RISA-3D

Rapid Interactive Structural Analysis – 3-Dimensional

User's Guide

26632 Towne Centre Drive, Suite 210
Foothill Ranch, CA 92610

(949) 951-5815
(949) 951-5848 (FAX)
www.risatech.com

Table of Contents

Introduction

How to Use this Book

Welcome to the RISA-3D User's Guide. If you are a ***first-time user*** of RISA-3D, we recommend that you start with this book. This book contains step-by-step tutorials that guide you through the entire modeling process using most RISA-3D features. You will create a real-world example of building and solving a model, making changes, and optimizing the model. Several tips and shortcuts will also be demonstrated along the way.

Begin by reviewing First Look at RISA-3D on page 9 to familiarize yourself with the toolbars and shortcuts, then proceed to the step-by-step tutorials.

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 program folder under **Tutorials**, and are named **Tutorial 2 Starter.r3d**, **Tutorial 3 Starter.r3d**, etc.

After you have gone through the tutorials in the this User's Guide, you can use Help menu and the *RISA-3D General Reference Manual* for complete, detailed information on every topic relating to RISA-3D. The topics are arranged in alphabetical order and are thoroughly indexed.

If you are a ***more experienced user*** and are not sure which book will be most helpful your situation, consider that the tutorials in this User's Guide are intended to teach the basics of defining and solving models in RISA-3D, how and when to apply RISA-3D features to help you be most productive, and how to review the results. However, the more in-depth analytical aspects of RISA-3D will be covered in *RISA-3D General Reference Manual* and the Help menu. For example, various material (hot rolled steel, wood, concrete) code checks will be performed in this Guide, but the specifics of how those code checks are calculated is covered in detail in the Help menu and the *RISA-3D General Reference Manual* (specifically the sections titled Hot Rolled Steel Design, Wood Design, and Concrete Design).

Where to Download RISA-3D 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 **RISA-3D**.

Document Conventions

The following conventions are used throughout this book:

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

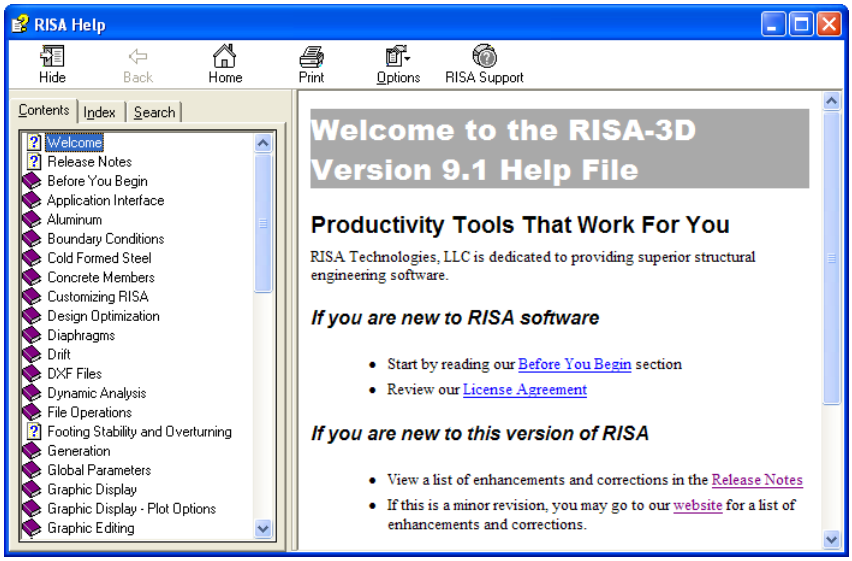
Using the Online Help

Whether you need help on general topics, specific features, or toolbars, it is all built in to the extensive RISA-3D online Help system. The RISA-3D Help menu 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.



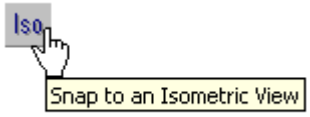
Help on a specific feature (context-sensitive help)

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 Technologies, and is available to all registered licensees at no additional charge. Our knowledgeable support team is ready to answer your most challenging questions.

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

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

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

- Your name, company name, and phone number;
- Product name and serial number or Key ID;
- A detailed problem description; and
- Your model (filename.r3d) as an e-mail attachment. If your model contains multiple members, plates, or load combinations, please specify which ones we should look at.

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

E-mail: support@risatech.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.

Mail:
If you prefer support via mail, please enclose all information listed above, and mail to:

RISA Technologies, Technical Support
26632 Towne Centre Drive, Suite 210
Foothill Ranch, CA 92610

RISA Technologies Online

Visit RISA Technologies 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

RISA-3D Overview

RISA-3D was designed to accelerate the time it takes to perform structural design. Computer-aided structural design is typically a three step process: (1) define your model, (2) solve the model to obtain solution results, then (3) review those solution results to see how the model performed and determine if changes are needed. Usually, this cycle must be repeated several times before arriving at a final, optimized design.

One main reason RISA-3D is able to speed up the design process so successfully is because of its unique ability to define the model and make revisions both graphically (using the drawing tools) and numerically (using the customized spreadsheets).

In RISA-3D, 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. As you perform the step-by-step tutorials in this guide, you will be exploring both methods using the drawing tools and the spreadsheets.

Modeling Tips

Keep in mind that almost all the model data can be edited either graphically (using the drawing tools) or numerically (using the spreadsheets). In most cases, it is simply a matter of preference as to which method you decide to use. Here are a few tips to consider when making your decision:

Some data such as cross-section properties, load combinations, oddball joint coordinates, etc., are most easily entered numerically using a spreadsheet. Other data such as regularly spaced joints, member connectivity, regular loadings, and wholesale changes are better handled using graphic editing tools.

Typically you may want to use the spreadsheets to define some preliminary information such as material and section properties, then proceed to the drawing grid and graphically draw the bulk of your model and apply loads.

Other graphic editing features are built around RISA-3D's graphic selection tools. These features let you graphically edit the model or parts of it. For example, to modify boundary conditions, you would choose the condition you want to apply and select the joints to modify.

Solving the Model and Tips for Reviewing Results

Once you have the model defined, continue by solving the model. RISA-3D features fast 32-bit solution speed, so your model solutions should not take long. Once the model is solved you will be presented with the results.

Once again, you can review your results graphically or with the spreadsheets. Spreadsheet solution results (displacements, forces, stresses, etc.) are listed on the **Results** menu. You can sort and filter these results to find exactly what you want. In the graphic views, click **Plot Options** and choose the results that you want to display (force diagrams, plate contours, deflected shapes, etc.).

Hardware Requirements

Minimum

- Any Windows compatible computer with a Pentium 3 or better processor
- Windows XP\Vista\Windows 7
- 256 MB of RAM
- 200 MB of hard disk space
- Two or three button mouse
- USB port (required for Stand-Alone version or the Network Host computer)

Recommended

- Windows XP\Vista\Windows 7
- As much extended RAM as possible
- As much free disk space as possible
- Two button mouse with wheel

Note: The amount of space needed by RISA-3D to solve a structural model is dependent on the size of the model. In general, 500 MB of RAM is adequate to solve most problems, but the more the better, especially if you will be solving large problems. RISA-3D has been designed to use as much available RAM as possible. If there is not enough RAM, RISA-3D will use hard drive space until enough memory is obtained to solve the problem (causing the solution to run much slower).

Program Limits

Joints	100,000
Members	32,000
Plates	100,000
Solids	100,000
Section Sets	5,000
Materials	500
Custom Wood Species	1,000
Diaphragms	500
Basic Load Cases	1,000
Loads	200,000
Moving Loads	500
Load Combinations	5,000
Mode Shapes	500

Demonstration Version: While you can open and solve a larger model, the largest model that can be saved to disk with the demonstration version is limited to 40 Joints, 40 Members, and 40 Plates.

License Agreement

END-USER LICENSE AGREEMENT FOR RISA TECHNOLOGIES, LLC® SOFTWARE:

The RISA-3D software product (SOFTWARE PRODUCT) includes computer software, the associated media, any printed materials, and any electronic documentation. By installing, copying 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 Technologies, LLC 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 Technologies, LLC within 60 days of purchase for a full refund.

1. SOFTWARE LICENSE. The SOFTWARE PRODUCT is licensed, not sold. All right, title and interest is and remains vested in RISA TECHNOLOGIES, LLC. 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. 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 Technologies, LLC. 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 Technologies, LLC or its officers be liable to anyone for any damages, including any lost profits, lost savings or lost data. In no event will RISA Technologies, LLC 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 Technologies, LLC or its officers have been advised of or should be aware of the possibility of such damages. RISA TECHNOLOGIES, LLC' entire liability shall be limited to the purchase price of the SOFTWARE PRODUCT.

3. LIMITED WARRANTY. RISA Technologies, LLC warrants that the SOFTWARE PRODUCT will operate but does not warrant that the SOFTWARE PRODUCT will operate error free or without interruption. RISA Technologies, LLC sole obligation and your exclusive remedy under this warranty will be to receive software support from RISA Technologies, LLC via telephone, e-mail or fax. RISA Technologies, LLC 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 Technologies, LLC 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. If the PROTECTION DEVICE becomes broken or inoperable it should be returned to RISA TECHNOLOGIES, LLC for a replacement. The replacement will not be provided if RISA TECHNOLOGIES, LLC cannot affirm that the broken PROTECTION DEVICE was originally provided by RISA TECHNOLOGIES, LLC for use with the SOFTWARE PRODUCT. A lost or stolen PROTECTION DEVICE will not be replaced by RISA TECHNOLOGIES, LLC.

5. TERMINATION. RISA TECHNOLOGIES, LLC 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 Technologies, LLC.

6. CHOICE OF LAW. By entering into this Agreement in accordance with Paragraph 1, you have agreed to the exclusive jurisdiction of the State and Federal courts of the State of California, USA for resolution of any dispute you have relating to the SOFTWARE PRODUCT or related goods and services provided by RISA Technologies, LLC. 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 within the State of California, USA. No choice of law rules of any jurisdiction apply.

"RISA" as applied to structural engineering software is a trademark of RISA Technologies, LLC.

Installation

Installation Instructions

To install RISA-3D:

- Place the RISA-3D CD in your computer CD drive. If the CD starts automatically, skip the remaining steps, and follow the on-screen instructions.
- If the CD does not start after 10 seconds, click the Windows **Start** button and select **Run**.
- In the **Run** dialog box, type "**d:\launchsetup**" (where "d" is the label of your CD drive), and then click the **OK** button.
- Follow the on-screen instructions.

RISA-3D Customization--Important Assumption!

Please ensure that when performing these tutorials, RISA-3D has not been customized in any way, and is in the default, installed state. If the installation of RISA-3D has been customized, you may reset the program defaults as follows: on the **Tools** menu, select **Preferences**, click the **Reset Customization Options to Their Original Settings** button.

Maintenance

Program maintenance provides all upgrades to RISA-3D, discounts on new products, and top priority for technical support.

When your maintenance expires, you will be given the opportunity to continue program maintenance on an annual basis. You are under no obligation to continue program maintenance, of course, but if you decide to discontinue maintenance you will no longer receive RISA-3D program upgrades and you will not be eligible for technical support once the version of the program you ended with becomes obsolete.

First Look at RISA-3D

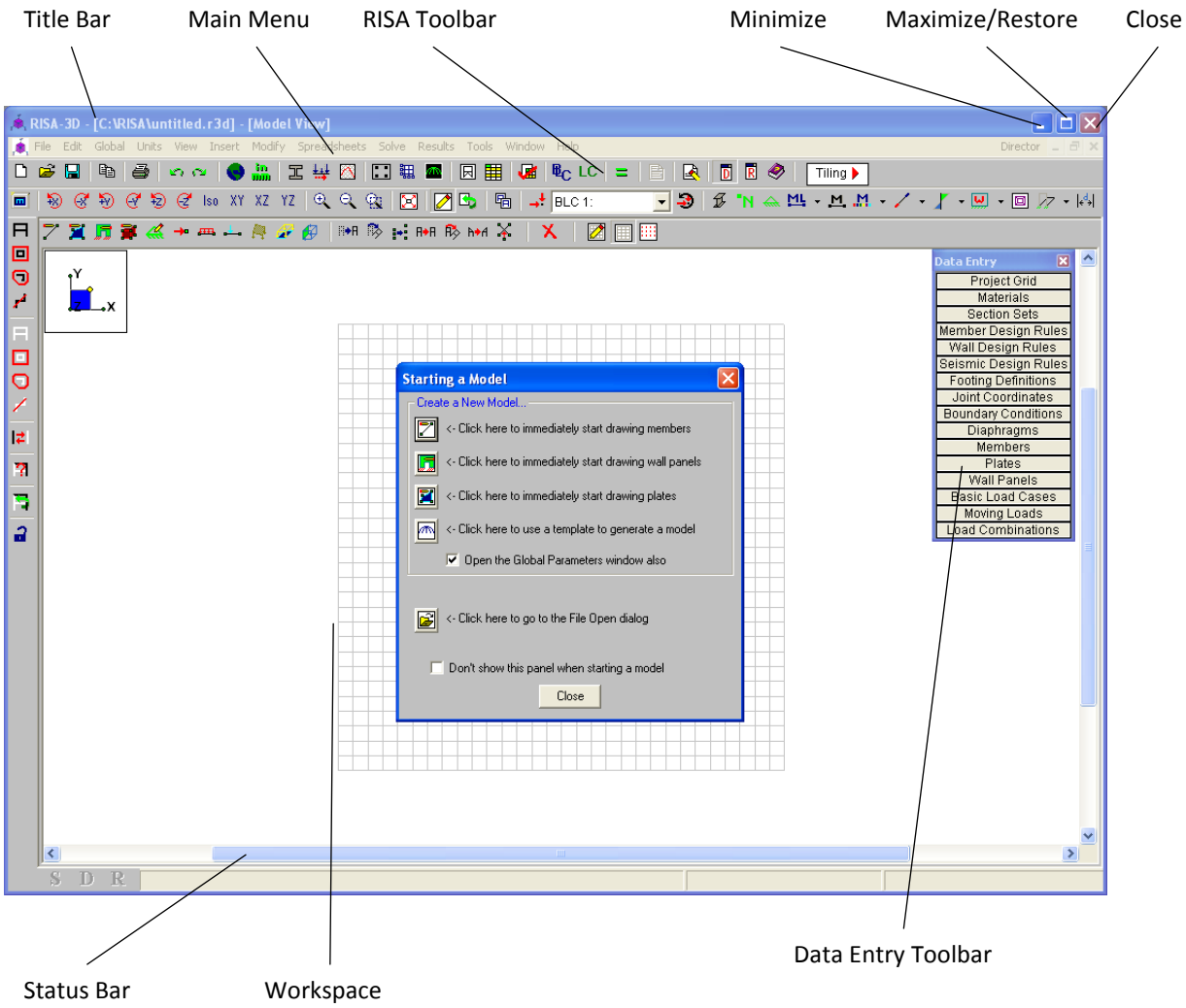
Starting RISA-3D

This section describes the RISA-3D user interface, the toolbars, and shortcuts. We recommend that you review this section before you begin the tutorials. However, if you are already familiar with the RISA-3D user interface, feel free to skip ahead to Tutorial 1 on page 17.

Start RISA-3D as follows:

- On the **Start** button, click **All Programs**, select **RISA**, then select **RISA-3D**.

The RISA-3D Windows and Dialog Boxes



Title bar

The title bar at the top of your RISA-3D 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 RISA-3D 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.

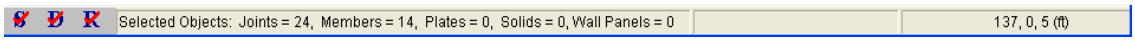
Workspace

The actual work that you do in RISA-3D will be in the main area on the screen, the workspace. Currently the workspace contains a white model view with the default drawing grid and the **Starting a Model** options (you will see how to change the default grid and many other defaults as you go through the tutorials). 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 in RISA-3D. The bar has four parts:

- The left side of the bar has the letters “S”, “D”, and “R” to indicate the solved state of the model for Static, Dynamic, and Response Spectra solutions. If the letters are dimmed, a solution has not been performed. After performing a solution, the letters will become blue in color with a red checkmark on them (as shown below).



To the right of the solved status flags are three status boxes which display information while you work:

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

Dialog boxes

Dialog boxes are windows that help you perform a specific function within RISA-3D. For example, on the **File** menu, click **Open** and the dialog box below will display which helps you locate the file you would like to open. Many of the dialog boxes also contain **Help** buttons that will direct you to the related topic in Help.

Menus

Main Menu

The Main menu and its submenus provide access to all features RISA-3D has to offer, as summarized below:

File	Provides access to file operations such as opening, saving, importing, exporting, and appending files.
Edit	Provides editing tools that help you modify and manipulate the spreadsheets. You may use this menu to add or remove information from the spreadsheets or to sort and mathematically manipulate current spreadsheet data.
Global	Allows you to set options that apply to the entire model such as design codes, shear deformation and warping effects, p-delta options, model descriptions, and notes.
Units	Allows you to set units or convert existing units.
View	Allows you to open a new model view or adjust the current model view.
Insert	Used to insert joints, members, plates, 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 while the **Spreadsheets** menu provides access to the spreadsheets.
Modify	Allows access to the graphic editing features and may be used to modify existing joints, members, plates, and loads.
Spreadsheets	Opens the spreadsheets.
Solve	Allows access to the analysis options provided with RISA-3D.
Results	Allows access to all analysis result spreadsheets. This button is dimmed when no results are available, such as before you run a solution.
Tools	Provides tools to help you organize, identify, and correct problems as you model the structure. **Preferences** are also located here.
Window	Manages all of the windows that you have open in RISA-3D, whether they are spreadsheets or model views. Special tiling options are also available that relate to specific modeling tasks.
Help	Allows access to the RISA-3D online Help. For more information on this, see the Using the Online Help section of the Help menu. If you are connected to the internet, you may also check to see if any product updates are available.
Director	If your RISA-3D model is linked to a RISAFloor model or a RISAFoundation model, this menu allows you to switch between programs.

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.

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

Note: You will discover many options to access the features available in RISA-3D. 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 Block Fill and Block Math.

Drawing Toolbar

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

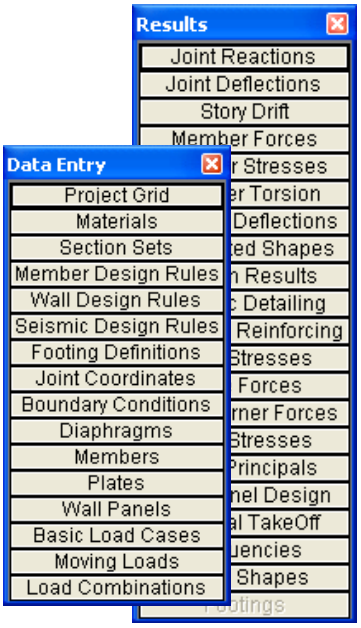
Selection Toolbar

…only visible in Model View

The Selection toolbar is the vertical toolbar along the left side of the screen. It provides selection tools to help you work with parts of the model.

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

Spreadsheet Toolbars (Data Entry and Results Toolbars)



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

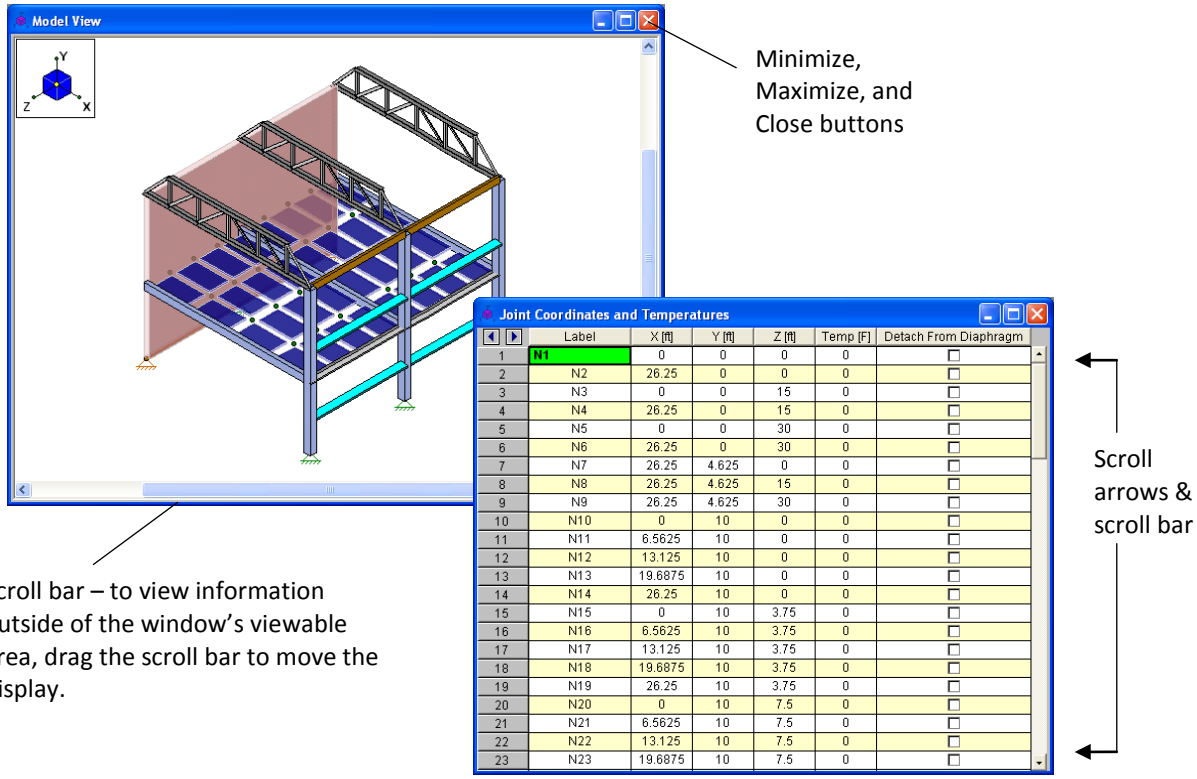
The **Data Entry** toolbar is a vertical toolbar on the right of your screen. It looks different than the other toolbars because its buttons consist of text instead of images.

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

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

Managing Windows

As you work in RISA-3D, 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 RISA-3D 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.

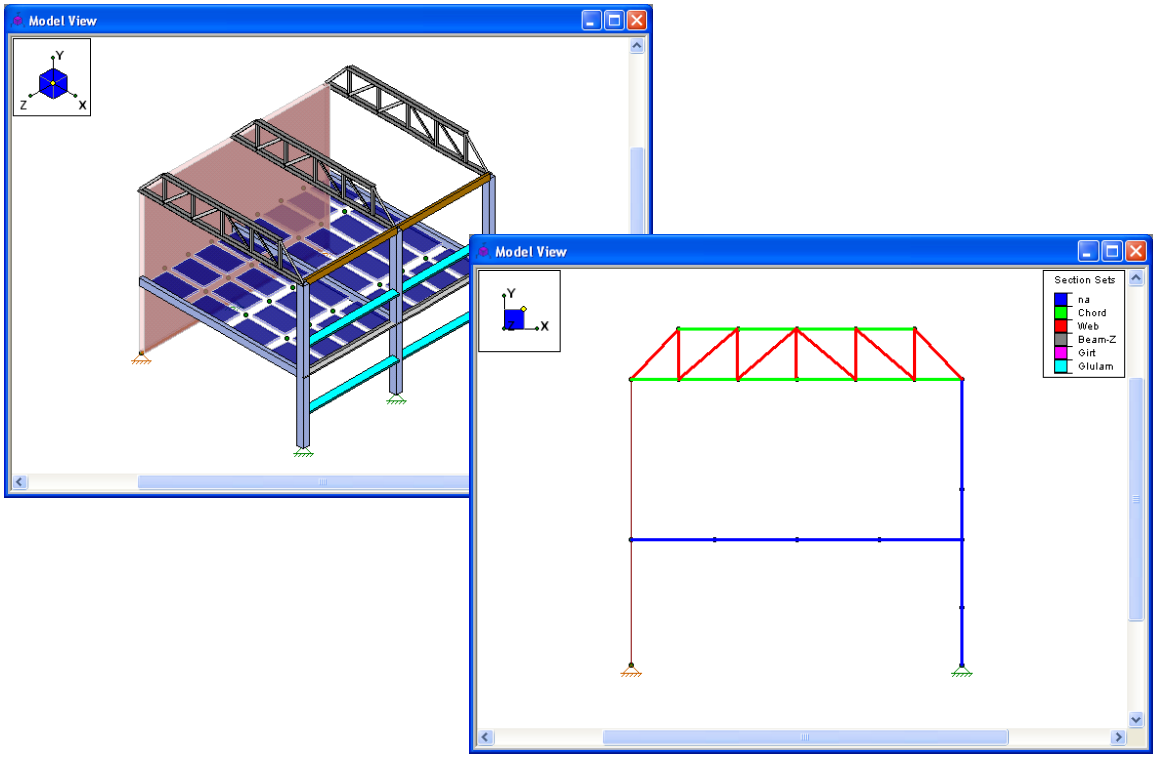


Managing Model Views

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 RISA-3D vary depending on which window is active (the window with a colored title bar is the active window).

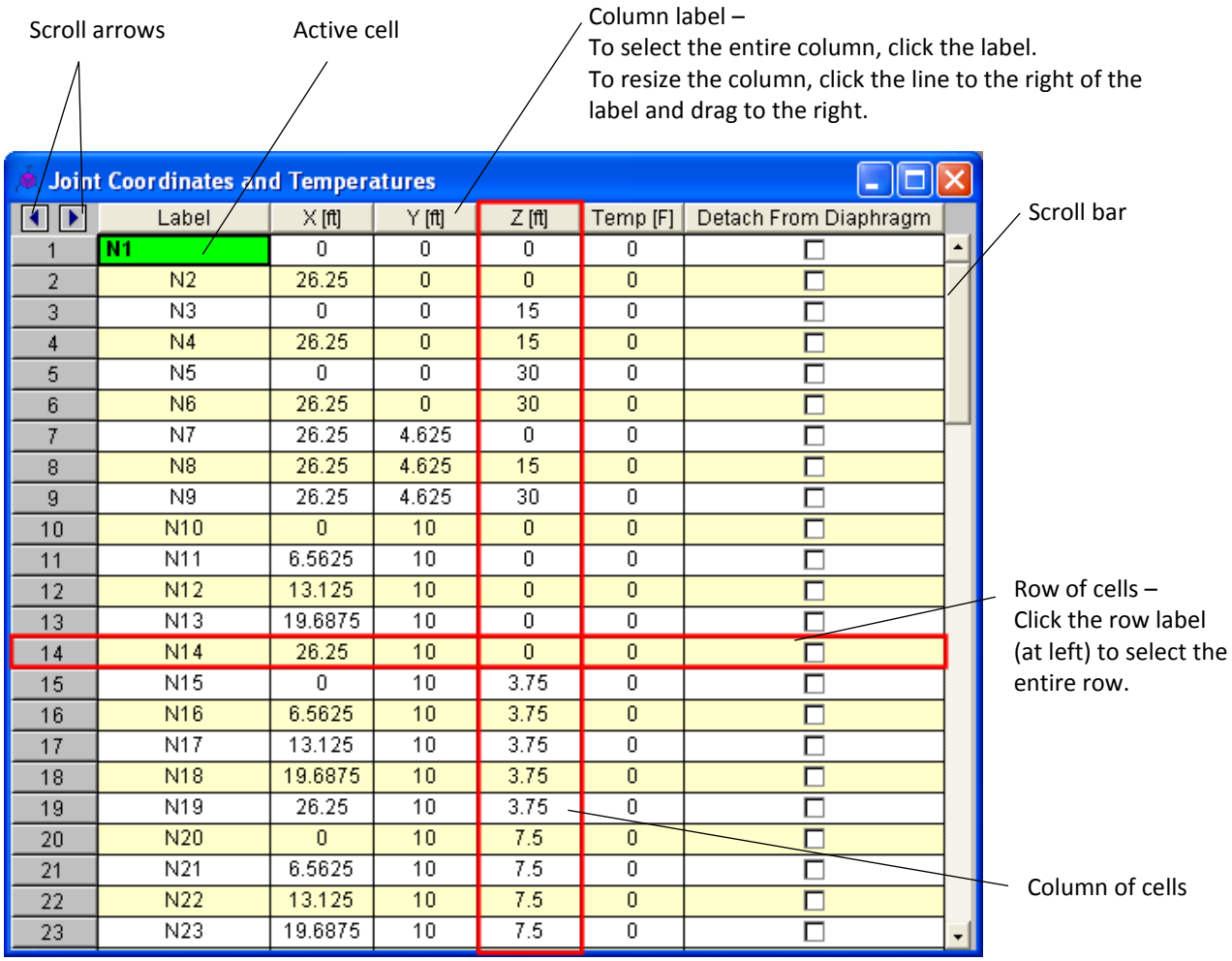
For example, if your active window is a spreadsheet, and you are looking for the zoom toolbar, you will not be able to locate it until you click on your model, switching to model view. Then you will be able to access the zooming tools, and all the other tools related to modeling.



Working in Spreadsheets

Spreadsheets provide a means to enter and edit model data numerically. Spreadsheets are comprised of rows and columns of cells. To add or edit data in a cell, click on the cell, making it the active cell, then type. There is always only one active cell at a time, and is indicated with green color. Notice 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.



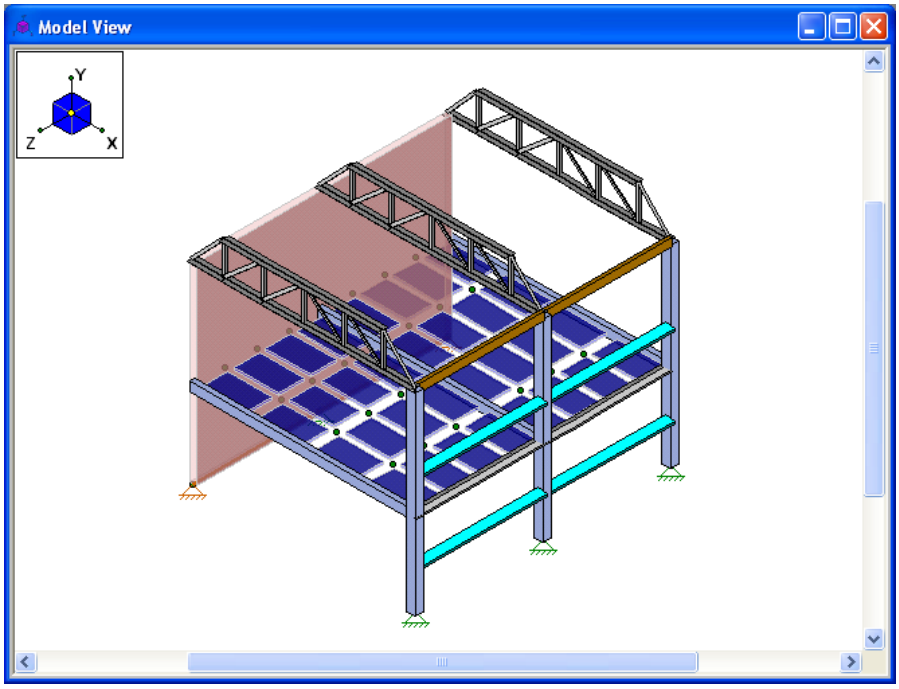
Joint Coordinates and Temperatures

	Label	X [ft]	Y [ft]	Z [ft]	Temp [F]	Detach From Diaphragm
1	N1	0	0	0	0	☐
2	N2	26.25	0	0	0	☐
3	N3	0	0	15	0	☐
4	N4	26.25	0	15	0	☐
5	N5	0	0	30	0	☐
6	N6	26.25	0	30	0	☐
7	N7	26.25	4.625	0	0	☐
8	N8	26.25	4.625	15	0	☐
9	N9	26.25	4.625	30	0	☐
10	N10	0	10	0	0	☐
11	N11	6.5625	10	0	0	☐
12	N12	13.125	10	0	0	☐
13	N13	19.6875	10	0	0	☐
14	N14	26.25	10	0	0	☐
15	N15	0	10	3.75	0	☐
16	N16	6.5625	10	3.75	0	☐
17	N17	13.125	10	3.75	0	☐
18	N18	19.6875	10	3.75	0	☐
19	N19	26.25	10	3.75	0	☐
20	N20	0	10	7.5	0	☐
21	N21	6.5625	10	7.5	0	☐
22	N22	13.125	10	7.5	0	☐
23	N23	19.6875	10	7.5	0	☐

Tutorial 1 – Modeling

This is the first tutorial in this series designed to guide you through the entire modeling process using many RISA-3D features. The tutorials are intended to simulate the "real world" design process. Just as in the real world, you will enter the data, solve the model, review results, then go back and make changes, etc. We believe the real test of any solutions software is in the design modification—not just the initial design. See for yourself how easily you can design *and change* your model using RISA-3D.

In the tutorials ahead, you will be modeling this structure:



As you perform the step-by-step tutorials, you will make use of both the spreadsheet and graphic drawing capabilities of RISA-3D. In Tutorial 1, you will create a 2-D frame. In Tutorial 2, you will "extrude" this into a 3D model, and add additional members. Tutorial 3 will show how to define loading. In Tutorial 4, you will run a static solution and review the results. Tutorial 5 will demonstrate dynamic analysis and seismic design. Then, Tutorial 6 will explore the various options for ineroperability of RISA-3D with other programs such as other RISA programs, Revit Structure, and CAD programs.

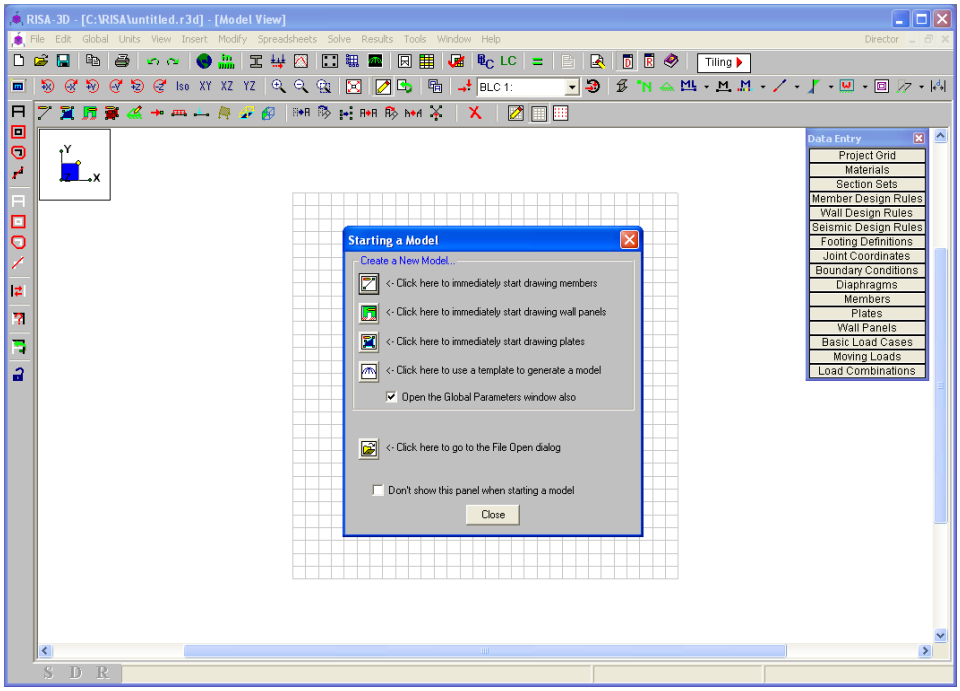
The tutorials build upon themselves from start to finish. You have the option of performing them all at one time, or performing each one separately. To make this possible, RISA provides model files for you to load at the beginning of each tutorial. These starter files are located in the RISA program folder under **Tutorials**, and are named **Tutorial 2 starter.r3d**, **Tutorial 3 starter.r3d**, etc.

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 here:

- 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.

Now, if you are ready to get started, start RISA-3D if you have not already done so:

- Double-click the **RISA-3D** icon to start the program. The **Starting a Model** dialog box will display, which allows you to create a new model or open an existing file.



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

Starting a Model Options

The **Starting a Model** dialog box presents you with several startup options, along with a blank drawing grid placed in the workspace. You can choose to start drawing your model (by defining members, plates, or using a template to generate it automatically), open an existing model, or work on your own (by clicking **Close**).

Automatically Generating with Templates

RISA-3D has powerful templates that automate the generation of structures. Although these templates are not used in the tutorials, we believe that once you understand how to model structures without the templates, you will easily understand the benefit of the templates for your future modeling projects, making RISA-3D even more powerful.

These templates can be accessed as follows:

On the **Insert** menu, click **Structure (Generate).**

-OR-

On the RISA toolbar, click the **Perform High Level Generation** button .

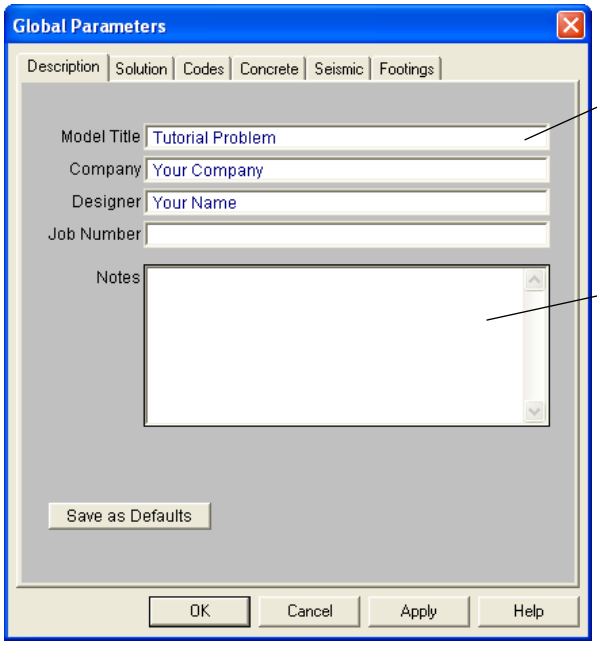
Some of the templates are very specific, such as the Rectangular Tank with Stiffeners or the General Truss generator. Some are general, such as the Circular Disk With Plates. Using these templates, you can generate almost any typical or repetitive model or model component.

Create a New Model

You will now begin to model your structure by first creating a 2D frame, defining your Global Parameters, your Drawing Grid, and be introduced to some basic graphical modeling.

- In the **Starting a Model** dialog, under **Create a New Model**, make sure the **Open the Global Parameters window also** check box is selected. This will save you a step by opening the **Global Parameters** dialog box once you make your starting selection.
- Select **Click here to immediately start drawing members**  .

The **Global Parameters** dialog box should now display.



Tip:
Press TAB to advance to the next field.

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

Global Parameters are settings that apply to the model as a whole.

Set Global Parameters

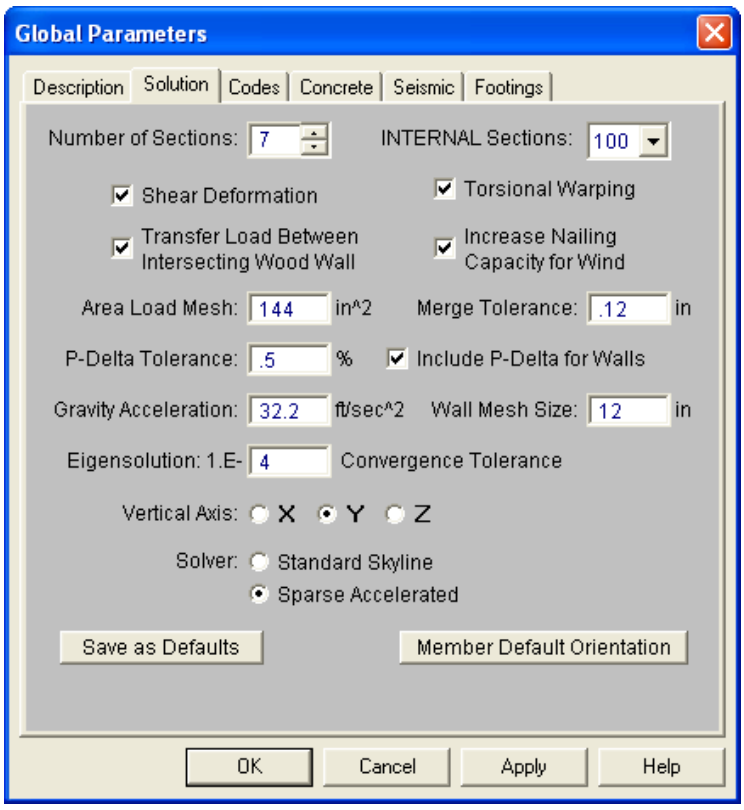
Set the Global Parameters as follows:

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

Review and modify the Solution parameters:

- Click the **Solution** tab.
- In the **Number of Sections** box, type **7** (or you may use the up/down arrows to increase/decrease the value). Make sure the the **INTERNAL Sections** selection is set to **100**.
- Click **Apply**.

The dialog box will remain open, and should now look like this:



Below is a description of all the available Solution parameters:

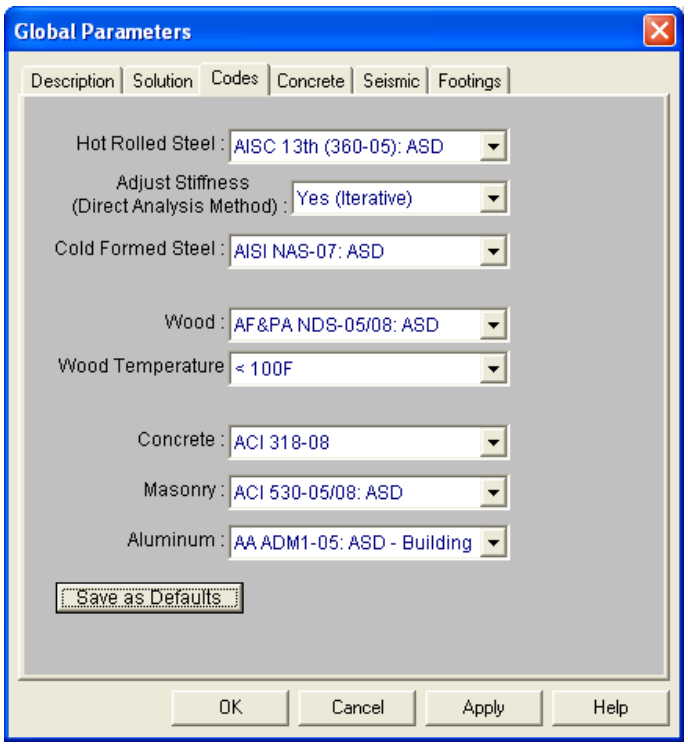
Number of Sections	Specify how many locations along each member are to be used for the reporting of forces, stresses, and deflections. For this tutorial, you will specify the calculations be reported at 7 section locations.
Internal Sections	Controls how many places along each member RISA-3D calculates and stores results, such as deflections and code checks. The member force diagrams displayed in the model view and the detail plot are also drawn from these results.
Shear Deformation	This check box indicates whether or not shear deformation considerations are to be included in the model solution. This check box is usually selected and is the default setting.
Torsional Warping	This check box indicates whether or not torsional warping effects are considered when calculating stiffness and stress values for wide flange and channel shapes.
Transfer Load Between Intersecting Wood Wall	This check box indicates whether or not to transfer loads between perpendicular wall panels framing into each other.
Increase Nailing Capacity for Wind	This checkbox indicates if you want to increase the shear capacity of wood wall panels and diaphragms by 1.4 for all load combinations that contain wind loads.

Area Load Mesh	Controls how the load is broken down for attribution to the members.
Merge Tolerance	Defines the maximum distance two joints can be apart from each other yet still be merged together. Also used when scanning for crossing members and for unattached joints along the spans of members.
P-Delta Tolerance	Used to set the convergence tolerance for the P-Delta analysis.
Include P-Delta for Walls	This checkbox indicates whether or not to include P-Delta analysis for wall panels. If this box is checked, the P-Delta analysis will only be performed on load combinations which include P-Delta.
Wall Mesh Size	Controls the base mesh size to be used when wall panel elements are solved.
Gravity Acceleration	Used to convert loads into masses for the purpose of dynamic analysis.
Eigensolution Convergence Tolerance	Used to set how close a subsequent solution must be to the previous solution for a mode to be considered converged (during dynamic analysis).
Vertical Axis	Indicates which of the three global axes (X, Y or Z) is to be considered the vertical axis.
Solver	Select the solver to be used during solution. Please see the help menu for more information on this option.

Review the options for the Code parameters:

- Click the **Codes** tab.

For this tutorial, the default settings will be used.

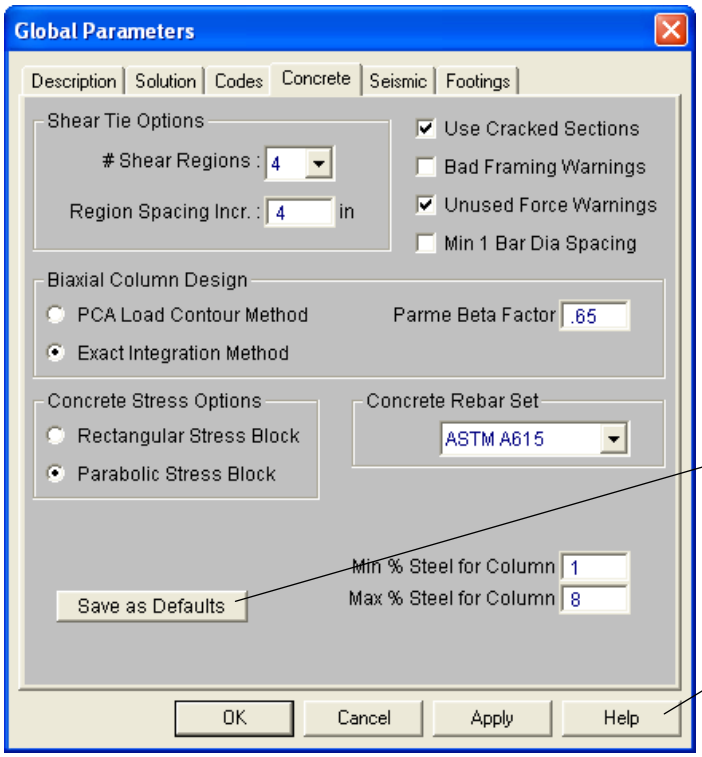


These settings control how code checking is performed, as described below:

Hot Rolled Steel	Code choices include *AISC 13th (360-05) ASD & LRFD, 9th Edition ASD, 2nd Edition LRFD, 3rd Edition LRFD*, and several international steel codes. If the *AISC 360-05* code is selected (either ASD or LRFD), the **Adjust Stiffness** menu will appear. The options in this menu control the stiffness reduction provisions per Appendix 7 of the code.
Cold Formed Steel	Code choices are *AISI NAS-07: ASD & LRFD, AISI NAS-04: ASD & LRFD, AISI NAS-01: ASD & LRFD, AISI 1999: ASD & LRFD*, and several international codes.
Wood	Code choices are *AF&PA NDS-05/08:ASD, AF&PA NDS-01:ASD, AF&PA NDS-97: ASD.*
Concrete	Code choices are *ACI 318-08, ACI 318-05, ACI 318-02, ACI 318-99*, and several international codes.
Masonry	Code choices are *ACI 530-05/08:ASD, ACI 530-05/08:Strength, ACI 530-02:ASD, ACI 530-02:Strength, ACI 530-99:ASD,UBC 1997: ASD & Strength.*
Alumninum	Code choices include *ADM1- 05:ASD Building & Bridge*. The Building and Bridge options refer to the safety factors that are used in the model.

Review and modify the options for the Concrete parameters:

- Click the **Concrete** tab.
- Under **Concrete Stress Options**, click **Parabolic Stress Block**. This method is the more accurate of the two methods.
- Click **Apply**.



Save as Defaults –

At any time during the modeling process, you can click this button to save all the settings to become the new default settings for all future models.

Click the **Help** button to open the Help Menu to the relavent topic.

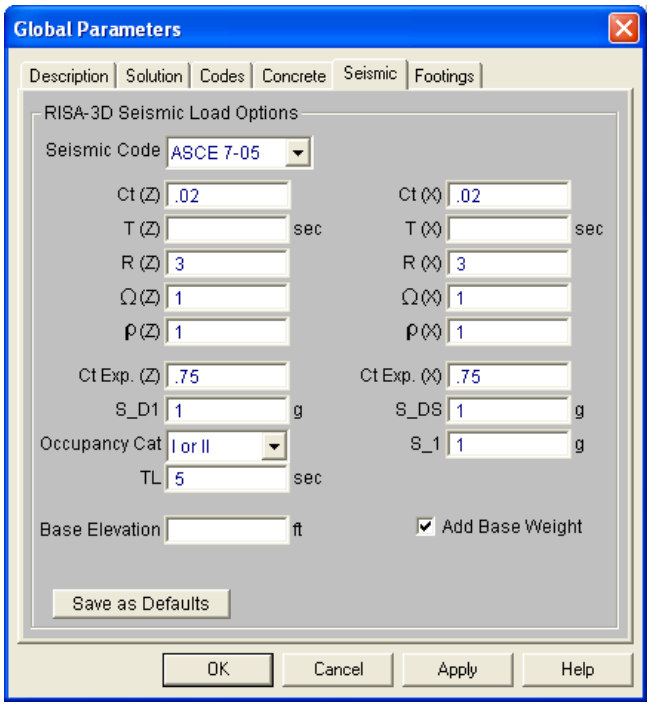
These settings contain options related to the analysis and design of concrete members as described below:

Shear Tie	Allows you to control the number of shear regions that will be used when detailing a beam or column span. It also allows you to specify an increment when increasing or reducing the spacing of the shear ties.
Biaxial Column Design	Controls which method will be used to determine the biaxial column capacity: the Exact Integration Method is more accurate than the PCA Load Contour Method and does not require you to make an estimate of the Parme Beta Factor.
Concrete Stress Options	Allows you to choose what type of stress block to consider in your analysis. The options are the constant Rectangular Stress Block and the Parabolic Stress Block. Since the Parabolic Block is more accurate, you will select this stress distribution for your design.
Concrete Rebar Set	Specify the grade of rebar you wish to use. You can also specify a Minimum % Steel and Maximum % Steel for column reinforcement.
Concrete check boxes	**Used Cracked Sections** - modifies the member stiffnesses by the Icr factor. For more information, refer to *RISA-3D General Reference*. **Bad Framing Warnings** - helps you detect modeling errors during solution. It produces a message in the error log whenever an unsupported member is detected at either of its end joints (can be common for models that contain cantilevers, rafters, etc.) **Unused Force Warnings** - produces messages on the member detail report when weak axis and torsional moments have been neglected. **Minimum 1 Bar Dia Spacing** - allows a minimum spacing of one bar diameter between parallel bars. Otherwise, RISA-3D will default to a two bar diameter or one inch clear spacing, whichever is greater, to allow for lap splices and continue to maintain adequate spacing between parallel bars.

Review and modify the options for the Seismic parameters:

- Click the **Seismic** tab.

For this tutorial, the default settings will be used.



This tab contains options that are specifically related to the calculation of code presecribed seismic loading, and seismic detailing checks.

This information can be used to generate the lateral loading on your structure if you choose to use the Seismic Load Generator.

You will not use this feature in this tutorial, however you may access the Help menu for more information on this feature.

