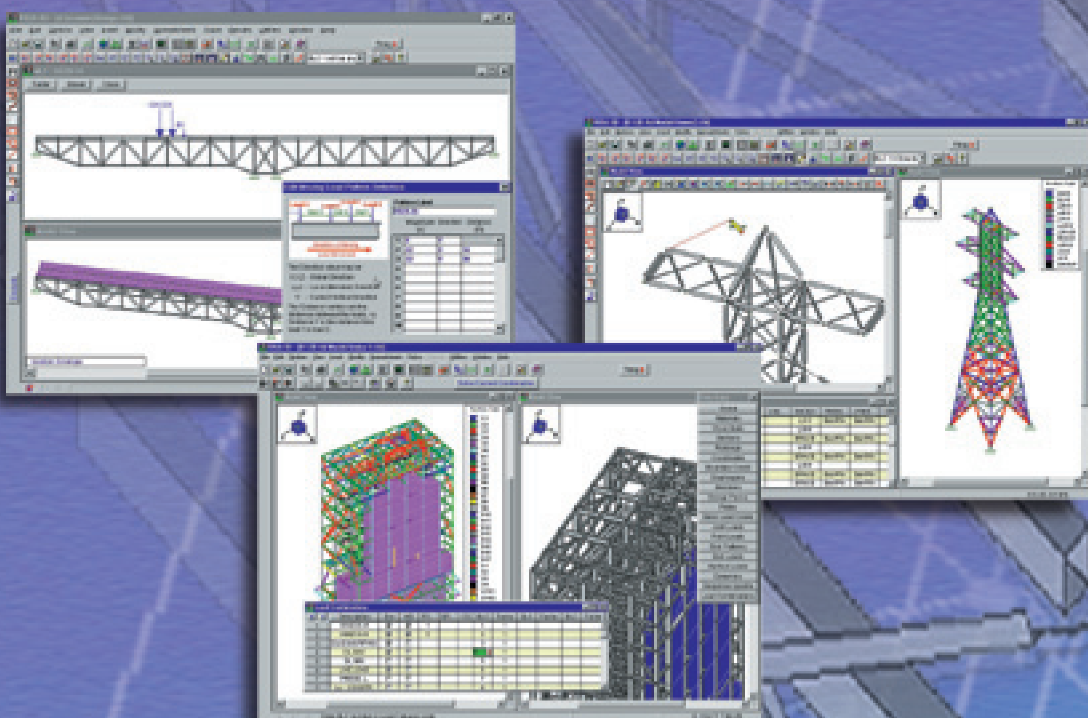
RISA TECHNOLOGIES

RISA-3D



WWW.RISATECH.COM

1.800.332.RISA

PRODUCTIVITY TOOLS THAT WORK FOR YOU.™

RISA-3D Demo

Rapid Interactive Structural Analysis – 3 Dimensional

Version 4.1 Demonstration Guide

RISA
TECHNOLOGIES

26212 Dimension Drive, Suite 200
Lake Forest, California 92630-7801

(949) 951-5815
(949) 951-5848 (FAX)

www.risatech.com

Installation

RISA-3D has been written specifically for 32 bit Windows operating systems. More specifically, RISA-3D has been written for Windows95/98/2000 and WindowsNT Version 4.0 (or later). There are no additional requirements; if your system is running Windows 95/98/2000/NT, you will be able to run RISA-3D.

To install the RISA-3D demonstration program please follow these instructions:

1) Put the RISA-3D demo disk 1 in your computers "**A**" (3.5inch) drive.

2) Click the Windows **Start** button and select **Run**.

3) In the Run dialog box type "**a:setup**" and then click the **OK** button.

4) Follow the on-screen instructions.

Uninstalling the Demo

Once you've finished running and evaluating the RISA-3D demonstration program you may wish to remove it from your computer. Here is how you do that:

Click **Settings** from the **Start** menu and then choose **Control Panel.** Double-click on the **Add/Remove Programs** icon to open the **Add/Remove Programs** dialog. Select **RISA-3D Demo** and click the **Add/Remove** button.

RISA-3D Demonstration Overview

Thank you for requesting the RISA-3D demonstration package!

The demo version allows you to enter and solve a model and this guide provides a tutorial that will walk you through the process from start to finish. Please allow about 3 hours to complete the demo tutorial in its entirety. This may seem like a lot of time but it should prove to be time well spent. Once you've completed the working demo you will be ready to use the full version of RISA-3D.

This demo program and manual are copyrighted to RISA Technologies, but you are free to copy and distribute both the disks and manual to anyone you wish, so long as no fee is associated with that distribution.

The RISA-3D demo program is licensed for use for demonstration purposes only. In no event should this demo program be used for professional design purposes.

RISA-3D Maintenance Policy

Program maintenance includes all upgrades to RISA-3D, defect notifications and discounts on new products.

The first year of program maintenance is included in the purchase price. After the first year, 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.

Complete program support is available to registered owners of RISA-3D and is included in the purchase price. This support is provided for the life of the program.

The "life of the program" is defined as the time period for which that version of the program is the current version. In other words, whenever a new version of RISA-3D is released, the life of the previous version is considered to be ended. RISA Technologies will support *only* the current version of RISA-3D.

Evaluating Software

This is not an easy task. Closely scrutinizing complicated features is no trivial matter. It requires a lot of time and effort and there aren't any shortcuts. Without manually verifying the results it is difficult to determine that things are being done correctly. The bottom line is that you need to use and verify software to know how good it is.

There are other considerations in addition to accuracy. We focus on making our software easy to use so that you and other engineers in your office can invest as little time as possible in getting up and running with RISA-3D.

All software seems to make similar claims so you be the final judge. Your expectations are the only measuring stick. We want you to put RISA-3D to the test and we provide the following tutorial to show you much of what RISA-3D can do. Should you have any questions you may call us and speak with a professional engineer.

Demonstration Limitations

This demo is a working program but certain limitations have been deliberately imposed. These include:

Only Five Sets of Section Properties - This demo limits you to only five sets of member section properties. Therefore any model you define with this demo can have no more than five different member sizes. The full version allows up to 500 different section property sets.

Shapes Database Modification Not Saved - Complete AISC, Trade ARBED and Canadian libraries of shapes are available for use. The demo version allows you to add shapes but will only keep the added shapes for the current session and will not save them for the next time that you run the demo. The full version allows you to permanently save any changes to the databases.

Model Size Limited to 100 Joints, 150 Members, 150 Plates - This demo limits you to no more than 100 nodes, 150 members and 150 plates. The full version allows up to 32000 nodes, 32000 members and 32000 plates.

TUTORIAL

The tutorial is intended to teach you how to use RISA-3D. It goes into a lot of detail on the program and is broken up into three parts so you don't have to do it all at once. After completing the working demo you will have a very good understanding of how RISA-3D works and will be ready to use the full version of RISA-3D.

We won't go into a lot of depth regarding the analytical aspects of RISA-3D here. For example, we'll perform steel and wood code checks in this tutorial, but we won't discuss the specifics of how those code checks are calculated; that's covered in excruciating detail in the full version manual.

Overview

Before we get started on the actual tutorial a general overview is in order. This is simply an introduction to the layout of the program and it's basic concepts. Don't worry if you don't fully understand the features and the process here because that is what the tutorial is for. This is simply a rough picture of where we are going. All the details will be explained in the tutorial sections that follow.

RISA-3D mainly has two complementary sets of tools; graphics based drawing tools and customized spreadsheets. We say "complementary" because these tools work together very closely. Everything that you do graphically is automatically recorded in the spreadsheets for you. You may view and edit these spreadsheets at any time. The converse is also true with few exceptions; what you enter and view in the spreadsheets may be viewed and edited graphically at any time. You will be thoroughly exposed to both sets of these tools in this tutorial.

Using computer software to perform structural design is essentially a three-step process. First, you define your model. Next you solve that model to obtain solution results. Last, you review those solution results to see how your model performed and determine if changes are needed. Usually you will need to repeat the cycle several times before you arrive at a final design. The purpose of RISA-3D is to speed up this process.

Defining the Model

Some data such as cross section properties, load combinations, oddball joint coordinates , etc. are most easily entered numerically, and the best tool for numeric input is a spreadsheet. Other data such as regularly spaced joints, member connectivity, regular loadings and wholesale changes are better handled using graphic drawing tools.

Typically you'll use the spreadsheets to define some preliminary information such as material and section properties. Then you will proceed to the drawing grid and graphically draw the bulk of your model and apply loads. RISA-3D allows you to specify drawing grids with equal and/or unequal increments to make drawing the model easier.

Other graphic edit options 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.

Keep in mind that almost all the model data can be edited either way (graphically or numerically in the spreadsheet). You can decide for yourself which method is preferable.

Solving the Model

Once you've got the model defined, you'll need to solve the model. RISA-3D features fast 32-bit solution speed, so your model solutions will be pretty quick. Once the model is solved you'll be presented with the results.

Reviewing the Results

Results review is similar to data editing in RISA-3D in that you can do it graphically and via the spreadsheet. All of the spreadsheet solution results (displacements, forces, stresses, etc.) will be listed in the **Results** menu. You can sort and filter these results as you are viewing them. For example you could go to the **Steel Code Check** results, sort based on code check values, then exclude all members with a code check value below 0.7, leaving results only for those members you're most interested in.

For the graphics, you can click on the **Plot Options** button and choose the results that you want to display. For example, you can specify that moment diagrams be displayed on all the members. You can also view rendered models, plate stress contours, animated deflections or mode shapes and much more.

Important Assumption!

The tutorial is written with the assumption that the RISA-3D demo has not been customized and is in the default, installed state. RISA-3D allows you to customize its settings so that it best suits your needs. If the installation of the RISA-3D demo that you are using has been customized then the options may not agree with those that are assumed here, which may lead to some confusion. You may reset the program defaults by selecting this option on the **Options** menu.

Conventions Used in the Tutorial

Action Items

This tutorial builds upon itself from start to finish and it is important that you do not miss a step and get off track. To help you avoid overlooking a step we have underlined the action items that call for you to actually do something on your computer. If you wish to work through the tutorial quickly you may scan for the action items and read the details that interest you.

Using the Mouse

For the purposes of this tutorial, when we say to "click" something with the mouse, we mean move the mouse such that the mouse pointer you see on the screen is on the object to be clicked and, once there, press and quickly release the left mouse button. From time to time we will want you to click the RIGHT button, and when we do we will expressly say to click the RIGHT button. If we say click something without mentioning which button, click the left button.

Using the Keyboard

During the tutorial, entries you are to type will be shown like this:

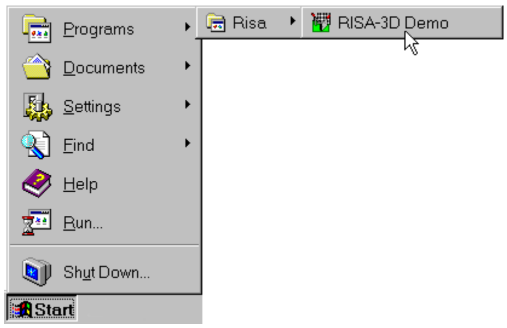
12 , 10 , ...

When you see an entry such as this, type it in exactly as listed, except *don't type the commas!* When something is listed in brackets ([ ]), it means press the particular key. For example, **[Enter]** means press the "Enter" key, **[Spacebar]** means press the space bar, etc. "**Alt-**" and "**Ctrl-**" mean press the listed key while holding down the Alt or Ctrl key. **[F1], [F2], ...** mean press the indicated function key at the top of the keyboard.

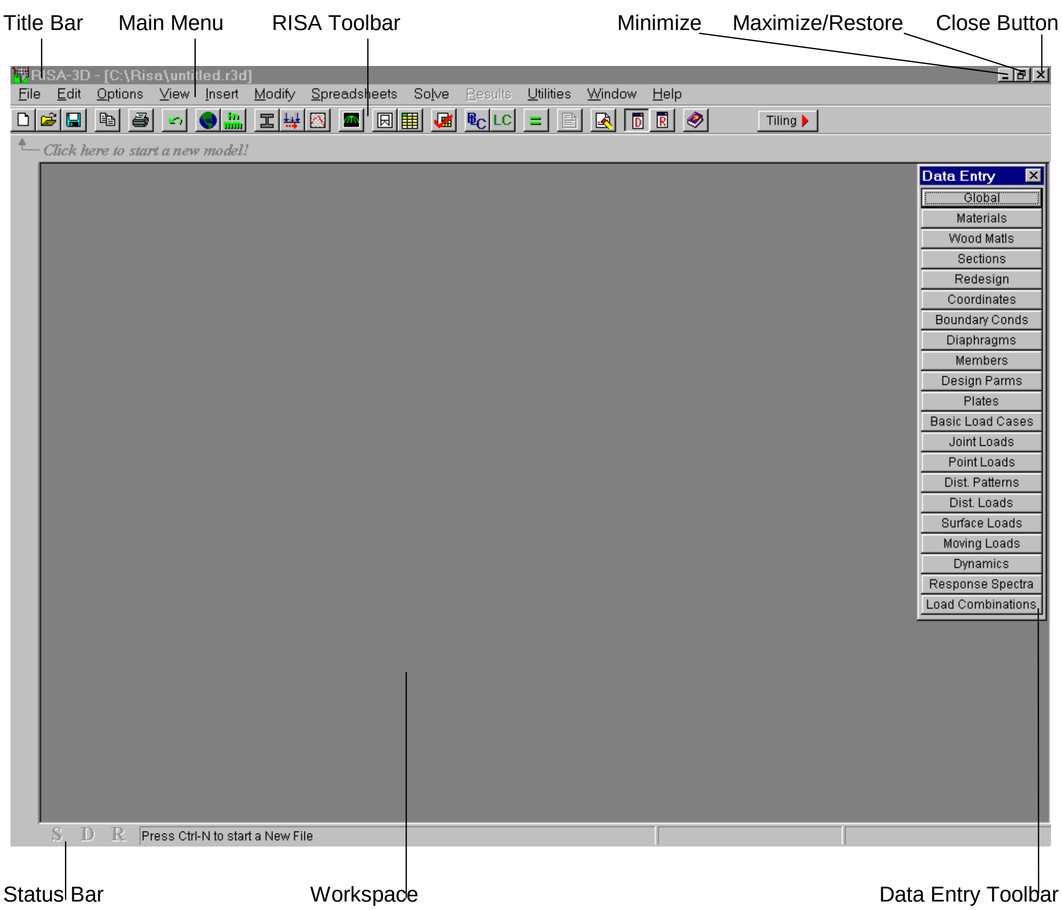
Starting the RISA-3D Demo

So lets take a quick first look at RISA-3D and then we'll begin the tutorial.

Start the program as shown below by clicking on the **Start** button and then choose **Programs** and then **RISA** and finally **RISA-3D Demo.**



The RISA-3D demo will open full screen as in the figure below.



The RISA-3D logo image in the center of the screen will disappear after a few seconds. Lets take a moment to explain what you have in front of you.

RISA-3D Screen

Title Bar

The bar along the top of the screen is called the title bar. It contains the name of the file that is currently open in RISA-3D. The three buttons on the far right side of the title bar are used to control the main window. The button will shrink RISA-3D to a button on the taskbar. The button will shrink the RISA-3D window on your screen. The button will close RISA-3D, prompting you to save changes if necessary. You will also see these buttons in other RISA-3D windows and they have basically the same functions there as well.

Caution!: We will not be using these buttons in the tutorial so do not click on them by accident. We will use similar buttons that will be in the spreadsheet titlebars and model view titlebars so be careful not to mistakenly click on these.

Main Menu

Just beneath the title bar is the RISA-3D main menu beginning with **File** on the far left and ending with **Help** on the far right. These menus provide access to all of the features RISA-3D has to offer. Clicking on each of these menus will display sub-menus that contain options that you may choose from. The toolbar buttons mentioned in the next section provide easy access to many of these menu options.

File Menu - The **File** menu helps you access file operations such as opening and saving, importing, exporting and appending files.

Edit – The **Edit** menu provides spreadsheet 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 operate current spreadsheet data.

Options – Program options may be set with this menu. The options include steel design code, program units, global axes orientation and the toolbars that you want to use.

View – Use the **View** menu to open a new model view or to adjust the current view.

Insert – The **Insert** menu is used to insert nodes, members, plates and loads into the model. All of these items may be input graphically or entered in the spreadsheets. This menu gives access to the graphical methods that RISA-3D provides and the **Spreadsheet** menu helps you access the spreadsheets.

Modify – The **Modify** menu is used to modify existing nodes, members, plates and loads. This menu also gives access to the graphical methods and the **Spreadsheet** menu helps you access the spreadsheets.

Spreadsheets – You may open any of the input spreadsheets from this menu.

Solve – Use this menu to access the analysis options provided with RISA-3D.

Results – All analysis result spreadsheets may be accessed from this menu.

Utilities – RISA-3D provides many utilities to help you organize, identify and correct problems as you model the structure.

Window – The **Window** menu can be used to manage all of the windows that you have open in RISA-3D whether they are spreadsheets or model views. Special tiling options are provided here that are geared to specific modeling tasks.

Help – Use this menu to access the extensive on-line help system provided with RISA-3D.

RISA Toolbar

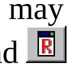
RISA-3D has toolbars with buttons to help you access common commands and popular options. All you need to note at this time is where the toolbar is. If at any time you are not sure what a particular button does, simply let your mouse hover over the button and a helpful tip will pop up and explain the button.

The RISA toolbar is the first horizontal row of buttons beneath the menu. These buttons perform general actions such as opening and closing files, changing design parameters, printing, and solving the model.

Data Entry Toolbar

The vertical toolbar on the right of your screen is the **Data Entry** toolbar. It looks a bit different because it has larger buttons with text on them, instead of pictures. This toolbar is to assist you in accessing the spreadsheets while building a model. The buttons are generally in the order in which you would typically build a model. We will use this toolbar frequently as we work through the tutorial.

A similar **Results** toolbar is presented after the model has been solved to help you access the result spreadsheets.

These toolbars may be turned on and off by clicking  and  on the **RISA Toolbar.**

Now is a good time to note that there are many ways to access the features available to you in RISA-3D and the method that you will use will simply be a matter of personal preference. The good news is that RISA-3D gives you the options. The tutorial will tend to lean to the use of toolbar buttons but be aware that the menu contains the same options and there are shortcut keys and hot keys that can't be beat in the interest of speed.

Workspace

The actual work that you do in RISA-3D will be in the main area on the screen, which we will call the workspace. Currently the workspace is blank (dark gray) except that it contains the floating **Data Entry Toolbar**.

Status Bar

The status bar across the bottom of the screen is to pass information to you as you work with RISA-3D. This 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. The letters are currently gray to indicate that none of these solutions have been performed. After performing a solution you will notice these letters become blue and a checkmark is placed in front of them.

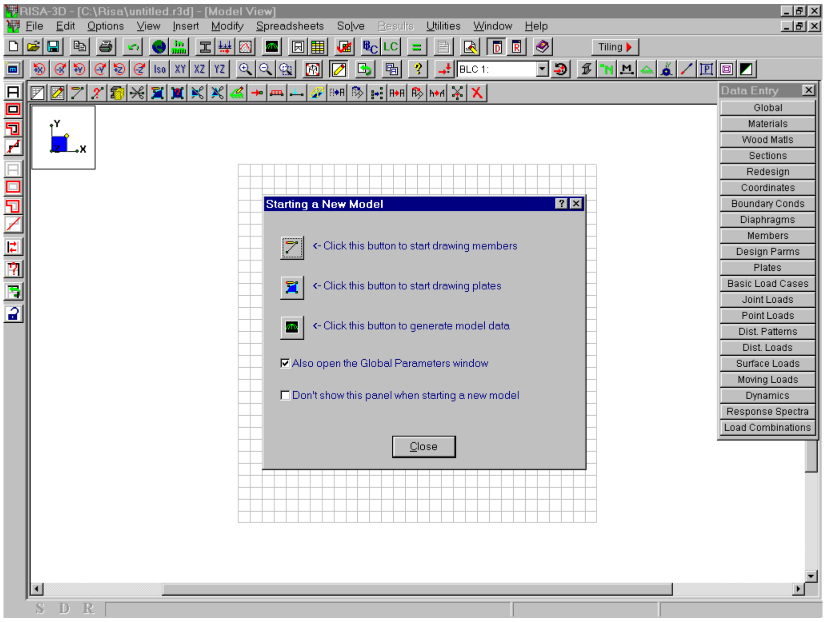
To the right of the solved status flags are three boxes which will pass you information while you work. These are called status boxes. The first status box is used to pass general information that is relevant to what you are currently doing. Currently the message is "Press Ctrl-N to start a new file". Look to this box for help as you need it.

The middle message box is used to report the units of the current spreadsheet cell. As you move from cell to cell look to the middle status box for the appropriate units. Currently the box is empty because we are not working in a spreadsheet.

The status box on the right is used to pass the mouse coordinates to you as you work in the model view. We will point this out again and use it to help us throughout the tutorial.

Starting a New File

Click on the **New File**  button in the upper left corner to start working on a new file.



Your screen should look like the figure above. A blank (white) model view is presented with a drawing grid. On top of the drawing grid is a dialog window to help you get started. Additional toolbars are provided to help you work with the model view. Let's explain these and then move on.

Window Toolbar

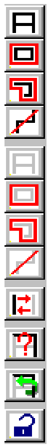
The Window Toolbar is the second horizontal toolbar. It gets its name because the buttons change as you move from window to window in order to help you with what you are currently doing. When you are working in a graphic model view the buttons above provide tools, such as rotate and zoom, to assist you with that view.

When you are working in a spreadsheet, this toolbar will contain the buttons similar to those below, which provide spreadsheet tools such as **Fill** and **Math**.

Drawing Toolbar

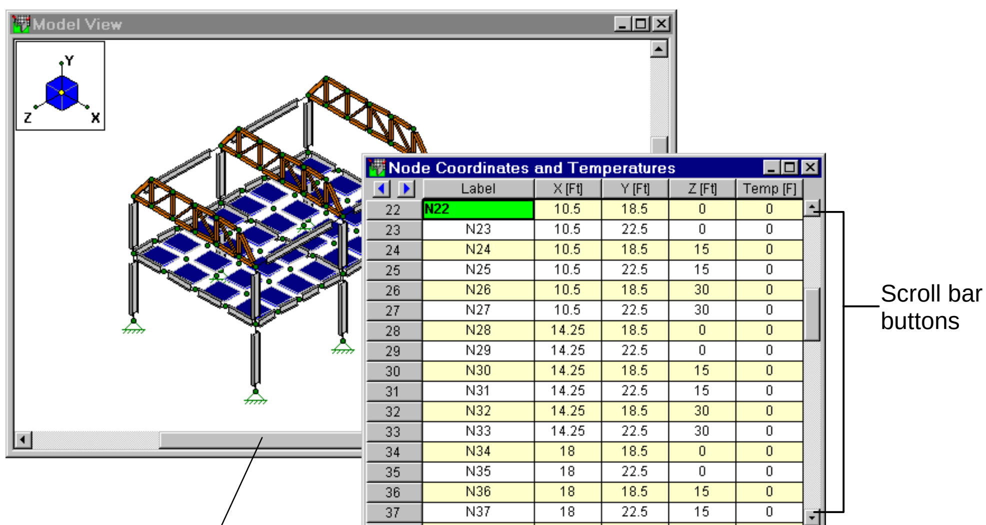
The Drawing toolbar is the third horizontal toolbar in the figure above. This toolbar may be turned on and off as needed to help you with graphic editing.

Selection Toolbar



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 make selections when you do things like graphically edit only part of the model or print only part of the results. This toolbar only works with the model view windows so if you are working in another window such as a spreadsheet, this toolbar will not be seen.

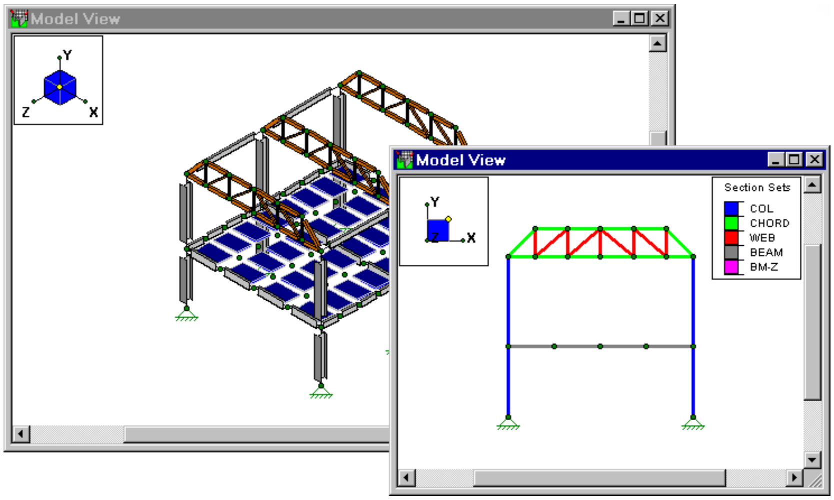
Windows



You will work within model views and spreadsheets, each in their own window that may be moved around the workspace and sized as you wish. A powerful feature of RISA-3D is the ability to have multiple model views and multiple spreadsheets open at one time. The options in the **Window** menu are provided to help you manage these windows.

Notice in the windows above, the same three buttons  are located in the upper right corner to help you minimize, maximize and close the window. There are also scroll bars to help you view information that is outside of the window viewing area. Click the scroll bar buttons or drag the scroll box to advance the display in one direction or another.

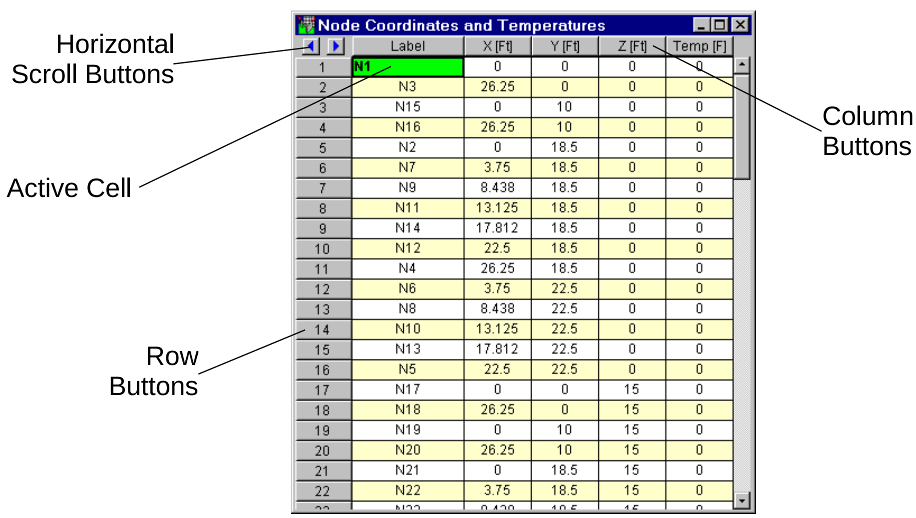
Model View Windows



You may open as many model view windows as you like. This is especially helpful when working in close 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 also have different information plotted in multiple views.

One thing to remember is that the toolbars that RISA-3D displays depends upon what window is active. The active window is the one with the colored titlebar. For example, if you are looking for the zoom toolbar button and the active window is a spreadsheet, you need to select a model view first before you can access the zooming tools.

Spreadsheet Windows

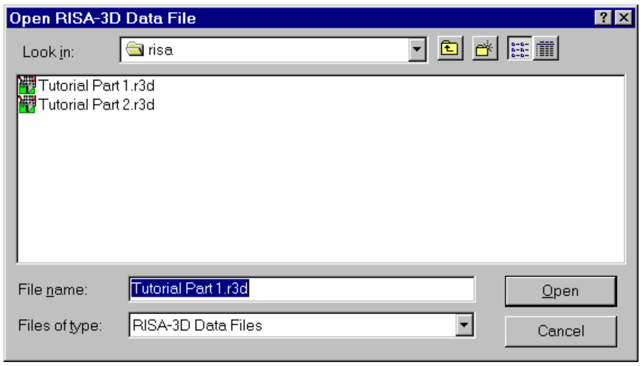


	Label	X [Ft]	Y [Ft]	Z [Ft]	Temp [F]
1	N1	0	0	0	0
2	N3	26.25	0	0	0
3	N15	0	10	0	0
4	N16	26.25	10	0	0
5	N2	0	18.5	0	0
6	N7	3.75	18.5	0	0
7	N9	8.438	18.5	0	0
8	N11	13.125	18.5	0	0
9	N14	17.812	18.5	0	0
10	N12	22.5	18.5	0	0
11	N4	26.25	18.5	0	0
12	N6	3.75	22.5	0	0
13	N8	8.438	22.5	0	0
14	N10	13.125	22.5	0	0
15	N13	17.812	22.5	0	0
16	N5	22.5	22.5	0	0
17	N17	0	0	15	0
18	N18	26.25	0	15	0
19	N19	0	10	15	0
20	N20	26.25	10	15	0
21	N21	0	18.5	15	0
22	N22	3.75	18.5	15	0

Spreadsheets are made up of rows and columns of data cells. If you wish to add or edit data in a spreadsheet cell you click on the cell, making it the “active” cell, and then edit the cell. There is always one and only one active cell, which is the cell that has the “attention” of the keyboard. This “active cell” is simply the green cell that moves around the spreadsheet as you hit the cursor keys (← ,→), Page Up, Page Down, Home, End, etc.

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

Dialog Windows



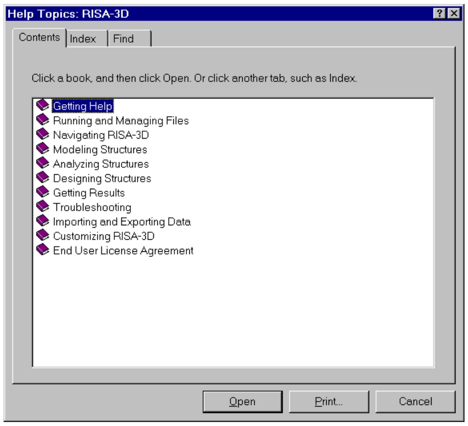
Dialog windows are windows that help you perform a specific function within the program. For example if you were to select **Open** from the **File** menu you will be presented with the dialog box above, which helps you find the file you wish to open. You will find that dialogs are very easy to work with. There are **Help** buttons that will bring you directly to the relevant topic in the help file. You may also click on the ? button in the titlebar, and then click on any item in the dialog to get help for that item.

Getting Help

RISA-3D offers a variety of ways to find help with the program in general or with a specific feature. There are three types of help available within RISA-3D: General Help Topics, Window Help and "What's This?".

Help Topics

Use the toolbar button to open the three-tabbed help window below and search on your own. The first tab, **Contents**, displays the list of major content areas that you may choose from by double clicking on them. The **Index** tab allows you to search for help topics based on keywords. The third tab, **Find**, is a more detailed search that allows you to search the actual text of the topics to locate the keywords that you enter.



Once you have a topic open there are a number of options available to you that make Help even more useful. Click the menu options to do such things as annotate the file with your own notes or place a bookmark.

Window Help

Window help accesses the above help topics for you. Use the toolbar button or the Help button in dialog boxes to open the appropriate help file topic that is relevant to what you are currently working on.

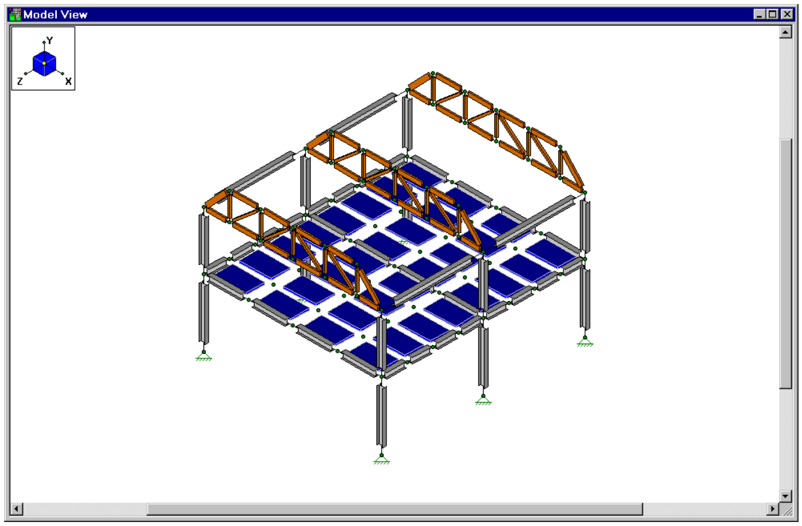
"What's This?"

"What's This?" is a Help option that lets you point to an item in a dialog box and find a brief definition of the item. Click the button on the right side of the dialog title bar and the mouse cursor changes to allow you to then click on an item in the dialog. A brief description of the item will be displayed. For toolbars you may simply let your mouse hover over the button to display a description of the button.

