# Table of Contents

## Introduction .................................................................  
  How to Use this Book ................................................. 1  
  Using the Online Help ............................................... 3  
  Technical Support Information ................................. 4  
  RISA Online .......................................................... 4  

## Before You Begin ..........................................................  
  RISAFloor Overview .................................................. 5  
  Hardware Requirements ........................................... 5  
  Program Limits .......................................................... 6  
  License Agreement ..................................................... 7  
  Installation ............................................................... 7  

## First Look at RISAFloor ..................................................  
  Starting RISAFloor .................................................... 10  
  Windows and Dialog Boxes ....................................... 10  
  Menus and Toolbars .................................................. 12  
  Managing Windows, Model Views, and Spreadsheets... 16  
  Working in Spreadsheets .......................................... 17  

## Part A: Beam Supported Floors .................................  
  Part A: Tutorial 1 – Columns and Walls ..................... 19  
  Starting a New File .................................................... 19  
  Project Grid .............................................................. 21  
  Model Settings .......................................................... 24  
  Columns ................................................................. 26  
  Wall Panels ............................................................... 34  
  Part A: Tutorial 2 – Modeling Beams ......................... 41  
  Primary Framing ........................................................ 42  
  Infill Framing ............................................................ 47  
  Additional Beam Drawing Tips (optional) ..................... 51
# Table of Contents

**Part A: Tutorial 3 – Decks & Diaphragms**
- Creating Diaphragm Edges ............................................. 56
- Deck Properties ............................................................. 59

**Part A: Tutorial 4 – Loading**
- Area Loads ................................................................. 67
- Line Loads ................................................................. 73
- Point Loads ................................................................. 78

**Part A: Tutorial 5 – Solution & Results**
- Load Combinations ...................................................... 81
- Solving and Reviewing Results ................................... 84
- Modifying the Model ................................................... 89

**Part A: Tutorial 6 – RISA-3D Integration**
- RISA-3D Lateral Load Generators ............................ 93
- RISA-3D Load Combinations .................................. 100

**Part A: Tutorial 7 – RISA Interoperability**
- RISA-Revit Link ......................................................... 104
- CIS/2 Translator ........................................................... 107
- RISAFloor & CAD ....................................................... 108
- Conclusion ................................................................. 111

**Part B: Elevated Slab Design** ........................................ 112

**Part B: Tutorial 1 – Columns & Walls**
- Starting a New File ..................................................... 114
- Drawing Grid ............................................................. 116
- Columns ................................................................. 119
- Wall Panels ............................................................... 127
- Wall Panel Openings ................................................ 132

**Part B: Tutorial 2 – Slab & Design Strips**
- Draw the Slab ............................................................ 137
- Modifying Your Slab .................................................. 141
- Creating a Slab Opening .......................................... 148
- Support Lines and Design Strips .............................. 152
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Part B: Tutorial 3 – Loading</td>
<td>165</td>
</tr>
<tr>
<td>Area Loads</td>
<td>166</td>
</tr>
<tr>
<td>Line Loads</td>
<td>173</td>
</tr>
<tr>
<td>Point Loads</td>
<td>175</td>
</tr>
<tr>
<td>Part B: Tutorial 4 – Solution &amp; Results</td>
<td>179</td>
</tr>
<tr>
<td>Load Combinations</td>
<td>180</td>
</tr>
<tr>
<td>Solving the Model</td>
<td>182</td>
</tr>
<tr>
<td>Spreadsheet Results</td>
<td>187</td>
</tr>
<tr>
<td>Viewing Graphical Results</td>
<td>189</td>
</tr>
<tr>
<td>Detail Reports</td>
<td>192</td>
</tr>
<tr>
<td>Part B: Tutorial 5 – RISA-3D Integration</td>
<td>199</td>
</tr>
<tr>
<td>Lateral Load Generators</td>
<td>200</td>
</tr>
<tr>
<td>RISA-3D Diaphragms</td>
<td>203</td>
</tr>
<tr>
<td>RISA-3D Loading</td>
<td>204</td>
</tr>
<tr>
<td>RISA-3D Load Combination Generator</td>
<td>205</td>
</tr>
<tr>
<td>Part B: Tutorial 6 – DXF Export</td>
<td>211</td>
</tr>
<tr>
<td>DXF Export</td>
<td>212</td>
</tr>
<tr>
<td>DXF Import</td>
<td>213</td>
</tr>
<tr>
<td>Appendix A – RISAFloor Toolbar Button Quick Reference</td>
<td>215</td>
</tr>
<tr>
<td>RISA Toolbar</td>
<td>215</td>
</tr>
<tr>
<td>Window Toolbar</td>
<td>217</td>
</tr>
<tr>
<td>Drawing Toolbar</td>
<td>221</td>
</tr>
<tr>
<td>Selection Toolbar</td>
<td>223</td>
</tr>
</tbody>
</table>
Introduction

How to Use this Book

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

Begin by reviewing First Look at RISAFloor on page 10 to familiarize yourself with the RISAFloor menus, toolbars, and shortcuts. Appendix A – RISAFloor Toolbar Button Quick Reference has also been included on page 215 to help you reference toolbar buttons.

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

Part A – Beam Supported Floors will guide you step-by-step through the RISAFloor modeling process to build and analyze a model using beam supported floors; Part B – Elevated Slab Design will guide you through building and analyzing a model with elevated slab floors in RISAFloor ES. In each part, you will create a real-world example of building and solving a model, making changes, and optimizing the model. Tips and shortcuts will also be demonstrated along the way.

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

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

The tutorials build upon themselves from start to finish. You have the option of performing them all at one time, or performing each one separately. To make this possible, RISA provides model files for you to load at the beginning of each tutorial. These starter files are located in the RISA folder under Tutorials, and are named Tutorial A2 Starter.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 http://www.risa.com. Click Downloads, Product Documentation, then RISAFloor.
**Document Conventions**

The following conventions are used throughout this book:

<table>
<thead>
<tr>
<th>This convention:</th>
<th>Indicates:</th>
</tr>
</thead>
<tbody>
<tr>
<td>CAPITAL LETTERS</td>
<td>Names of keys on the keyboard – for example, SHIFT, CTRL, or ALT.</td>
</tr>
<tr>
<td>KEY+KEY</td>
<td>One key should be held down and then another key pressed – for example, CTRL+P or ALT+F4.</td>
</tr>
<tr>
<td><strong>Bold</strong> text</td>
<td>User interface options – for example, <strong>File</strong> menu.</td>
</tr>
<tr>
<td><strong>Boxed</strong> text</td>
<td>Notes or modeling tip information.</td>
</tr>
<tr>
<td>Bulleted text</td>
<td>Action item for building the tutorial model.</td>
</tr>
<tr>
<td><img src="lightning_bolt.png" alt="Lightning bolt" /></td>
<td>Tutorial action item for building the model.</td>
</tr>
</tbody>
</table>
Using the Online Help

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

Help on general topics

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

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

Help on a specific feature (context-sensitive help)

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

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

Help on toolbar buttons

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

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

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

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

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

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

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

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

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

**Phone:** (949) 951-5815 or (800) 332-RISA (7472)
Technical support personnel are available from 6:00 A.M. to 5:00 P.M. Pacific Standard Time, Monday through Friday.

**RISA Online**

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

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

Subscription License

One of the following operating systems is required:

• Microsoft Windows 10 (64 bit only)
• Microsoft Windows 8.1 (64 bit only)
• Microsoft Windows 7 SP1 (64 bit only)

Perpetual License (Standalone / Network)

One of the following operating systems is required:

• Microsoft Windows 10 (32 bit or 64 bit)
• Microsoft Windows 8.1 (32 bit or 64 bit)
• Microsoft Windows 7 SP1 (32 bit or 64 bit)

Note: RISAFloor ES is only available for 64-bit operating systems
**Before You Begin**

**Hardware**
The following hardware is required:

- 1 GHz or faster processor
- 1024x768 or higher monitor resolution
- 2 (or more) button mouse, mouse wheel recommended
- 1 GB of RAM for 32 bit computers 8 GB for 64 bit computers
- 4 GB of hard disk space

**Hardware Limitations**
- The 32 bit version of RISAFloor cannot address more than 2 GB of memory on a 32-bit operating system, or 4 GB of memory on a 64-bit operating system. This limitation affects how large of a model can be solved.
- 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**