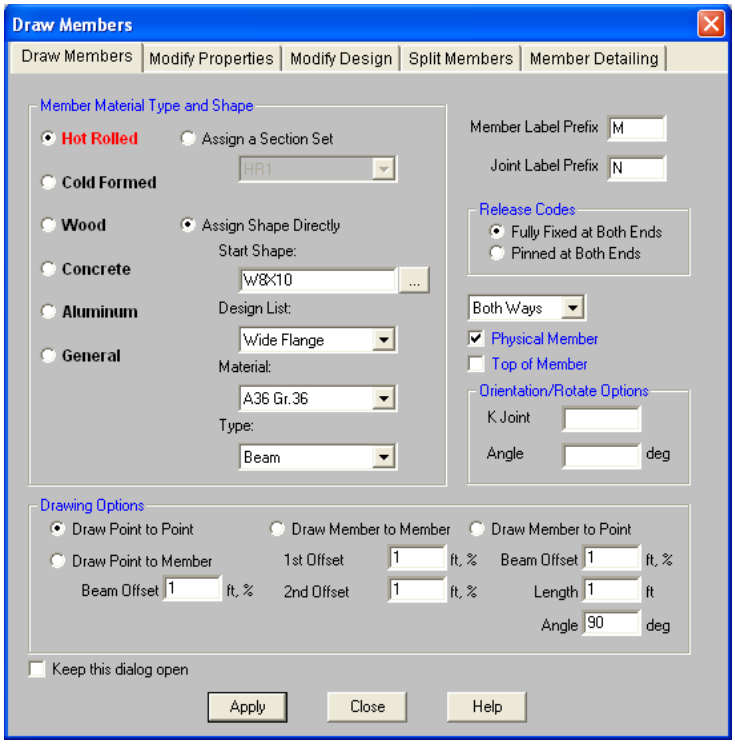
Before closing the **Global Parameters** dialog box, notice you will also see a tab labeled **Footings**. If you have RISAFoot installed, this will allow you control over footing design input. If you do not have RISAFoot installed, this tab will only state that you need to install the program in order to utilize the integration. For more information on this integration, please click **Help** or refer to *RISA-3D General Reference Manual*.

You may now close the **Global Parameters** dialog box.

- Click **OK** to exit.

Members

Once you close the **Global Parameters** dialog box, the **Draw Members** dialog box will display:



Before drawing the members, first define the material type to be used: Hot Rolled, Cold Formed, Wood, Concrete, Aluminum, or General. Once the materials are stored in RISA-3D, the spreadsheets and solution results will be sorted based on that material. When performing a solution, RISA-3D will only analyze and/or design members for load combinations that are appropriate for that material (this will be discussed in more detail in Tutorial 3 – Loading).

Define concrete as the material type for your new members you will be creating:

- Under **Member Material Type and Shape**, click **Concrete**.

Next, you need to decide how you want to define the cross-sectional properties of the member. You can assign a shape directly, or define your properties based on section sets.

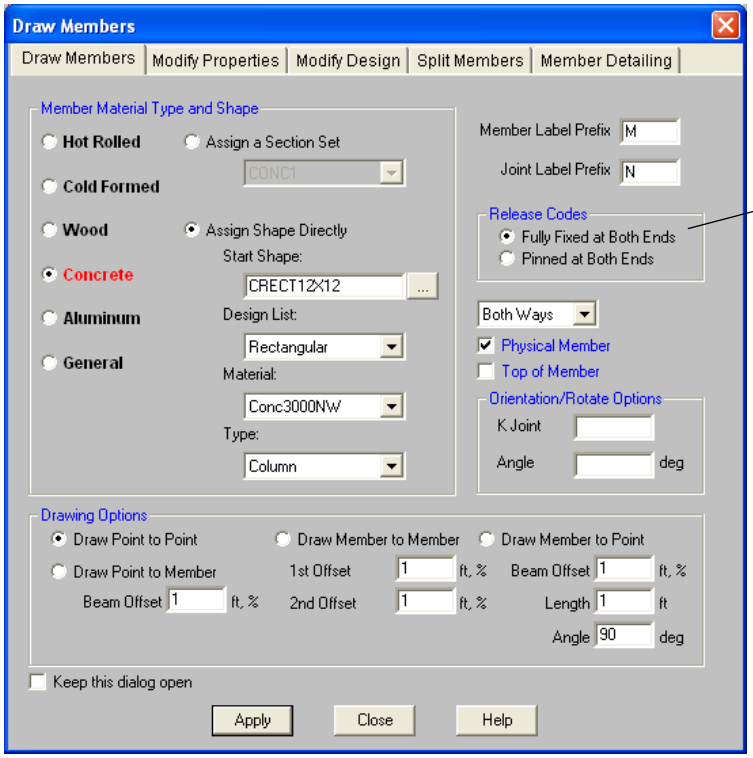
For now, start with a 12x12 column, and assign your concrete shapes directly. (Later you will use Section Sets for your wood truss and steel members.)

- Click **Assign Shape Directly**.

This bypasses the Section Sets and allows you to assign an explicit shape for these new members:

- Under **Start Shape**, type: **CRECT12X12**
- Under **Design List**, select **Rectangular**.
- Under **Type**, select **Column**.

The dialog box should now look like this:



The **Release Codes** selection designates how each member is considered connected at its end joints.

Tip:
Use Release Codes to specify how members are attached to one another. Later, you'll explore using Boundary Conditions to specify how the structure is attached to its external points of support (reactions).

Member Releases

You may now begin drawing columns.

- Click **Apply** to begin drawing columns.

Note: When you enter the drawing mode, your cursor changes to  . Also, on the Drawing toolbar, notice the **Draw New Members** button is pressed in  .

Drawing Grid

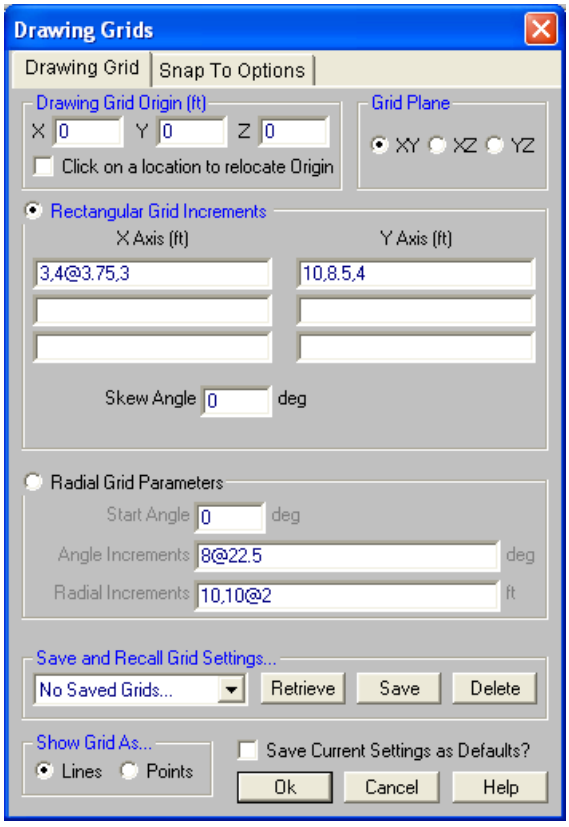
Before you begin drawing, notice that RISA-3D will always open a default 30x30 grid spaced at one foot. To make it easier to find your column and beam locations for this tutorial, redefine the grid as follows:

- On the Drawing toolbar, click the **Modify Drawing Grid** button  to redefine the grid. The **Drawing Grid** tab should already be selected.

The drawing grid may be specified in plane or in either elevation. For this model, leave the drawing grid in the XY plane, and change the grid settings as follows:

- Under **Rectangular Grid Increments**, in the first column under **X Axis**, highlight and delete any current data and type: **3,4@3.75,3** (including the commas).
- In the first column under **Y Axis**, highlight and delete any current data and type: **10,8.5,4** (including the commas).

The dialog should now look like this:



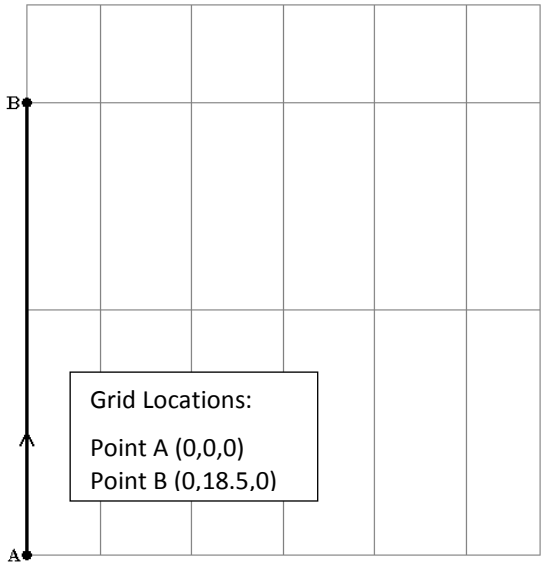
Close the **Drawing Grids** dialog box and the **Data Entry** toolbar.

- Click **OK** to close the dialog box.
- Click **Close** to close the **Data Entry** toolbar (upper right corner of your workspace).

Drawing Members

As you move your cursor around the drawing grid (specifically over grid intersections), notice RISA-3D displays the coordinate location of the cursor in the lower right corner of the Status bar.

The coordinates displayed on the Status bar are the coordinates of the grid point or joint location closest to the cursor. Notice the coordinates change as you move the mouse around. Also, when you are in the drawing mode (as you are now), the points are highlighted by a red asterisk any time the tip of your mouse approaches the point.



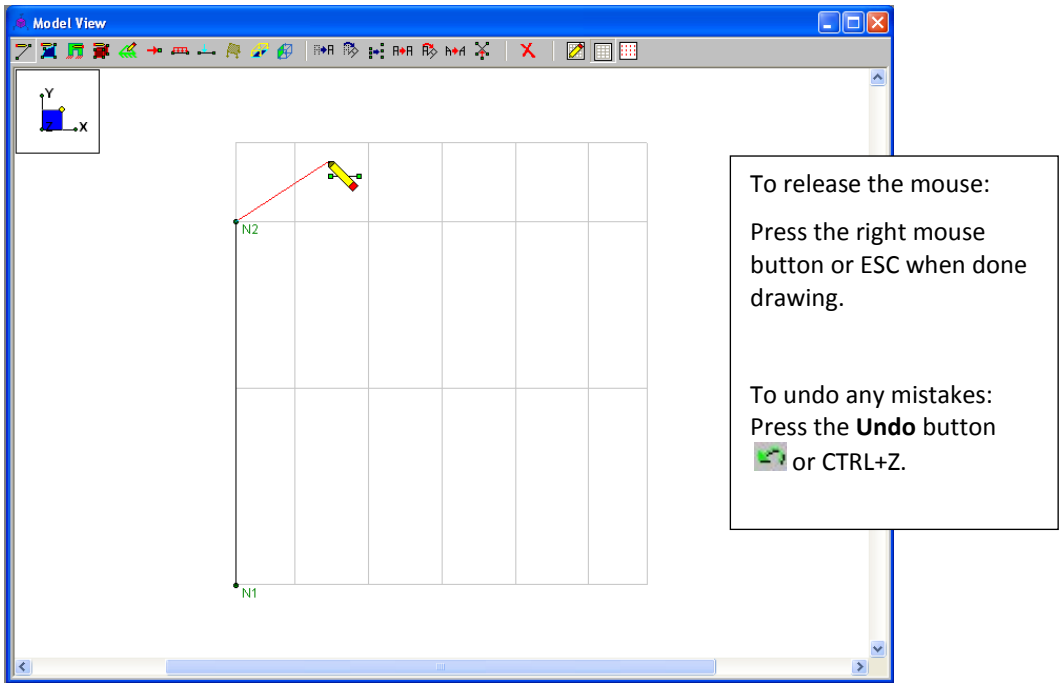
As you continue the tutorial, ensure that you draw the members in the specified location by watching the Status bar to verify the exact coordinate location.

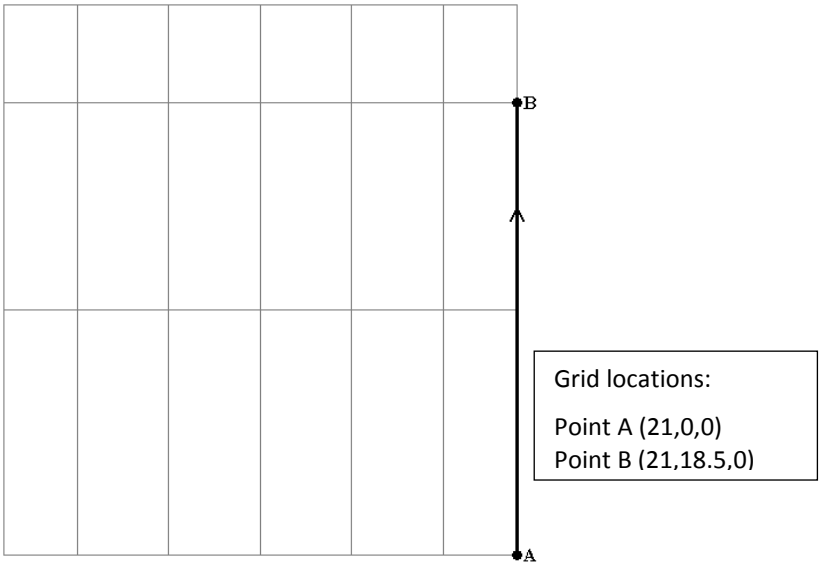
Start by drawing a line from point A to point B, as shown in the image above:

- Place the cursor on the lower left corner of the grid to create point A (coordinates **0,0,0**) and click once. (Notice a joint has been created and labeled **N1**.)
- Next, move the mouse straight up to point B (coordinates **0,18.5,0**) and click again. (Another joint has been created and labeled **N2**.)

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

- When finished drawing the line, press the right mouse button to release the mouse.





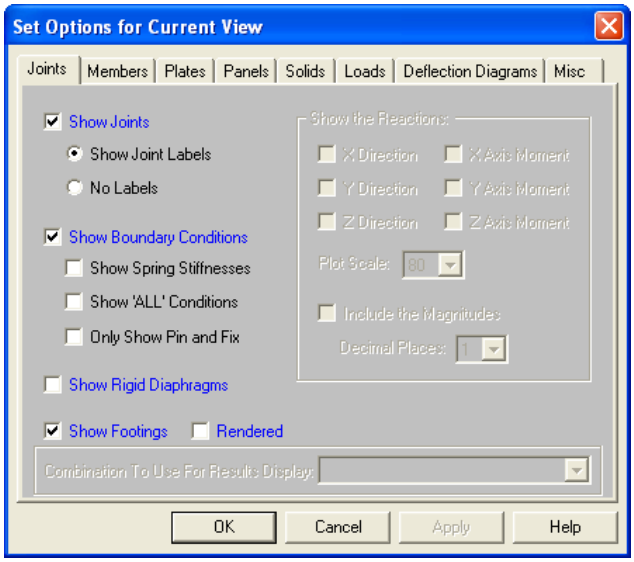
Draw another line from point A to point B, as shown in the image above:

- To create point A, place the cursor on the lower right corner of the grid (coordinates **21,0,0**) and click once. (The new joint has been created and labeled **N3**.)
- Next, move the mouse straight up to point B (coordinates **21,18.5,0**) and click again. (The new joint has been created and labeled **N4**.)
- When finished drawing the line, press the right mouse button to release the mouse. Click the right mouse button a second time to exit the drawing mode.

Now you should have a column on the left, extending from joint N1 to joint N2, and another on the right, from joint N3 to N4.

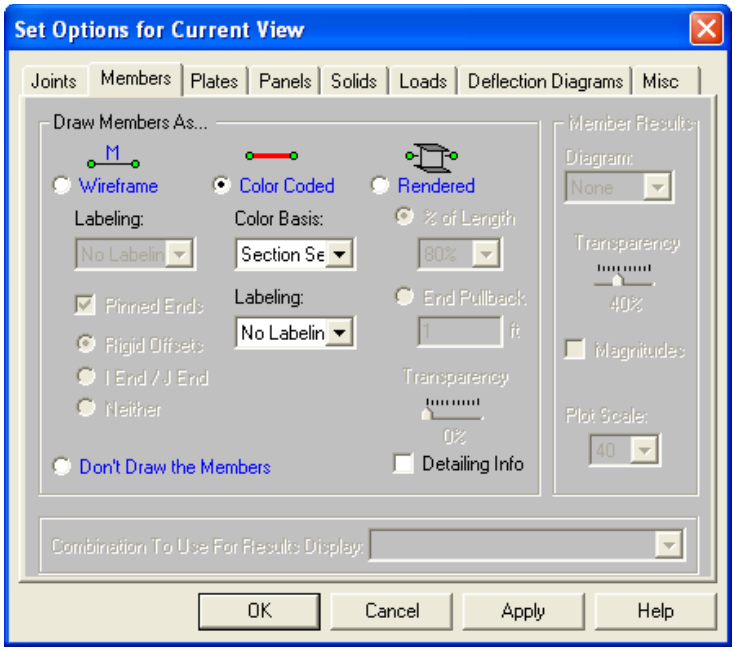
Before drawing the truss, take a moment to change the way that members are shown and color-code them by their section sets.

- On the Window toolbar, click **Plot Options**  .



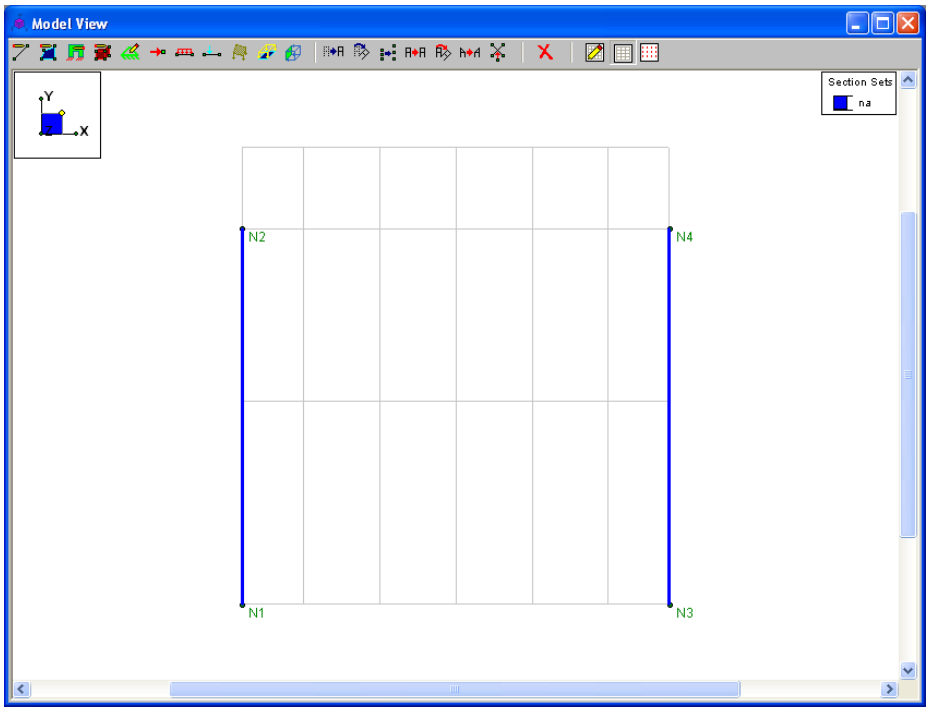
The Plot Options will be discussed in more detail later. For now, just change the color coding:

- In the dialog box, click the **Members** tab.
- Click **Color Coded**. Under **Color Basis**, select **Section Set**. Under **Labeling**, select **No Labeling**.



- Click **OK** to close the dialog box and return to the plot.

The members should now be displayed in color and a legend is now displayed in the upper right corner of the workspace (underneath where the **Data Entry** toolbar was). Notice that the description appears in the legend as "**na**" because the Section Sets have not yet been defined.

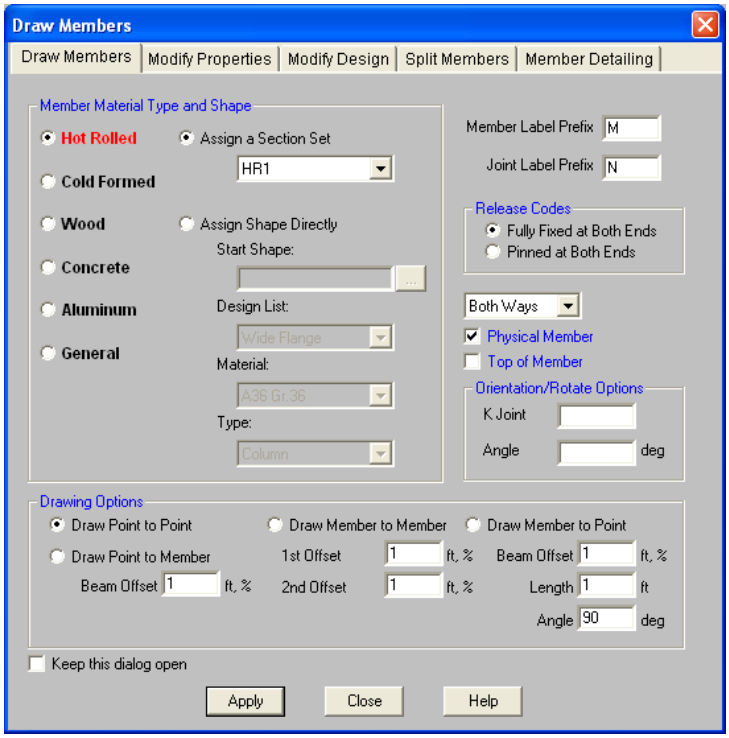


Now that the columns are defined, move ahead to the truss members. To resume drawing:

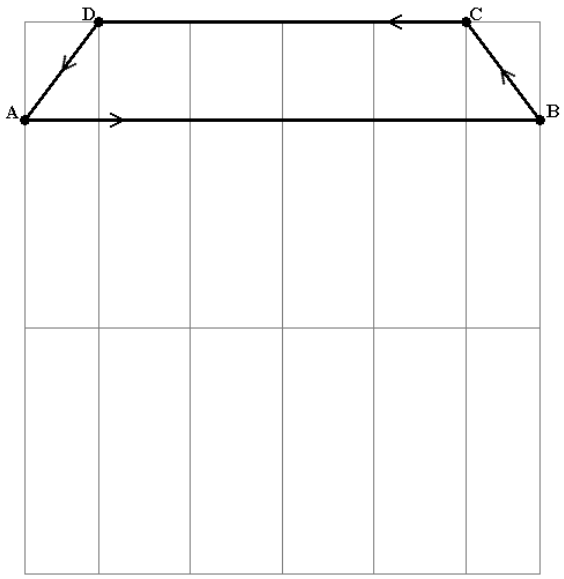
- On the Drawing toolbar, click the **Draw New Members** button  .

You will now define your truss members as you would a section set. This allows you to group similar members for easier modeling and modifying. Because you have not yet defined any section sets, you will first place the members in the sets as you build the model, then define the properties (section sets) later.

- Under **Member Material Type and Shape**, select **Hot Rolled**.
- Then click **Assign a Section Set** to assign a section set for your new members. (The available options for Hot Rolled Steel will appear.)



- Click **Apply** to return to the model view and resume drawing.



Grid Locations:

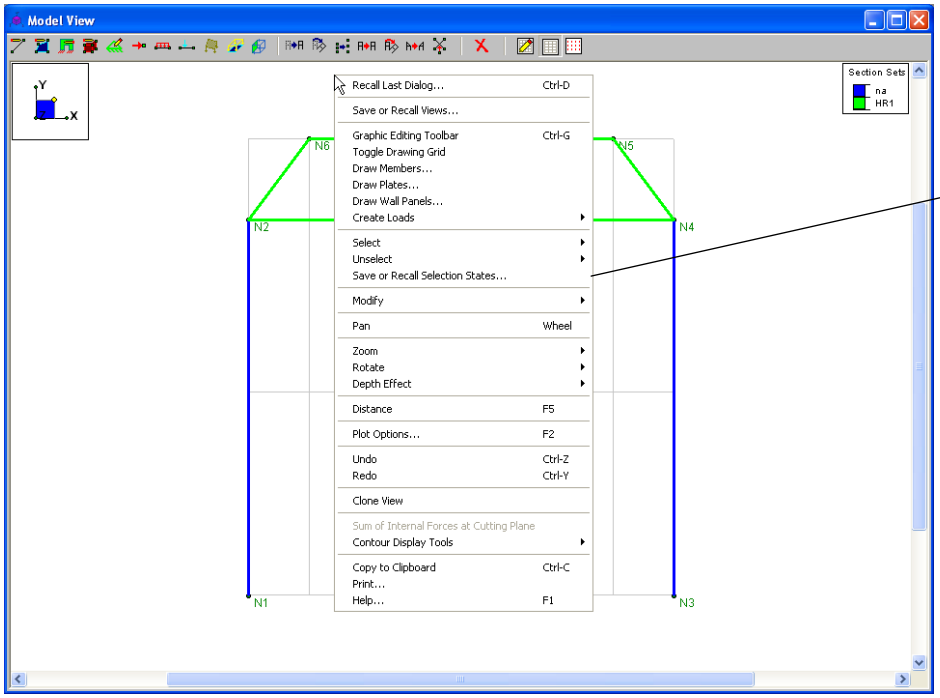
Point A (0,18.5,0)
Point B (21,18.5,0)
Point C (18,22.5,0)
Point D (3,22.5,0)

Draw the truss chord, as shown on the image above:

- Draw members from points A to B, then B to C, then C to D, and lastly from D to A.
- Click the right mouse button or ESC to release the mouse.
- Click the right mouse button or ESC again to exit the drawing mode.

Next, you will draw the web members using the HR1 section set again. Later, you will modify this to differentiate the web members from the truss chords using separate section sets.

- Click the right mouse button (from anywhere in your workspace) to activate the shortcut menu.

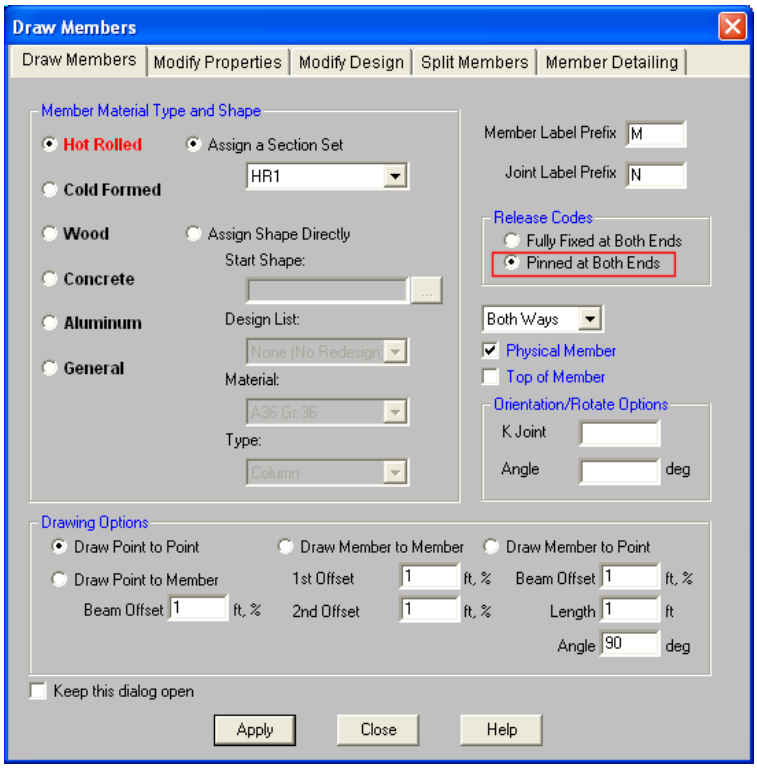


Press the right mouse button to activate the Shortcut menu.

The Shortcut menu displays options available that relate to the tasks you are performing. These options are specific to your current view. For example, if you were viewing a spreadsheet, the choices would be relevant to spreadsheets.

- Select **Recall Last Dialog** to display the **Draw Members** dialog box again.
- Under **Release Codes**, click **Pinned at Both Ends**.

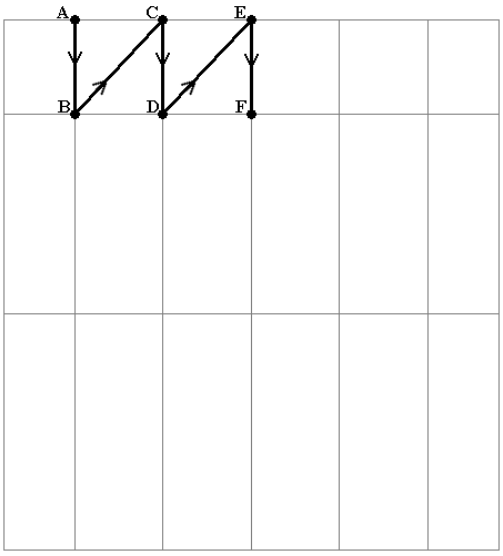
The dialog box now looks like this:



Exit the dialog box:

- Click **Apply**.

Draw the web members, as shown below:

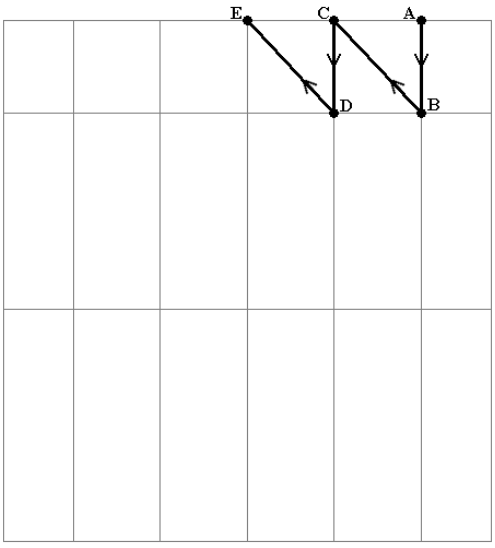


Grid Locations:

Point A (3,22.5,0)
Point B (3,18.5,0)
Point C (6.75,22.5,0)
Point D (6.75,18.5,0)
Point E (10.5,22.5,0)
Point F (10.5,18.5,0)

- Draw members from points A to B, B to C, C to D, D to E, then lastly from E to F.
- Click the right mouse button or press ESC to release the mouse.

Choose a new starting point for the next series of members.



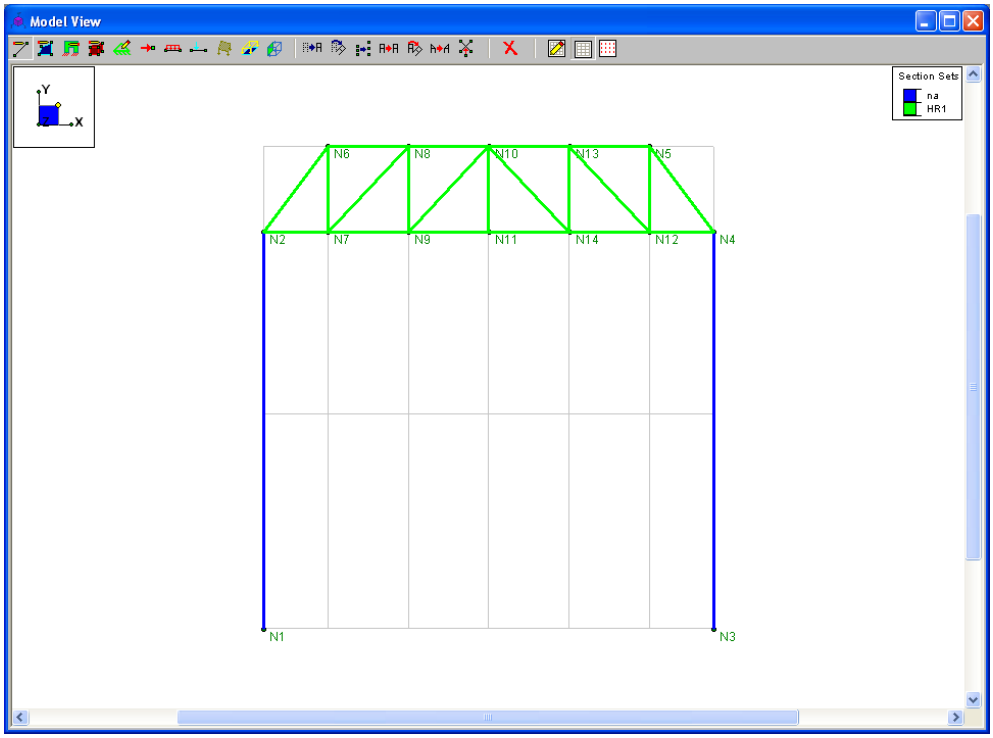
Grid Locations:

Point A (18,22.5,0)
Point B (18,18.5,0)
Point C (14.25,22.5,0)
Point D (14.25,18.5,0)
Point E (10.5,22.5,0)

Draw the next series of members, as shown above.

- Draw members from points A to B, B to C, C to D, then lastly from D to E.
- Click the right mouse button or press ESC to release the mouse.

The image should now look like this:



Physical Members

Notice that you have defined your truss chords with one member, rather than multiple members between each joint. RISA-3D's Physical Member feature allows you to quickly draw in members this way, rather than making you draw every member as split between nodes.

Physical Members automatically connect the members and nodes that frame into them along their midspan. Therefore, you do not have to use multiple members to model what is one Physical Member in the field. Then, if you later have to make changes to your model, you do not have to apply changes to multiple smaller members. Most important, the results values for this continous member will not be spread out over multiple members, making it difficult to find design values, it will instead be presented as that of only one member.

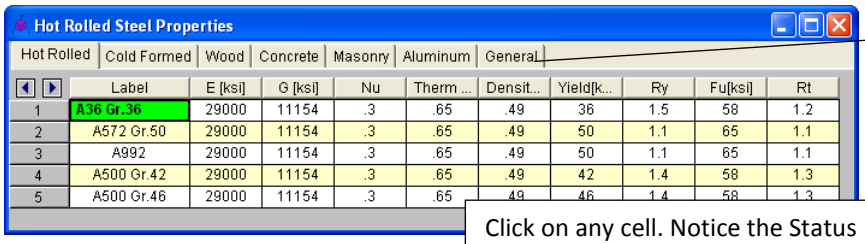
Material Properties

The material properties are defined in the **Materials** spreadsheet. This spreadsheet contains default material properties that are common for design in the United States. However, you can always edit or add to this spreadsheet at any time if you use different properties.

Re-open the **Data Entry** toolbar you closed earlier:

- On the RISA toolbar, click the **Spreadsheet Shortcuts (Data Entry)** button to display the **Data Entry** toolbar.
- On the **Data Entry** toolbar, click **Materials**.

The **Materials** spreadsheet will display.



Notice the tabs contain the various material types.

Click to review the information under each tab: **Hot Rolled**, **Cold Formed**, etc.

Click on any cell. Notice the Status bar (bottom of the screen) gives an explanation of the current column.

- Review each of the different material tabs and the default program values. For this tutorial, we will use the default values, so do not make any changes.

Note: The United States default material values are built in to RISA-3D. However, you can create new default values for your future models by adding those materials into any spreadsheet and, on the Window toolbar, clicking **Save as Defaults**  .

For example, if you are working on a Canadian job with a steel that conforms to the G40.21 specification, you can add that material and save it for use on future models. On the Window toolbar, click **Save as Defaults**  . Any new models will start with this steel material automatically.

Design Rules – Size / UC

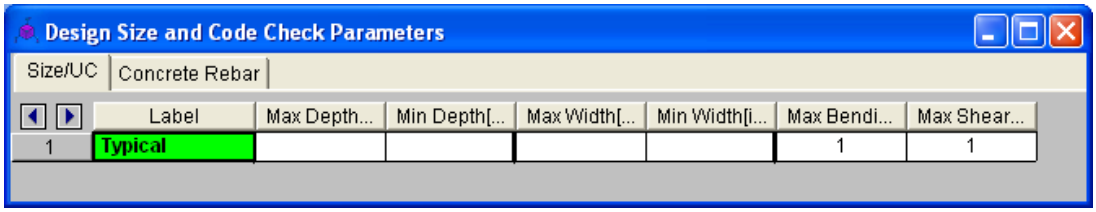
In an effort to optimize hot rolled, cold formed, wood, concrete, aluminum, and masonry members, RISA-3D will suggest a lighter, more efficient shape for each of the members. When suggesting an alternate shape, RISA-3D will only consider shapes that are included in the designated Design List. The Design List is a set of members that will be used in the design and optimization of a member. You may edit the default redesign lists at any time to create custom lists of your own. These lists are contained in C:\RISA\risa_redesign_lists and you may open/edit them in a text formatting program, such as Notepad.

For example, a Section Set could be defined with the Wide Flange Design List. This list references the redesign list which contains the US list of wide flange sections from the *AISC Manual of Steel Construction*. In this case, RISA-3D will never suggest that a channel or H pile would be a more efficient shape. For information about editing the existing Design Lists or creating your own custom Design Lists, refer to *RISA-3D General Reference*.

Next, designate specific sections you would like redesigned by RISA-3D using the optimization parameters set in the **Design Rules** spreadsheet.

- On the **Data Entry** toolbar, select **Member Design Rules**.

The **Member Design Rules** spreadsheet will display:



Enter the following data in row 1:

- In row **1**, under **Label**, type **Col-Rules**. (Press the TAB key two times.)
- Under **Min Depth**, type **14**. (Press the TAB key two times.)
- Under **Min Width**, type **14**.
- Press ENTER.

Enter the following data in row 2 (continue pressing the TAB key to advance between columns):

- In row **2**, under **Label**, type **Steel-Bms**.
- Under **Min Depth**, type **12**.
- Press ENTER.

Enter the following data in row 3:

- In row **3**, under **Label**, type **Conc-Bms**.
- Under **Min Depth**, type **12**.
- Under **Min Width**, type **10**.
- Press ENTER.

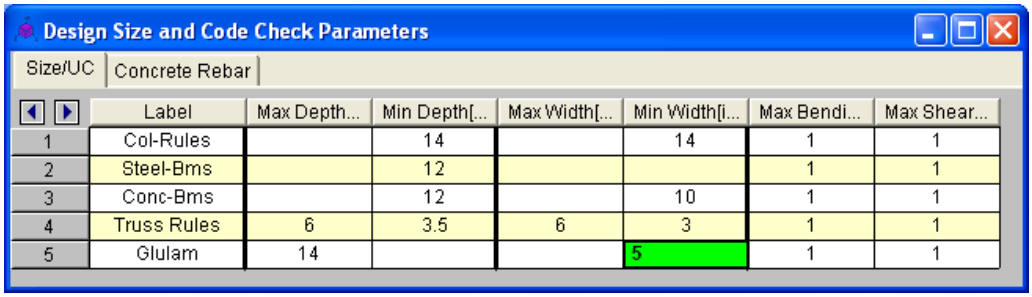
Enter the following data in row 4:

- In row **4**, under **Label**, type **Truss Rules**.
- Under **Max Depth**, type **6**.
- Under **Min Depth**, type **3.5**.
- Under **Max Width**, type **6**.
- Under **Min Width**, type **3**.
- Press ENTER.

Enter the following data in row 5:

- In row **5**, under **Label**, type **Glulam**.
- Under **Max Depth**, type **14**.
- Under **Min Width**, type **5**.

This spreadsheet should look like this:



	Label	Max Depth...	Min Depth[...	Max Width[...	Min Width[i...	Max Bendi...	Max Shear...
1	Col-Rules		14		14	1	1
2	Steel-Bms		12			1	1
3	Conc-Bms		12		10	1	1
4	Truss Rules	6	3.5	6	3	1	1
5	Glulam	14			5	1	1

Notice there are several parameters available to control the suggested alternate shapes. You can define acceptable member depth and width ranges, and also code check ranges.

Design Rules – Concrete Rebar

In addition to controlling the size of your concrete members, RISA-3D also allows you to control the type of reinforcement and cover to be used.

- Click on the **Concrete Rebar** tab.

This spreadsheet controls the size and placement of the flexural and shear reinforcement used in the design of your concrete beams and columns.

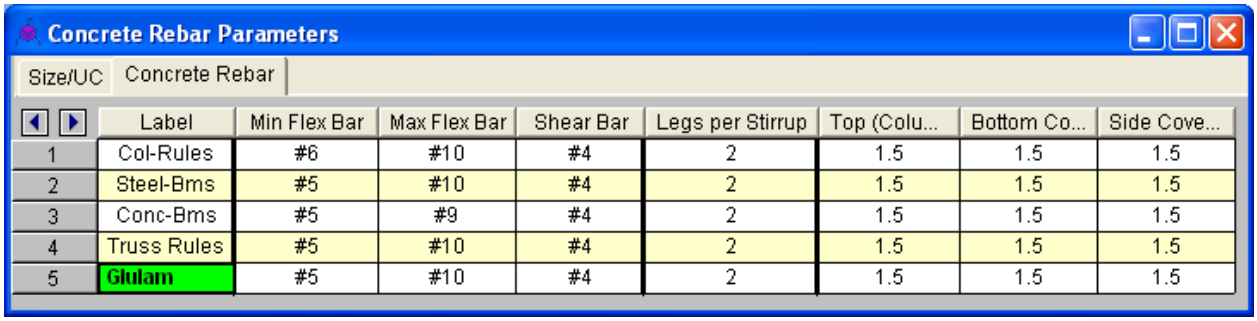
Change the minimum bar size for **Col-Rules**:

- In the **Col-Rules** row click the first cell in the **Min Flex Bar** column. Click a second time in the cell to reveal the list of options, and select **#6**.

Note: It may be helpful to lengthen the column heading to read the full column label.

Change the maximum bar size for **Conc-Bms**:

- In the **Conc-Bms** row click the cell under **Max Flex Bar** column. Click a second time in the cell to reveal the list of options, and select **#9**.



	Label	Min Flex Bar	Max Flex Bar	Shear Bar	Legs per Stirrup	Top (Colu...	Bottom Co...	Side Cove...
1	Col-Rules	#6	#10	#4	2	1.5	1.5	1.5
2	Steel-Bms	#5	#10	#4	2	1.5	1.5	1.5
3	Conc-Bms	#5	#9	#4	2	1.5	1.5	1.5
4	Truss Rules	#5	#10	#4	2	1.5	1.5	1.5
5	Glulam	#5	#10	#4	2	1.5	1.5	1.5

Note: Although rules for concrete rebar are given for each Design Rule row, these rebar entries will be ignored if they are applied to anything other than concrete members.

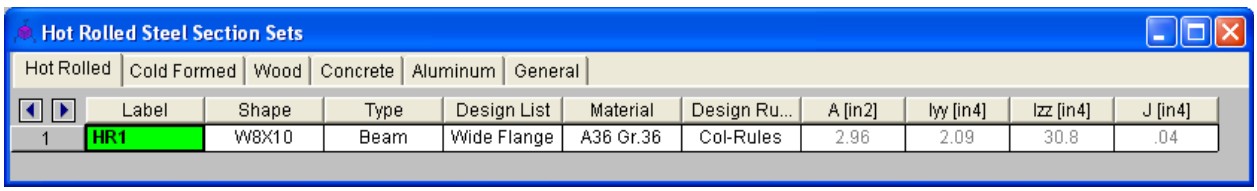
Section Sets Spreadsheet

RISA-3D starts you off with one default Section Set for each basic material type. To differentiate between the new members being created, you will create new Section Sets for each as you go along, using the **Section Sets** spreadsheet. While in this spreadsheet, you will also define the section properties for the members you have defined so far.

Open the **Section Sets** spreadsheet.

- On the **Data Entry** toolbar, click **Section Sets**.

The **Section Sets** spreadsheet should look like this:



	Label	Shape	Type	Design List	Material	Design Ru...	A [in2]	Iyy [in4]	Izz [in4]	J [in4]
1	HR1	W8X10	Beam	Wide Flange	A36 Gr.36	Col-Rules	2.96	2.09	30.8	.04

Section sets provide an easy way to manage members that are repetitive. This will become even more apparent when you have RISA-3D choose new and/or optimized member sizes.

So far, your truss has only nine web members. But, what if you had three trusses and were required to change the section (as you will do in Tutorial 2). Changing 27 member properties one at a time could take some time. By defining Section Sets, you will be able to globally change the truss web member properties by changing only one section on the spreadsheet.

Hot Rolled Sections and Database

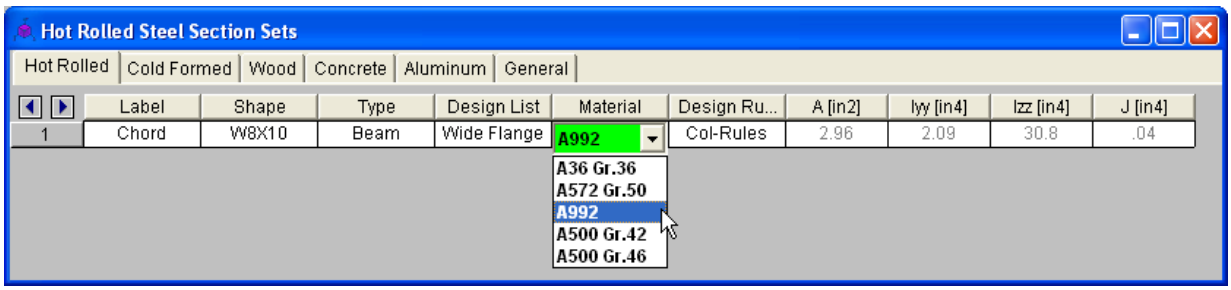
You currently have one Section Set for each material type. Next, you will add some Section Sets and change the labels so that they are more useful. The active cell (denoted by green color) should contain the text **HR1A**. You used that section to draw in the truss members.

Give the section a more descriptive label:

- In the row labeled **1**, under the **Label** column, type **Chord**.

Next, change the material:

- Click the cell under the **Material** column. Click the cell a second time to display the available hot rolled materials (these come directly from the default entries in the **Hot Rolled** tab in the **Materials** spreadsheet). Select **A992** for the beam material as shown below:



Notice that the Section Set data consists of a number of entries. The **Shape** column entry indicates the current size that will be used for the member. The **Design List** and **Design Rules** entries define the list of shapes to be used and the design restrictions to impose when redesigning and/or optimizing the member. For more information on customizing design lists, refer to Appendix A in *RISA-3D General Reference*.

Modify the Design Rules:

- Click the cell under the **Design Rules** column. Click the cell a second time to reveal the list of available options. Select **Truss Rules**.

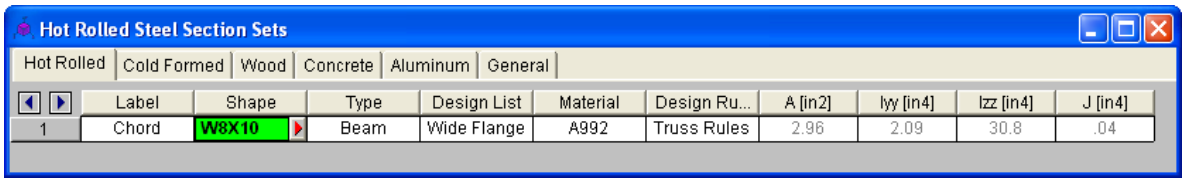
The **Type** column entry indicates whether the member is a beam, column, or brace. This is especially important for the design of concrete sections. Finally, the **A**, **Iyy**, **Izz,** and **J** columns list the area, moments of inertia, and torsional constant for the current shape.

By default the current shape is a W8X10. This is because it is the first shape available in the Wide Flange Design List. Since this is not realistic for your steel truss, you will want to change it. If you know what shape you want to use, you may type it in directly. Or, if you are not sure, you may open the database and review the full list of available shapes.

Define the Hot Rolled shape:

- Click the first cell under the **Shape** column.

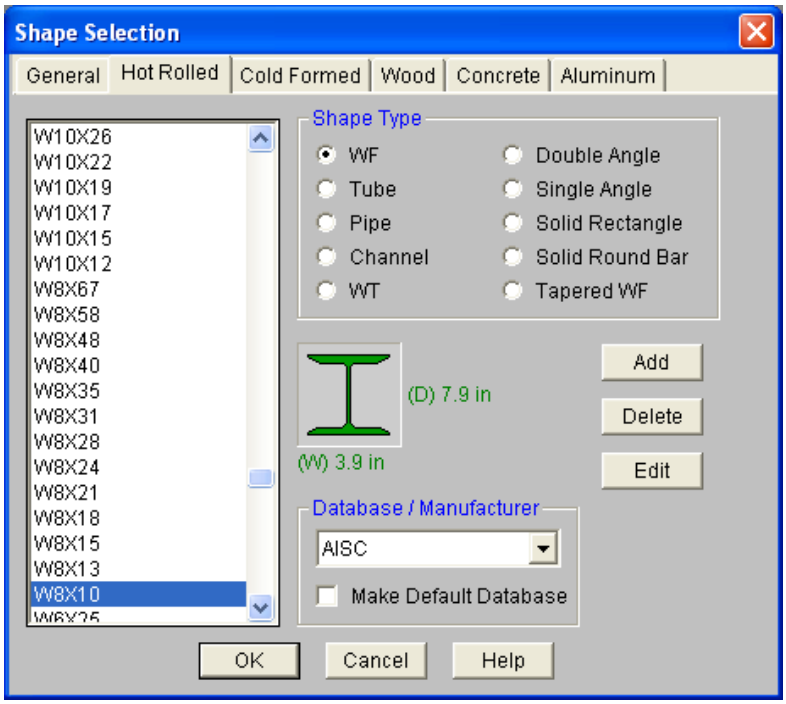
The spreadsheet should look like this:



Open the **Hot Rolled** shape database:

- Click the red arrow on the right to open the **Hot Rolled** tab in the **Shape Selection** database.

The **Hot Rolled** tab **Shape Selection** database looks like this:



You may want to review the **Shape Type** and **Database/Manufacturer** settings for future reference.

The left side of the dialog box contains a list of available shapes. The selected shape is currently W8X10. Change the shape selection:

- Select **Double Angle** as your **Shape Type**.

Notice that the list of available shapes populates now with all the available double angle sections.

- Select **LL6X6X12X3** from the list and click **OK** to return to the spreadsheet.
- On the **Section Sets** spreadsheet, notice that **LL6X6X12X3** is now in the **Shape** column. Press the TAB key to advance to the next cell and update the geometric properties (A, Iyy, Izz, J).
- Click in the **Design List** cell of the first row. Chose **Double Angle** from the menu.

Now add two more section sets for the truss web members and the steel beam:

- Click ENTER to create a new line.
- In the row labeled **2**, under the **Label** column, type **Web**.

Note: Clicking ENTER creates a new line with all entries (except the **Label**) identical to that of the line above it. Because of this, it is not neccessary for you to modify the **Material** and **Design Rule** entries for your second row in this spreadsheet.

Change the shape selection:

- Click in the **Shape** cell in the second row and click the red arrow to open the **Hot Rolled** tab in the **Shape Selection** database.
- Select **Single Angle** as your **Shape Type**.
- Select **L4X4X4** from the list and click **OK** to return to the spreadsheet.
- Click in the **Design List** cell of the second row. Chose **Single Angle** from the menu.

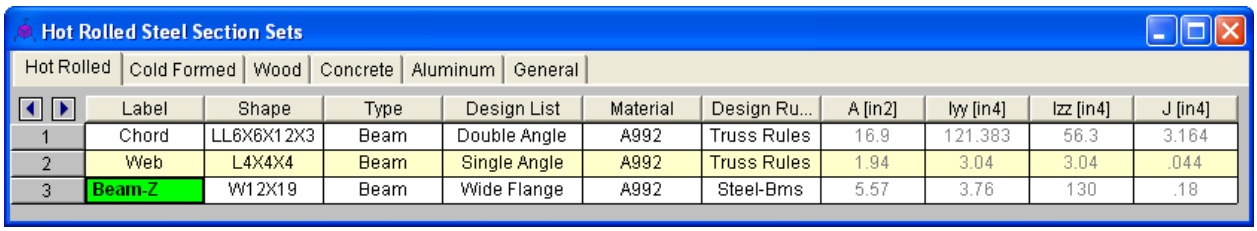
Last, enter in your steel beam:

- Click ENTER to create a new line.
- In the row labeled **3**, under the **Label** column, type **Beam-Z**.