As you proceed through this tutorial, you may wish to review the help information for any windows you have questions about.

What's Ahead

For this tutorial we will be designing this frame:



In part one of the tutorial we will create a 2-D frame comprised of two steel columns supporting a wood truss. We'll make use of both the spreadsheet and graphic drawing capabilities of RISA-3D. In part two we'll "extrude" this into a 3D model, add plates and solve the model. Part three will demonstrate seismic analysis and design.

This tutorial is intended as a "real world" design example. This means we won't just enter the data once, solve the model and miraculously have a completed design. Just as in the real world, we'll enter the data, solve the model, review results, go back and make changes, etc. Any program can look good for a "one time through, no mistakes" data entry procedure; the *real* test is how easily can you *change* your model!

RISA-3D Tutorial, Part 1

So we have started RISA-3D and clicked the **New File** button. We should note here that your picture might not look exactly like the screen above due to the fact that your computer could be set up differently. Your screen resolution and font sizes can change the appearance of the menu and toolbars. This is simply cosmetic and should not affect your ability to go through this tutorial.

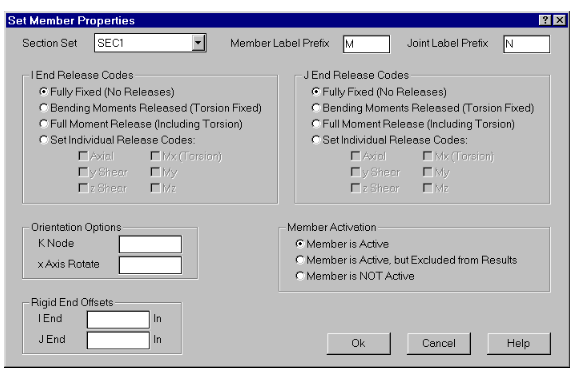
New Models

Clicking the **New File** button or selecting **New** from the **File** menu produce the same results, they present you with a clean slate with some initial options. A blank drawing grid is placed in the workspace and you are given options in a dialog box with the title **Starting a New Model.** You can choose to start drawing your model, generate it automatically, or click **Cancel** and work on your own. Notice that a box is checked that reads “Also open the Global Parameters window.”. We will see what this means in just a minute.

Drawing Members

Click the button to start drawing members.

You should now see a dialog box with the title **Set Member Properties**, which is shown below.



This panel is used to define the parameters for the members to be drawn such as what section set to use, end releases and member end offsets.

The “Section Set” field is simply the label that refers to the cross-sectional properties to be used for the members. We will define these properties in a minute.

To the right of the Section Set there are some labeling options. As you work, RISA-3D automatically labels your nodes, members and plates. Unless you specify otherwise the members will be labeled M1, M2 etc. and the nodes will be labeled N1, N2 etc. You can even change as you go so that say the nodes on level one of the structure have the labels N1, N2 etc. and those on level two might have N100, N101 etc. This can help later with understanding and sorting the results.

Member Releases

Beneath the Section Set are the member's end-releases. The end releases are used to designate whether the forces and moments at the ends of the member are considered fixed to or released from the member's points of attachment (the I and J nodes). Each member has 6 force components at each end (these are: axial, y-y & z-z shear, torque, y-y & z-z bending). Any or all of these force components can be "released" from the member's point of attachment by setting the individual release codes. If a force component is released, that force is not transferred either way, either from the attachment point into the member or from the member to the attachment point.

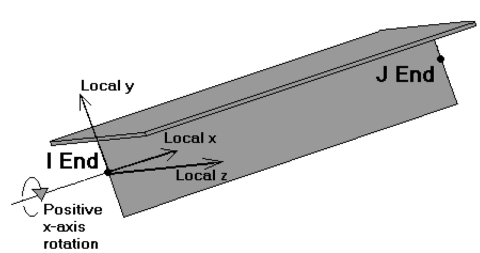
You may set the end condition either by choosing from the two standard choices or by setting them yourself. These standard choices are included because 99% of the release configurations you'll ever want to define will be one of these. For example a truss member would not be able to transfer moment through its ends so you might choose the bending moment release option for both the I-end and the J-end.

Member Releases vs. Boundary Conditions

A quick note on member releases and boundary conditions: You should use member releases to model how members are attached to each other, for example how beams are connected to columns or other beams. Use boundary conditions to model how the structure is attached to its external points of support, for example how the columns are attached to the foundation (pinned in this model). We'll be discussing boundary conditions a little later.

Member Orientation

To discuss member orientations we need to understand the member's local axes. Please review this diagram:

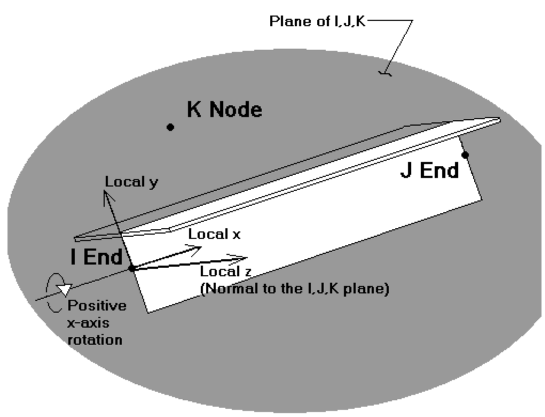


As can be seen from the diagram, the local x-axis corresponds to the member centerline. The positive direction is from the I-node towards the J-node. The complicated part is defining the orientation for the local y and z axes. Of course, we only have to define the direction for one of these two (y and z) axes. The third axis direction is based on the directions of the first two.

If you do not explicitly define the orientation for a member, the default is for the member's local z-axis to lie in the global X-Z plane, or as near as possible. If the member is defined in the global Y direction, the member's local y and z axes both lie in the global X-Z plane, so the local z-axis is made parallel to the global Z-axis.

Note that global axes are referred to with the capital X, Y and Z and that local axes are referred to with the lower case letters x, y and z.

RISA-3D provides two ways to explicitly set the orientation of the y-axis. The first is by defining a K-node for the member. If a K-node is defined, the three nodes (I, J, K) entered for the member are used to define the plane of the member's x and y axes. The z-axis is defined based on the right hand rule. See the following diagram:



The second way to explicitly define the orientation is by rotating the member about the local x-axis. Positive rotation is counter-clockwise about the x-axis, with the x-axis pointing towards you.

The dialog has fields for a K-node entry and an "x-Axis Rotate" entry. We can use either or both of these to define the orientation of the members we will be drawing.

We will discuss member offsets when we use them later in the tutorial.

Inactive Members

Above the **OK** button are the activation options for the members that will be drawn. Active members are part of the model solution and results. The second option will provide members that are part of the model solution but exclude the member from the results in order to simplify output.

Inactive members will not be included when the model is solved or plotted. This is handy if you want to try a frame with and then without certain members, without having to actually delete the member data.

Common Dialog Buttons

The buttons at the bottom right of the **Set Member Properties** dialog box are standard buttons and will appear in most of the dialog boxes that you work with. The **OK** button simply accepts and stores all of the changes that you have made in the box and lets you proceed, in this case, drawing. The **Cancel** button will close the box as if you have never opened it. Changes made will not be stored and, again for this case, you will not be in drawing mode. The **Help** button and the **What's This?** button access the help options that we mentioned in the previous section. Feel free to try them both out at this time. If you open the help file you may close it by clicking the button in its title bar.

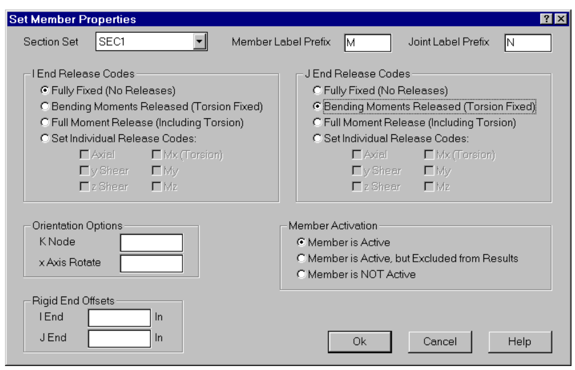
Column Members

Now that we've explained all that, let's fill in this dialog and start drawing the columns. We will assign them to the "SEC1" section set and we won't change the member labeling.

We do, however, want our columns to be pinned at the top so we will have to change the J End Release Code.

Select the second option under the **J End Release Code,** which is **"Bending Moments Released (Torsion Fixed)"**.

We will leave everything else the way it is so the dialog should look like this:



Click **OK** and a second dialog opens for the **Global Parameters.**

Global Parameters

This window is used to enter information that applies to the problem as a whole, I.e. global information. It is opened automatically because of this checked box in the **Starting a New Model** dialog that we saw earlier:

Click in the first field, labeled **Model Title** so that a flashing cursor appears there. Now type:

Tutorial Problem, [TAB]

***(Type your name)*, [TAB]**

J1000

Notice that you hit the TAB key to move from field to field. The mouse may also be used to go directly to a field. Just put the mouse pointer on the desired field and click.

The next 3 boxes control steel code checking. Currently the **Steel Code** choices are **9th edition ASD, 2nd Edition LRFD,** the **Canadian steel code**, or **"None"** for no steel design. We will use the ASD specification for the tutorial, which is already selected. The Allowable Stress Increase Factor defaults to 1.333, reflecting the 1/3 increase allowed by the 9th edition ASD for transient loads (wind, seismic, etc.). The checked box below indicates you want RISA-3D to **Do Redesign** for you as part of the steel design.

The **Number of Sections** field is where we specify how many locations along each member are to be used for the reporting of forces, stresses, and deflections. We will have the calculations reported at 7 section locations.

Click the up arrow two times to increase the number from "5" to "7".

The checked **Include Shear Deformation** box indicates shear deformation considerations are to be included in the model solution. This will almost always be checked (the default). For more information on this see the help file.

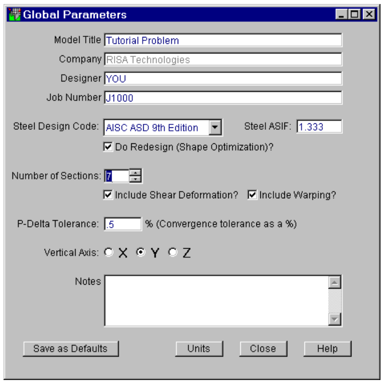
The **Include Warping** field indicates whether you want RISA-3D to consider torsional warping effects when calculating stiffness and stress values for wide flange and channel shapes. For more information on this see the help file.

Moving down, the **P-Delta Tolerance** is used to set the convergence tolerance for the P-Delta analysis. Leave this set to 0.5%.

The **Vertical Axis** option is used to indicate which of the three global axes (X, Y or Z) is to be considered the vertical axis. For this model, leave this set to "Y"; indicating the global Y-axis is the vertical axis.

Any notes that you would like to keep with the model may be typed in the **Notes** area.

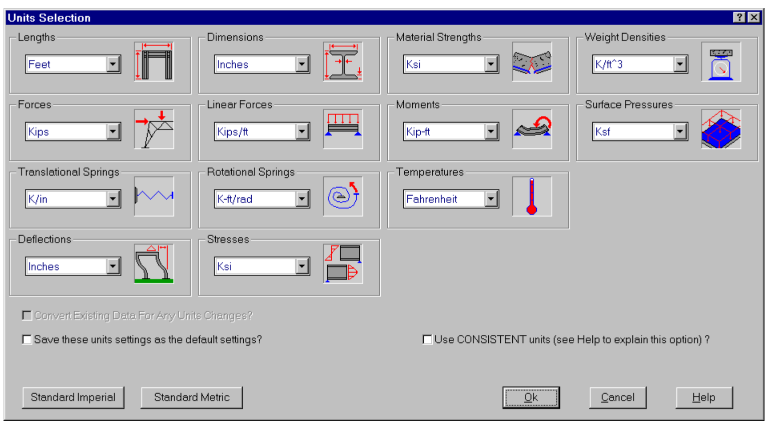
The **Global Parameters** dialog should now look like this:



Before we close this window we'll look at the buttons at the bottom. On the left is a button that you will see in a few different places, "Save as Defaults". If we were to click it now all of the changes we have made in the **Global Parameters** window would become the defaults so that when we start a new model they are already filled in.

Now click the **Units** button.

You will see this dialog that allows you to change to and convert existing data to different units:



These may be changed at any time in the modeling process. You may click on any of the down arrows to look at the options for parameters. Again the "Save as Defaults" option at the bottom of the dialog may be used to make any changes more permanent. You may also quickly apply two standard units settings by selecting **Standard Imperial** or **Standard Metric** options.

Click **Cancel** to close the **Units** dialog. Now close the **Global Parameters** window by clicking **Close**.

Drawing Grid

RISA-3D will open with a 30 x 30 grid spaced at one foot. This would work with our model but lets change it just to show how it is done.

Click on the Drawing Toolbar to redefine the grid.

RISA-3D employs a “point to point” drawing approach, i.e. when drawing a member, you click on one discrete point as the start location, and then click on another point as the end location. These “points” you click on can be either grid points (whose locations will be defined on this dialog) or nodes that already exist.

The drawing grid may originate at any location and may be set in any of the three global planes (XY, YZ, XZ). For our model we will leave the drawing grid in the XY plane and it’s origin at the global axes origin. We will also leave the grid as a grid of lines rather than the other option of points.

To make our change, double-click on the current entry of “30@1”in the first box in the X-Axis column. This should highlight the entire entry in blue.

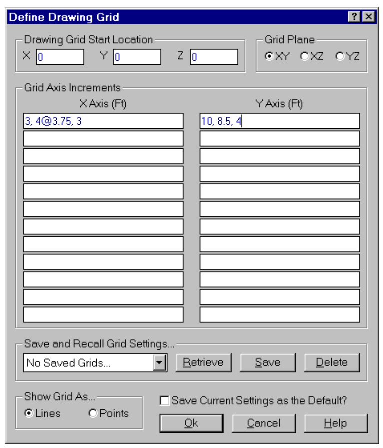
Now type:

3, 4@3.75, 3

Now double-click on the current Y-Axis entry of “30@1” and type:

10, 8.5, 4

The dialog should now look like this:



What we’ve done is define a drawing grid with an initial X-increment of 3 ft followed by 4 equal increments of 3.75 ft each and a final increment of 3 ft. Note the use of the “@” symbol to define a series of equal increments.

Our Y increments are 10 ft, 8.5 ft and 4 ft. Notice how RISA-3D allows you to define your drawing grid to match your model, using series of equal and unequal increments in both directions as needed. These increments are all offset from the Grid Start Location which we let default to 0., 0, 0.

Other things to notice here are that you may save this grid layout and any other for later recall. You may also specify that the layout should be the default grid layout each time the program starts up.

As a side note, if you have a few joints with odd locations such as 10.72, 9.635, etc., it’s sometimes better to define them in the spreadsheet by typing in the coordinates directly rather than trying to define a drawing grid that includes them.

Ok, lets return to the model view and start drawing!

Click **OK,** which closes the **Define Drawing Grid** dialog. Click the **Close** button at the top of the toolbar to the right of the title "**Data Entry**".

Notice that your mouse cursor has changed to and the **Draw Members** button on the **Drawing Toolbar** is "down". We are ready to draw.

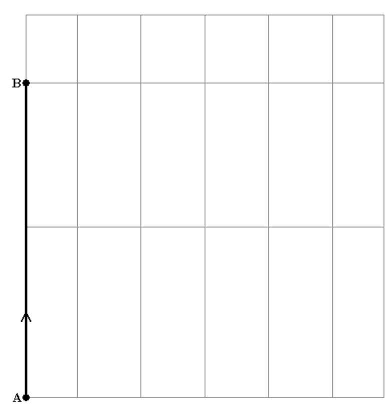
Start Drawing

Once drawing is activated RISA-3D displays the coordinate location of the mouse in the lower right corner of the model (you'll see this when you start moving the mouse). These coordinates are actually the coordinates of the graphic editing point closest to the mouse (either a grid point or a joint location); you'll see them change as you move the mouse around.

It's a bit cumbersome to dictate the drawing that will be performed next, so we will provide sketches showing what is to be drawn, as well as dictating the required operations in terms of grid coordinates, for instance we'll say "move to grid location such and such and click". A "grid location" is any point where grid lines are intersecting.

Watch the coordinates display at the bottom right to make sure you're moving to the right location. For your model to be the same as that in this tutorial it is important to draw just as it is described in here so that member labels and orientations will agree. Also, be sure to move *all the way to the specified grid location!* The coordinate display just shows the point you're closest to; you have to make sure the mouse is directly on the grid location (the point where the grid lines are crossing) before you click.

We will start by drawing a line from point A to point B as shown here:

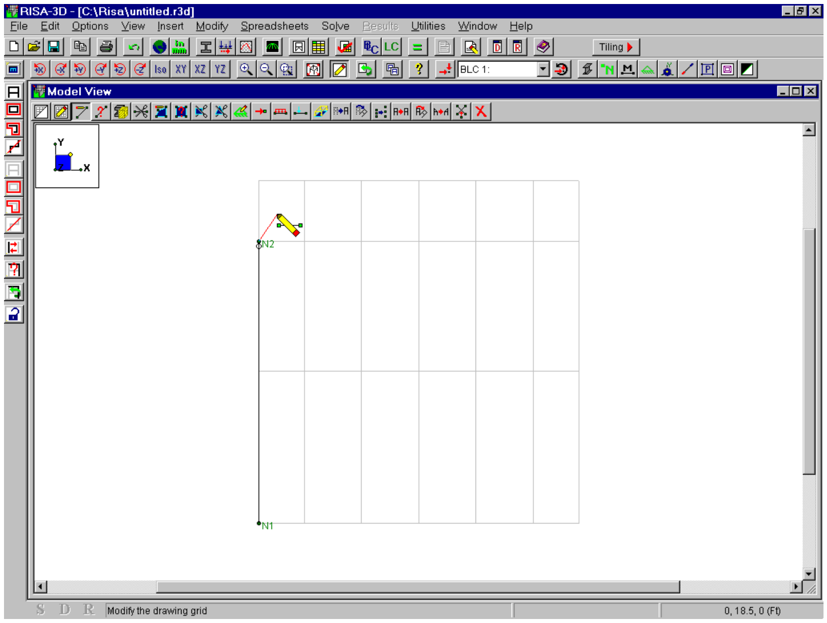


Move the mouse to the lower left corner of the grid (point A, location 0,0,0) and click once.

You should now have a line drawn from that point to the mouse, this line follows the mouse as you move it. Also a node has been created and labeled N1. If you don't see this, it means you weren't very close to the lower left grid point when you clicked.

Now move straight up to point B, grid location (0,18.5,0) and click again.

The line that follows the mouse should now be anchored to this new grid point, node N2, and your screen should now look like this (note the pin release at the top of the column):

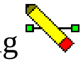


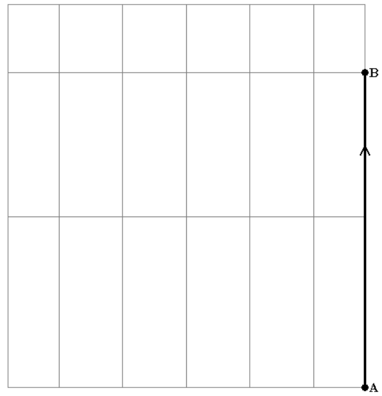
If your screen *doesn't* look like this, again it means you probably clicked when the mouse wasn't directly on the correct grid point. (Sorry we're being so picky about this, but the sequence of the tutorial depends upon the model being drawn correctly, so we want to make sure everything is in order).

Now is a good time to point out the **Undo** button  on the **RISA Toolbar**, which will undo any mistakes that you make. Click it as many times as necessary.

Let's continue with our drawing. The next line we draw will be from A to B as shown in the next figure. Since the second column is not attached to the member we have already drawn we need to "pick up" our pencil and move to the other side of the drawing grid to resume drawing.

Click the RIGHT mouse button and your mouse cursor should no longer have a member trailing behind it.

We are still in drawing mode as indicated by the drawing  cursor.



Move the mouse to the lower, right grid point (A) at coordinate (21,0,0) and click. Now move straight up to the grid point (B) at coordinate location (21,18.5,0) and click again.

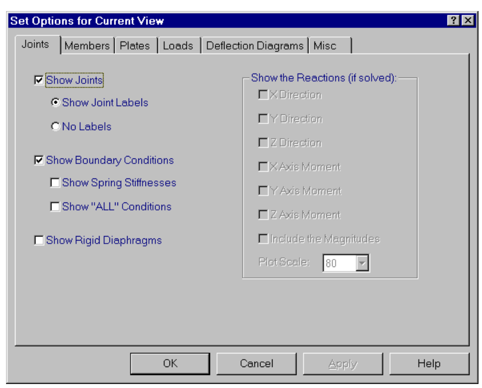
We now have a column on the left from node N1 to node N2, and another on the right from node N3 to N4.

Click on the **Draw Members**  button in the **Drawing Toolbar** to stop drawing.

This will pop the button back "up" and the cursor becomes the standard arrow .

It's a little difficult to see the members since the drawing grid is displayed with lines as well, so let's do something about that. We'll tell RISA-3D to draw our members using thicker, color-coded lines.

Click on the **Set Plot Options** button . It is the first button on the **Window Toolbar** (the second horizontal toolbar).

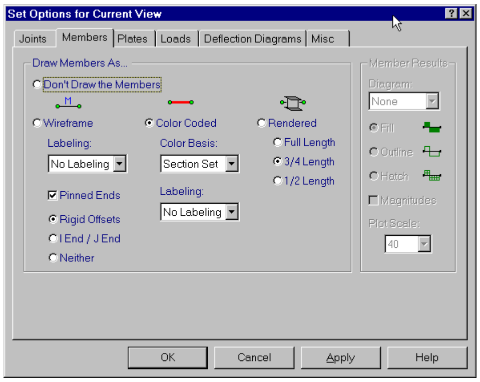


We'll discuss the plot options in more detail later.

For now, click on the second tab at the top, which is labeled **Members.** Now click on the circular button next to the words **"Color Coded"** and a black dot will indicate you have made this choice.

Beneath this the first indented option is **"Color Basis"** and **"Section Set"** is already selected. We have now told RISA-3D to color code the members by their section sets.

Now the dialog box looks like this:

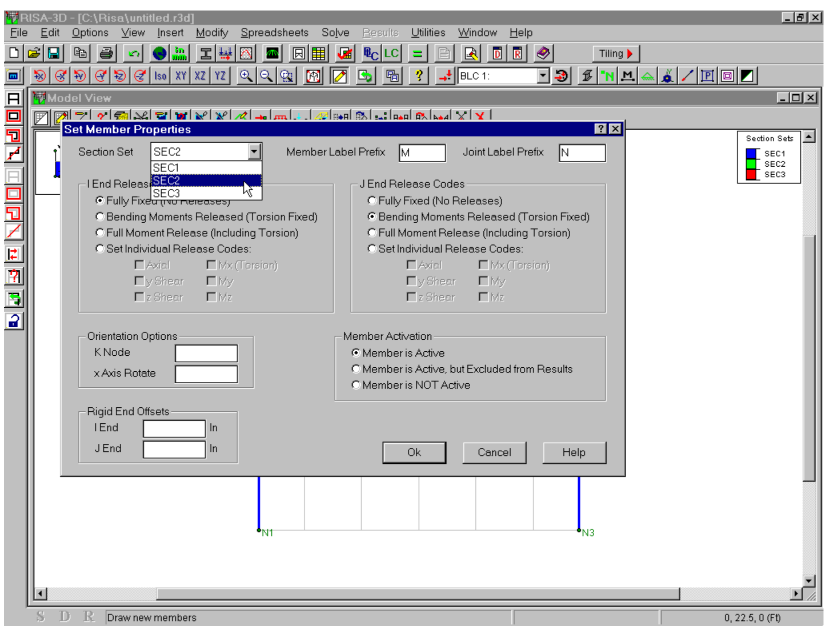


Now click on **OK** to close the **Set Plot Options** dialog and go back to the plot.

The members should now be displayed in color. The legend is displayed in the upper right hand corner of the model view to tell you what the colors mean. We haven't used "SEC2" and "SEC3" yet but that is about to change.

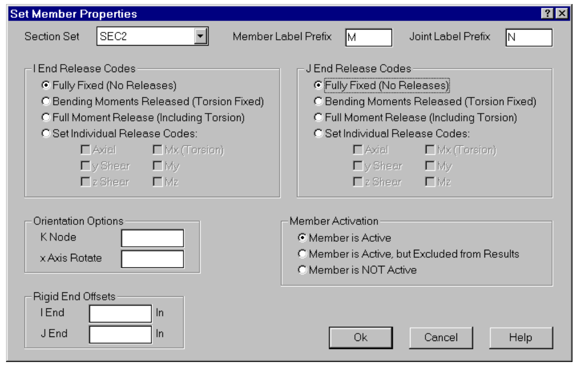
Our columns are defined, now lets do the wood truss. Since these members are going to be different we will use a different section set.

Click on the button to start drawing again and we see the **Set Member Properties** dialog again. Change the **Section Set** to "SEC2" by clicking the down arrow and choosing "SEC2" from the drop-down list as shown here:



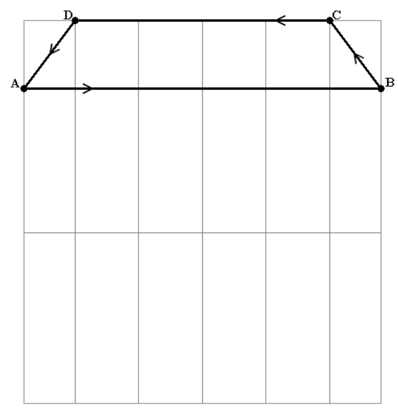
We will make both the ends of the member fixed so click on the **"Fully Fixed"** option for the **J End Release Code** .

The dialog should look like this:



Click **OK** to start drawing again and we are back at the model view.

This sketch shows what you'll draw next; lines from A to B, then B to C, then C to D and lastly from D to A:



To begin drawing the truss chord, move the mouse to the top of the left side column at node N2 (point A) at grid location (0,18.5,0) and click. Now move to the right to the top of the right side column at node N4 (point B), grid location (21,18.5,0), and click again.

A new member should appear, and the mouse should still have a line connected to it from the end point of the new member. This enables you to easily continue drawing new members.

Notice that the truss chords are being drawn with a different color, which is due to the different section set we are using.

Move up and to the left to grid location (18,22.5,0) (point C) and click again.

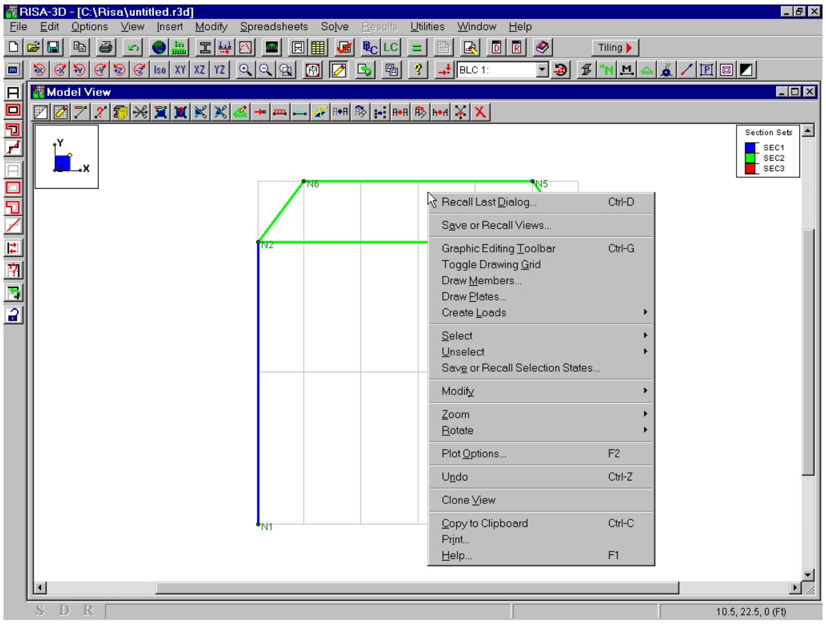
Now move left to grid location (3,22.5,0) (point D) and click again.

Finally, move down and to the left to grid location (0,18.5,0) (point A) and click once more. "Pick up" your pencil by clicking the RIGHT mouse button. Click the RIGHT mouse button again to stop drawing members.

The truss chord members are now defined. Next we'll do the web members and again we will use a different section set because the web members will have different properties.

To get back to the **Set Member Properties** dialog box we can use a specially tailored menu called the shortcut menu. This menu is accessed by clicking the RIGHT mouse button anywhere in the window area and is window specific. This means that RISA-3D presents you with a menu that is relevant to what you are currently doing.

Click the RIGHT mouse button somewhere in the middle of your screen and you will see this shortcut menu:



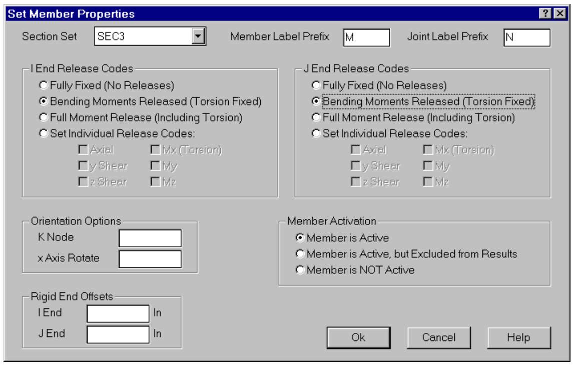
This menu brings relevant options right to where you are currently working. Notice that all of the options have to do with the model view that we are currently working with. If we were working in a spreadsheet the choices would be relevant to that spreadsheet.

Note that the first option is called **Recall Last Dialog** and note also that next to it is "Ctrl-D". This just means that executing the key combination Ctrl-D is the same as choosing that option. We will use Ctrl-D later.

Now with the left mouse button click on **Recall Last Dialog** and the **Set Member Properties** dialog is opened.

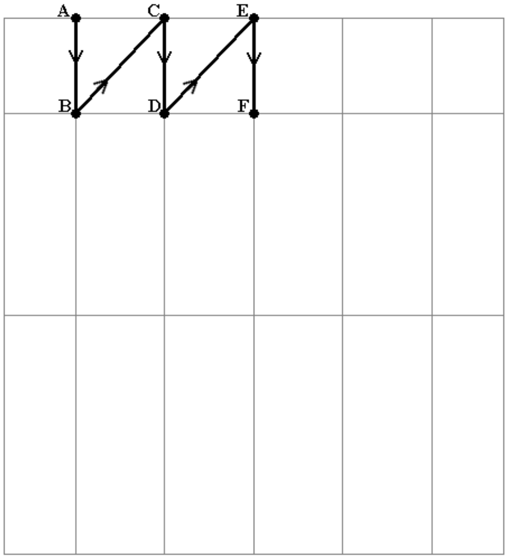
Change the Section Set by choosing "SEC3" from the drop down list. The web members will be pinned at both ends so click on "**Bending Moments Released**" for both the I-end and the J-end.

The dialog now looks like this:



Now click **OK**

This sketch shows what will be drawn next; lines from A to B, then to C, etc. ending at point F:



Move to point A, grid location (3, 22.5, 0), and click.

Now move down to point B, grid location (3, 18.5, 0), and click.

Move up and to the right to point C, grid location (6.75, 22.5, 0), and click.

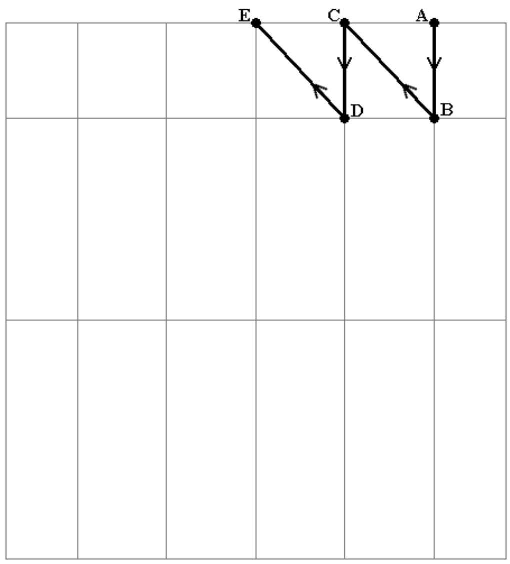
Next move down to point D, grid location (6.75, 18.5, 0), and click.

Move up and to the right to point E, grid location (10.5, 22.5, 0), and click.

Now move down to point F, grid location (10.5, 18.5, 0), and click.

RIGHT click to pick up the pencil again so we can choose a new starting point for the next series of members.

This sketch shows what will be drawn next, lines from A to B, to C, etc. ending at point E:



Move to point A, grid location (18, 22.5, 0), and click.