<table>
<thead>
<tr>
<th></th>
<th>32 Bit</th>
<th>64 Bit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Floors</td>
<td>200</td>
<td>200</td>
</tr>
<tr>
<td>Walls (per Entire Model)</td>
<td>5,000</td>
<td>10,000</td>
</tr>
<tr>
<td>Openings/Regions (per Wall Panel)</td>
<td>25/100</td>
<td>25/100</td>
</tr>
<tr>
<td>Columns per Floor</td>
<td>1,000</td>
<td>2,500</td>
</tr>
<tr>
<td>Beams per Floor</td>
<td>3,000</td>
<td>10,000</td>
</tr>
<tr>
<td>Points per Floor</td>
<td>5,000</td>
<td>25,000</td>
</tr>
<tr>
<td>Point Loads per Floor</td>
<td>2,000</td>
<td>10,000</td>
</tr>
<tr>
<td>Line Loads per Floor</td>
<td>2,000</td>
<td>10,000</td>
</tr>
<tr>
<td>Area Load Polygons per Floor</td>
<td>200</td>
<td>1,000</td>
</tr>
<tr>
<td>Deck Polygons per Floor</td>
<td>200</td>
<td>1,000</td>
</tr>
<tr>
<td>Materials</td>
<td>500</td>
<td>500</td>
</tr>
<tr>
<td>Custom Wood Species</td>
<td>500</td>
<td>500</td>
</tr>
<tr>
<td>Load Combinations</td>
<td>5,000</td>
<td>5,000</td>
</tr>
<tr>
<td>Concrete Slabs per Floor (RISAFloor ES Only)</td>
<td>NA</td>
<td>20</td>
</tr>
<tr>
<td>Support Lines per Slab (RISAFloor ES Only)</td>
<td>NA</td>
<td>300</td>
</tr>
</tbody>
</table>
**Demonstration Version:** You can open and solve a model of any size, however you may not:

- Create more than one Concrete Floor Slab per Floor
- 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 one concrete floor slab
- Save a model with more than 20 columns or walls
- Save a model with more than 10 support lines

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

**License Agreement**

**END-USER LICENSE AGREEMENT FOR RISA TECH, INC. SOFTWARE**

The RISAFloor software product (SOFTWARE PRODUCT) includes computer software, the associated media, any printed materials, and any electronic documentation. By installing, copying, accessing or otherwise using the SOFTWARE PRODUCT, you agree to be bound by the terms of this agreement. If you do not agree with the terms of this agreement RISA Tech, Inc. is unwilling to license the SOFTWARE PRODUCT to you. In such event, you must delete any installations and destroy any copies of the SOFTWARE PRODUCT and return the SOFTWARE PRODUCT to RISA Tech, Inc. within 60 days of purchase for a full refund.

Copyright © 2018 by RISA Tech, Inc. All rights reserved. RISA is part of the Nemetschek Group. The SOFTWARE PRODUCT is protected by United States copyright laws and various international treaties. All rights not specifically granted under this agreement are reserved by RISA Tech, Inc. “RISA” as applied to structural engineering software is a registered trademark of RISA Tech, Inc.

**1. SOFTWARE LICENSE.** The SOFTWARE PRODUCT is licensed, not sold. All right, title and interest is and remains vested in RISA Tech, Inc. You may not rent, lease, or lend the SOFTWARE PRODUCT. You are specifically granted a license to the use of this program on no more than one CPU at any given time. The Network Version of the SOFTWARE PRODUCT is licensed for simultaneous use on a certain maximum number of network stations that varies on a per license basis. The Subscription Version of the SOFTWARE PRODUCT is licensed for simultaneous use on a certain maximum number of CPU’s that varies on a per license basis. As part of the license to use the SOFTWARE PRODUCT, the program user acknowledges the reading, understanding and acceptance of all terms of this agreement. The SOFTWARE PRODUCT may not be reviewed, compared or evaluated in any manner in any publication without expressed written consent of RISA Tech, Inc. You may not disassemble, decompile, reverse engineer or modify in any way the SOFTWARE PRODUCT. If the SOFTWARE PRODUCT was purchased at a discounted price for educational purposes it may in no event be used for professional design purposes. The terms of this license agreement are binding in perpetuity.

**2. DISCLAIMER.** We intend that the information contained in the SOFTWARE PRODUCT be accurate and reliable, but it is entirely the responsibility of the program user to verify the accuracy and applicability of any results obtained from the SOFTWARE PRODUCT. The SOFTWARE PRODUCT is intended for use by professional engineers and architects who possess an understanding of structural mechanics. In no event will RISA Tech, Inc. or its officers be liable to anyone for any damages, including
any lost profits, lost savings or lost data. In no event will RISA Tech, Inc. or its officers be liable for incidental, special, punitive or consequential damages or professional malpractice arising out of or in connection with the usage of the SOFTWARE PRODUCT, even if RISA Tech, Inc. or its officers have been advised of or should be aware of the possibility of such damages. RISA Tech, Inc.’s entire liability shall be limited to the purchase price or subscription price of the SOFTWARE PRODUCT.

3. LIMITED WARRANTY. RISA Tech, Inc. warrants that the SOFTWARE PRODUCT will operate but does not warrant that the SOFTWARE PRODUCT will operate error free or without interruption. RISA Tech, Inc.’s sole obligation and your exclusive remedy under this warranty will be to receive software support from RISA Tech, Inc. via telephone, email or fax. RISA Tech, Inc. shall only be obligated to provide support for the most recent version of the SOFTWARE PRODUCT. If your version of the SOFTWARE PRODUCT is not the most recent version RISA Tech, Inc. shall have no obligation to provide support in any form. Except as stated above the SOFTWARE PRODUCT is provided without warranty, express or implied, including without limitation the implied warranties of merchantability and fitness for a particular purpose.

4. PROTECTION DEVICE. In the event the SOFTWARE PRODUCT requires the use of a PROTECTION DEVICE to operate, you are specifically prohibited from attempting to bypass the functionality of the PROTECTION DEVICE by any means. A lost or stolen PROTECTION DEVICE will not be replaced by RISA Tech, Inc.

5. SUBSCRIPTION SERVICE. In the event the SOFTWARE PRODUCT is accessed via a SUBSCRIPTION SERVICE, RISA Tech, Inc. shall make the SUBSCRIPTION SERVICE available for your use via an internet connection for the Term of the Subscription Agreement. At the end of the Term of the Subscription Agreement your right to access the SOFTWARE PRODUCT via the SUBSCRIPTION SERVICE will terminate unless the Subscription Agreement is renewed. RISA Tech, Inc. does not guarantee the SUBSCRIPTION SERVICE will be available error-free or uninterrupted. You acknowledge RISA Tech, Inc. does not control the transfer of information over the internet and that the SUBSCRIPTION SERVICE may be subject to interruptions or delays beyond the control of RISA Tech, Inc. RISA Tech, Inc. is not responsible for any delivery failures or any other damages resulting from any problems with the SUBSCRIPTION SERVICE.

6. TERMINATION. RISA Tech, Inc. may terminate your right to use the SOFTWARE PRODUCT if you fail to comply with the terms and conditions of this agreement. In such event, you must delete any installations and destroy any copies of the SOFTWARE PRODUCT and promptly return the SOFTWARE PRODUCT to RISA Tech, Inc.

7. CHOICE OF LAW. By entering into this Agreement in accordance with Paragraph 1, above, you have agreed to the exclusive jurisdiction of the State and Federal courts of Orange County, California, USA for resolution of any dispute you have relating to the SOFTWARE PRODUCT or related goods and services provided by RISA Tech, Inc. All disputes therefore shall be resolved in accordance with the laws of the State of California, USA and all parties to this Agreement expressly agree to exclusive jurisdiction of, and venue in, the courts in Orange County, California, USA. No choice of law rules of any jurisdiction apply.

8. PRIVACY POLICY. All information you provide and have provided, or which we otherwise collect and have collected, through or in connection with the SOFTWARE PRODUCT, including personally identifiable information about you, is subject to, and will be treated in accordance with, the then-current Privacy Policy located at http://risa.com/privacy, which is incorporated herein by this reference. By accessing, downloading, installing, using and providing information to or through the SOFTWARE PRODUCT, you consent to all actions taken by us with respect to such information in compliance with the Privacy Policy.
Installation

*Installation Instructions*

To install RISAFloor, please follow these instructions:

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

*RISAFloor Customization–Important Assumption!*

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

Starting RISAFloor

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

Start RISAFloor as follows:

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

Windows and Dialog Boxes
Title bar

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

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

Minimize, Maximize, Close

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

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

Workspace

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

Status bar

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

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

To the right of the “S” are 3 status boxes:

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

Dialog boxes

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

### Main Menu

<table>
<thead>
<tr>
<th>Menu</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>File</td>
<td>Provides access to file operations such as opening, saving, and exporting files.</td>
</tr>
<tr>
<td>Edit</td>
<td>Provides editing tools that help you modify and manipulate the spreadsheets. You may use this menu to add or remove information from the spreadsheets or to sort and mathematically manipulate current spreadsheet data.</td>
</tr>
<tr>
<td>Settings</td>
<td>Allows you to open model settings.</td>
</tr>
<tr>
<td>View</td>
<td>Allows you to open a new model view or adjust the current model view.</td>
</tr>
<tr>
<td>Insert</td>
<td>Used to insert drawing columns, beams, walls, and loads into the model. All of these items may be drawn graphically or entered in the spreadsheets. This menu provides access to the graphical methods that RISAFloor provides, while the Spreadsheets menu gives you access to the spreadsheets.</td>
</tr>
<tr>
<td>Modify</td>
<td>Allows access to the graphic editing features and may be used to modify existing model elements.</td>
</tr>
<tr>
<td>Spreadsheets</td>
<td>Opens the spreadsheets.</td>
</tr>
<tr>
<td>Solve</td>
<td>Solves the model.</td>
</tr>
<tr>
<td>Results</td>
<td>Allows access to all analysis result spreadsheets. This button is dimmed when no results are available, such as before you run a solution.</td>
</tr>
<tr>
<td>Tools</td>
<td>Provides tools to help you organize, identify, and correct problems as you model the structure. Program Application Settings are also located here.</td>
</tr>
<tr>
<td>Window</td>
<td>Manages all of the windows that you have open in RISAFloor, whether they are spreadsheets or model views.</td>
</tr>
<tr>
<td>Help</td>
<td>Provides access to the RISAFloor online Help menu. For more information on Help, see Using the Online Help on page 3.</td>
</tr>
<tr>
<td>Director</td>
<td>This feature allows you to integrate your model with RISA-3D and/or RISAConnection if you also own those products.</td>
</tr>
</tbody>
</table>
**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

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

...in Spreadsheet View

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

**Drawing Toolbar**

...on Beam Supported Floors

...on Concrete Floor Slabs

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

**Selection Toolbar**

...only visible in Model View

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

You will need to make selections when you do things like graphically edit a part of the model or print only part of the results.
Spreadsheet Toolbars
(Data Entry and Results toolbars)

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

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

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

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

Note: The toolbars above include several slab-specific spreadsheets that are only available in RISAFloor ES. For more information about these and how to use them, please see Part B: Elevated Slab Design on page 112.
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 comprised of rows and columns of data cells. To add or edit data in a cell, click the cell, making it the active cell, then type. Only one cell can be active at a time, and it is denoted in green. You can change which cell is active using the LEFT ARROW, RIGHT ARROW, PAGE UP, PAGE DOWN, HOME keys, etc.

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

Note: The view shown above is from the Demonstration version of RISAFloor. If you are running the full version of the program, this dialog will look slightly different.
To create a new model, begin by creating a floor plan, or click Close and work on your own.

- Click Create a New Floor Plan to begin.

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

The Create New Floor Plan dialog box also allows you to set the Elevation, the Default Area Load, and the Default Deck for the floor. These settings are discussed in more detail on page 60.

- In the Elevation box, enter 15 ft.

- Click Ok.

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

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

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

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

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

For Z Direction/Radial Line Labeling:

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

For X Direction/Radial Line Labeling:

- In the **Start Label** box, type **J**.
  - For the **Label Order**, click **Z to A**.
The **Project Grid Lines** dialog box should look similar to below:

Click **Apply**. Then click **OK** to close the dialog box.
You can verify the values in the **Project Grid Lines** spreadsheet and also modify the grid lines if needed.

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

![Project Grid Lines Spreadsheet](image)

**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 look like this:

![Workspace View](image)
Model Settings

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

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

Click on the Solution tab.

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

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.

25
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**.

![Column Shape Options](image)

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

**Note:** When you enter the drawing mode, your cursor changes to .
You can now start defining the columns by clicking or drawing, as shown below:

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

- Repeat for the grid intersections between G6 and J7.

To further define the gravity columns:

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

![Drawing Grids dialog box](image)

- **Click Ok** 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, I5, and J5.

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

- When you finish adding these columns, use the right mouse button (right-click) or ESC to terminate the column drawing tool.
Notice that by default all the columns are oriented with their webs parallel to the vertical axis. To change the orientation of the columns toward the center column:

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

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

Note: The cursor will change to .
Now, you can select the columns as follows:

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

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

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

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

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

- On the Drawing toolbar, click the **Drawing Grid** button to turn off the **Drawing Grid**.
Next, rotate the horizontal columns 90 degrees, so that they are oriented properly:

- 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 Properties](image)

- Click **Apply**.

**Note:** The cursor will once again change to 🔄.
Click on or draw a box around the columns circled in the next image. When you are done, your columns should have rotated 90 degrees, and oriented as shown below.
Wall Panels

Prepare to draw your wall panels:

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

![Wall Panel Settings](image)

- 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 (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
Part A: Tutorial 1 – Columns and Walls

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.

![Floor Plan 1](image)

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

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

![Grid Settings](image)

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 Status Bar 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 Status Bar.

Note: If there is an existing region then this 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 lined and the lintel label.

Right-click or click ESC to cancel the opening drawing tool.
Wall Panel Design Regions

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

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

Turn the rendered view back on for a better view.

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

Now you can see the 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 http://www.risa.com/d_documentation.html.

- Click OK to close the Wall Panel Editor.

Your model should look similar to the following image:
This is the end of Tutorial A1.

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

- On the File menu, click Save As and enter a file name.
**Part A: Tutorial 2 – Modeling Beams**

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

If you are continuing from the previous tutorial:
- From the **Window** menu, select **Single View**.
- On the Window toolbar, click the **Graphic Editing** button to activate the Drawing toolbar.
- On the **Data Entry** toolbar, click **Close** to close it. Skip ahead to the next page.

-OR- If you are starting here from scratch, follow the steps below to load the starter file provided by RISA:
- Double-click on the **RISAFloor** icon to start the program.
- Click **Open Model**. Double-click the **Tutorials** folder, select **Tutorial A2 Starter.rfl** and click **Open**. Click **Close** (or **Cancel**) to exit the **Model Settings** dialog box.
- On the Window toolbar, click the **Graphic Editing** button to activate the Drawing toolbar.

Your screen should now look like this:
Primary Framing

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

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

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

**Note:** The cursor will change to , to signify that you are now in drawing mode.
Draw a box around the entire model, as shown in the figure below:

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

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

On the Drawing toolbar, click Modify Drawing Grid.

Click the Snap To Options tab.

Under Universal Snap Increments, make sure the Z Axis and X Axis Increments are both set to 1 ft. Check the Use Universal Increments check box (this will allow you to draw freehand without being limited to a project or drawing grid).

Click Ok to return to the model view.

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.
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 Lca | 54, 119, 0 (ft) |

Continue drawing the beams until your model looks like this:

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

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

Click **Apply Entries by Clicking Beams Individually**, and then click **Apply**.
Now, click on the four beams that are shown circled in the image below (you can click anywhere on the beams).

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

Your model should now look like this:

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

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

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

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

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

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

- On the Drawing toolbar, click **Generate Beams** again.
- Under **Beam Orientation**, click **Horizontal**, and then click **Apply**.
- Draw a box around the area between grid lines **F5** and **J10** to select the entire area.
When finished, your model should look similar to the one shown below:

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

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

Continue with other bays until the radial portion of your model is similar to that shown below:
Terminate the drawing tool using the right mouse button (right-click) or press ESC.
Additional Beam Drawing Tips (optional)

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

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

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

- Click at the start of the first beam (C2) and drag to the end of the last beam (C5). Right-click to release the mouse.

Your model should now look like the image below. The single span beams will be automatically broken at the supports for you.
Since these members should not be a part of your completed model, click the **Undo** button to undo the creation of the beams.

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

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

- On the Drawing toolbar, click the **Draw Beams** button to again open the **Draw Beams** dialog box.
- Under **Drawing Options**, click **Draw Beam to Beam**. In both the **1st Offset** and **2nd Offset** boxes, enter **5 ft**, and then click **Apply**.
Experiment with this tool by clicking to create members parallel to the diagonal beam between D4 and C5, as shown in the image below.

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

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

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

On the Drawing toolbar, click the Modify Drawing Grid button and select the Snap To Options tab.

Under Beam/Wall Snap Locations, make sure that the Quarter Points and Third Points check boxes are checked.

Under Universal Snap Increments, make sure that the Use Universal Increments check box is cleared.

Click Ok.

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

On the Drawing toolbar, click the Draw Beams button again to open the Draw Beams dialog box.

Under Drawing Options, select Draw Point to Point, and click Apply.

Draw by clicking in the member from the quarter point of one beam, and dragging to the half point of another. Right-click to release the mouse.

Note: The cursor provides additional information regarding the cursor coordinates. As you move your cursor around on your screen, a box adjacent to your cursor will appear and populate with the coordinates of the cursor.
The mouse coordinates that are displayed are the coordinates of the grid point or joint that is nearest to the cursor. These are the same coordinates displayed in the Status Bar in the third box when a model view is active.

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

![Coordinates Display](image)

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

![Diagram](image)

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

This is the end of Tutorial A2.

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

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

**Note:** If you are running the Demonstration version, the file is now too large to save. You will need to directly continue on with the remaining tutorials or use the Starter Files for the following tutorials.
Part A: Tutorial 3 – Decks & Diaphragms

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

If you are continuing from the previous tutorial:

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

-OR- If you are starting here from scratch, follow the steps below to load the starter file provided by RISA:
  - Double-click on the RISAFloor icon to start the program.
  - Click Open Model. Double-click the Tutorials folder, select Tutorial A3 Starter.rfl and click Open. Click Close (or Cancel) to exit the Model Settings dialog box.
  - On the Window toolbar, click the Graphic Editing button to activate the Drawing toolbar.