Change the shape selection:

- Click in the **Shape** cell in the third row and click the red arrow to open the **Hot Rolled** tab in the **Shape Selection** database.
- Select **WF** as your **Shape Type**.
- Select **W12X19** and click **OK** to return to the **Section Sets** spreadsheet.
- Click in the **Design List** cell of the third row. Chose **Wide Flange** from the menu.
- Lastly, click in the **Design Rules** cell of the third row and select **Steel-Bms**.

When you are finished, the spreadsheet should look like this:



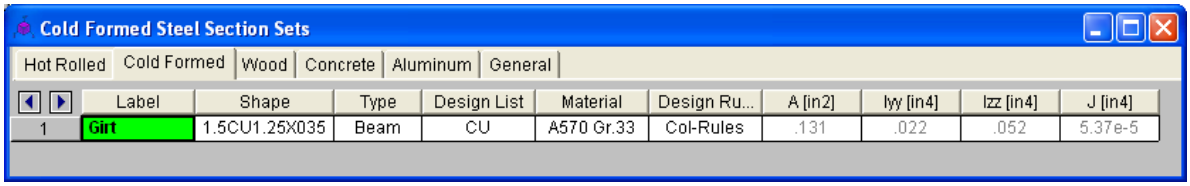
	Label	Shape	Type	Design List	Material	Design Ru...	A [in2]	Iyy [in4]	Izz [in4]	J [in4]
1	Chord	LL6X6X12X3	Beam	Double Angle	A992	Truss Rules	16.9	121.383	56.3	3.164
2	Web	L4X4X4	Beam	Single Angle	A992	Truss Rules	1.94	3.04	3.04	.044
3	Beam-Z	W12X19	Beam	Wide Flange	A992	Steel-Bms	5.57	3.76	130	.18

Cold Formed Sections and Database

Now, you will add in your Cold Formed Section Set. Open the Cold Formed tab:

- Click the **Cold Formed** tab.
- Under **Label**, type **Girt** to make the label more informative.

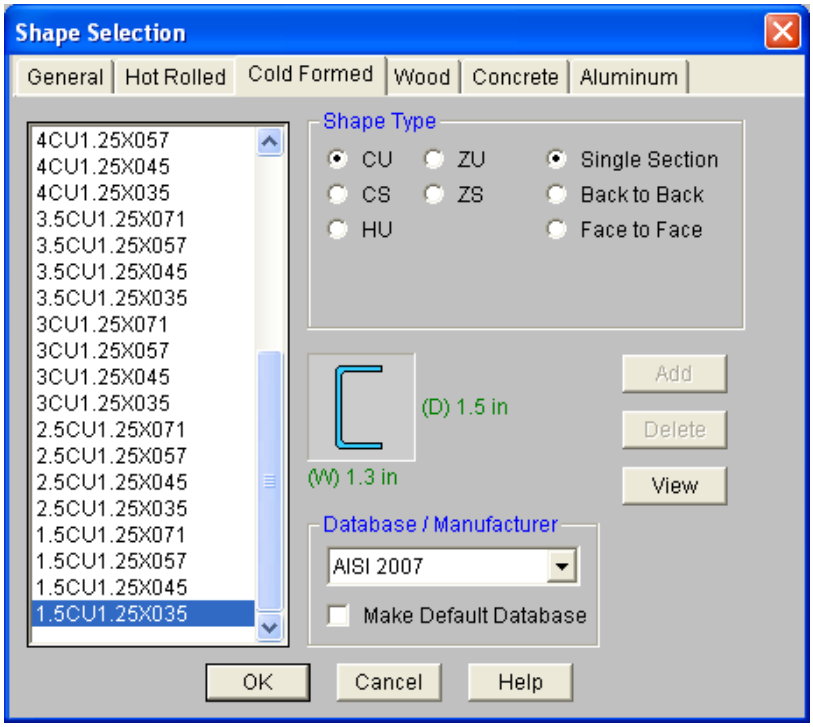
Your screen should now look like this:



Once again, select your shape from the shape selection database.

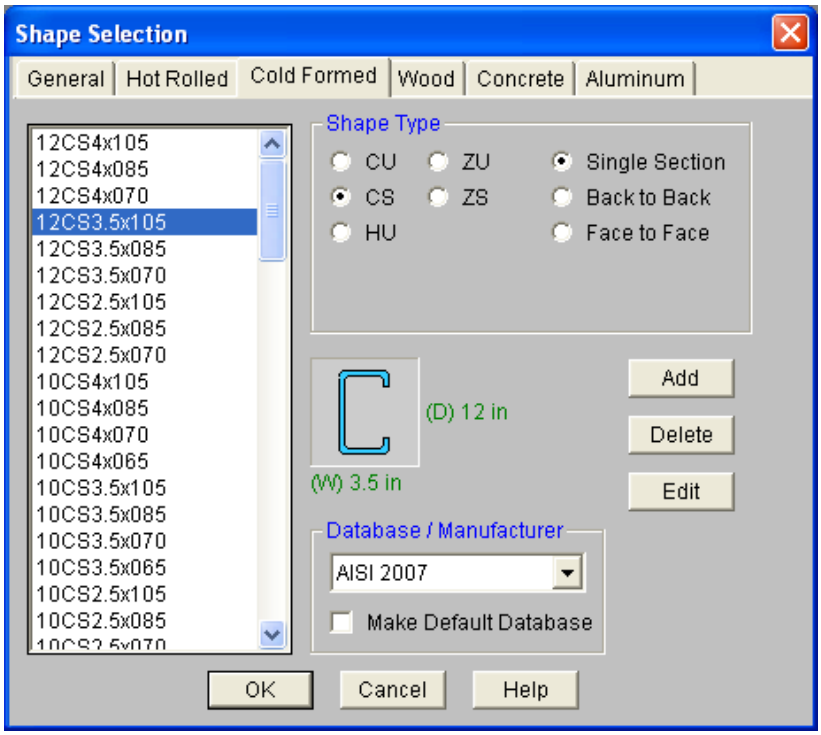
- Click the first cell under the **Shape** column.
- Click the red arrow on the right to open the **Cold Formed** tab of the **Shape Selection** database.

The **Cold Formed** tab is very similar to the **Hot Rolled** tab, and looks like this:



Review and modify the shape settings:

- Under **Shape Type**, select **CS**.
- Under the list of available shapes (on the left), select **12CS3.5x105**.



- Click **OK** to return to the **Section Sets** spreadsheet.

To ensure that when performing member redesign and/or optimization, RISA-3D will suggest a new CS from the appropriate list of shapes, modify the design list as follows.

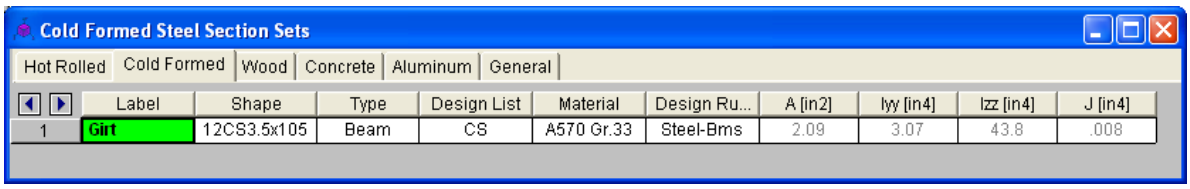
Note: When selecting cells in a spreadsheet, you may click directly in the cell or press the TAB to advance from cell to cell.

- On the **Section Sets** spreadsheet, click in the cell under the **Design List** column, making this the active cell. Click the cell a second time to display the available design lists.
- Select **CS**.

Next, modify the Design Rules:

- Click in the cell under the **Design Rules** column two times to display the available design lists.
- Select **Steel-Bms**.

When finished, your spreadsheet should look like this:

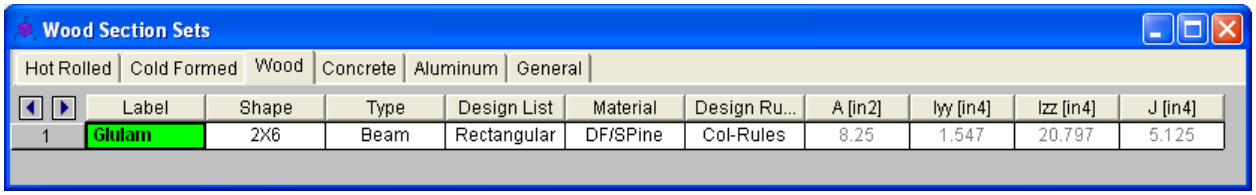


Wood Sections and Database

Moving now to wood, define the section used for the glulam beams.

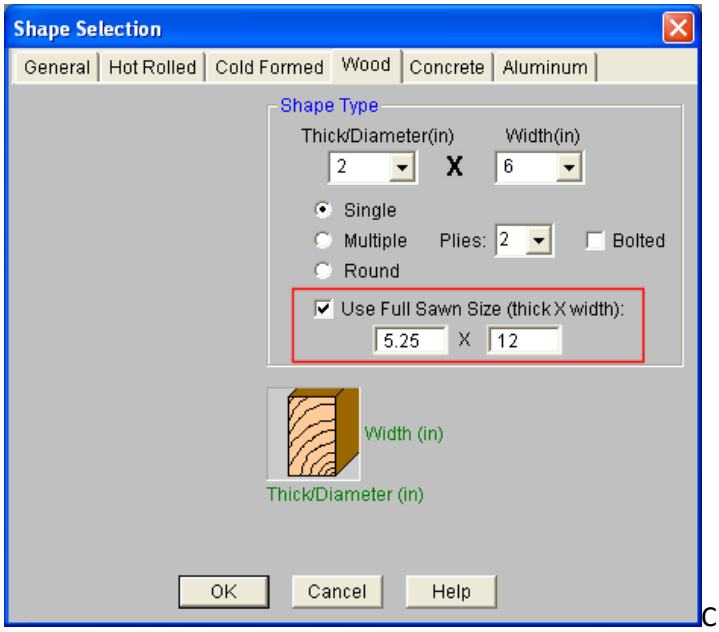
- Click the **Wood** tab.
- Under **Label**, in row **1**, type **Glulam**.

The spread sheet should now look like this:



Next, enter in your full sawn member size:

- Click in the **Shape** cell. Click the red arrow to open the **Wood** tab of the Shape Selection database.
- Under **Shape Type**, select **Use Full Sawn Size (thick X width).**
- Type in **5.25** X **12**, as shown in the image below:



C

Now you can return to the **Section Sets** spreadsheet:

- Click **OK.**

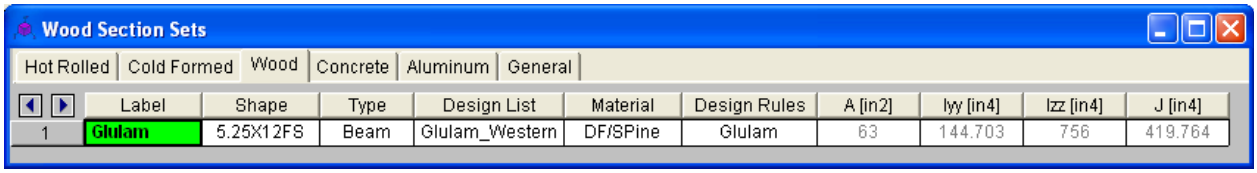
You have now changed the current shape to a full sawn glulam size. If you want RISA-3D to suggest appropriate full sawn shapes when performing member redesign and/or optimization, you must also change the Design List.

- Click under the **Design List** column to make it the active cell. Click the cell a second time to view the available design lists. (You may need to expand the **Design List** column to view the longer labels.)
- Select **Glulam_Western**.

These settings ensure that when RISA-3D performs member redesign and/or optimization, it will suggest an appopriate member for your beams. To limit the depth and thickness of the suggested shapes, you will now change the **Design Rules**.

- On the **Wood Section Sets** spreadsheet, click the first cell under the **Design Rules** column to make this the active cell. Click the cell a second time to view the available Design Rules.
- Select **Glulam**.

Your spreadsheet should now look like this:



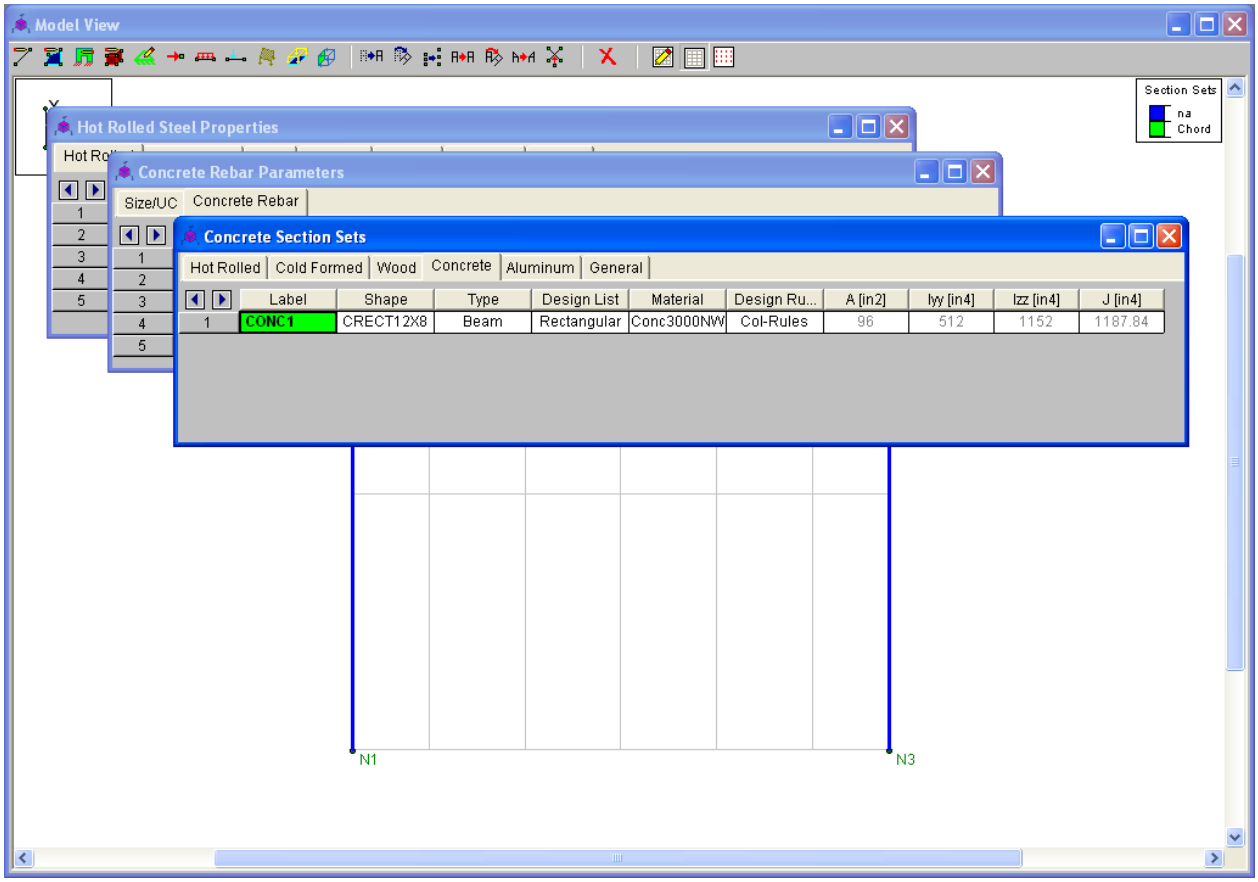
Concrete Sections

Open the **Concrete** tab:

- Click the **Concrete** tab.

Because you initially defined your concrete member using **Assign Shape Directly** (rather than using section sets), you will not need to define any members using the section sets listed under the **Concrete** tab. Take a minute to review the information listed under the **Concrete** tab. It is similar to the information in the other tabs for steel and wood.

Multiple Windows



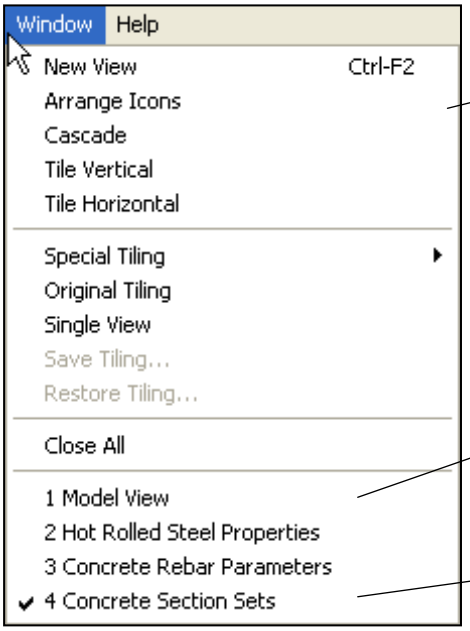
At this point, you should have a model view and three spreadsheets open and visible on your screen. Notice the "active window" is denoted by a colored title bar (in this case, the **Concrete Section Sets** spreadsheet). The title bars of the other inactive windows are dimmed. The windows are tiled in the order that you opened them. You can return to any of the windows by clicking anywhere on them, preferably on the title bars, so you do not accidentally click a button or a grid point.

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

You can also access the open windows and dialog boxes through the **Window** menu. This menu allows you access to windows that may be hidden from view.

- Click on the **Window** menu from the Main menu.

The **Window** menu looks like this:



The **Window** menu presents some window arrangement and tiling options.

The open windows are listed in the order they were opened. You can select any of the open windows to make it active.

The active window is denoted by a check mark.

Change the active window to model view:

- On the **Window** menu, select **1 Model View**.

The model view becomes the active window and moves to the front of the other windows. The spreadsheets are still open beneath the model view window, but not currently visible.

You can also recall open windows the same way you originally opened them (by selecting from a menu or clicking a toolbar button). For example, to make the currently open **Section Sets** spreadsheet become the active window so you can work on it:

- On the **Data Entry** toolbar, click **Section Sets** to open the **Section Sets** spreadsheet.

Close the active window:

- Click **Close** or press CTRL+F4 to close the **Section Sets** spreadsheet.

Now, the model view should be your active window. Notice that RISA-3D resumes where you left off for all open windows.

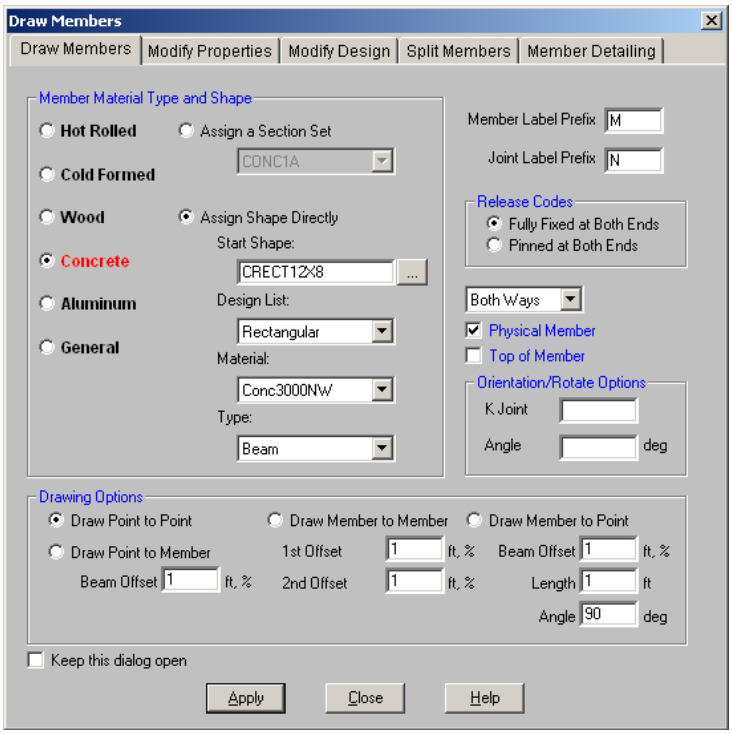
Open the **Draw Members** dialog box and prepare to draw the crossbeam:

- Press CTRL+D to recall the **Draw Members** dialog box.
- Under **Member Material Type and Shape**, click **Concrete**.

Bypass the section sets and assign this shape directly:

- Under **Member Material Type and Shape**, click **Assign Shape Directly**. Under **Start Shape**, type **CRECT12X8**. In the **Type** list, click **Beam**.
- Under **Release Codes**, click **Fully Fixed at Both Ends**.

The **Draw Members** dialog box should now look like this:

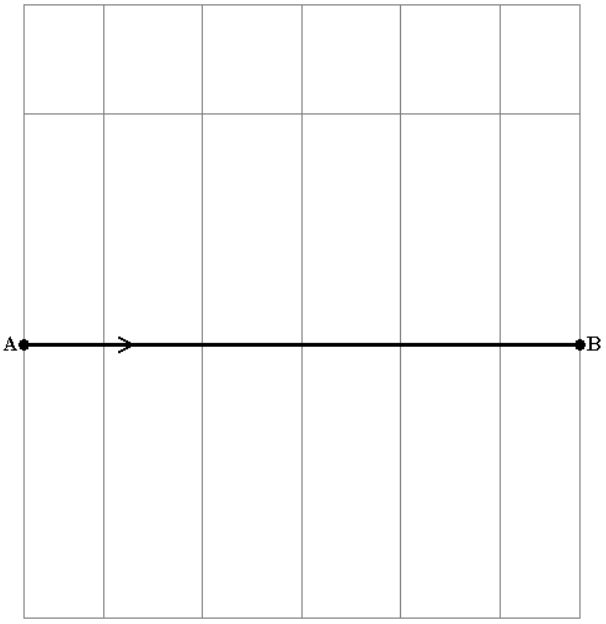


Exit the dialog box and return to the model view.

- Click **Apply**.

Begin drawing the cross beams:

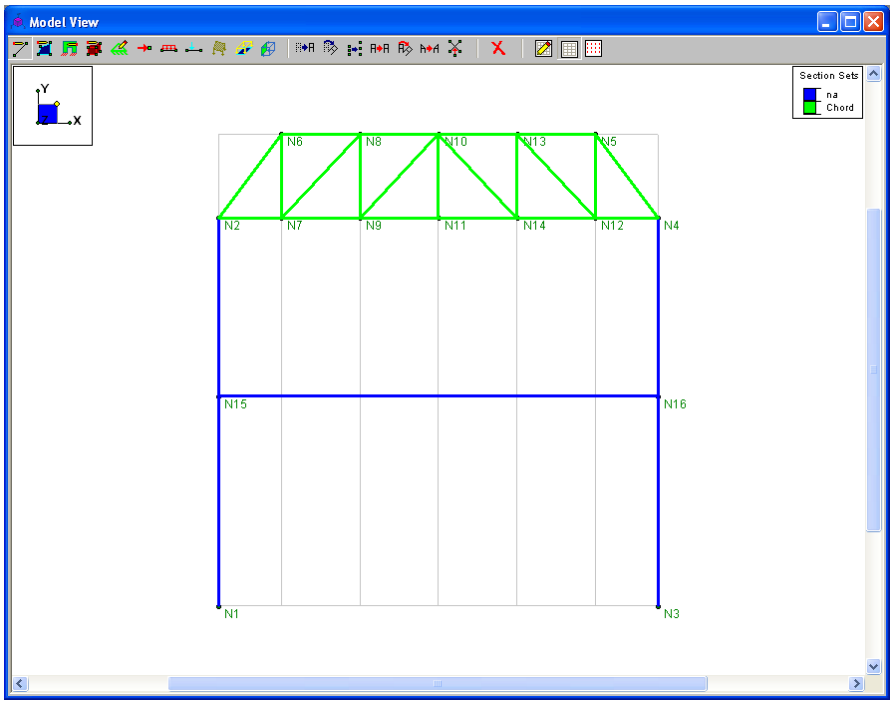
- Draw members from point A to point B using the grid locations shown below.
- Click the right mouse button or press ESC to release the mouse.



Grid Locations:

Point A (0,10,0)
Point B (21,10,0)

Your image should now look like this:



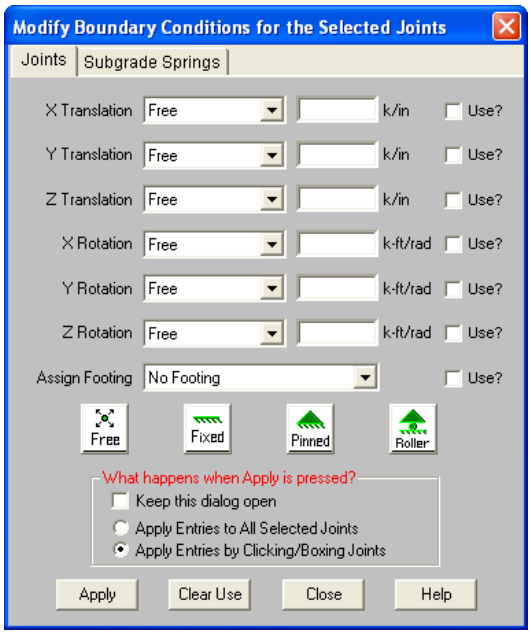
Boundary Conditions

Boundary conditions define how the model is constrained. All models must be attached to some external point(s) of support otherwise the program will produce an error message telling you that your model is unstable.

Prepare to add boundary conditions:

- On the Drawing toolbar, click **Boundary Conditions**  .

The following dialog box will display:



Boundary conditions can be applied to any joint. There are six degrees of freedom for each joint; therefore, there are six releases provided; one for each direction.

Review the options for assigning boundary conditions. Each option is described below:

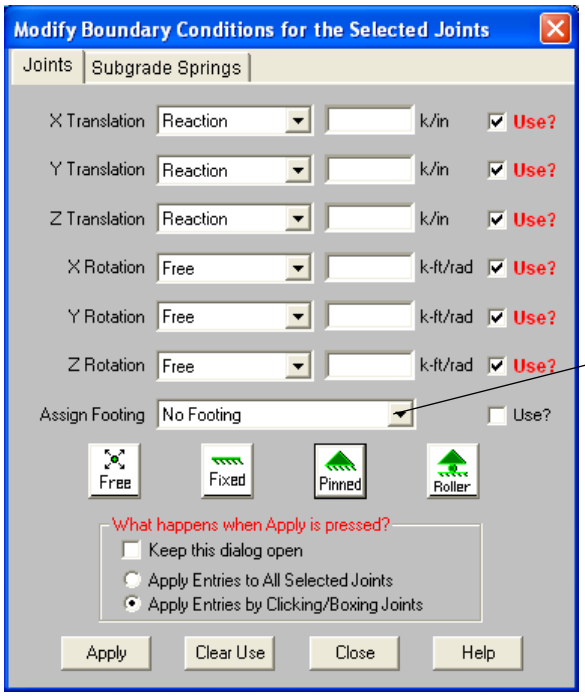
- In the **X Translation** list, click the down arrow to see the available boundary condition options.

Free	No restraint.
Reaction	Specifies full restraint for the indicated direction. No movement will be allowed in the indicated direction for this joint. A reaction force will be calculated and reported.
Fixed	Also specifies full restraint for the joint in the indicated direction; however, no reaction force is calculated. This setting actually removes the degree of freedom from the solution. If you are not interested in reactions, using this option will result in a slightly smaller model.
Spring	This option models a spring attached to the joint in the indicated direction and must be accompanied by a spring stiffness. The units for the spring stiffness depend upon whether the spring is translational or rotational and are displayed next to the edit box. For example, to specify a spring of stiffness 100 kips per inch, select **Spring** from the list and type **100** in the adjacent box.
Compression Spring	This option models a compression-only spring and must be accompanied by a spring stiffness. This type of spring will only resist compression (negative displacement); if a tension force (positive displacement) is applied to it, it will release and provide no resistance.
Tension Spring	This option models a tension-only spring and must be accompanied by a spring stiffness. This type of spring will only resist tension (positive displacements); if a compression force (negative displacement) is applied to it, it will release and provide no resistance.
Slave	This option “slaves” the degree of freedom of one joint to another that you specify.

Next, specify pin supports at the base of your columns. For this tutorial, the X, Y, and Z translations will be restrained, but the three rotations will be unrestrained. This is a common boundary condition that is built in to RISA-3D, and can be applied as shown below:

- Click the **Pinned** button and the boundary condition settings described above will be completed for you.

The dialog box now looks like this:



The **Use?** check boxes must be selected for a modification to be applied. This allows you to change a condition for just one of the parameters, instead of requiring all six to match.

The **Assign Footing** option allows you to place a footing at a joint if you also have RISAFoot authorized and installed on your computer. Use the menu to select the footing from those specified in the **Footing Definitions** spreadsheet.

Finish your selections and close the dialog box:

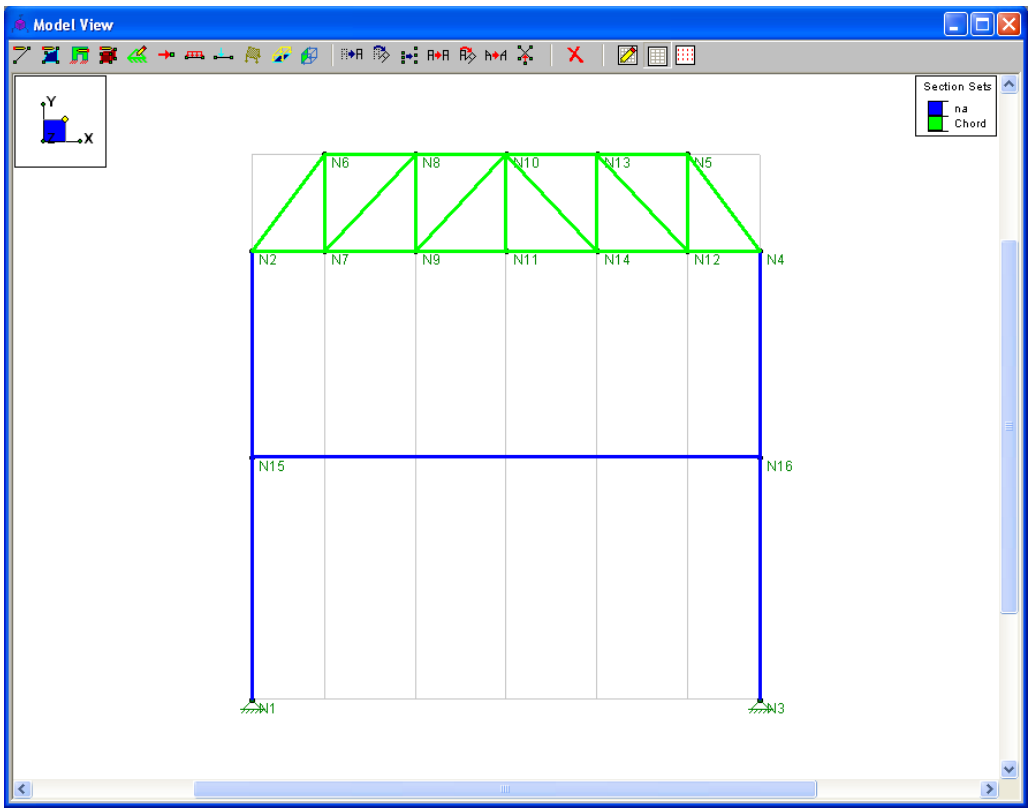
- Under **What happens when Apply is pressed?**, select **Apply Entries by Clicking/Boxing Joints**.
- Click **Apply**.

Note: Your cursor will change to , indicating you are ready to apply the boundary condition when you click on any joint.

Apply the boundary condition to the base of both columns at joints N1 and N3.

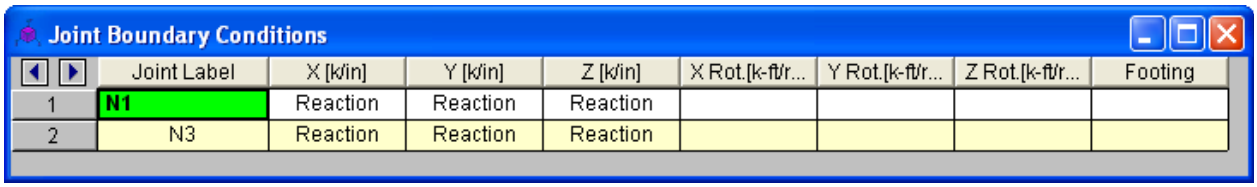
- Click joint **N1** and then on joint **N3**.

Now that you are finished adding the boundary conditions, your model should look like this:



Review how the boundary condition data you just entered graphically was recorded on the **Boundary Conditions** spreadsheet:

- On the **Data Entry** toolbar, click **Boundary Conditions** to open the spreadsheet.



	Joint Label	X [k/in]	Y [k/in]	Z [k/in]	X Rot.[k-ft/r...	Y Rot.[k-ft/r...	Z Rot.[k-ft/r...	Footing
1	N1	Reaction	Reaction	Reaction				
2	N3	Reaction	Reaction	Reaction				

RISA-3D allows you to edit existing boundary conditions here, or add new ones. To add a new boundary condition, simply type directly on the spreadsheet (you can type just the first letter such as “R” for reaction, then press ENTER). Or, you can click on the red arrow to view the available options.

This is the end of Tutorial 1. The next tutorial demonstrates how to make modifications to the model.

You may save your model to be used as the starting point for the next tutorial, or begin the next tutorial using the .r3d starter file in the RISA-3D **Tutorials** folder. To save the model:

- On the **File** menu, click **Save As** and enter a file name. Note that it will have an .r3d extension.

Tutorial 2 – Modify

In this tutorial, you will modify the 2D frame you defined in Tutorial 1 and use RISA-3D's graphic editing features to expand it into a complete 3D frame. This tutorial continues where the previous tutorial ended, so follow these steps to get your model up and running:

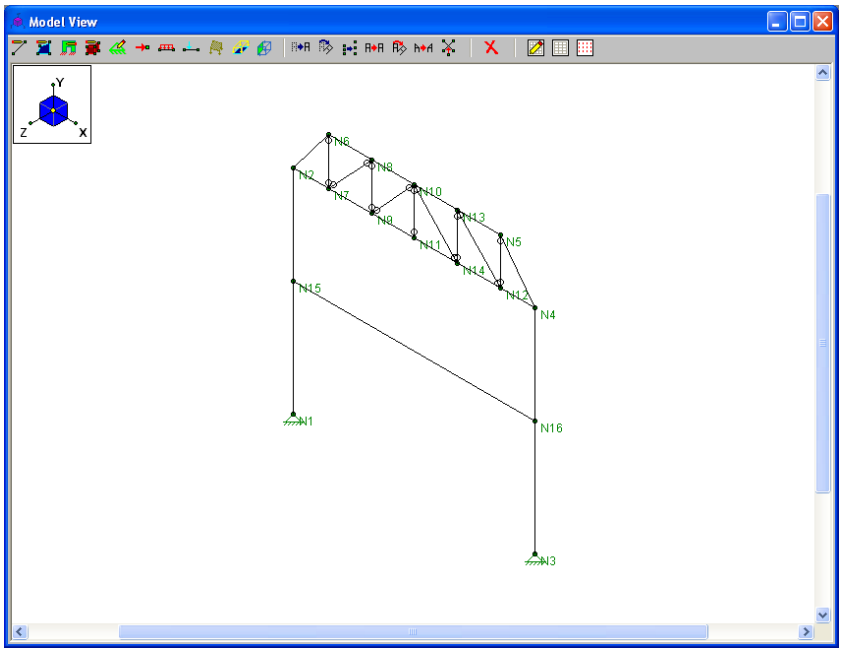
If you are continuing from the previous tutorial:

- From the **Window** menu, select **Single View**.
- On the Window toolbar, click the **Graphic Editing Toolbar** 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 Technologies:

- Double-click the **RISA-3D** icon to start the program.
- Click **Open Model**.
  Double-click the **Tutorials** folder, select **Tutorial 2 Starter.r3d** and click **Open**.
  Click **Close** (or **Cancel**) to exit the **Global Parameters** dialog box.
- On the Window toolbar, click the **Graphic Editing Toolbar** button to activate the Drawing toolbar.
- On the **Data Entry** toolbar, click **Close** to close it.

Your screen should look like this:

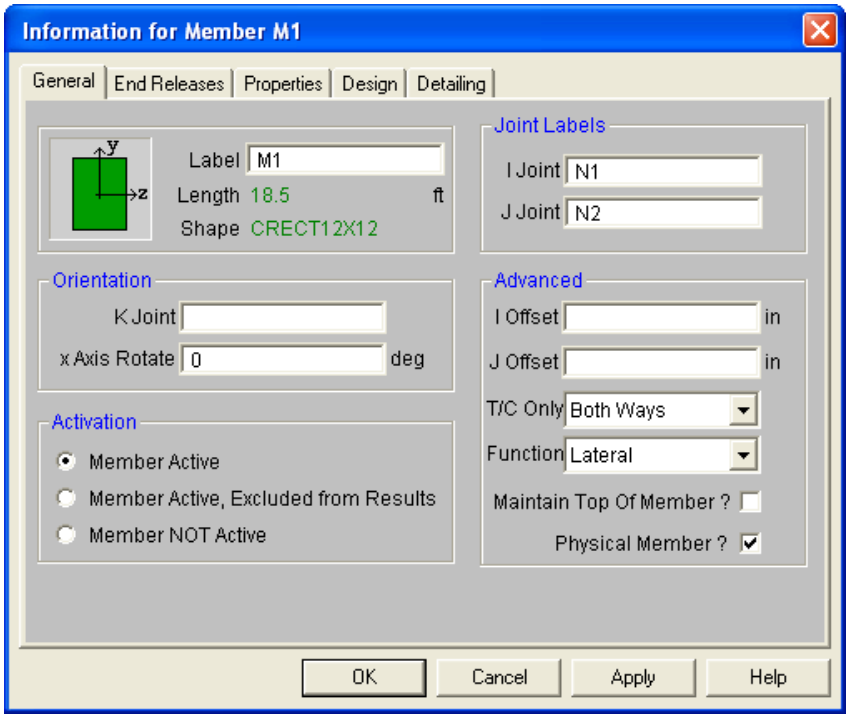


To get started, you will modify the end conditions at the tops of the columns. Notice that the web members of the truss are pinned at the ends (indicated by the circle at the end of each member). All other members are fixed at the end. You will modify the column top ends so that they are pinned as well.

Modifying an Item

RISA-3D provides a quick and easy way to view and edit properties of any item in the model. You may simply double-click any joint, member, or plate to view and edit all of the information for that item.

- Double-click the left column (anywhere between **N1** and **N2**) to open the following dialog box:



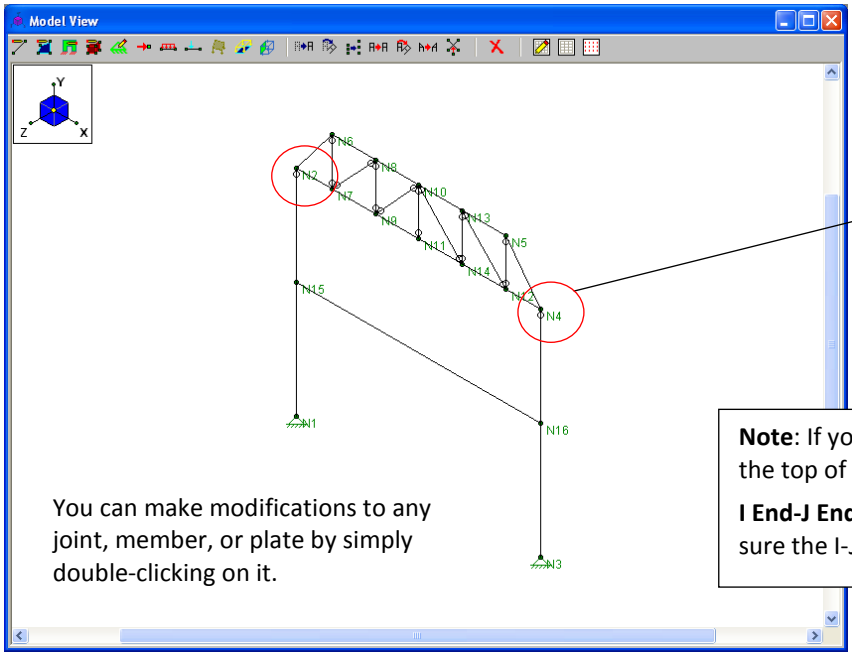
Modify the end releases for the first column:

- Click the **End Releases** tab.
- Under **J End Release Codes**, click **Bending Moment Released (Torsion Fixed)**.
- Click **OK** to close the dialog box and return to the model view.

Next, double click the column on the right to define the end releases for that column:

- Double-click the right column (anywhere between **N3** and **N4**).
- Click the **End Releases** tab.
- Under **J End Release Codes**, click **Bending Moment Released (Torsion Fixed)**.
- Click **OK** to close the dialog box.

Your model should look like this:



The open circles at the top of each column represent pinned end releases

You can make modifications to any joint, member, or plate by simply double-clicking on it.

Note: If you do not see the open circles at the top of each column, click the **I End-J End** button a few times to make sure the I-J end flags are on.

Before proceeding, inspect one of the joints:

- Double-click the bottom left joint (**N1**) to review the information.
- Click **Cancel** to close the dialog box without making any changes.

Note: This method of reviewing and editing model information is sufficient when modifying one or two items in your model. However, to modify several members at once, it is easier and more efficient to use the graphical selection and editing tools, described next.

Editing the Model View

This section explains how to modify the model view using the graphical selection and editing tools. Since you are currently working in a model view the Window toolbar will look like this:

The first button on the left is the **Plot Options** button (used earlier to color the members). This will be explained in more detail later.

Rotating the Model

The **Rotate** buttons are used to rotate the model counter-clockwise or clockwise about the global axes. The **Snap View** buttons are used to "snap" the view to the default isometric view, or the XY, XZ, or YZ planar view, respectively.

- Click some of the **Rotate** and **Snap View** buttons (shown above) and experiment with rotating and snapping the plot to different views.

As you click the **Rotate** and **Snap View** buttons, notice the direction the plot is rotating. It may help to reference the global axes icon at the top left corner of your model that looks like this:

When you are finished experimenting, place the model in an isometric view:

- Click **Isometric** Iso.

Zooming the Model

The **Zoom** buttons are used to zoom the model view in and out. The first two **Zoom In** and **Zoom Out** of the model as a whole. The third, **Box Zoom**, allows you to zoom in on a specific area of interest. Try the box zoom now:

- Click the **Box Zoom** button, then box any area in your model by pressing and dragging your mouse. Release the mouse.

Return to your normal model view:

- Click **Redraw** to redraw the full model view.

Panning

If your mouse has a wheel, you will be able to use RISA-3D's panning feature. Simply press down on the wheel anywhere on your model, then move the mouse to the desired location. RISA-3D "grabs" the model at the current location and "moves" the model to the new location.

Try the panning feature, then zoom in and out:

- Position your mouse anywhere on your model. Press down on the mouse wheel and drag to the left. Your model should move to the new location.
- Zoom in and out by rolling the mouse wheel up and down.

Return to your normal view:

- Click **Redraw** to redraw the full model view.

Drawing with the Graphic Editing Toolbar

The **Graphic Editing Toolbar** button is used to toggle the Drawing toolbar on and off in the current model view.

- Click the **Graphic Editing Toolbar** button two times. Notice the Drawing toolbar will toggle off and on again.

Saving Views

If you create a customized view that you think you may use again, you can save it for future recall using the **Save or Recall View** button  . (It may even be a good idea to use this feature whenever you create a view that takes some time.) Simply click the **Save or Recall View** button  and assign a name to the saved view. All of the plot options will be recorded and saved for later recall.

Cloning Views

Cloning a view, using the **Clone** button  , creates a new window and makes an identical copy of the current model view. Cloning allows you to keep your original model view intact and make graphical adjustments to the copy (in the new window).

Clone your current view and use the new model view (the copy) to explore the rest of the toolbar.

- On the Window toolbar, click **Clone**  . The new window should be the active window (denoted by the colored title bar). You should also be able to see the original window tiled beneath it.

You are now working in a new model view (in a new window). The former model view will remain intact and unchanged.

Load Viewing Operations

Load viewing functionality is provided to help you display and switch the loads. This will be discussed in more detail after your loads are defined.

Remaining Viewing Buttons

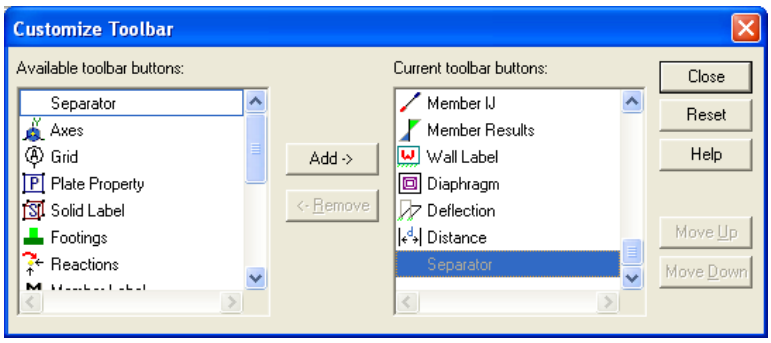
The remaining toolbar buttons control identifying information in the new view. Click each item a few times to turn the viewing buttons on and off.

Customizing the Window Toolbar

All the viewing buttons described above are the defaults of RISA-3D. However, once you become more familiar with RISA-3D, you may decide to add or remove buttons to customize the Window toolbar to make it more useful for your modeling needs.

- On the **Tools** menu, select **Customize Toolbar**.

The **Customize Toolbar** dialog box looks like this:



Note: Please use care not to customize any of the Window toolbar buttons at this time. For the tutorials to be performed accurately, it is imperative that the Window toolbar remain in its default state.

- Click **Close** without making any changes to the toolbar.

Close the cloned view:

- Press Ctrl-F4 to close the cloned view and return to the original view.

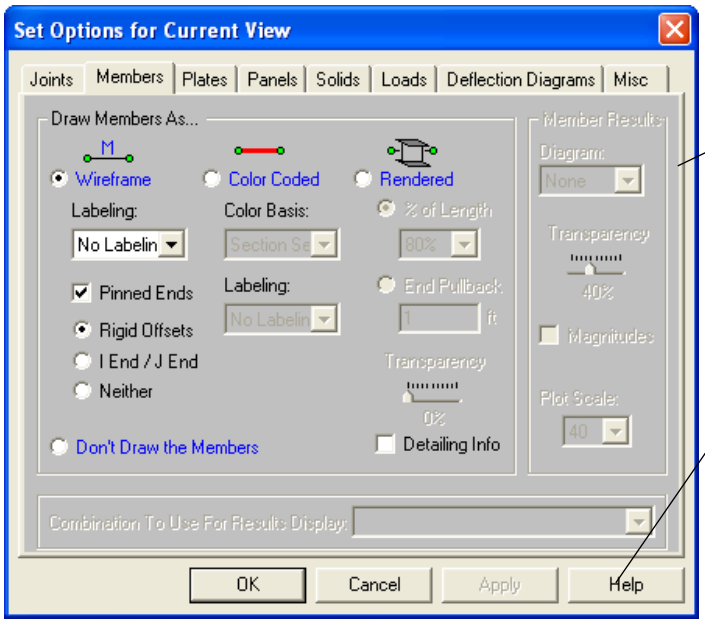
Plot Options

The **Plot Options** dialog box allows you to control the information displayed on your plot.

- On the Window toolbar, click the **Plot Options** button  .

Notice the tabs separating the various types of information.

- Click the **Members** tab.
- Under **Draw Members As**, click each option (**Wireframe**, **Color Coded**, and **Rendered**) one at a time. Notice the dimming of available options changes depending on your selections.



Because the model has not been solved, everything under **Member Results** is dimmed and unavailable.

OK – Apply changes and close the dialog box.

Cancel – Cancel changes and close the dialog box.

Apply – Apply changes and leave the dialog open.

Help – Open the Help Menu for this dialog box.

Next, define the plot options for your model:

- Under **Draw Members As**, click **Color Coded**. Under **Color Basis**, select **Section Set**. Under **Labeling**, select **No Labeling**.
- Click **Apply** to apply the changes and leave the dialog box open.

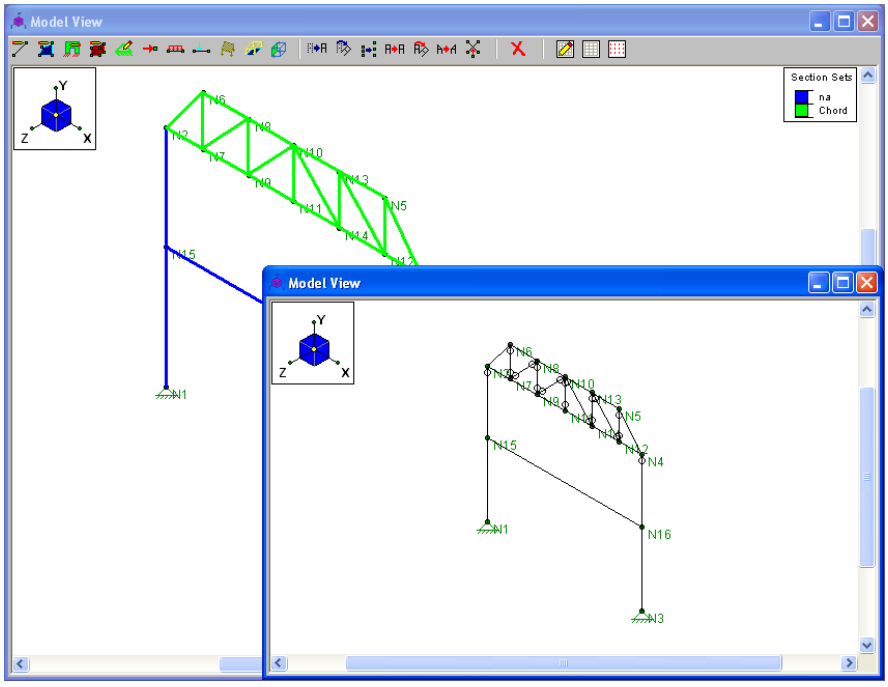
Finish by exploring the options on the other tabs, reviewing the many available options. Be sure to click the **Misc** tab. You may make any of these options your default for future plots by pressing the button at the bottom, **Make the Current Plot Settings the Default**.

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

Multiple Views

Previously, you saw that you could have more than one view using the clone feature. With the clone feature, each view is independent and can be rotated, rendered, selected, etc., without affecting other views. Now you will open a New Model View which simply opens a new model view in the default view, such as you see when you first open the file.

- On the RISA toolbar, click **New Model View**  and you will be presented with a new model view its the original state (as if you just opened the file). In other words, no rendering or view changes will be shown. Feel free to use the buttons on the Window toolbar. You will see that you can adjust this view without affecting your original view.



The importance of multiple views cannot be stressed enough. If you do not want to change your existing view, but need to view the other side of the model, simply open a new window to view the other side. You can have a different drawing grid open in each view. For example, you can draw beams on one grid in one view and columns in another.

As you move between each view, RISA-3D will even keep track of any modeling/model modifications in the new view and update the other views. For example, you might be drawing new beam members in one view, new columns in another view, and plates in a third view and these modeling changes are made in all views. Only the view state (selection state, plot options, etc.) is kept separate for each new view. Any modeling changes are considered global and are made to all views.

Multiple views are also useful when looking at the results because you can plot different information in each view. The options are endless, so make sure that you put this feature to work for you.

- Click **Close** to close the second view.

Selection Tools

Selection tools are necessary to identify and isolate specific parts of the model.