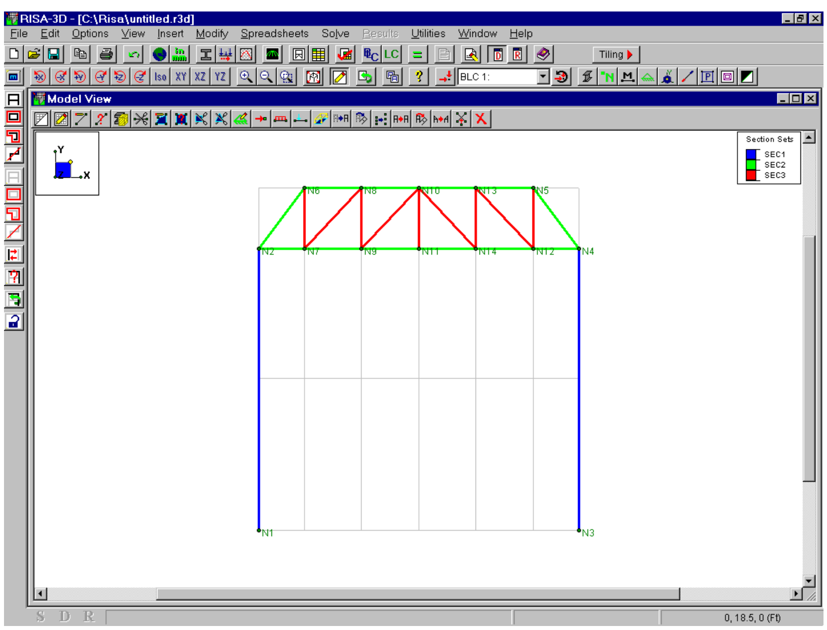
Now move down to point B, grid location (18, 18.5, 0), and click.

Next move up and left to point C, grid location (14.25, 22.5, 0), and click.

Move down to point D, grid location (14.25, 18.5, 0), and click.

Finally, move up and left to point E, grid location (10.5, 22.5, 0), and click.

RIGHT click to pick up the pencil again and the screen now looks like this:



So our geometry is almost complete. We only need to define the crossbeam at the 10ft level and we'll be done.

RISA-3D starts you off with three section sets. We will have to create a fourth one for the crossbeam. (Later you will learn how to specify the information RISA-3D starts new files with using the **Save as Defaults** feature.)

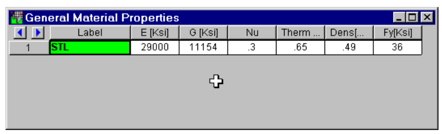
Creating new sections is accomplished in the **Sections** spreadsheet**.** While we are there we will define the section properties for all the members that we have defined thus far. We have not defined any material properties either, which will be done in the **Material** spreadsheet. We could have defined all of this before we started drawing but since we didn't we'll do it now.

Click  on the **RISA Toolbar** to bring back the **Data Entry Toolbar.**

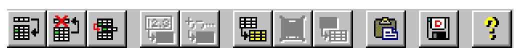
Material Properties

On the **Data Entry Toolbar** click on 

and this window will open:



You should also notice that the **Window Toolbar** has changed. It now looks like this:



Remember that this toolbar is geared towards what you are currently doing. These buttons will help edit the spreadsheet. Also, notice that the vertical **Selection Toolbar** is gone because the active window is a spreadsheet, not a model view.

In RISA-3D, material properties may be entered on two separate spreadsheets; this spreadsheet and the **Wood Materials** spreadsheet. The **Materials** spreadsheet is for general material properties while the **Wood Materials** spreadsheet is specifically for wood properties. The reason for this is simple. Unlike general material properties, wood material properties are closely tied to the cross-sectional properties of the section that you are using.

You can see the default properties that come up on the **Materials** spreadsheet are for A36 steel, and that's what we will use for our steel members, so nothing needs to be changed.

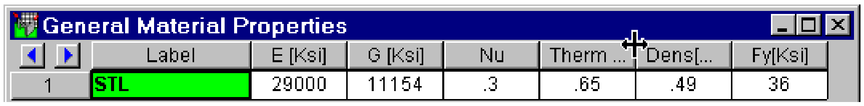
Save as Defaults

A few quick notes about spreadsheets in general and then we will move on. You may save any changes to this and certain other spreadsheets as the default values. If, for instance, you always use 50ksi steel you can make this change and save it by clicking the  button on the **Window Toolbar**. New files will then start with 50ksi steel automatically.

Adjusting Spreadsheets

You may notice that some of the column headings in the spreadsheet are truncated with the "…" symbol because they are too long to display in the column width. You may look to the status bar for a full explanation of the current entry. The units will be displayed when appropriate. You can also adjust the column widths to view the headings.

To increase or decrease the width of a column, drag the line to the right of the column heading. To do this let your mouse hover over the line until the cursor changes as shown below:



Once you get this double-arrow cursor you may click and drag the column width.

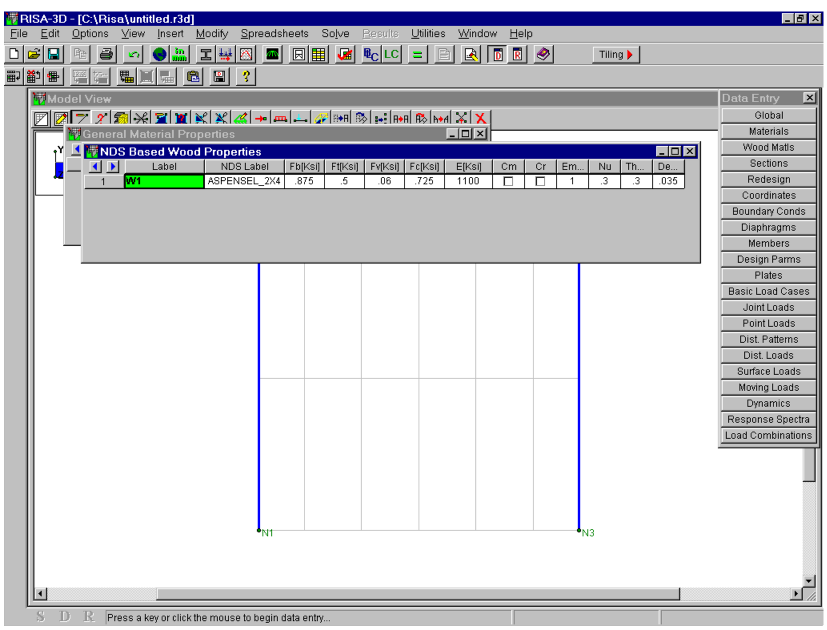
You should also note the two blue arrows in the upper left corner of the spreadsheet. You may use these cursors to move the active cell to the left or right. These are especially useful when you want to move to spreadsheet columns that are not displayed in the window viewing area. To view rows that are not in the viewing area use the scroll bars as mentioned previously.

Wood Properties

Click Wood Matls to open the **Wood Materials** spreadsheet. Create the first line of data by pressing:

[Enter]

Your screen should now look like this:



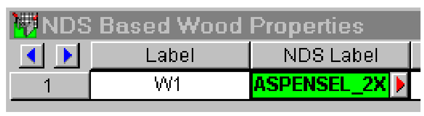
Wood gets it's own spreadsheet in RISA-3D. This is because the material properties used for wood are based on the species, grade and size, unlike steel which has the same properties no matter what the shape. The complete NDS species database is built-in, so for any species and grade listed there, RISA-3D will provide the design values automatically. You can also enter design values directly.

For our model we will define two wood property sets, one for the truss chord members and one for the truss web members.

Remember that the green cell is the *active cell*. Currently the active cell is the Wood Label, which is "W1". We are going to change the species from the Aspen currently in the **NDS Label** field to a 4x4 Douglas Fir Larch and use this for the web members in the truss.

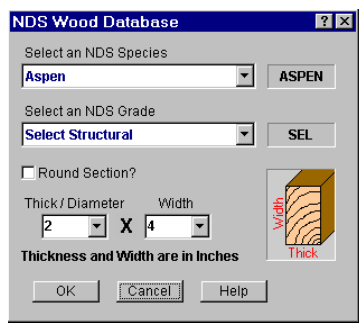
Click on the cell that currently holds "ASPENSEL_2X4".

It should now look like this:



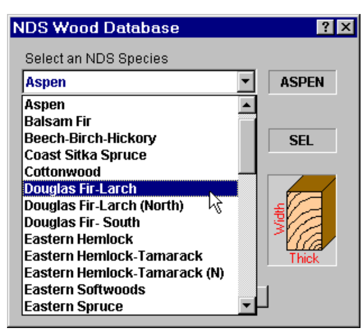
The active (green) cell should now be in the **NDS Label** field. Notice that there is also an arrow in the cell that looks like this .

Click the arrow and the **NDS Database** dialog opens:

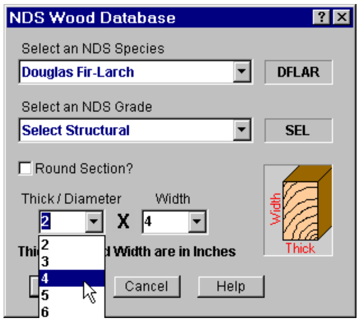


This dialog helps you enter NDS timber sections. The top edit field in the box contains the species, which is now "Aspen".

Click on the down arrow on the right side of this field and pick "Douglas Fir-Larch" like this:



Similarly change the thickness to “4”:



Click **OK**.

You are taken back to the **NDS** spreadsheet and the cell should now contain “DFLARSEL_4X4”. The properties will not be entered until we actually leave this cell.

Now we will add a larger section for the chord members but this time we will type it in directly. The syntax is fully documented in the help file so we won’t explain it here except to say that the first five letters are the species and the next three are the grade. The size comes after the underscore.

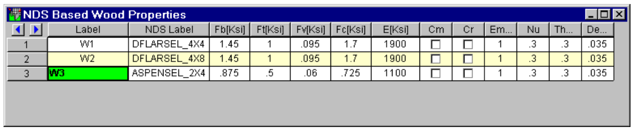
Press:

[ENTER], [TAB]

Note that the properties in the first row have been updated with the allowable stresses from the database. A new line is created and the active cell moves to “ASPENSEL_2X4”.

Now type:

DFLARSEL_4X8, [ENTER]

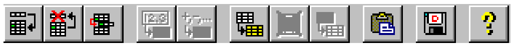


NDS Based Wood Properties

	Label	NDS Label	Fb[Ksi]	Ft[Ksi]	Fv[Ksi]	Fc[Ksi]	E[Ksi]	Cm	Cr	Em...	Nu	Th...	De...
1	W1	DFLARSEL_4X4	1.45	1	.095	1.7	1900	☐	☐	1	.3	.3	.035
2	W2	DFLARSEL_4X8	1.45	1	.095	1.7	1900	☐	☐	1	.3	.3	.035
3	W3	ASPENSEL_2X4	.875	.5	.06	.725	1100	☐	☐	1	.3	.3	.035

When you do this the value that you typed is processed and the remainder of the properties are updated. Pressing ENTER has also created a new line, which we do not need for our model. This will allow us to demonstrate how to delete a line.

Remember that the **Window Toolbar** will change as you move from window to window so that the buttons are relevant to what you are currently doing. It now looks like this:

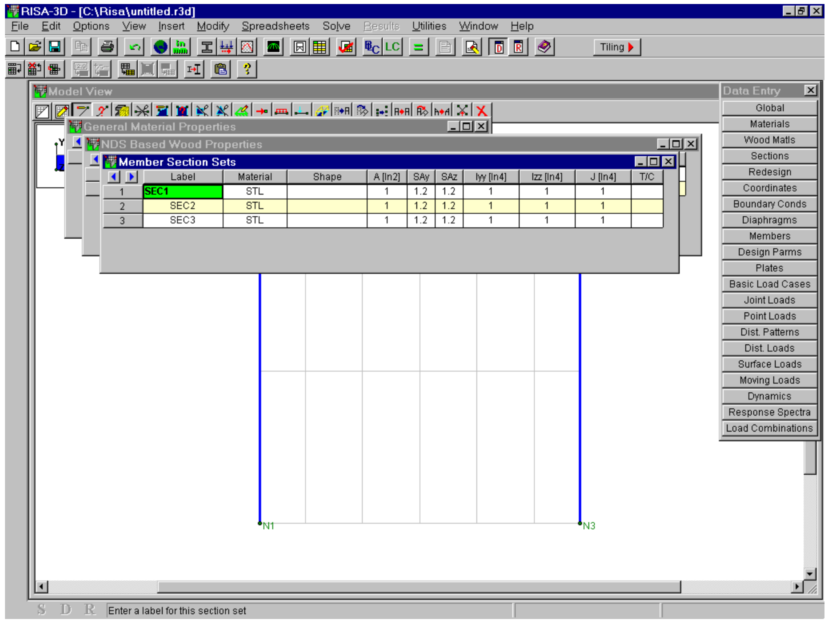


Click the second button , which deletes the current line.

Now we have just the two wood materials and we can proceed. Again, you could use the **Save as Defaults** button  to make the changes the default values for new files. Now we’ve defined our steel and material property sets, next we’ll use them when we define the section sets.

Sections Sets Spreadsheet

On the **Data Entry** toolbar click Sections. This spreadsheet appears:



Section Sets provide you with the ability to quickly make changes. Our truss has only nine web members but what if we had three trusses and we had to change the section, which we will in part two of this tutorial. Changing 27 member properties could take some time if you had to do one at a time. The ability to define section sets means you will only need to change one section on this spreadsheet to globally change the truss web members. Repetitive members are inherent in structural design and this provides an easy way to handle them. This will become even more apparent when we have RISA-3D pick new steel sizes.

You see the three section sets that we have already. They do not have any properties yet but they will soon. Also there is room for plenty more and remember we need another for the crossbeam. First lets change the current section set labels so that they are more meaningful.

The active cell should already be the cell that contains the text “SEC1” so lets change the three Section Set Labels.

Type:

COL, [ENTER]

CHORD, [ENTER]

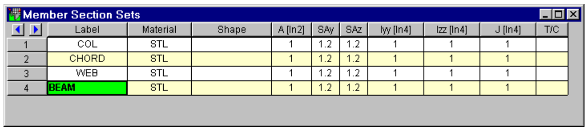
WEB, [ENTER]

A new line is created and the active cell holds “S4”. RISA-3D will make labels for you and you need only change them for your convenience. The spreadsheet line number is 4 so the label created is “S4”. We could have changed the Wood Labels (W1 & W2) if we wanted, and we still can. They would be automatically updated throughout the model wherever we had used them.

Now for the fourth Section Set type:

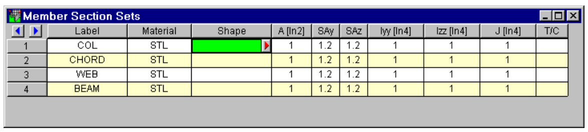
BEAM

So far we have:



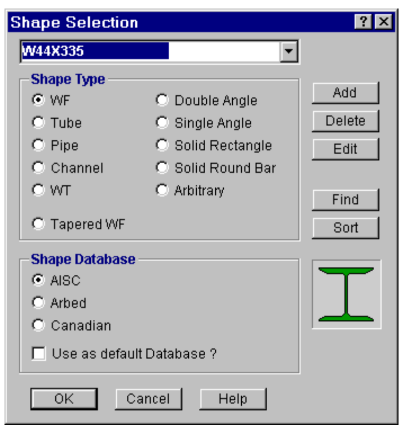
Now we will bring in the section properties from the databases and then get back to drawing the model.

Click on the first cell in the **Shape** column and it will look like this:



It becomes the active cell and that familiar arrow appears which will take us to the Steel Shape Database.

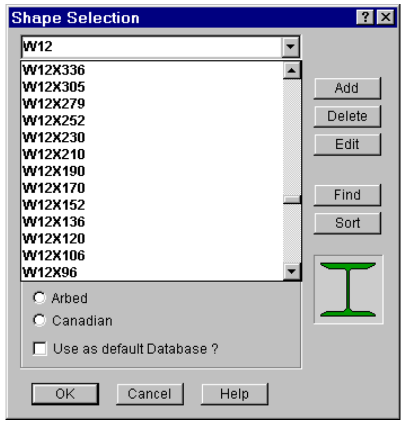
Click the arrow in the cell and the **Shape Selection** dialog box is opened which looks like this:



Shape Database

The edit box at the top of the dialog contains a W44X335 as the current shape. We will use a W12X40 for the columns. We could click the down arrow and choose the shape like we did for the wood species. However there are a lot of steel shapes and the W12's are certainly not at the top.

Instead, type "W12" and see what happens,



The edit box drops down just as if we had clicked on the down arrow but we are also taken to the W12 section of the database.

Now scroll down and click on the W12X40 to select it.

Of course we could have just finished typing the shape since we were already half way there. But if you don't know the database by heart you may not know what shape you want until you get there.

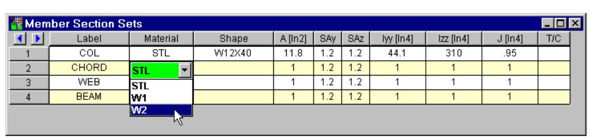
The rest of this dialog box is pretty self-explanatory. The Shape Type is currently a Wide Flange but can be any of the others listed such as a WT or channel section. The Shape Database is currently the AISC database but you can also choose from Trade ARBED and the Canadian shape database and even make it the default.

The shape in the edit box is now a W12X40 so click **OK**.

You will see the W12X40 is now in the **Shape** column of the first row in the spreadsheet. The rest of the properties will be updated as soon as you leave the cell.

Now the second line in the **Section Sets** spreadsheet is for the chord of the wood truss. The material is currently "STL" which is the label for the A36 steel on the **Materials** spreadsheet so we will need to change this.

Click that cell to make it the active cell and click the down arrow.



Select "W2" from the material labels which is the label we specified for the 4X8.

The properties will be updated after we leave the cell. Remember that for wood, material and section properties are closely tied in the NDS spreadsheet. All that is needed is the wood material label and the rest of the properties are filled in from the database, including the name of the shape.

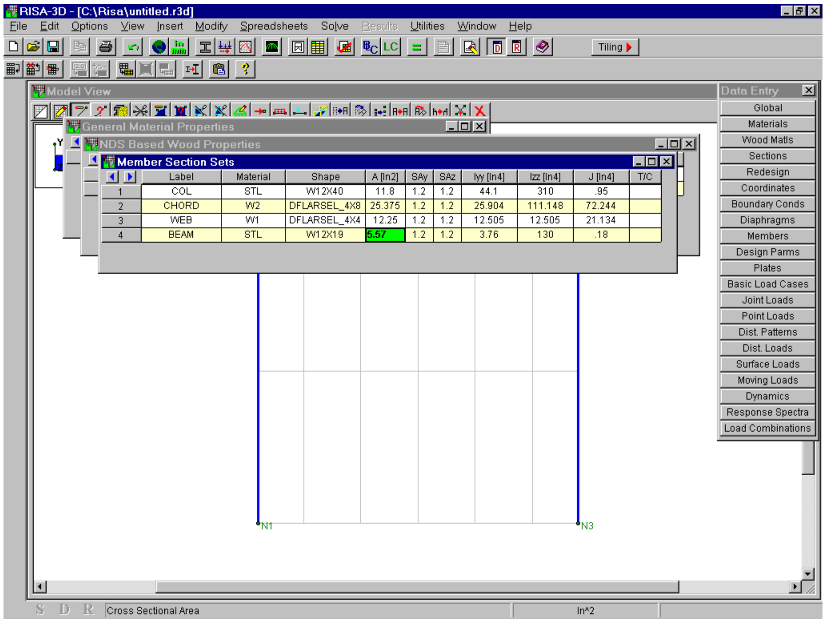
Now click on the material field for the WEB section, which is currently "STL". Click the down arrow and select "W1" which is the label for the 4X4.

Now only the BEAM section remains. The material is "STL" which is correct so we will just specify a shape. This time we'll just type which is faster when you know what you want.

Click on the **Shape** field in the fourth row and type:

W12X19, [TAB]

The properties aren't filled in until you leave the field by pressing TAB.



Note that our steel section sets are using a W12 shape. These are just initial shapes that RISA-3D will optimize for us a little later.

We do not have to use the databases to enter properties. For simple shapes or those not in the database we could have left the **Shape** field blank and simply typed the values for the area and moment of inertias ourselves.

Another feature, called "On-line shapes", allows you to type any basic shape, such as a pipe, and the properties are calculated on the spot. We won't use this here but we will mention it briefly because this is an important, timesaving, feature.

On-line Shapes

For this model we're using database shapes for the four section sets, but you do have the option of entering in the **Shape** field an "on-line" shape definition. RISA-3D has the ability to accept direct (on-line) definitions of solid rectangular and circular shapes and also pipes.

The following are the on-line entries RISA-3D recognizes:

RE**depth**X**width**	(rectangular; depth X width)
PI**dia**X**thick**	(pipe; diameter X thickness)
BAR**dia**	(bar; diameter)

So, as an example, if you were to enter “RE12x4” in the **Shape** field, you would be defining a 12" deep by 4" wide solid rectangular shape. RISA-3D automatically calculates all necessary section properties for on-line shapes (A, I, etc.) For more information on on-line shape definitions, see the help file.

As we mentioned before, when you want help with a spreadsheet click the button on the Window toolbar. Just as with the help buttons in the dialog boxes you will be taken to the appropriate topic in the help file. Go ahead and try it now and you will open the Sections Spreadsheet topic in the help file. Look around all you want. When you are finished, close the help file by clicking the button in the help window title bar.

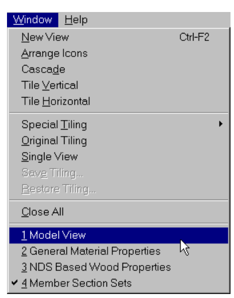
Multiple Windows

Now is a good time to talk about all of the windows that we have open on the screen. We have a model view and three spreadsheets. The “current window” is the one with the colored title bar (**Member Section Sets**) and the other titlebars are gray. The windows are laid one on top of the other in the order that we worked with them. You can return to any of the windows by clicking anywhere on them although you will want to be careful not to click on a button, or near a grid point if you are in drawing mode. It is best to click on title bars and other areas that don’t have actions associated with them.

If you want to move a window to get a better view of it or move it out of the way, click and hold the mouse button on it’s title bar while dragging the window to a new position.

But what if you can’t see any part of the window that you want to work with so that you can’t click on it?

Click on the **Window** menu.



The top of the **Window** menu presents some arrangement and tiling options that we will get to later. The bottom four options are the open windows. Notice that they are numbered in the order that we opened them starting with the **Model View** and ending with the **Member Section Sets** we just finished. The current window has a checkmark next to it. We can choose any of the open windows from this list at any time. So if you hide a window or just want to see what is already open you can always come here.

Now select the **Model View** by clicking on “**1: Model View**”.

The **Model View** comes to the front of the workspace and the spreadsheets are no longer visible. They are still open however but are behind the larger model view window.

You can also recall open windows the same way that you originally opened them whether it was with a menu or a toolbar button.

As an example, click on the Sections Sections button in the **Data Entry Toolbar.**

This window was already open so it was brought to where you can work with it.

You may close windows either by clicking the close button or by pressing the ESC key.

Press the [ESC] key to close the **Sections** spreadsheet.

We will leave the other spreadsheets open behind the model view.

So we are back at the model view and our cursor is still which indicates that we are still in drawing mode. RISA-3D remembers what mode you are in for all the open windows. We need to draw the crossbeam. We are still set up for drawing web members however because that is what we were doing when we left the model view.

We will use the **Rigid End Offsets** feature with the crossbeam so let's cover that next.

Member Offsets

Member offsets reflect the fact that the member ends may not be attached at the centerline of the member being attached to. For example, a beam connected to the flange of a column is offset from the centerline of the column by a distance of half the column depth.

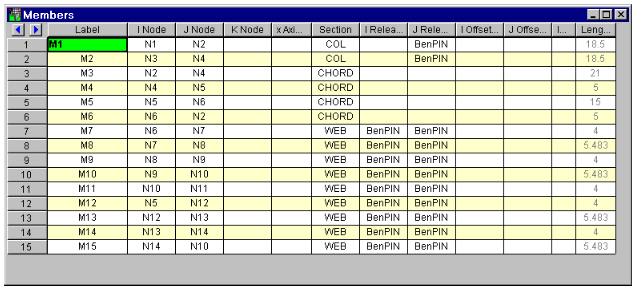
Instead of entering explicit offset distances, RISA-3D gives you the option of tying the offset distance to a particular member and calculating it for you. Say your member is framing into the flange of a 12" deep column. The offset distance would be 6", so you could enter 6" for the offset. Now, if that column gets changed to a W14 shape, you would have to go back and change the offset distance to 7" (or, more likely, something like 6.79" or 7.21"). If you have several members with offsets, it would be very time consuming to keep updating the offsets every time you change the column sizes.

To link the offset distance to the depth of some other member, enter the member label as the offset entry. For example, if the column your beam attaches to is member M10, you would enter for the offset "M10". Then half the depth of member M10 will be used for the offset distance. Of course, you can always directly enter a numeric offset distance.

We want to offset the beam end from the centerlines of the column members so we need to know the labels of the column members. The model view currently shows the node labels and we could plot the member labels as well. Instead we will visit the member spreadsheet.

Members Spreadsheet

Click the Members button on the Data Entry Toolbar, which brings up the **Members** spreadsheet.



Members

	Label	I Node	J Node	K Node	x Axi...	Section	I Relea...	J Rele...	I Offset...	J Offse...	I...	Leng...
1	M1	N1	N2			COL		BenPIN				18.5
2	M2	N3	N4			COL		BenPIN				18.5
3	M3	N2	N4			CHORD						21
4	M4	N4	N5			CHORD						5
5	M5	N5	N6			CHORD						15
6	M6	N6	N2			CHORD						5
7	M7	N6	N7			WEB	BenPIN	BenPIN				4
8	M8	N7	N8			WEB	BenPIN	BenPIN				5.483
9	M9	N8	N9			WEB	BenPIN	BenPIN				4
10	M10	N9	N10			WEB	BenPIN	BenPIN				5.483
11	M11	N10	N11			WEB	BenPIN	BenPIN				4
12	M12	N5	N12			WEB	BenPIN	BenPIN				4
13	M13	N12	N13			WEB	BenPIN	BenPIN				5.483
14	M14	N13	N14			WEB	BenPIN	BenPIN				4
15	M15	N14	N10			WEB	BenPIN	BenPIN				5.483

This demonstrates the close integration between the spreadsheet and the graphics. Every member that we have drawn thus far has been recorded here. Alternatively, we could have defined our members here instead of drawing them and they would be plotted in the model view just as they are now.

The **Members** spreadsheet contains all the information that we have specified with the **Set Member Properties** dialog. The labels we are interested in are in the first column. They are “M1” and “M2”. The other information here includes the I and J node for each member. The **K Node** and the **x Axis** information is for member orientation which we will cover soon. The section indicates which member has what properties. The end release information is given and the offsets follow which are blank right now but won’t be for long. The second column from the right handles inactive members, which will be covered later. The member length column on the far right is for information only and is the only field that can not be edited.

We now know the member labels for our columns and can use them when we draw the crossbeam. (Later we will see how to plot the labels in the model view.)

Press the [ESC] key to close the **Members** spreadsheet.

We are back at the model view. Remember the option **Recall Last Dialog** on the shortcut menu? Remember the “Ctrl-D” key combination that is associated with it? Let’s use that now.

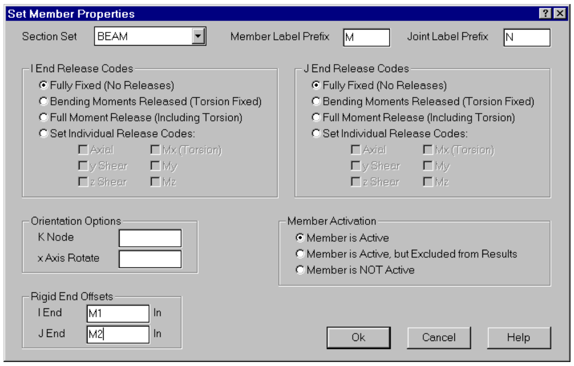
Press the key combination “CTRL-D” to recall the **Set Member Properties** dialog. We want to draw the crossbeam so click the down arrow next to “WEB” and change this to “BEAM”. Also, we want the beam to be fully fixed at both ends so select that option for the **I End Release** and for the **J End Release**.

Now, under **Rigid End Offsets,** click in the white edit box for the I-End and type:

M1 , [TAB] , M2

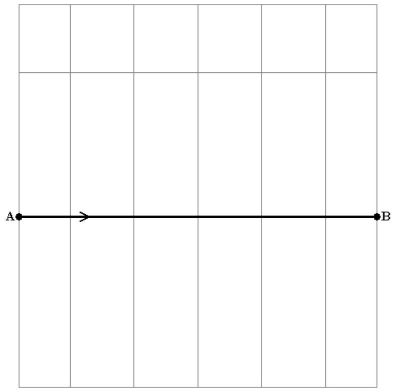
These entries mean use 1/2 the depth of member M1 for the I-end offset, and use 1/2 the depth of member M2 for the J-end offset.

The dialog now looks like this:



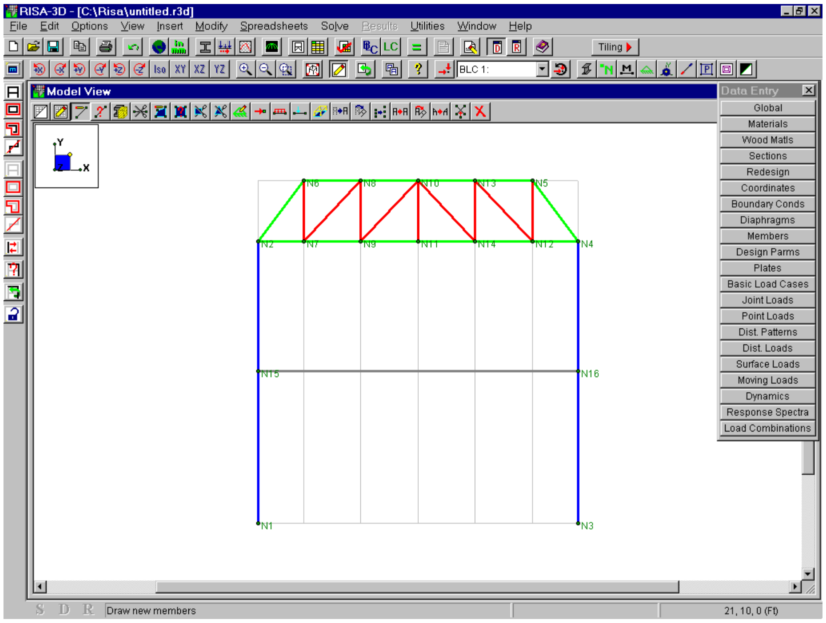
Click **OK** and we go back to the model view.

Here is what you'll be drawing; a line from A to B:



Move to point A, grid location (0, 10, 0), and click.

Now move right to point B, grid location (21, 10, 0), and click. RIGHT click to stop drawing.

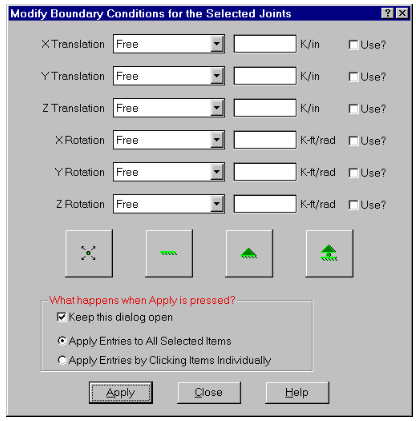


Boundary Conditions

We will now add boundary conditions at the base of the columns and then learn a little more about the model view and how to control it.

Click on the **Modify Boundary Conditions** button on the **Drawing** toolbar.

You will be presented with the following dialog:



Boundary conditions define how the model is externally constrained. All models must be attached to some external point or points of support. RISA-3D allows you to define these points of support as completely restrained or partially restrained by a spring.

Boundary conditions are applied to particular nodes. There are six degrees of freedom for each node (X, Y and Z translation and also rotation about the X, Y and Z axes), so there are six parameters provided, one for each direction.

Click the first down arrow in the **X Translation** field and you will see all the options for the boundary conditions:

Reaction - The "Reaction" code specifies full restraint for the indicated direction. No movement will be allowed in the indicated direction for this node. Furthermore RISA-3D will calculate a "reaction" at this node, for this direction.

Fixed - The "Fixed" code specifies full restraint for the node in the indicated direction. The difference between "Fixed" and "Reaction" is that for the "Fixed" code, no reaction is calculated. The "Fixed" condition actually removes the degree of freedom from the solution. If you aren't interested in reactions using the "Fixed" code will result in a slightly smaller model.

Spring - This models a spring attached to the node 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, if a spring of stiffness 100 Kips per Inch is desired, you would choose **Spring** from the drop down list and enter “100” in the adjacent box.