Your screen should now look like this:
Creating Diaphragm Edges

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

- On the Selection toolbar, click the Select All button to make sure the entire model is selected.
- On the Drawing toolbar, click the Create Diaphragm Perimeters button to open the following dialog box.

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

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

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

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

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

- For the Concrete Deck, change Max Span to 12 ft.

- Next, click within the Material Type cell of the second row (Composite Deck). Click on the red arrow to open the Deck Selection dialog.

- Click through the various Deck Types to see the available options. Click Cancel when you are finished.

- Next, click the Loads tab and review the loading input options.

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

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

- Close the spreadsheet by clicking Close.
Next, you will learn how to apply these deck properties to your floor. Each floor has a **Default Deck** and **Deck Angle** that will be used for the entire floor if no other deck is defined. To review and modify these defaults:

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

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

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

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

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

- Click **Apply**.
Part A: Tutorial 3 – Decks & Diaphragms

- 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 loads as applied:

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

- Click **OK**.
Notice the differences between this plot and the previous plot. In this plot, you can verify the location of openings.
Move your mouse over the various decks. Notice the deck type and orientation information are displayed in the Status bar as you move the cursor.

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

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

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

This is the end of Tutorial A3.

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

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

**Note:** If you are running the Demonstration version, the file is now too large to save. You will need to directly continue on with the remaining tutorials or use the Starter Files for the following tutorials.
Part A: Tutorial 4 – Loading

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

If you are continuing from the previous tutorial:

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

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

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

Your screen should now look like this:
Area Loads

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

Edit the Area Loads spreadsheet, as shown below:

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

![Spreadsheet Image]

The entries in the Area Loads spreadsheet are described below:

<table>
<thead>
<tr>
<th>Label</th>
<th>The name you will later use to refer to the load when you apply or view it.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Additive</td>
<td>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.</td>
</tr>
<tr>
<td>PreDL</td>
<td>These entries allow you to specify load magnitudes for different categories that will later be used in the load combinations that you solve.</td>
</tr>
<tr>
<td>PostDL</td>
<td>PreDL and PostDL are for dead loads that are pre- and post-composite. Remember that other dead loads (deck self-weight and construction dead load) were already defined within the deck definitions spreadsheet.</td>
</tr>
<tr>
<td>LL</td>
<td>LL and LL Type load columns are for the live load magnitude and the type of live load which you may choose from a list.</td>
</tr>
<tr>
<td>VL</td>
<td>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.</td>
</tr>
<tr>
<td>Dyn Load</td>
<td>The dynamic load will be used to determine the diaphragm mass and mass moment of inertia for a RISA-3D seismic analysis.</td>
</tr>
</tbody>
</table>

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

- Click Close to exit the spreadsheets and return to the model view.
To apply local area loads:

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

![Create Area Loads dialog box](image)

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

- **OR-**

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

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

Draw the area loads:

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

- Click **Apply**.

**Note:** The cursor will change to 🗻.
Draw the polygons for the two area loads by clicking on points, as shown in the following figure.

**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**.
Part A: Tutorial 4 – Loading

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

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

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

**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.
Part A: Tutorial 4 – Loading

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:

- **DL PostComp-Pc**
  - Toggle between load categories and combinations.
- **Toggle the graphical display of the loads.**
- **Select the various load categories or combinations.**

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

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

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

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

![Floor Line Loads dialog box](image)

- Click Apply.

**Note:** The cursor will change to .
To see the model better while you are drawing in the load, change to an isometric view and zoom in:

- On the Window toolbar, click on the **Isometric** button.
- 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.
- On the Window toolbar, click **Redraw** to return to a full view.

Next, the loads need to be modeled due to the exterior cladding. To do this, apply a uniform load of 2.5 kips/ft to all beams on the perimeter of the building. RISAFloor provides a selection tool that makes this easy.
First, unselect your entire model and make the following selection changes:

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

- Click OK to apply the selection.

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

On the Drawing toolbar, select the Line Load button again to open the Floor Line Loads dialog box.

For the DL Post load, enter a starting and ending magnitude of 2.5 kip/ft.

Under What happens when Apply is pressed?, click Apply to All Selected Members/Walls.

Click Apply.

Your structure should look like this:
Now, add the last two cladding loads.

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

Prepare to apply point loads:

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

![Floor Point Loads dialog box](image)

- 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:

![Diagram of a floor plan with point loads drawn in](image)

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

This is the end of Tutorial 4.

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

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

**Note:** If you are running the Demonstration version, the file is now too large to save. You will need to directly continue on with the remaining tutorials or use the Starter Files for the following tutorials.
Part A: Tutorial 5 – Solution & Results

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

If you are continuing from the previous tutorial:

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

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

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

Your screen should now look like this:
RISAFloor solves load combinations that are defined in the **Load Combinations** spreadsheet. As RISAFloor designs members, it chooses members that satisfy the codes specified in the **Model Settings** and the rules that are defined in the **Design Rules** spreadsheet.

### Load Combinations

To view the load combinations:

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

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

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

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

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

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

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

- Click **Generate**.
The following load combinations will be generated by RISAFloor:

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.

- Click Generate.
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.

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

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

- Uncheck the **Hot Rolled Steel** checkboxes for the strength level combinations (lines 8-14).
- Uncheck the **Concrete** checkboxes for the service level combinations (lines 1-7).
Part A: Tutorial 5 – Solution & Results

When finished, your spreadsheet should look like this:

![Spreadsheet Image](image)

- Close the spreadsheet by clicking the Close button.

Solving and Reviewing Results

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:

![Solution Warning Image](image)

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.

![Warning Log]

**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.

![Design Results]

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:

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

Click Apply.
Your plot should now look like this:

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

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

- **Click the Box Zoom button** and select the portion of the model where the most obvious deck span violation occurred, as shown in the figure below:

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

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

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

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

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

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

To perform an interactive redesign on any member:

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

![Member Redesign Dialog Box](image1)

- Click Cancel to close the dialog box.

You can now make a framing change to your model.

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

- On the Drawing toolbar, click Draw Beams.

This is the same button that you initially used to draw the beams.
Part A: Tutorial 5 – Solution & Results

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

![Clear Results Warning]

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.

![Diagram]

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 directly continue on with the remaining tutorials or use the Starter Files for the following tutorials.
Part A: Tutorial 6 – RISA-3D Integration

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

If you are continuing from the previous tutorial:

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

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

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

Your model will look like the following:
To see the interaction between RISAFloor and RISA-3D, solve your RISAFloor model, and then use RISA-3D to analyze and design your lateral system.

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

When you enter into 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 Dialog Box]

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

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

- **Click OK** to accept the loads.
After the **Wind Loads** dialog box closes, the **Seismic Loads** dialog box will display:

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

Be sure to also check the information generated in the **Seismic Load Results** area. This gives the equivalent static force distribution and base shear 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**.
An isometric view of your model will now be displayed in RISA-3D. Although the interface looks very similar to that of RISAFloor, you are now working completely in RISA-3D.

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

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

Your screen should now look like this:
On the **Main** menu, click **Spreadsheets**, 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.

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

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

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

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

Click **Close** to exit the **Diaphragms** spreadsheet and return to the model view.
On the Window toolbar, click on the **Diaphragm Display** button.

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:

1. On the Window toolbar, click the **Graphic Editing** button to activate the Drawing toolbar.
2. On the Drawing toolbar, click the **Draw New Members** button. The **Draw Members** dialog box will appear.
Click Assign Shape Directly. Under Start Shape, type HSS4x4x4, under Type, select VBrace, under Design List, select Square Tube, and under Material, select A500 Gr. B Rect.

Under Release Codes, select Pinned at Both Ends.

Click Apply.
Draw in the X braces (as shown in the image below) by clicking and dragging from one point to the next. 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.

- On the Window toolbar, click the LC Generator button to display the following dialog box.

- 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.
- 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).

![Load Combination Generator - Wind](image)

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).

![Load Combination Generator - Seismic](image)

Click **Generate**. Then click **Close** to close the dialog.
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 \Users\Documents\RISA\Load Combinations (or C:\RISA\risa_LC_lists) 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 and the interfaces of these two programs are quite similar. 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.

CIS/2 Translator: http://www.risatech.com/p_cis2.html?tab=2
DXF Export/Import: http://risanews.com/tag/dxf /

RISA-Revit Link

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

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

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

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

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

Solve the model and save:

- Click Solve to solve the model. Click OK from the Solution Warning dialog.
- From the File menu, click Save and be sure to Save Results.
Your model should now display design data such as member sizes, end forces, and number of studs:

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

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

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

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

- Open a 3D Analytical view.
- On the Main menu, click Add-Ins, and select Import from RISA.
Click **Browse** to locate the Autodesk Revit (.exc) exchange file, as shown below.