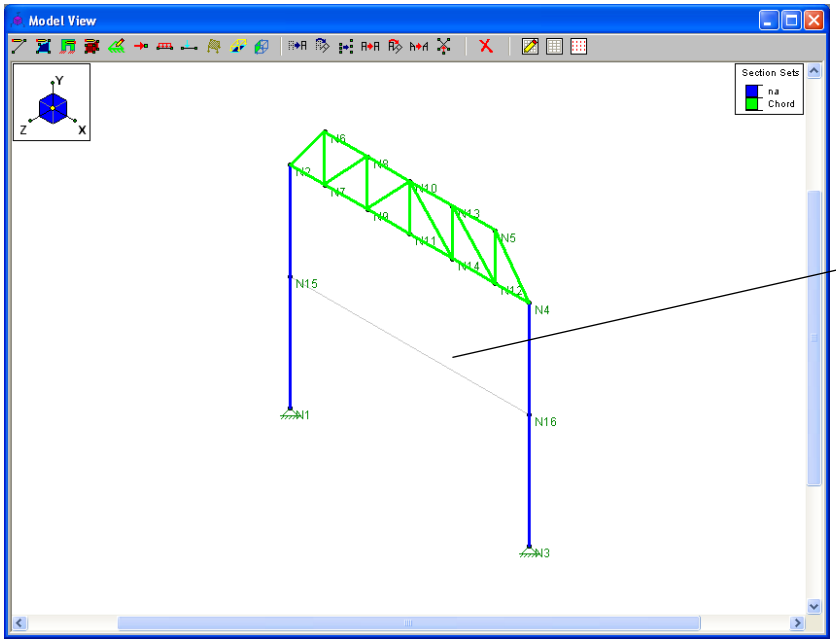
Note: Currently, your cursor looks like this , indicating you are in the default selection mode. While in this mode, click any item to select or unselect it.

Selected items appear in Color Fill view. Unselected items appear as grey wireframe lines. Next, compare the two.

Currently, all your model elements are selected. Unselect the crossbeam:

- Click the crossbeam (between **N15** and **N16**) to unselect it. The color fill beam will become light grey in color, denoting that it is now unselected.

Your model should now look like this:



Making Selections One at a Time

Experiment with selecting and unselecting members and joint locations, and viewing information on them.

- Click a few of the lines to see how the members can be unselected, and then again to select them.
- Click a joint location to select/unselect the joint.
- Double click on a member to open the **Information** dialog box. Click **Close** or **Cancel**.

Using the Selection Toolbar to Make Selections

Rather than clicking to select one item at a time, you may use RISA-3D's Selection toolbar to help you quickly achieve any selection you want.

The Selection toolbar is the vertical toolbar along the left side of the screen.
It provides selection tools to help you work with parts of the model. The toolbar is visible when your model is the active window (it will not be visible when working on a spreadsheet).

Experiment with making selections using the selection tools. Feel free to try all of them as you cannot affect the model solely by selecting items.

Note: When using the select and unselect tools, the cursor will remain in that selection tool mode so that you can make multiple selections. To terminate the chosen selection mode, click the right mouse button.

The selection tools are described below:

	Select All and **Unselect All** Select/unselect the entire model.
	Box Select and **Box Unselect** Draw a box around the part to be selected/unselected. Only items that are *entirely* in the box will be affected.
	Polygon Select and **Polygon Unselect** Draw a polygon around the part to be selected/unselected (double click to end). When you have finished drawing the polygon, double click to complete the selection.
	Line Select and **Line Unselect** Draw a line through all items to be selected or unselected. Simply click the mouse to begin drawing the line, drag to draw the line, then release the mouse to complete the selection. This tool is ideal for selecting the columns or web members in a truss.
	Invert Selected Used to invert, or reverse, the selected state of the model. All selected items are made unselected, and all unselected items are made selected. This can be *very* useful when just a few items are to be selected in a large model; just click to unselect the items, then click **Invert Selected** to invert (or "flip") the selection.

	Criteria Selection Allows you to use other criteria to make your selections. Using the **Select Items for Current View** dialog box, you can: (1) Select items between defined coordinates; (2) Select items using their labels, such as all joints from N7 to N15; (3) Select members by defining multiple types of criteria such as orientation, section set, database shape, and material; and (4) Select plates by defining material, thickness, and orientation.
	Save/Recall Selection Save or recall selection states of the model. If the model is altered after a selection state has been saved, the saved selection state will also be altered. After saving a selection state, any new items (joints, members, etc.) will be set to "selected." To save a selection state, click **Save** and provide a name for the saved selection. You can have up to 16 different saved selections in a model. To retrieve a saved selection, choose the selection state from the list and click **Retrieve**. To delete a saved selection, choose the selection state from the list and click **Delete**.
	Lock Unselected (unlocked) and (locked) Lock all currently unselected parts of the model. When **Lock Unselected** is pressed, this selection tool changes to . When locked, all unselected parts of the model will remain unselected and invisible--no matter what other selection buttons are pressed. To turn the lock off, press **Lock Unselected** a second time (returning it to the unlocked position). This is useful to isolate one part of the model from the rest. For example, if you are designing one particular floor in a multi-story building, and you need to select and unselect different parts of that floor, lock the part of the model you do not want affected. Specifically: click **Unselect All** to unselect the entire model; click **Box Select** to select the floor; then press **Lock Unselected** (the button will change to). The rest of the model will now remain unselected until you press again to unlock it.

When you are finished experimenting with the selection tools:

- Ensure that the **Lock Unselected** button is unlocked.

Now that you have explored the various selection tools, use them to modify the model.

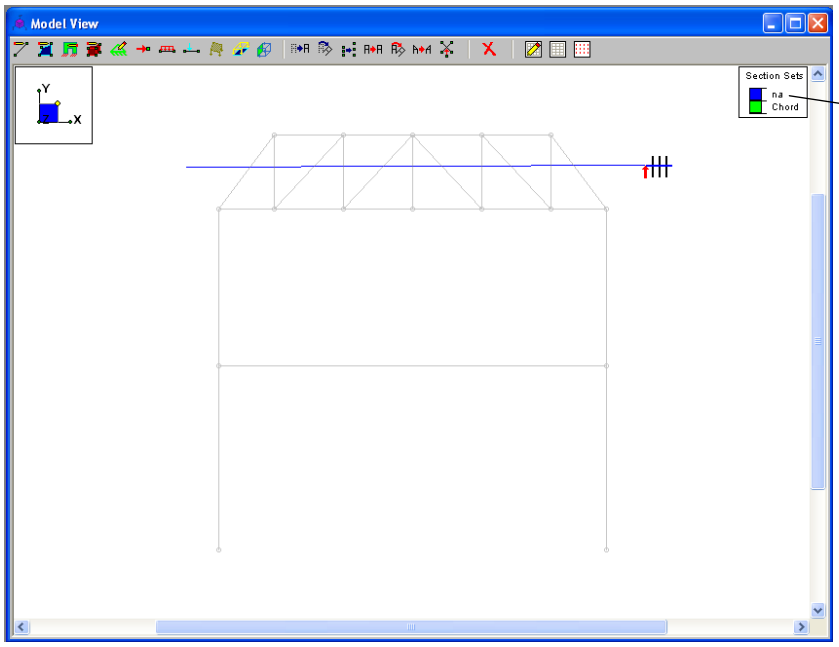
Making Changes to Selections

Unselect the entire model and change the viewing plane:

- On the Selection toolbar, click **Unselect All** to unselect the entire model.
- On the Window toolbar, click the **XY Planar** button to change the model view to an XY elevation. Then click **Redraw** to maximize the viewing plane.

Make some selections using the **Line Select** button:

- Click the **Line Select** button  . Your cursor will change to  , indicating that you are now in the line selection mode.
- Draw a line through the truss web members, as shown below. Click your mouse to begin drawing the line near the leftmost truss web member, and drag past the rightmost web member, then release the mouse.



Notice the section set for all of your web members is listed as Chord. This is because you only had one Hot Rolled section set available to you when you first drew your truss.

All of the truss web members should now be selected. You could now easily modify those selected members, as you will do in the next section.

Modify Members

In this section, you will use the **Modify** tools to rotate your columns.

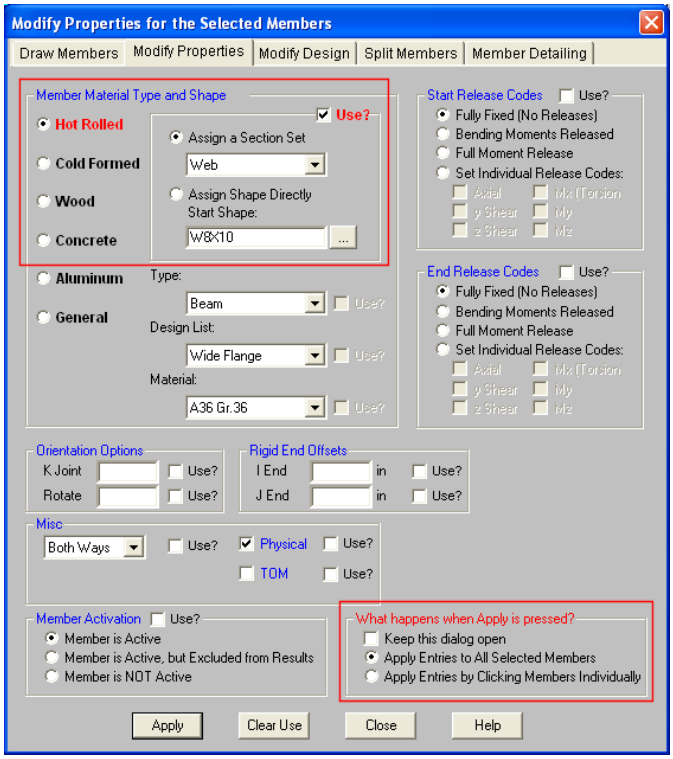
- On the Drawing toolbar, click **Draw New Members**  .
- Select the **Modify Properties** tab.

Note: Each section of the dialog box contains a **Use?** check box. Selecting any of these **Use?** check boxes determines whether or not the parameter will be applied.

This dialog box looks similar to the **Draw Members** dialog box because, after the members are drawn, this dialog box allows you to change any of those initial parameters.

- Under **Member Material Type and Shape**, click **Hot Rolled**.
- Click **Assign a Section Set**, and select **Web**.
- At the top of the **Member Material Type and Shape** section, select the **Use?** check box. Once selected, the text will turn red.
- Under **What Happens when Apply is pressed?** click **Apply Entries to All Selected Members**.

The dialog box should now look like this:



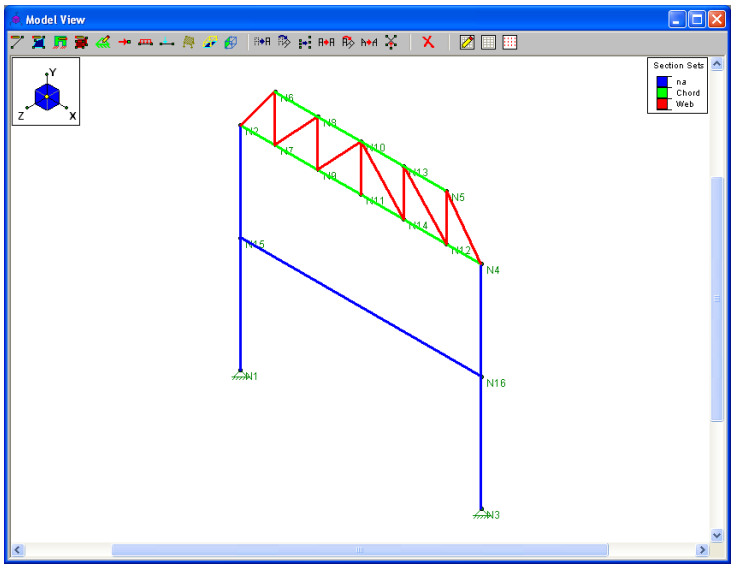
Apply the changes and close the dialog box:

- Click **Apply**.

Modify the section set for all of the selected members:

- On the Selection toolbar, click **Select All** to select the entire model.
- On the Window toolbar, click **Isometric** to display an isometric view of the entire structure.

Your model should now look like this:



Model Rendering

A powerful graphics tool in RISA-3D is the ability to display a rendered image of the model. A rendered image shows each member plotted using a true scale representation of the shape assigned to the member.

A rendered view offers some important benefits: besides producing a very detailed image, it also allows you to easily review member orientations and member connectivity. There are also rendering options (within the **Members** tab of the **Plot Options** dialog box) that allow you to adjust pullback length and rendering transparency.

Turn on the model rendering:

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

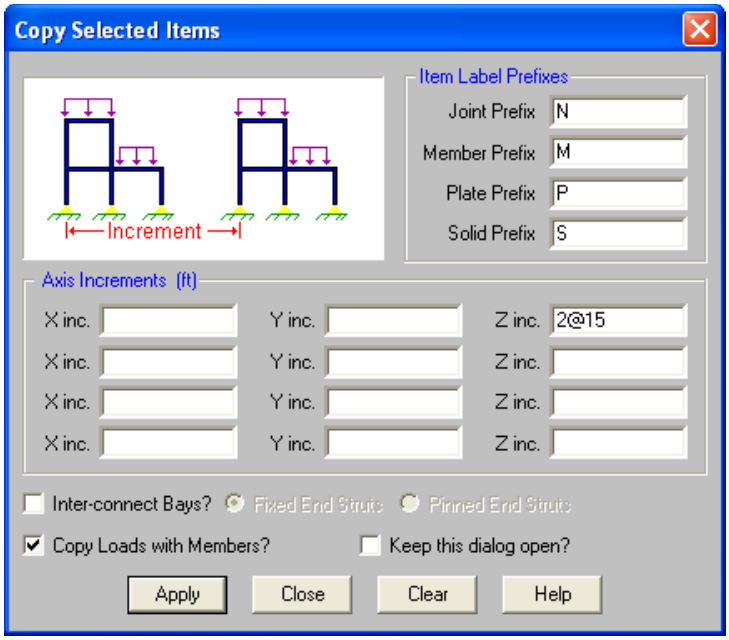
Copying Model Elements

RISA-3D allows you to copy selected portions of the model. You will use this feature to turn your two-dimensional model into a three-bay structure.

- On the Drawing toolbar, click **Linear Offset Copy** to open the **Copy Selected Items** dialog box. This feature is used to copy the currently selected portion of the model (in increments) in any or all three of the global directions.

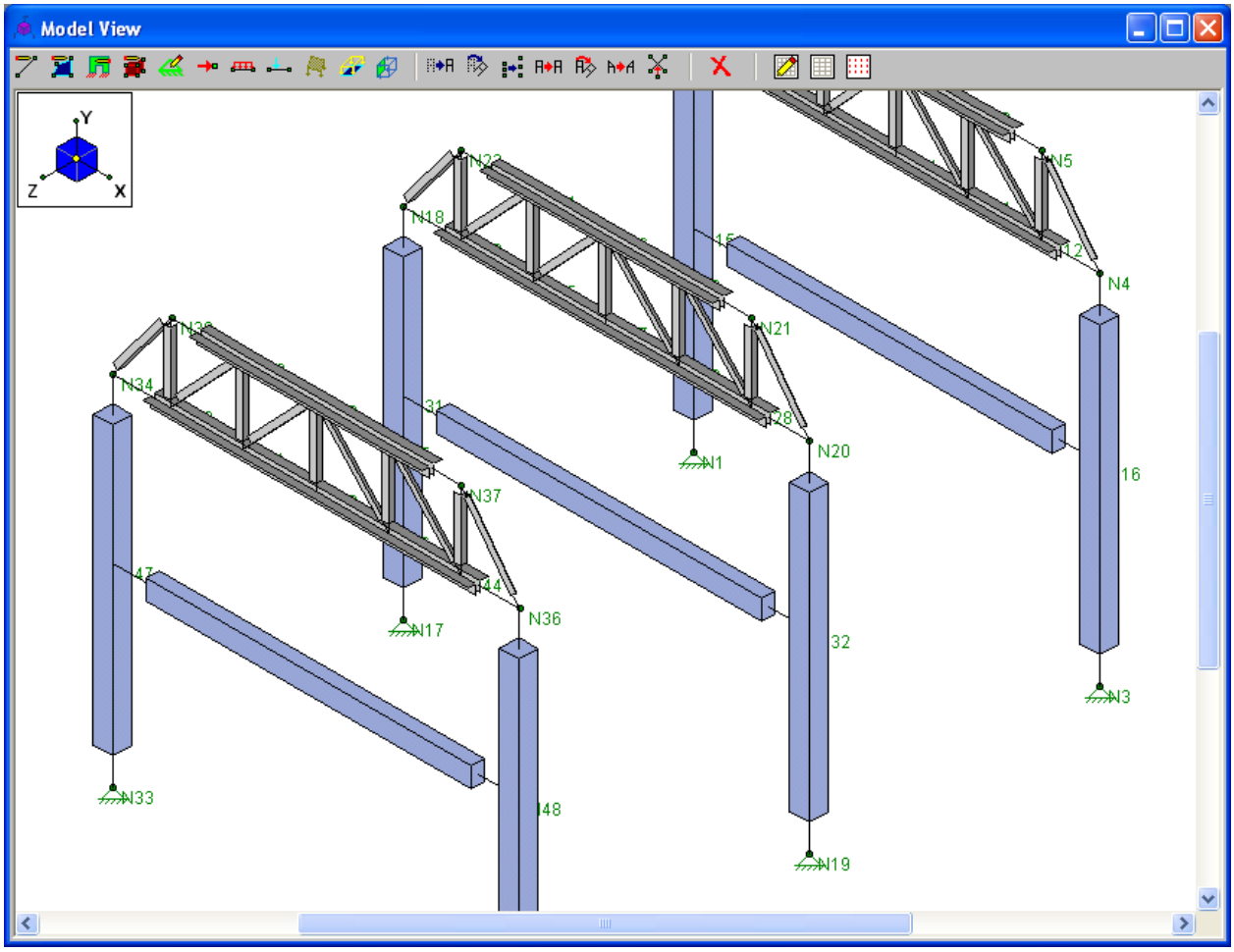
Notice there are several fields used to specify increments in the X, Y, and Z directions. Use the "@" symbol to specify multiple equal increments. For this tutorial, replicate your model in the Z direction.

- Under **Axis Increments**, in the first **Z inc.** column, type **2@15**.



- Click **Apply**.

Because your entire frame was selected, all of the frames will be copied. If you had any loads on the frames, they would have also been copied. Your structure should look like this:



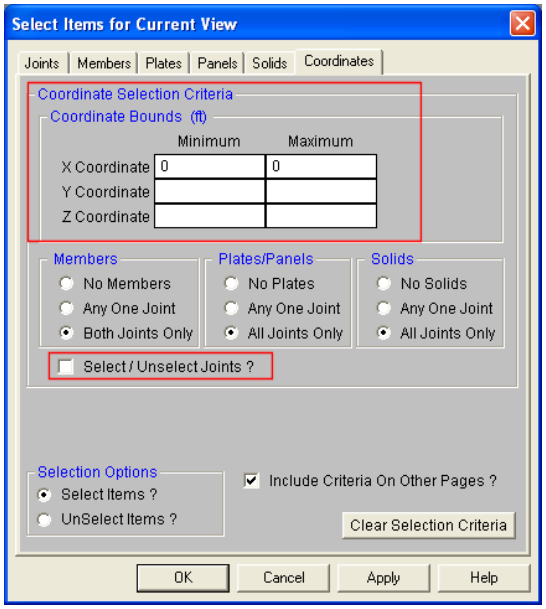
Next you will add a masonry wall panel to the back side of your structure. But before you draw the wall panel, delete the concrete columns from this side of the structure:

- On the Window toolbar, click **Redraw** to view the entire model.
- On the Selection toolbar, click **Un-Select All**.

Now that the entire model is unselected, use the Selection Criteria tool to select just the back three columns.

- On the Selection toolbar, click the **Criteria Selection** button. Click the **Coordinates** tab.
- Under **Coordinate Bounds**, in the **X Coordinate** box, type **0** in both the **Minimum** and **Maximum** columns.
- Near the bottom of the **Coordinate Selection Criteria** section, click to clear the **Select/Unselect Joints** check box so that joints on the columns will not be selected.

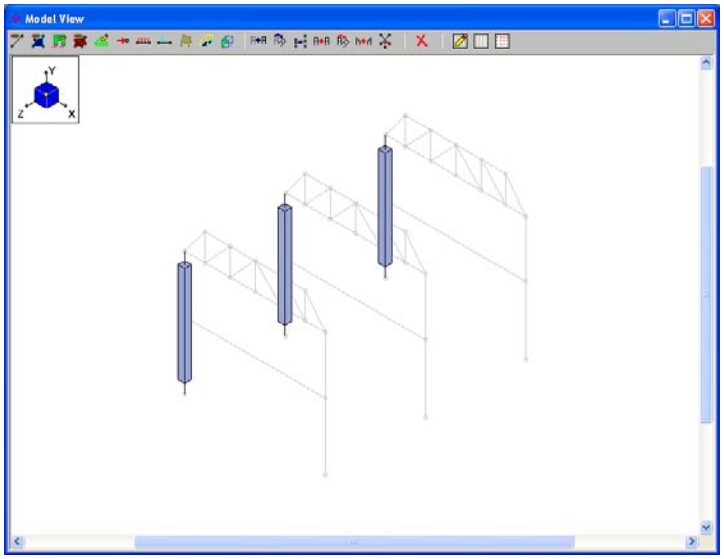
Your dialog window should look like this:



Close the dialog box and return to the model view.

- Click **OK**.

Notice that just the back three columns are selected:



Now, delete these columns.

- On the Drawing toolbar, click **Delete** to open the **Delete Items** dialog box.
- Click **Delete Based on This Criteria**, then select the **Delete Selected Members** check box.
- Click **Apply**.

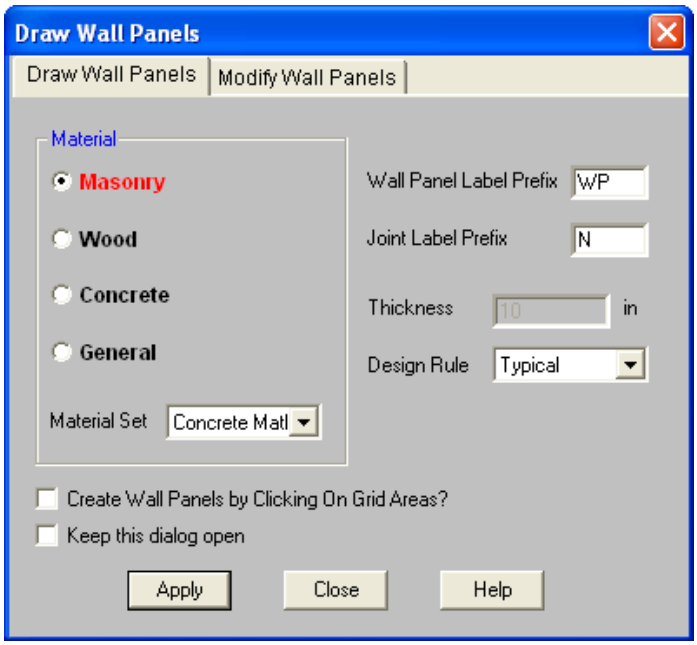
Notice that the three columns have now been deleted.

- Click **Select All** to select the entire model once again.

Drawing Wall Panels and Additional Framing

Now, draw in the wall panel:

- On the Drawing toolbar, click **Draw New Wall Panels**  .
- Click **Apply** to draw in the wall panel using the default values.

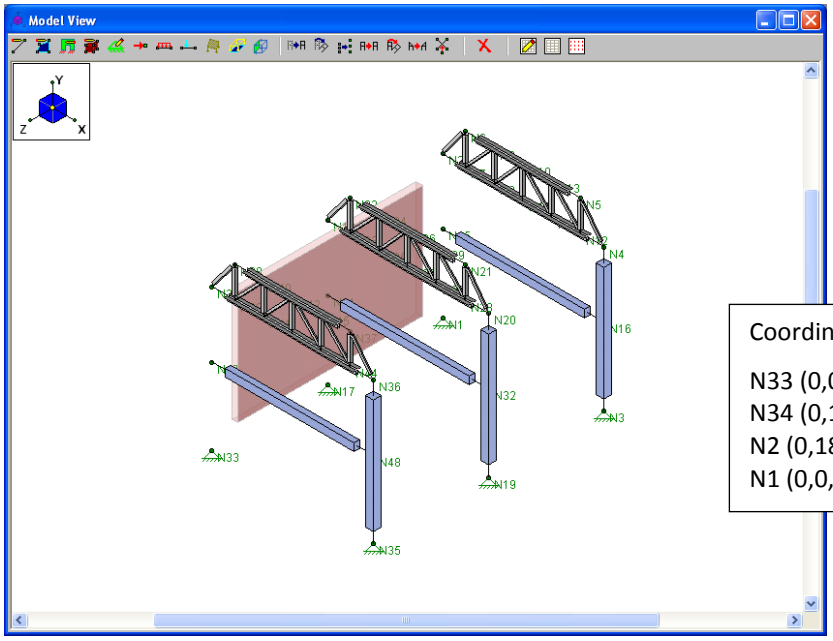


Note: Your cursor will now change to  , indicating you are ready to draw your wall panel.

Begin drawing the wall panel:

- Click each node to define the four corners of the structure: start with **N33** at front lower left (coordinates **0,0,30**), then move clockwise to **N34** (coordinates **0,18.5,30**), **N2** (coordinates **0,18.5,0**), and finally **N1** (coordinates **0,0,0**).
- When finished drawing the wall panel, press the right mouse button or ESC to exit the drawing mode.

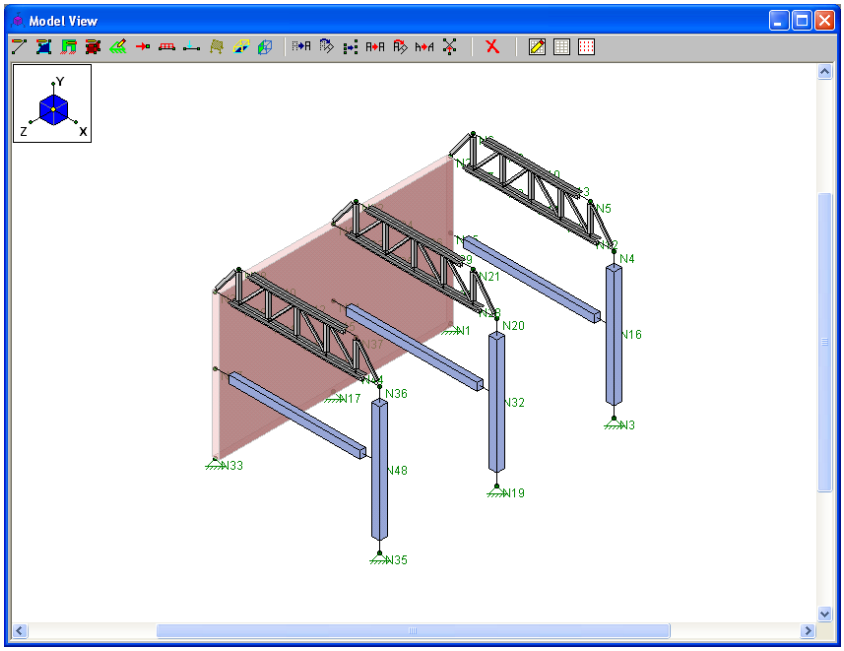
Your screen should now look like this:



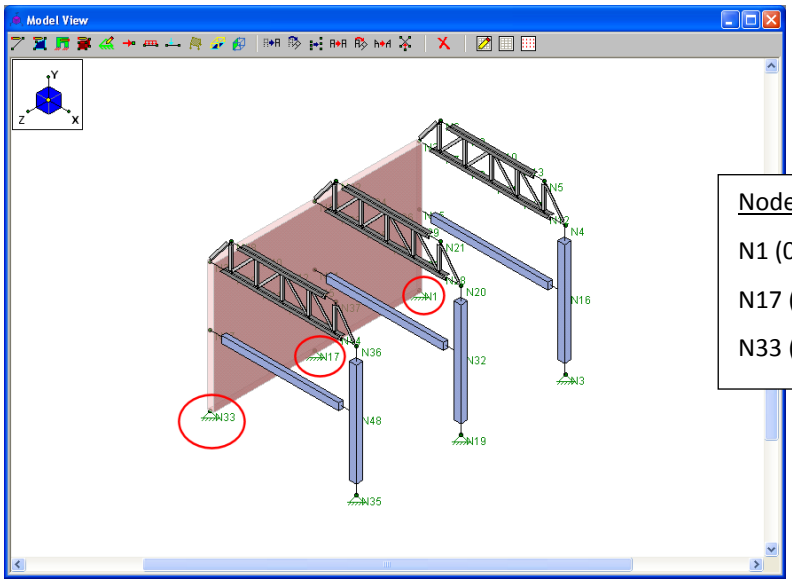
Notice the default rendered wall panel does not fully extend to its corner nodes. This is because the default view for wall panels has a rendered size of 80%. Change this to a size of 100% for a better view:

- On the Window toolbar, click **Plot Options** . Select the **Panels** tab.
- Under **Draw Wall Panels As**, click **Rendered**. In the **Size** list, click **100%.**
- Click **OK** to exit the dialog box and return to your model.

Your wall should now be rendered at 100%.



Notice that the three pinned boundary conditions still are present at the base of the wall. During solution, the wall panel elements will automatically generate a continuous boundary condition at the base, therefore you will want to delete these three boundary conditions to avoid duplicate restraint.



Nodes to delete:

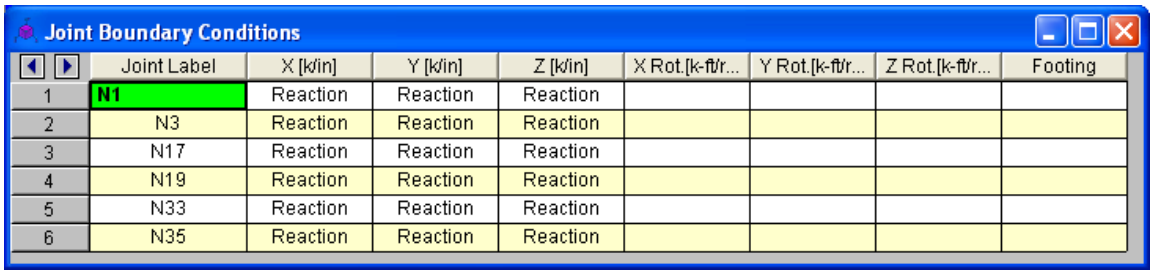
N1 (0,0,0)

N17 (0,0,15)

N33 (0,0,30)

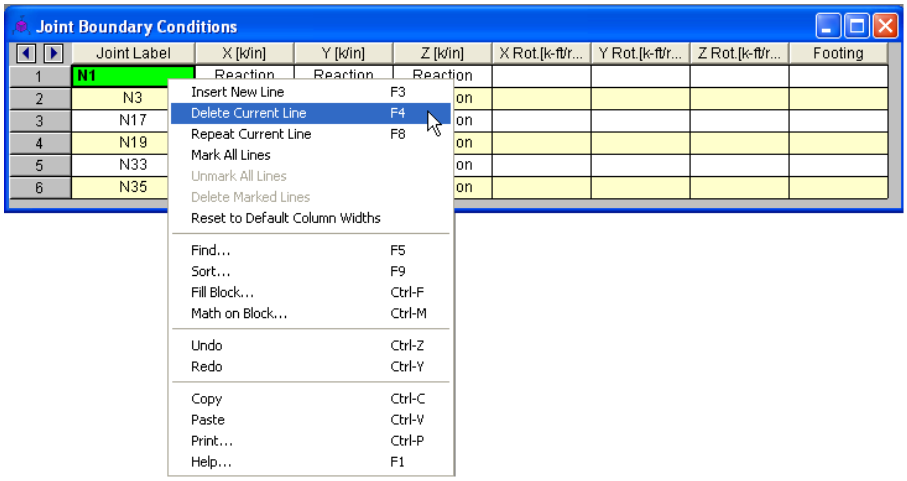
Make note of the boundary condition labels and then delete them directly out of the **Boundary Conditions** spreadsheet:

- On the **Spreadsheets** menu, select **Boundary Conditions**.
- Click in the **N1** cell to highlight it in green as shown below:



	Joint Label	X [k/in]	Y [k/in]	Z [k/in]	X Rot.[k-ft/r...	Y Rot.[k-ft/r...	Z Rot.[k-ft/r...	Footing
1	N1	Reaction	Reaction	Reaction				
2	N3	Reaction	Reaction	Reaction				
3	N17	Reaction	Reaction	Reaction				
4	N19	Reaction	Reaction	Reaction				
5	N33	Reaction	Reaction	Reaction				
6	N35	Reaction	Reaction	Reaction				

- Right-click your mouse and select **Delete Current Line** from the shortcut menu.



- Repeat for the remaining two boundary conditions (N17 and N33).

When complete, your **Boundary Conditions** spreadsheet should be left with just three entries, as shown below:

Joint Boundary Conditions

	Joint Label	X [k/in]	Y [k/in]	Z [k/in]	X Rot.[k-ft/r...	Y Rot.[k-ft/r...	Z Rot.[k-ft/r...	Footing
1	N3	Reaction	Reaction	Reaction				
2	N19	Reaction	Reaction	Reaction				
3	N35	Reaction	Reaction	Reaction				

- Close the spreadsheet.

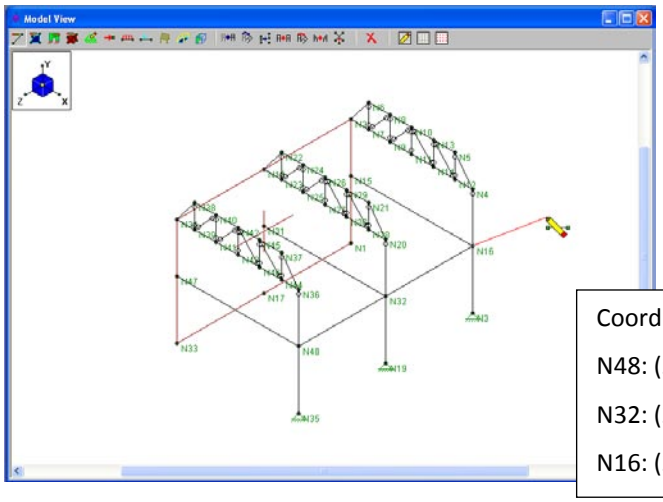
This was just a simple overview of the masonry wall panel feature. For more detailed information on this feature and how to use it to set up design rules, regions, and openings for both masonry and wood walls, refer to the Wall Panel Tutorials available on our website: http://www.risa.com/d_documentation.html.

Next, prepare to draw the beams on the front of the structure.

- First, click the **Toggle Rendering** button  once to go back to wireframe view.
- On the Drawing toolbar, click **Draw New Members**  .
- Click the **Draw Members** tab.
- Under **Member Material Type and Shape**, click **Hot Rolled**. Click **Assign a Section Set** and select **Beam-Z** from the list.
- Under **Release Codes**, click **Fully Fixed at Both Ends**.
- Click **Apply**, and get ready to draw.

Draw the beams:

- Click the joint in the middle of the front column, **N48** (coordinates **21,10,30**), to define the I-end of the first member. Then click the middle joint on the middle column, **N32** (coordinates **21,10,15**) to define the J-end of the first member and the I-end of the second member. Finally, click the middle joint of the right-most column, **N16** (coordinates **21,10,0**), to define the J-end of the second member.



Coordinate Locations:

N48: (21,10,30)

N32: (21,10,15)

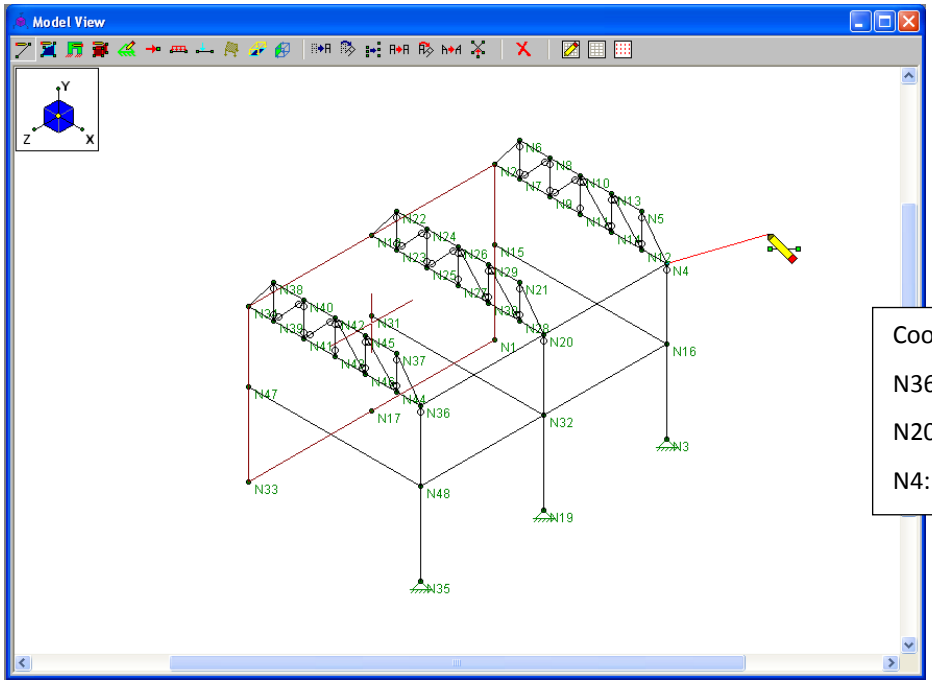
N16: (21,10,0)

Now, draw in glulam beams bewteen the top nodes of the same columns.

- Click CTRL+D to reopen the **Draw Members** dialog window.
- Under **Member Material Type and Shape**, click **Wood**. Click **Assign a Section Set** and select **Glulam** from the list.
- Under **Release Codes**, click **Fully Fixed at Both Ends**.
- Click **Apply**, and get ready to draw.

Draw the beams:

- Click the joint in the middle of the front column, **N36** (coordinates **21,18.5,30**), to define the I-end of the first member. Then click the middle joint on the middle column, **N20** (coordinates **21,18.5,15**) to define the J-end of the first member and the I-end of the second member. Finally, click the middle joint of the right-most column, **N4** (coordinates **21,18.5,0**), to define the J-end of the second member.



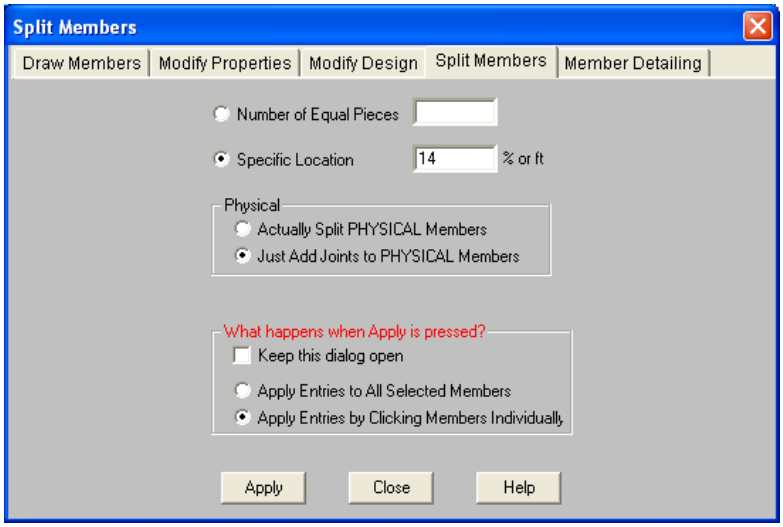
- When finished, click ESC twice to exit the drawing mode.

Drawing Along Members

Next, you will add cold formed steel girts to the columns between the beams. Because there are no joints at these locations, you can either create a drawing grid (as you did in Tutorial 1), or you can insert joints on the columns (as you will do next).

- On the Drawing toolbar, click **Draw New Members**  . Click the **Split Members** tab.
- Click **Specific Location** and type **14** in the box.
- Under **Physical**, click **Just Add Joints to Physical Members**.
- Under **What happens when Apply is pressed?**, click **Apply Entries by Clicking Members Individually**.

Your dialog box should now look like this:



Close the dialog box and begin to draw:

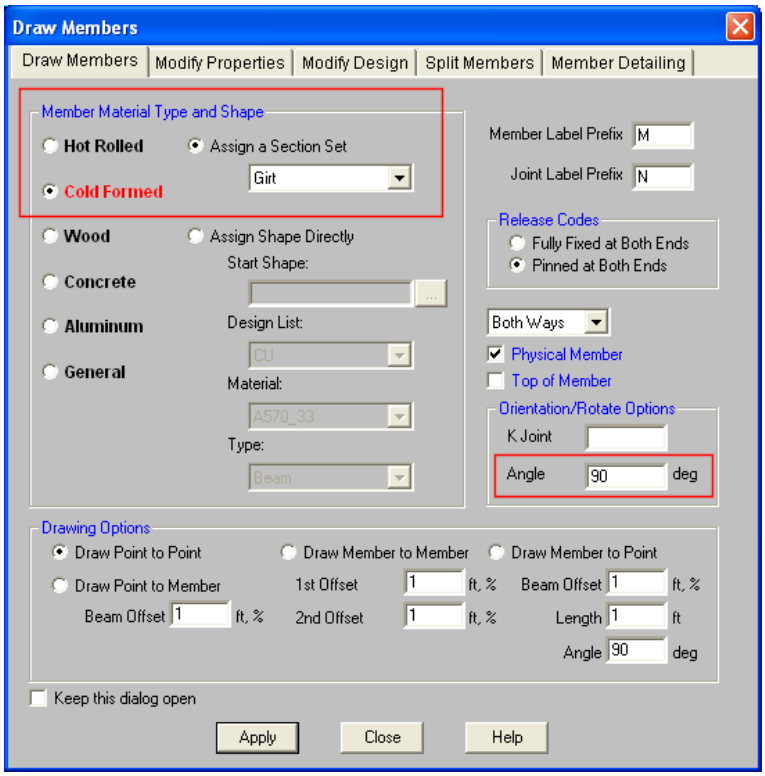
- Click **Apply**.
- Click once on each of the three columns in the front (the same columns you just connected with beams).

New nodes will appear on the upper half of the columns between the two beams. You may use the mouse wheel to zoom in to see them closer.

- Press CTRL+D to re-open the **Draw New Members** dialog box. Click the **Draw Members** tab.
- Under **Member Material Type and Shape**, click **Cold Formed**. Click **Assign a Section Set**, and select **Girt** from the list.
- Under **Release Codes**, click **Pinned at Both Ends**.

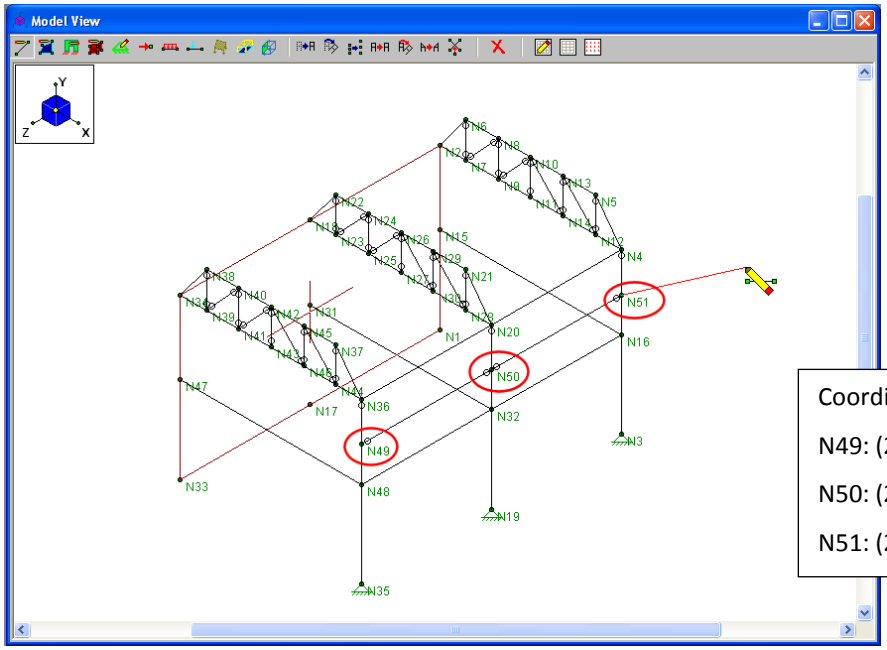
You also want the cold formed members to be turned so that their strong axis resists wind load. Specify the rotation before drawing them:

- Under **Orientation/Rotate Options**, in the **Angle** box, type **90**.



Close the dialog box and prepare to draw again.

- Click **Apply** and you are ready to draw.
- Click on the three new nodes, from left to right, **N49** (coordinates **21,14,30**), **N50** (coordinates **21,14,15**), and **N51** (coordinates **21,14,0**). The three new nodes are shown circled in the figure below.
- When finished, right click to release the mouse.



Coordinate Locations:

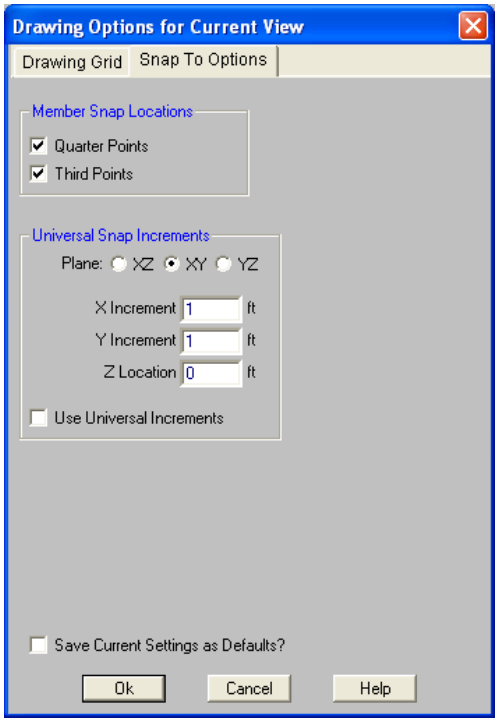
N49: (21,14,30)

N50: (21,14,15)

N51: (21,14,0)

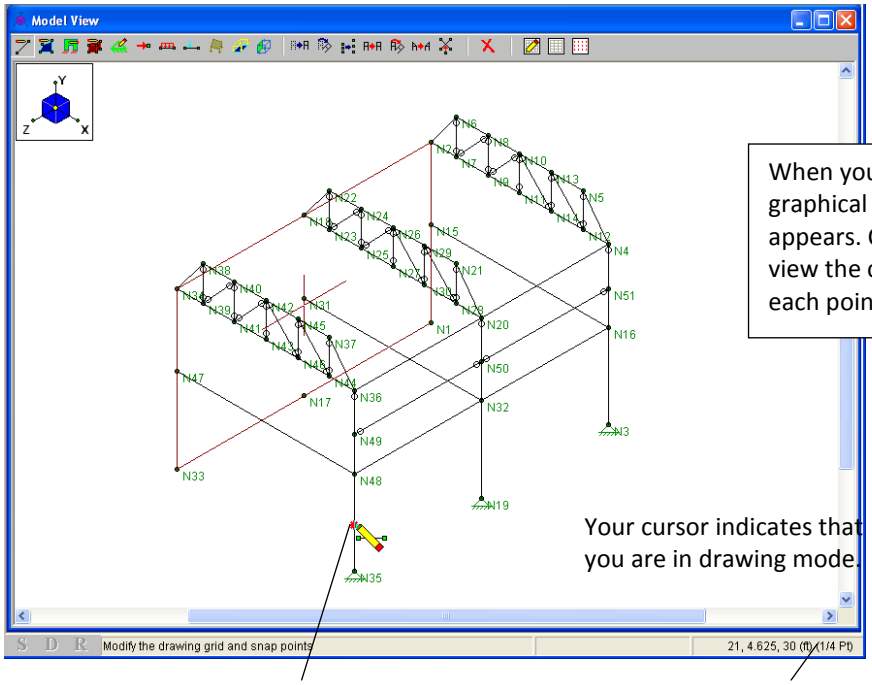
Next, modify the drawing grid to make the addition of the next members easier. These settings allow you to set up a grid of graphical editing points along any orthogonal plane. They also allow you to automatically snap to the quarter or third points of existing members.

- On the Drawing toolbar, click **Modify Drawing Grid**  . Click the **Snap To Options** tab.
- Under **Member Snap Locations**, select the **Quarter Points** and **Third Points** check boxes (if not already selected).
- Under **Universal Snap Increments**, make sure the **Use Universal Increments** check box is cleared, as shown below:



Continue drawing your cold formed girts:

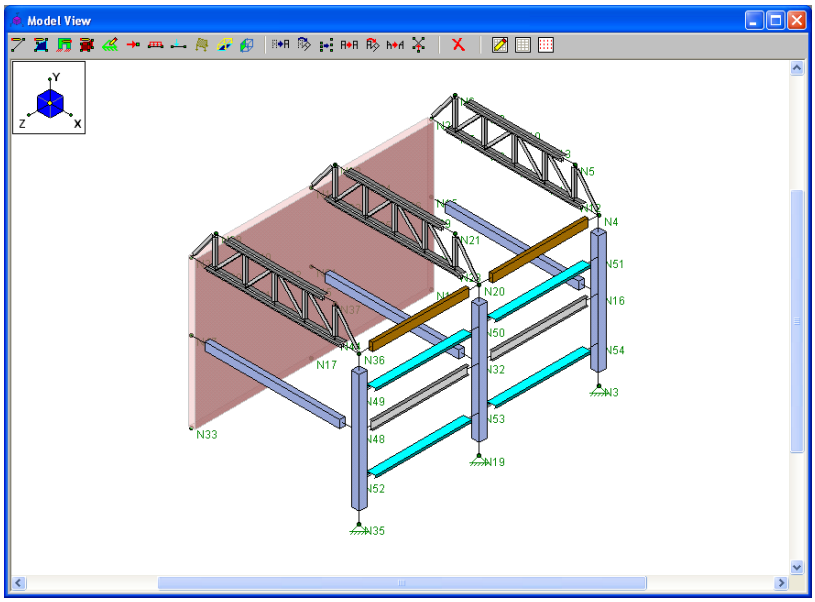
- Click **Ok** to close the dialog box and begin drawing.



Draw the lower girts using the drawing points defined at the quarter points of the columns.

- Click the left column coordinate (**21, 4.625, 30**). Then click the middle column coordinate (**21, 4.625, 15**) and the right column coordinate (**21, 4.625, 0**).
- When you are finished drawing, right click to release the mouse. Right click a second time to exit the drawing mode.
- Click the **Toggle Rendering** button  twice to return to rendered view.

When you are finished your screen should look like this:



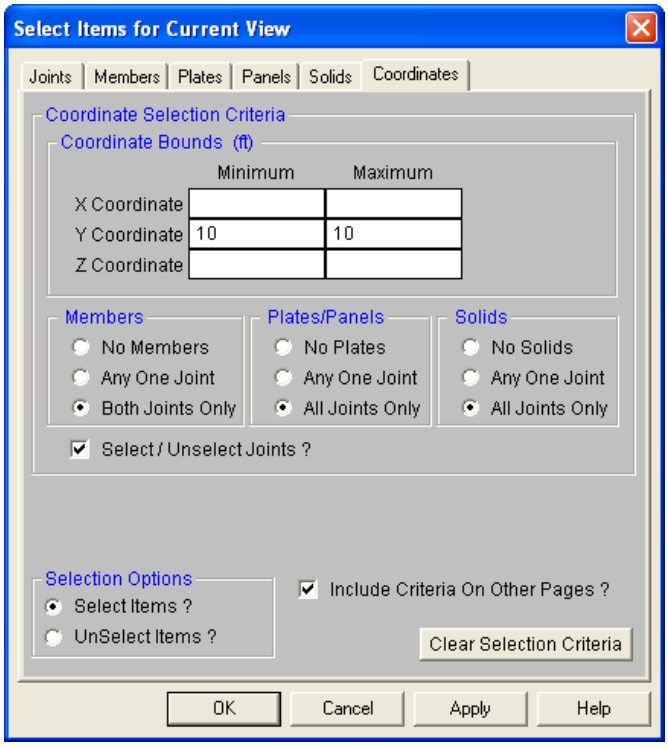
Selecting Planes

Next, you will add a floor composed of plate elements. Use **Criteria Selection** again to select the joints to facilitate drawing at the 10 foot elevation. This selection option allows you to select elevations, plans, or any block of the model.

