shortcut |
|--------|--|------------------------------------|----------|
| li.    | Draw or Modify Columns 📗                                 | Draw or modify columns             |          |
|        | Draw or modify columns                                   |                                    |          |
| 17     | Draw or Modify Walls п                                   | Draw or modify walls               |          |
|        | Draw or modify walls                                     |                                    |          |
| 1      | Draw or Modify Beams 🦨                                   | Draw or modify beams               |          |
|        | Draw or modify beams                                     |                                    |          |
|        | Generate Infill Beams 🧾                                  | Generate infill beams              |          |
|        | Generate infill beams                                    | within bays                        |          |
| 1      | Elevate Points 🚄   | Change the elevation of            |          |
|        | Change the elevation of selected points                  | selected points in the model       |          |
| Ð      | Draw Slabs/Diaphragms 뙵                                  | Draw or modify slabs or diaphragms |          |
|        | Draw slabs or diaphragms                                 | ulaphilagins                       |          |
|        | Assign Deck  | Assign deck properties             |          |
|        | Assign deck properties                                   |                                    |          |
|        | Add Wood Diaphragm Regions 🖾                             | Add wood diaphragm                 |          |
|        | Add wood diaphragm regions                               | regions                            |          |
|        | Draw Area Loads 🔲  | Draw area loads                    |          |
|        | Draw area loads  |                                    |          |
| щ      | <b>Draw Line Loads</b><br>Draw line loads point to point | Draw line loads point to point     |          |
| +      |  | Draw point loads                   |          |
|        | Assign Point Loads 🔹<br>Draw point loads                 |                                    |          |
|        |  |                                    |          |

👔 🗖 🖍 🛅 🚣 🛅 📓 📓 🛲 🚦 🤣 🛤 🖬 🕨 🥕 🗶 📜 🧮

## **Drawing Toolbar**

| <b>%</b>            | Project Grid 🔗  | Draw Project Grid Lines   |
|---------------------|---|---|
|                     | Draw Project Grid Lines   |   |
| <b>∐</b> ♦ <b>用</b> | Move  | Move selected parts of the model                                |
| R◆R                 | Сору Н+Я  | Copy selected parts of the model                                |
| I+I                 | Offset Copy   | Copy selected parts of the<br>model using an offset<br>distance |
| $\times$            | Extend ×  | Extend the member   |
|                     | Extend the member   |   |
| 1                   | Trim  | Trim the member   |
|                     | Trim the member   |   |
| X                   | <b>Delete</b> X<br>Delete parts of the model                        | Delete parts of the model                                       |
|                     | Modify Drawing Grid 🕅<br>Modify the drawing grid and snap<br>points | Modify the drawing grid<br>and snap points                      |
|                     | <b>Drawing Grid</b><br>Toggle the drawing grid on or off            | Toggle the drawing grid on or off                               |
|                     | Universal Snap Points III<br>Toggle the universal snap points       | Toggle the universal snap points on or off                      |

# Selection Toolbar

| Button | Title   | Label in RISAFoundation  | Shortcut |
|--------|---|--|----------|
| H      | Select All 用<br>Select the entire model   | Make the entire model selected   | CTRL+A   |
|        | Box Select <a>Draw a box around the part to be selected</a>                                     | Draw a box around the<br>part of the model to be<br>selected                   |          |
|        | Polygon Select<br>Draw a polygon around the<br>part to be selected (double<br>click to end)     | Draw a polygon around<br>the part to be selected<br>(double click to end)      |          |
| P      | Line Select r <sup>d</sup><br>Draw a line through the beams<br>and plates to be selected        | Draw a line through the<br>beams and plates to be<br>selected                  |          |
| Ξ      | Unselect All 🔲<br>Unselect the entire model   | Make the entire model<br>UNselected  | Ctrl+U   |
|        | Box Unselect <a>Draw a box around the part to be unselected</a>                                 | Draw a box around the<br>part of the model to be<br>UNselected                 |          |
| O      | Polygon Unselect<br>Draw a polygon around the<br>part to be unselected (double<br>click to end) | Draw a polygon around<br>the part to be<br>UNselected (double click<br>to end) |          |
| /      | Line Unselect<br>Draw a line through the beams<br>and plates to be unselected                   | Draw a line through the beams and plates to be UNselected                      |          |
| 2      | Invert Selected<br>Invert the selected areas of<br>the model                                    | Invert the selected state of the model   | Ctrl+I   |
| ?      | Criteria Selection  Page 2 Select or unselect   | Select or Unselect based on other criteria                                     |          |
| 2      | Save/Recall Selection<br>Save or recall selections of the<br>model                              | Save or recall selection states for the model                                  |          |
| 3      | Lock Unselected 🔒   | Lock the unselected part of the model  | Ctrl+L   |