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

**Note:** This was just a basic description of the RISA-Revit Link interoperability. For more information, you may download documentation from the RISA website specific to the Link and how to use it. Go to the RISA website: [http://www.risa.com/partners/prt_revit.html](http://www.risa.com/partners/prt_revit.html).
CIS/2 Translator

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

Here are instructions on how to install the translator:

- Click Close to exit RISAFloor if it is open.
- Go to http://www.risa.com/products.html, click RISA CIS/2 Translator 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.

![Translator Interface]

- 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: http://www.risa.com/d_documentation.html.
RISAFloor & CAD

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

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

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

Solve the model and save:

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

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

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

You should be able to open this DXF file with any standard drafting package. The following is an example of the information that can be written to the drawing file.
Part A: Tutorial 7 – RISA Interoperability

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

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

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

- Enter the information shown below:

![Import DXF File dialog box](image)

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

![New floor from imported DXF file](image)

This is the end of Tutorial 7.
Conclusion

This completes Part A: Beam Supported Floors.

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

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

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

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

The tutorials build upon themselves from start to finish. You have the option of performing them all at one time, or performing each one separately. To make this possible, RISA provides model files for you to load at the beginning of each tutorial. These starter files are located in the RISA program folder under Tutorials, and are named [Tutorial B2 starter.rfl](#), [Tutorial B3 starter.rfl](#), etc.

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

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

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

Starting a New File

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

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

![Starting a New File](image)

**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.**

![Create New Floor Plan dialog box]

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

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

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

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

- Click on the **Browse for File** button and then browse to the **Tutorials** file under your **Documents\RISA\Model Files\Tutorials** folder.
- Select the file called Tutorial B1 Drawing Grid.dxf.
Next you are given the option to select which layers in your DXF drawing to import. For this tutorial we will bring in all layers.

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

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

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

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

Close the **Model Settings** dialog box:

- Click **Cancel**.

**Drawing**

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

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

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

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

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

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

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

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

Create the design rule:

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

![Diagrams of design rule setup]

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

![Diagram of rebar setup]

- Click **OK** to close the dialog and save your design rule.
Return to the **Draw Concrete Columns** dialog to finish defining the columns:

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

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

**Note:** When you enter the drawing mode, your cursor changes to 🗑️.
You can now start defining the columns by clicking on the drawing grid locations as shown below:

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

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

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

- Use whichever method you prefer to select the remaining column grid locations.
Right-click your mouse to cancel the drawing tool when done.

When finished, your model will look like this:
Modifying Your Columns

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

**Column Stack Editor**

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

![Column Stack Manager dialog](image)

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

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

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

- Select **Columns** from the **Modify** menu.
- Under **Function**, select **Lateral**. Check the “**Use?**” checkbox.
- At the bottom, select **Apply Entries to All Selected Columns** and then click **Apply**.
All of the columns will change to red, indicating that they are now all lateral members.
Wall Panels

Prepare to draw your wall panels:

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

![Wall panel settings dialog box]

- Click Apply.

**Note:** Your cursor will change to ✏️ to indicate that you are now ready to draw wall panels.
Part B: Tutorial 1 – Columns & Walls

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

- Click on **Box Zoom** from the Drawing toolbar.
- Box select the elevator opening on the drawing grid.
Now that you have a zoomed in view, start drawing your wall panels:

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

- Continue around the outside of the elevator opening.

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

- To release the drawing tool, use your right mouse (right-click) or ESC.

- Return to the full model view by clicking on the Full Model View button from the Drawing toolbar.
When finished, your model should look like the following.

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

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

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

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

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

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

- Click the **Toggle Rendered View** button at the top center of the **Editor** Drawing toolbar. Now only the drawing grid is visible in the plane of the wall.

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

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

Now draw in the opening.

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

**Note:** Your cursor will change to .
Use the coordinate information in the lower right Status Bar to help select the opening coordinates.

₁ First click on the top left of the opening (96.09, 11, 24).

₂ Next click the lower right corner of the opening (96.09, 8, 30).
When complete, the opening will show with the X lined and the lintel label.

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

Wall Panel Design Regions

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

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

Turn the rendered view back on for a better view.

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

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

For more details on how to model wall panels, including information on masonry design and wood design, refer to http://www.risa.com/d_documentation.html.
Click **OK** to close the **Wall Panel Editor**.

This is the end of Tutorial B1.

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

If you are using the full version of the program, you may save by doing the following: **On the File menu, click **Save As** and enter a file name.**
Part B: Tutorial 2 – Slab & Design Strips

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

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

Re-Import the DXF Drawing Grid

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

Start by drawing in the slab.

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

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

**Note:** If you are using the Demonstration version, this option is not available, so you will need to skip down to the Draw Slab section on page 139 to accept the default slab definition and continue drawing.

![Slab Definition](image)

**Slab Definition**

If you are not using the Demonstration version, click on the New button.
Type in the label “Tutorial Slab”, change the Thickness to 8”, and set the Diaphragm type to Semi-Rigid.

Click OK to exit the dialog and return to the Draw Slabs dialog. The new slab definition should now be selected in the dialog.
**Draw Slab**

- **Click** Apply to start drawing.

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

Draw your Slab edge:
- **Using the snap points from our DXF drawing grid background to draw in the edge of the slab.**

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

When you click back on the starting point a second time to finish, the tool will release and your slab edge will turn green as shown above.
- **Right-click** your mouse once more to cancel the drawing tool.
When finished, your model should look like the following.

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

For more direction on how to use this tool, please see the Redraw Edges section below on page 145.
Modifying Your Slab

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

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

Here you can see your slab properties in spreadsheet form.

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

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

Next, you will modify the slab edges.

**Offset Slabs**

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

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

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

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

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

Click **Undo** once to return to the original slab edge.
Modifying the Slab Edges

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

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

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

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

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

**Note:** Your cursor in the graphical model view will now look like this in order to designate that you are in modification mode.
Box select the lower right side of the slab edge as shown here:

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

Click ESC on your keyboard or right-click your mouse to escape the modification tool.

Click Undo once to return back 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:

![Instructional Dialog]

**Click OK to continue.**

**Note:** Your cursor in the graphical model view will now look like this in order to designate 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 back to the original slab:

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

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

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

![Create Slab Openings dialog]

- Click Apply.

**Note:** Your cursor will change to in order to designate that you are now in drawing mode.

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

- Click on the Universal Snap Points button from the Drawing toolbar.
Draw in the opening:

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

  (32,20,0)
  (42,20,0)
  (43,12,0)
  (31,12,0)

Once again on (32,20,0)

Note:
You can draw an opening polygon of any shape.
Zoom in on the lower portion of the model for an easier view.

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

- Unselect the **Universal Snap Points** button from the Drawing toolbar.
- Unselect the **Drawing Grid** button from the Drawing toolbar.
Next, you will draw in the elevator shaft opening.

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

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 wall-beam intersection at this location.

- Right-click to release the opening tool.
When complete, the model will look like the following image.
Support Lines and Design Strips

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

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

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

Click Apply.

Note: Your cursor will change to in order to designate that you are now in drawing mode.
Draw in your support lines by clicking column to column. Note that you can click over existing columns and the support line will break at the column as long as it’s collinear with your line. Start by clicking on the red lateral column in the lower left corner.

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

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

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

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

The North-South Lines will be toggled off.
Return to the Draw Support Line dialog by clicking CTRL+D.

Change the Strip Orientation to East-West and the Design Rule to E-W Rebar.

Click Apply.

Draw in all the Support Lines as shown below.

Note: There is no need to draw support lines over the lateral beams.
Toggle back on the display of all Support Lines.

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

Note: The Support Line labels may vary depending on the order in which they were input, so it is okay if your lines do not exactly match those shown above.
Since there are some significant cantilevered portions of the slab floor, you will next extend some of the Support Lines out to the edge of slab.

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

![Modify Area dialog](image)

- 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 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.
Part B: Tutorial 2 – Slab & Design Strips

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 dialog](image)

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

![Model zoomed in](image)
First click on the Design Strip edge and then the line that you want to align with.

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

When finished your model should look like this:

**Delete Items**

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

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

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

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

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

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

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

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

This is the end of Tutorial B2.

