RISA-2D

Rapid Interactive Structural Analysis – 2-Dimensional

Version 7.0 User's Demonstration Guide

Table of Contents

Before You Begin

How to Use the Manuals

This Guide is designed to be read in two ways. If you are already familiar with structural modeling in general you can skip the supporting text and read only the underlined action items to move through the tutorials very quickly. To do this read the rest of this **Before You Begin** section and then skip to page 7 and look for the underlined action items. If you want more thorough explanations of the modeling process you may read all or some of the supporting text as you see fit.

The tutorials are intended for the first time user of RISA-2D or those who need a refresher on how the program works. To complete this guide in its entirety will take less than an hour if you perform only the underlined action items mentioned above or 2 to 3 hours if you read all of the supporting text. We've broken the tutorials up so you don't have to do it all at once.

The tutorial is intended to teach you the basics of defining and solving models in RISA-2D, and also how to review the results. We won't go into a lot of depth regarding the analytical aspects of RISA-2D here; that's what the RISA-2D General Reference manual and on-line help file are for. What we will cover is how and when to apply RISA-2D features to help you be most productive. For example, we'll perform steel, wood, and concrete code checks in this tutorial, but we won't discuss the specifics of how those code checks are calculated; that is covered in excruciating detail in the General Reference manual and also in the help file.

After you have gone through the User Guide, use the General Reference and On-line Help for detailed information on any topic. The topics are summarized in the table of contents and are thoroughly indexed.

License Agreement

END-USER LICENSE AGREEMENT FOR RISA TECHNOLOGIES® SOFTWARE:

The RISA-3D software product (SOFTWARE PRODUCT) includes computer software, the associated media, any printed materials, and any electronic documentation. By installing, copying or otherwise using the SOFTWARE PRODUCT, you agree to be bound by the terms of this agreement. If you do not agree with the terms of this agreement RISA Technologies is unwilling to license the SOFTWARE PRODUCT to you. In such event you must delete any installations and destroy any copies of the SOFTWARE PRODUCT and return the SOFTWARE PRODUCT to RISA Technologies within 60 days of purchase for a full refund.

Copyright 2006 by RISA Technologies. All rights reserved. The SOFTWARE PRODUCT is protected by United States copyright laws and various international treaties. All rights not specifically granted under this agreement are reserved by RISA TECHNOLOGIES.

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

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

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

4. PROTECTION DEVICE. In the event the SOFTWARE PRODUCT requires the use of a PROTECTION DEVICE to operate, you are specifically prohibited from attempting to bypass the functionality of the PROTECTION DEVICE by any means. If the PROTECTION DEVICE becomes broken or inoperable it should be returned to RISA TECHNOLOGIES for a replacement. The replacement will not be provided if RISA TECHNOLOGIES can not affirm that the broken PROTECTION DEVICE was originally provided by RISA TECHNOLOGIES for use with the SOFTWARE PRODUCT. A lost or stolen PROTECTION DEVICE will not be replaced by RISA TECHNOLOGIES.

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

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

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

Installation

To install RISA-2D please follow these instructions:

> 1) Put the RISA-2D CD in your computer CD drive.
>
> 2) If the CD starts automatically go to step 4. If the CD does not start after 10 seconds click the Windows **Start** button and select **Run**.
>
> 3) In the Run dialog box type "**d:\launch**" (where "d" is the label of your CD drive) and then click the **OK** button.
>
> 4) Follow the on-screen instructions.

Technical Support

Complete program support is available to registered owners of RISA-2D and is included in the purchase price. This support is provided for the life of the program. See the RISA-2D General Reference or Help file for a list of your support options.

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-2D is released, the life of the previous version is considered to be ended.

RISA Technologies will support only the current version of RISA-2D.

Maintenance

Program maintenance provides all upgrades to RISA-2D.

When your maintenance expires, you will be given the opportunity to continue program maintenance on an annual basis. You are under no obligation to continue program maintenance, of course, but if you decide to discontinue maintenance you will no longer receive RISA-2D program upgrades and technical support.

Overview

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-2D is to speed up this process.

RISA-2D mainly has two complimentary sets of tools; graphics based drawing tools and customized spreadsheets. We say "complimentary" 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; 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.

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.

Other graphic edit features are built around RISA-2D'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-2D 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

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

Important Assumption!