Unselect the entire model and define the coordinate elevation as follows:

- On the Selection toolbar, click **Unselect All** .
- Also on the Selection toolbar, click **Criteria Selection** .
- Select the **Coordinates** tab.
- Click the **Clear Selection Criteria** button to clear the entries from the previous selection.
- Under **Coordinate Bounds**, in the **Y Coordinate** box (in the **Minimum** column), type **10**. Press TAB to move to the **Maximum** column and type **10** again.

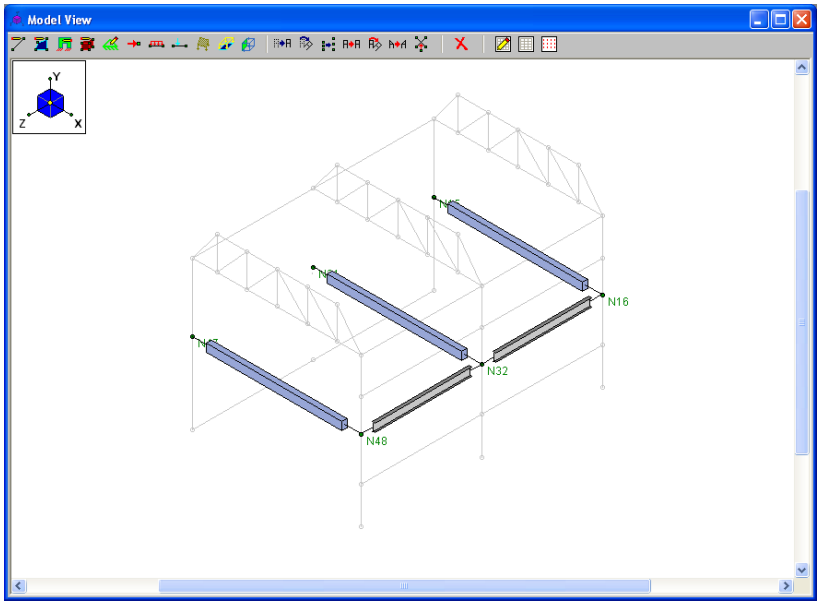
The dialog box should look like this:



Close the dialog box:

- Click **OK.**

Items at the 10 foot elevation are selected like this:

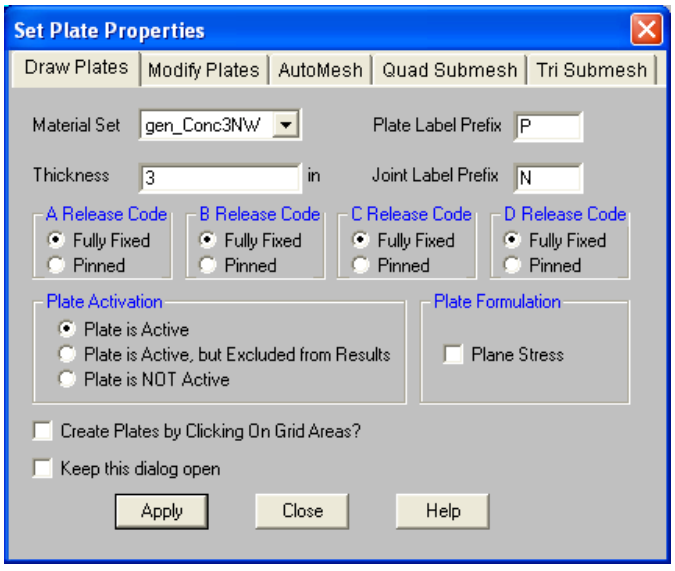


Drawing Plates

Next, define the floor by drawing and submeshing plates.

- On the Drawing toolbar, click **Draw New Plates** .
- In the **Material Set** box, select **gen_Conc3NW**. In the **Thickness** box, type **3**.

The dialog box should look like this:



Close the dialog box and return to wireframe view for easier drawing.

- Click **Apply** to close the dialog box and enter the drawing mode.

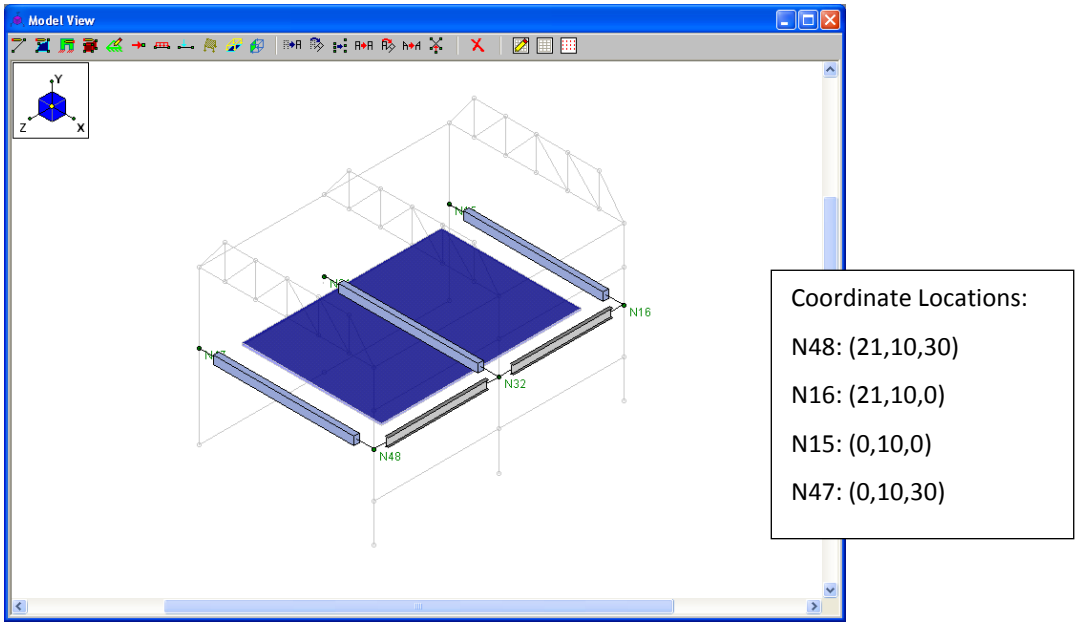
Note: The cursor changes to , indicating you are in the plate drawing mode.

- Click the **Toggle Rendering** button  once to return to wireframe view.

Now draw in the plates.

- To begin, click the closest corner joint (**N48**) then move counter-clockwise and click the joints (**N16**), (**N15**), and finally (**N47**). As you move, notice the double lines defining the interior of the plate element.
- Right click to exit the drawing mode.
- Click the **Toggle Rendering** button  twice to return to rendered view.

The first plate appears in rendered form with its thickness shown to scale, as shown below:



Sub-meshing Plates

How many plate elements do you need to get accurate results? Many books have been written addressing the topic of "mesh size." The finite element method is an "approaching method." This means the more fine your mesh, the more accurate your results. See the *RISA-3D General Reference* for parametric studies that demonstrate this, along with some design examples for shear walls, horizontal diaphragms, and spread footings. Our studies have shown that, to get accurate results, you should try to have a 4x4 mesh of plates between points of support and load concentrations.

RISA-3D provides two distinct submeshing options: *Automesh* and *Quad Submesh*.

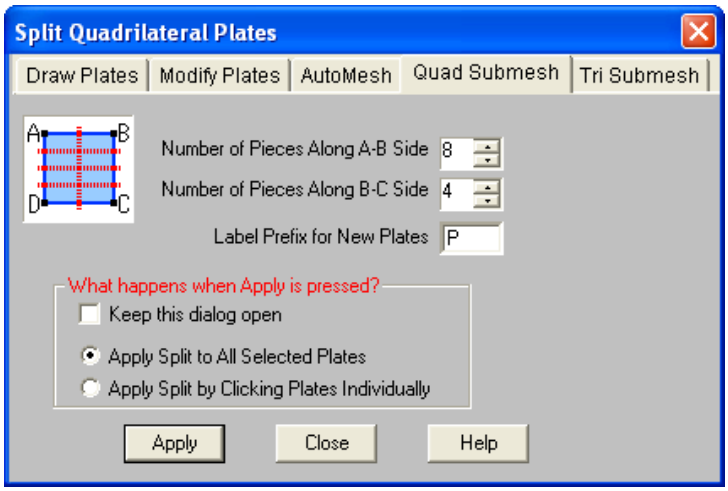
Automesh is the most advanced of the two options. This feature allows you to draw any arbitrary polygon and have RISA-3D fill the polygon with well-sized quadrilateral plates. The AutoMesh feature will not be demonstrated in this tutorial; however, you are encouraged to learn more about AutoMesh in the *RISA-3D General Reference*.

Quad Submesh is the simpler option. It involves creating a single plate and then replacing that plate with a submesh of smaller plates. You will use this option to mesh the plates next.

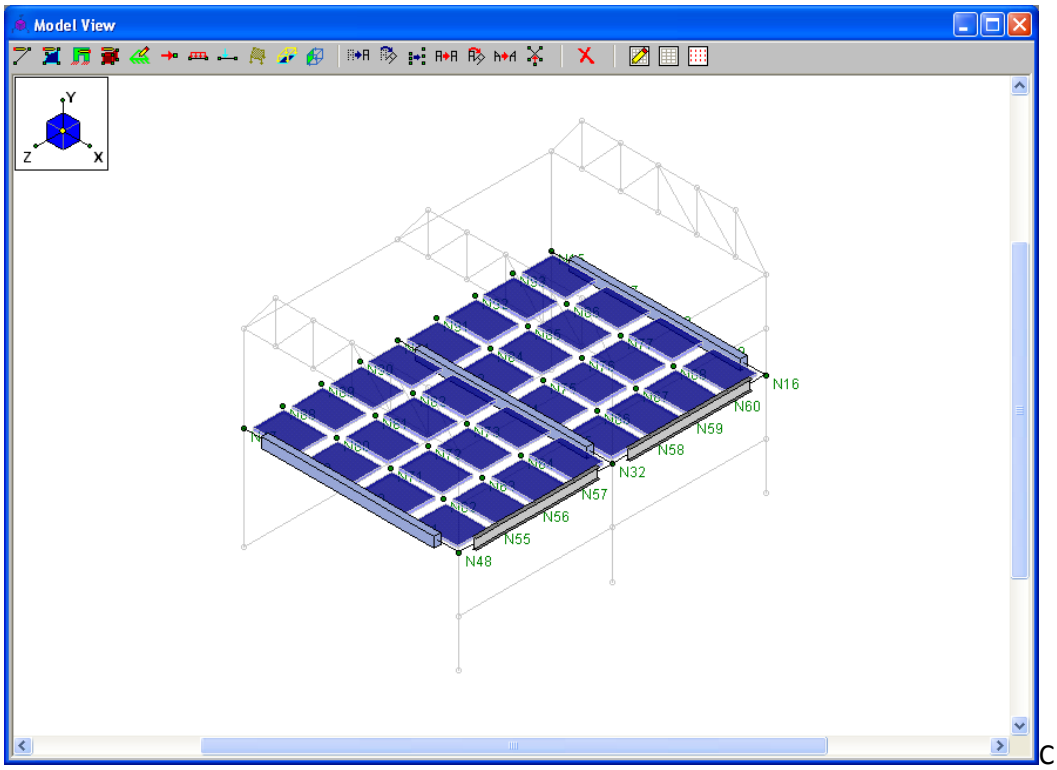
- Press CTRL+D to display the dialog box again.
- Click the **Quad Submesh** tab.

The default entries would divide your plates into a 4x4 submesh which would create rectangular submeshed plates, so modify it to create square plates for more accurate results.

- In the **Number of Pieces Along A-B Side** box, type **8**.
- Click **Apply Split to All Selected Plates**.
- Click **Apply**.



You should now have a 4x4 mesh of plates in each bay, as shown below:

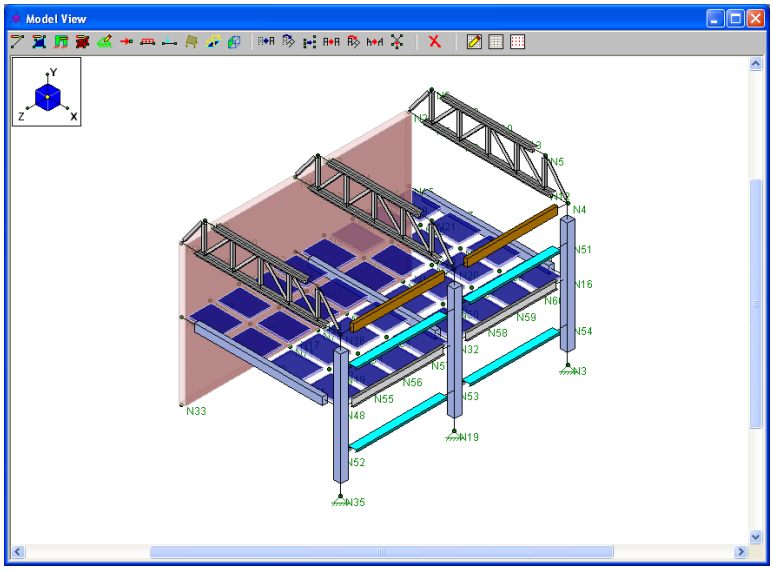


C

Select the model and continue:

- On the Selection toolbar, click **Select All** .

You should now see:



Notice that submeshing the plates has added new joints along the existing beam members. Because of the Physical Member feature, these new joints are automatically attached.

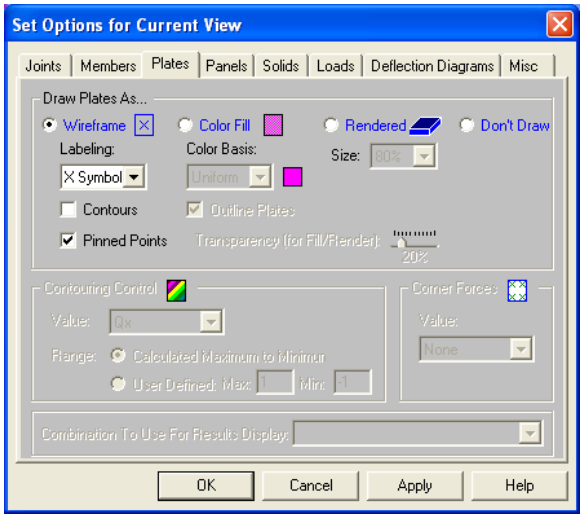
The plates are shown at 80% rendered view, which can be adjusted in the **Plot Options** dialog box. The shrink factors only apply to the length and width of the plates, not the thickness. The full thickness is always shown. You may also notice that duplicate joints have been created at the middle columns. Later, you will use the Model Merge feature to eliminate those.

Viewing Plates

Remove the labels and display the plot options for plates:

- On the Window toolbar, click **Joint Labels** to turn them off.
- Also, on the Window toolbar, click **Plot Options**. Select the **Plates** tab.

These options control how the plates are displayed graphically, and are similar to the member display buttons.



The four main drawing options:

Wireframe – view in wireframe.
Color Filled – view in color fill and plotted as a surface.
Rendered – renders the plot
Don't Draw – will not draw the plates.

Note: Both **Color Filled** and **Rendered** may have a transparancy factor applied.

0% = fully opaque
100% = completely transparent

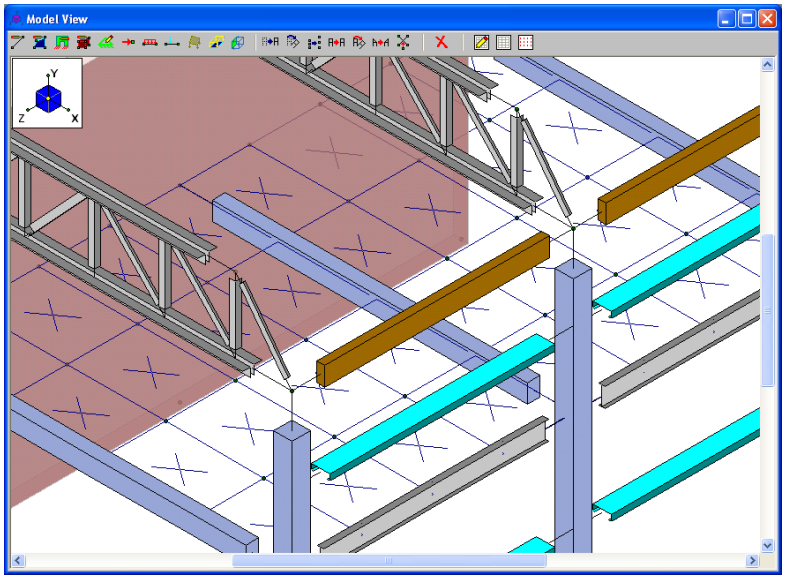
Display the plates in wireframe and take a closer look at them:

- Under **Draw Plates As**, click **Wireframe**. Under **Labeling**, select **X Symbol**.
- Click **OK**.

Now, zoom in and take a closer look at the wireframe plates.

- Press the PLUS SIGN [**+**] on the numeric keypad four times to zoom in.

You should now see this:



Return to full model view:

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

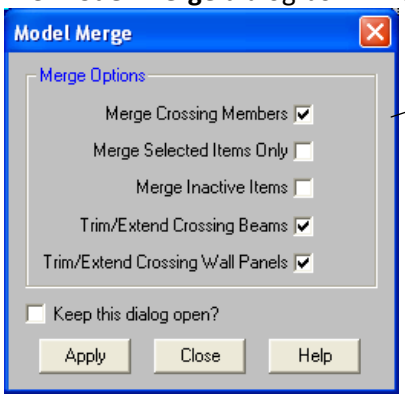
Model Merge

The Model Merge feature searches for duplicate nodes and members (with the same coordinates) and merges them into one.

Use Model Merge to merge the duplicate nodes and members:

- On the Drawing toolbar, click **Model Merge** .

The **Model Merge** dialog box will appear:



Merge Crossing Members - Members crossing each other will be merged (a typical case might be X-bracing).

You have the option of merging the entire model or only a selected part of the model.

- Click **Apply**. You will be presented with the results of the merge.
- Click **OK** to close the **Model Merge Results** dialog box.

Scaling Elements

Assume your client has just submitted a change order to raise the truss from 18'-6" to 22'-9", and would like the frame 25% wider (from 21' to 26'-3"). The client would like the completed design this afternoon!

These are significant changes to the model. If you were using a batch input program or a program heavily reliant on "parametric" data generators, you would probably be required to start the data entry process completely from scratch to create the new model.

A major strength of RISA-3D is the ability to make these types of changes to existing data, realizing that data modification is probably one of the most common repetitive tasks, it is therefore the true test of any software product. RISA-3D allows you to make all the requested changes graphically (using the built-in graphical editing tools), or numerically (using the spreadsheets), or a combination of both—whichever you prefer.

For this tutorial, you will make the changes graphically. Later, an explanation will be given about how you could have accomplished the same changes within the spreadsheets.

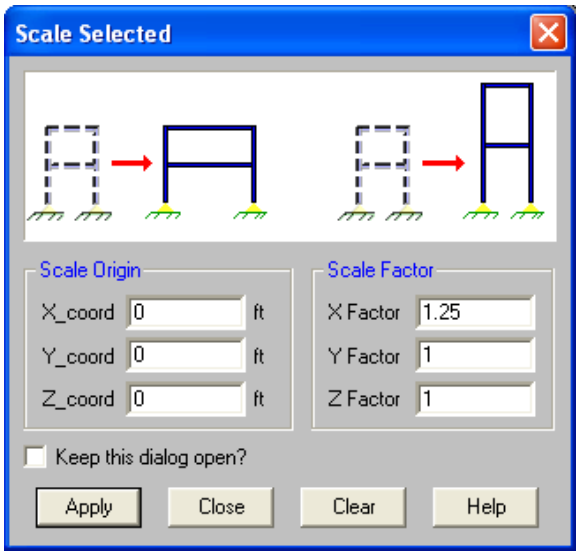
To begin, widen the truss using the **Scale** feature.

- Click the **Modify** menu and select **Scale**.

Note: On the Drawing toolbar, there is also a **Scale Factor Move** button  .

To widen the truss, scale the model in the global X direction:

- Under **Scale Factor**, in the **X Factor** box, type **1.25**.



The **Scale Origin** is the point that will remain stationary as the selected items are scaled.

Scale Factor defines the scale factors in each global direction.

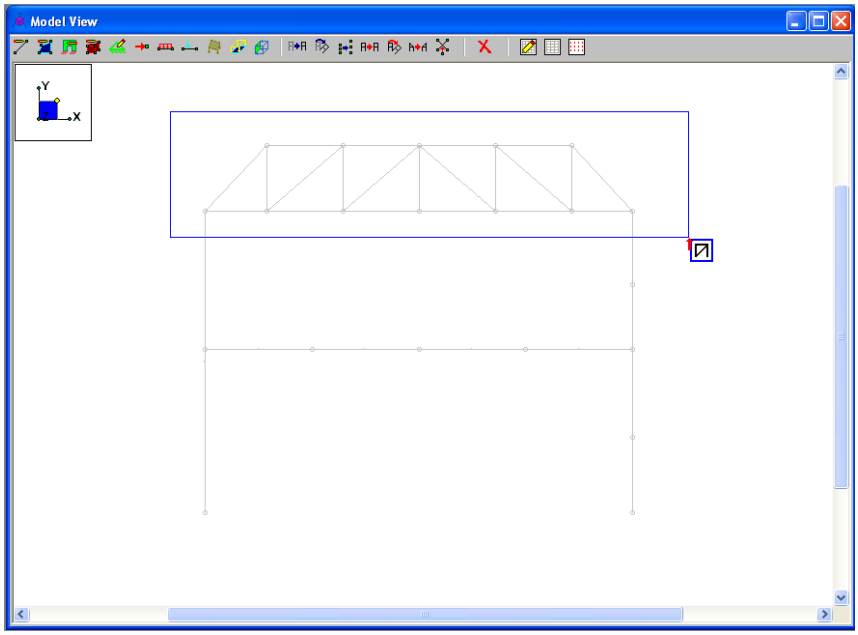
- Click **Apply**.

You are now finished widening the truss. Next, you will raise the truss. You cannot use Scale feature to raise the truss because it would also lift the crossbeam. So instead, use the Move feature. Because you want to move the truss and nothing else, you must first make a selection of the truss.

It will be easier to work with an elevation, so change the elevation, then select your truss:

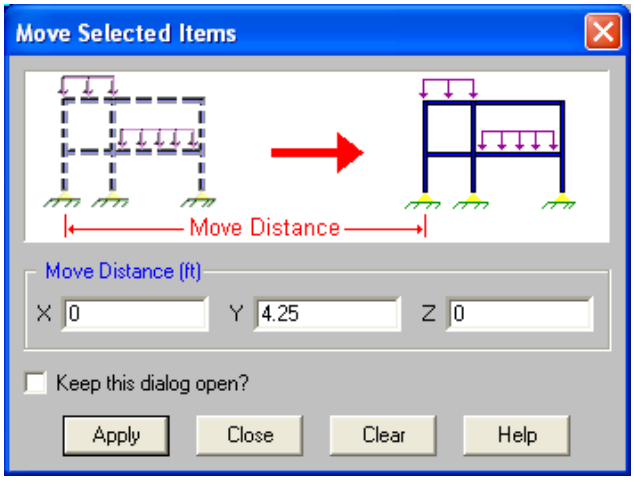
- On the Window toolbar, click **XY Planar**  . Then, click **Redraw**  .
- On the Selection toolbar, click **Unselect All**  . Then, click **Box Select**  . Draw a box around the entire truss. Right-click to release the selection tool.

Your selection of the truss should look like this:



Change the elevation as follows:

- On the Drawing toolbar, click **Linear Translation Move**  . A dialog box will appear.
- Under **Move Distance (ft)**, double-click in the **Y** box and type **4.25**.



- Click **Apply**.

Note: You could have also used the Block Math feature to accomplish both modifications numerically in the spreadsheets. To widen the structure, you would have simply multiplied the joint X coordinates by 1.25. To raise the truss, you would have first sorted the Y coordinates and then selected the coordinates that are at an elevation of 8.5 ft or higher. The final step would have been to use the Block Math feature to add 4.25 to these selected Y coordinates.

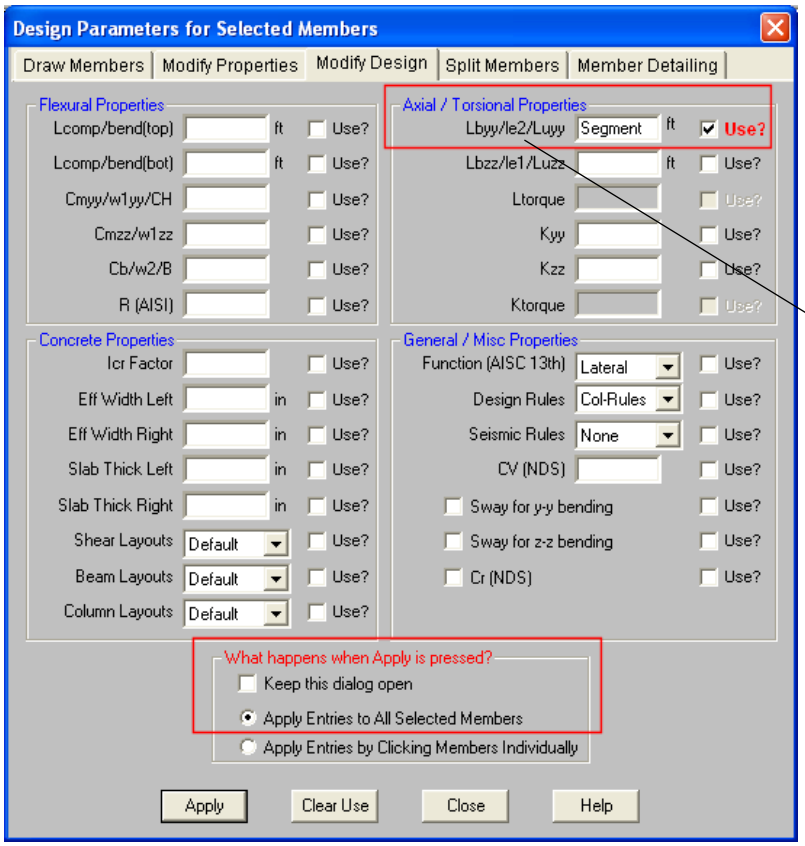
Now, select the model and change your view to prepare for the next step:

- On the Selection toolbar, click **Select All** .
- On the Window toolbar, click **Isometric** . Then, click **Redraw** .

Design Parameters

Before solving the model, steel and timber design code parameters, such as unbraced lengths, must be specified. Once again, you will assign these values graphically. To view the parameters for any one member you can also double-click that member. The values will be recorded on the **Member Design Parameters** spreadsheet.

- On the **Modify** menu, select **Design Parameters**. Click the **Modify Design** tab (if it is not already selected).
- Under **Axial/Torsional Properties**, in the **Lbyy/le2/Luyy** box, type **Segment** (or **Seg**). Also, select the **Use?** check box. (It will display in red once it is selected.)
- Under **What happens when Apply is pressed?**, click **Apply Entries to All Selected Members** as shown below:



Many parameters have three labels separated by a slash (/), indicating steel/ timber/concrete design parameters.

Each field is described below. Refer to the **Help** or *RISA-3D General Reference* for more information.

Currently, all the options are blank except the first. For the options left blank, RISA-3D will use the default values when the code check calculations are performed. For weak axis bending, as a result of using the **Segment** code, each node will be treated as a brace point.

- Click **Apply**.

To actually see the settings for the unbraced lengths, you can double-click any member and review the Unbraced Lengths in the **Design** tab. This information is also accessible from the **Members** spreadsheet.

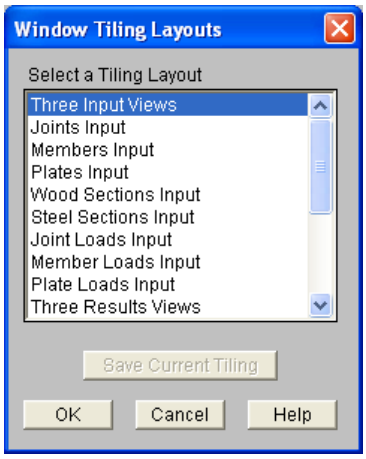
RISA-3D calculates most of these parameters for you, although you may choose to specify your own values. The table on the next page explains how each parameter is handled, enabling you to take advantage of the capabilities of RISA-3D.

Lb/le/Lu	Unbraced lengths with respect to column type buckling. These are used in the KL/r calculations as well as the allowable axial strength.
Lb-comp/le-bend	Unbraced lengths of the compression flanges for buckling due to flexure. These are used in the calculation of allowable bending strength.
K factors	Effective length factors for column type buckling.
Sway Flags	These flags indicate whether the member is to be considered subject to sidesway or not. These are considered in the calculation of K factors, Cm and Cb factors.
Repetitive Member	This is the Cr factor used in NDS wood design.
Design Rules	Design rule for design optimization.
Beam Layouts	Beam reinforcement layout for flexural design. This can be set to default for program design, or set to a custom rebar layout.
Column Layouts	Column reinforcement layout for flexural design. This can be set to default for program design, or set to a custom rebar layout.
Cm/w1/CH	Interactive bending coefficients. CH is the NDS Timber design shear stress factor.
Cb/w2/B	Lateral-torsional buckling modification factors.
R (AISI)	Per section C3.1.3 of the AISI. Used to calculate the moment capacity of cold formed beams that have one flange fastened to decking.
Icr Factor (ACI)	Cracked moment of inertia factor for concrete design.
Cv (NDS)	This is the Cv (volume) factor used for NDS wood design.
Eff Widths (ACI)	Effective widths for concrete T-beam and L-beam design.
Slab Thicknesses (ACI)	Slab thicknesses for concrete T-beam and L-beam design.
Function	Function for stiffness reduction per the *AISC 13th Edition Steel Code* Direct Analysis Method design.
Shear Layouts	Beam reinforcement layout for shear design. This can be set to default for program design, or set to a custom rebar layout.

Sorting

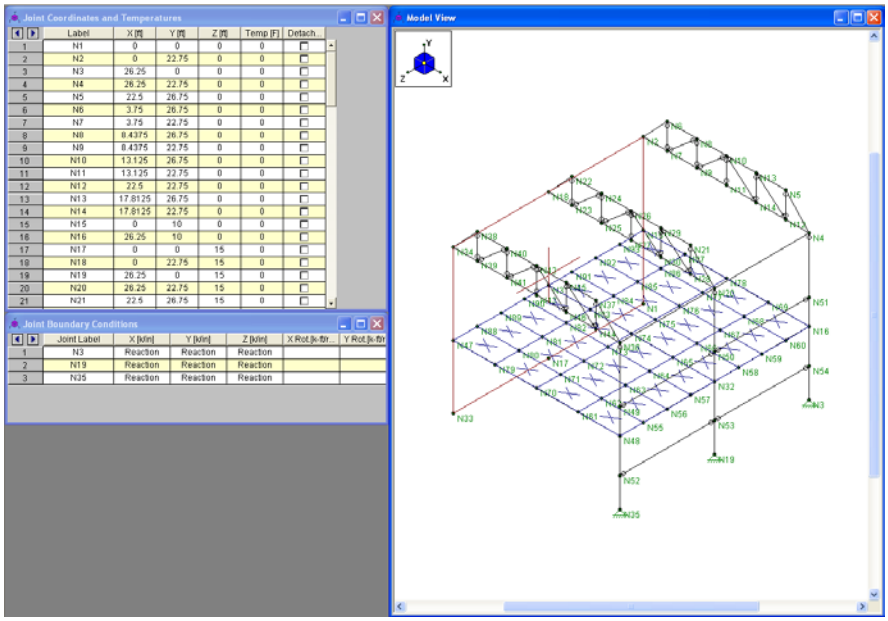
Next, you will sort information in your model using the **Joint Coordinates** spreadsheet for the first time. You will also use a special window arrangment to do this.

- On the RISA toolbar, click the **Tiling** button.



- Select **Joints Input**.
- Click **OK**.

This tiling option has opened the **Joint Coordinates** spreadsheet in the upper left corner and placed the **Joint Boundary Conditions** spreadsheet below it.



The joints are listed in the order in which they were created as you defined your model. However, it would be more useful to list them based on their coordinate locations. Sort the joints in ascending X coordinate order:

- On the **Joint Coordinates** spreadsheet, click anywhere in the **X** column.
- On the Window toolbar, click **Sort**  , select **Sort Low to High**, and click **OK**.

Next, add a Z-coordinate sort:

- Click anywhere in the **Z** column.
- Right-click the mouse, select **Sort**  , click **Sort Low to High**, and click **OK**.

Last, sort on the Y-coordinate:

- Click anywhere in the **Y** column.
- Right-click the mouse, select **Sort**  , click **Sort Low to High**, and click **OK**.

Now you have the joints in ascending Y coordinate order, and at each Y coordinate level. They are also sorted in ascending Z and X coordinate order. Notice that the labels stayed with their assigned joints, and the model plot looks exactly the same.

Relabel the joints as follows:

- On the **Tools** menu, select **ReLabel Joints.**
- Click **OK**.

As you can see from the model view window, the joints are now relabeled from the bottom up. Next, sort the members.

- On the RISA toolbar, click **Tiling** again.
- Select **Members Input**.
- Click **OK.**

This displays all spreadsheets that contribute to the definition of members. The **Members** spreadsheet is located in the lower left corner. Use this to sort the members, first by their I-Joint:

- On the **Member Primary Data** spreadsheet, click anywhere in the **I-Joint** column.
- Right-click the mouse, select **Sort**  , click **Sort Low to High**, and click **OK**.

Then sort by the section set:

- Click anywhere in the **Section/Shape** column.
- Right-click the mouse, select **Sort**  , click **Sort Low to High**, and click **OK**.

Again, the member labels remain with the same members and the view has not been affected by the sort (because nothing has actually changed other than how the members are arranged in the spreadsheet).
Next, sort your beams by section. This will make them easier to identify and work with in your next operation (assigning parameters for steel and wood design):

- On the **Tools** menu, select **Relabel Members**.
- Click **OK**.

This is the end of Tutorial 2. The next tutorial demonstrates how to load the model.

You may save your model to be used as the starting point for the next tutorial, or begin the next tutorial using the .r3d starter file in the RISA-3D **Tutorials** folder. To save the model:

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

Tutorial 3 – Loading

In this tutorial, you will apply loads and explore how to generate load combinations from pre-defined load cases. This tutorial continues where the previous tutorial ended, so follow these steps to get your model up and running:

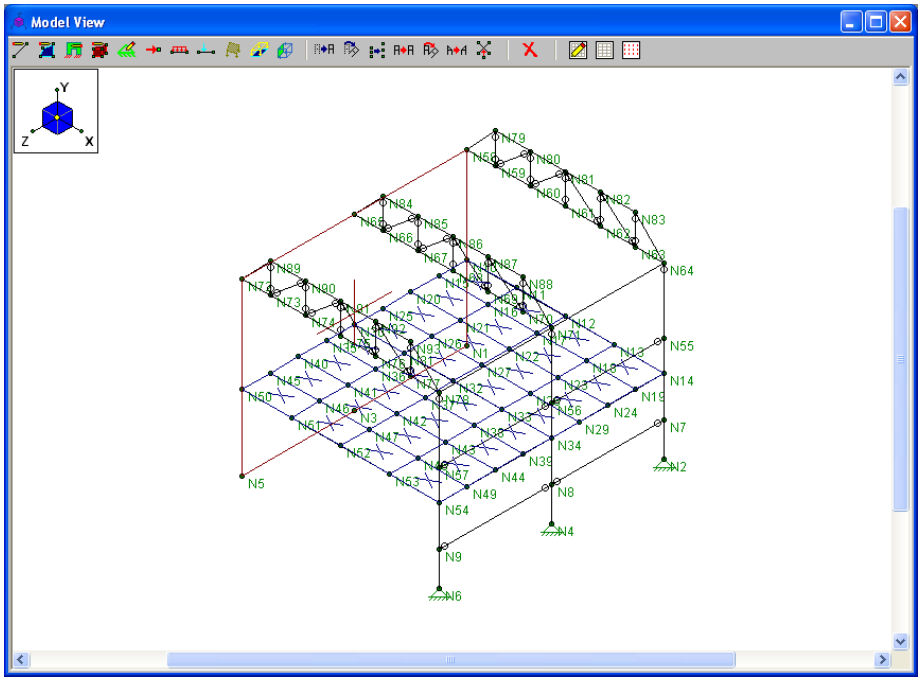
If you are continuing from the previous tutorial:

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

- Double-click the **RISA-3D** icon to start the program.
- Click **Open Model**.
  Double-click the **Tutorials** folder, select **Tutorial 3 Starter.r3d** and click **Open**.
  Click **Close** (or **Cancel**) to exit the **Global Parameters** dialog box.
- On the Window toolbar, click the **Graphic Editing Toolbar** button to activate the Drawing toolbar.
- On the **Data Entry** toolbar, click **Close** to close it.

Your screen should look like this:



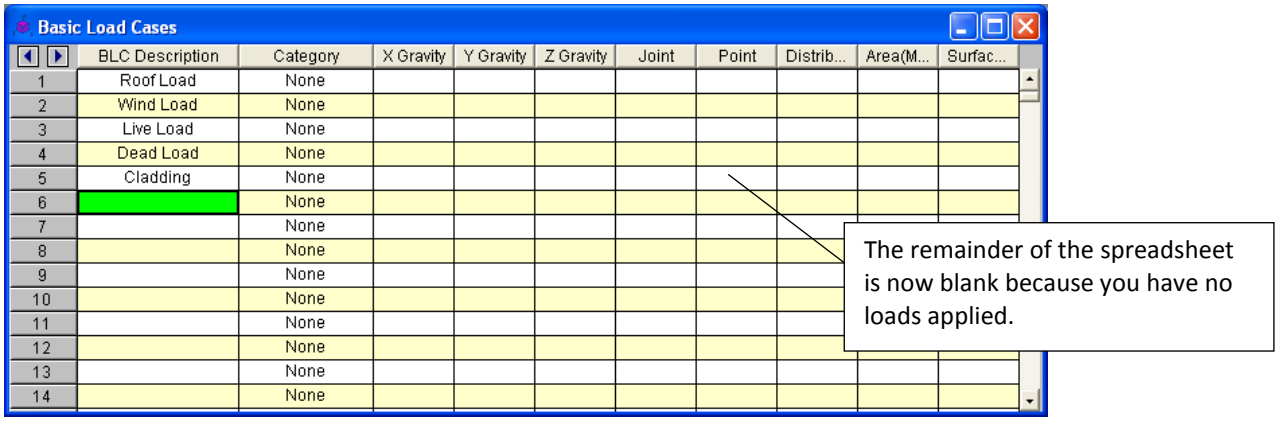
Load Cases, Categories, and Combinations

Before you load the model, it is important to understand how RISA-3D manages loads. When loads are defined in RISA-3D, they are assigned to a "Basic Load Case," which is simply a set of loads. You can have up to 1000 separate Basic Load Cases (BLCs) defined in the **Basic Load Cases** spreadsheet. RISA-3D also provides an option to assign the BLCs to categories (DL, LL, etc.), which allows you to easily combine them later in the load combinations. When you are ready to solve the model, the BLCs (or BLC categories) are combined with multipliers to create load combinations in the **Load Combinations** spreadsheet.

In order to keep your loads organized, first define your Basic Load Cases. Then, when you apply your loads, specify which Load Case each will be assigned to. Although a description is not required by RISA-3D, below you will give a description for the first five BLCs.

- On the **Spreadsheets** menu, click **Basic Load Cases** to open the spreadsheet.
- Under **BLC Description**, in row **1**, type **Roof Load**. Press ENTER.
  In row **2**, type **Wind Load**. Press ENTER.
  In row **3**, type **Live Load**. Press ENTER.
  In row **4**, type **Dead Load**. Press ENTER.
  In row **5**, type **Cladding**. Press ENTER.

Your spreadsheet should now look like this:



Next, you will apply the loads and then come back to this spreadsheet to discuss it further.

- Click **Close** to exit the spreadsheet.

Load Direction Options

RISA-3D allows you to specify the direction of your loads in different ways. The three basic choices are:

- Apply loads aligned with global axes, designated with upper case labels such as **X**, **Y,** and **Z**.
- Apply loads aligned with member or plate local axes, designated with lower case labels such as **x**, **y,** and **z**.
- Apply projected loads along global axes, designated with upper case labels such as **PX**, **PY,** and **PZ**.

There is no need to worry about remembering the options, as you will be presented with this list each time you define the loads in RISA-3D. Also, the Help menu will provide explanations as needed.

Area Loads

The roof load and the wind load will be applied as area loads. This means you will specify a load over an area and RISA-3D will attribute that load to the appropriate members. RISA -3D gives the option of one- or two-way load attribution for area loads. One-way loads are attributed to the closest member in a certain direction; and two-way loads are attributed to the closest member, regardless of direction.

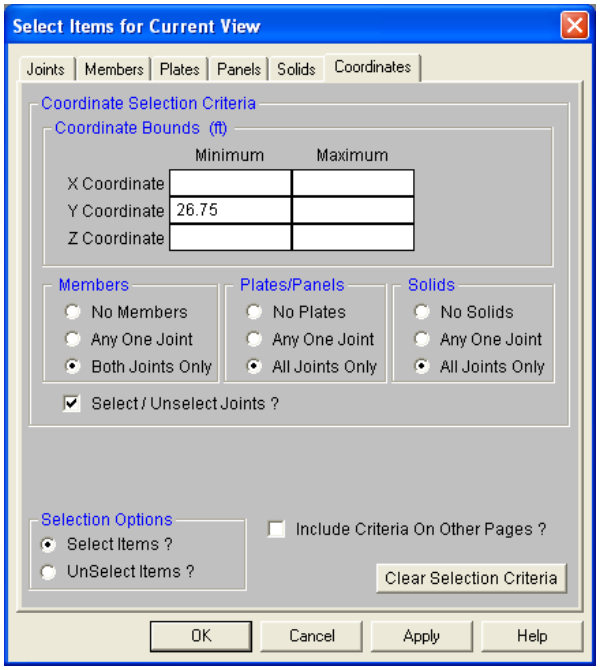
For this tutorial, the roof load will be a 20 psf vertical load attributed to the truss top chords with a one-way area load. The wind load will be a 40 psf lateral load attributed to the members on the side of the structure with a two-way area load.

To apply the load, start by selecting just the roof plane:

- On the Selection toolbar, click **Unselect All** to unselect the entire model. Then, click the **Criteria Selection** button. A dialog box will appear. Click the **Coordinates** tab.
- Click the **Clear Selection Criteria** button (at the bottom). Then, under **Coordinate Bounds**, in the **Y Coordinate** row, in the **Minimum** column, type **26.75**.

Note: The **Maximum** value can be left blank, since there is nothing above that elevation.

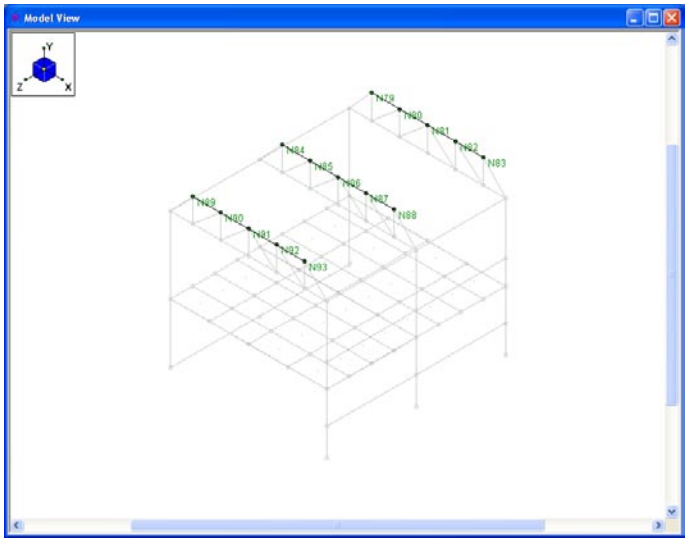
The dialog box should look like this:



Close the dialog box:

- Click **OK.**

Items at the top chord elevation are now selected, as shown below:



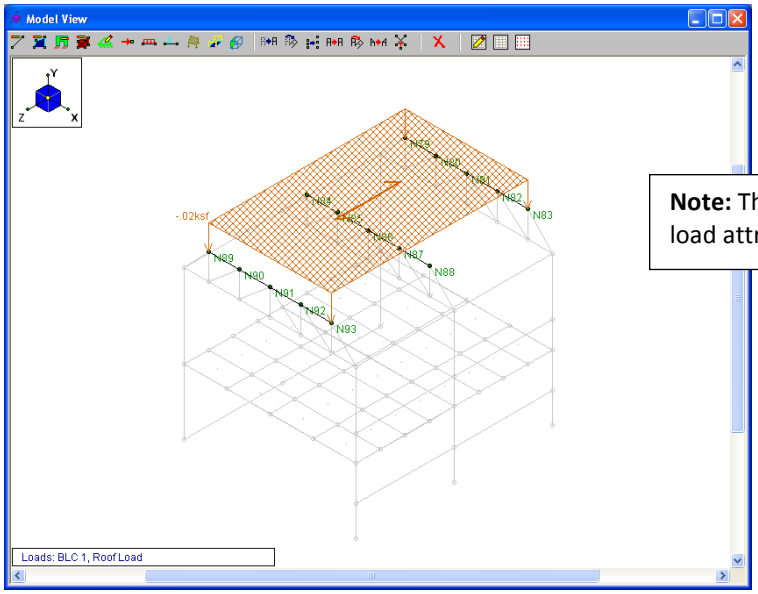
Next, draw the roof load area:

- On the **Insert** menu, select **Area (Member) Load.** In the **Magnitude** box, type **-0.02**. In the **Distribution** list, select **A-B** to specify the direction. The **Basic Load Case** list should already be set to **Roof Load.**
- Click **Apply** to begin drawing the roof load area.

Because you selected **A-B** for the **Distribution** direction, the first two clicks (**A** to **B**) define the direction of the load attribution. You can verify this graphically by checking the direction of the arrow on the load graphic.

- First, click the leftmost node, **N89** (coordinates **3.75,26.75,30**), and then proceed clockwise clicking on the corner nodes **N79** (coordinates **3.75,26.75,0**), **N83** (coordinates **22.5,26.75,0**), and finally **N93** (coordinates **22.5,26.75,30**) to define the area.

Now, your model should look like this:



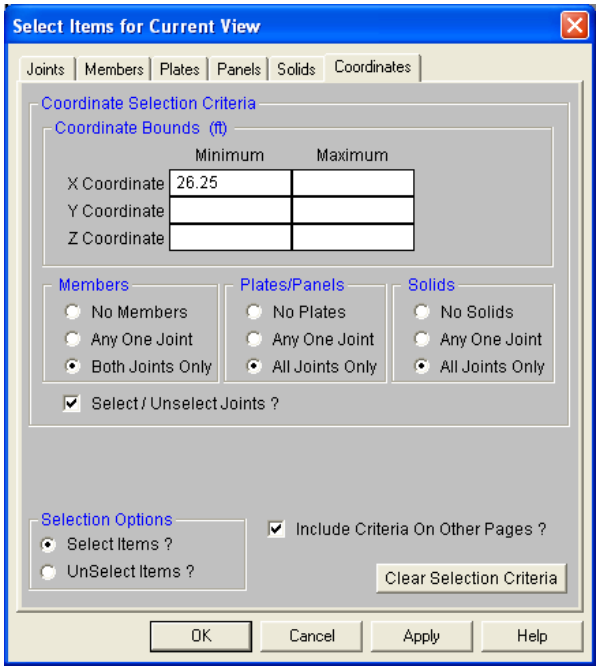
Note: The arrow denotes the load attribution direction.

Next, you will select the wind load plane.

- On the Selection toolbar, click **Unselect All** to unselect the entire model. Then, click the **Criteria Selection** button. Make sure the **Coordinates** tab is still selected.
- Again, click the **Clear Selection Criteria** button (at the bottom). Then, under **Coordinate Bounds**, in the **X Coordinate** row, in the **Minimum** column, type **26.25**.

Note: The **Maximum** value can again be left blank, since there is nothing above that elevation.

The dialog box should look like this:



Close the dialog box.

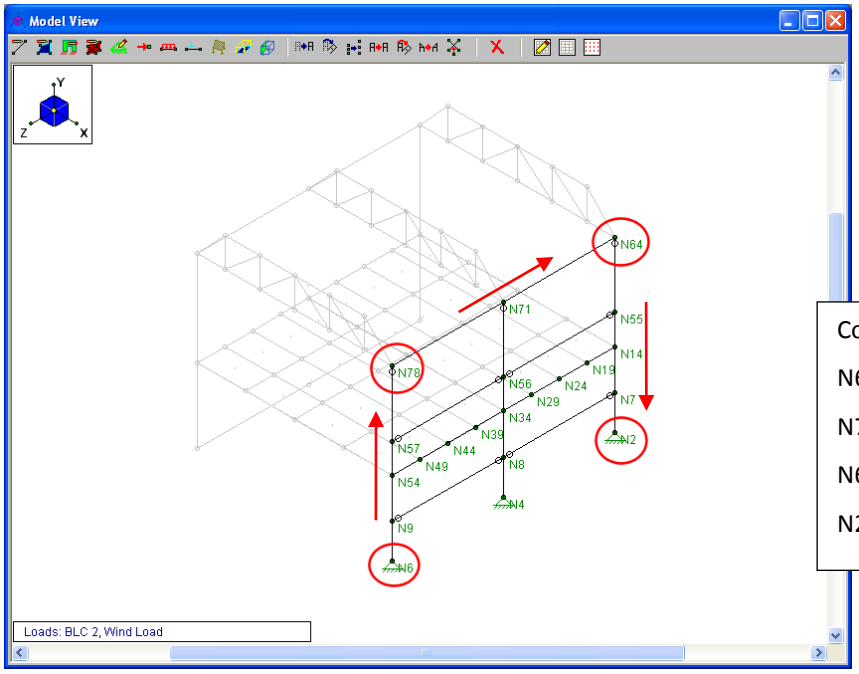
- Click **OK.**

The front face elements are now selected. Next, change the load direction:

- Press Ctrl+D to recall the **Area Loads for Members** dialog box.
- In the **Load Direction** list, select **X**. In the **Magnitude** box, type **-0.04**. In the **Distribution Direction** list, select **Two Way**. In the **Basic Load Case** list, select **2: Wind Load**. Click **Apply** to draw wind load area.

Return to your model to define the area:

- First, click the bottom-most node, **N6,** and then proceed clockwise clicking on the corner nodes **N78**, **N64,** and finally **N2,** as shown below.