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

If you are using the full version of the program, you may save by doing the following: On the **File** menu, click **Save As** and enter a file name.
Part B: Tutorial 3 – Loading

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

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

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

Your screen should now look like this:
Area Loads

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

Edit the Area Loads spreadsheet, as shown below:

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

![Area Loads Spreadsheet](image)

The entries in the Area Loads spreadsheet are described below:

<table>
<thead>
<tr>
<th>Label</th>
<th>The name you will later use to refer to the load when you apply or view it.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Additive</td>
<td>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.</td>
</tr>
<tr>
<td>PreDL</td>
<td>These entries allow you to specify load magnitudes for different categories that will later be used in the load combinations that you solve.</td>
</tr>
<tr>
<td>PostDL</td>
<td></td>
</tr>
<tr>
<td>LL</td>
<td></td>
</tr>
<tr>
<td>LL Type</td>
<td></td>
</tr>
<tr>
<td>VL</td>
<td>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.</td>
</tr>
<tr>
<td>Dyn Load</td>
<td>The dynamic load will be used to determine the diaphragm mass and mass moment of inertia for a RISA-3D seismic analysis.</td>
</tr>
</tbody>
</table>

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

- Click Close to exit the spreadsheet and return to the model view.
Drawing Area Loads

To apply local area loads:

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

![Create Area Loads dialog box]

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

- Click **Apply**.

**Note:** The cursor will change to 🔐.

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

- Click the **Universal Snap Points** button from the Drawing toolbar.
Draw the polygons for the area load by clicking on the drawing grid snap points, as shown in the following figure.

Storage Area Load
Corner Coordinates:
(42,104,0)
(42,117,0)
(5,117,0)
(5,104,0)

Click on **Area Loads** button to reopen the **Create Area Loads** dialog box.

Under Uniform Area Loads, in the Area Load box, select Public.

Click **Apply**.
**Part B: Tutorial 3 – Loading**

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

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

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

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

- Right-click to release the drawing tool.
**Plotting Area Loads**

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

To plot the area loads:

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

Your screen should look similar to the figure shown below:

![Figure showing area loads as applied](image)

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.
**Part B: Tutorial 3 – Loading**

**Toggle Load Display**

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

- **Switch Load Display** button - Toggle between load categories and combinations.
- **Area Loads** button - Toggle the graphical display of the loads.
- **Load** menu - Select the various load categories or combinations.

Use these buttons to change the display of loads.

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

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

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

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

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

![Floor Line Loads dialog box](image)

- Click **Apply**.

**Note:** The cursor will change to \( \downarrow \) 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.
- 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.

![Isometric View](image-url)

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.
- On the Window toolbar, click **Redraw** to return to a full view.
Point Loads

Prepare to apply point loads:

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

![Floor Point Loads dialog box]

- Click **Apply**.

**Note:** The cursor will change to \( \leftrightarrow \) 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 \( \bullet \) on the Drawing toolbar is turned on (the button will be depressed).
Part B: Tutorial 3 – Loading

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

Draw in the point loads:

- Use the coordinates (notice they are displayed in the Status bar) and the snap points to draw in a point load at each of the following coordinates:
  - (61,86,0)
  - (61,76,0)
  - (56,81,0)
  - (66,81,0)

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

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

- Right-click to release the drawing tool.
The last modification step is to create a second floor.

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

![Create New Floor Plan](image)

**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. Close both views and return to one 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**.

![Floors menu](image)

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

![Full Model](image)

**Note:** Notice that the wall opening 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:

1. Double-click on the RISAFloor icon to start the program.
2. Click Open Model. Double-click the Tutorials folder, select Tutorial B4 Starter.rfl and click Open. Click Close (or Cancel) to exit the Model Settings dialog box.
3. On the Window toolbar, click the Graphic Editing button to activate the Drawing toolbar.
4. 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:

 Ember On the Data Entry toolbar, select Load Combinations.

![](image.png)

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

 Ember 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.

 Ember 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.

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

![](image.png)

 Ember Click Generate.
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.

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.

- Click **Generate**.
Part B: Tutorial 4 – Solution & Results

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.

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

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

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

◆ Click OK when the next Solution Warning dialog comes up.
When the solution is finished, the chosen member sizes are displayed on the plot (or, if you are in rendered mode, the model is rendered with the sizes) and the solution Warning Log is displayed.

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

**Reviewing the Results**

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

The Results toolbar should now open on the upper right of your model space on top of the Data Entry toolbar.
Take a look at some of the results.

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

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 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.
- From the **Modify** menu, select **Columns**.
- Click Yes to close the Clear Results Warning.

Use the **Modify Column Properties** dialog to add in a shear cap.
Under **Shear Cap Options**, select the **Use?** checkbox.

Select Add/Modify Rectangular Shear Cap (in).

Enter a **Thickness** of 18”, a **Depth** of 60” and a **Width** of 60”.

![Image of Shear Cap Options]

- Click **Apply**.

**Note:** Your cursor will change to ![modified cursor] to designate that you are in modification mode.
To find the location of the columns, turn on the column labels.

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

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

Now add a Shear Cap to these columns.

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

Right-click or click ESC to escape the modification tool.
When complete, the model should show the shear cap perimeters.

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

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

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

**Spreadsheet Results**

Open the **Strip Reinforcing** spreadsheet:

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

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.

Return to your model view:
- On the **Main** menu, click **Window** and select **Single View**.
**Viewing Graphical Results**

Explore the graphic result viewing options:

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

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

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

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

- Click **Apply**.
Your plot should now look like this:

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

- Click Cancel to exit the dialog box and return to your model view.
Next, take a look at the design strip graphical results but this time you will use the Window toolbar buttons. These give you quicker access to the graphical plot options.

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

Your model view should now look like this:
Detail Reports
Take a look at the details of the design for the Support Lines and Design Strips.

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

**Note:** Your cursor will change to .

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

The Detail Report displays enveloped shear and moment diagrams for each span of the Support Line. It also gives Analysis Results (Max and Min Shear and Moment values used for reinforcement design) and Deflection Results for each span.
You can use the drop-down menu in the upper left to see the Total report or the more detailed reports for the Column Strip, Mid-Strip Right, or Mid-Strip Left.

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

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

Next, you will take a look at the governing **Cut** results.
The reinforcement for each Design Strip is governed by an internal Design Cut.

- Click on the Cut button from the Design Strip Detail Report. This will open the Design Cut Detail Report for the governing cut for Column Strip Span 1.

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

Note: You can click on the forward and backward arrows at the top left to scroll quickly through the Design Cut Detail Reports.
Part B: Tutorial 4 – Solution & Results

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

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

Turn off the display of the support lines.

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

Now, take a look at the Design Cut Results:

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

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

![Detail Report](image)

When finished reviewing these results, close the Detail Report.

- Click Close to close the Detail Report and return to the model view.
Now you can review the column design results in the **Detail Report**.

- With your **Detail cursor** still active, click on the top left column (with the shear cap).
This will open the Envelope Column Detail Report.

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

Next, take a look at the Punching Shear results.

From the drop-down menu in the upper left, select Punching Shear.
This will open the **Punching Shear Detail Report** for the column which now shows the passing code check.

![Punching Shear Detail Report](image)

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

This is the end of Tutorial B4.

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

If you are using the full version of the program, you may save by doing the following: On the **File** menu, click **Save As** and enter a file name.
Part B: Tutorial 5 – RISA-3D Integration

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

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

- Double-click on the **RISAFloor** icon to start the program.

- Click **Open Model**. Double-click the **Tutorials** folder, select **Tutorial B5 Starter.rfl** and click **Open**. Click **Close** (or **Cancel**) to exit the **Model Settings** dialog box.

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

- Turn off the view of the design strips by clicking on the **Design Strip** button on the Drawing toolbar. Click on the black menu arrow and select **None**.

Your screen should now look like this:
To see the interaction between RISAFloor ES and RISA-3D, solve your RISAFloor ES model, and then use RISA-3D to analyze and design your lateral system.

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

### Lateral Load Generators

When you enter into 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 dialog box](image)

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

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

- Click **OK** to accept the loads.
After the Wind Loads dialog box closes, the Seismic Loads dialog box will display:

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

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

- Click OK to accept the loads.
- If the Model Settings dialog box displays, close it by clicking OK or Cancel.
An isometric view of your model will now be displayed in RISA-3D. Although the interface looks very similar to that of RISAFloor ES, you are now working completely in RISA-3D.

Notice that only the beam, 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**.

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

- Click **Close** to exit the **Diaphragms** spreadsheet and return to the model view.
RISA-3D Loading

Next, look at the loading information that was brought over from RISAFloor ES.

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

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

- When you are finished reviewing the data, click Close to close the spreadsheet and return to your model view.
RISA-3D Load Combination Generator

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

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

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

- On the Window toolbar, click the LC Generator button to display the following dialog box.

![LC Generator Dialog Box]

- 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.
- Clear the Generate Deflection LCs box.
- 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).  

![Wind Load Options](image)

- 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).  

![Seismic Load Options](image)
- 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**.

![Load Combination Generator - Gravity](image)

- Click **Generate**.
- Click on the **Wind** tab. Make sure that the **LC Code** also updated on this tab.

![Load Combination Generator - Wind](image)
Part B: Tutorial 5 – RISA-3D Integration

- Click Generate.
- Click on the **Seismic** tab. Make sure that the **LC Code** also updated on this tab.

![Load Combination Generator - Seismic](image)

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

This will generate the many load combinations shown below.

![Load Combinations](image)

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 Combinations (or C:\RISA\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.