The tutorial is written with the assumption that RISA-2D has not been customized and is in the default, installed state. RISA-2D allows you to customize its settings so that it best suits your needs. If the installation of RISA-2D 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 in the **Preferences** on the **Tools** menu.

Conventions Used in the Tutorials

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

When we say to "click" something with the mouse, we mean move the mouse so the mouse pointer on the screen is on the object, once there, press and release the left mouse button. Occasionally we will want you to click the RIGHT button. In this case 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.
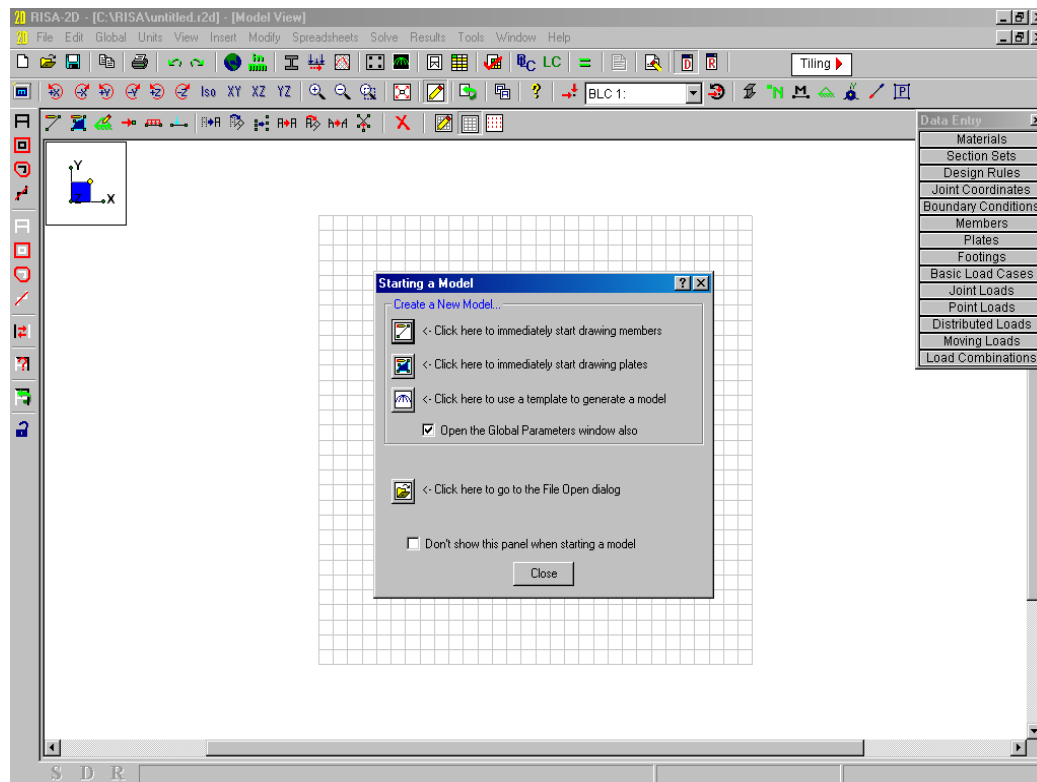
First Look at RISA-2D

Starting RISA-2D

So lets take a quick first look at RISA-2D 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-2D**.
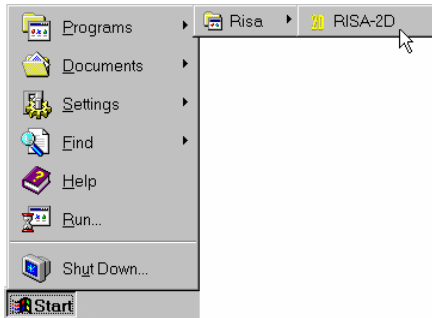
The remainder of this chapter will give an overall guide of what you have in front of you. (The next action item is on page 15.)

The RISA-2D 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.

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-2D. The three buttons on the far right side of the title bar are used to control the main window. The button will shrink RISA-2D to a button on the taskbar. The button will shrink the RISA-2D window on your screen. The button will close RISA-2D, prompting you to save changes if necessary. You will also see these buttons in other RISA-2D windows and they have basically the same functions there as well.

Main Menu
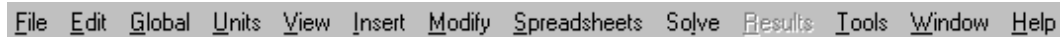
Just beneath the title bar is the RISA-2D main menu beginning with **File** on the far left and ending with **Help** on the far right. These menus provide access to all of the features RISA-2D has to offer. Clicking on each of these menus will display sub-menus that contain options that you may choose 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 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.