Coordinate Locations:

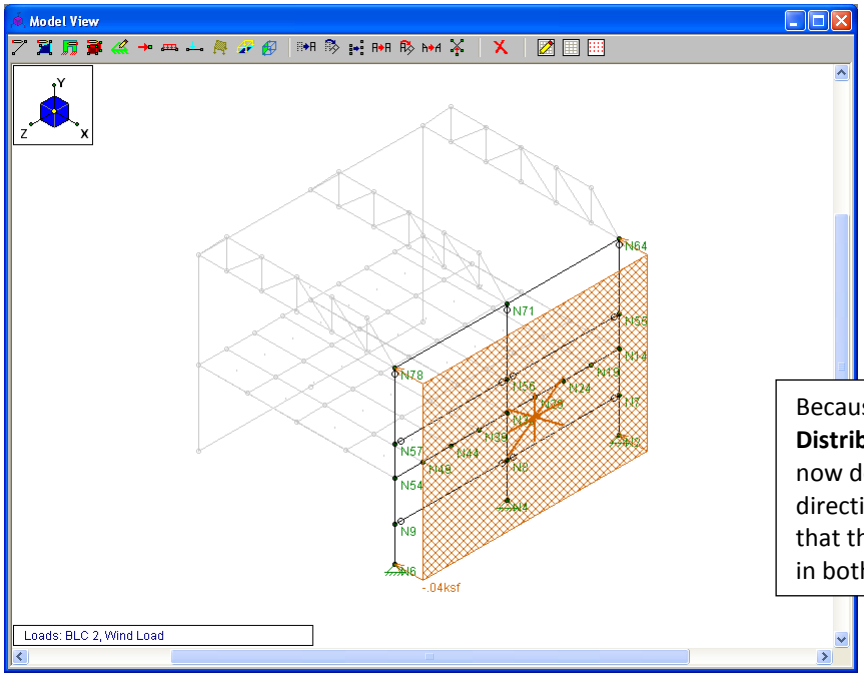
N6: (26.25,0,30)

N78: (26.25,22.75,30)

N64: (26.25,22.75,0)

N2: (26.25,0,0)

Your screen should now show the wind load applied as shown below:



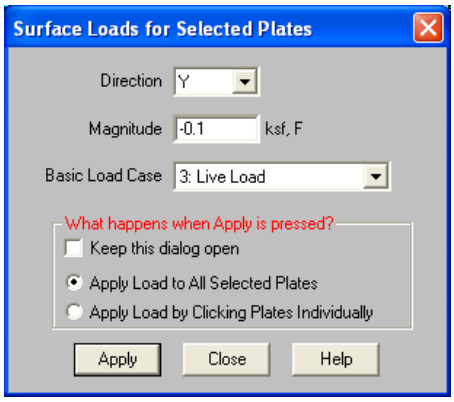
Because you selected a **Two Way Distribution** direction, the load is now denoted by a star (rather than a directional arrow). The star indicates that the area load will be attributed in both directions.

Surface Loads

Next, you will apply surface loads on the plates. You will apply a uniform 0.1 ksf (100 psf) load to all plates, as part of Basic Load Case 3. The surface load will be applied uniformly over the surface of the plate.

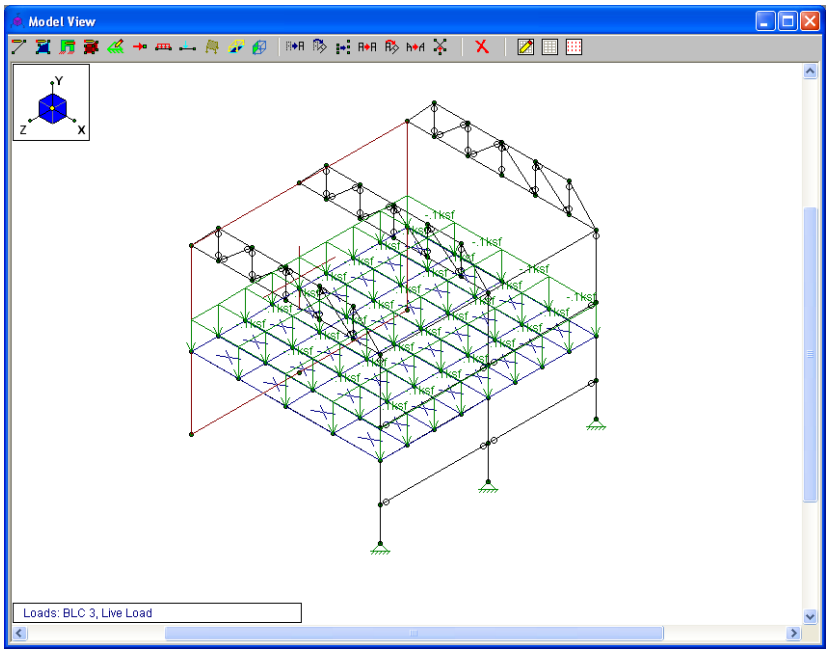
Select the entire model and remove the labels:

- Click **Select All** .
- On the Window toolbar, click **Joint Labels** . This will toggle off the display of the joint labels.
- On the Drawing toolbar, click **Surface Loads for Plates** .
- In the **Direction** list, select **Y** (be sure to select the upper case "Y" for the global Y direction). In the **Magnitude** box, type **-0.1**. In the **Basic Load Case** list, select **3: Live Load**. Finally, click **Apply Load to All Selected Plates**.



- Click **Apply**.

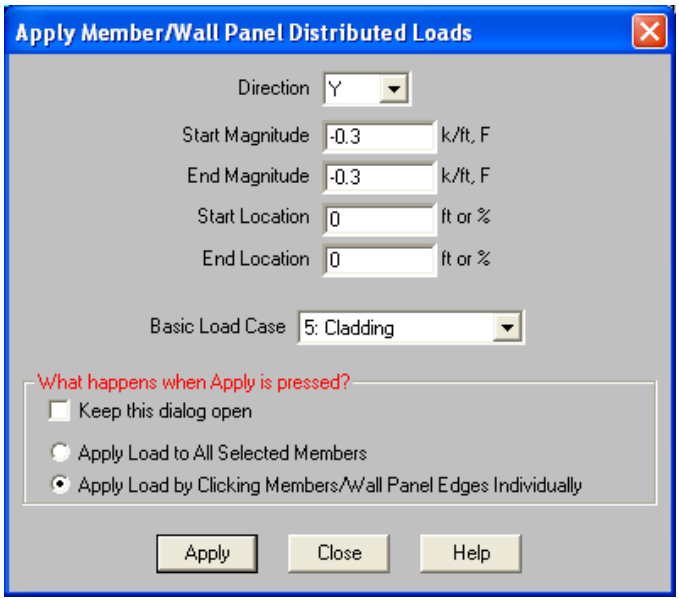
The loads will now be applied to all plates, as shown below.



Distributed Loads

The cladding on the structure will be applied as a 300 plf distributed vertical load on the Glulam beams and the Beam-Z members. Apply these loads by clicking on the members to demonstrate this method.

- On the Drawing toolbar, click **Distributed Loads**  . In the **Start Magnitude** box, type -**0.3**. Notice the same value is automatically entered as the **End Magnitude** value.
- In the **Basic Load Case** box, select **5: Cladding**.



Close the dialog box and prepare to select the members.

- Click **Apply** and you can now select the members.

At this point, you could simply click the four members to apply the load. However, for demonstration purposes, make a selection that designates the correct members.

- On the Selection toolbar, click **Unselect All**  . Then, click the **Criteria Selection** button  . Select the **Members** tab.
- Under **Member Properties**, in the **Section Set/Shape** list, choose **Beam-Z**.
- Click **Apply** and the members will be selected.

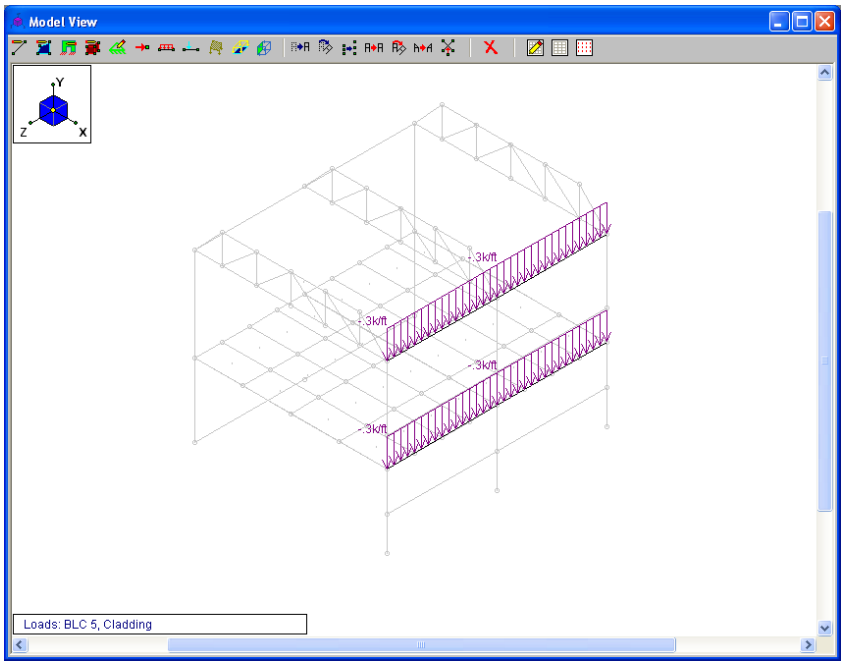
Repeat to select the Glulam beams:

- Under **Member Properties**, in the **Section Set/Shape** list, choose **Glulam**.
- Click **OK** and the members will be selected and the dialog window will close.

Now, click these four members to load them with the distributed cladding load:

- Click each of the four members one at a time. The loading will be applied.
- Press ESC to stop drawing.

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

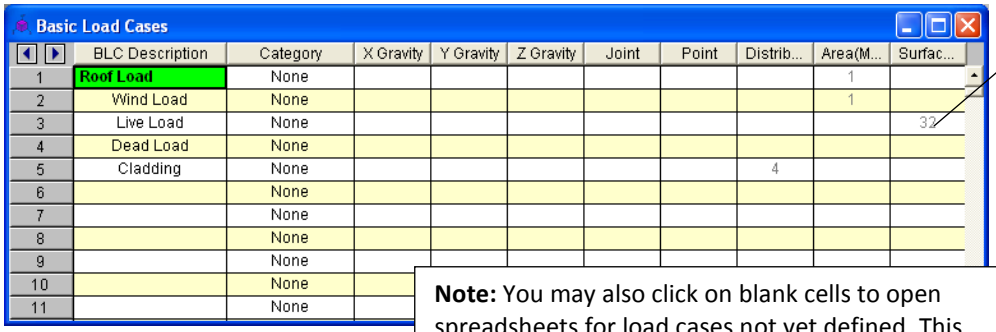


Basic Load Case Spreadsheet

To verify that the loads you have applied graphically are correct, review them in the spreadsheets.

- On the **Spreadsheets** menu, select **Basic Load Cases** to open the **Basic Load Cases** spreadsheet.

The **Basic Load Cases** spreadsheet is where you named your basic load cases earlier. Notice that there are now some numbers on the right side of the spreadsheet. These numbers summarize the loads you just defined: the first two lines contain a "1" which represents the one area load that you placed in each case; the "32" indicates you have 32 plates loaded with surface loads; and the "4" represents the 4 cladding loads you just applied. They are greyed, indicating they cannot be edited in this spreadsheet. But, clicking on any of the entries will cause the appropriate spreadsheet to open, where any editing can be performed.



The greyed out numbers cannot be edited from this spreadsheet. However, you may click on the entry to open the appropriate spreadsheet where the editing can be performed.

Note: You may also click on blank cells to open spreadsheets for load cases not yet defined. This "linking" provides an easy way for you to get an overall view of the loads with the **Basic Load Cases** spreadsheet and then quickly open the specific load spreadsheets for further investigation.

Open the distributed loads with the loads that were assigned to **BLC 5**.

- Click the number **4** (in the **Distributed** column of the **Cladding** case).

The **Member Distributed Loads** spreadsheet will open. Notice the member list (at the top of the spreadsheet) now displays **BLC 5: Cladding**. You can also use this list to view distributed loads that belong to another BLC.

- Click **Close** to close the **Distributed Loads** spreadsheet and return to the **Basic Load Cases** spreadsheet.

Copying Loads

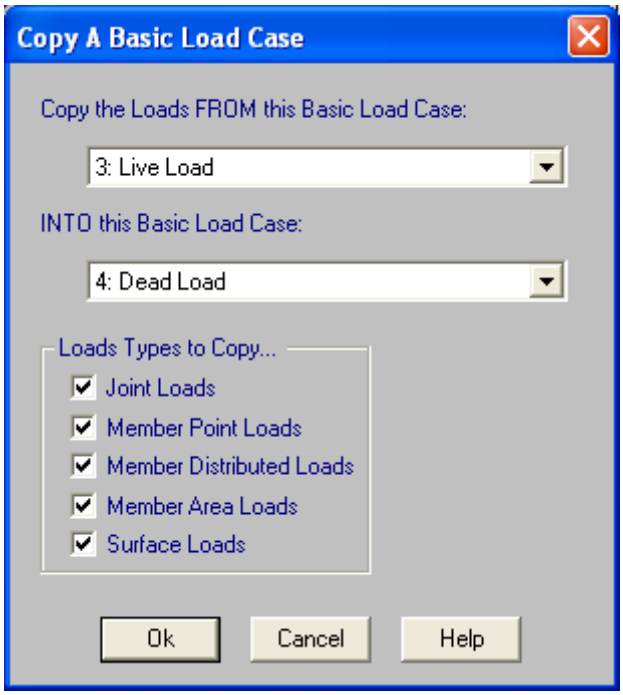
After taking the time to define a complex load case, you may realize that another load case is very similar to it. To save time when defining the second load case, you may find it useful to copy the first load case and modify the copy rather than start from scratch. For this purpose, RISA-3D provides the Copy BLC tool that allows you to copy all or part of a basic load case to another basic load case.

Use the Copy BLC tool to create a 90 psf dead load from the 100 psf live load that you just created.

- On the Window toolbar, click the **Copy BLC** button . The **Copy Basic Load Case** dialog box will appear.
- Under **Copy the Loads FROM this Basic Load Case**, select **3: Live Load**.

- Under **INTO this Basic Load Case**, select **4: Dead Load**.

The dialog box should look like this:

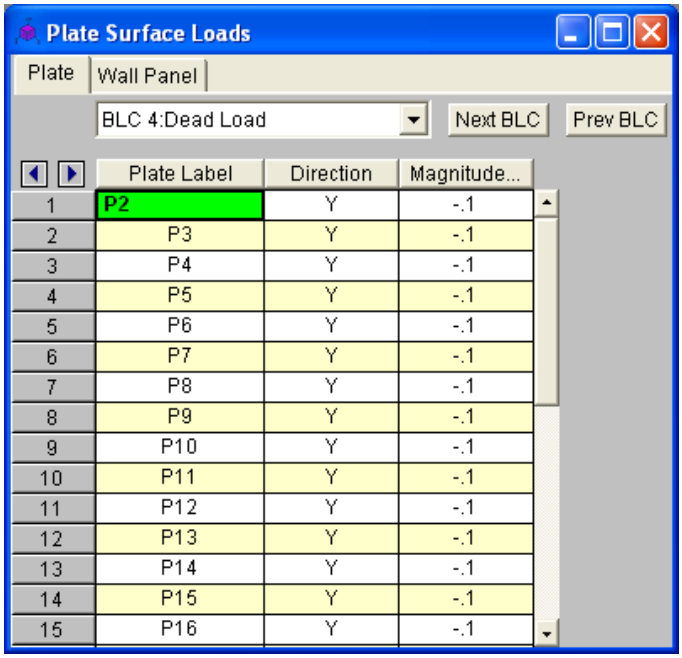


Close the dialog box:

- Click **OK.**

There is now a "**32**" in the **Surface** column of the **Dead Load** case.

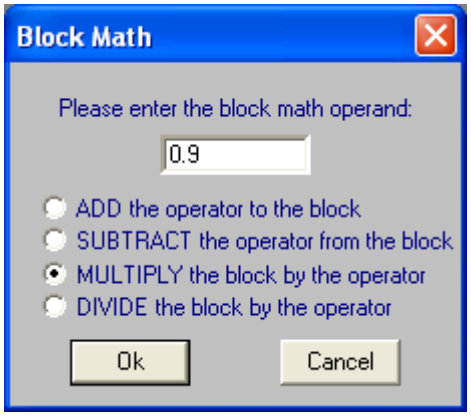
- Click the "**32**" to open the **Surface Loads** spreadsheet, where **BLC 4: Dead Load** will be displayed in the load list.



Use the Block Math feature to change this load from 100 psf to 90 psf, or 90% of the copied live load.

- First, select all of the cells in the **Magnitude** column. A fast way to select the entire column is to click on the **Magnitude** column heading. The entire column is now selected and is colored magenta.
- On the Window toolbar, click **Block Math**  .
- Under **Please enter the block math operand**, type **0.9**.
- Click **MULTIPLY the block by the operator**.

The dialog box should look like this:



- Click **Ok**. The surface loads now have a magnitude of -**0.09** ksf (90 psf downward).
- Press Ctrl+F4 to close the **Plate Surface Loads** spreadsheet.

Self-weight

The last step in defining your loads is to define the self weight of the structure. You will add it to the same basic load case where you just placed the surface dead loads. To accomplish this, simply enter a factor in the **Y Gravity** column on the Dead Load basic load case.

- Click in the **Y Gravity** column of the **Dead Load** case (row 4) and type **-1**.

RISA-3D will now calculate and include the self-weight of the model during solution. The factor entered is a multiplier for the self-weight. In this case, "-1" is used because you want the full self-weight applied downward (the negative Y-direction). You may apply self-weight multipliers in the X, Y, and/or Z directions.

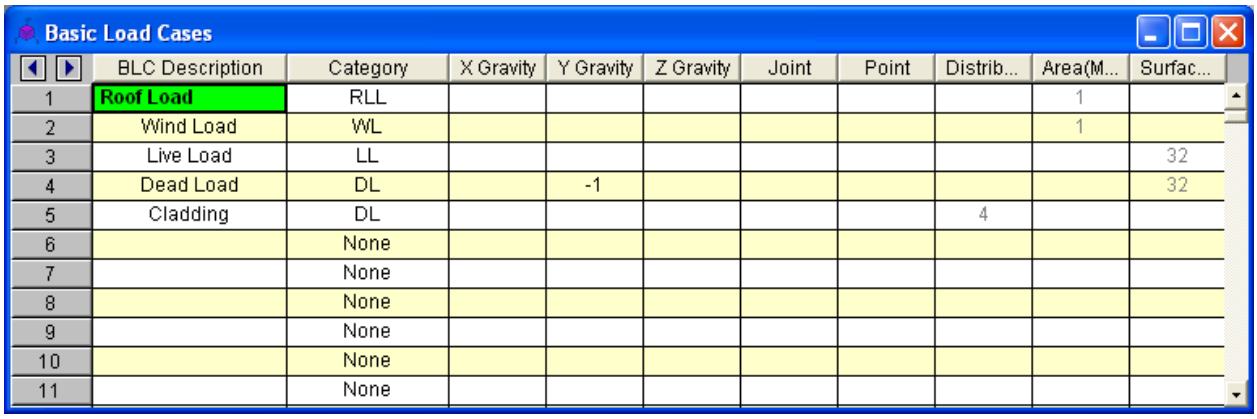
Load Categories

The final thing to do on the **Basic Load Cases** spreadsheet is to assign each load case to a category. Categories make it easier to later combine them for Load Combinations. In this model, the loads are quite manageable and you could just refer to them by their BLC number. Many structures, however, will be modeled with many separate load cases that are part of the same family.

Assign the associated categories to each of our Basic Load Cases:

- Click in the **Category** column of the **Roof Load** case. Select **RLL (Roof Live Load)**.
- Click in the **Category** column of the **Wind Load** case. Select **WL (Wind Load)**.
- Click in the **Category** column of the **Live Load** case. Select **LL (Live Load).**
- Click in the **Category** column of the **Dead Load** case. Select **DL (Dead Load)**.
- Click in the **Category** column of the **Cladding** case. Select **DL (Dead Load)**.

The spreadsheet is now complete and looks like this:



	BLC Description	Category	X Gravity	Y Gravity	Z Gravity	Joint	Point	Distrib...	Area(M...	Surfac...
1	Roof Load	RLL							1	
2	Wind Load	WL							1	
3	Live Load	LL								32
4	Dead Load	DL		-1						32
5	Cladding	DL						4		
6		None								
7		None								
8		None								
9		None								
10		None								
11		None								

Close the spreadsheet:

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

Load Combinations

You will now define how the loads are to be combined. The Load Combinations combine the loads applied to the Basic Load Cases, along with multiplying factors to apply to the model for solution and results. Most standard code combinations are already built into RISA-3D. Apply the ASCE minimum design loads and then modify them.

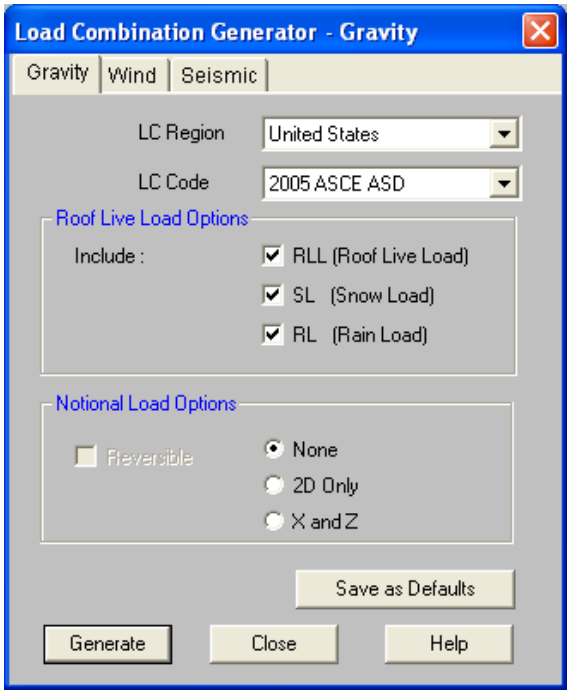
- On the **Spreadsheets** menu choose **Load Combinations**.

Currently, there is no data available for the spreadsheet. Generate the load combinations using the LC Generator:

- On the Window toolbar, click the **Load Combination Generator** button LC Generator.

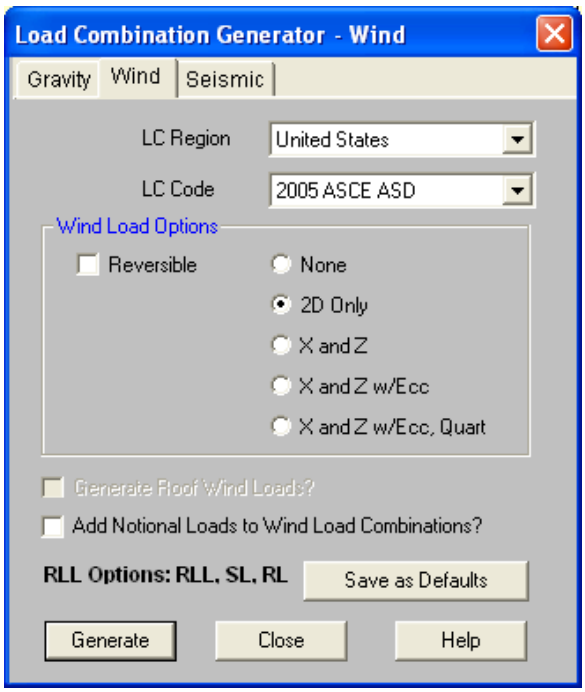
Generate the Load Combinations:

- Click on the **Gravity** tab. In the **Region** list, select **United States**.
- In the **Code** list, select **2005 ASCE ASD**.
- Under **Roof Live Load Options**, select **RLL (Roof Live Load)**, **SL (Snow Load)**, and **RL (Rain Load)**.

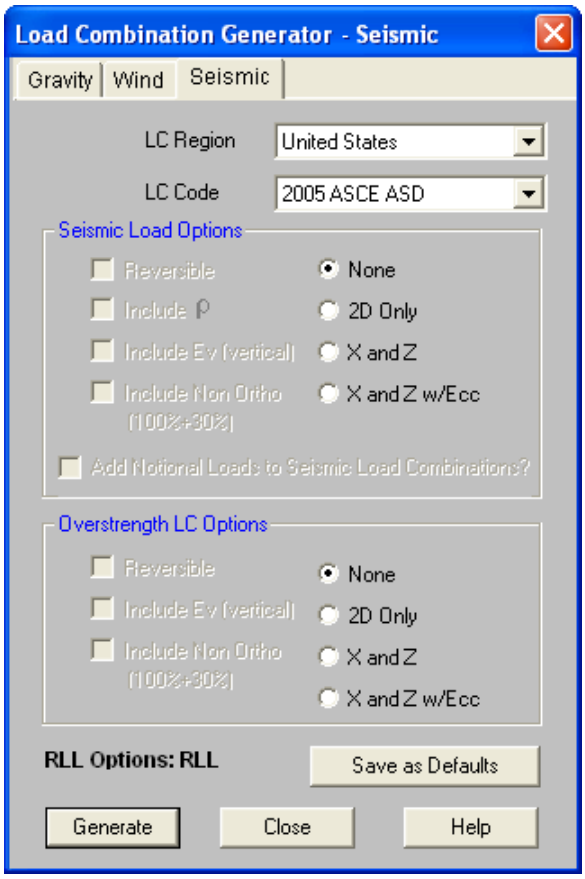


- Click **Generate**. Next, click on the **Wind** tab.

- Under **Wind Load Options**, click **2D Only**.



- Click **Generate**.
- Lastly, click on the **Seismic** tab.

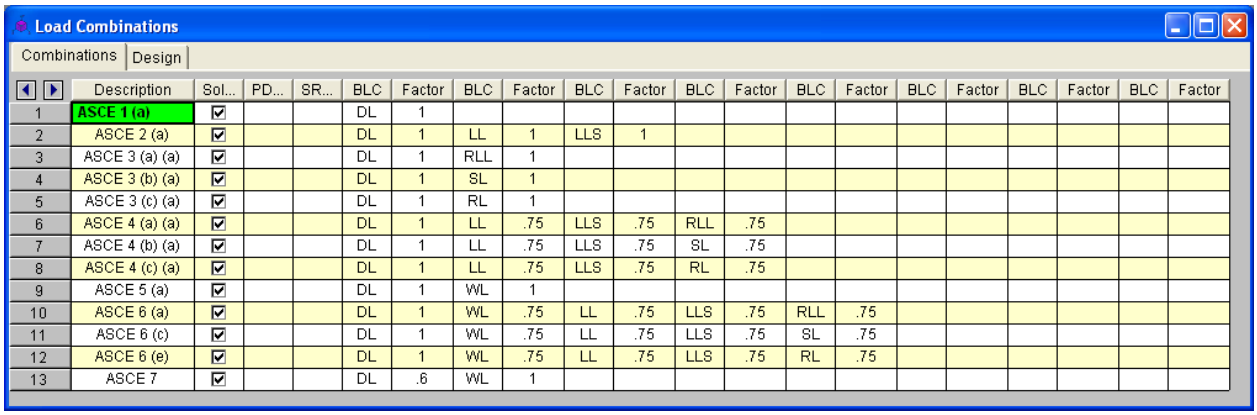


- Because you did not apply any earthquake loads to your structure, it is appropriate to leave both the **Seismic Load Options** and the **Overstrength LC Options** set to **None**.

For more information on the Wind Load Options and Seismic Load Options refer to the Generating Building Code Combinations section of *RISA-3D General Reference*.

- Finally, click **Close**.

The following **Load Combinations** spreadsheet will be generated.



Load Combinations

Combinations | Design

	Description	Sol...	PD...	SR...	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor
1	ASCE 1 (a)	☑			DL	1														
2	ASCE 2 (a)	☑			DL	1	LL	1	LLS	1										
3	ASCE 3 (a) (a)	☑			DL	1	RLL	1												
4	ASCE 3 (b) (a)	☑			DL	1	SL	1												
5	ASCE 3 (c) (a)	☑			DL	1	RL	1												
6	ASCE 4 (a) (a)	☑			DL	1	LL	.75	LLS	.75	RLL	.75								
7	ASCE 4 (b) (a)	☑			DL	1	LL	.75	LLS	.75	SL	.75								
8	ASCE 4 (c) (a)	☑			DL	1	LL	.75	LLS	.75	RL	.75								
9	ASCE 5 (a)	☑			DL	1	WL	1												
10	ASCE 6 (a)	☑			DL	1	WL	.75	LL	.75	LLS	.75	RLL	.75						
11	ASCE 6 (c)	☑			DL	1	WL	.75	LL	.75	LLS	.75	SL	.75						
12	ASCE 6 (e)	☑			DL	1	WL	.75	LL	.75	LLS	.75	RL	.75						
13	ASCE 7	☑			DL	.6	WL	1												

The generated combinations may be edited in the spreadsheet. Each field is described below:

Description	A descriptive label of your choice. Enter any descriptive label, and it will be displayed with the results when the load combination is solved.
Solve	This check box indicates whether this combination is to be included in the Batch or Envelope solution (that will be performed later). All of your combinations are currently selected, indicating that they are to be included.
PDelta	The **PDelta** flag indicates what type of P-Delta analysis you wish to run. P-Delta calculations account for the secondary effects resulting from load eccentricities due to model deflections.
SRSS	The **SRSS** field is used to combine response spectra analysis results for different directions by taking the square root of the sum of the squares.
(the remaining fields)	The remaining fields define the actual combinations, with pairs of **BLC** columns and **Factor** columns. The values are exactly as listed in the code.

Note: Instead of using categories, as shown, you may also call out loads by the BLC number. For example, instead of using LL, you could reference the same load by typing the BLC number (**3**). Other options include nesting one combination within another and including response spectra results. For more information, refer to Help or the *RISA-3D General Reference*.

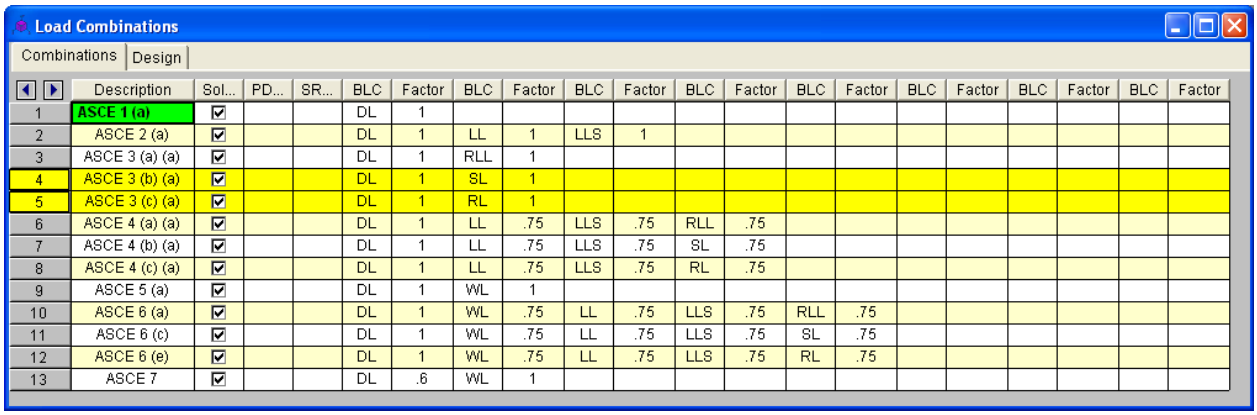
Before moving on, make the following changes to the spreadsheet. Assuming you do not have a snow load, or rain load, remove the unnecessary combinations from the list by deleting those lines from the spreadsheet.

First, select the rows to be deleted:

- Select rows **4** and **5** (load combinations **ASCE 3(b)(a)** and **ASCE 3(c)(a)**) by clicking directly on row **4** (the row label), then drag down to row **5**. Release the mouse.

These lines should now be shaded yellow, as shown below.

Note: If the cells are highlighted in magenta, you selected cells from within the spreadsheet, not the row label.



	Description	Sol...	PD...	SR...	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor
1	ASCE 1 (a)	☑			DL	1														
2	ASCE 2 (a)	☑			DL	1	LL	1	LLS	1										
3	ASCE 3 (a) (a)	☑			DL	1	RLL	1												
4	ASCE 3 (b) (a)	☑			DL	1	SL	1												
5	ASCE 3 (c) (a)	☑			DL	1	RL	1												
6	ASCE 4 (a) (a)	☑			DL	1	LL	.75	LLS	.75	RLL	.75								
7	ASCE 4 (b) (a)	☑			DL	1	LL	.75	LLS	.75	SL	.75								
8	ASCE 4 (c) (a)	☑			DL	1	LL	.75	LLS	.75	RL	.75								
9	ASCE 5 (a)	☑			DL	1	WL	1												
10	ASCE 6 (a)	☑			DL	1	WL	.75	LL	.75	LLS	.75	RLL	.75						
11	ASCE 6 (c)	☑			DL	1	WL	.75	LL	.75	LLS	.75	SL	.75						
12	ASCE 6 (e)	☑			DL	1	WL	.75	LL	.75	LLS	.75	RL	.75						
13	ASCE 7	☑			DL	.6	WL	1												

Now, delete rows **4** and **5**:

- On the Window toolbar, click **Delete Lines** .

Note: After deleting rows, the row numbers will renumber.

The rows are now deleted, leaving you with the remaining combinations. Next, delete combinations **5 (ASCE 4 (b)(a))** and **6 (ASCE 4 (c)(a))**, this time using a different method:

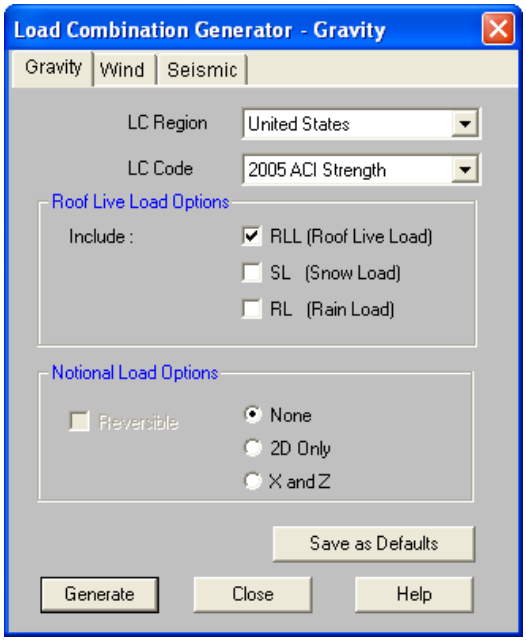
- Under the **Description** column, place your cursor in the **ASCE 4 (b)(a)** cell (the fifth row down). Press the F4 key two times to delete this load combination as well as **ASCE 4 (c)(a).**
- Repeat to delete combinations **7 (ASCE 6 (C))** and **8 (ASCE 6(e)).**

The remaining load combinations are sufficient for your steel and wood members. However, you will need to add more load combinations for the design of your concrete members.

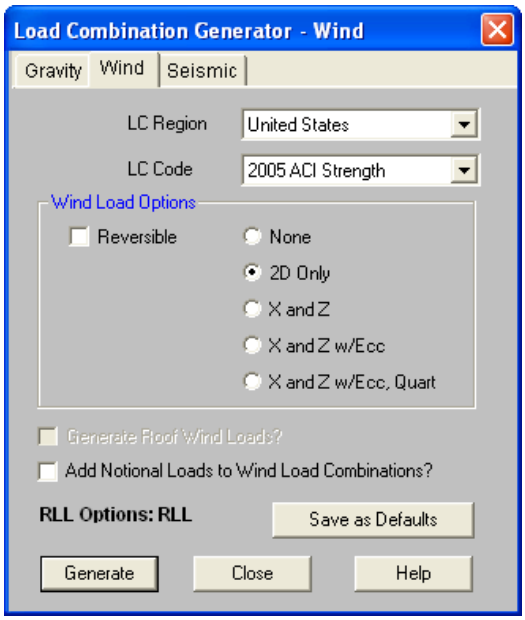
- Click the **Load Combination Generator** button again.

Generate the Load Combinations:

- Click on the **Gravity** tab.
- In the **Region** list, select **United States**.
- In the **Code** list, select **2005 ACI Strength**.
- Uncheck the **SL (Snow Load)** and **RL (Rain Load)** checkboxes so you don't generate unnecessary Load Combinations this time.



- Click **Generate**. Next, click on the **Wind** tab.
- Under **Wind Load Options**, click **2D Only**.



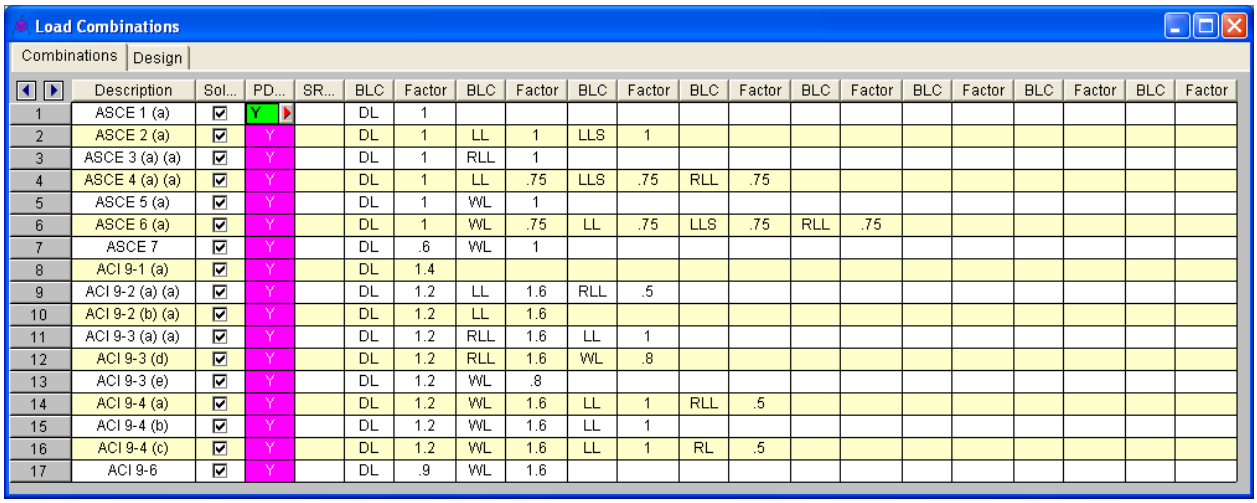
- Click **Generate** and then click the **Close** button.

The **Load Combinations** spreadsheet will now include your ACI load combinations.

With the requirements of the *AISC 13th Edition Steel Code* and the *ACI 318-08 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.



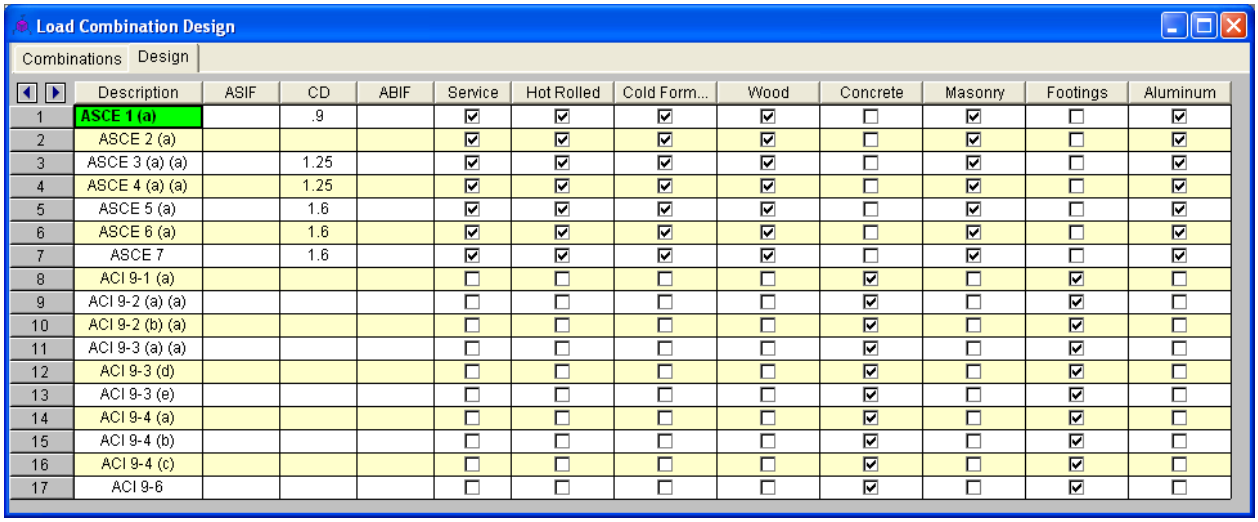
Load Combinations

Combinations | Design

	Description	Sol...	PD...	SR...	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor
1	ASCE 1 (a)	☑	Y		DL	1														
2	ASCE 2 (a)	☑	Y		DL	1	LL	1	LLS	1										
3	ASCE 3 (a) (a)	☑	Y		DL	1	RLL	1												
4	ASCE 4 (a) (a)	☑	Y		DL	1	LL	.75	LLS	.75	RLL	.75								
5	ASCE 5 (a)	☑	Y		DL	1	WL	1												
6	ASCE 6 (a)	☑	Y		DL	1	WL	.75	LL	.75	LLS	.75	RLL	.75						
7	ASCE 7	☑	Y		DL	.6	WL	1												
8	ACI 9-1 (a)	☑	Y		DL	1.4														
9	ACI 9-2 (a) (a)	☑	Y		DL	1.2	LL	1.6	RLL	.5										
10	ACI 9-2 (b) (a)	☑	Y		DL	1.2	LL	1.6												
11	ACI 9-3 (a) (a)	☑	Y		DL	1.2	RLL	1.6	LL	1										
12	ACI 9-3 (d)	☑	Y		DL	1.2	RLL	1.6	WL	.8										
13	ACI 9-3 (e)	☑	Y		DL	1.2	WL	.8												
14	ACI 9-4 (a)	☑	Y		DL	1.2	WL	1.6	LL	1	RLL	.5								
15	ACI 9-4 (b)	☑	Y		DL	1.2	WL	1.6	LL	1										
16	ACI 9-4 (c)	☑	Y		DL	1.2	WL	1.6	LL	1	RL	.5								
17	ACI 9-6	☑	Y		DL	.9	WL	1.6												

View more load combination data:

- On the **Load Combinations** spreadsheet, click the **Design** tab to view more load combination data.



Load Combination Design

Combinations | Design

	Description	ASIF	CD	ABIF	Service	Hot Rolled	Cold Form...	Wood	Concrete	Masonry	Footings	Aluminum
1	ASCE 1 (a)		.9		☑	☑	☑	☑	☐	☑	☐	☑
2	ASCE 2 (a)				☑	☑	☑	☑	☐	☑	☐	☑
3	ASCE 3 (a) (a)		1.25		☑	☑	☑	☑	☐	☑	☐	☑
4	ASCE 4 (a) (a)		1.25		☑	☑	☑	☑	☐	☑	☐	☑
5	ASCE 5 (a)		1.6		☑	☑	☑	☑	☐	☑	☐	☑
6	ASCE 6 (a)		1.6		☑	☑	☑	☑	☐	☑	☐	☑
7	ASCE 7		1.6		☑	☑	☑	☑	☐	☑	☐	☑
8	ACI 9-1 (a)				☐	☐	☐	☐	☑	☐	☑	☐
9	ACI 9-2 (a) (a)				☐	☐	☐	☐	☑	☐	☑	☐
10	ACI 9-2 (b) (a)				☐	☐	☐	☐	☑	☐	☑	☐
11	ACI 9-3 (a) (a)				☐	☐	☐	☐	☑	☐	☑	☐
12	ACI 9-3 (d)				☐	☐	☐	☐	☑	☐	☑	☐
13	ACI 9-3 (e)				☐	☐	☐	☐	☑	☐	☑	☐
14	ACI 9-4 (a)				☐	☐	☐	☐	☑	☐	☑	☐
15	ACI 9-4 (b)				☐	☐	☐	☐	☑	☐	☑	☐
16	ACI 9-4 (c)				☐	☐	☐	☐	☑	☐	☑	☐
17	ACI 9-6				☐	☐	☐	☐	☑	☐	☑	☐

The data from the **Design** tab is described below:

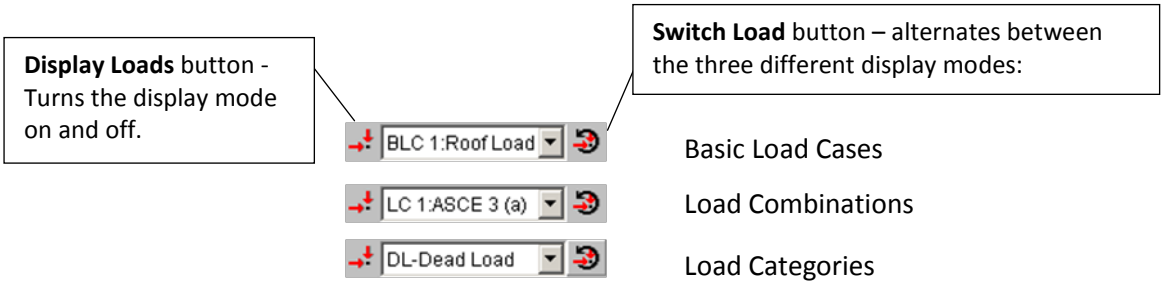
ASIF	Used to enter the allowable stress increase factor (ASIF) for the design of hot rolled steel per *AISC 9th Edition* specifications. Since you are using the ASCE load combinations, this increase factor (usually 1.333 for wind or seismic loads) will not be used. An entry in this field greater than 1.0 is also used to indicate whether the seismic provisions of the LRFD code are to be used.
CD	The load duration factor used only in timber design. This factor is dependent on the load with the shortest duration that is present in the load combination.
ABIF	Used to set the allowable bearing stress factor which will be used for the soil bearing check in Footing design. This factor is only available when you have installed a current copy of the RISAFoot program.
Service	This check box indicates whether the load combination is for service level loads. This will affect the stiffness used for concrete members during the solution.
The remaining fields	The remaining fields define which material types will be designed and/or checked for that load combination. Therefore, you will never get an NDS wood code check for the strength level load combinations used for concrete design.

Review, then close the spreadsheet.

- Review the information on your spreadsheet.
- Click **Close** to close the spreadsheet.

Displaying Loads

RISA-3D provides easy ways to view your loads using the loads display buttons described below. The loads can be displayed as Basic Load Cases, Load Categories, or Load Combinations.



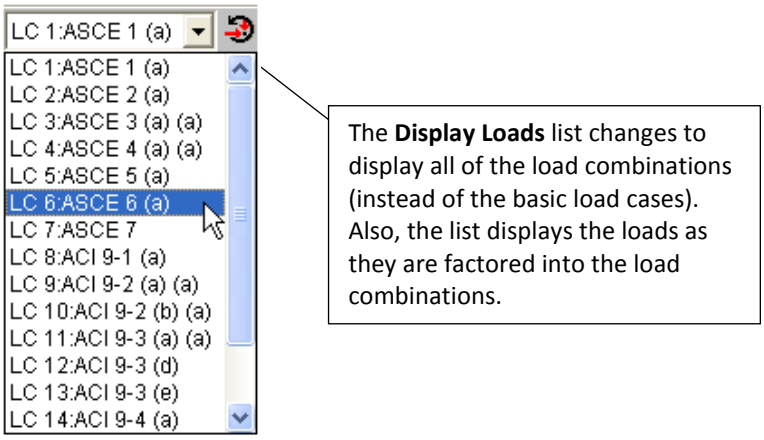
To view the basic load cases, first select the entire model:

- On the Selection toolbar, click **Select All** .
- On the Window toolbar, click the **Display Loads** list and select **BLC 2:Wind Load**. If your loads do not appear, click the **Display Loads** button to turn them on.

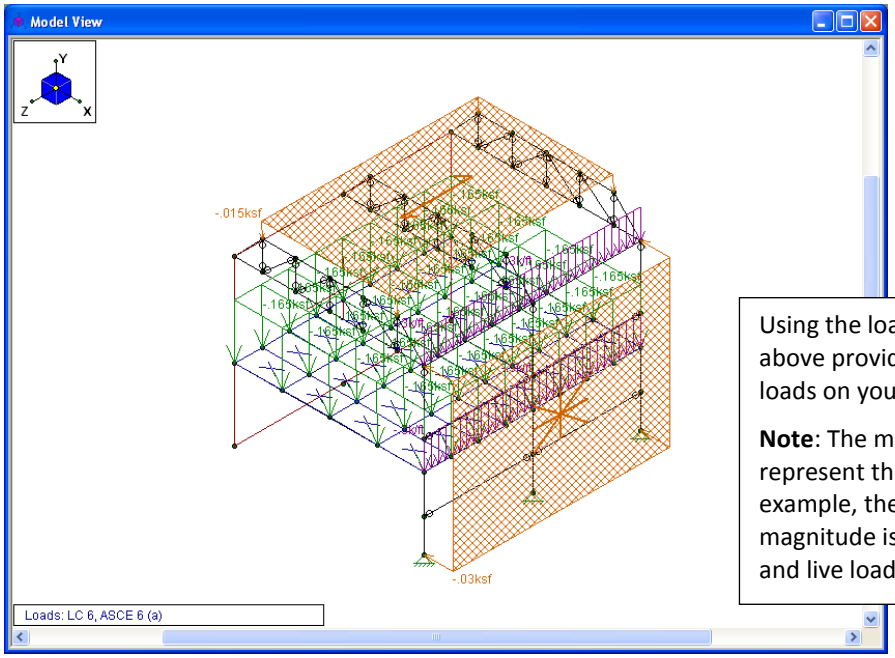
To view the load combinations:

- On the Window toolbar, click the **Switch Loads** button one time.

Select your load combination:



- Select **LC 6:ASCE 6 (a).**



Using the load display techniques described above provide a great way to verify the loads on your model.

Note: The magnitudes of the loads represent the factored combination. For example, the displayed floor load magnitude is the combination of the dead and live loads multiplied by their factors.

When you are finished viewing the load combinations, view the load categories:

- On the Window toolbar, click the **Switch Loads** button  one more time. The **Display Loads** list will again change, now displaying the Load Categories.
- View the loads by category by selecting them from the list.

This concludes Tutorial 3. The next tutorial demonstrates how to solve the model.

You may save your model to be used as the starting point for the next tutorial, or begin the next tutorial using the .r3d starter file in the RISA-3D **Tutorials** folder. To save the model:

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

Tutorial 4 – Solving & Results

In this tutorial, you will solve the model defined in the last tutorial and review the results. Make sure you start off correctly by opening the file mentioned next. This tutorial continues where the previous tutorial ended, so follow these steps to get your model up and running:

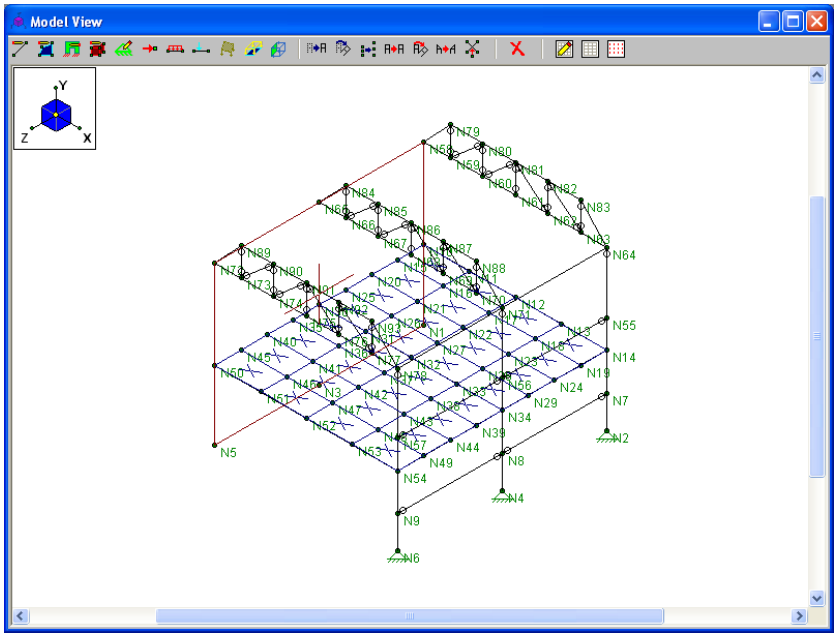
If you are continuing from the previous tutorial:

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

- Double-click the **RISA-3D** icon to start the program.
- Click **Open Model**.
  Double-click the **Tutorials** folder, select **Tutorial 4 Starter.r3d** and click **Open**.
  Click **Close** (or **Cancel**) to exit the **Global Parameters** dialog box.
- On the Window toolbar, click the **Graphic Editing Toolbar** button to activate the Drawing toolbar.
- On the **Data Entry** toolbar, click **Close** to close it.