Compression Spring - This 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 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.

Now, we want to specify pin supports at the base of our columns, so we want X, Y and Z translations restrained, but we want the three rotations unrestrained. This is a standard boundary condition that RISA-3D has built in.

Click the **Pinned** 

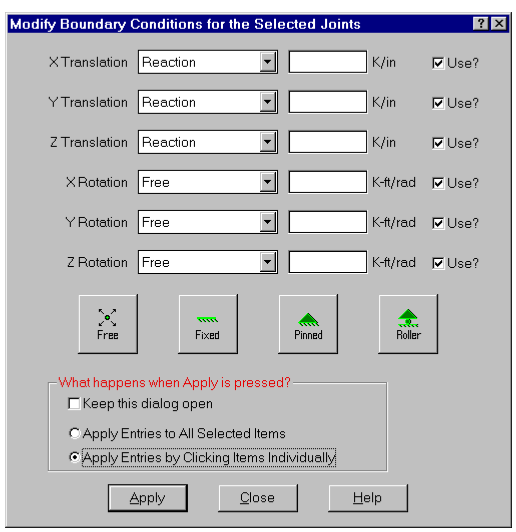
button and the conditions are filled in for you.

The boxes to the right labeled “**Use?**” must be checked for a modification to be applied. This way if you later decide to change a condition for just one of the parameters, you do not have to set all six to match the desired condition.

The section at the bottom of the box asks **“What happens when apply is pressed?”**. We have not covered the RISA-3D selection tools yet so just be aware that this is an important feature that we will come back to.

For now, click to remove the check from the box that is labeled **“Keep this dialog open”.** Choose the last option **“Apply Entries by Clicking Items Individually”.**

The dialog now looks like this:

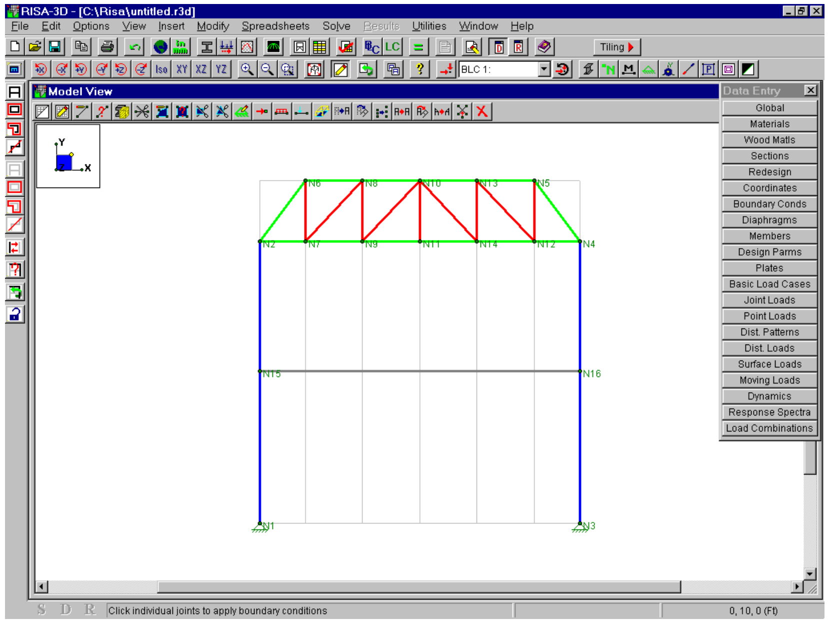


Click Apply.

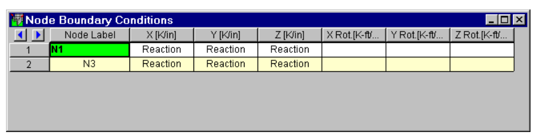
The mouse cursor now looks like this  , which indicates that you are ready to apply the boundary condition to any node that you click on. We want to apply the boundary condition to the base of both columns at nodes N1 and N3.

Click on node N1 and then on node N3.

We are finished with the boundary conditions and your screen should look like this:



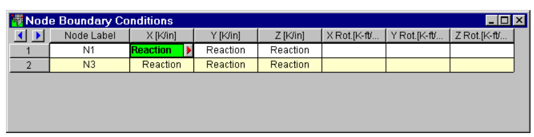
To see how these boundary conditions were recorded click Boundary Conds to open the **Boundary Conditions** spreadsheet:



Node Boundary Conditions

	Node Label	X [K/in]	Y [K/in]	Z [K/in]	X Rot.[K-ft/...	Y Rot.[K-ft/...	Z Rot.[K-ft/...
1	N1	Reaction	Reaction	Reaction			
2	N3	Reaction	Reaction	Reaction			

We could always edit the boundary conditions here, or add new ones. If we were to click on a cell we see that familiar arrow:



Node Boundary Conditions

	Node Label	X [K/in]	Y [K/in]	Z [K/in]	X Rot.[K-ft/...	Y Rot.[K-ft/...	Z Rot.[K-ft/...
1	N1	Reaction	Reaction	Reaction			
2	N3	Reaction	Reaction	Reaction			

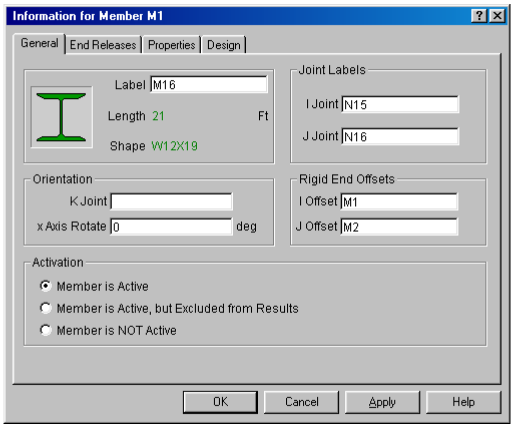
We could type in a boundary condition directly, usually by just typing the first letter such as “R” for reaction. We can also click on the arrow to get a helpful dialog much like the one that we just used.

Press [ESC] to close the spreadsheet.

Double-click for Information

RISA-3D provides a quick and easy way to get all of the information for any item in the model. Simply double-click on any joint, member or plate and you will be presented with a dialog that presents all of the information for that item.

Double-click on the crossbeam to obtain the following dialog:



Not only are you presented with all of the information for the member, you can edit it as well.

Move through each tab to see the **End Releases**, **Properties** and **Design** information (some of this we have not covered yet). Close the dialog without making any changes by clicking **Cancel**.

Now double-click on the bottom left joint to see information for joint N1.

You can make modifications to any joint, member or plate simply by double-clicking on it and making the adjustments. How much easier could it be? Well, if you had to change a lot of members there is an easier way. This is where the graphical tools in RISA-3D become very useful. Let's learn more about the model view and how we can best put it to use.

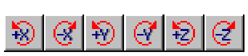
Close the dialog without making any changes by clicking **Cancel** and click  to remove the **Data Entry Toolbar**.

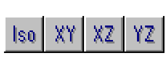
Model View

Now lets learn about some of the graphics features. Since we are currently working in a model view the **Window** toolbar will look like this:

The first button is the **Set Plot Options** button. You have already used this to color the members. We will save this button for last, as it is an important one and will get a lot of attention.

Rotating

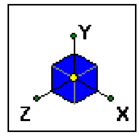
These buttons;  , are used to rotate the model plot counter-clockwise or clockwise about the global axes.

These buttons; 

, are used to "snap" the view to the particular plane (**XY , YZ , XZ**). The button labeled **Iso** snaps to the default isometric view.

The best way to understand what these buttons do is to try them, so, with the mouse, click on some of these buttons and experiment with rotating and snapping the plot.

As you click on these buttons, look at how the plot is rotating. Also look at the axes orientation shown in the top left hand corner of the model view window. That's this box:



When you're finished experimenting place the model in an isometric view by clicking .

Zooming

These buttons; , are used to zoom the model view in and out. This button; , is used to zoom in on part of the model. To use this option, first click on this button, then move the mouse over to the model plot. You will then draw a window around the part of the model you wish to zoom by holding down the left mouse button while dragging the mouse. Press and hold down the left mouse button and then start moving the mouse. You will see a box drawn from the anchor point to the mouse. Move the mouse such that the part of the model you wish to zoom in on is enclosed in the box, then let the mouse button go.

You can cancel the box zoom by dragging the mouse off of the model view window area.

Go ahead and try a box zoom now.

The other two buttons, and , zoom in or out on the model as a whole. The actual zooming is towards the center of the model view. You would typically use the box zoom to get to an area of interest and then use these buttons to adjust the magnification.

Try these buttons and see what they do. After you are finished click the **Full Model View** button.

Graphic Editing

This button, , is used to toggle the **Drawing Toolbar** on and off in the current model view. Remember you can have more than one view of the model open at one time and clicking this button will only affect the current view.

Click the button a few times to see what it does and when you are finished leave the **Drawing** toolbar ON.

Saving Views

You may use to save a view or recall a saved view. If you have a view that you like to work with a lot or have created a view that took some time, save it. Simply click this button and give the view a name. All of the plot options are saved with the model for later recall.

Cloning Views

You can clone a view by clicking . Cloning a view creates a new model view that is identical to the first. This allows you to preserve a view that you want to keep and make adjustments in the new view. We will clone our current view and then use the new view to explore the rest of the toolbar.

Click to clone the view and move the new window up by dragging it's titlebar.

Help

We mentioned the help button when we were working with the spreadsheets. Remember, this help button will open the help file to the topic that you are currently working in. If you were to click this button while in a model view the help file would open in the "Navigating Graphics" topic.

Loads

The next three items, BLC 1: , are provided to help you view the loads. We will come back to this once we have defined our loads.

Toggles

The remaining buttons are toggles and will control information in the new view.

Click to toggle model rendering. We'll cover rendering later on.

Click to toggle the node labels.

Click to toggle the member labels.

Click to toggle the boundaries.

Click to toggle the global axes orientation on and off in the view.

Click to toggle the I-J end flags.

If we had plates in our model would show the labels and if we had diaphragms defined would display them.

Click to change the background color between black and white.

Press ESC to close the view we created by cloning. The original view is unchanged.

Plot Options

Now click the first button on the **Window Toolbar**; .

This dialog contains the various options you have for controlling what is displayed on the plot. Related options are grouped together on a page marked by the tabs across the top. Remember that we used this dialog to color the members by their section sets. It opened back to the "**Members**" page, which is where we left it.

Round radio buttons and drop down lists control most of the options. Radio buttons are for controlling mutually exclusive options such that clicking one button will clear another. For instance when we turned on the color-coded plot we also turned off the **"Wireframe"** plotting that we started with. Lists are similar in that only one option may be chosen at one time.

Click on the down arrow next to the lists to view the options they give you.

Check boxes, on the other hand, are independent of each other so that multiple boxes may be checked at the same time. This brings us to another point about the hierarchy of the options in this dialog.

Some display options depend on other options, and that is reflected in how the various buttons are listed here. For example there are three ways to display the members, which are the main options across the top. They are: **"Wireframe"**, **"Color Coded"** and **"Rendered"**.

Each of these member options has a subgroup of buttons indented beneath it. None of the options in the subgroups will be shown unless the main option controlling that sub-group is on. For example, the **"Rigid Offsets"** option indicates whether you want each member's offset displayed. If you click this button on, the offsets won't be shown unless the **"Wireframe"** option is also chosen.

Some options are drawn in gray and can't be selected. Here, for example, the results review keys are all shown as inactive (gray) because the model has not been solved.

The buttons at the bottom may be clicked at any time and will accept or cancel all of the choices you have made on all of the pages. The difference between the **OK** and the **Apply** buttons is that clicking **Apply** will leave the dialog open after applying the choices you have made.

Model Rendering

A powerful graphics option offered by 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.

The subgroup beneath this button indicates how much of the member length is to be rendered. Can you tell which members are steel and which are wood? A rendered view offers some big benefits; besides producing a very nice image, we can also very easily review member orientations and, with this 3/4 length render, member connectivity.

Select the **Rendered** option by clicking the radio button next to it and click **OK** to close the **Plot Options** dialog.

For example, look at the columns. You can see instantly that the column orientations are such that they are not bending about their weak axis in the plane of the model, as they should be. We'll fix this soon.

What about member connectivity? This view allows us to also see how the members are connected to each other. In the non-rendered (colored line) view, the model looked fine, but closer inspection of the rendered plot reveals some problems caused by the way we drew the model.

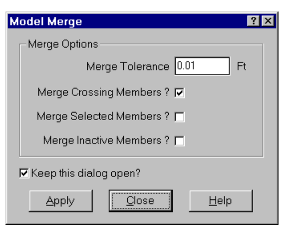
For example, we drew each column from the bottom up to the top as a single member; we didn't stop to include the joint near the middle of each column (the joints the crossbeam is connected to). This means the crossbeam is essentially "floating in space", not attached to any other members. The same situation exists for the truss top and bottom chords; most of the joints that define the web members are not incorporated into the chord members, which are drawn straight through these joints. The model as currently defined can not be solved.

So now you're thinking "Sure, skipping all those joints made drawing the model a lot easier, but NOW what do I do?". There is a solution! Model Merge!

Model Merge

First let's get to the model merge dialog, then we'll explain what it is.

Click on the **Model Merge** button on the **Drawing Toolbar** in the model view to see this dialog:



What happens when RISA-3D does a model merge is that the program first examines the model looking for duplicate nodes (nodes with the same coordinates). The tolerance specified in the panel determines how close two nodes need to be to be considered duplicates. Any that are found are merged into a single node.

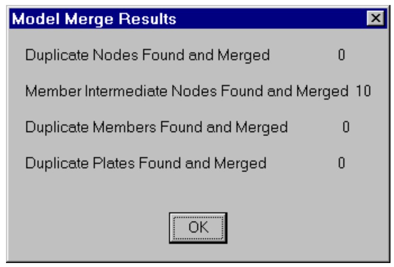
Next, the members are all checked to see if there are nodes defined along the span of the member. The tolerance value is used for this as well. If so, the member is split into pieces that are connected to these nodes along the span.

Next, if the "Merge crossing members?" option is checked in the model merge panel, RISA-3D will look for members crossing each other, and if any are found, a node is created at the crossing point and the members are split to incorporate this new node. A typical case might be X-bracing.

Finally, the model is checked for duplicate members. Any duplicates found are merged into a single member.

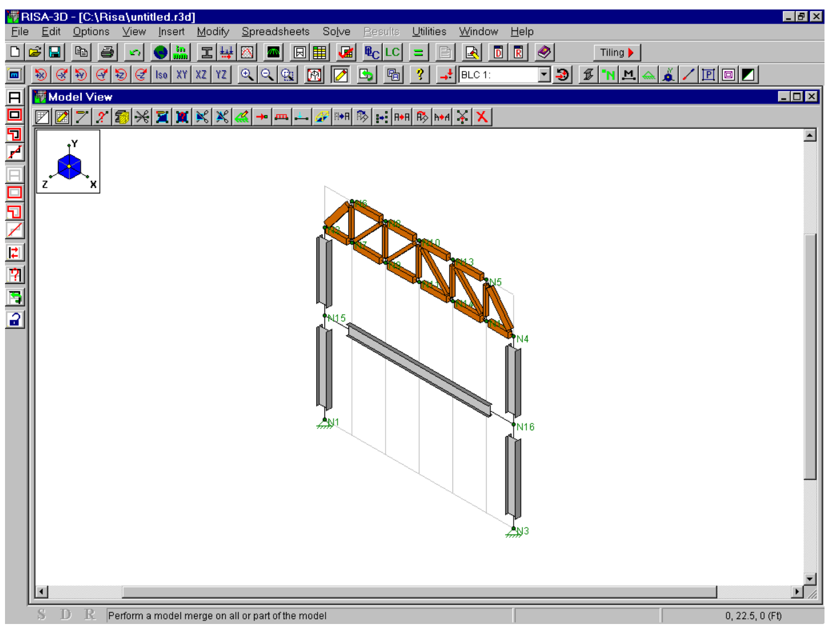
You have the option of having RISA-3D only look at part of the model for the merge, this is related to the "select and unselect" features still to be discussed. If the "**Merge Selected Items Only**" option is NOT checked, the merge will be applied to the entire model.

Click on **Apply** and you will be presented with the results of the merge:



Click **Ok** then click **Close** on the **Model Merge** dialog box.

Now look at the model and note how the columns and truss chords have been split up.



Powerful Feature

It's important to realize how this merge feature can make drawing models so much easier; you don't need to worry about intermediate nodes along member spans or even about members that cross each other, just draw the members from start to end and use the merge to split them up.

Click on the **Toggle Grid** button on the **Drawing Toolbar** to hide the drawing grid since it is no longer needed.

Selection Tools

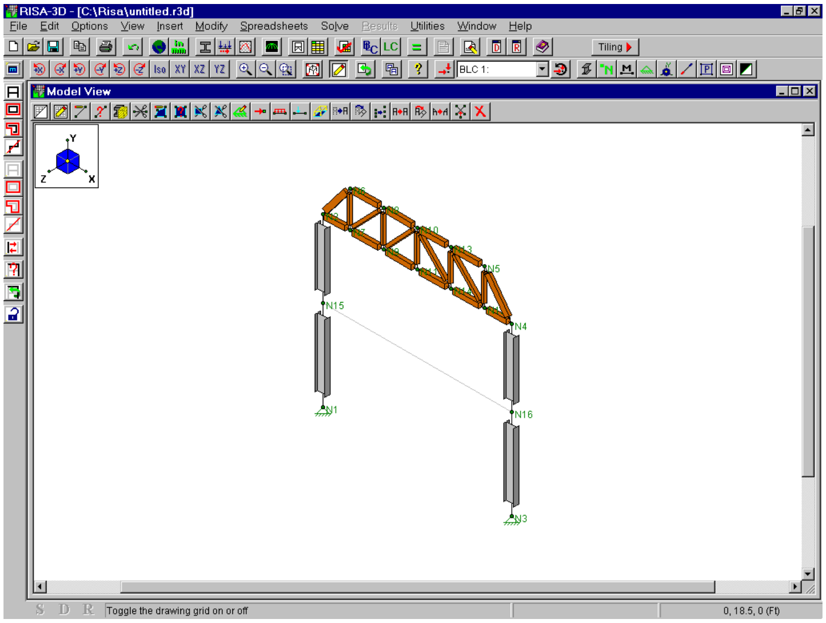
Before we proceed any further, a very important concept needs to be explained. This is the concept of items being graphically "selected" or "unselected". Selections are important when we start changing our model because we need to specify where the change is to take place.

Stated simply, an item is "selected" if it is fully displayed graphically. An item is "unselected" if, instead of a display of the item, only a gray line shadow of the item is displayed. An "item" could be a joint, member or plate.

In the model that we are working on, all of the nodes and members are currently selected. They are fully drawn, which means they are selected. This is the default state when you define a new item.

Click on the crossbeam that we just created and you will see that the rendered shape disappears and that a gray ghost line takes it's place.

Your model should now look like this:



The crossbeam is now "unselected".

Currently your mouse cursor is the standard arrow when in the model view. (The cursor changes when it is over a button or menu item.) This indicates that you are in the default selection mode, which simply means that you can select or unselect one item at a time by clicking on it.

Try this now, click on a few of the lines to see how the members can be unselected, and then again selected. (Remember, if you double-click you will open the **Information** dialog! If you do this by accident click **Cancel**.)

You can also click on the joint locations to select and unselect individual nodes so try this as well.

Rather than clicking one item at a time you may use RISA-3D's powerful tools to help you quickly achieve the selection that you want. They are located on the vertical **Selection** toolbar on the left side of the screen.

Note that this toolbar will not be displayed when a spreadsheet is active. Let's try the selection tools. Feel free to try all of them as you can not change the model solely by selecting items.

Click on this button .

This "unselects" the entire model. See how the model is now displayed in gray?

Now try each of the selection tools below.

The **Box Select** and **Box Unselect** tools work as you might expect; you use them to draw a box around the part of the model you wish to select or unselect. Only items that are *entirely* in the box will be affected.

To use these, first click the button then move the mouse to the model view. The button stays down and the cursor changes to indicate that you are in the box mode.

Click and hold the mouse button to establish an anchor point for the box; then move the mouse such that the box encloses that part of the model to be selected or unselected. Then release the mouse to perform the select or unselect operation.

The box cursor remains and you may draw as many boxes as you need until you are finished.

You can end the operation by re-clicking the box button to turn the option off. You can cancel a box that is underway by dragging the mouse off the model view while still holding the mouse button down.

The **Polygon Select** and **Polygon Unselect** tools work in a similar fashion. You use them to draw a polygon around the part of the model you wish to select or unselect. The polygon may have many sides, so to indicate that you are finished drawing click the mouse twice at the last location.

With the **Line Select** and **Line Unselect** tools you simply draw a line through all of the items that you want to select or unselect. Just click and hold the mouse at the start of the line and then draw the line until you release the mouse button. This is a great tool for selecting the columns or the web members in the truss.

The **Invert Selected** option is 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 on the desired items to unselect them, then click "**Invert Selected"** to "flip" the selection.

The **Criteria Selection** button allows you to select items in ways other than choosing them on the screen. You can select items between certain coordinates.

You can also select items by their labels such as all joints from N7 to N15. For members, you can select by many criteria such as orientation, section set, database shape and material. Plates can be selected by material, thickness and orientation.

The **Save/Recall** button is used to save and recall selection states for the model. This is very helpful for large models that require more than just a few clicks to obtain a particular selection state.

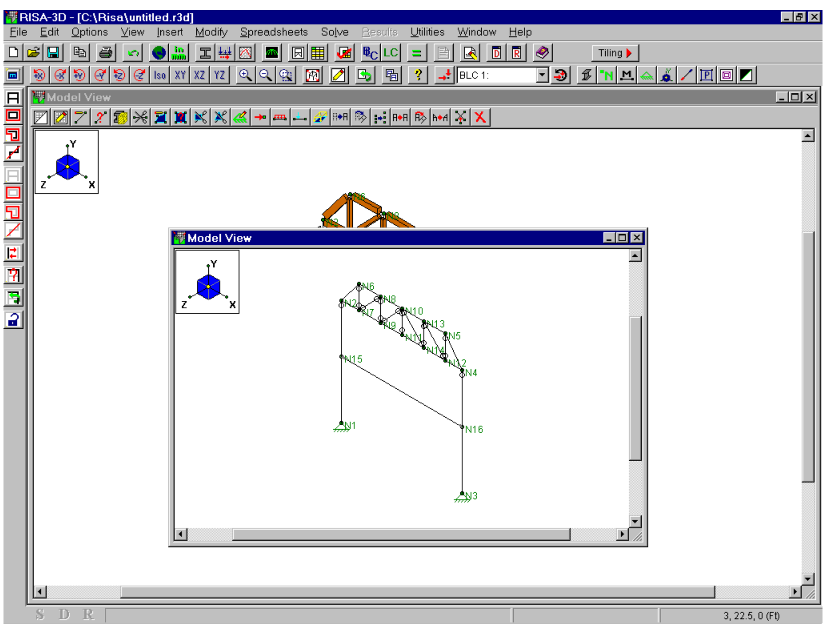
The **Lock Unselected** button, when pressed, causes all currently unselected items to stay unselected, no matter what other selection buttons are pressed, until this button is pressed again. This is useful when you are working on just part of a model, say one particular floor in a multistory building, and you need to select and unselect different parts of that floor and you want the rest of the model to "stay out of the way". Just unselect the entire model, then use **Box Select** to select the floor, then press "**Lock Unselected**" (the button changes to) and the rest of the model will stay unselected until you press "**Lock Unselected**" again to turn it off.

When you are finished trying all of the selection tools make sure that the **Lock Unselected** button is up and looks like this:

Now click the **Select All** button to select the entire model.

Multiple Views

We demonstrated that you can have more than one view when we used the clone feature. Each view is independent and can be rotated, rendered, selected, etc. without affecting other views.



On the **RISA Toolbar** click on the **New View** button and you will be presented with an original model view. Feel free to use the buttons on the

Window Toolbar and the **Select Toolbar** and you will see that you can adjust this view without affecting our original view.

The importance of multiple views can not be stressed enough. If you don't 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 what you are doing in the view. For instance you might be drawing new beam members in one view, new columns in another and plates in a third.

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. There are plenty more features to showcase so we will do just that.

Close the second view by clicking in the title bar.

Modify Options

The **Drawing** toolbar has different options, some of the options create new data such as new beams or distributed loads. Other options make changes to already existing data such as "Modify Boundary Conditions".

For these "modify" options a "**Use?**" checkbox is available for each parameter that may be modified.

This little box is used to indicate whether the particular parameter is to be applied. We saw this when we defined the boundary conditions. If the little box is checked, the parameter *will* be used, if the little box is not checked, the parameter *won't* be used. The usefulness of this feature will become obvious in this next section.

Modify Members

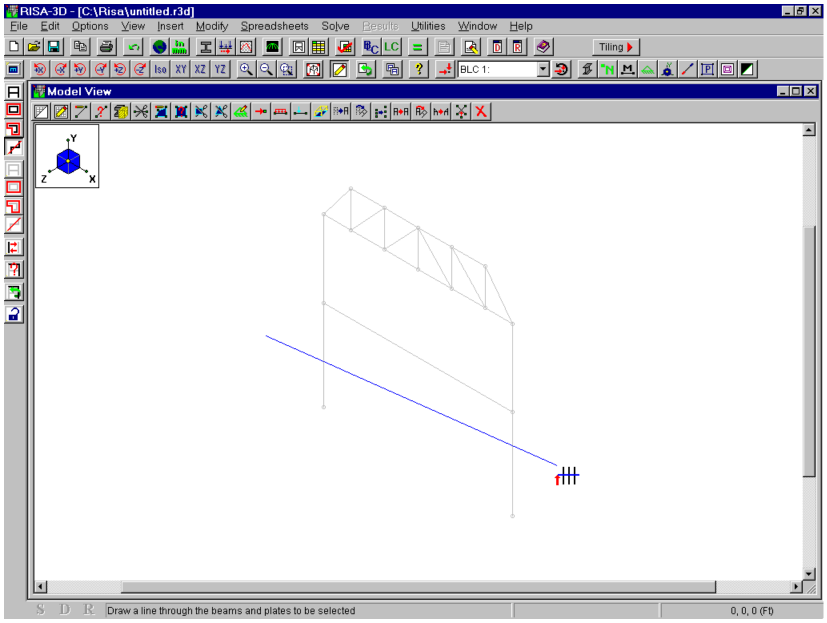
Now that we know how to select parts of the model we will use this and rotate the columns 90 degrees so that they bend about the weak axis.

First click **Unselect All** to unselect the entire model.

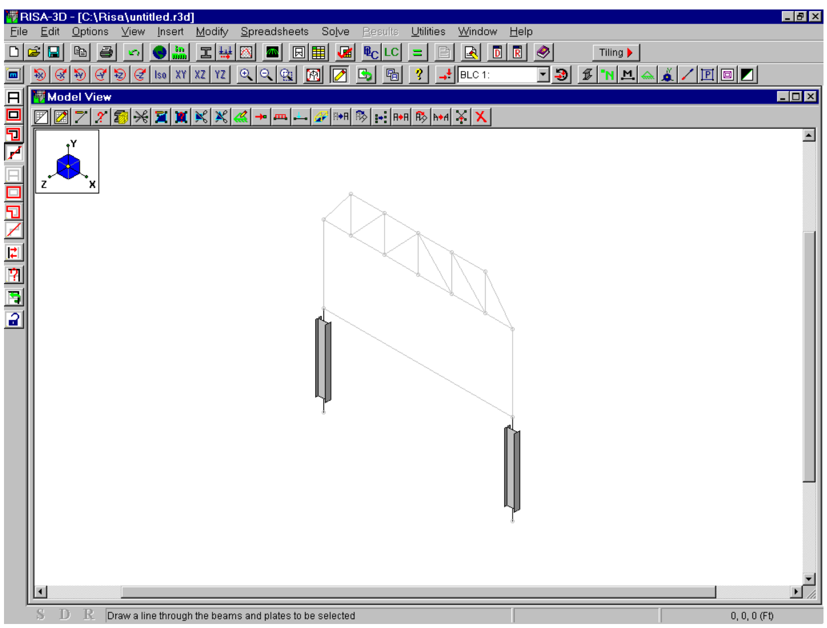
We will use **Line Select** to select the columns so click and your mouse cursor becomes .

The line select is useful for columns because we can draw lines through the columns without selecting other members.

Click and hold the mouse to the left of the model and draw a line through the two columns below the crossbeam. Release the mouse when it is on the right side of the model as shown here.

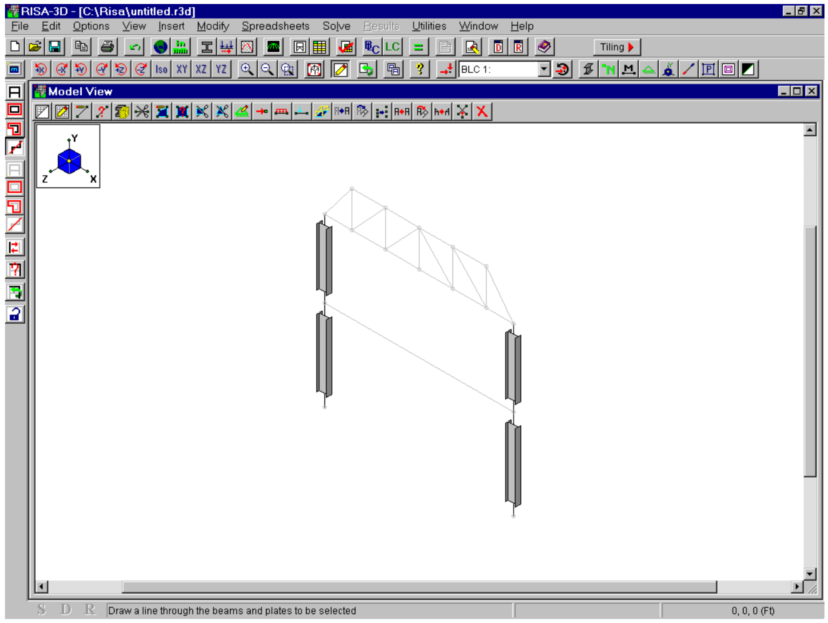


The two lower columns are now selected as shown below.



Now drag a line through the two upper columns between the truss and the crossbeam.

The end result should be that the four column members are selected and the crossbeam and the entire truss are unselected as shown below.



Click on the **Line Select** button to pop the button back up and end the selection mode. Now click on the **Modify Members** button on the **Drawing Toolbar** and a dialog window presents modification options.

This dialog looks a lot like the **Set Member Properties** dialog that we used to draw the members. This is because it allows you to change any of those parameters *after* drawing the members.

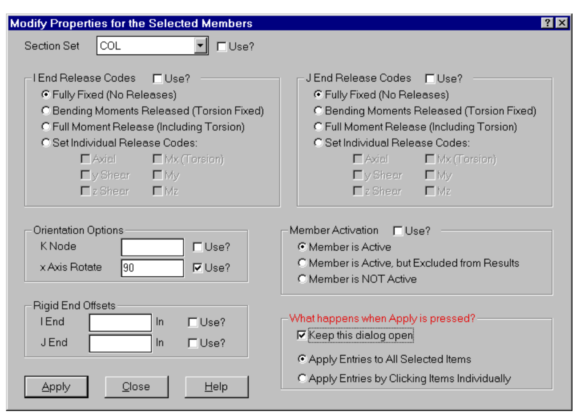
Halfway down on the left is a field that is labeled "**x Axis Rotate**".

Click in the **"x-Axis Rotate" field and type:**

90

Now click the box just to the right of that field that is labeled "**Use?**" so that the box is checked.

The dialog is now set and should look like this:



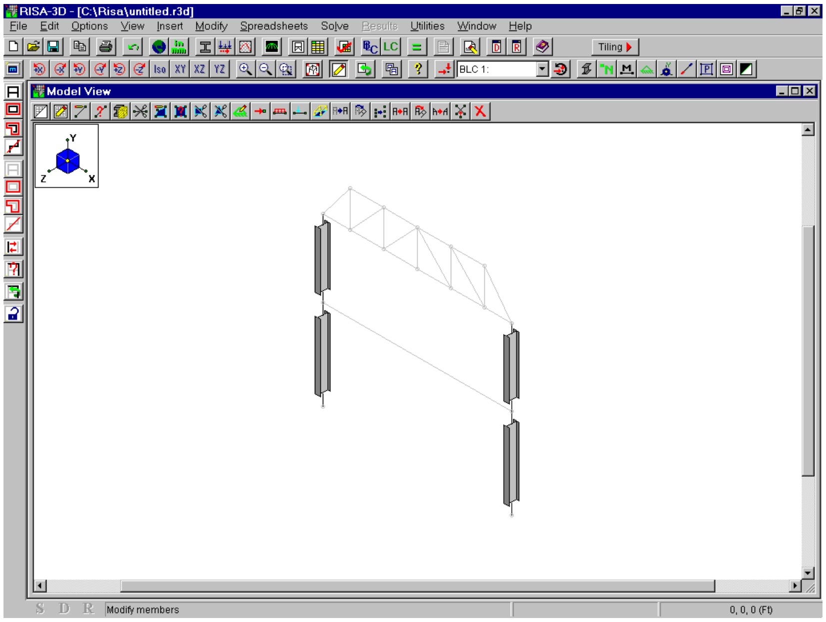
What we have done here is specify that we want to rotate the members 90 degrees about their local x-axis. The rest of the settings will not be used because the **Use?**

boxes are not checked. For example, the release codes specify fully fixed, but the column end conditions will not be modified due to the unchecked **Use?** box. If the **Use?** box was not a feature in RISA-3D then we would have to reset these releases even though the column ends don't need to be modified.