Global – The **Global** parameters allow you to set options that apply to the entire model such as steel code, shear deformation and warping effects, p-delta options and model descriptions and notes.

Units – This option takes you to the **Units** dialog where you can choose different units and convert existing data.

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 joints, 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-2D provides and the **Spreadsheet** menu helps you access the spreadsheets.

Modify – The **Modify** menu gives access to the graphic editing features and may be used to modify existing joints, members, plates and loads.

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

Solve – Use this menu to access the analysis options provided with RISA-2D.

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

Tools – RISA-2D provides features to help you organize identify and correct problems as you model the structure. Program preferences are also located here.

Window – The **Window** menu can be used to manage all of the windows that you have open in RISA-2D 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-2D.

Toolbars

RISA-2D 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.

RISA Toolbar

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.

Now is a good time to note that there are many ways to access the features available to you in RISA-2D and the method that you will use will simply be a matter of personal preference. The good news is that RISA-2D 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.

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**.
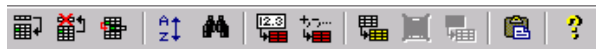
Drawing Toolbar

The **Drawing Toolbar** is the third horizontal toolbar and is displayed 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.

Spreadsheet Toolbars

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 [D] and [R] on the **RISA Toolbar.**

Workspace

The actual work that you do in RISA-2D will be in the main area on the screen, which we will call the workspace. Currently the workspace contains a white model view with the default drawing grid and the **New Model** options. (We will see how to change the default grid and many other defaults as we go along.) As we open new model views and spreadsheets they will also appear in the workspace. This brings us to the types or windows that we can work with.
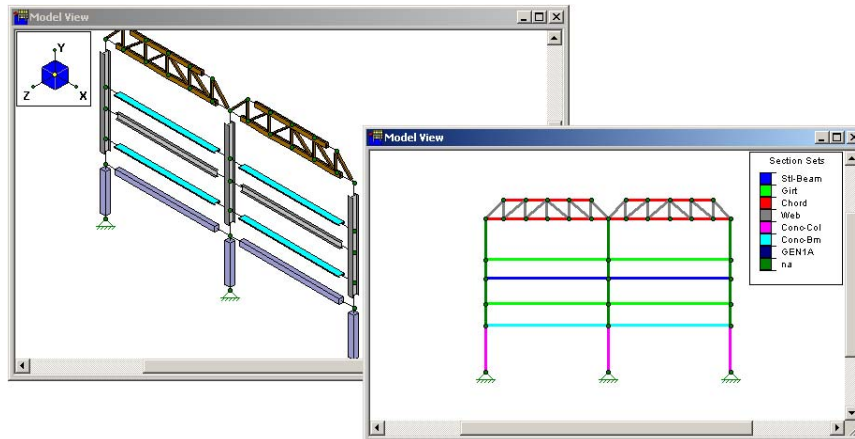
Windows
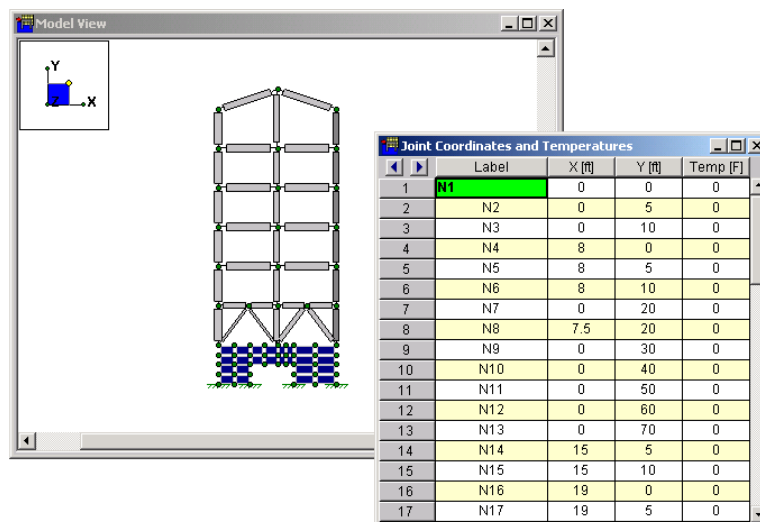
You will work within model views and spreadsheets, each in their own window that may be moved around the workspace and sized as you wish. A powerful feature of RISA-2D is the ability to have multiple model views and multiple spreadsheets open at one time. The options in the **Window** menu are provided to help you manage these windows.