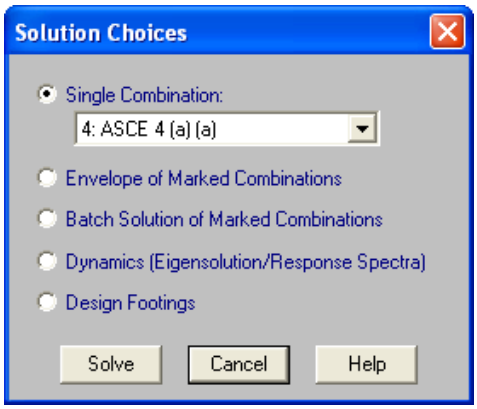
Your screen should look like this:



Solving the Model

In this tutorial, you will solve a few single load combinations, then solve all of the combinations and look at the enveloped results.

- On the RISA toolbar, click **Solve** .
- Click **Single Combination**. In the **Single Combination** list, select **4: ASCE 4(a)(a)**.



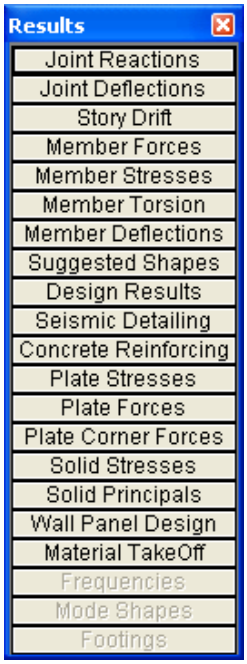
- Click **Solve**.

RISA-3D reports the solution steps as they occur. When the solution is complete, the **Results** toolbar is displayed to the right of the model view for access to other results.

You may view results in the spreadsheets, plotted on the model view, or review the details for each member, as described below.

Using the Results Toolbar

The results spreadsheets may be accessed from the **Results** menu or the **Results** toolbar. For now, use the **Results** toolbar to quickly review the results.



Later, you will click on spreadsheets to review the data. While in the spreadsheets, you may refer to the Status bar for an explanation of any active cell. Use the scroll bars to view information not displayed in the window.

The keyboard can be useful to move your cursor around in the spreadsheets. Use the PAGE UP and PAGE DOWN keys to move your cursor up or down a page at a time, or the UP ARROW and DOWN ARROW keys to move your cursor up or down one line at a time. The HOME and END keys move your cursor to the top (HOME) or bottom (END) of the spreadsheet.

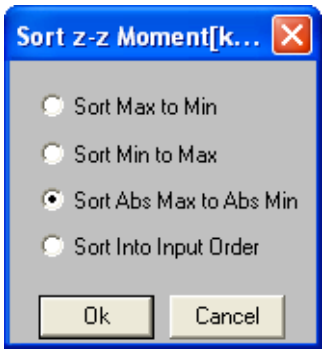
The Window toolbar also provides helpful tools that allow you to locate and manipulate the spreadsheet data. You can use the Find feature to locate specific results or the Sort feature to sort the results. The Exclude feature allows you to hide results that are not of interest.

Now that you have an idea of what results are available, in the next section you will see how easily and quickly you can manipulate them.

- On the **Results** toolbar, select **Joint Deflections** to review the results.
- Next, select **Member Forces** to view the **Member Section Forces** spreadsheet. (Note that seven locations are listed for each member. This is because you specified seven sections when defining the Global Parameters settings.)

A powerful feature of the spreadsheets is that the data can be sorted various ways. Next, you will use the Sort feature to sort the strong axis moment results.

- Under the **z-z Moment** column, click any cell.
- On the Window toolbar, click **Sort**  to access the sorting options.
- Click **Sort Abs Max to Abs Min**.



- Click **OK**.

The members are now sorted according to their absolute maximum z-axis bending moment value. Review the remaining spreadsheet data:

- On the **Results** toolbar, select **Member Stresses** to view the **Member Section Stresses** spreadsheet.
- Select **Member Torsion** to view the **Member Torsion Stresses** spreadsheet, including warping and shear stresses for the members that warp.
- Select **Member Deflections** to view the **Member Section Deflections** spreadsheet. This spreadsheet also shows the deflections as a ratio of member length (the L/y Ratio) providing an easy check against deflection criteria, such as L/360.
- Select **Suggested Shapes** to view the **Suggested Shapes** spreadsheet for redesign (you will use this spreadsheet later).

Review the Design Results data:

- Select **Design Results** and choose the **Hot Rolled Steel** tab**.** The data relating to the AISC code checks is presented here; i.e., ratio of actual to allowable stresses. (Any failing members would be denoted with red text.) The allowable stresses, Cb and Cm values, and controlling equation are listed. If you were performing LRFD-based design, the member strengths would also be listed here (along with the code check value).
- Select the **Cold Formed Steel** tab to view the cold formed steel code checks. The design strengths Pn, Mny, and Mnz are listed, along with the code check values and controlling equation. Factored design stresses and controlling equation are also listed.

- Select the **Wood** tab to view the wood code checks for our glulam members. The factored design stresses and controlling equation are also listed.
- Select the **Concrete Beams** tab. Notice that there are no results for this load combination. This is because when you defined this load combination, the check box for concrete design was not selected. Later, when you solve batch and envelope solutions you will see how this prevents you from getting unnecessary steel or wood code checks for load combinations that are specifically intended for concrete design.

Now, finish reviewing the **Results** toolbar data:

- On the **Results** toolbar, select **Plate Stresses** to view the **Plate Principal Stresses** spreadsheet.
- Select **Plate Forces** to view the **Plate Forces** spreadsheet.
- Select **Plate Corner Forces** to view the **Plate Corner Forces** spreadsheet.
- Select **Wall Panel Design** to view the **Wall Panel Design** spreadsheet.
- Select **Material TakeOff** to view the **Material Takeoff** spreadsheet.

Graphic Results

Since you have solved the model, you now have several more plotting options. But, you must first close all the spreadsheets, turn off the joint labels, and return to the model viewing options:

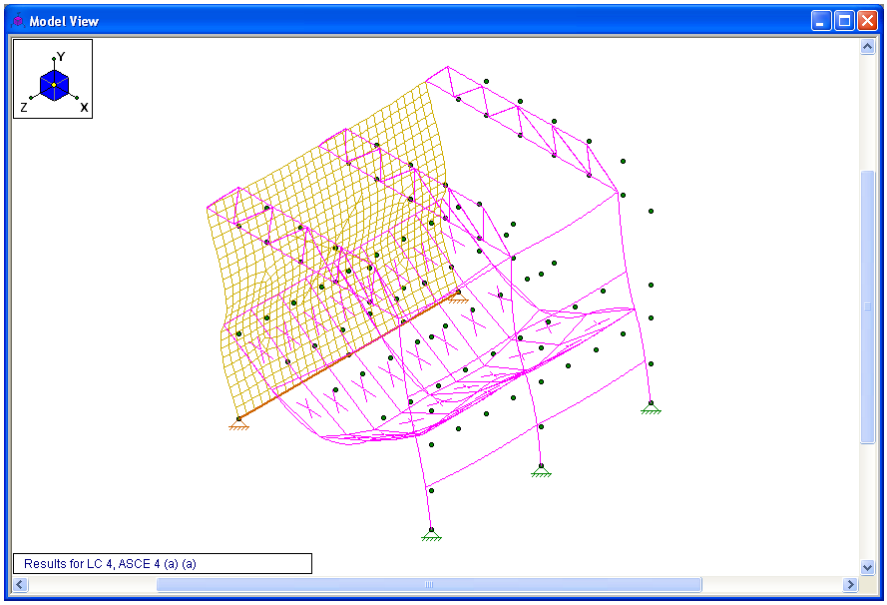
- On the **Window** menu, select **Single View**. All the open spreadsheets will be closed.
- Click **Joint Labels**  to turn off the joint labels.
- Finally, on the **Results** toolbar, click **Close**  to close the toolbar.

Plotting Deflected Shapes

Another important feature that makes quick review possible is the ability to view deflected shapes. You will utilize this feature next.

- On the Window toolbar, click **Plot Options**  to open the **Set Options for Current View** dialog box.
- Click the **Deflection Diagrams** tab.
- Under **Show Deflected Shape For**, click **Load Combination (pick from list at bottom)**. The combination you solved will now appear in the list at the bottom.
- Click **OK**.

Your screen should now look like this:



This is a simple representation of the model deflection under these loads. Next, you will use some of RISA-3D's more advanced graphics features.

- On the Window toolbar, click **Render**  two times.

Animation

Now, you can go a step further and animate this deflected shape plot.

- On the Window toolbar, click **Plot Options**  again. Make sure you are still on the **Deflection Diagrams** tab.
- Click the **Animate This Deflected Shape** button.

A progress bar indicates that the animation is rendering, the speed of which will depend on the speed of your computer. Once your new animation displays, you can close the dialog box.

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

Once the animation begins, you can change the speed using the **Faster** or **Slower** buttons. The deflected shape animation is a powerful tool that allows you to clearly see how the model is deflecting, and helps you identify parts of the structure that may have been incorrectly modeled or loaded.

When you are finished watching the animation, close the window:

- Click **Close** to close the animation window.

Color-Coding Member Results

You can also display members and highlight areas of concern. To demonstrate this, perform a plot of the model with each member color-coded by axial stress.

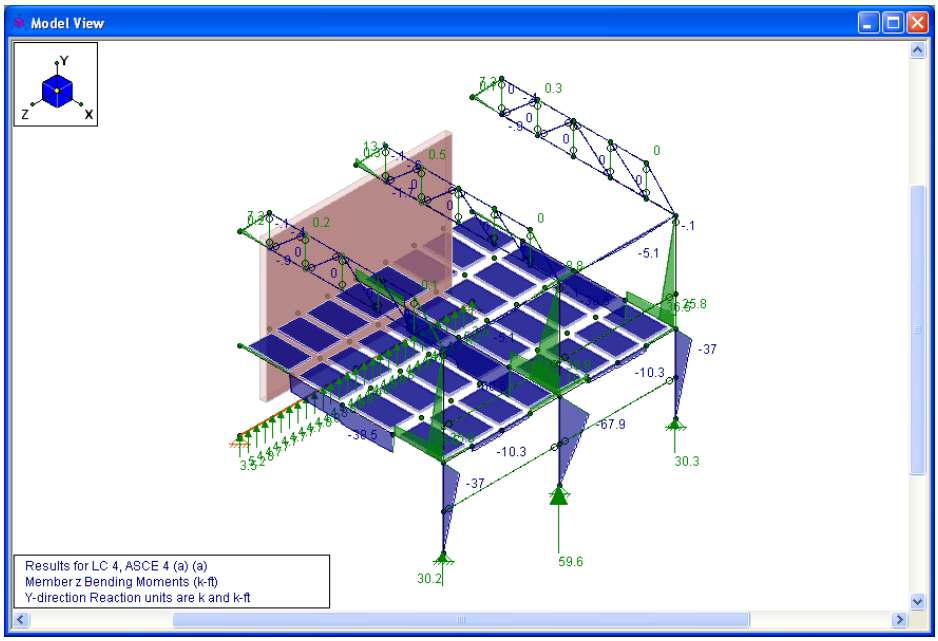
- On the Window toolbar, click **Plot Options** to return to the **Set Options for Current View** dialog box. You should still be on the **Deflection Diagrams** tab.
- Click **Don't Show Deflected Shape** to turn off the deflected shape.
- Next, click the **Members** tab.
- Under **Draw Members As**, select **Color Coded**. Under the **Color Basis** list, select **Compression**.
- Click **OK**.

The color legend appears in the upper right side of the model view window. With the color-coded members you can quickly identify members that require further attention.

Lastly, display moments and reactions in the model view.

- On the Window toolbar, click **Plot Options** again. You should still be in the **Members** tab.
- Under **Draw Members As**, select **Wireframe**.
- Under **Member Results**, in the **Diagram** list, select **z-z Moment**. Also, select the **Magnitudes** check box.
- Now, click the **Joints** tab.
- In the **Show the Reactions** section, select the **Y Direction** check box. Then, select the **Include the Magnitudes** check box.
- Click **OK**.

The moment diagrams and reactions are now plotted with magnitudes, as shown below:

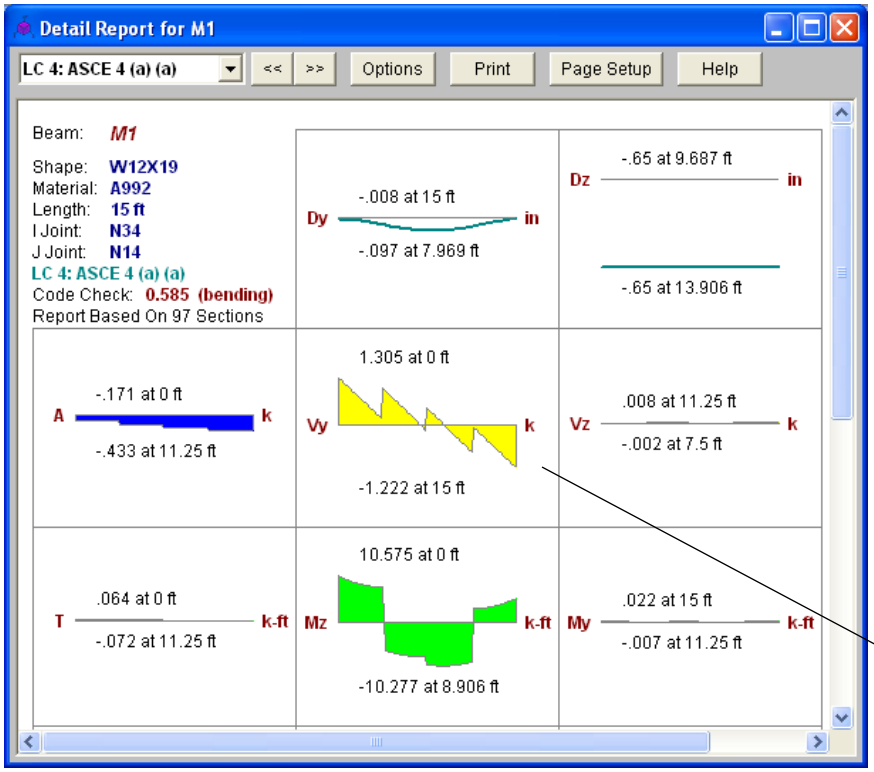


Member Detail Report

Next, use the member detail report to take a closer look at some members.

- From the **Results** menu, select **Members - Member Detail**. Click **OK** to display a report on member **M1** (which is the crossbeam on the middle right).

The report looks like this:

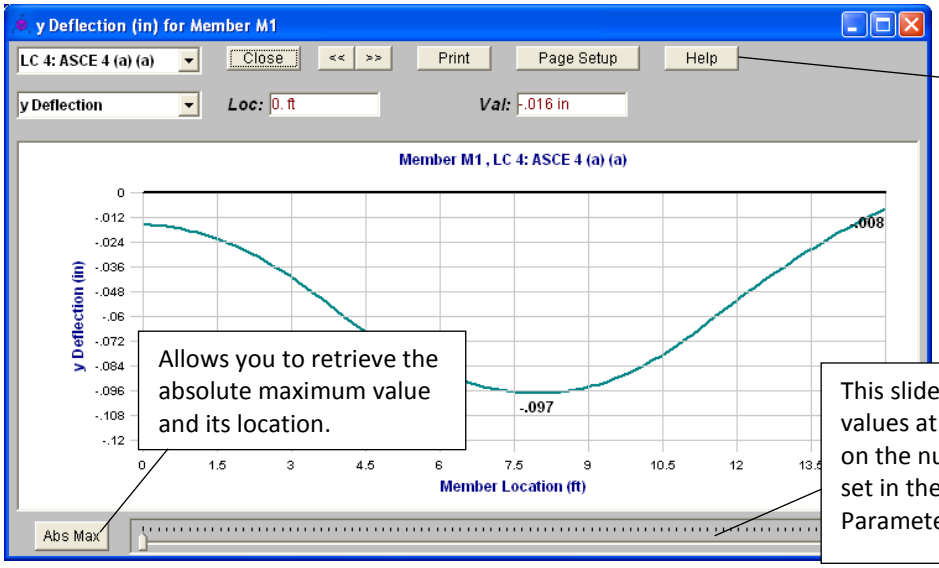


This detailed report displays all the forces, stresses, and deflections for any individual member.

Click on any diagram in the report to enlarge (see below).

- Click the local y-axis (**Dy**) deflection diagram (top center diagram).

The deflection diagram for member M1 will display:



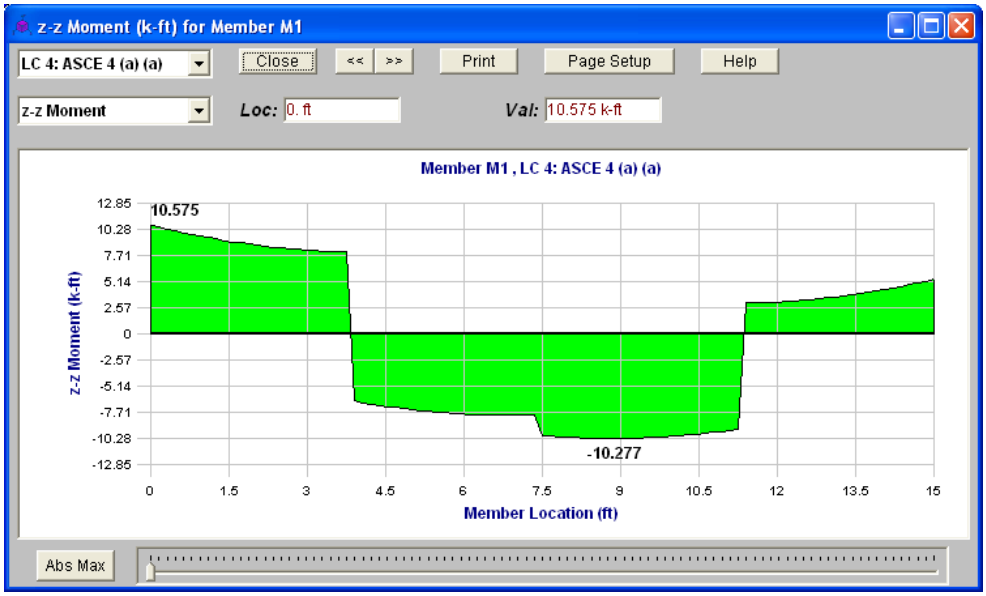
The buttons across the top allow you to close the diagram, view the previous member or the next member, or print.

Allows you to retrieve the absolute maximum value and its location.

This slider allows you to retrieve values at 97 sections. This is based on the number of Internal Sections set in the **Solution** tab of Global Parameters.

Besides the many ways to navigate these diagrams that are described above, you can also access diagrams for other values without returning to the model view or the detail report.

- Click the list containing **y Deflection**. The entire list of available diagrams will display. Select **z-z Moment** to display the z-z moment values for member M1.



Feel free to try any of the options or view another diagram from the list.

- When you are finished, click **Close** to close any diagrams you may have open.

You will be returned to the member detail report.

- Scroll down in the detail report to view code check information: code check value along with the controlling equation and all of the design values.
- Finally, click **Close** to close the detail report.

You may also open the member detail report by clicking on a member in the model view.

- On the Selection toolbar, click the **Detail** button. The cursor will now change to . Click on any member to generate a report for that member. Try clicking on a few members to see the various detail reports.

You may also open detail reports from within any spreadsheet.

- From the **Results** menu, select **Members - Design Results**. When the spreadsheet displays, click the **Hot Rolled Steel** tab.
- On the Window toolbar, click **Detail Report for Current Item**.

Plotting Plate Results

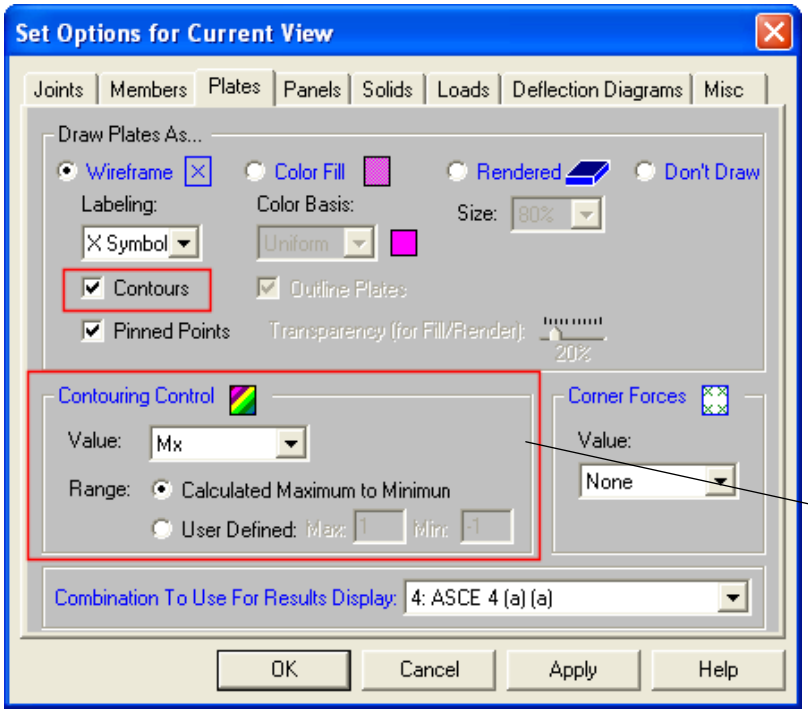
Before viewing the plate results, close any extra windows, turn off the joint labels, and set the plot options as follows:

- On the Window menu, select **Single View** to close the extra windows you may have open.
- Click **Joint Labels** to to turn off the joint labels.

- On the Window toolbar, click **Plot Options**  . Select the **Plates** tab.

Try contouring using the Wireframe and the Color Fill options to see the difference. Begin with the Wireframe option:

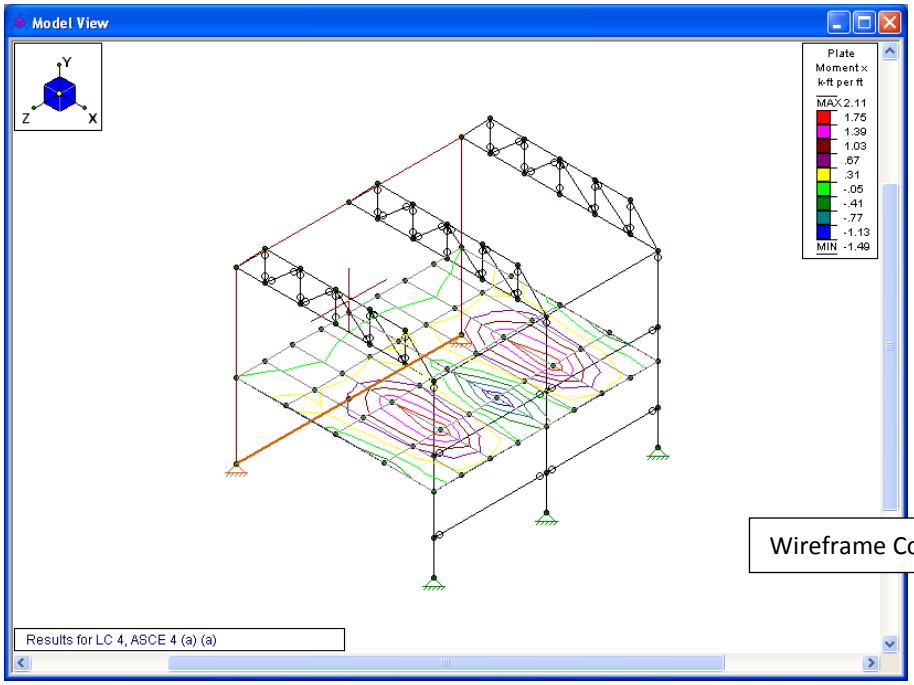
- Under **Wireframe**, select the **Contours** check box.
- In the **Contouring Control** section, in the **Value** list, select **Mx**.



Contours refers to the color coding to be displayed on the plates (or panels, or solids), where different colors represent different magnitudes for the particular result being "contoured." Under **Wireframe**, the contours are displayed as lines. Under **Color Fill**, the contours are displayed as color-filled areas.

Contouring Control sets the value to be contoured (for either **Wireframe** or **Color Fill** mode).

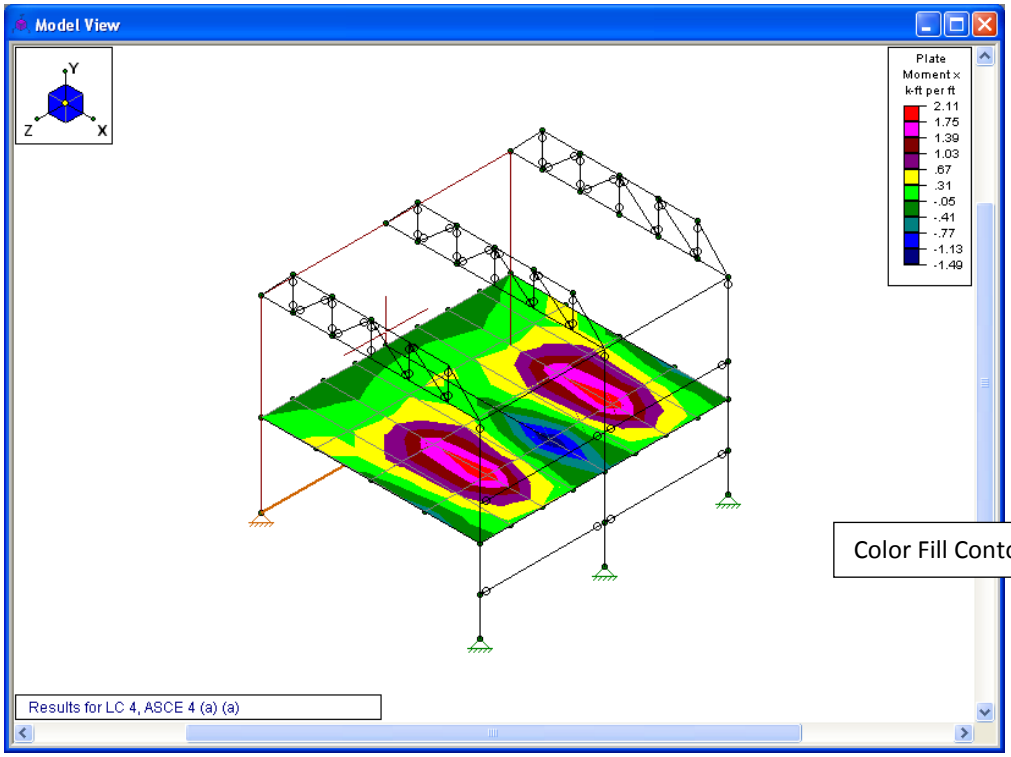
- Click **OK.**



Wireframe Contour

The colored lines represent certain values of Mx and the contouring enables you to see graphically how the Mx moment is distributed on the floors. Next, try the color filled contour.

- On the Window toolbar, click **Plot Options** to return to the plot options for the plates.
- Under **Color Fill**, in the **Color Basis** list, select **Contours**.
- Click **OK**.

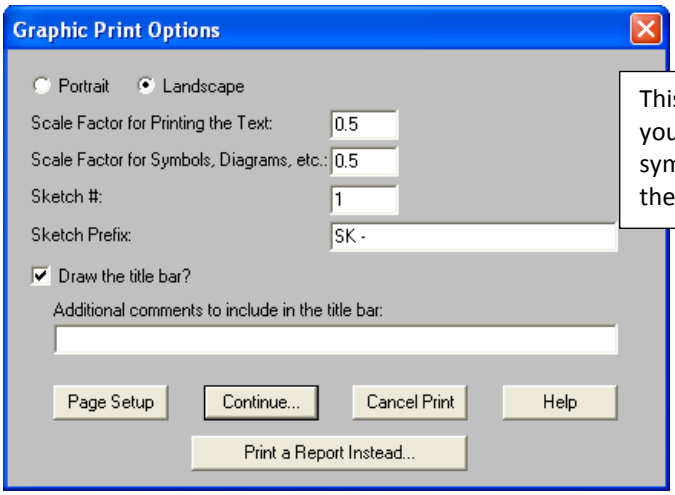


This "color fill" gives a more colorful representation of the Mx distribution. Note that for the previous wireframe contour, each line represented a specific value; however, for the color filled contouring, the colors represent a range of values (as displayed in the legend).

Printing Graphic Results

Print your results:

- On the Window toolbar, click **Plot Options**. Then, click the **Members** tab.
- Under **Draw Members As**, click **Rendered**.
- Click **OK**.
- On the RISA toolbar, click **Print**. The **Graphic Print Options** dialog box will appear.
- Click **Landscape** (paper orientation).



This dialog box allows you to customize your printout by: scaling fonts and symbols for labels and legends, including the title bar, or adding any comments.

- Click **Continue**. The **Print** dialog box will display. Select your printer, then click **OK** and the plot will be printed.

Tiling Results Windows

Window tiling provides some very useful options within RISA-3D.

- On the RISA toolbar, click the **Tiling** button. The **Window Tiling** dialog box will appear. Use the scroll bar to view the entire list of tiling options. Select **Plate Results**. Click **OK** to accept your tiling selection.
- On the RISA toolbar, click the **Tiling** button again. This time, select **Three Results Views**. This tiling option allows you to display three views at once. Click **OK**.

Batch Solution

The Batch solution option allows you to solve multiple combinations concurrently. When using this option, the results for each will be recorded and can then be viewed in the results spreadsheets and organized either by load combination or by item.

- On the RISA toolbar, click **Solve** again.
- Select **Batch Solution of Marked Combinations**. Then, click **Solve**. A message will display notifying you that the results will be cleared. Select **Yes**.

Note: Whenever you solve the model, RISA-3D will display a message notifying you that the results will be cleared (this alleviates the possibility of you having results data that does not match the input data). If you prefer to disable the warning message, you may do so in the Preferences settings (on the **Tools** menu, click **Preferences**).

After the solution is performed, RISA-3D does retain the decomposed stiffness matrix, so as long as the data items you edit do not impact the stiffness matrix, subsequent solutions will be much faster.

With the Batch solution complete, you may now choose different load combinations when plotting results. Each view may show results from a different combination.

- Click in the lower left view to make it the active window.
- On the Window toolbar, click **Plot Options**  . You should still be on the **Members** tab.
- In the **Combination to Use for Results Display** list, select **8: ACI 9-1 (a)**. Click **OK**.

The lower left view now shows results for Load Combination 8.

- On the **Results** menu, select **Reactions**.

For the Batch solution, the spreadsheets display all results for each load combination. Currently, the results are grouped by combination. All reactions for Load Combination 1 are listed first, then for each combination thereafter. The combination number is listed in the first column.

You can also group the results so that all the reactions for joint N1 are together, then N2, etc.

- On the **Results** menu, select **Results Presentation - Batch Results Listed By Item** to display them joint by joint. The first column shows the load combination.

Now, take a closer look at your concrete results.

- On the **Results** menu choose **Members - Design Results**.
- Click the **Concrete Beams** tab.

Take a moment to browse these beam results. This spreadsheet presents a maximum code check for the top reinforcing (negative bending), the bottom reinforcing (positive bending), and beam shear. In addition, the factored beam capacities are given for each critical location.

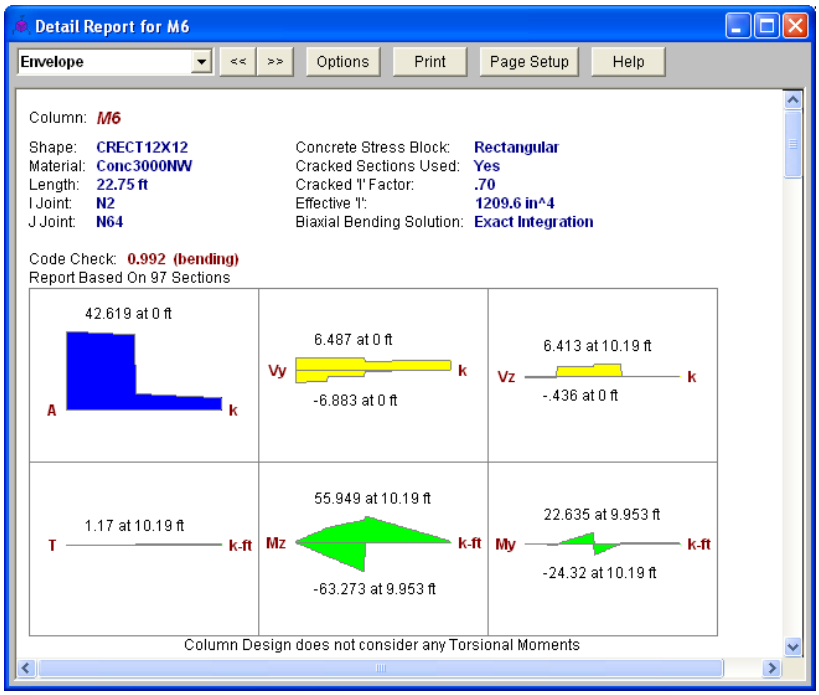
Notice that no governing load combination is listed for the concrete code check. This is because the rebar layout for a concrete beam is always based on an *envelope* of the design forces.

- Next, click the **Concrete Columns** tab.

The column design results are similar to the beam, except that the bending and shear results are given for *biaxial* loads.

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

This will display an envelope detail report for the concrete column that looks like this:



Because this detail report is based on moment and shear envelopes, it is somewhat different that the one you just reviewed for a steel member.

- Scroll down to take a look at the **Span Information** and reinforcing steel.

In concrete design, each column or beam is divided into spans, depending on how many points of support it has. This portion of the detail report also contains information on the reinforcing steel required for each region of each span.

Envelope Solution

An envelope solution is different from a single combination or a batch solution because it only stores the maximum and minimum values. Each of the spreadsheets will contain minimum and maximum values for each load combination included. Because the rest of the data is not stored, the detail reports and the deflected shapes are not available when this solution type is selected.

Solve the model again, this time as an envelope solution:

- First, select **Single View** from the **Window** menu to return back to the model view.
- Click **Solve**  .
- Select **Envelope of Marked Combinations**, then click **Solve**. Click **Yes** to agree to clear the results.
- Explore several of the spreadsheets to see how the results differ from the Batch solution.

Code Check Results

The program offers several ways to review the code check results. You can use a tiling option to organize your screen to include spreadsheet results as well as graphical model views showing the design data.

Change the tiling options:

- Click the **Tiling** button. Select **Member Steel Results**, then click **OK.**

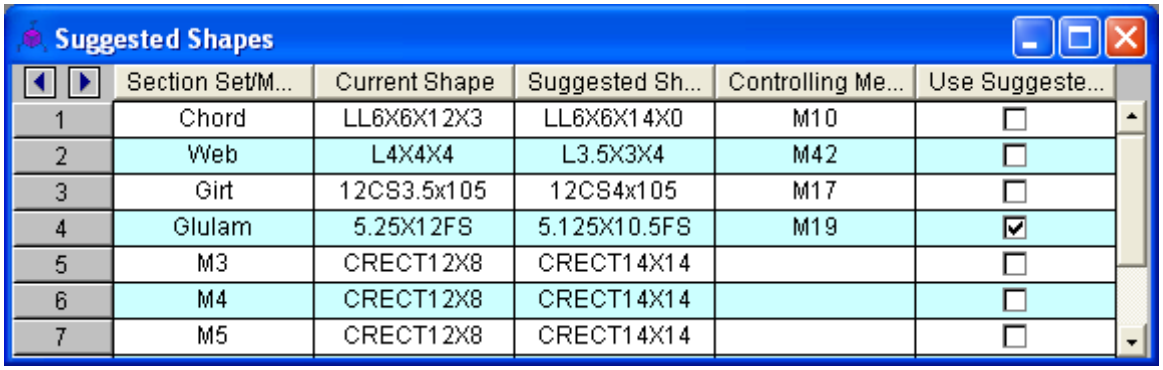
Notice the red members in the model view indicating that you have members that are failing the code check. The worst case results are now being reported in the spreadsheet, as a result of the envelope solution you ran. The columns labeled **LC** report the load combination that generated the controlling values. Other results, such as the member forces, display maximum and minimum results in order to capture the worst of both negative and positive results.

Optimizing Member Sizes

The **Suggested Shapes** spreadsheet reports RISA-3D's recommendations for alternate shape sizes, along with the member that is controlling the design.

Note: If you are already using the optimum shape for a section set or member, it will not be listed on this spreadsheet.

- Click the **Suggested Shapes** spreadsheet to make it active.
- Uncheck all the boxes in the **Use Suggested?** column except that in row **4** for the **Glulam** section.



	Section Set/M...	Current Shape	Suggested Sh...	Controlling Me...	Use Suggeste...
1	Chord	LL6X6X12X3	LL6X6X14X0	M10	☐
2	Web	L4X4X4	L3.5X3X4	M42	☐
3	Girt	12CS3.5x105	12CS4x105	M17	☐
4	Glulam	5.25X12FS	5.125X10.5FS	M19	☑
5	M3	CRECT12X8	CRECT14X14		☐
6	M4	CRECT12X8	CRECT14X14		☐
7	M5	CRECT12X8	CRECT14X14		☐

- On the Window toolbar, click **Replace and Resolve**  . Then, click **Yes** to agree to resolve.

Excluding Results

Next, you will use the **Steel Code Checks** spreadsheet to sort the members and hide the results for members that have a code check value lower than 0.7. This exemplifies how you can cut down the amount of data for review and verification in your spreadsheets, making it more manageable.

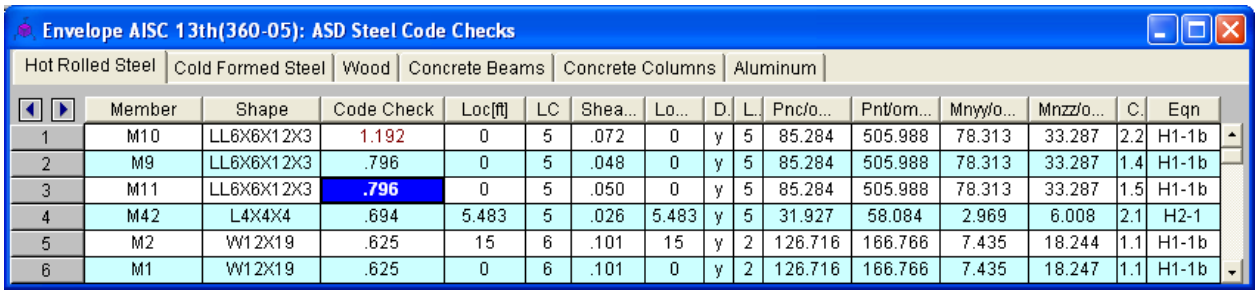
- Click on the **Envelope AISC 13th(360-05): ASD Steel Code Checks** spreadsheet to make it the active window. Click and drag the lower right corner of the window to enlarge it, so that you can view more of the results.

Now, sort the results based on the maximum code check to see which members are closest to failure.

- On the **Envelope AISC 13th(360-05): ASD Steel Code Checks** spreadsheet, click anywhere in the **Code Check** column.
- On the Window toolbar, click **Sort**, then select **Sort Max to Min,** and click **OK**.

Now, look for the last member with a code check value that is **0.7** or higher (which is **M11** in row **3**).

- In the **Code Check** column, click in row **3** to make **M11** the active row, as shown in the image below.



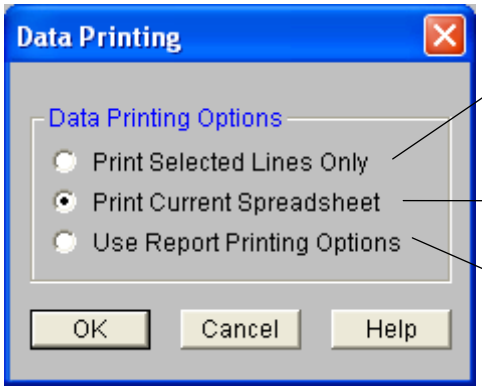
	Member	Shape	Code Check	Loc[ft]	LC	Shea...	Lo...	D...	L...	Pnc/o...	Pnt/om...	Mnyy/o...	Mnzz/o...	C...	Eqn
1	M10	LL6X6X12X3	1.192	0	5	.072	0	y	5	85.284	505.988	78.313	33.287	2.2	H1-1b
2	M9	LL6X6X12X3	.796	0	5	.048	0	y	5	85.284	505.988	78.313	33.287	1.4	H1-1b
3	M11	LL6X6X12X3	.796	0	5	.050	0	y	5	85.284	505.988	78.313	33.287	1.5	H1-1b
4	M42	L4X4X4	.694	5.483	5	.026	5.483	y	5	31.927	58.084	2.969	6.008	2.1	H2-1
5	M2	W12X19	.625	15	6	.101	15	y	2	126.716	166.766	7.435	18.244	1.1	H1-1b
6	M1	W12X19	.625	0	6	.101	0	y	2	126.716	166.766	7.435	18.247	1.1	H1-1b

- On the Window toolbar, click **Exclude After**. All rows below the active cell **(M11)** will be deleted from the spreadsheet (keep in mind, you can always click **Unexclude** to retrieve them).

Printing Reports and Spreadsheets

Next, you will explore printing options for the spreadsheets. Because you have excluded some results from the spreadsheet, note that the excluded information will not print.

- On the RISA toolbar, click **Print** to open the **Data Printing** dialog box.
- Click **Print Current Spreadsheet**, and click **OK**.

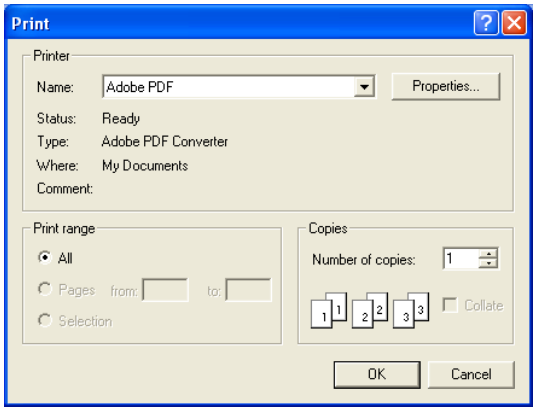


Allows you to print selected lines in a spreadsheet (select the lines you would like printed, then click **Print**).

Select this option to print a single spreadsheet.

This option allows you to print an entire report of your input and/or output.

Once you select the printing option you prefer, the **Print** dialog box will display:



- On the **Print** dialog box, select a printer, and click **OK**. The printer will then print your spreadsheet or report.

Printing Graphical Results

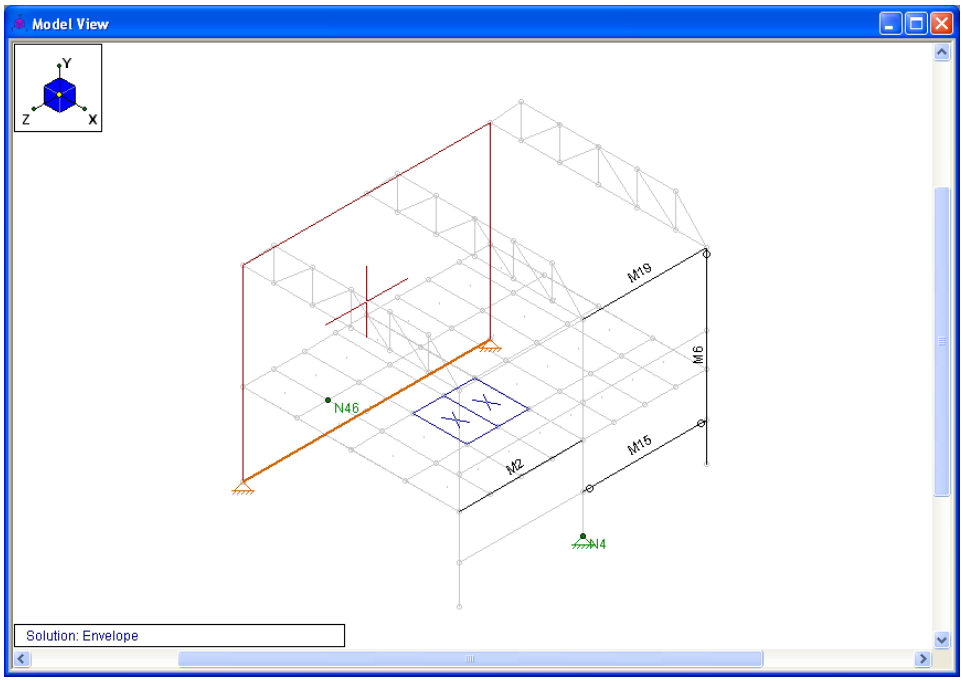
You can also graphically select items you would like to print using the graphical exclude feature.

- On the **Window** menu, select **Single View**.
- On the Window toolbar, click **Member Labels** to display the member labels.
- On the Selection toolbar, click **Unselect All**.

Then, make your selection of the items you would like to print:

- Click on the wall, one concrete member, one steel member, one wood member, one cold formed steel member, two joints, and two plates.

Your screen should now look something like this:



This next step will "exclude" the unselected items from the results:

- On the Selection toolbar, click **Exclude**.
- This will automatically open the **Exclude Results Confirmation** dialog. Click **Yes** to continue.

Now it is time to print:

- On the RISA toolbar, click **Print**  .
- Click **Print a Report Instead**.

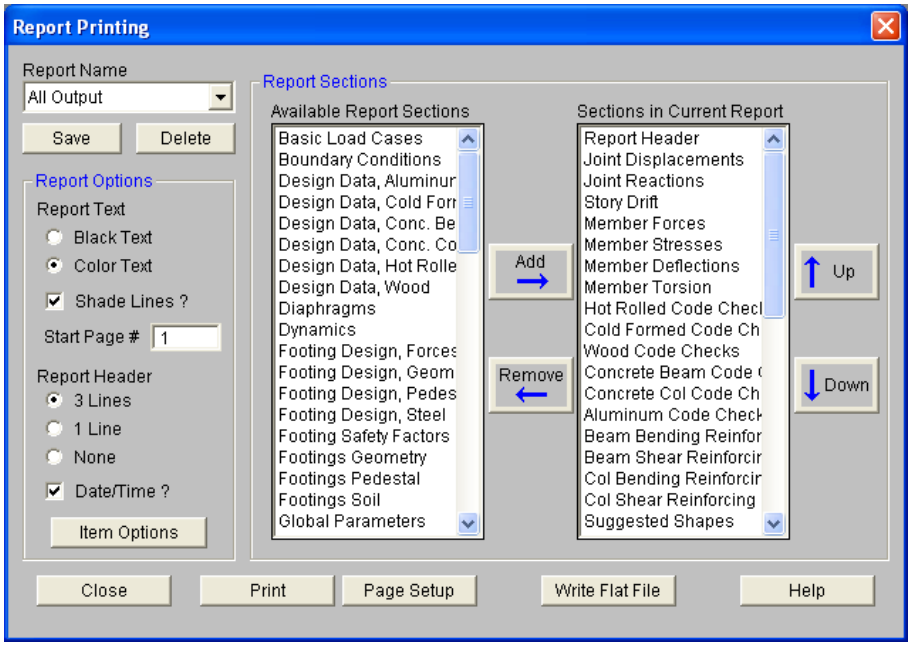
The **Report Printing** dialog box will display. As demonstrated below, you can select a standard report (by making a selection from the **Report Name** list) or you can build your own customized report (by making selections from all the available criteria).

- In the **Report Name** list, select **All Output**.

Notice that when you make this selection, items under **Report Sections** move from the **Available Report Sections** column to the **Sections in Current Report** column. Before you print, go ahead and remove a few items from the report.

- Under **Report Sections**, in the **Sections in Current Report** list, double-click **Story Drift**. It will now move to the other column. Also double-click **Member Torsion**, **Frequencies,** and **Mode Shapes** to remove them from the list so they will not be printed.

The dialog box should now look like this:



To move a section from one column to the other: highlight the section, and click the **Add** and **Remove** buttons.

Shortcut: Double-click an item to move it from one column to the other.

To adjust the printing order, use the **Up** and **Down** buttons.

Print your customized report:

- Click the **Print** button (at the bottom of the **Report Printing** dialog box). The **Print** dialog box will display. Click **OK**.
- Click **Close** to close the **Report Printing** dialog box.

This concludes Tutorial 4. The next tutorial demonstrates how to solve a dynamic analysis of the model.

Note: The Tutorial 5 starter file contains additional load combinations that were not created as part of Tutorial 3 or 4. You will use these additional load combinations to explore the dynamic solution. Exit the file without saving.

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

Tutorial 5 – Dynamic Analysis

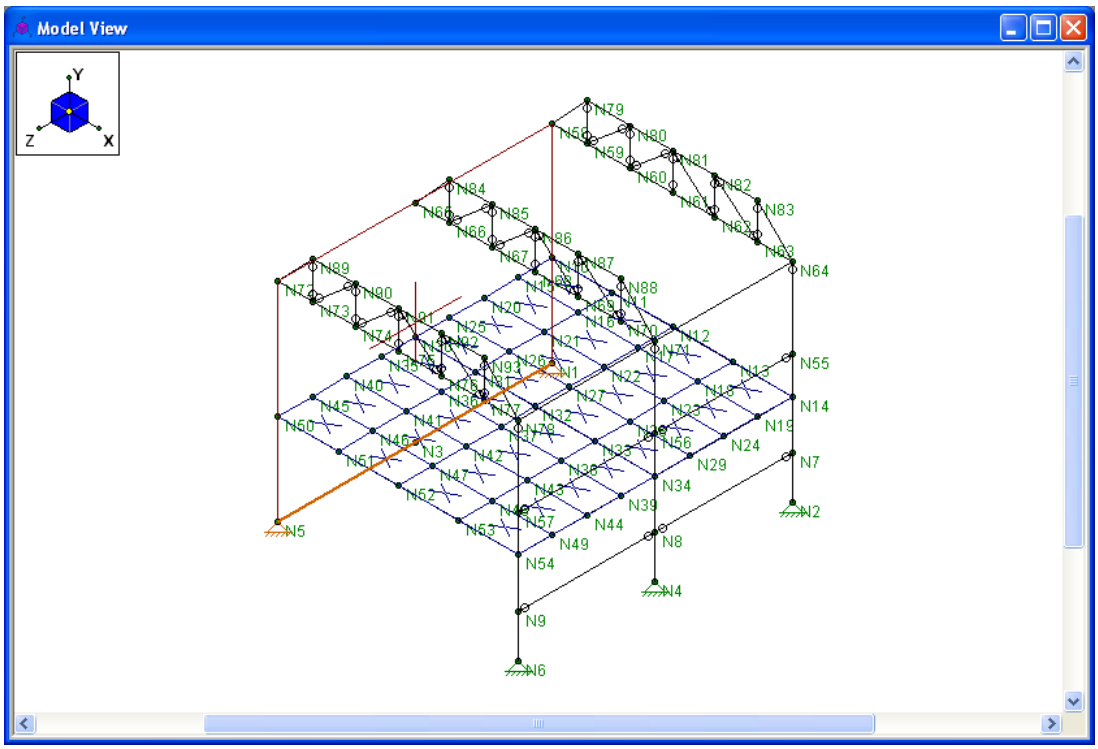
In this tutorial, you will perform a dynamic solution on the model you updated in the last tutorial. Make sure you start off correctly by opening the file mentioned next.

Note: The Tutorial 5 starter file contains additional load combinations that were not created as part of Tutorial 3 or 4. You will use these additional load combinations to explore the dynamic solution.

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

- Double-click the RISA-3D icon to start the program.
- Click **Open Model** .
  Double-click the **Tutorials** folder, select **Tutorial 5 Starter.r3d** and click **Open**.
  Click **Close** (or **Cancel**) to exit the **Global Parameters** dialog box.
- On the Window toolbar, click the **Graphic Editing Toolbar** button to activate the Drawing toolbar.
- On the **Data Entry** toolbar, click **Close** to close it.