Another important thing to note is that we are using the option that states "**Apply Entries to All Selected Items**". We made our selection prior to opening this dialog but we could have waited unit now to select the members. Regardless, clicking Apply will rotate the selected members 90 degrees.

Click **Apply** and then click **Close**.

The columns have been rotated to the orientation shown below.

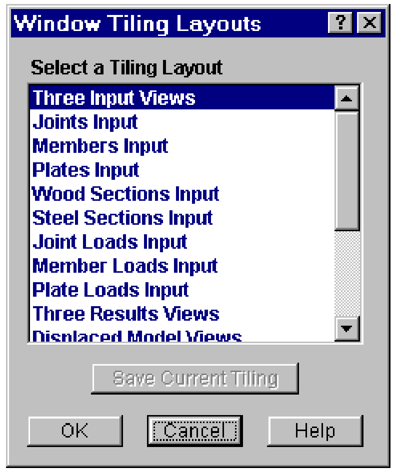


Click **Select All** to select the entire model.

Sorting

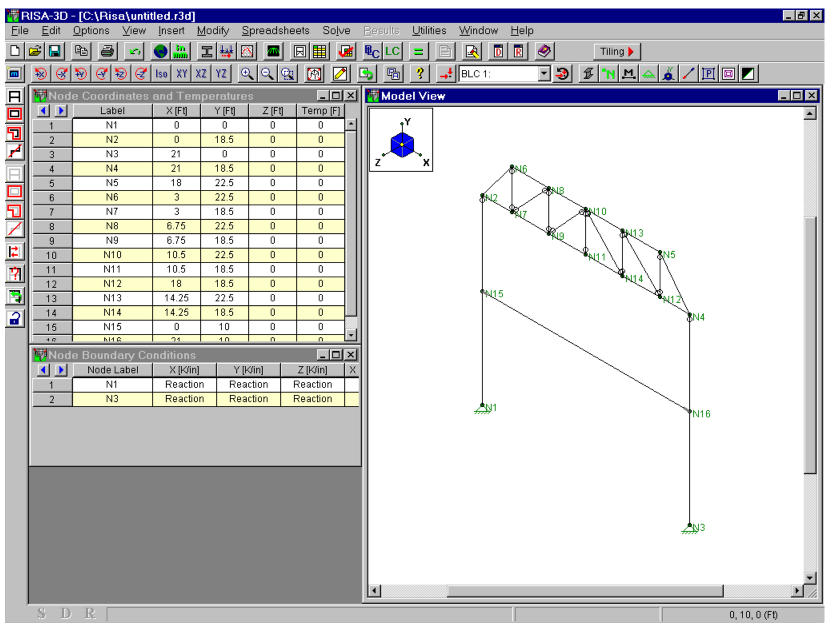
Let's look at the **Joint Coordinates** spreadsheet for the first time and let's use a special window arrangement to do this.

Click the **Tiling** button on the **RISA Toolbar** and the following list of window tiling options is presented.



Later on we will experiment with these options. For now, select the second option which is "**Joints Input**" and click **OK**.

Your screen should look like this:



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

The nodes in the **Node Coordinates** spreadsheet are listed in the order in which they were created as we graphically drew, and then merged, our model. It would probably be more useful to list them based on their coordinate locations, so let's do a sort.

Click anywhere in the X-coordinate column.

Now click on the **Window Toolbar** and a dialog with the title "Sort X" is presented. Select "**Sort Low to High**" and click **OK**.

The joints are now sorted in ascending X coordinate order. Now lets add a Y-coordinate sort.

Click in the Y coordinate column.

Click to open the dialog titled "**Sort Y**". "**Sort Low to High**" is already selected so click **OK**.

So now we have the joints in ascending Y coordinate order, and at each Y coordinate level, they are further sorted in ascending X coordinate order. Notice that the labels stayed with their assigned nodes and the model plot looks exactly the same.

Now RIGHT-click on the **Joints** spreadsheet and select **Relabel Joints** from the pop-up menu**.** You are given the option to change the prefix for the labels but we will stick with "N". Click **OK**.

The joints have been relabeled and this can be seen in the model view window as the nodes are now labeled from the bottom up.

Now let's quickly sort our members as well.

Click again and select "**Members Input**" and then click **OK.**

This presents all of the spreadsheets that contribute to the definition of members. The **Members** spreadsheet is located in the lower left corner. We will use this to sort the members first by their I-Node and then the section set.

To sort the members by I-Node click anywhere in the **I-Node** column and click . Choose "**Low to High**" as the sort option and click **OK**.

To sort the members by section set click anywhere in the **Section** column and click . Choose "**Low to High**" as the sort option 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 displayed in the spreadsheet.

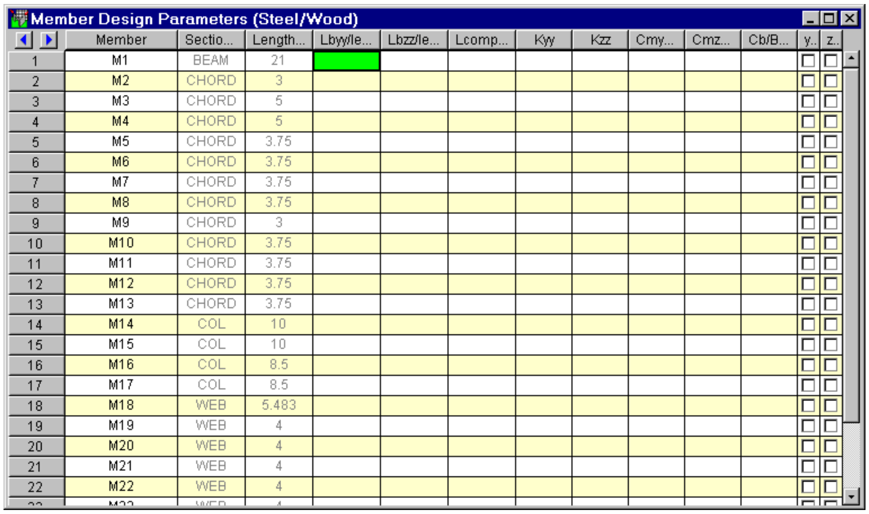
Now RIGHT-click on the **Members** spreadsheet and select **Relabel Members** from the pop-up menu**.** Click **OK**.

This will sort our beams in ascending section set order, making them easier to identify and work with in our next operation which is assigning parameters for steel and wood design.

Design Parameters

The **Member Design Parameters** spreadsheet is where code parameters, such as unbraced length, are recorded.

Click to recall the **Data Entry Toolbar** and press Design Parms.


Member Design Parameters (Steel/Wood)

	Member	Sectio...	Length...	Lbyy/le...	Lbzz/le...	Lcomp...	Kyy	Kzz	Cmy...	Cmz...	Cb/B...	y...	z...
1	M1	BEAM	21										
2	M2	CHORD	3										
3	M3	CHORD	5										
4	M4	CHORD	5										
5	M5	CHORD	3.75										
6	M6	CHORD	3.75										
7	M7	CHORD	3.75										
8	M8	CHORD	3.75										
9	M9	CHORD	3										
10	M10	CHORD	3.75										
11	M11	CHORD	3.75										
12	M12	CHORD	3.75										
13	M13	CHORD	3.75										
14	M14	COL	10										
15	M15	COL	10										
16	M16	COL	8.5										
17	M17	COL	8.5										
18	M18	WEB	5.483										
19	M19	WEB	4										
20	M20	WEB	4										
21	M21	WEB	4										
22	M22	WEB	4										

This spreadsheet is used to enter the parameters that are required for AISC code checking for steel members and also NDS code checking for wood members. Notice that most columns have two labels, one for steel and one for wood. You may have to stretch the columns to view the entire label. We will briefly mention each column below. Refer to the help file for more information.

The first three columns are for information only and may not be edited. These are the member labels, the section set and the actual node to node length of the member.

The **Lb/le** columns are the unbraced lengths for buckling about the member's y-y and z-z axes. These are used in KL/r ratio calculations for steel, and le/d calculations for wood.

The **Lcomp** value (used for steel) is the laterally unbraced length of the compression flange used in the calculation of allowable bending stress for ASD design, or member strength for LRFD design. The **le-bend** value (used for wood) is used in the calculation of RB.

When these cells are blank RISA-3D will use the full node to node length of the member for these values (Lb's, le's and Lcomp).

The **K** factors are the effective length factors, which RISA-3D is capable of approximating for you.

The **Cm** factor, used in the calculation of the bending stress ratio for steel ASD design, may be entered if you wish to use a specific value. If left blank it will be calculated exactly by RISA-3D. **CH** shares the same column as Cmyy and is the NDS shear stress factor.

If you leave the **Cb** factor blank RISA-3D will calculate it for you if you're doing ASD design. For LRFD design, the **Cb** value is set to "1" if not entered.

The final two fields are the sway check boxes to indicate whether the member is to be considered subject to sidesway for bending about its y-y and z-z axes. Sway comes into play in the calculation of **Cm** and **Cb** and in the approximation of **K** factors.

Currently, all the entries are blank, which means RISA-3D will use the default values when the code check calculations are performed.

Redesign

RISA-3D has the ability to optimize the steel section sets. This means RISA-3D will suggest an alternate database shape for our steel sections that should perform better than the currently defined shapes. Only those shapes of the same type as the shape originally defined for the section set will be considered. For example, section set COL is defined with a wide flange, so only wide flange shapes will be suggested as alternates.

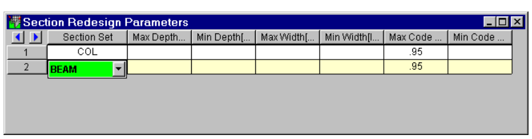
To have RISA-3D do this, we must tell the program which section sets are to be redesigned and how we want it done. We do this on the **Redesign** spreadsheet.

Press Redesign and the spreadsheet opens with no data. Press ENTER.

The first section set (COL) is inserted automatically to create the first data line. Let's enter the label of the beam section.

Press ENTER so that a new line is created and then press the letter "B".

RISA-3D looks in the drop down list for a section that begins with "B" and enters it. If there were more than one such section you could press "B" multiple times until it found the one you wanted.



If you come to this spreadsheet and you can't remember the labels for the section sets or have many sets to redesign you may click the **All** All button on the **Window Toolbar** to have RISA-3D automatically fill in *all* the section sets here.

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. We will specify that our beam is at least 12" deep.

Click in the second row of the **Min Depth** column and enter:

12

Basic Load Cases

Before we load the model it is important to understand just how RISA-3D lets you manage loads. When loads are defined in RISA-3D they are assigned to a "Basic Load Case". You can have up to 1000 separate Basic Load Cases (BLC's). You also have the option to assign the BLC's to categories, which help you easily combine them later in the model solutions. When you are ready to solve the model, the BLC's and categories are combined (with multipliers) into one or more load combinations.

What we're going to enter are a set of joint loads as a roof load and a set of distributed loads as the wind load. These loads will be used differently in each combination so we will define them in different BLC's.

To apply our loads graphically we need to get back to the model view. Since we have a lot of windows open we will clean up the workspace before we move on.

From the **Window** menu select **Single View** and then turn on the drawing toolbar by clicking the button on the **Window Toolbar.**

Joint Loads

First let's apply node loads to the top of the truss. These will represent our roof load and will be grouped in Basic Load Case 1.

Click the **Joint Load** button on the **Drawing Toolbar**.

The joint load dialog should now be open. At the very top of the dialog we can specify that we want to apply a joint load or an enforced displacement on the joint. We will leave this set to joint load.

The direction that we want is the vertical direction. The global axes are plotted in the model view and the vertical is Y. Remember that positive is up.

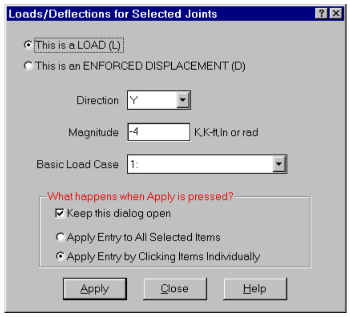
In the **Direction field**, click the down arrow and select "Y".

Next, click in the **Magnitude** field and type:

-4

We will use Basic Load Case 1 so skip down to the bottom section to specify how the loads are applied. Like we did with the boundary conditions, we will click the joints to apply the load. This time we will leave the dialog open so that we can change the load magnitude as we go.

Select the last option **"Apply Entry by Clicking Items Individually"** and the dialog will look like this:



We have specified that the dialog remains open while we apply the joint loads because we will apply loads of different magnitudes for the outer joints. This is an important feature and one that you will come to appreciate.

Click **Apply,** the dialog remains open but the mouse cursor changes to  . You may have to move the dialog down to view the top chord joints.

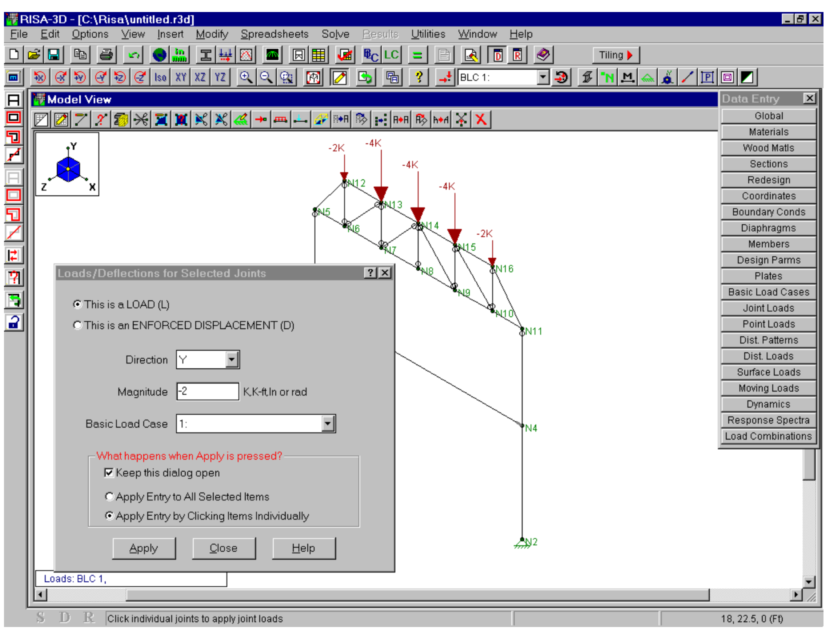
Now click on nodes N13, N14 and N15.

As you click, the load will be drawn for you with along with its magnitude. Don't forget about the **Undo**  button should you make any mistakes.

Now go back to the **Joint Loads** dialog and change the magnitude to "-2" and click **Apply** again. (This is important. If you don't click **Apply** then the change is not accepted.)

Return to the model and click on the two exterior panel points on the top chord which are Nodes N12 and N16.

When you are finished your screen should look like this:



We are finished applying joint loads so close the dialog by clicking **Close**.

We will now apply distributed member loads as Basic Load Case 2, which will be "wind load". RISA-3D allows you to define load patterns and then use those patterns to apply distributed loads.

Load Patterns

Open the **Distributed Load Patterns** spreadsheet by clicking Dist. Patterns.

Generally, the distributed loads applied to various members of a structure have the same basic pattern (usually full-length and uniform). Rather than require the entry of repetitious data for every member, RISA-3D allows you to define basic distributed load patterns for use with multiple members. For the individual members, you only have to call out a pattern and specify a multiplier. Once the basic patterns needed for the model are defined, the same information doesn't have to be repeated over and over.

The best way to use these patterns is to specify the pattern magnitudes with a value of 1.0. That way, when you need to apply the same sort of load to different members, where only the magnitude of the load changes, you can use the same pattern label and enter the actual load magnitude as the pattern multiplier.

Three patterns are provided by default for the most common loads. They define full-length constant uniform loads in the global directions (X, Y and Z). Actually they use negative magnitudes so the load is defined in the negative X, Y and Z directions. The UNIFORMX pattern will model the wind load.

Note that if the start and end locations are both zero, RISA-3D will distribute the load across the full length of the member!

Of course, these patterns are not limited to full length uniform. Simply add new lines to the spreadsheet (by pressing ENTER) and define the loads you want. You can enter any start and end magnitudes and locations. This way you can define partial length, triangular or trapezoidal loads.

The load locations can also be entered as a percentage of member length. To do this, enter the percentage value *preceded* by the symbol "%". For example, you could define the start and end locations as "%25" and "%75" respectively, to specify a load over the middle 2 quarters of the member, whatever the member length. The percentage is of the full node to node length of the member, ignoring any offsets.

Click ☒ to close the spreadsheet.

Distributed Loads

Click on the **Create Distributed Loads** button to open the dialog.

Before filling in the panel, let's select only those members we wish to load. Previously we used the **"Click to Apply"** option both with the dialog closed and then most recently with it remaining open. This method essentially allowed us to select one node at a time as we applied boundary conditions and joint loads. But what if we had to apply loads to 100 members? Certainly we would not want to click on each one. Now we will use the **"Apply to Selected"** option to demonstrate the use of selections once again.

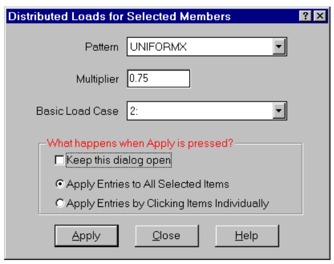
We could have selected the members first but we will do it now to demonstrate that you may change your selection at any time as you work. You don't have to close the dialog to make the selection and then open it again to apply a load. This is very important if you want to apply different loads as we did with the joint loads. Opening and closing dialogs would hinder this process.

Click to unselect the entire model. Now select the RIGHT side columns by clicking on each one. (Two members in all, one on top of the other joined at the crossbeam).

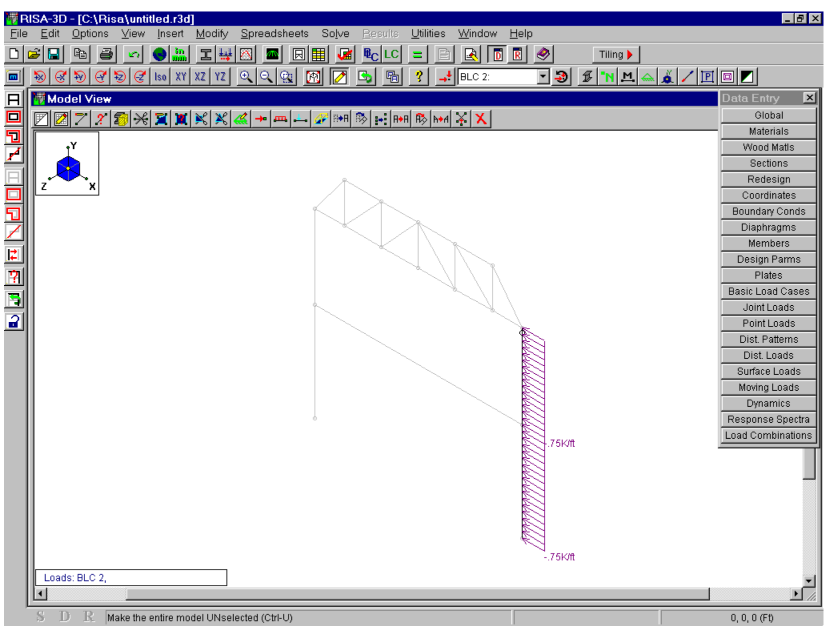
Now we will fill in the Distributed Loads dialog.

Change the multiplier to "0.75" for a 750plf load. Change the Basic Load Case by clicking the down arrow in that field and select "2". Clear the checkmark from the box labeled **"Keep this dialog open".**

The option to **"Apply Entries to All Selected Items"** should be selected.



Click **Apply** and the screen should now look like this:



Select the entire model by clicking **Select All** .

Basic Load Case Spreadsheet

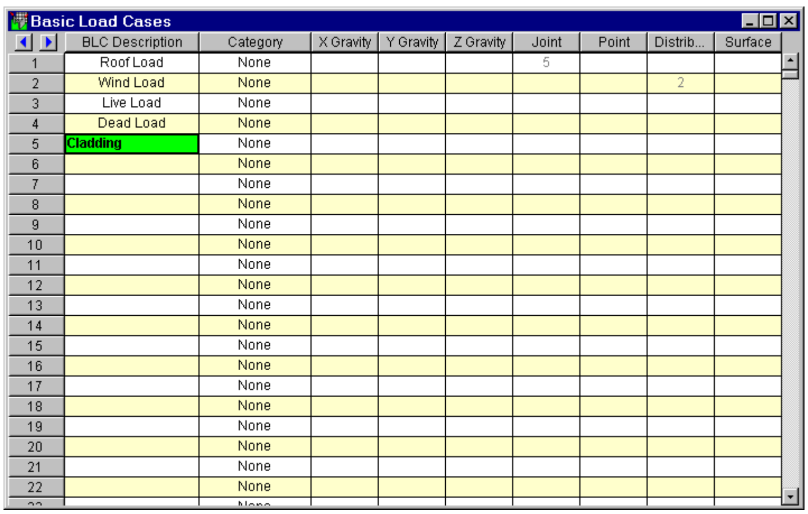
Let's view the **Basic Load Cases** spreadsheet by pressing [Basic Load Cases]

Let's enter a description for each of the BLC's we'll be using. The active cell should be in the first row and first column, which is the **BLC Description** field for BLC 1.

Type:

> **Roof Load, [ENTER]**
>
> **Wind Load, [ENTER]**
>
> **Live Load, [ENTER]**
>
> **Dead Load, [ENTER]**
>
> **Cladding**

So we've given descriptions to five BLC's, two of which we have already defined. Keep in mind these descriptions are strictly for your benefit. RISA-3D doesn't require a description.


Basic Load Cases

	BLC Description	Category	X Gravity	Y Gravity	Z Gravity	Joint	Point	Distrib...	Surface
1	Roof Load	None				5			
2	Wind Load	None						2	
3	Live Load	None							
4	Dead Load	None							
5	Cladding	None							
6		None							
7		None							
8		None							
9		None							
10		None							
11		None							
12		None							
13		None							
14		None							
15		None							
16		None							
17		None							
18		None							
19		None							
20		None							
21		None							
22		None							

Now lets look at the loads in the spreadsheets. Notice that there is a "5" in the **Joint** column. This "5" represents our five joint loads in basic load case 1. The "2" in the **Distributed** column represents the two distributed loads in basic load case 2.

RISA-3D provides special interaction between these cells and the spreadsheets that hold the actual loads such that clicking the mouse on the cells will open the appropriate spreadsheet. This is why the numbers are gray, which means that these fields may not be edited.

Click on the cell that contains "5" in the first row.

The **Joint Loads** spreadsheet is opened for basic load case 1 (BLC 1). The joints and their loads are listed and of course may be edited. Note that the title bar of this spreadsheet verifies that we are looking at "BLC1".

Now back in the **Basic Load Case** spreadsheet click on the cell that contains the number “2”.

The **Distributed Loads** spreadsheet opens with the loads that were assigned to BLC 2. Note that the drop-down list on the right side of the **Window Toolbar** now contains “BLC 2:Wind Load”. To view distributed loads that belong to another BLC you can use this list to select the BLC of interest.

You may also click on blank cells in the **Basic Load Case** spreadsheet to open spreadsheets for load cases that have yet to be defined. This interaction provides an easy way for you to get an overall view of the loads with the **Basic Load Cases** spreadsheet and then quickly move through the loads you need to work with.

We have yet to define the last three loads listed on the **Basic Load Case** spreadsheet. Also we haven’t discussed the **Category** and the **Gravity** columns. These will be addressed in Part Two of the tutorial.

Saving Your Data

At this point we’ve defined a 2D frame with loads. Let’s save the model.

Select **Save** from the **File** menu and name the file “Tutorial Part 1”. (We will assume that you have installed RISA-3D in the c:\risademo directory and that the file will be saved there as well.)

If you or someone else has previously been through this tutorial you may get an overwrite warning for this file; if you do, select **Yes** when prompted for whether the file should be overwritten.

This concludes the first part of the tutorial.

If you wish to press on with part 2 of the tutorial right now, select **Single View** from the **Window** menu and jump ahead to the section “**Continuing On**” now.

If you wish to exit RISA-3D and do part 2 later, select **Exit** from the **File** menu.

RISA-3D Tutorial, Part 2

First, restart the RISA-3D demo.

Now read in the file by selecting **Open** from the **File** menu.

In the **File Open** dialog select the file named "Tutorial Part 1" and click **Open**. Close the **Global Parameters** window by clicking **Close** and we will begin working in the model view.

Continuing On

In this part of the tutorial we're going to take the 2D frame we defined in the first part of the tutorial and use RISA-3D's replication features to expand it into a 3D frame. We will also add some plates to the model before solving it and looking at RISA-3D's result browsing features.

On the **Window Toolbar** select the **Drawing Toolbar** button and the toolbar is displayed.

Copying Model Elements

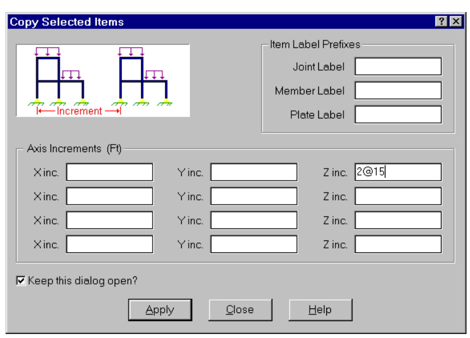
RISA-3D allows you to copy selected portions of the model and we will use this feature to turn our two-dimensional model into a three bay structure.

Click the **Copy** button to open the **Copy Selected Items** dialog. (Note that the copy button has a red arrow.)

This feature is used to copy the currently selected portion of the model in increments in any or all three of the global directions.

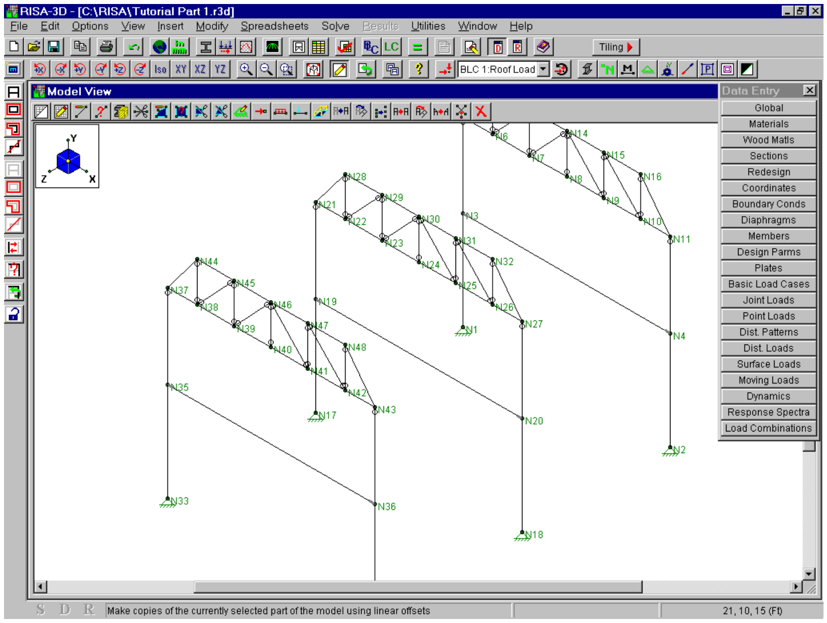
You should see three fields each for increments in the X, Y and Z directions. You can use the "@" symbol to specify multiple equal increments. We'll replicate our model in the Z direction.

In the section marked **"Axis Increments"** click in the top right field, which is labeled "Z inc", and type **"2@15".**



Click **Apply** and then click **Close**.

The screen should now look like this:



Note that in addition to copying the nodes and members, we have copied the loads as well although they are not shown here. Later we will demonstrate how to copy loads in another way. We will zoom out after drawing in some beams to connect the frames

To define a new section before we start drawing click on the **Sections** spreadsheet button Sections . Once the spreadsheet is open press ENTER four times which will create a fifth line in the spreadsheet. Now type:

BM-Z, [TAB], [TAB], W12X19

Here is what the spreadsheet should look like:

Member Section Sets

	Label	Material	Shape	A [In2]	SAy	SAz	Iyy [In4]	Izz [In4]	J [In4]	T/C
1	COL	STL	W12X40	11.8	1.2	1.2	44.1	310	.95	
2	CHORD	W2	DFLARSEL_4X8	25.375	1.2	1.2	25.904	111.148	72.244	
3	WEB	W1	DFLARSEL_4X4	12.25	1.2	1.2	12.505	12.505	21.134	
4	BEAM	STL	W12X19	5.57	1.2	1.2	3.76	130	.18	
5	BM-Z	STL	W12X19	5.57	1.2	1.2	3.76	130	.18	

Note that all three of our steel section sets are using a W12 shape. These are just initial shapes that RISA-3D will optimize for us a little later.

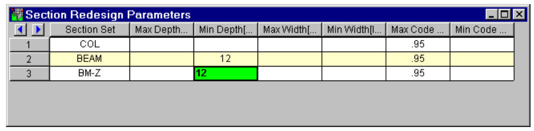
Close the spreadsheet window by clicking on the button in its title bar.

We also need to add this new section set to the list of section sets to be optimized, so now click Redesign to bring up the **Section Redesign** spreadsheet and then press ENTER twice to create a new line in the third row.

The active cell should be in the first column which is the S**ection Set** field.

Click the down arrow and select the "BM-Z" section. Now press [TAB] twice and enter "12" for the **Minimum Depth**.

Here is what you should have:



Section Redesign Parameters

	Section Set	Max Depth...	Min Depth[...	Max Width[...	Min Width[...	Max Code ...	Min Code ...
1	COL					.95	
2	BEAM		12			.95	
3	BM-Z		12			.95	

Close the spreadsheet by clicking the close button.

Now click the **Draw Members** button on the **Drawing Toolbar**. In the **Set Member Properties** dialog change the Section Set to the new section by clicking the down arrow and selecting "BM-Z".

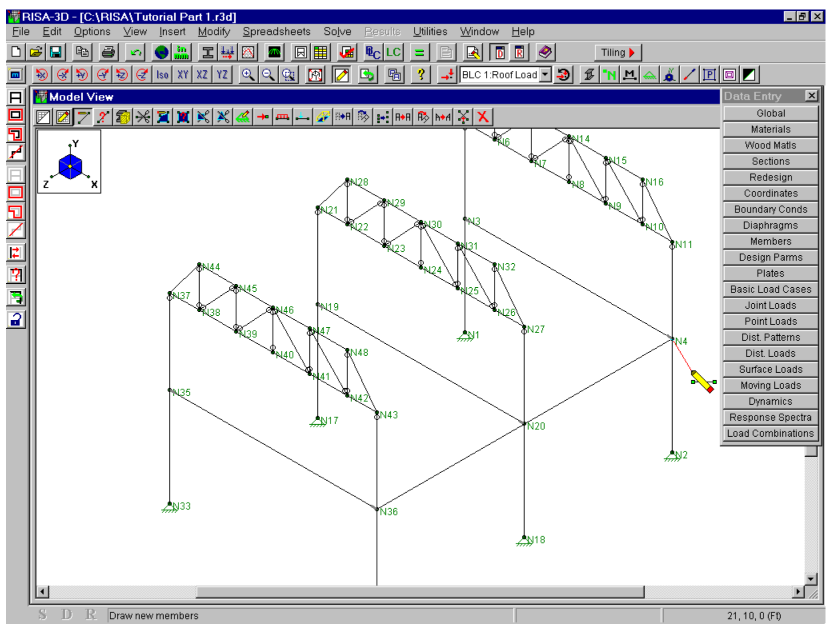
The end releases should already be set to **Fully Fixed** for both ends.

Click **OK** and we are ready to draw.

We will draw one member, load it, and then demonstrate how to combine the selection and copy features to define the rest of the members to connect the frames.

Click on the joint that is labeled N36 to define the I-end of the member and then click on the node labeled N4 to define the J-end.

Your screen should look like this:



RIGHT click to pick up the pencil and stop drawing and RIGHT click again because we are finished drawing.