With the requirements of the *AISC 14th Edition Steel Code* and the *ACI 318-14 Concrete Code*, you will need to include P-Delta effects in your results. This is not included by default, but you can include it in the Load Combinations spreadsheet.

- Use your cursor to highlight the PDelta 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.
As you can see from this tutorial, the interaction between RISAFloor ES and RISA-3D is completely seamless and the interfaces of these two programs are quite similar. 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.
Part B: Tutorial 6 – DXF Export

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 dialog box]

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.

![Floor Plan 1]
DXF Import

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

- On the RISA toolbar, click on **New Model** to start a fresh model.
- Click **Close** to exit the **Starting a Model** dialog box.
- On the **Main** menu, click **File**, click **Import**, then select **DXF File**. Select the first floor tutorial file you recently exported and saved. It should be named **Tutorial FLOOR_1.dxf**.

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

- Enter the information shown below:

![Import DXF File dialog box](image)

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

![Imported DXF file](image)

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

<table>
<thead>
<tr>
<th>Button</th>
<th>Title</th>
<th>Label in RISAFoundation</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>![Folder]</td>
<td><strong>New Model</strong> ![Document]</td>
<td>Start a new model</td>
<td>(CTRL+N)</td>
</tr>
<tr>
<td>![Folder]</td>
<td><strong>Open Model</strong> ![Document]</td>
<td>Open an existing model</td>
<td>(CTRL + O)</td>
</tr>
<tr>
<td>![Folder]</td>
<td><strong>Save</strong> ![Document]</td>
<td>Save current model to disk</td>
<td>(CTRL+S)</td>
</tr>
<tr>
<td>![Folder]</td>
<td><strong>Copy</strong> ![Document]</td>
<td>Copy to the clipboard</td>
<td>(Ctrl+C)</td>
</tr>
<tr>
<td>![Folder]</td>
<td><strong>Print</strong> ![Document]</td>
<td>Print a report or graphic image</td>
<td>(Ctrl+P)</td>
</tr>
<tr>
<td>![Folder]</td>
<td><strong>Undo</strong> ![Document]</td>
<td>Undo the last operation</td>
<td>(Ctrl+Z)</td>
</tr>
<tr>
<td>![Folder]</td>
<td><strong>Redo</strong> ![Document]</td>
<td>Reverse the most recent undo operation</td>
<td>(Ctrl+Y)</td>
</tr>
<tr>
<td>![Folder]</td>
<td>(Global) <strong>Model Settings</strong> ![Document]</td>
<td>Set Model Settings</td>
<td></td>
</tr>
<tr>
<td>![Folder]</td>
<td><strong>Define Units</strong> ![Document]</td>
<td>Define units to be used</td>
<td></td>
</tr>
<tr>
<td>![Folder]</td>
<td><strong>Shape Database</strong> ![Document]</td>
<td>Edit or view the shape database</td>
<td></td>
</tr>
<tr>
<td>![Folder]</td>
<td><strong>Rebar Layout</strong> ![Document]</td>
<td>Create and edit a custom rebar layout</td>
<td></td>
</tr>
<tr>
<td>![Folder]</td>
<td><strong>Project Grid</strong> ![Document]</td>
<td>Open the Project Grid spreadsheet</td>
<td></td>
</tr>
<tr>
<td>![Folder]</td>
<td><strong>Create New Floor Plan</strong> ![Document]</td>
<td>Create a new floor plan</td>
<td></td>
</tr>
<tr>
<td>Button</td>
<td>Description</td>
<td>Keyboard Shortcut</td>
<td></td>
</tr>
<tr>
<td>-----------------</td>
<td>------------------------------------------------</td>
<td>------------------</td>
<td></td>
</tr>
<tr>
<td>Delete Floor Plans</td>
<td>Delete existing floor plans</td>
<td></td>
<td></td>
</tr>
<tr>
<td>New Model View</td>
<td>Create a new model view</td>
<td>(CTRL + F2)</td>
<td></td>
</tr>
<tr>
<td>Open Spreadsheets</td>
<td>Select spreadsheets to open</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Refresh</td>
<td>Refresh all open windows with current data</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Load Combinations</td>
<td>Open the load combinations spreadsheet</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Solve</td>
<td>Perform the analysis and design calculations</td>
<td>(F7)</td>
<td></td>
</tr>
<tr>
<td>Browse Results</td>
<td>Select results to browse</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Erase Results</td>
<td>Erase all solution results</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Data Entry toolbar</td>
<td>Turn the spreadsheet shortcuts menu on or off</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Results toolbar</td>
<td>Turn the results shortcuts menu on or off</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Help</td>
<td>View Help menu topics</td>
<td>(CTRL + F1)</td>
<td></td>
</tr>
</tbody>
</table>
## Window Toolbar

...in Model View

<table>
<thead>
<tr>
<th>Button</th>
<th>Title</th>
<th>Label in RISA Foundation</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>![Button]</td>
<td><strong>Model Display Options</strong></td>
<td>Bring up the Model Display Options dialog</td>
<td>F2</td>
</tr>
<tr>
<td>![Button]</td>
<td>Floor Menu</td>
<td>Select which floor you want to view</td>
<td></td>
</tr>
<tr>
<td>![Button]</td>
<td>Snapshot</td>
<td>Save an image of the current model view for later use in the printed report</td>
<td></td>
</tr>
<tr>
<td>![Button]</td>
<td>Isometric</td>
<td>Snap to an Isometric View</td>
<td></td>
</tr>
<tr>
<td>![Button]</td>
<td>Plan</td>
<td>Snap to an XZ Planar View</td>
<td></td>
</tr>
<tr>
<td>![Button]</td>
<td>Rotate buttons</td>
<td>Rotate the view counter-clockwise about the X axis</td>
<td></td>
</tr>
<tr>
<td>![Button]</td>
<td></td>
<td>Rotate the view counter-clockwise about the Y axis</td>
<td></td>
</tr>
<tr>
<td>![Button]</td>
<td></td>
<td>Rotate the view counter-clockwise about the Z axis</td>
<td></td>
</tr>
<tr>
<td>![Button]</td>
<td>3 buttons below are collectively called <strong>Zoom</strong> buttons</td>
<td></td>
<td></td>
</tr>
<tr>
<td>![Button]</td>
<td>Zoom In</td>
<td>Zoom IN (closer view) on the model</td>
<td>PLUS (+)</td>
</tr>
<tr>
<td>![Button]</td>
<td>Zoom Out</td>
<td>Zoom OUT (view farther away) on the model</td>
<td>MINUS (-)</td>
</tr>
<tr>
<td>![Button]</td>
<td>Box Zoom</td>
<td>Draw a box around the part of the model to be zoomed</td>
<td></td>
</tr>
<tr>
<td>![Button]</td>
<td>Redraw</td>
<td>Redraw full model view</td>
<td></td>
</tr>
</tbody>
</table>
### Appendix A – RISAFloor Toolbar Button Quick Reference

<table>
<thead>
<tr>
<th>Graphic Editing</th>
<th>Activate the Graphic Editing (AKA Drawing) toolbar</th>
<th>Ctrl+G</th>
</tr>
</thead>
<tbody>
<tr>
<td>3 buttons below collectively refer to <strong>Loads Display</strong></td>
<td><strong>Display Loads</strong> Toggle display of the loads (LC or CAT)</td>
<td><strong>Switch Loads</strong> Switch loads display between combinations and categories</td>
</tr>
<tr>
<td><strong>Loads List</strong> – lists the available loads</td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Show Wall Properties</strong> Wall Panel toggle</td>
<td>Toggles on the display of Wall Panel properties</td>
<td></td>
</tr>
<tr>
<td><strong>Show Deck Assignment</strong> Show Deck Assignment</td>
<td>Toggles on the display of the deck for beam supported floors only</td>
<td></td>
</tr>
<tr>
<td><strong>Show Area Load Types</strong> Show Area Load</td>
<td>Toggles on the display of the area loads</td>
<td></td>
</tr>
<tr>
<td><strong>Show Project Grid</strong> Show Project Grid toggle</td>
<td>Toggles on the display of the Project Grid</td>
<td></td>
</tr>
</tbody>
</table>
Appendix A – RISAFloor Toolbar Button Quick Reference