Your screen should look like this:

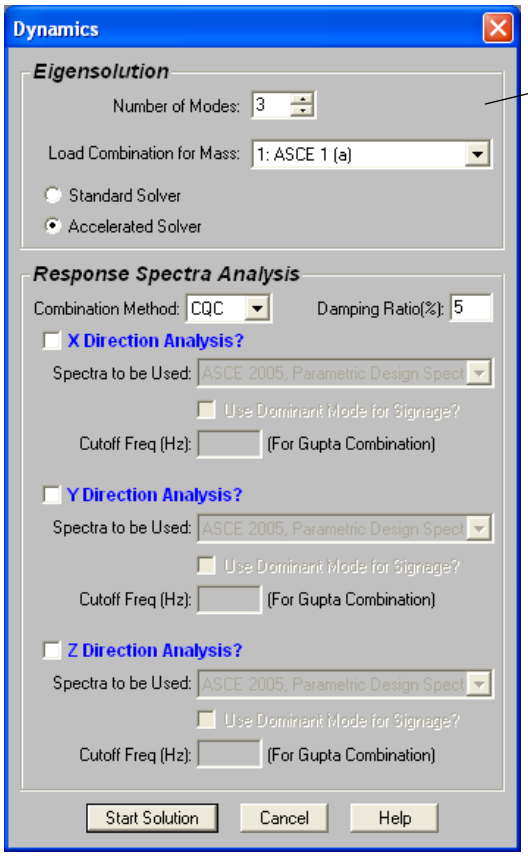


Dynamic Solution

In addition to static solutions, RISA-3D can also perform a dynamic analysis to calculate the modes and frequencies of your model. This then can be used in a response spectra analysis which uses the frequencies to calculate forces, stresses, and deflections in your model.

- On the RISA toolbar, click **Solve** .
- Click **Dynamics (Eigensolution/Response Spectra)**, then click **Solve**.

The **Dynamics** dialog box will display, with several options.



For the **Eigensolution** options, simply specify how many modes you want calculated, and which load combination should be used for mass calculation. Be sure to use the **Accelerated** solver as it is an advanced solver and will run the solution quicker.

For a detailed description of all available options, refer to the **Help** sections on **Dynamic Analysis** and **Response Spectra Analysis**. Not all features are covered in this tutorial.

Now, using the Eigensolution options, solve for three modes using load combination 1 to calculate mass.

- In the **Eigensolution** section, in the **Load Combination for Mass** list, select **1: ASCE 1(a)**.
- Click **Accelerated Solver**.
- Click **Start Solution** and the dynamic solution will begin.

The term “dynamic analysis” typically means the analysis of a model for dynamic loading, including the calculation of stresses and displacements. For RISA-3D, the analysis of a model for dynamic effects is considered to be composed of two parts: the dynamic analysis and the response spectra analysis.

The dynamic analysis you are performing now refers to the calculation of the modes and frequencies of vibration for the model. The response spectra analysis uses these modes to calculate forces, stresses, and deflections in the model.

Once the dynamic analysis is complete, the **Frequencies and Participation** spreadsheet will display.

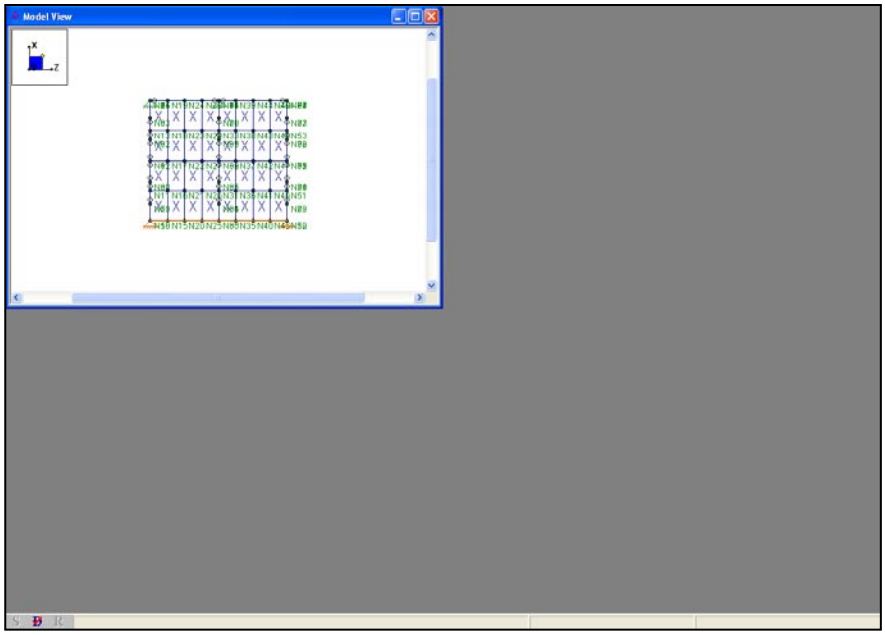
Frequencies and Mode Shapes

This spreadsheet lists the frequencies and periods calculated for the model. The last three columns are the modal participation percentages (which will not be available until after you perform the Response Spectra Analysis).

You can also review the mode shapes graphically. Set up your screen a bit differently to look at the mode shapes.

- On the RISA toolbar, click the **Tiling** button, select **Three Input Views**, then click **OK**.
- Click **Close** to close the model view on the right and the bottom of the workspace.
- Click **Close** to close the **Results** toolbar.

You should be left with just this upper left model view:



Now change to an isometric view:

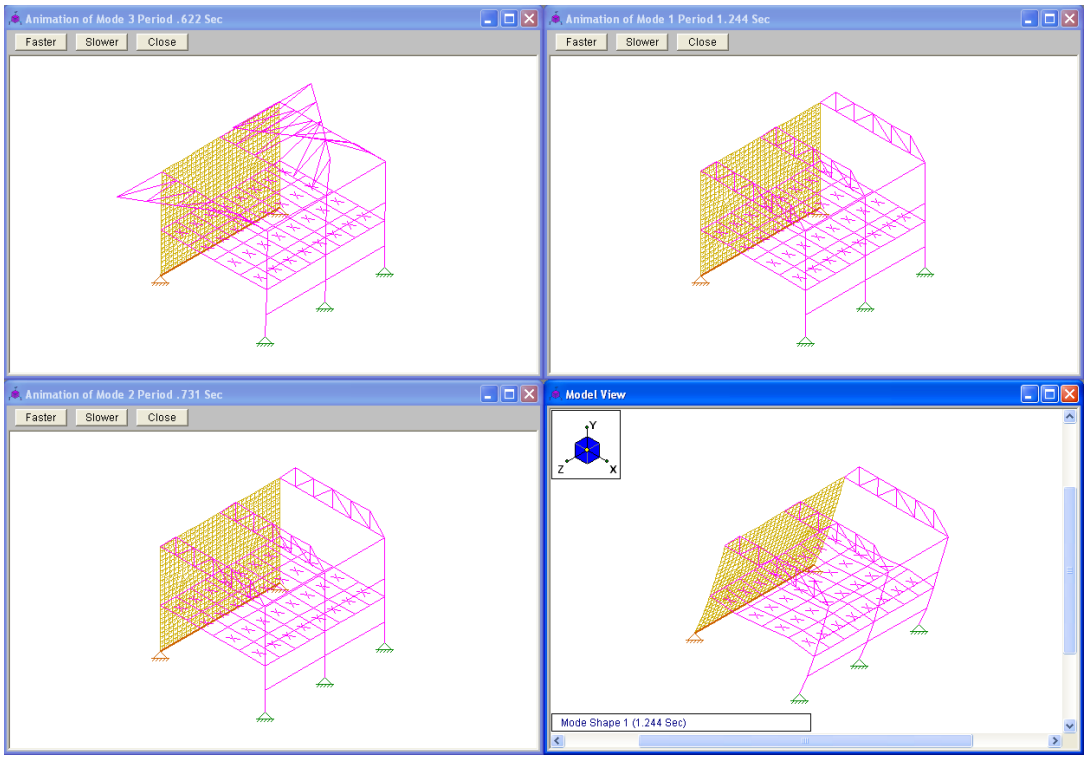
- On the Window toolbar, click **Isometric**, then click **Plot Options**.

The **Plot Options** dialog box will appear.

- Click the **Joints** tab. Click to clear the **Show Joints** check box.
- Click the **Deflection Diagrams** tab. Under **Show Deflected Shape For**, click **Mode Shape**. Then, click the **Animate ALL the Mode Shapes** button. A message will display alerting you that this may take a while (however, it should not for just three modes). Click **Yes**.

Three different progress bars will display, one after the other, as the three different animations are being built. After the animations are built and the progress bars no longer display.

- Click **OK** to close the **Plot Options** dialog box.
- On the **Window** menu, click **Tile Horizontal**.



When you have finished viewing the animations, change to a single view:

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

Response Spectra Analysis

Solve the model again, this time for Response Spectra Analysis results:

- Click **Solve** .
- Once again, select **Dynamics (Eigensolution/Response Spectra)**, then click **Solve**.

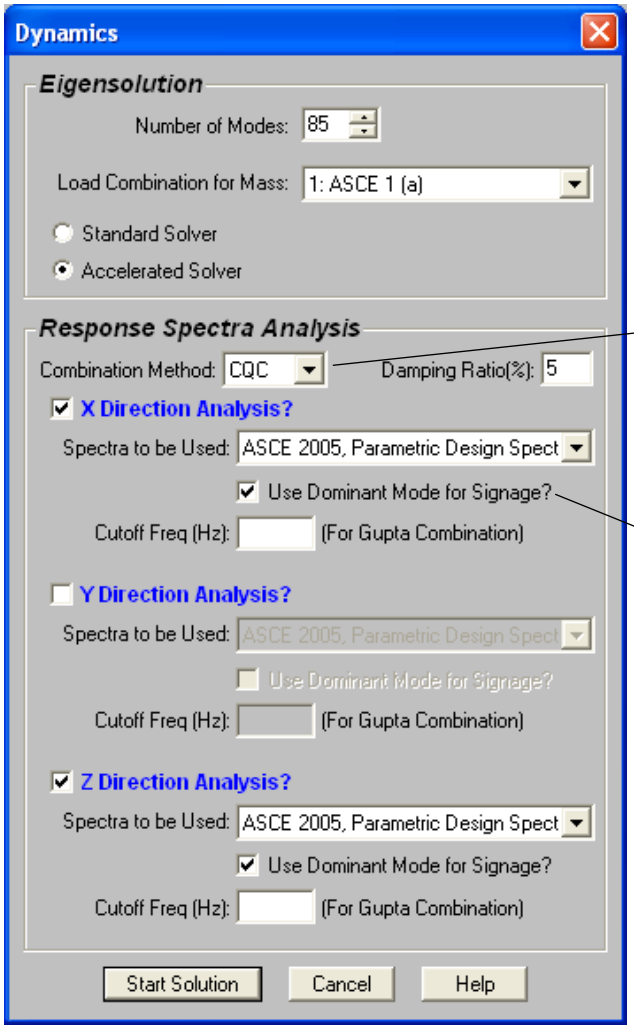
The **Dynamics** dialog box will appear.

Begin defining the solution. Specify the current combination method and damping ratio:

- In the **Eigensolution** section, in the **Number of Modes** list, type **85**.
- In the **Response Spectra Analysis** section, in the **Combination Method** list, select **CQC**. In the **Damping Ratio (%)** box, type **5**.

For this RISA-3D analysis, you will use the ASCE 2005, Parametric Design Spectra. See Figure 11.4-1 in the *ASCE 7-05* for the equations used to build the spectra.

- Select the **X Direction Analysis?** check box. Then, in the **Spectra to be Used** box, select **ASCE 2005, Parametric Design Spectra**. Then, select the **Use Dominant Mode for Signage?** check box. This specifies that RISA-3D automatically detect and use the dominant mode for signage.
- Select the **Z Direction Analysis?** check box. In the **Spectra to be Used** box, select **ASCE 2005, Parametric Design Spectra**. Then, select the **Use Dominant Mode for Signage?** check box.



In the **Response Spectra Analysis** section, each direction offers multiple analysis options. The first field is a list of the available spectra in the RISA-3D spectra library.

The **Combination Method** option gives you control over how the individual modal results will be combined together. Most building codes require you to use the **CQC** method of combination.

The **Use Dominant Mode for Signage** option uses the sign convention of the dominant mode to set the sign convention of all other modes. This can help avoid confusion when comparing dynamics results or when viewing the deflected shape.

For a detailed description of all available options, refer to the **Help** sections on **Dynamic Analysis** and **Response Spectra Analysis**. Not all features are covered in this tutorial.

You have specified the spectra to be applied in the X and Z directions, and use the dominant mode for results signs. You can now start the solution:

- Click the **Start Solution** button.

When this solution is complete, this **Frequencies and Participation** spreadsheet will display with mass participation percentages in the **SX Participation** and **SZ Participation** columns.

Modal Participation

The modal participation percentages are now recorded for the X and Z directions and summed at the bottom of the spreadsheet.

Note: Make sure the participation for each direction totals to 90% or more! For this model, you have met these criteria, but if you are running a model where the participation total is less than 90%, you must return to the dynamics dialog box, increase the number of modes and rerun the dynamic solution and also the RSA.

The final step is to include these RSA results with the static loads in a set of load combinations to obtain the final, overall model solution.

Scaling Factors

Next, you will explore the calculation of the spectra scaling factors. The actual procedure will not be detailed here--only a quick overview (for a detailed example of the procedure, search the **Help** index using the keywords **Scaling Factor**, then select Dynamic Analysis- Response Spectra - RSA Scaling Factor).

The following overview of the procedure is based on the requirements in *Chapter 12 of the ASCE 7-05*. If you are not familiar with these requirements, or if you use another building code, refer to the code you will be using to understand the basis for the procedure.

There are two reasons for having to calculate scaling factors: First, if a "normalized" spectra was used to calculate the spectral results, you must scale the normalized results to match your site-specific and structure-specific criteria. Second, the ASCE sets minimum values for the Seismic Response Coeffiecient, Cs, which limits the value of the design base shear.

In summary, what has to be done is:

- Calculate the static design base shear (V).
- Obtain the unscaled RSA Elastic Response base shear.
- Scale the RSA base shear such that the program-calculated value matches that calculated per the ASCE 7-05 hand-calculation methods.

Note: Remember that you can get the details for the scaling factor calculation from the Help file as described above.

RISA-3D has a feature that will automatically calculate the Equivalent Lateral Force Base Shear and scale your RSA results to match. This feature is described in the next section.

Load Combinations with RSA Results

These scale factors will now be applied to your spectral results so you can combine them with your static results. To see how this is done, return to the **Load Combinations** spreadsheet:

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

Notice the spreadsheet has some additional load combinations not seen in the previous tutorials. These have been entered as examples to demonstrate how to apply the RSA results to your load combinations:

Load Combinations

Combinations | Design

	Description	Sol...	PD...	SR...	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor
1	ASCE 1 (a)	☑	Y		DL	1														
2	ASCE 2 (a)	☑	Y		DL	1	LL	1	LLS	1										
3	ASCE 3 (a) (a)	☑	Y		DL	1	RLL	1												
4	ASCE 4 (a) (a)	☑	Y		DL	1	LL	.75	LLS	.75	RLL	.75								
5	ASCE 5 (a)	☑	Y		DL	1	WL	1												
6	ASCE 6 (a)	☑	Y		DL	1	WL	.75	LL	.75	LLS	.75	RLL	.75						
7	ASCE 7	☑	Y		DL	.6	WL	1												
8	ACI 9-1 (a)	☑	Y		DL	1.4														
9	ACI 9-2 (a) (a)	☑	Y		DL	1.2	LL	1.6	RLL	.5										
10	ACI 9-2 (b) (a)	☑	Y		DL	1.2	LL	1.6												
11	ACI 9-3 (a) (a)	☑	Y		DL	1.2	RLL	1.6	LL	1										
12	ACI 9-3 (d)	☑	Y		DL	1.2	RLL	1.6	WL	.8										
13	ACI 9-3 (e)	☑	Y		DL	1.2	WL	.8												
14	ACI 9-4 (a)	☑	Y		DL	1.2	WL	1.6	LL	1	RLL	.5								
15	ACI 9-4 (b)	☑	Y		DL	1.2	WL	1.6	LL	1										
16	ACI 9-4 (c)	☑	Y		DL	1.2	WL	1.6	LL	1	RL	.5								
17	ACI 9-6	☑	Y		DL	.9	WL	1.6												
18	DL+LL+SX+SZ*0.3	☑	Y		L2	1	SX	1	SZ	.3										
19	DL+LL+SX*0.3+SZ	☑	Y		L2	1	SX	.3	SZ	1										

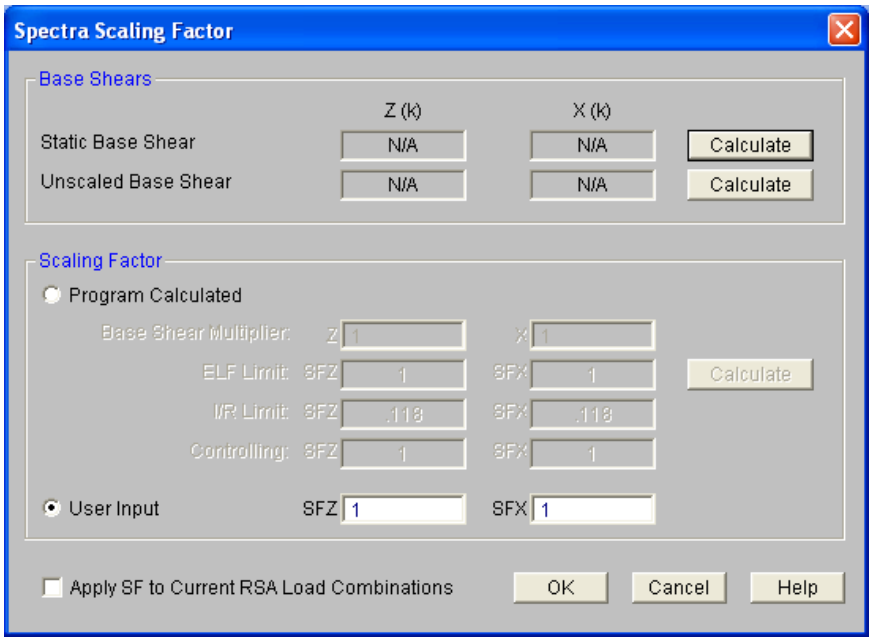
Notice on the last two lines, the spectra analysis results (SX and SZ) have been combined with the combination of loads previously defined in Load Combination 2 (L2).

Using 100% dynamic response in one direction with 30% in the other direction is a common way of accounting for directional effects from RSA's in two directions at the same time. An alternative to this method is an SRSS combination, which is discussed in more detail in *RISA-3D General Reference*.

The only problem is that the SX and SZ entries reflect the full *unscaled* spectra results. Therefore, you must scale them down using the SF_X and SF_Z scaling factors that you calculated in the previous section.

- On the Window toolbar, click the **Spectra Scaling Factor** button SF.

The **Spectra Scaling Factor** dialog box will display:



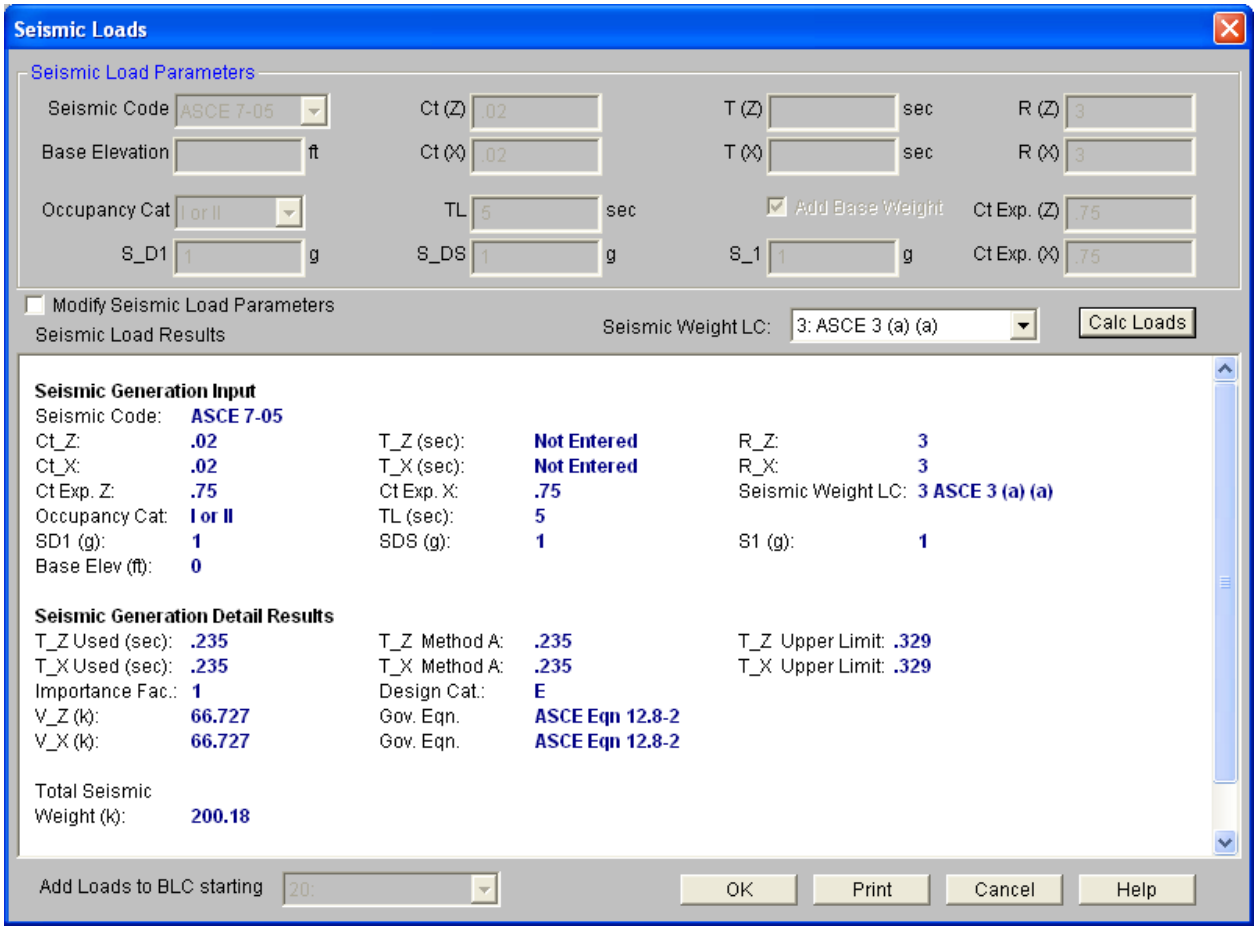
Now, have RISA-3D calculate the base shear values and scale the RSA results to match.

- In the **Base Shears** section, on the **Static Base Shear** row, click **Calculate**.

The **Seismic Loads** dialog box will display.

- In the **Seismic Weight LC** list, select **3: ASCE 3 (a)(a)** and click **Calc Loads**.

RISA-3D will calculate the static base shear based on the *ASCE 7-05* code. The **Seismic Loads** dialog box should look like this:

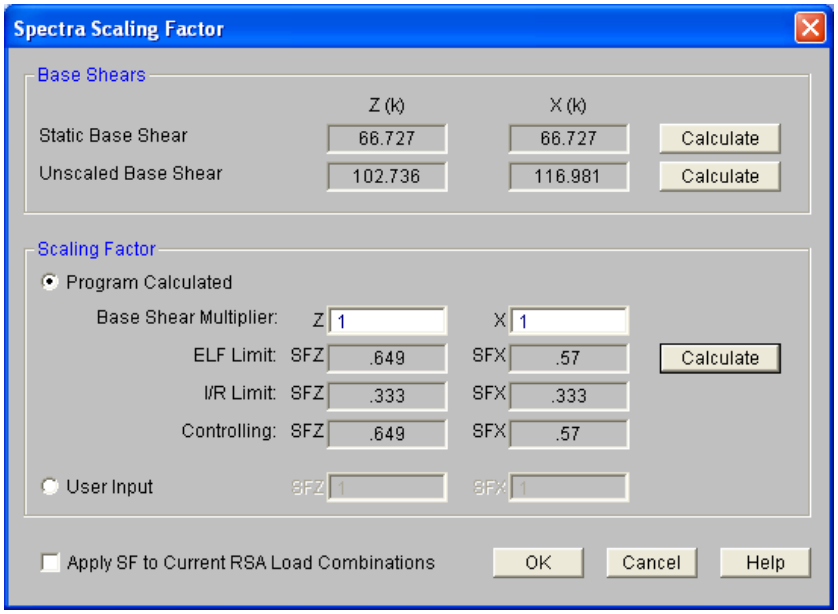


- Click **OK** to return to the **Spectra Scaling Factor** dialog box.
- In the **Base Shears** section, on the **Unscaled Base Shear** row, click **Calculate**.

RISA-3D has now calculated the unscaled base shear and you are ready to scale the RSA results to match the Unscaled Base Shear and Static Base Shear.

- Under **Scaling Factor**, click **Program Calculated**. Then, in the same area, click **Calculate**.

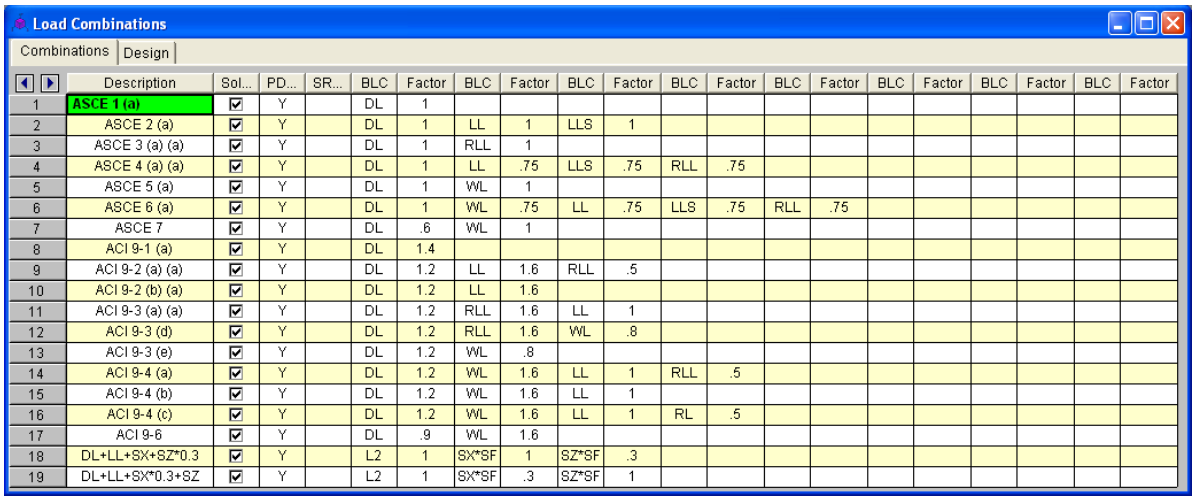
The **Spectra Scaling Factor** dialog box should look like this:



Once you have completed the calculations, apply the scaling factors to current load combinations:

- Select the **Apply SF to Current RSA Load Combinations** check box.
- Click **OK**.

Review the load combinations below and see how the SF scale factors allow the RSA load combinations to more closely reflect the code required load combinations.



	Description	Sol...	PD...	SR...	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor
1	ASCE 1 (a)	☑	Y		DL	1														
2	ASCE 2 (a)	☑	Y		DL	1	LL	1	LLS	1										
3	ASCE 3 (a) (a)	☑	Y		DL	1	RLL	1												
4	ASCE 4 (a) (a)	☑	Y		DL	1	LL	.75	LLS	.75	RLL	.75								
5	ASCE 5 (a)	☑	Y		DL	1	WL	1												
6	ASCE 6 (a)	☑	Y		DL	1	WL	.75	LL	.75	LLS	.75	RLL	.75						
7	ASCE 7	☑	Y		DL	.6	WL	1												
8	ACI 9-1 (a)	☑	Y		DL	1.4														
9	ACI 9-2 (a) (a)	☑	Y		DL	1.2	LL	1.6	RLL	.5										
10	ACI 9-2 (b) (a)	☑	Y		DL	1.2	LL	1.6												
11	ACI 9-3 (a) (a)	☑	Y		DL	1.2	RLL	1.6	LL	1										
12	ACI 9-3 (d)	☑	Y		DL	1.2	RLL	1.6	WL	.8										
13	ACI 9-3 (e)	☑	Y		DL	1.2	WL	.8												
14	ACI 9-4 (a)	☑	Y		DL	1.2	WL	1.6	LL	1	RLL	.5								
15	ACI 9-4 (b)	☑	Y		DL	1.2	WL	1.6	LL	1										
16	ACI 9-4 (c)	☑	Y		DL	1.2	WL	1.6	LL	1	RL	.5								
17	ACI 9-6	☑	Y		DL	.9	WL	1.6												
18	DL+LL+SX+SZ*0.3	☑	Y		L2	1	SX*SF	1	SZ*SF	.3										
19	DL+LL+SX*0.3+SZ	☑	Y		L2	1	SX*SF	.3	SZ*SF	1										

Now that you have added the dynamic results to your load combinations, you may run a solution that includes them.

- On the Window toolbar, click **Solve Envelope**.
- When the solution is complete, click the **Tiling** button, select **Member Steel Results** and click **OK.**

You are now finished with Tutorial 5. If you want more practice, you can go back and increase member sizes, rerun the dynamics, solve different load combinations, or experiment with the plot in any way you like.

- To exit RISA-3D, on the **File** menu, select **Exit**.

Tutorial 6 – Interoperability

RISA-3D has several options for importing or exporting projects to and from other industry software. This tutorial will introduce you to several of those options and explore how to make these options assist your modeling.

RISA-Revit Structure Link

Because the RISA-Revit link is a bi-directional exchange link, new models can be created in either RISA-3D or Revit Structure and edited in either program. The model will update in both programs as you move between the two during your design process.

Start by exploring the BIM exchange file export options from RISA-3D to Revit Structure. If you do not have the RISA-Revit link, you may download it at no cost from our website. Below are instructions on how to install the link.

Exit RISA-3D, and download the RISA-Revit link:

- Click **Close** to exit RISA-3D if you have it open.
- Go to http://www.risa.com/partners/prt_revit.html, click **Downloads**, and then select the **RISA-Revit** version you want to download.
- Download and install the link.

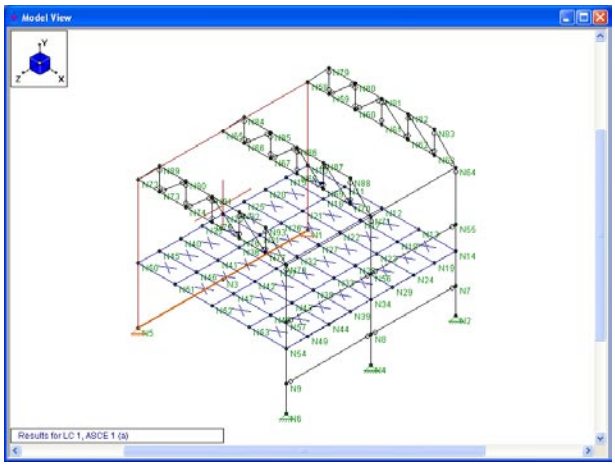
Follow the steps below to load the RISA-3D starter file for this tutorial:

- Double-click the **RISA-3D** icon to start the program.
- On the **File** menu, click **Open**. Double-click the **Tutorials** folder. Select **Tutorial 6 Starter.r3d** and click **Open**. Click **Close** (or **Cancel**) to exit the **Global Parameters** dialog box.

Solve the model and save:

- Click **Solve** to solve the model.
- Select a **Batch** solution.

Your model should look like the following image:



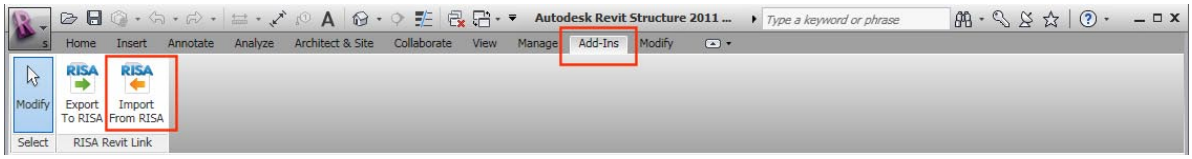
- On the **File** menu, click **Save As** and enter a new file name.
- On the **File** menu, click **Export**, then select **BIM Exchange File**, and designate a name for your exchange file.
- On the **File** menu, click **Exit** to close RISA-3D.

Next, open Revit Structure and import this exchange file:

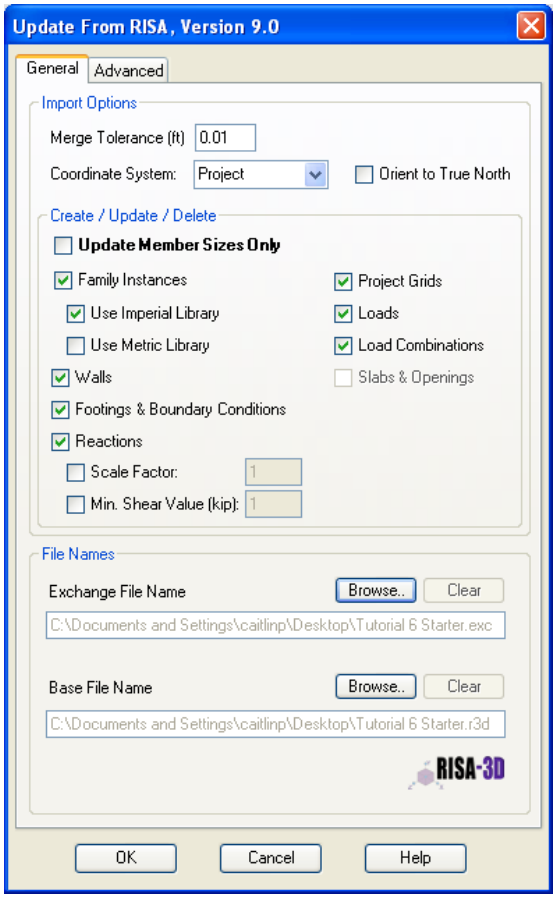
- Open a new file in Revit Structure.
- On the **Main** menu, click **Save**, and specify **Tutorial_6.rvt** as the file name.

Note: This tutorial is based on the RISA interface with Revit Structure 2011, which is identical to the procedure for the RISA-Revit 2010 link.

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



- Click **RISA-3D**, and then click **Browse** to locate the BIM exchange file (.exc), as shown below.



Once in Revit Structure, you can modify your model and then export back to RISA-3D. Because of its two-way functionality, you can modify the model in either program and the other will recognize the updates.

This was just a basic description of the RISA-Revit Link export option. For more information, download the 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, click **Downloads**, under **RISA-Revit Link Documentation**, select a document to download, either *RISA-Revit Link Documentation* or *RISA-Revit Quick Tips*.

CIS/2 Translator

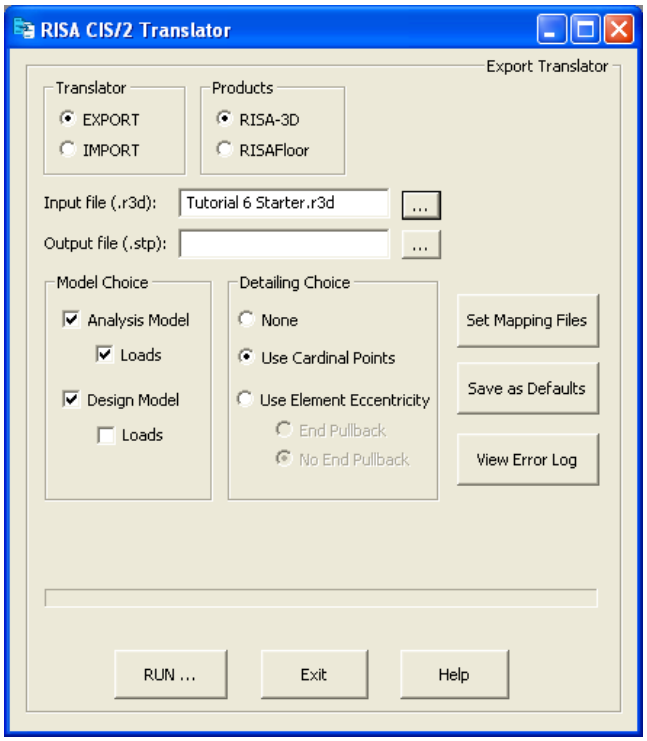
The RISA CIS/2 Translator is a tool for importing or exporting data from RISA to the CIM steel part 21 file format. You may download this translator at no cost from the RISA website.

To download and install the translator:

- Click **Close** to exit RISA-3D if you have it open.
- To download the translator, go to http://www.risa.com/products.html, click **RISA CIS/2 Translator**.
- Install the translator.

To open the RISA CIS/2 Translator:

- On the **Start Menu,** select **All Programs**, click **RISA**, and then click **RISA CIS/2 Translator**. A dialog box will display.
- Enter the following information to match the image below:



Run the export:

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

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

RISA-3D & CAD

RISA-3D 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, RISA-3D produces CAD-quality drawings that include your beam sizes, camber, stud layout, end reactions, etc.

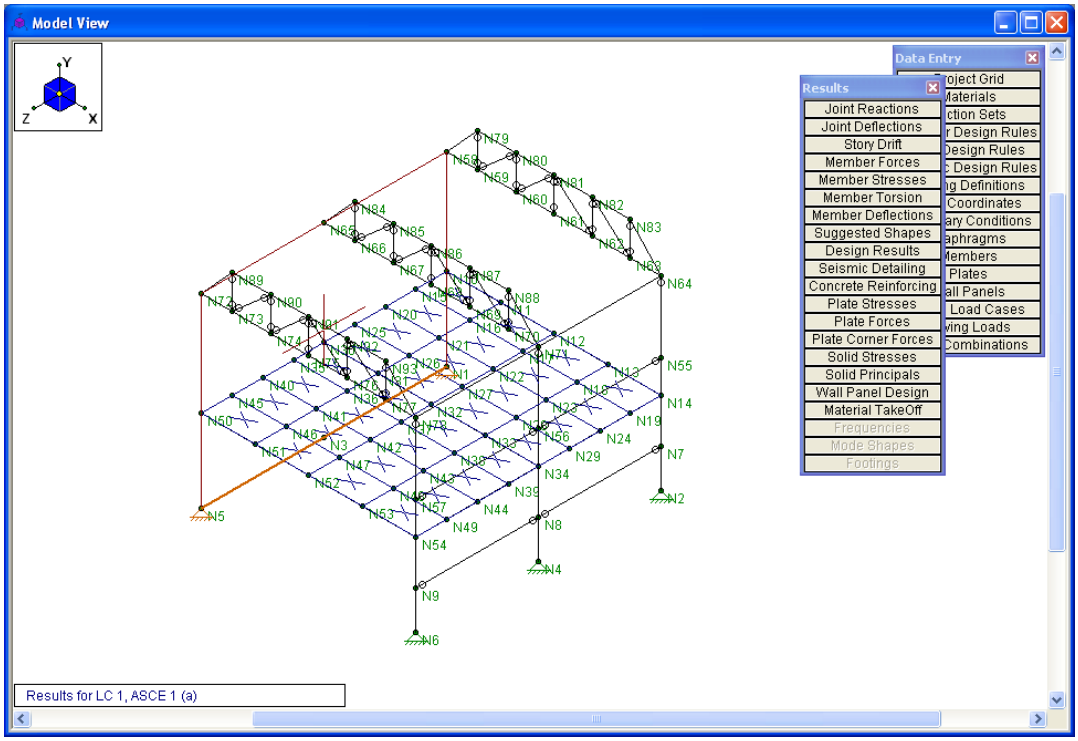
Follow the steps below to load the RISA-3D starter file:

- Double-click the **RISA-3D** icon to start the program.
- On the **File** menu, click **Open**  . Double-click the **Tutorials** folder. Select **Tutorial 6 Starter.r3d** and click **Open**. Click **Close**  (or **Cancel**) to exit the **Global Parameters** dialog box.

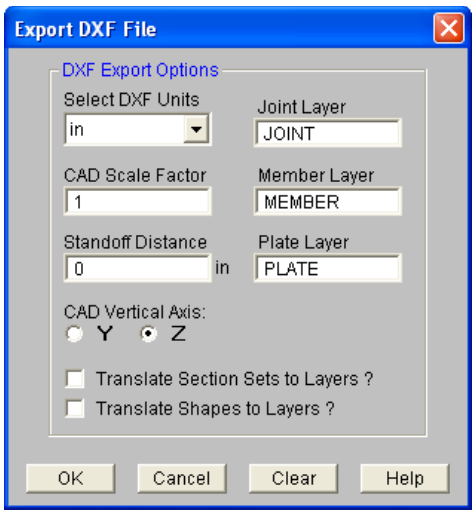
Solve the model and save:

- Click **Solve**  to solve the model.
- Select a **Batch** solution.

Your model should look like the following image:



- On the **File** menu, click **Save As** and enter a new file name.
- On the **Main** menu, click **File**, click **Export**, and select **DXF File**.
- Enter the file name **Tutorial 6.dxf** and click **Save**. The **Export DXF File** dialog box will display.
- Enter the information shown below and click **OK**.

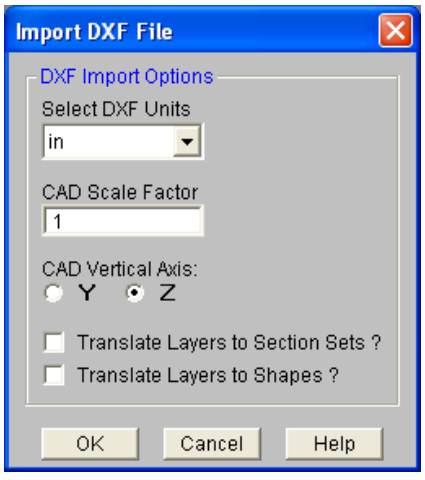


You should be able to open this DXF file with any standard drafting package. It will include simple model geometry to assist you with your drafting.

Similarly, you can import a DXF file into RISA-3D for analysis.

- On the RISA toolbar, click **New Model** to create a new model.
- Because you want to import geometry rather than start from scratch, click **Close** to exit the **Starting a Model** dialog box.
- On the **File** menu, click **Import**, then select **DXF File**.
- Select the tutorial file you recently exported and saved.

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



- Click **OK**. This will now create the model from the imported DXF file.

Conclusion

Congratulations on completing your introductory tour of RISA-3D! 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, fax at (949) 951-5848, or email at info@risatech.com.

Appendix A – RISA-3D Toolbar Buttons

RISA Toolbar

Button	Title	Label in RISA-3D	Shortcut
	New Model Start a new model	Start a new model	(CTRL+N)
	Open Model Open an existing model	Reload a previously saved model	
	Save Save current model	Save the current model	(CTRL+S)
	Copy Copy to the clipboard	Copy to the clipboard	(Ctrl+C)
	Print Print a report or graphic image	Print a report or graphic image	(Ctrl+P)
	Undo Undo the last operation	Undo the last operation	(Ctrl+Z)
	Redo Reverse the most recent undo operation	Reverse the most recent undo operation	(Ctrl+Y)
	Set Global Parameters	Set Global Parameters	
	Define Units Define units to be used	Define units to be used	
	Shape Database Edit the shape database	Edit the Shape Database	
	Moving Load Patterns View/edit the moving load patterns	View/edit the moving load patterns	
	Response Spectra Library Edit the response spectra library	Edit the response spectra library	
	Rebar Layout Create and edit a rebar layout	Create and edit a rebar layout	

	**Project Grid**  Open project grid spreadsheet	Project grid spreadsheet
	**High Level Generation** 	Perform High Level Generation operations
	**New Model View**  Create a new model view	Create a new model view
	**Open Spreadsheets**  Select spreadsheets to open	Select spreadsheets to open
	**Refresh**  Refresh all windows with current data	Refresh all windows with the most current data
	**Basic Load Case**  Open the basic load case spreadsheet	Open the Basic Load Case (BLC) spreadsheet
	**Load Combinations**  Open the load combinations spreadsheet	Load combinations spreadsheet
	**Solve**  Perform the analysis and design calculations	Perform the analysis and design calculations
	**Browse Results**  Select results to browse	Select results to browse
	**Erase Results**  Erase all solution results	Erase all solution results
	**Data Entry** toolbar  Turn the spreadsheet shortcuts window on or off	Turn the spreadsheet shortcuts window off or on
	**Results** toolbar  Turn the results shortcuts window on or off	Turn the results shortcuts window off or on
	**Help**  View help topics	View Help topics
	**Tiling**	Tile Window Options

Window Toolbar

…in Model View

Button	Title	Label in RISA-3D	Shortcut
	Plot Options Display plot options	Bring up the plot options dialog	F2
	Rotate buttons	Rotate the view counter-clockwise about the X axis	
		Rotate the view clockwise about the X axis	
		Rotate the view counter-clockwise about the Y axis	
		Rotate the view clockwise about the Y axis	
		Rotate the view counter-clockwise about the Z axis	
		Rotate the view clockwise about the Z axis	
	View buttons		
Iso	**Isometric** Display an isometric view	Snap to an Isometric View	
XY	**XY Planar** Display an XY planar view	Snap to an XY Planar View	
XZ	**XZ Planar** Display an XZ planar view	Snap to an XZ Planar View	
YZ	**YZ Planar** Display a YZ planar view	Snap to an YZ Planar View	
	3 buttons below are collectively called **Zoom** buttons		
	Zoom In	Zoom IN (closer view) on the model (+)	PLUS (+)
	Zoom Out	Zoom OUT (view farther away) on the model (-)	MINUS (-)
	Box Zoom	Draw a box around the part of the model to be zoomed	

	**Redraw**  Redraw full model view	Redraw full model view	
	**Graphic Editing**  Activate the graphic editing (or drawing) toolbar	Activate the Graphic Editing toolbar	Ctrl+G
	**Save or Recall View**  Save or recall view states	Save or recall view states	
	**Clone View**  Make an exact copy of current model view	Clone (make an exact copy of) the current model view	
	3 buttons below collectively refer to **Loads Display**  		
	**Display Loads**  Toggle display of the loads (LC or CAT)	Toggle display of the loads (LC or CAT)	
	Loads List – lists the available loads  		
	**Switch Loads**  Switch loads display (combinations or categories)	Switch loads display between combinations and categories	
	**Rendering**  Toggle between wireframe and rendering of beams and plates	Toggle between wireframe and rendering of beams and plates	
	**Joint Labels**  Joint labels toggle	Toggle the joint labels	
	**Boundary Conditions**  Toggle display of the boundary conditions	Toggle display of the boundary conditions	
	**Member Labels**  Member labels toggle	Toggle the member labels	
	**Member Color Basis**  Toggle display of member color coding	Toggle display of member color coding	
	**I End-J End**  Toggle display of member I end, J end representation	Toggle display of member I end, J end representation	

	Show Member Results Toggle display of member results (only available after solution)	Toggle display of member results (only available after solution)	
	Wall Panel Label Toggle display of wall panel labeling options	Toggle display of wall panel labeling options	
	Diaphragm Display Toggle display of the diaphragms	Toggle display of the diaphragms	
	Show Deflection Diagram Toggle display of deflection diagram	Toggle display of deflection diagram	
	Distance Tool Initializes the Distance tool which can be used to measure the distance between two nodes	Initializes the Distance tool which can be used to measure the distance between two nodes (see the Status bar for the distance result)	F5

…in Spreadsheet View

Button	Title	Label in RISA-3D	Shortcut
	New Line Insert a new line before the current line	Insert a new line before the current line	F3
	Delete Line Delete the current line	Delete the current line	F4
	Repeat Line Repeat the current line	Repeat the current line	F8
	Sort	Sort based on values in the current column	F9
	Find	Find an item in the current spreadsheet	F5
	Fill Fill the currently marked block	Fill the currently marked block	Ctrl+F

	**Math**  Perform math on the currently marked block	Perform math on the currently marked block	Ctrl+M
	**Mark Lines**  Mark all the lines	Mark all the lines	
	**Delete Lines**  Delete the currently marked lines	Delete the currently marked lines	Ctrl+D
	**Unmark Lines**  Unmark all the lines	UnMark all the lines	Ctrl+L
	**Paste**  Paste from the clipboard	Paste from the clipboard	Ctrl+V
	**Save As Default**  Save the current data as the default	Save the current data as the default	
	**Help**  Help on the current window	Help for the current window	SHIFT+F1

These additional buttons appear after results are generated

	**Replace and Resolve**  Replace current shapes and then solve again	Only available while in the Suggested Shapes spreadsheet - Replace current shapes and then solve again	
	**Exclude After**  Exclude all results after the current item	Available in most Results spreadsheets - Exclude all results after the current item	
	**Unexclude**  Un-Exclude all previously excluded results	Available in most Results spreadsheets - Un-Exclude all previously excluded results	

Drawing Toolbar

Button	Title	Label in RISA-3D	Shortcut
	Draw New Members	Draw new members	
	Draw New Plates	Draw new plates	
	Draw New Wall Panels	Draw new wall panels	
	Draw New Solids	Draw new solids	
	Modify Boundary Conditions	Modify boundary conditions	
	Joint Loads or Moments Apply joint loads or moments to joints	Apply joint loads or moments to joints	
	Distributed Loads Apply distributed loads to members	Apply distributed loads to members	
	Point Loads or Moments Apply point loads or moments to members	Apply point loads or moments to members	
	Member Area Loads Apply member area loads	Apply member area loads	
	Surface Loads to Plates Apply surface loads to plates	Apply surface loads to plates	
	Surface Loads to Wall Panels Apply surface loads to wall panels	Apply surface loads to wall panels	
	Linear Translation Move Move via linear translation the currently selected part	Move via linear translation the currently selected part of the model	
	Rotational Move Move via rotation the currently selected part	Move via rotation the currently selected part of the model	
	Scale Factor Move Move via a scaling factor the locations for the currently selected joints.	Move via a scaling factor the locations for the currently selected joints	

	**Linear Offset Copy**  Copy the currently selected part using linear offsets	Make copies of the currently selected part of the model using linear offsets
	**Rotational Offset Copy**  Copy the currently selected part using rotational offset	Make copies of the currently selected part of the model using rotational offset
	**Mirror Image Copy**  Mirror image copy of the selected part	Make a mirror image copy of the currently selected part of the model
	**Model Merge**  Perform a model merge on all or part of the model	Perform a model merge on all or part of the model
	**Delete**  Delete parts of the model	Delete parts of the model
	**Modify Drawing Grid**  Modify the drawing grid and snap points	Modify the drawing grid and snap points
	**Drawing Grid**  Toggle the drawing grid on or off	Toggle the drawing grid on or off
	**Universal Snap Points**  Toggle the universal snap points on or off	Toggle the universal snap points on or off

Selection Toolbar

Button	Title	Label in RISA-3D	Shortcut
	Select All Select the entire model	Make the entire model selected	CTRL+A
	Box Select Draw a box around the part to be selected	Draw a box around the part of the model to be selected	
	Polygon Select Draw a polygon around the part to be selected (double click to end)	Draw a polygon around the part to be selected (double click to end)	
	Line Select Draw a line through the beams and plates to be selected	Draw a line through the beams and plates to be selected	
	Unselect All Unselect the entire model	Make the entire model UNselected	Ctrl+U
	Box Unselect Draw a box around the part to be unselected	Draw a box around the part of the model to be UNselected	
	Polygon Unselect Draw a polygon around the part to be unselected (double click to end)	Draw a polygon around the part to be unselected (double click to end)	
	Line Unselect Draw a line through the beams and plates to be unselected	Draw a line through the beams and plates to be UNselected	
	Invert Selected Invert the selected areas of the model	Invert the selected state of the model	Ctrl+I
	Criteria Selection Select or unselect based on other criteria	Select or unselect based on other criteria	
	Save/Recall Selection Save or recall selections of the model	Save or recall selection states for the model	

	**Lock Unselected** (unlocked) or (locked) Lock the unselected parts of the model	Lock the unselected part of the model	Ctrl+L