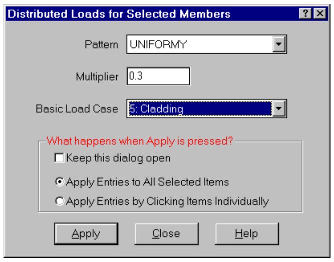
Next we will load the member, copy it to other locations and then merge so that it is connected to the center frame. First we will select the member so that we can work with it.

Click the **Unselect All** button and then click on the member that we just drew so that it is the only member that is selected.

It should be the only member drawn in black and everything else is gray.

Now click the **Distributed Loads** button. This will be a 300plf load in the vertical direction so set the **Pattern** to "UNIFORMY" by clicking on the down arrow in the first field. Next, change the value in the **Multiplier** field to "0.3".

Now click on the down arrow in the **Basic Load Case** field and select **"5: Cladding"** and clear the **Keep this dialog open** checkbox.



Click **Apply** and the result is the newly drawn member with the distributed load.

Next we will copy the member, and it's load, to two other locations.

Now click on the **Copy** button again. Clear the **Z inc** entry and click in the **X inc** field and type:

-21

Click **Apply.**

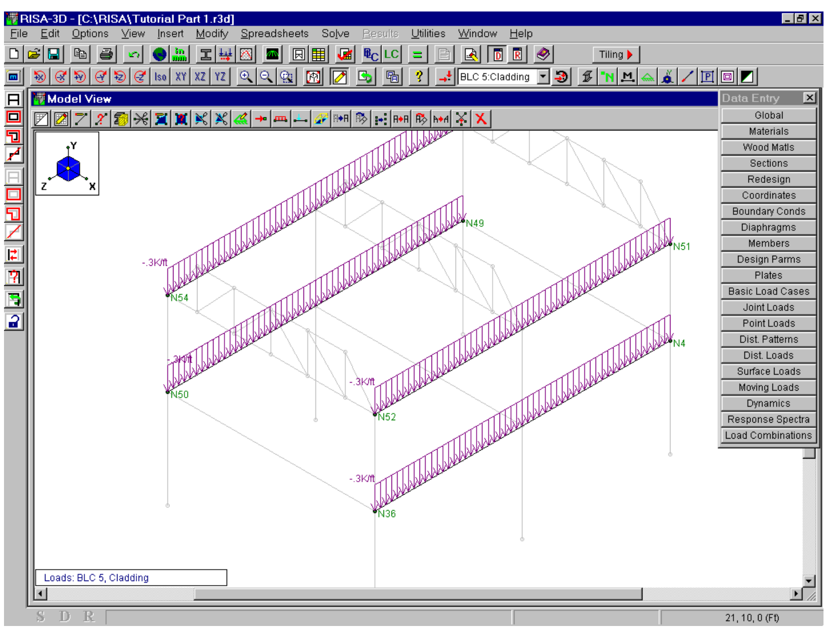
The member and load is copied to the other side of the frame, although you may not be able to see it as it is behind the **Copy** dialog. Now we will copy the members up 8.5ft to the tops of the columns and close the dialog.

On the **Copy** dialog, clear the **X Inc.** entry and click in the **Y inc.** field and type:

8.5

Click **Apply** and then click **Close**.

The frames are now connected with members that have a cladding load.



Turn off the loads by clicking on the **Toggle Loads** button on the **Window Toolbar**.

We will perform another model merge to split these members and connect them to the center frame.

Select **Full Model Merge** from the **Utilities** menu. Click **Apply** and you will be informed that duplicate and intermediate nodes have been corrected. Click **OK** and we are back to drawing.

Now we're going to add a floor composed of plate elements. We'll use concrete as the material, so let's go back to the spreadsheet and define the concrete properties.

Click Materials to open the spreadsheet and press [ENTER] to create a new line. Starting in the **Label** field, type:

> **Conc, [TAB], 3500, [TAB], [SPACEBAR], [TAB], [TAB], [TAB], .15, [TAB]**

Pressing the SPACEBAR to clear the shear modulus field causes RISA-3D to determine the value from the Young's modulus and Poisson's ratio.

Close the spreadsheet by pressing [ESC].

Using Criteria Select

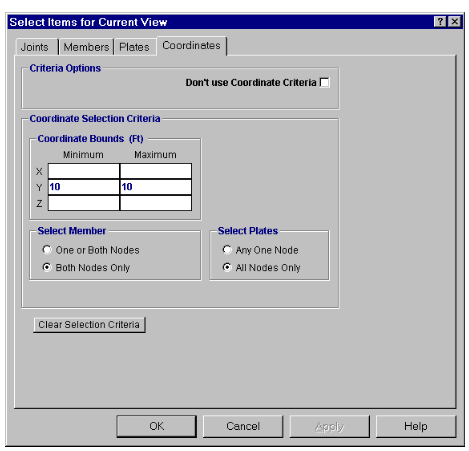
We will use **Criteria Selection to** select the nodes that we will draw to.

Unselect the entire model by clicking on . Now click the **Criteria Selection** button. Once the dialog is open first select the **Members** tab and check the box at the top that is labeled "**Don't Select / Unselect Members**". Next select the **Coordinates** tab.

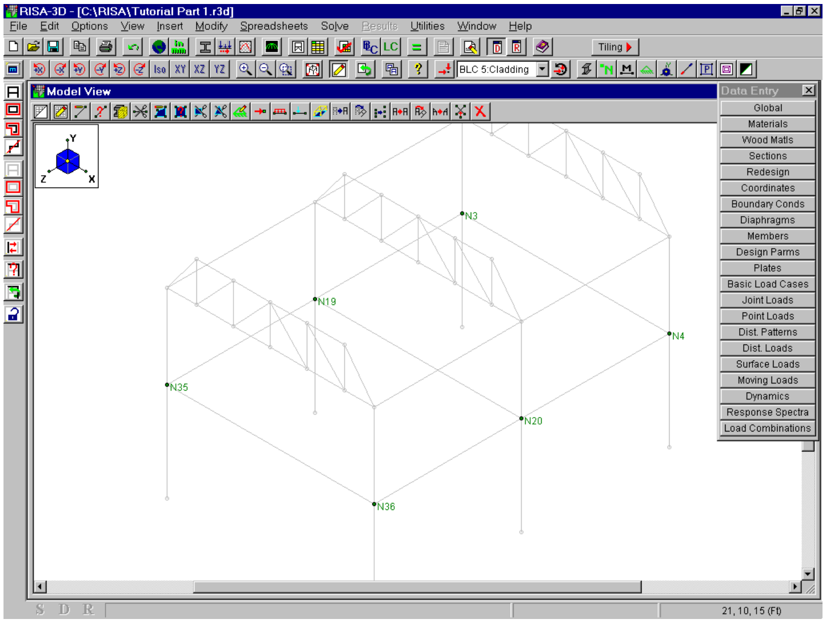
We will use the coordinate bounds to select the nodes at an elevation of 10ft.

Click in the first Y coordinate field and enter "10" for the minimum value. Now press TAB to move to the maximum Y coordinate and enter "10" there as well.

The dialog should look like this:



Click **OK** and the screen now looks like this:



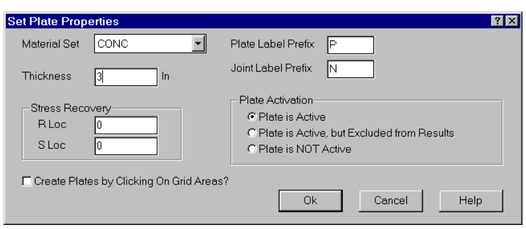
Plate/Shells

Click on the **Draw Plates** button.

Now we're ready to draw the plates. The entries on this panel are pretty straightforward; you define a material set and thickness for the plates to be drawn. The "R" and "S" entries have to do with where on each plate the plate stresses are calculated; these will almost always be set to 0. R and S are discussed in more detail in the Help file.

Go to the **Material** field click the down arrow to change to the "CONC" material set. Then click in the **Thickness** field and change this to "3".

The dialog should look like this:



Now click **OK** to begin drawing the plates.

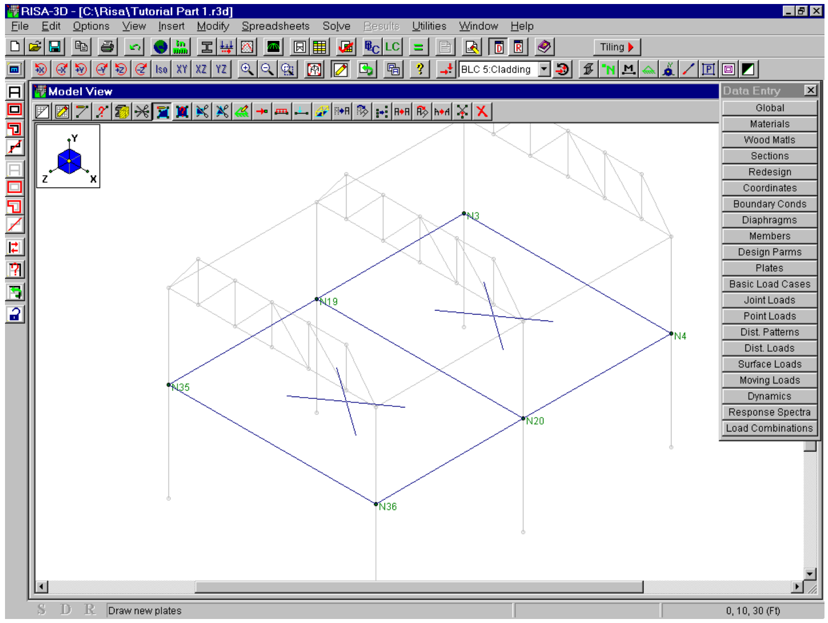
To draw the first plate, click on node N35, then move to node N36 and click again.

As you move the mouse from the node you'll notice the double lines attached to the mouse cursor, indicating that a plate is being drawn.

Now move to node N20 and click, then move to node N19 and click again and the first plate appears in wireframe form with an "X" in the middle pointing to the corners.

To draw the second plate, click on node N19, then N20 , then node N4 and lastly on node N3.

You should see this:



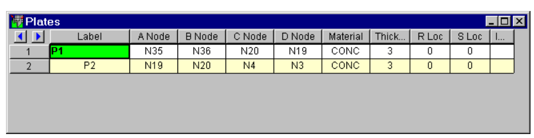
We now have our plates which we will have RISA-3D submesh for us in a minute.

Let's first take a moment to learn about the RISA-3D plate element. Let's go look at the plate data in the spreadsheet.

Select **Plates** from the **Spreadsheets** menu.

The **Spreadsheets** menu is an alternate method to using the **Data Entry** toolbar when opening spreadsheets.

The following spreadsheet is now open:


Plates

	Label	A Node	B Node	C Node	D Node	Material	Thick...	R Loc	S Loc	I...
1	P1	N35	N36	N20	N19	CONC	3	0	0	
2	P2	N19	N20	N4	N3	CONC	3	0	0	

RISA-3D's plate/shell finite element allows you to easily model shear walls, floors, domes, shells, spread footings, mat foundations and most any other surface structures. We will refer to these elements as "plate" elements, but they would more accurately be called flat shell elements because they model membrane and transverse shear as well as plate bending behavior.

The A, B, C and D node entries are used to define the 4 corner nodes of a quadrilateral element. (To define a 3-node element leave the D-node entry blank or make it the same as the C-node.) The nodes must all lie on the same plane and be entered in sequence, in either a clockwise or counter-clockwise direction.

See the help file for information on plate local axes and stress recovery options.

The material set label links the plate with the desired material set. The thickness entry is the desired thickness for the element, which is constant over the entire element. The last field is the "Inactive" flag, which works the same way that it does for the members.

Plate Mesh

Click to close the spreadsheet and go back to the model view.

One further topic to be discussed on the plates is "mesh size". How many plate elements do you need to use to get accurate results? As with any finite element, the RISA-3D plate element gives more accurate answers as you use more plates to model a given surface. On our web site (www.risatech.com) we present some parametric studies that demonstrate this along with some design examples for shear walls, horizontal diaphragms and spread footings.

Our studies have shown you should try to have a 4x4 mesh of plates between points of support to get highly accurate results. Looking at the plot, the columns represent the points of support for the plates, so we currently have only a 1x1 plate mesh between support points. In fact, what we have now (two large plates both supported at all four corners) would give us poor results if we were to solve this model in its current state.

RISA-3D provides an extremely easy way to refine a plate mesh. This feature allows us to draw one plate where we need many just as we took advantage of the model merge feature and drew one member where we needed many.

Submesh the plates by clicking on the **Submesh Quads** button.

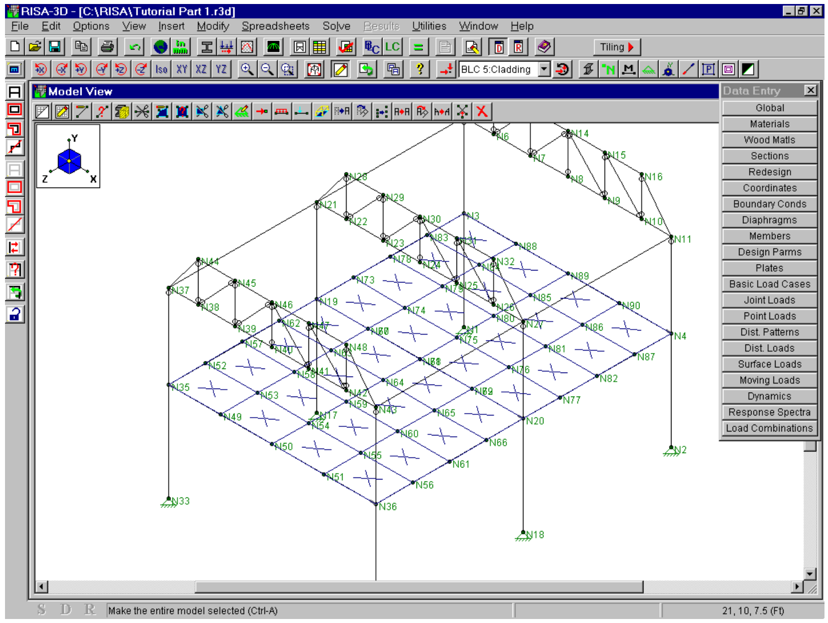
Since we currently have a 1x1 mesh between support points and we want a 4x4 mesh, we need to submesh each plate into a 4x4 grid of plates.

The default entries will divide our plates into a 4x4 submesh, just click **Apply** and then click **Close**.

Both original plates have now been submeshed into 4x4 grids of smaller plates.

The plates are displayed as rectangles with an "X" in the center. You select and unselect individual plates just like you do with members, by pointing at them with the mouse and clicking. Naturally the select/unselect tools apply to the plates as well.

Click the **Select All** button and you should now see:



Notice that submeshing the plates has added new nodes along the existing beam members. Currently these nodes are not attached to the beams. It's time we did another merge to integrate all these new plates and nodes into our model.

Click on the **Drawing** toolbar. Now click **Apply** and the merge results are presented. Clear the results by clicking **OK** and close the **Model Merge** dialog by clicking **Close**.

The lower members that we drew with the "BM-Z" section were divided into 4 members each. At this point we've integrated the plates thus completing the structural model. We will look at some plate graphics and then define some more loads before solving the model.

Plate Graphics

On the **Window** toolbar first click on the **Node Labels** button which will remove the labels from the model view. Now click on and select the tab marked **"Plates"**.

These options control the graphic display of the plates and to a certain degree mimic the member display buttons. **Wireframe**, **Color Filled**, **Rendered** and **Don't Draw** are the four main options across the top and are mutually exclusive. These determine how the plates are displayed. Beneath each of these are additional sub-options that may be included.

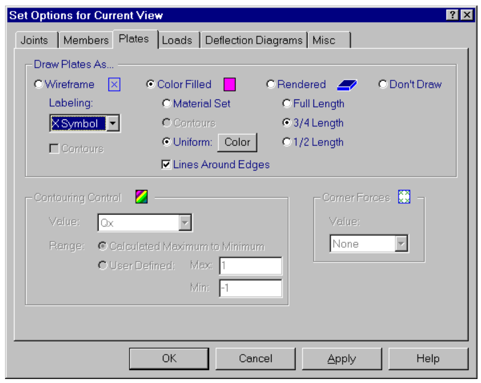
For example we are currently viewing the plates in wireframe form. When this option is on, you may additionally display the plate labels, material set, numerical thickness or local axes alongside each plate. Remember that the indented sub-options below each main option don't do anything unless the main option is selected.

The **Color Filled** button causes the plates to be “filled” when displayed. They will be plotted as a surface. To see what this does, lets try it.

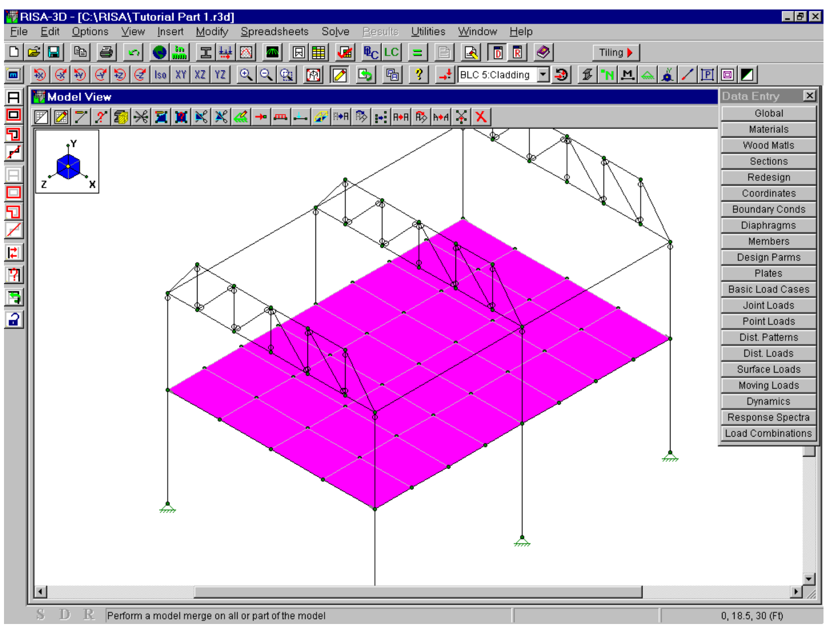
Click the button next to **Color Filled** to select this option.

The **Color Filled** option has three sub-options beneath it. The **Material Set** option will cause the plates to be color-coded according to which material set is assigned to them. The **Contours** only applies to solved models and may not be selected now. The **Uniform** option is currently selected and simply allows you to specify a color for the plates in the model view.

The dialog looks like this:



Now click **OK** to apply the choices to the model view and close the dialog.



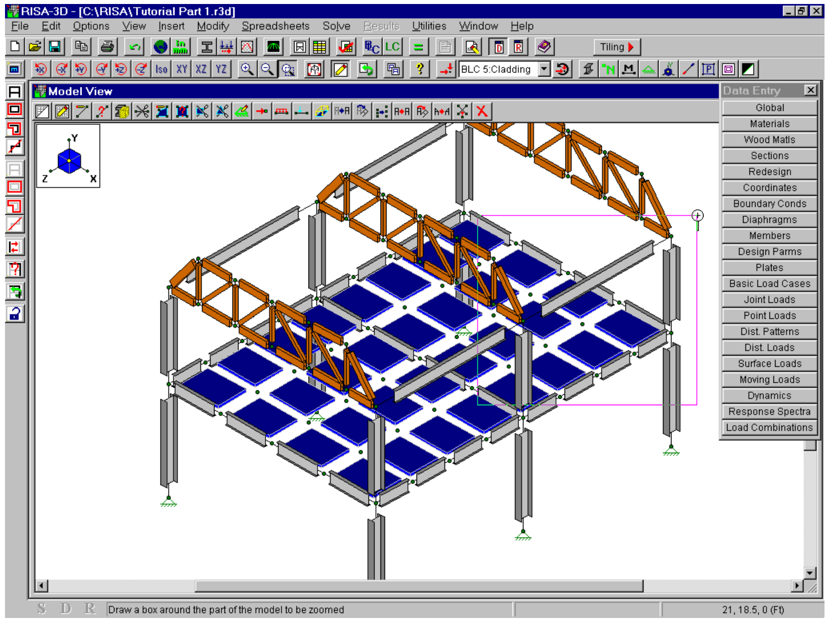
As you can see, this gives a “hidden line” display of the plates, meaning anything behind the plates is hidden from view.

We have seen the plates in wireframe and colored forms. The remaining option provides rendered plates.

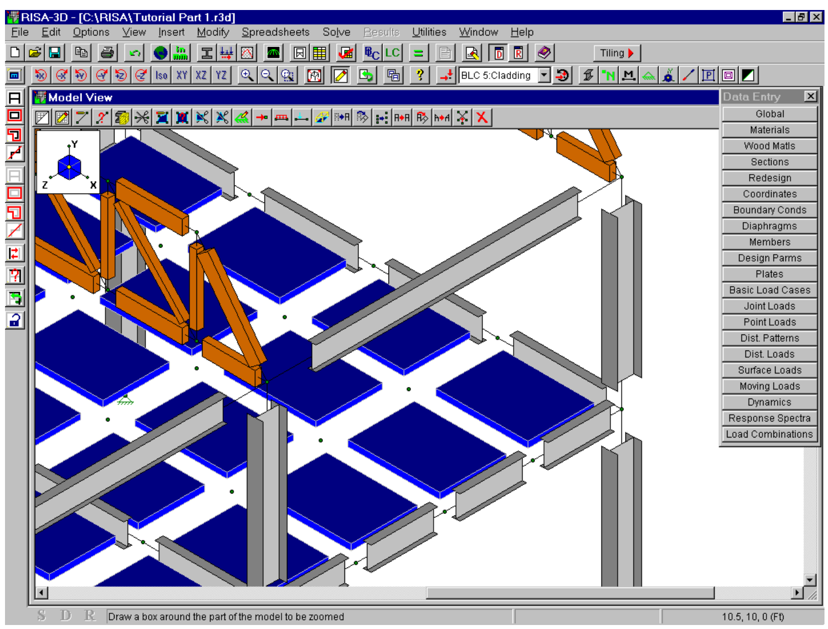
Click on  which will render the plates as well as the members.

We’ll take a closer look at the rendered plates.

Click on the **Box Zoom** button on the **Window** toolbar and then draw the zoom box like this (remember, move to one corner of the box, click and hold the left mouse button, then drag to the opposite corner and release):



To see:



The plates are shown with a 25% shrink factor. The shrink factors only apply to the length and width of the plates, not the thickness. The full thickness is always shown. They can also be shown full size or shrunk 50%. These choices must be accessed in the **Set Plot Options** dialog.

Scroll the image horizontally to see just how responsive the graphics are. There is essentially no redraw time. Now return to the full plot by clicking on . Click on again and the rendered view is turned off.

Surface Loads

The next thing we need to do is to put surface loads on the plates. We'll assign these surface loads to Basic Load Case 3, the "Live Load".

On the **Drawing** toolbar click the **Surface Loads** button.

The first field in the **Surface Loads** dialog is the **Direction** for the surface loads to be created. The valid surface load directions are:

x,y,z : Applied in the direction of the plate local axes

X,Y,Z : Applied in the direction of the global axes

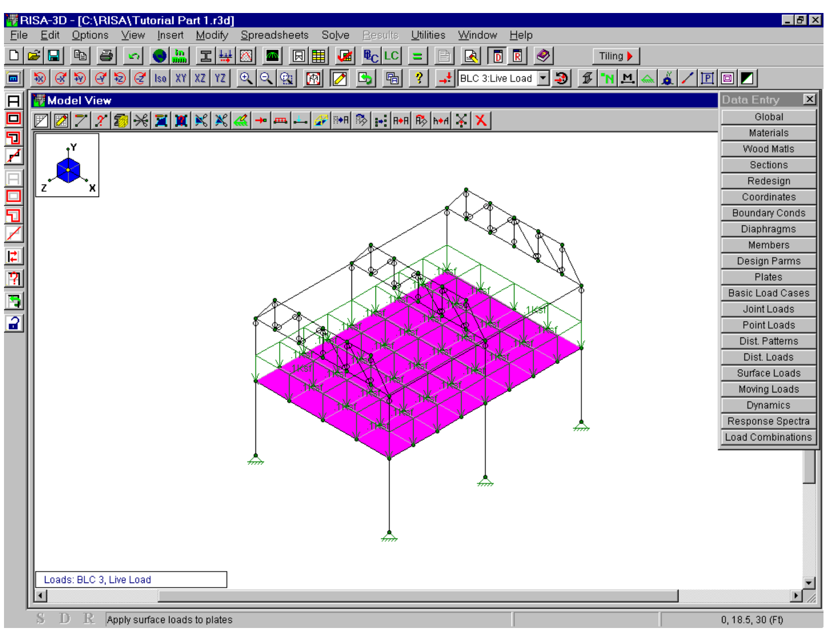
L,V,H : Projected loads in the X,Y or Z direction

We will stick with the lower case "z" which will give us loads normal to the plate. Choosing an upper case "Y" would yield the same result.

Click in the **Magnitude** field and enter "0.1". Click the down arrow in the Basic Load Case field and choose "3: Live Load".

This will apply a uniform .1ksf (100psf) load to all selected plates, as part of Basic Load Case 3. The surface load is applied uniformly over the surface of the plate.

Click **Apply** and the loads are created and displayed. Now click **Close** to exit the dialog.



Copying Loads

Now lets finish up by defining the dead load so we can solve the model.

Click on Basic Load Cases.

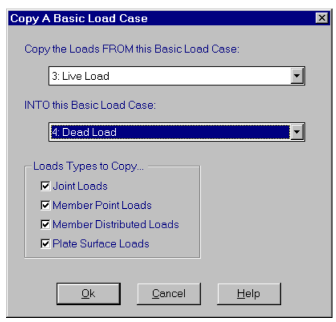
The spreadsheet now shows the surface live load on 32 plates and the cladding load on 20 members.

You may also copy entire basic load cases from one to another. After taking the time to define a complex load case you may realize that another load case is very similar to it. In such a situation it is might be best to copy the first case and then modify the copy rather than start from scratch.

We will use the copy feature to create a 90psf dead load from the 100psf live load that we just created.

Click the **Copy BLC** button and you are presented with the **Copy Basic Load Case** dialog. Click on the down arrow in the first field and select "3: Live Load". Click on the down arrow in the second field and select "4: Dead Load".

The dialog should look like this:



Click **OK** and there is now a "32" in the **Surface** column of the Dead Load case. Click on this "32" to open the **Surface Loads** spreadsheet.

We now need to change this load from 100psf to 90psf.

We will use the **Block Math** feature to change the magnitude of the load. First we will select all of the cells in the **Magnitude** column.

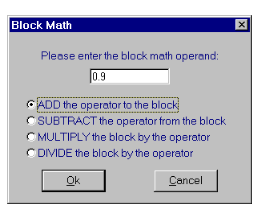
Click on the column heading for the **Magnitude** (this is the gray area at the top of the column that contains **"Magnitude…")**.

The entire column is now selected and is colored magenta.

Now click on the **Block Math** button to open the **Block Math** dialog.

We want the dead load to be 90psf, or 90% of the copied live load.

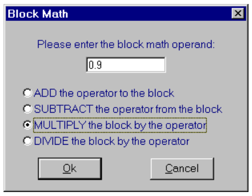
Enter "0.9" and the dialog looks like this:



Click **OK** and the surface loads now have a magnitude of 1.0ksf (1000psf).

Oops, we forgot to choose **Multiply** and have *added* 0.9ksf instead. We can easily undo this mistake or any other.

Click on the **Undo** button on the **RISA** Toolbar to undo our mistake. Again click on the spreadsheet column heading for the **Magnitude.** Click on the **Block Math** again and enter “0.9” in the field and select the **Multiply** option.



Click **OK** and the surface loads now have a magnitude of 0.09ksf (90psf). Press [ESC] to close the **Plate Surface Loads** spreadsheet.

Self-Weight

Now all we have to do is include the self-weight of the structure and we are finished modeling. We will add it to the same load case that we placed the surface dead loads. Simply entering a factor in the **Y Gravity** column on the Dead Load case will do this.

Click in the **Y Gravity** column on the fourth row and type:

-1

This is how you tell RISA-3D to calculate and include the self-weight of the model. The factor entered is a multiplier for the self-weight. In this case, we used “-1” because we want the full self-weight applied downward (the negative Y-direction). You can apply self-weight in the X, Y and/or Z directions.

Load Categories

The last thing we will do on the **Basic Load Case** spreadsheet is assign each load case to a category. Categories are especially helpful in bringing all of your loads together. In this model the loads are quite manageable and we 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. Being able to categorize the loads makes it easier to later combine them for solutions.

Click in the first cell of the **Category** column and then click on the down arrow in that cell to see the list of categories. Choose **“RLL (Roof Live Load)”**.

Click in the next cell down which represents BLC 2 and select the down arrow again and pick the **“WL (Wind Load)”** category.

In the same manner assign the third load case to the **“LL (Live Load)”** category and then assign BLC 4 and BLC 5 to the **“DL (Dead Load)”** category.

The spreadsheet is complete and looks like this:

Basic Load Cases

	BLC Description	Category	X Gravity	Y Gravity	Z Gravity	Joint	Point	Distrib...	Surface
1	Roof Load	RLL				15			
2	Wind Load	WL						6	
3	Live Load	LL							32
4	Dead Load	DL		-1					32
5	Cladding	DL (Dead Lo						20	
6		None							
7		None							
8		None							
9		None							
10		None							
11		None							
12		None							
13		None							
14		None							
15		None							
16		None							
17		None							
18		None							
19		None							
20		None							
21		None							
22		None							

Close it by clicking the ☒ in the titlebar.

Load Combinations

Our model and loads are now defined, so let's solve it and take a close look at the results handling features of RISA-3D. In order to solve the model we must enter load combinations that we wish to be solved.

Let's go to that spreadsheet by selecting Load Combinations from the **Data Entry Toolbar.**

Most standard code combinations are already built into RISA-3D. We will apply the ASCE minimum design loads and then modify them.

On right side of the **Window Toolbar** click the arrow on the drop down list and select the third choice: "ASCE 1995 ASD". Now click the **ADD** button next to the list. Stretch the bottom of the window down to view the 20 combinations that were created.

Load Combinations

	Description	Env	WS	PD...	SR...	CD	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor
1	ASCE 1	☑	☐			1	DL	1								
2	ASCE 2 (a)	☑	☐			1	DL	1	LL	1	RLL	1				
3	ASCE 2 (b)	☑	☐			1	DL	1	LL	1	SL	1				
4	ASCE 2 (c)	☑	☐			1	DL	1	LL	1	RL	1				
5	ASCE 3 (a)	☑	☐			1	DL	1	WL	1						
6	ASCE 3 (b)	☑	☐			1	DL	1	WL	-1						
7	ASCE 3 (c)	☑	☐			1	DL	1	EL	1						
8	ASCE 3 (d)	☑	☐			1	DL	1	EL	-1						
9	ASCE 4 (a)	☑	☐			1	DL	1	LL	.75	RLL	.75	WL	.75		
10	ASCE 4 (b)	☑	☐			1	DL	1	LL	.75	RLL	.75	WL	-.75		
11	ASCE 4 (c)	☑	☐			1	DL	1	LL	1	RLL	1	EL	1		
12	ASCE 4 (d)	☑	☐			1	DL	1	LL	1	RLL	1	EL	-1		
13	ASCE 4 (e)	☑	☐			1	DL	1	LL	.75	SL	.75	WL	.75		
14	ASCE 4 (f)	☑	☐			1	DL	1	LL	.75	SL	.75	WL	-.75		
15	ASCE 4 (g)	☑	☐			1	DL	1	LL	1	SL	1	EL	1		
16	ASCE 4 (h)	☑	☐			1	DL	1	LL	1	SL	1	EL	-1		
17	ASCE 4 (i)	☑	☐			1	DL	1	LL	.75	RL	.75	WL	.75		
18	ASCE 4 (j)	☑	☐			1	DL	1	LL	.75	RL	.75	WL	-.75		
19	ASCE 4 (k)	☑	☐			1	DL	1	LL	1	RL	1	EL	1		
20	ASCE 4 (l)	☑	☐			1	DL	1	LL	1	RL	1	EL	-1		

The combinations that were entered in the spreadsheet may be edited. We will explain all of the fields before going further.

The first field, the description, is strictly for your reference. You may enter any descriptive label that you wish and it will be displayed with the results when the load combination is solved.

The next five fields are flags for special instructions for the load combinations. We will mention them here briefly although we will not use them all. See the help file for more information about these options.

The **Env** checkbox indicates whether this combination is to be included in the envelope solution. RISA-3D allows you to solve a single load combination or an enveloped solution. The envelope solutions where *all* combinations with a checkmark in their **Env** fields are solved simultaneously, with the maximum and minimum results from all the combinations reported. All of our combinations are currently included in the envelope.