<table>
<thead>
<tr>
<th>Button</th>
<th>Title</th>
<th>Label in RISA.Foundation</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="77x672" alt="Distance Tool" /></td>
<td>Distance Tool</td>
<td>Measures the distance between two points</td>
<td>F5</td>
</tr>
<tr>
<td><img src="212x672" alt="Design Strip Rebar" /></td>
<td>Design Strip Rebar</td>
<td>Toggles on the display of the Design Strip rebar</td>
<td></td>
</tr>
<tr>
<td><img src="98x529" alt="Design Strip" /></td>
<td>Design Strip</td>
<td>Toggles on the display of the Design Strips and/or the Support Lines</td>
<td></td>
</tr>
<tr>
<td><img src="77x628" alt="Design Strip Diagram" /></td>
<td>Design Strip Diagram</td>
<td>Toggles on the display of the force diagram on the Design Strip</td>
<td></td>
</tr>
<tr>
<td><img src="72x43" alt="New Line" /></td>
<td>New Line</td>
<td>Insert a new line before the current line</td>
<td>F3</td>
</tr>
<tr>
<td><img src="72x268" alt="Delete Line" /></td>
<td>Delete Line</td>
<td>Delete the current line</td>
<td>F4</td>
</tr>
<tr>
<td><img src="72x229" alt="Repeat Line" /></td>
<td>Repeat Line</td>
<td>Repeat the current line</td>
<td>F8</td>
</tr>
<tr>
<td><img src="72x179" alt="Fill" /></td>
<td>Fill</td>
<td>Fill the currently marked block</td>
<td>Ctrl+F</td>
</tr>
<tr>
<td><img src="72x131" alt="Math" /></td>
<td>Math</td>
<td>Perform math on the currently marked block</td>
<td>Ctrl+M</td>
</tr>
<tr>
<td><img src="72x98" alt="Mark Lines" /></td>
<td>Mark Lines</td>
<td>Mark all the lines</td>
<td></td>
</tr>
</tbody>
</table>

...in Spreadsheet View
<table>
<thead>
<tr>
<th></th>
<th>Function</th>
<th>Description</th>
<th>Keyboard Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="icon" /></td>
<td><strong>Delete Lines</strong></td>
<td>Delete the currently marked lines</td>
<td>Ctrl+D</td>
</tr>
<tr>
<td><img src="image" alt="icon" /></td>
<td><strong>Unmark Lines</strong></td>
<td>Unmark all the lines</td>
<td>Ctrl+L</td>
</tr>
<tr>
<td><img src="image" alt="icon" /></td>
<td><strong>Paste</strong></td>
<td>Paste from the clipboard</td>
<td>Ctrl+V</td>
</tr>
<tr>
<td><img src="image" alt="icon" /></td>
<td><strong>Save as Defaults</strong></td>
<td>Save the current data as the default</td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="icon" /></td>
<td><strong>Help</strong></td>
<td>Help on the current window</td>
<td>SHIFT+F1</td>
</tr>
</tbody>
</table>
## Drawing Toolbar

<table>
<thead>
<tr>
<th>Button</th>
<th>Title</th>
<th>Label in RISAFoundation</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>![Column Button]</td>
<td>Draw or Modify Columns</td>
<td>Draw or modify columns</td>
<td></td>
</tr>
<tr>
<td>![Wall Button]</td>
<td>Draw or Modify Walls</td>
<td>Draw or modify walls</td>
<td></td>
</tr>
<tr>
<td>![Beam Button]</td>
<td>Draw or Modify Beams</td>
<td>Draw or modify beams</td>
<td></td>
</tr>
<tr>
<td>![Infill Button]</td>
<td>Generate Infill Beams</td>
<td>Generate infill beams within bays</td>
<td></td>
</tr>
<tr>
<td>![Elevation Button]</td>
<td>Elevate Points</td>
<td>Change the elevation of selected points</td>
<td></td>
</tr>
<tr>
<td>![Slab Button]</td>
<td>Draw Slabs/Diaphragms</td>
<td>Draw or modify slabs or diaphragms</td>
<td></td>
</tr>
<tr>
<td>![Deck Button]</td>
<td>Assign Deck</td>
<td>Assign deck properties</td>
<td></td>
</tr>
<tr>
<td>![Diaphragm Button]</td>
<td>Add Wood Diaphragm Regions</td>
<td>Add wood diaphragm regions</td>
<td></td>
</tr>
<tr>
<td>![Load Button]</td>
<td>Draw Area Loads</td>
<td>Draw area loads</td>
<td></td>
</tr>
<tr>
<td>![Line Button]</td>
<td>Draw Line Loads</td>
<td>Draw line loads point to point</td>
<td></td>
</tr>
<tr>
<td>![Point Button]</td>
<td>Assign Point Loads</td>
<td>Draw point loads</td>
<td></td>
</tr>
<tr>
<td>Icon</td>
<td>Tool Name</td>
<td>Description</td>
<td></td>
</tr>
<tr>
<td>------</td>
<td>--------------------</td>
<td>--------------------------------------------------------------</td>
<td></td>
</tr>
<tr>
<td><img src="https://via.placeholder.com/150" alt="Image" /></td>
<td>Project Grid</td>
<td>Draw Project Grid Lines</td>
<td></td>
</tr>
<tr>
<td><img src="https://via.placeholder.com/150" alt="Image" /></td>
<td>Move</td>
<td>Move selected parts of the model</td>
<td></td>
</tr>
<tr>
<td><img src="https://via.placeholder.com/150" alt="Image" /></td>
<td>Copy</td>
<td>Copy selected parts of the model</td>
<td></td>
</tr>
<tr>
<td><img src="https://via.placeholder.com/150" alt="Image" /></td>
<td>Offset Copy</td>
<td>Copy selected parts of the model using an offset distance</td>
<td></td>
</tr>
<tr>
<td><img src="https://via.placeholder.com/150" alt="Image" /></td>
<td>Extend</td>
<td>Extend the member</td>
<td></td>
</tr>
<tr>
<td><img src="https://via.placeholder.com/150" alt="Image" /></td>
<td>Trim</td>
<td>Trim the member</td>
<td></td>
</tr>
<tr>
<td><img src="https://via.placeholder.com/150" alt="Image" /></td>
<td>Delete</td>
<td>Delete parts of the model</td>
<td></td>
</tr>
<tr>
<td><img src="https://via.placeholder.com/150" alt="Image" /></td>
<td>Modify Drawing Grid</td>
<td>Modify the drawing grid and snap points</td>
<td></td>
</tr>
<tr>
<td><img src="https://via.placeholder.com/150" alt="Image" /></td>
<td>Drawing Grid</td>
<td>Toggle the drawing grid on or off</td>
<td></td>
</tr>
<tr>
<td><img src="https://via.placeholder.com/150" alt="Image" /></td>
<td>Universal Snap Points</td>
<td>Toggle the universal snap points on or off</td>
<td></td>
</tr>
</tbody>
</table>
### Selection Toolbar

<table>
<thead>
<tr>
<th>Button</th>
<th>Title</th>
<th>Label in RISAFoundation</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="icon" alt="Select All" /></td>
<td>Select All</td>
<td>Select the entire model</td>
<td>Make the entire model selected</td>
</tr>
<tr>
<td><img src="icon" alt="Box Select" /></td>
<td>Box Select</td>
<td>Draw a box around the part to be selected</td>
<td>Draw a box around the part of the model to be selected</td>
</tr>
<tr>
<td><img src="icon" alt="Polygon Select" /></td>
<td>Polygon Select</td>
<td>Draw a polygon around the part to be selected (double click to end)</td>
<td>Draw a polygon around the part to be selected (double click to end)</td>
</tr>
<tr>
<td><img src="icon" alt="Line Select" /></td>
<td>Line Select</td>
<td>Draw a line through the beams and plates to be selected</td>
<td>Draw a line through the beams and plates to be selected</td>
</tr>
<tr>
<td><img src="icon" alt="Unselect All" /></td>
<td>Unselect All</td>
<td>Unselect the entire model</td>
<td>Make the entire model UNselected</td>
</tr>
<tr>
<td><img src="icon" alt="Box Unselect" /></td>
<td>Box Unselect</td>
<td>Draw a box around the part to be unselected</td>
<td>Draw a box around the part of the model to be UNselected</td>
</tr>
<tr>
<td><img src="icon" alt="Polygon Unselect" /></td>
<td>Polygon Unselect</td>
<td>Draw a polygon around the part to be unselected (double click to end)</td>
<td>Draw a polygon around the part to be UNselected (double click to end)</td>
</tr>
<tr>
<td><img src="icon" alt="Line Unselect" /></td>
<td>Line Unselect</td>
<td>Draw a line through the beams and plates to be unselected</td>
<td>Draw a line through the beams and plates to be UNselected</td>
</tr>
<tr>
<td><img src="icon" alt="Invert Selected" /></td>
<td>Invert Selected</td>
<td>Invert the selected areas of the model</td>
<td>Invert the selected state of the model</td>
</tr>
<tr>
<td><img src="icon" alt="Criteria Selection" /></td>
<td>Criteria Selection</td>
<td>Select or unselect</td>
<td>Select or Unselect based on other criteria</td>
</tr>
<tr>
<td><img src="icon" alt="Save/Recall Selection" /></td>
<td>Save/Recall Selection</td>
<td>Save or recall selections of the model</td>
<td>Save or recall selection states for the model</td>
</tr>
<tr>
<td><img src="icon" alt="Lock Unselected" /></td>
<td>Lock Unselected</td>
<td>Lock the unselected part of the model</td>
<td>Ctrl+L</td>
</tr>
</tbody>
</table>