The field labeled **WS** indicates whether the Wind/Seismic allowable stress increase allowed by the AISC 9th Edition specifications is to be used. Remember, the increase factor (usually 1.333) is entered on the **Global** window. This field is also used to indicate whether the seismic provisions of the LRFD code are to be used, if LRFD design is being done.

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.

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 define the actual combinations with pairs of **BLC** columns and **Factor** columns. The values should look familiar to you since they are just as they are listed in the code.

Instead of using categories, as we have here, you may also list loads by the BLC number. For example instead of using “LL” we could call the same load by using the number “3”. Other options include nesting one combination within another and including response spectra results. See the on-line help file for more on this.

We will make some changes before moving on. Since we do not have an earthquake load, snow load or rain load we will remove these combinations from the list. We will do this by deleting lines from the spreadsheet.

Select lines 7 and 8 by clicking the mouse on the button on the left side of row 7 (the button with a “7” on it) and drag the mouse to the button for row 8 before releasing.

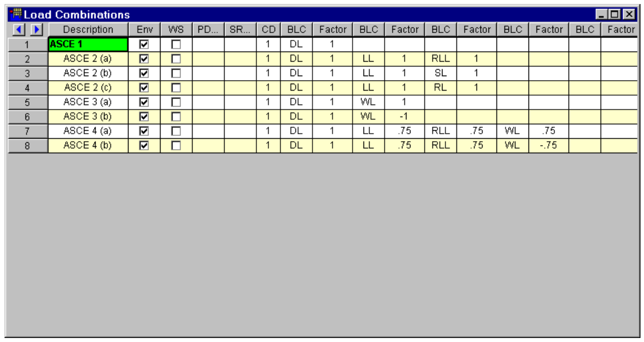
Both lines should be shaded yellow. If they are magenta you selected cells in the spreadsheet, not the header on the very left. If you have selected the rows correctly the line delete button on the **Window Toolbar** should be active.

Once you have the two lines selected, click the **Delete Lines** button on the **Window Toolbar**.

The lines are deleted leaving you with 18 combinations. Next we will delete the last 10 combinations.

Select lines 9 through 18 by clicking the mouse on the row button for row 9 and drag the mouse to row 18 before releasing. Click the **Delete Lines** button on the **Window Toolbar**.

The **Load Combinations** spreadsheet should look like this:



Load Combinations

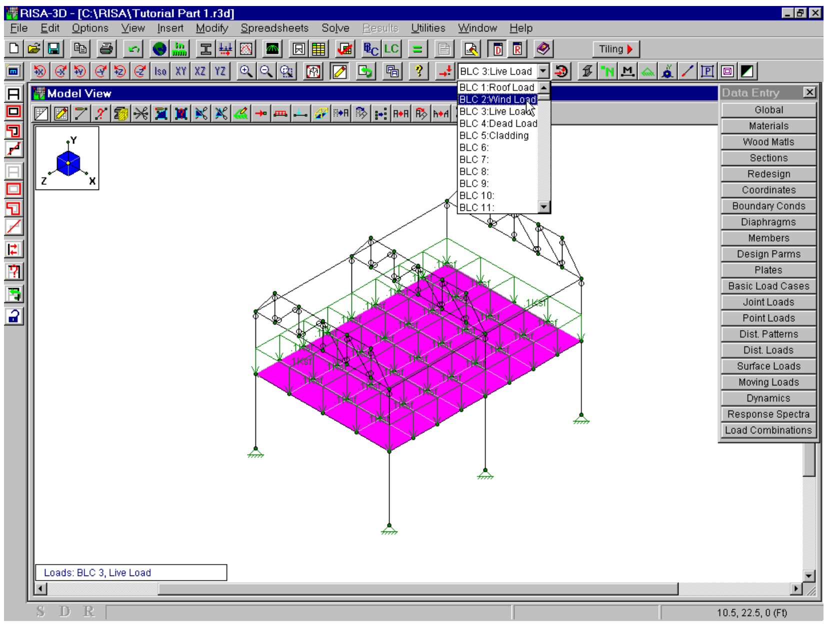
	Description	Env	WS	PD...	SR...	CD	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor
1	ASCE 1	☑	☐			1	DL	1								
2	ASCE 2 (a)	☑	☐			1	DL	1	LL	1	RLL	1				
3	ASCE 2 (b)	☑	☐			1	DL	1	LL	1	SL	1				
4	ASCE 2 (c)	☑	☐			1	DL	1	LL	1	RL	1				
5	ASCE 3 (a)	☑	☐			1	DL	1	WL	1						
6	ASCE 3 (b)	☑	☐			1	DL	1	WL	-1						
7	ASCE 4 (a)	☑	☐			1	DL	1	LL	.75	RLL	.75	WL	.75		
8	ASCE 4 (b)	☑	☐			1	DL	1	LL	.75	RLL	.75	WL	-.75		

Look over your load combinations to make sure they are the same as above and then close the spreadsheet by pressing ESC.

Displaying Loads

RISA-3D provides easy ways to view your loads.

On the **Window Toolbar,** just to the right of the **Toggle Loads** button, click on the down arrow in the drop-down list and choose "BLC 2:Wind Load".

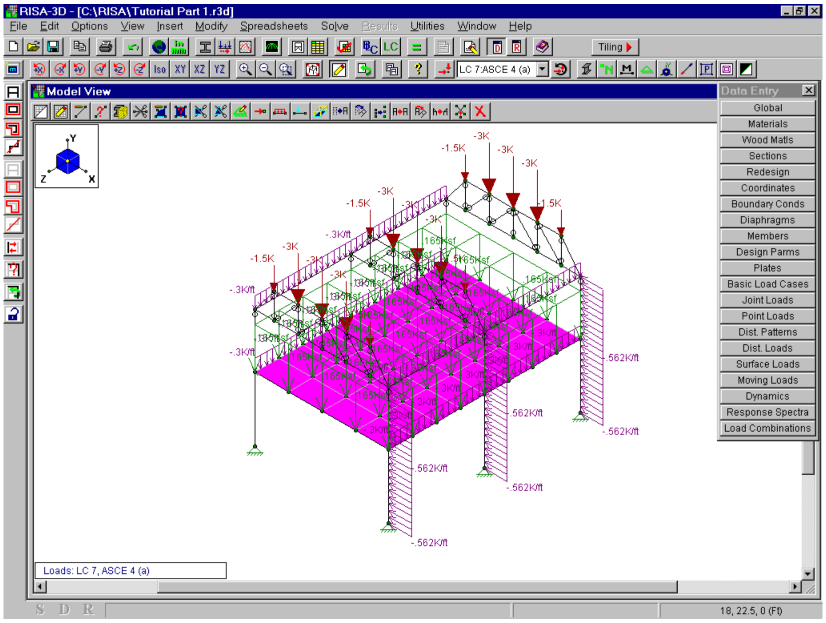


The plotted loads change to the wind loads on the columns.

You may go through all of the Basic Load Cases in this manner if you like. When you are finished viewing the BLC's, click the **Switch Loads** button one time.

The drop-down list box on the **Window Toolbar** now displays all of the load combinations instead of the basic cases and the view now shows the loads as they are factored in the load combinations.

Choose "LC 7:ASCE 4 (a)" from the drop down list.



This is a great way to verify that our model is solving the problem that we want it to solve. One thing to note here is that the magnitudes of the loads represent the factored combination. For example the displayed roof load magnitude is the combination of the dead and live roof loads multiplied by their factors.

When you are finished viewing the load combinations click the **Switch Loads** button once more.

Now the drop-down list on the **Window Toolbar** contains the Load Categories.

View the loads by category by selecting them from the list.

Saving the Data

Before we solve the model we'll save our data (a good habit to get into).

Select **Save As** from the **File** menu and this time name the file "Tutorial Part 2".

If you or someone else has previously been through this tutorial you may get an overwrite warning for this file; if you do, select **Yes** when prompted for whether the file should be overwritten.

Solving the Model

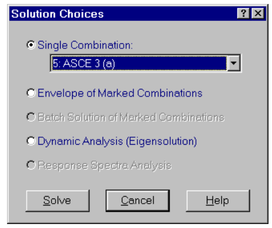
First we will solve a few single load combinations and then we will solve all of the combinations and look at the enveloped results.

Click on the **Solve** button and the **Solution Choices** dialog is presented.

The **Single Combination** option should already be selected.

Click on the down arrow and select "5: ASCE 3 (a)".

The dialog should look like this:



Click **Solve**.

RISA-3D reports the solution steps as they occur. When the solution is complete, you are presented with the **Joint Reactions** spreadsheet. You will notice that the reactions are summed at the bottom of the spreadsheet. The center of gravity of the vertical loads is reported in the last row.

Solution Spreadsheet Results

You may access the result spreadsheets from the **Results** menu or from the **Results Toolbar** which is placed on the right side after the solution is performed. Let's use the toolbar to quickly review the results. You may use the **Topic Help** and **Help** buttons for a more detailed explanation of the results. You may also look to the status bar for an explanation of the active cell. Take a moment to look over each spreadsheet. Use the scroll bars to view information not displayed in the window.

Select **Joint Deflections** from the **Results Toolbar**.

Select **Member Forces** from the **Results Toolbar**.

Notice the forces are listed for 7 locations on each member. This is because we specified 7 sections back on the Global window.

Select **Member Stresses** from the **Results Toolbar** to view the member stresses.

Select **Member Torsion** from the **Results Toolbar**.

These are the torsional stresses, including warping normal and shear stresses for the members that warp.

Select **Member Deflections** from the **Results Toolbar**.

These are the member span deflections. This spreadsheet also shows the deflections as a ratio of member length (the "L/y" ratios) providing an easy check against deflection criteria such as "L/360".

Select **Steel Code Checks** from the **Results Toolbar.**

Here we have the AISC code checks, I.e. ratio of actual to allowable stresses, per ASD 9th Edition criteria. Notice that the failing members are highlighted with red text. The allowable stresses, Cb and Cm values and controlling equation are listed. If we were doing LRFD based design, we would see the member strengths listed here (along with the code check value).

Select **Alternate Shapes** from the **Results Toolbar**.

These are the RISA-3D suggested redesign shapes (we'll use these later).

Select **Wood Code Checks** from the **Results Toolbar.**

These are the NDS code checks for our wood truss members. The factored design stresses and controlling equation are also listed.

We won't discuss the plate results just yet. We'll get to those a little later.

Select **Material Take-Off** from the **Results** menu to view the material take-off.

Browsing the Results

Select Member Forces from the **Results Toolbar** to go back to the **Member Forces** spreadsheet.

To review the results, you can page up and down or scroll up and down a line at a time. [Home] and [End] take you to the top and bottom of the results. You can use the **Find** feature to locate specific results and the **Sort** feature to sort the results. The **Exclude** feature allows you to hide results that are not of interest. Let's try some of these features.

Sorting the Results

We will use the **Sort** feature to sort the strong axis moment results.

Click in any cell in the last column, which is the **z-z Moment** column. Now click the **Sort** button on the **Window Toolbar** to access the sorting options. Select "**Abs Max to AbsMin**" as shown below:

Click **OK**.

You'll be returned to the Forces spreadsheet with the members sorted according to their absolute maximum z-axis bending moment value.

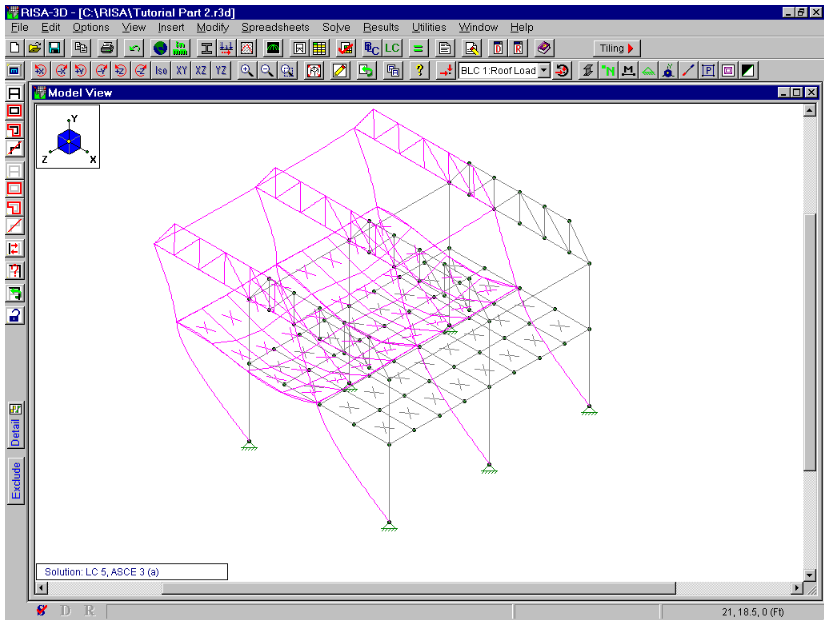
Solution Graphic Results

Since we've solved the model, we now have several more plotting options.

First select **Single View** from the **Window** menu and the spreadsheets will be closed. Then turn off the node labels by clicking . Finally, close the **Results toolbar** and the **Data Entry Toolbar** by clicking their close buttons.

Deflected Shape Plot

Click to Open the **Plot Options** dialog and select the **Deflection Diagrams** tab. Now select the option labeled "**Load Combination**" which will plot the combination we have solved. Check the box labeled **Include Undeflected Shadow** and click **OK.**



This is a fairly nice representation of how the model deflects under these loads, but now let's use some of RISA-3D's more advanced graphics features.

Click on the **Render** button on the **Window** toolbar.

It takes a second or two to create this image because of all the calculations RISA-3D has to do to produce it, but now you can really see what's happening with the model!

Animation

Let's go a step further and animate this deflected shape plot.

Click and then click Animate This Deflected Shape

A progress bar indicates that the animation is building. Depending on the speed of your computer, it may take a few seconds to build the animation since it is rendered. Behind the **Set Plot Options** dialog a new window is created with the animation.

Click **OK** on the **Set Plot Options** dialog to close it.

The animation window is behind the model view window.

From the **Window** menu select the last option which is "Animation of LC:5 ASCE 3 (a)" and the animation is brought back to the front. You may drag the window up to get a better view.

Once the animation begins, you can speed it up or slow it down. This provides a very vivid presentation of how the model is deflecting.

When you're finished watching the animation press the **Close** Close button.

Deflected shape animation is a powerful tool that will help you identify parts of the structure that have been incorrectly modeled or loaded and thus are not behaving correctly.

Color Coded Plot

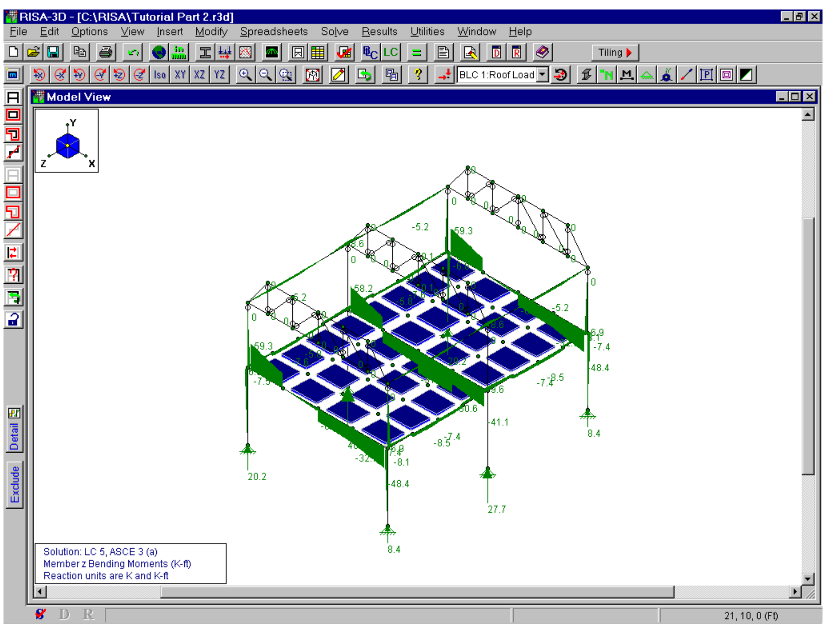
You can also display members and highlight areas of concern. We're now going to do a plot of the model with each member color-coded by axial stress.

Click to return to the options window. Turn off the deflected shape by selecting "**Don't Show Deflected Shape**" then click on the **Members** tab. Select **Color Coded** and then **Compression** from the **Color Basis** drop down list beneath it. Click **OK.**

The color legend is 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, we'll display moments and reactions in the model view.

Click on again and select **Wireframe** for the members. On the right side of the dialog is a section for **Member Results**. On the **Diagram** drop down list pick **z-z Moment** and then click the **Magnitudes** box so that it has a checkmark in it. Click the **Joints** tab and select the **Y Direction** and "**Include the Magnitude**" options.

Click **OK**. The moment diagrams and reactions are plotted with magnitudes.

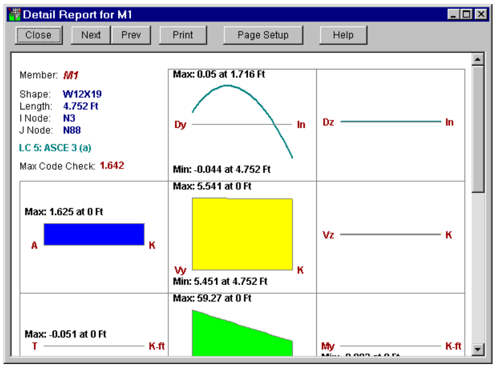


Remember that the columns are bending about their weak axis, which would require a y-Moment plot to view those diagrams.

Member Detail Report

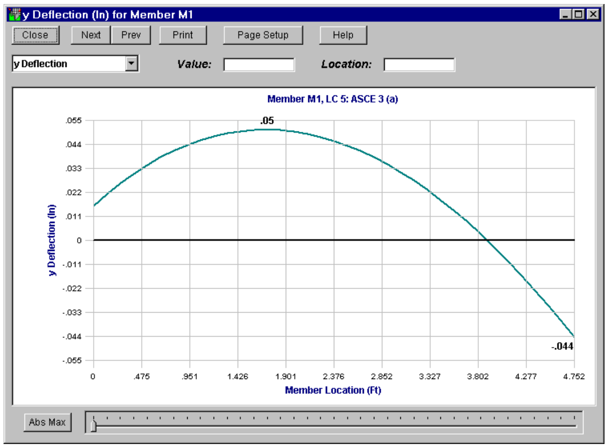
Next let's use the member detail report to take a closer look at some members.

From the **Results** menu, select **Members**‣**Member Detail**. Click **OK** and you will get this report on member M1 which is the crossbeam on the left:



This report gives you the ability to see a detailed report showing all the forces, stresses and deflections for any individual member. You can expand any of these diagrams for closer inspection by clicking on them.

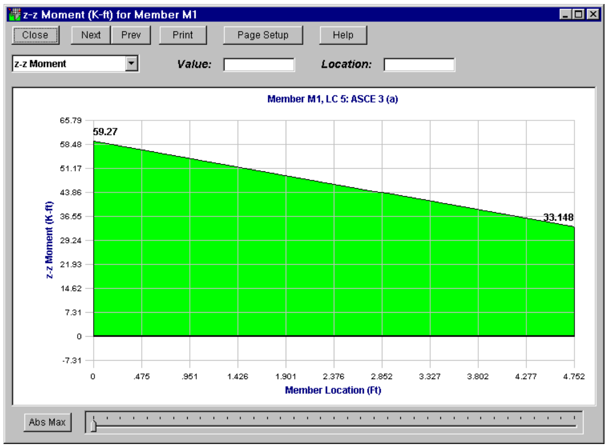
Click on the local y-axis deflection diagram, which is the first diagram in the middle column.



You should now be looking at the deflection diagram for member M1. You may click the **Abs Max** button to retrieve the absolute maximum value and it's location. The slider control to the right of the **Abs Max** button allows you to retrieve values at approximately 40 section locations.

The buttons across the top allow you to close the diagram, move to the same diagram for the previous member or the next member, or print the diagram. You may also access diagrams for other values without going back to the model view or the detail report so let's try this next.

Click the down arrow in the drop-down list box that now contains **"y Deflection"**. Select **"z-z Moment"** to get this diagram:



Feel free to try any of the options or view another diagram from the list. When you are finished click **Close** and you will be returned to the member detail report.

Scroll down to view code check information and you will see that the code check value exceeds the 1.0 limit along with the controlling equation and all of the design values. Click **Close** when you are finished.

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

On the **Selection Toolbar** click the **Member Detail** Detail button.

The mouse cursor will change to and you may now click on any member to open the report.

Try clicking on a few members; you may even open multiple reports at one time.

You may also open the detail report when reviewing the spreadsheets.

Select the **Results** menu and then click on **Members**‣**Steel Code Checks**. Click on any cell in the spreadsheet. Now click on this button on the **Window Toolbar**: Detail Report for Current Member.

We are finished with the detail reports and will now solve the fourth load combination.

Select **Single View** from the **Window** menu.

Plate Results

Click to turn on the **Results Toolbar** and select **Plate Stresses** from the **Results Toolbar** to view the plate stresses.

Here is what you should see:

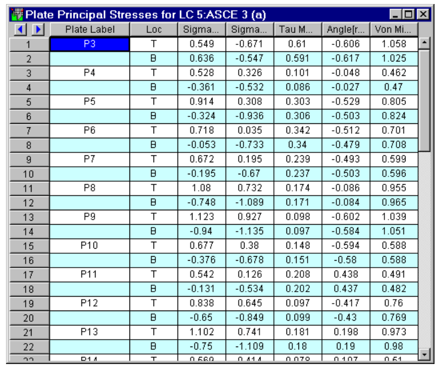


Plate Principal Stresses for LC 5:ASCE 3 (a)

	Plate Label	Loc	Sigma...	Sigma...	Tau M...	Angle[r...	Von Mi...
1	P3	T	0.549	-0.671	0.61	-0.606	1.058
2		B	0.636	-0.547	0.591	-0.617	1.025
3	P4	T	0.528	0.326	0.101	-0.048	0.462
4		B	-0.361	-0.532	0.086	-0.027	0.47
5	P5	T	0.914	0.308	0.303	-0.529	0.805
6		B	-0.324	-0.936	0.306	-0.503	0.824
7	P6	T	0.718	0.035	0.342	-0.512	0.701
8		B	-0.053	-0.733	0.34	-0.479	0.708
9	P7	T	0.672	0.195	0.239	-0.493	0.599
10		B	-0.195	-0.67	0.237	-0.503	0.596
11	P8	T	1.08	0.732	0.174	-0.086	0.955
12		B	-0.748	-1.089	0.171	-0.084	0.965
13	P9	T	1.123	0.927	0.098	-0.602	1.039
14		B	-0.94	-1.135	0.097	-0.584	1.051
15	P10	T	0.677	0.38	0.148	-0.594	0.588
16		B	-0.376	-0.678	0.151	-0.58	0.588
17	P11	T	0.542	0.126	0.208	0.438	0.491
18		B	-0.131	-0.534	0.202	0.437	0.482
19	P12	T	0.838	0.645	0.097	-0.417	0.76
20		B	-0.65	-0.849	0.099	-0.43	0.769
21	P13	T	1.102	0.741	0.181	0.198	0.973
22		B	-0.75	-1.109	0.18	0.19	0.98

See the following diagram for a graphic representation of the plate results and see the Plate section of the help file for more detailed explanation.

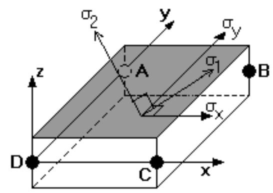
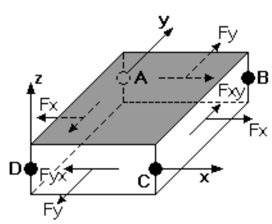


Plate Principal Stresses

Select **Plate Forces** from the **Results Toolbar** to view the plate forces.

The Plate Forces spreadsheet displays the "plane stress" forces, (F_x, F_y and F_{xy}), the out-of-plane shears (Q_x and Q_y), and the plate moments (M_x, M_y and M_{xy}).

See the following figures for a description of these results and see the help file for more information:



Plane Stress Forces

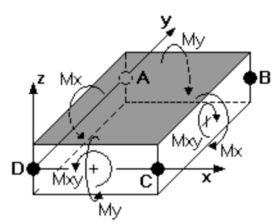


Plate Moments

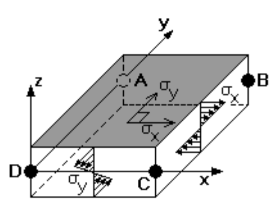
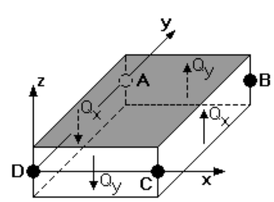


Plate Bending Stresses



Transverse Shears (Out of Plane)

Select **Plate Corner Forces** from the **Results Toolbar** to view the corner forces.

Plate Corner Forces for LC 5:ASCE 3 (a)

	Plate Label	Joint	X[K]	Y[K]	Z[K]	MX[K-ft]	MY[K-ft]	MZ[K-ft]
1	P3	N35	2.365	9.959	-0.234	8.606	0	19.796
2		N49	-1.112	-4.257	0.386	0.889	0	17.011
3		N53	-1.188	-3.645	0.509	3.087	0	2.284
4		N52	-0.066	-2.057	-0.661	8.802	0	2.397
5	P4	N49	1.66	7.186	-0.386	-0.834	0	16.195
6		N50	-1.19	-4.97	-0.009	0.817	0	17.675
7		N54	-1.121	-3.418	0.345	4.659	0	6.276
8		N53	0.65	1.202	0.05	3.668	0	3.89
9	P5	N50	1.176	4.216	0.009	-0.8	0	5.366
10		N51	-1.706	-3.323	-0.407	-1.183	0	9.302
11		N55	-0.627	-0.696	0.029	2.14	0	5.618
12		N54	1.157	-0.197	0.369	3.192	0	0.817
13	P6	N51	1.13	-0.519	0.406	1.187	0	-8.394
14		N36	-2.429	6.543	-0.234	8.548	0	-4.833
15		N56	0.078	-3.48	-0.694	9.745	0	0.006
16		N55	1.221	-2.544	0.521	3.109	0	-2.859
17	P7	N52	0.07	0.096	0.613	0.813	0	-2.339
18		N53	-0.203	1.263	-0.032	-2.692	0	-0.816
19		N58	-0.025	-1.249	-0.062	2.929	0	2.489
20		N57	0.158	-0.11	-0.518	4.048	0	0.593
21	P8	N53	0.741	-1.33	-0.527	-4.063	0	-5.358
22		N54	-0.601	1.684	-0.372	-3.792	0	-1.54

These are the forces and moments calculated at the corners of the plates, all in the global directions. The corner forces are obtained by multiplying the plate element matrix with the joint displacements. They represent the forces applied to the corners of the plate in order to hold the plate in equilibrium, very similar to a beam's end forces.

Ok, so now that you know all there is to know about plate results, let's go back to the graphics and try RISA-3D's contouring options.

Select **Single View** from the **Window Menu** and click on the **Results Toolbar.**

Plotting Plate Results

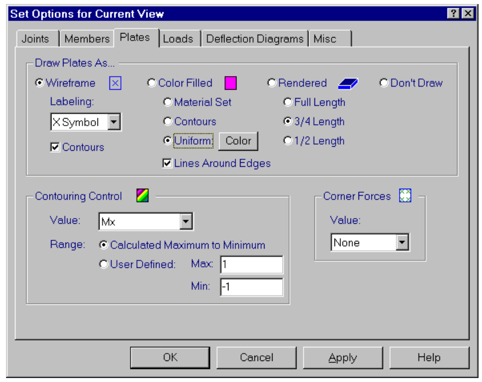
Turn off the node labels by clicking . Once again click on . Select the **Plates** tab.

Notice that there is a **Contour** option beneath **Wireframe** and also beneath **Color Filled**. These options control how results contours are displayed. By "contour" we mean that the solution results may be shown graphically on the plates via color-coding, where different colors represent different magnitudes for the particular result being "contoured". The contours may be drawn as either lines or as color filled areas.

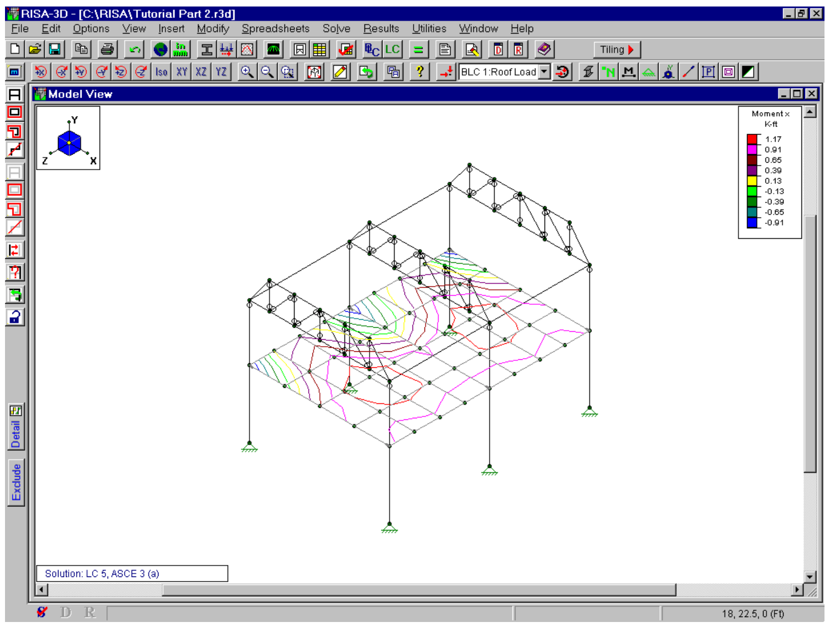
Half way down there is a section labeled **Value for Contouring (if solved)**. These options control what value is to be contoured. You can contour any of these results by just clicking on the appropriate button in this list. Let's give it a try. First, let's do a line contour.

Click the **Contours** option beneath the **Wireframe** option so that the box is checked. Now go down to the **Contouring Control** section and choose **Mx** from the **Value** list.

The dialog should look like this:

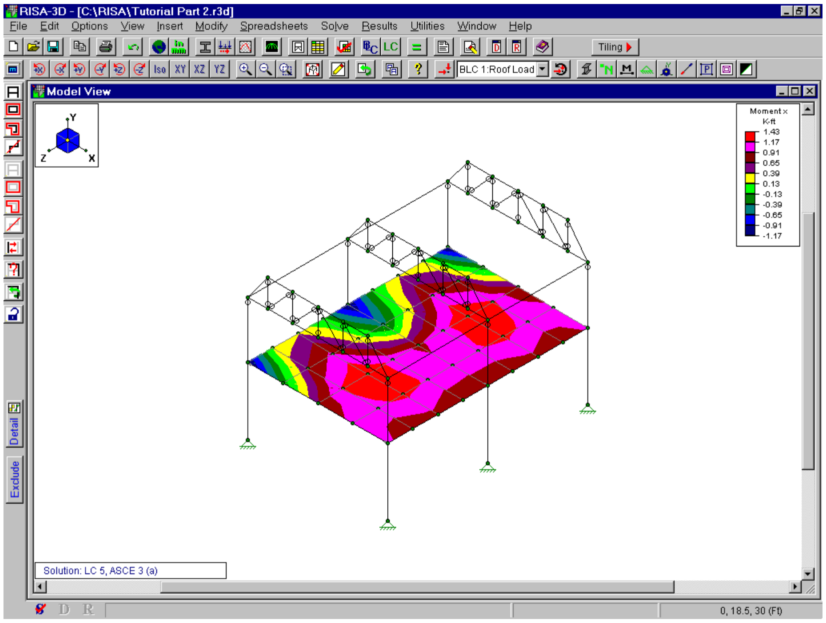


Click **OK.**



You can see that 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 lets try the color filled contour.

Click to get back to the plot options for the **Plates**. Now select **Color Filled** and the **Contour** option indented beneath it. Click **OK.**



This gives a more colorful representation of the Mx distribution. Note that for the previous line contour each line represented a specific value; for this color fill contouring the colors represent a range of values that are displayed in the legend.

Let's print the model view and then move on to animation.

First click on the **Plot Options** button and on the **Members** tab select the **Rendered** option and click **OK**. Now click the **Print** button on the **RISA Toolbar.** Select **Landscape** because our model view window is similar to a landscape paper orientation.

We can adjust the plot here if necessary. The scale factors scale fonts and symbols. You may also omit the title bar or add comments if you wish.

Click **Continue** and go to the printer dialog. Once you have selected your printer click **OK** and the plot will be sent.

Remember that the scale factors control the labels and legends and can be adjusted to whatever you like.

Click to go back to the options dialog and select the **Deflection Diagrams** tab. Turn on the deflected shape as before by selecting **Load Combination.** Now click on Animate This Deflected Shape .

You will see the animation behind the dialog.

Close the dialog by clicking **Cancel** and you may move the animation window to get a better view.

Note that had we clicked **OK** instead of **Cancel** the animation would be placed behind the model view again and we would select the last option from the **Window** menu which is **Animation of LC4: .9(D+L)+W+R.**

Notice how the Mx contour animates from zero out to its full value as the animated shape moves from an undeflected to fully deflected position? This shows how the Mx value builds in the plates as the structure is loaded.

Close the animation by clicking .

Now that we have covered all of the result spreadsheets and graphics take a minute to look at the special tiling options that RISA-3D provides.

To start, click on the **Tiling** button, scroll down, and select **Plate Results.** You may look at as many of the tiling options as you wish. Don't miss the **Three Results Views** option.

Envelope Solution

Click on the **Solve** button. Select **Envelope of Marked Combinations** and click **Solve.** Again you will be warned that the current results will be cleared. Select **Yes.**

RISA-3D always clears the results when you are making changes to the model thus avoiding the possibility that you could have results that don't match the input data. You will be prompted to make sure you wish to clear the results before doing so. This is because you may not want to purge results for a large model that took some time to solve. 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.

Steel Code Checks

Click on the **Tiling** button, select **Member Steel Results** and click **OK.**

The first things you will notice are the red members in the model view and the red text in the code check spreadsheet indicating that we have members that are failing. None of the wood truss members are in trouble but some of the steel columns and beams will require larger sizes.

Another thing to notice is that the spreadsheet results now have maximum and minimum values for each member, indicated by the "max" and "min" labels in the spreadsheet rows. In addition the spreadsheets have columns labeled "**lc"** that report the load combination that generated the maximum or minimum values. When you perform an envelope solution the spreadsheet results will contain this additional information.

Suggested Alternate Shapes

The fourth spreadsheet from the top reports what RISA-3D is suggesting for alternate shape sizes along with the member that is controlling the design.

Alternate Shapes

	Sectio...	Member	1st Choice	2nd Choice	3rd Choice	4th Choice
1	COL	M41	W12X87	HP14X89	W14X90	W12X96
2	BEAM	M96	W14X30	W16X31	W14X34	W12X35
3	BM-Z	M81	*W12X19	W12X22	W14X22	W12X26

On this spreadsheet RISA-3D suggests up to 4 alternative shapes for each section set, with the shapes in order of increasing weight. The asterisk in the last row indicates that we are already using the optimum shape for this section set.

Note that if you have modified your database by adding or removing shapes RISA-3D may pick different shapes than are shown here. Keep this in mind as you finish Part 2 of the tutorial. When you get to Part 3 make sure the shapes on the **Sections** spreadsheet are the 1st choice shapes shown on the last page of Part 2 otherwise your dynamic results will not agree with those shown here.

Let's go back to the **Sections** spreadsheet and change to the newly suggested shapes.

Click on the **Spreadsheets** menu and select **Sections.** Now click on the **Replace Shapes** button to replace the redesigned shapes in the spreadsheet automatically. Click on **Yes** to clear the results.

Now solve the model again by clicking on the **Solve** button and clicking **Solve**. When the solution is finished click on the **Tiling** Tiling button, select **Member Steel Results** and click **OK.**

Now only one column is in trouble. Looking at the **Alternate Shapes** spreadsheet (it is the second from the bottom.) we see that a new column is suggested as well as a new beam section.

A faster way to substitute suggested shapes can be accessed from this **Alternate Shapes** spreadsheet.

Click on the **Alternate Shapes** spreadsheet to make it active. The **Window Toolbar** presents the **Replace and Resolve** button. Click this button and agree to resolve by clicking **Yes.**

Click on the **Tiling** Tiling button, select **Member Steel Results** and click **OK.**

Now the section sizes are all optimized and the model view verifies that we no longer have failing members. See how easily we changed our model data and re-solved the problem? We could have just as easily added, changed or deleted any other data before re-solving.

Making Changes

Wait, what's that coming in on the FAX machine?! The client has just submitted a change order! He wants to raise the wood truss from 18'-6" to 22'-9", and he wants the frame 25% wider (from 21' to 26'3"). And he wants the completed design this afternoon!

These are significant changes to the model. If you were using a program that's batch input or heavily reliant on "parametric" data generators, at this point you would probably start the data entry process over from scratch to create the new model. Even many "graphical" programs would have serious problems dealing with these changes. Remember that the true test of a program is how easy it is to *modify* existing data because that is what you're going to be doing most of the time.

Fortunately, as we saw in optimizing our steel members, a major strength of RISA-3D is the ability to make changes to existing data. We could make the above changes entirely with graphic editing or entirely within the spreadsheets. We will make our changes graphically so that we may use a few more of these features. We will then mention how the changes could have been accomplished within the spreadsheets.

First we will widen the truss and to do this we will use the **Scale** feature.

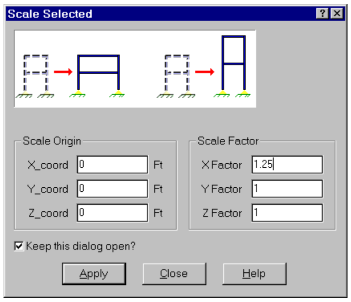
Maximize the **Model View** window by clicking on the **Maximize** button in its title bar. Click on the **Modify** menu and select **Scale**. You will be prompted to clear the results. Click **Yes**.

You may notice that the **Drawing Toolbar** is now available in the **Model View**. This is because we have chosen a graphic-editing feature and RISA-3D provides the toolbar to assist with further editing.

The **Scale** dialog box is also presented. The fields on the left contain the scale origin. This is simply the point that will remain stationary as the selected items are scaled. We will leave these fields set to "0".

The fields on the right are for the scale factors in each of the global directions. We only need to scale the model in the global X direction to widen the truss.

Change the **X Factor** to "1.25" and the dialog should look like this:



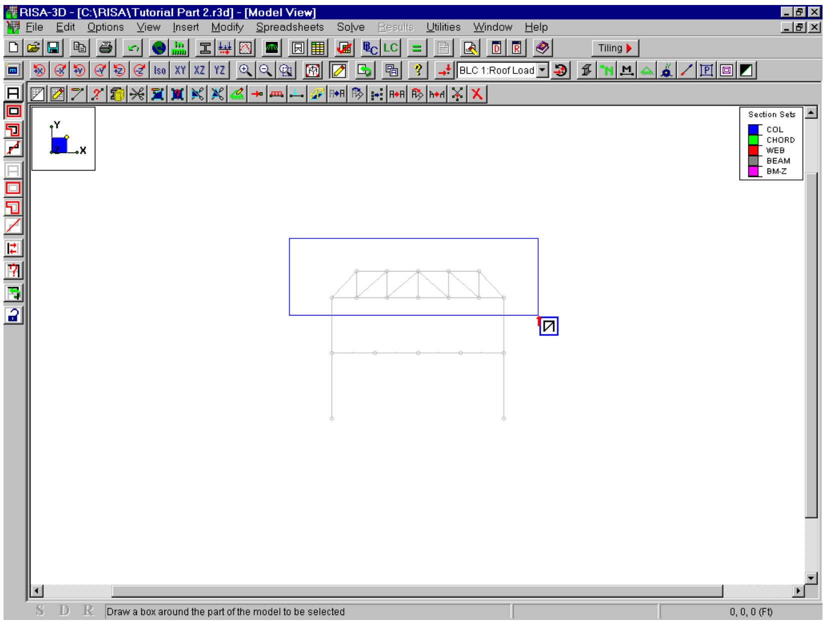
Click **Apply** and then **Close**.

The model view displays the change and we are finished widening the truss. We can not use **Scale** feature to raise the truss because it would lift the crossbeam as well which is not what we want. For this change we will use the **Move** feature. We will move the truss but we don't want to move everything so we need make our selection first.

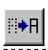
It will be easier to work with an elevation so click  on the **Window** toolbar. Now, on the **Selection** toolbar, click **Unselect All**  and then **Box Select** .

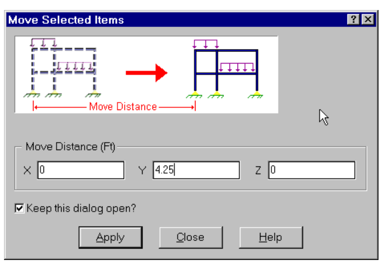
Moving over to the model we want to select the entire truss by drawing a box that contains it. The box should not contain nodes lower than the bottom chord but members crossing the window are ok.

Click above and to the left of the truss and hold the mouse button down while dragging the mouse down and right until the box contains the entire truss like this:



Release the mouse button and the entire truss, and nothing else, should be selected.

Now click the **Move**  button on the **Drawing Toolbar** (it has a blue arrow) and a dialog appears. We only want to change the elevation so double-click in the field labeled **Y** and type "4.25".



Now click **Apply** and then **Close**.

Our modifications are complete. The shape sizes will also need to be adjusted, but we'll let the RISA-3D redesign feature take care of that.

If we wanted to make the changes within the spreadsheets we could have used the block math feature to accomplish both modifications. To widen the structure we simply would have multiplied the node X coordinates by "1.25". To raise the truss we would have first sorted the Y coordinates and then selected the coordinates that are at an elevation of 8.5ft or higher. The final step would be to use the **Math** feature to add "4.25" to these selected Y coordinates.

Select the entire model by clicking on **Select All** . Also click on the **Isometric** button and then the **Full View** button.

Redesigning with Suggested Shapes

Now solve the model again by clicking on the **Solve** button and clicking **Solve**.

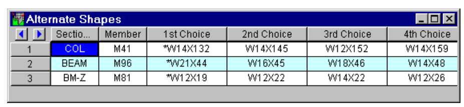
Since it's safe to assume our existing shape sizes won't be adequate, we'll also go through one redesign cycle before reviewing results.

Last time we solved we showed you the **Replace and Resolve** button but there is still a faster way. The key combination Ctrl-Alt-F7 will perform the same function and does not require you have any particular spreadsheet open.

Press Ctrl-Alt-F7 to substitute the shapes and resolve the model. Again, agree to clear the results.

Click on the **Tiling** button, select **Member Steel Results** and click **OK.**

Now once again we have run the solution with the optimum shapes.



Alternate Shapes

	Sectio...	Member	1st Choice	2nd Choice	3rd Choice	4th Choice
1	COL	M41	*W14X132	W14X145	W12X152	W14X159
2	BEAM	M96	*W21X44	W16X45	W18X46	W14X48
3	BM-Z	M81	*W12X19	W12X22	W14X22	W12X26

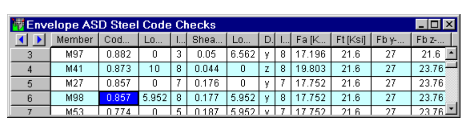
We will wrap up Part 2 by showing you how to exclude results and then print.

Excluding Results

What we will do is use the **Steel Code Checks** spreadsheet to sort the members and then hide results for members that have a code check value lower than "0.8".

Click in the **Code** column in the third spreadsheet down on the left. Now click on the **Sort** button and select **Sort Max to Min** and click **OK**. Now scroll down to look for the last member with a code check value that is 0.8 or higher. On the 6th line, member M98 has a value of "0.857".

Click anywhere in this row to place the active cursor like this:



Envelope ASD Steel Code Checks

	Member	Cod...	Lo...	I...	Shea...	Lo...	D.	I...	Fa [K...	Ft [Ksi]	Fb y-...	Fb z-...
3	M97	0.882	0	3	0.05	6.562	y	8	17.196	21.6	27	21.6
4	M41	0.873	10	8	0.044	0	z	8	19.803	21.6	27	23.76
5	M27	0.857	0	7	0.176	0	y	7	17.752	21.6	27	23.76
6	M98	0.857	5.952	8	0.177	5.952	y	8	17.752	21.6	27	23.76
7	M53	0.774	0	5	0.187	5.952	y	7	17.752	21.6	27	23.76

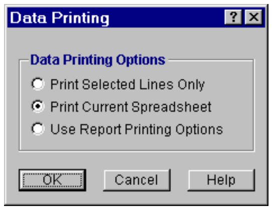
Now click on the **Exclude After** button (it has a red "X" on it) which is on the **Window Toolbar**.

You will be returned to the top of the spreadsheet but if you scroll down again you will see that the results for members after M53 are no longer displayed. (Don't despair, you can always click **Unexclude** to get them back.)

Printing

At this point, let's assume our design is complete (hasn't this tutorial gone on long enough?) and we wish to do some printing. For starters we will print the current spreadsheet to see that the excluded results do not print.

Press the **Print** button to open the print dialog. Click the middle option to print the current spreadsheet.



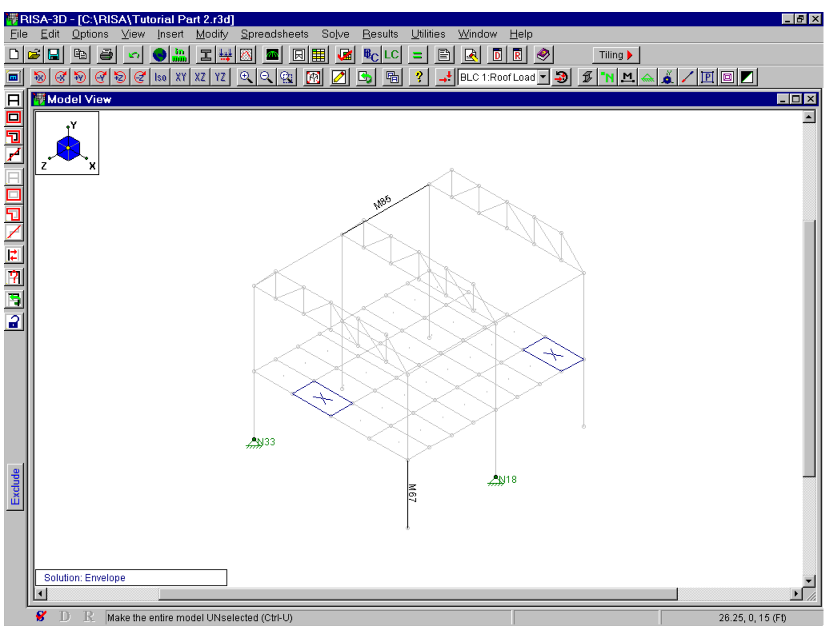
Click **OK** and you will be taken to the printer dialog where you will be able to choose a printer. Once you have done that click **OK** in that dialog and printer will give you your first report.

In the dialog above you can see that there are two other printing options. The first option will print lines from a spreadsheet and requires you to have selected the lines before you pressed **Print** .

The last option will print an entire report. We will first exclude most of the items on our screen so that the report only requires a few pages. We have seen how to hide results in the spreadsheets, now we will show you the graphical exclude feature which allows you to graphically select the items that you wish to print.

Select **Single View** from the **Window** menu. Display the member labels by clicking on the **Window Toolbar.**

Click **Unselect All** and then click on two members, two joints , and two plates so your screen looks something like this:



Now click on the **Exclude** Exclude button located on the **Select Toolbar.** Select **Yes** to exclude the results and click **OK** when the operation is complete.

We have excluded the unselected items from the results. Now it is time to print.

Click and then select Print a Report Instead... .

What you are looking at is the **Report Printing Options** dialog. This dialog allows you to select exactly what you want to print. You may select a standard report from the **Report Name** box in the upper left corner or you can build your own report. We do a little of both.

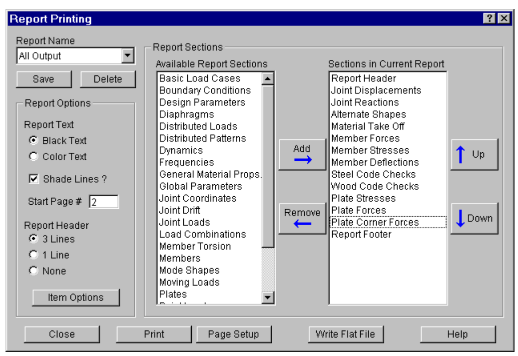
Click on the down arrow in the **Report Name** box and select **All Output**.

You will see that items have moved from the **Available Report Sections** to the **Current Report**. You may also manually move these sections by selecting them and then clicking the **Add** and **Remove** buttons. Another way to do this is to simply double-click on the item to move it in and out of the report. Once items are in the report you may use the **Up** and **Down** buttons to adjust the printing order. We will remove a few items from the report before we print.

Double-click on **Joint Drift,** which is the fourth item in the **Current Report**.

When you do this the section is removed from the report and placed back on the **Available** list.

Likewise, double-click on **Member Torsion** and on **Frequencies** and **Mode Shapes** to remove them from the list and the dialog now looks like this:



Click **Print** to get to your systems print dialog where you can select your printer and click **OK**. Click **Close** on the **Report Printing Options** dialog box.

On the **RISA Toolbar** click on **Save** to save the model.

This concludes part 2 of the tutorial.

If you wish to press on with part 3 of the tutorial right now click the **Clear Results** button on the **RISA Toolbar** and select **Single View** from the **Window** menu. Now jump ahead to the section "**Continuing the Tutorial**".

If you wish to exit RISA-3D and do part 3 later, select **Exit** from the **File** menu.

RISA-3D Tutorial, Part 3

First, restart the RISA-3D demo.

Now read in the file by selecting **Open** from the **File** menu.

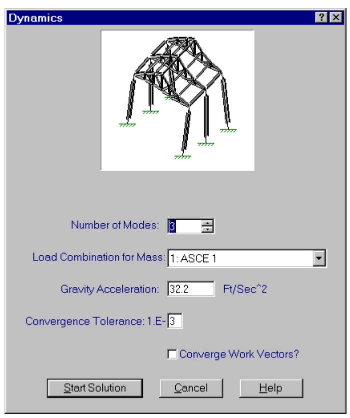
In the **File Open** dialog select the file named “Tutorial Part 2” and click **Open**. Close the **Global Parameters** window by clicking **Close** and we will start on the dynamics. Click and to remove the **Data Entry Toolbar** and prevent the **Results Toolbar** from displaying.

Continuing On

In this part of the tutorial we’re going to use the model we defined in the first and second parts of the tutorial to perform a dynamic analysis.

Click on the **Solve** button and select **Dynamic Analysis** and click **Solve**.

You should now see this dialog:



This dialog is the “gateway” into RISA-3D's dynamic capabilities. The entries on this dialog actually are quite simple (RISA-3D does all the hard stuff). Just tell RISA-3D how many modes to calculate and which load combination should be used for mass calculations. The other entries here rarely need to be altered. For a complete discussion of the entries on this dialog, see the **Dynamic Analysis** section in the help file.

We'll solve for 3 modes and we’ll use load combination 3 to calculate mass.

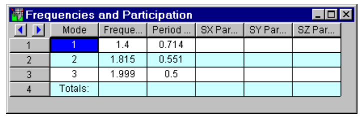
Select Load Combination 3 for the mass. Click and the dynamic solution begins.

This solution may take a couple of minutes, depending upon how fast your computer is, so read on.

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

The first part is the actual “dynamic analysis” (what we're doing here), which means the calculation of the modes and frequencies of vibration for the model. The second part is the “response spectra analysis”, which uses these modes to calculate forces, stresses and deflections in the model.

Once the dynamic analysis is complete, you'll find yourself at this spreadsheet:


Frequencies and Participation

	Mode	Freque...	Period ...	SX Par...	SY Par...	SZ Par...
1	1	1.4	0.714			
2	2	1.815	0.551			
3	3	1.999	0.5			
4	Totals:					

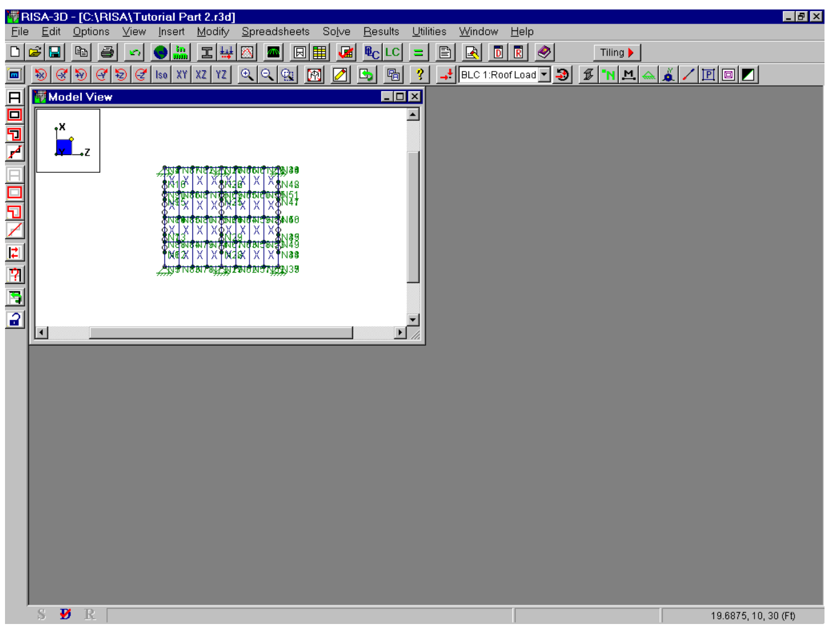
Frequencies and Mode Shapes

These are the frequencies/periods calculated for the model. The last three columns are the modal participation percentages, which will be filled in after we perform the response spectra analysis.

You can also review the mode shapes graphically. Let's set up our screen a bit differently to look at the mode shapes.

Click Tiling and then select **Three Input Views**. Now close the bottom and right side views by clicking on the ☒ button in their titlebars.

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



Now click on Iso and then which will open the **Plot Options** dialog.

Select the **Joints** tab. Clear the **Show Joints** checkbox. Click on the **Deflection Diagrams** tab and change the **Magnification Factor** to “20”. Now click on Animate ALL the Mode Shapes .

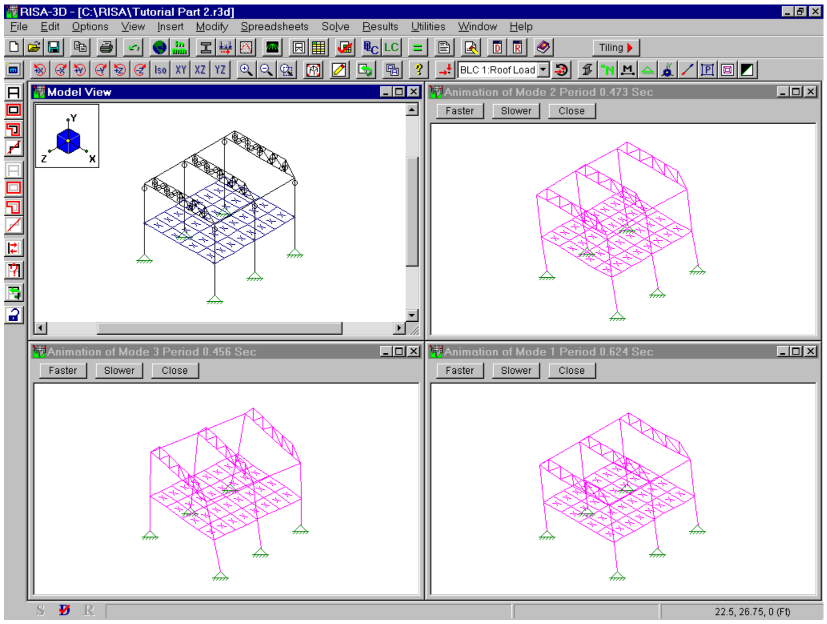
You will get a warning that this may take a while (it shouldn't for just three modes).

Click **Yes**.

You will see three different progress bars, one after the other, build three different animations.

After the animations are built and the progress bars are gone click **OK** to close the **Plot Options** dialog. Now click on **Tile Horizontal** from the **Window** menu.

Your screen should look like this with animated mode shapes in all but the upper left window:

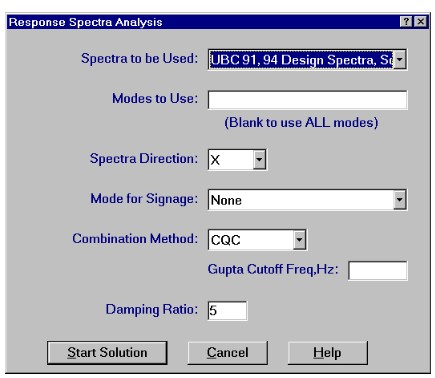


When you've finished viewing the animations select **Close All** from the **Window** menu.

Response Spectra Analysis

Click on the **Solve** button and select **Response Spectra Analysis** and click **Solve**.

This brings us to the **Response Spectra Selection** window:



The first field is a list of the available spectra in the RISA-3D spectra library. For this analysis, we'll have RISA-3D build a 97 UBC site specific spectra. See Figure 16-3 in the UBC for the equations used to build the spectra. See Tables 16-Q and 16-R to obtain the Ca and Cv values.

Click on the down arrow in the **Spectra to be Used** field and select "UBC 97, Parametric Design Spectra".

A detailed explanation for all the fields on this dialog can be found in the **Response Spectra Analysis** section of the help file, so we'll only provide a brief description here.

The "**Modes to be Used**" field allows you to select which of the solved modes should be used in the Response Spectra Analysis (RSA). Typically you will usually leave this field blank to use ALL the solved modes.

The "**Spectra Direction**" specifies the direction of action for the base excitation represented by the response spectra. For this model, we'll be applying the spectra in the X and Z directions so we will leave this as is.

The "**Mode No. for Signs**" field gives you some control over the signs of the final RSA results. The modal combination methods available all are derivatives of the SRSS method (Square Root of the Sum of the Squares), so the final results come out ALL POSITIVES. This is because an RSA is intended to predict the MAXIMUM values for the model response. This presents a problem if the RSA results are being combined with a static solution, which will have both positive and negative results. RISA-3D gives you the option of assigning signs to the final RSA results based on the signs for the results for any particular mode. If you choose not to use this option, just leave it set to "None".

For the tutorial we will have RISA-3D automatically detect and use the dominant mode (the mode with the highest participation factor, discussed later).

Click on the down arrow in the **Mode for Signage** field and select "Dominant".

We will stick with the current combination method and damping ratio which are CQC and 5% respectively.

So we've specified the spectra be applied in the X direction, and use the dominant mode for results signs.

Click Start Solution and the **UBC 97 Spectra** dialog is opened.

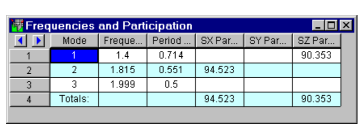
The default values listed are for Seismic Zone 3, Soil Type "Se" (Soft Soil Profile).

Click **OK.**

When complete, you are returned to the **Frequencies** spreadsheet. We'll go right back in and perform a RSA for the Z direction.

Again click on the **Solve** button and click **Solve** to bring up the **Response Spectra Selection** Window. This time change the **Spectra Direction** to "Z". Click Start Solution and then click **OK** in the **UBC 97 Spectra** dialog.

The **Frequencies Spreadsheet** is presented and looks like this:


Frequencies and Participation

	Mode	Freque...	Period ...	SX Par...	SY Par...	SZ Par...
1	1	1.4	0.714			90.353
2	2	1.815	0.551	94.523		
3	3	1.999	0.5			
4	Totals:			94.523		90.353

Modal Participation

The modal participation percentages are now recorded for the X and Z directions. About these participation values: Be sure the participation for each direction totals to 90% or more! For this model, we've met this criteria, but if you're running a model where the participation total is less than 90%, you must return to the dynamics dialog, increase the number of modes and redo the dynamic solution and also the RSA.

OK, so the dynamic analysis and response spectra analyses are done. 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

Now comes the calculation of the spectra scaling factors. We won't be going through the actual procedure here, but we'll give a quick overview. (You can see an example of the procedure in the Help file. Just search the Help index using the keywords "scaling factor", then select "RSA Scaling Factor" from the choices.) The procedure we use is based on the requirements in Section 1630 of '97 UBC. If you are not familiar with these requirements or if you use another building code you will want to refer to the code to understand the basis for the procedure.

The reasons for having to calculate scaling factors are twofold. First, if a "normalized" spectra was used to calculate the spectral results, you must scale the normalized results to match your site specific criteria. Second, the UBC sets minimum values for the design base shear.

In a nutshell, what has to be done is:

1) Calculate the UBC static design base shear (V).
2) Obtain the unscaled RSA Elastic Response base shear.
3) Scale the RSA base shear such that it satisfies the requirements of sections 1631.5.4.

After applying the UBC requirements, the calculation for the scaling factors (SF's) gives:

$SF_x = 0.16$

$SF_z = 0.16$

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

These scale factors will now be applied to our spectral results so we can combine them with our static results. All that's left to do is define our load combinations and do the final analysis of the model.

Return to the **Load Combinations Spreadsheet** by clicking on the LC button. Move the active (green) cell to the last line and press [ENTER] to create a new line. Type the next load combination:

DL+LL+SX+.3SZ , → , [SPACEBAR] , → , [SPACEBAR] , → , → , → , → , L2 , → , 1 , → , SX , → , .16 , → , SZ , → , .05

Create a new line by pressing ENTER and type the next load combination:

DL+LL+SZ+.3SX , → , [SPACEBAR] , → , [SPACEBAR] , → , → , → , → , L2 , → , 1 , → , SZ , → , .16 , → , SX , → , .05

Note that the SPACEBAR has set the envelope and the wind/seismic flags.

Load Combinations

	Description	Env	WS	PD...	SR...	CD	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor
1	ASCE 1	☑	☐			1	DL	1								
2	ASCE 2 (a)	☑	☐			1	DL	1	LL	1	RLL	1				
3	ASCE 2 (b)	☑	☐			1	DL	1	LL	1	SL	1				
4	ASCE 2 (c)	☑	☐			1	DL	1	LL	1	RL	1				
5	ASCE 3 (a)	☑	☐			1	DL	1	WL	1						
6	ASCE 3 (b)	☑	☐			1	DL	1	WL	-1						
7	ASCE 4 (a)	☑	☐			1	DL	1	LL	.75	RLL	.75	WL	.75		
8	ASCE 4 (b)	☑	☐			1	DL	1	LL	.75	RLL	.75	WL	-.75		
9	DL+LL+SZ+.3SX	☑	☑			1	L2	1	SX	.16	SZ	.05				
10	DL+LL+SX+.3SZ	☑	☑			1	L2	1	SZ	.16	SX	.05				

What we have done here is combine the spectra analysis results (SX and SZ) with the combination of loads previously defined in load combination 2 (L2).

Note that "0.05" is 30% of the "0.16" scale factor. 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 we will mention briefly.

RSA SRSS Combination

The SRSS flag indicates the RSA results from different directions are to be combined using an SRSS combination. This will generally ensure that reasonable maximum responses are obtained. Setting the SRSS flag will cause the RSA results to be ALL POSITIVES. The flag itself is entered as a "+" or "-" to indicate whether the combined RSA results are to be added (+) or subtracted (-) from the other loads in the combination. We haven't used the SRSS flag here.

Final Solutions

Click Solve Envelope on the **Window Toolbar.** Once again, click on the **Tiling** Tiling button, select **Member Steel Results** and click **OK.**

We have no members in trouble so we are finished.

Select **Save As** from the **File** menu and name the file "Tutorial Part 3".

If you or someone else has previously been through this tutorial you may get an overwrite warning for this file; if you do, select **Yes** when prompted for whether the file should be overwritten.

At this point, we're finished with the tutorial. If you like, you can go back and change some data, solve different Load Combinations, experiment with the plot; whatever you like.

To leave RISA-3D at this point, select **Exit** from the **File** menu.

Conclusion

This tutorial is rather long and detailed, but if you've completed all three parts you should have a very good feel for how to use RISA-3D. Some of the topics we have not covered are listed below. You may not use all of these features but it is always a good idea to be aware of what is available.

Of course, if you wish to know more about specific features, you can refer to the on-line help system. You may use the index to locate a particular topic.

Model Generation

RISA-3D has an expanding library of automatic generators that can quickly generate beams, arcs, trusses, cylinders and numerous mesh layouts of our plate shell element. Click the  button to access this feature. If nothing else, try the rectangular tank generator.

Moving Loads

The standard AASHTO loads are built into the moving loads database, however you can add and save custom moving loads as well.

Thermal Loads

You can model the effects of temperature differentials in members. These effects cause the axial expansion or contraction of the member along its length.

Rigid Diaphragms

RISA-3D has rigid membrane and rigid plane diaphragms to handle different modeling situations.

Tension/Compression Only Members and Springs

These features allow you to model one-way stiffness such as braces that cannot take compression or soil that can only resist compression.

Story Drift, Joint Slaving and Enforced Displacements

You may calculate and report inter-story drift, slave joints together or specify the displacement at a joint.

Tapered Members and Unequal Flanges

You may specify tapered wide flanges or members with unequal flanges and design them per the Appendix F criteria.

If you have any questions, comments or suggestions feel free to call us at 949-951-5815, or FAX us at 949-951-5848 or e-mail us at "support@risatech.com".

Thank you for using RISA-3D!