Notice in the windows above, the same three buttons [- □ X] 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.

Views

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 have different information plotted in each view.

One thing to remember is that the toolbars that RISA-2D displays depend upon what window is active. The active window is the one with the colored title bar. 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.

Spreadsheets

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.

Dialogs
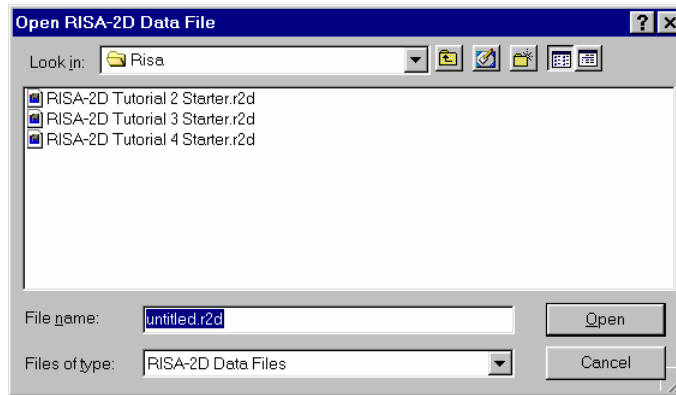
Dialog windows are windows that help you perform a specific function within the program. For example if you were to select **Open** from the **File** menu you will be presented with the dialog box above, which helps you find the file you wish to open. You will find that dialogs are very easy to work 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 title bar, and then click on any item in the dialog to get help for that item.

Status Bar

The status bar across the bottom of the screen is to pass information to you as you work with RISA-2D. 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. 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.

Getting Help

RISA-2D 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-2D: **General Help**, **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, **Search**, is a more detailed search that allows you to search the actual text of the topics to locate the keywords that you enter.
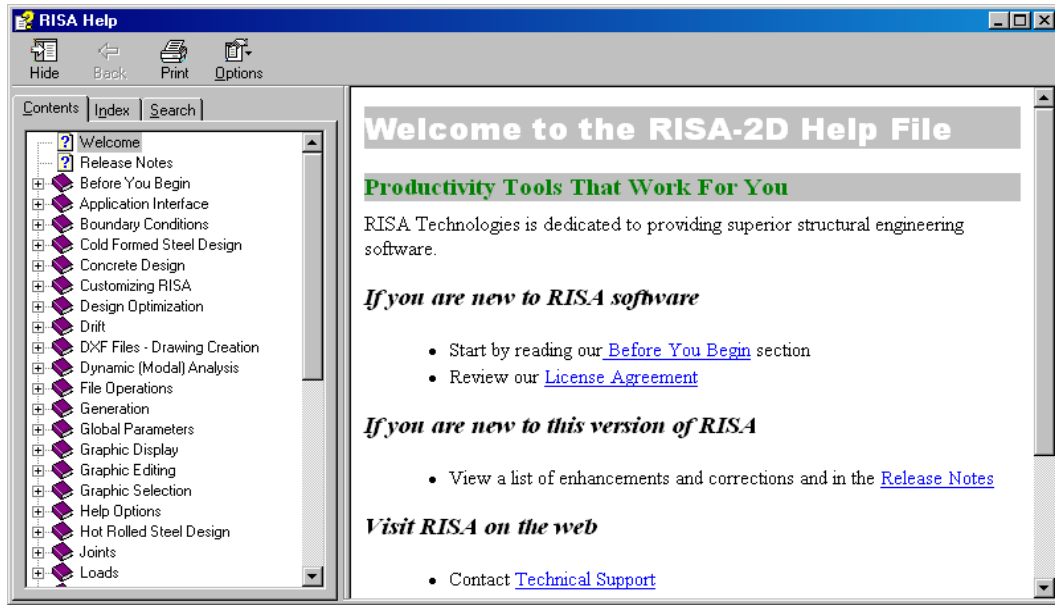
Once you have a topic open there are a number of options available to you that make Help even more useful. Click the menu options to do such things as annotate the file with your own notes or place a bookmark.

Window Specific 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. **This is very useful!**

"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

In the tutorials we will be modeling this structure:
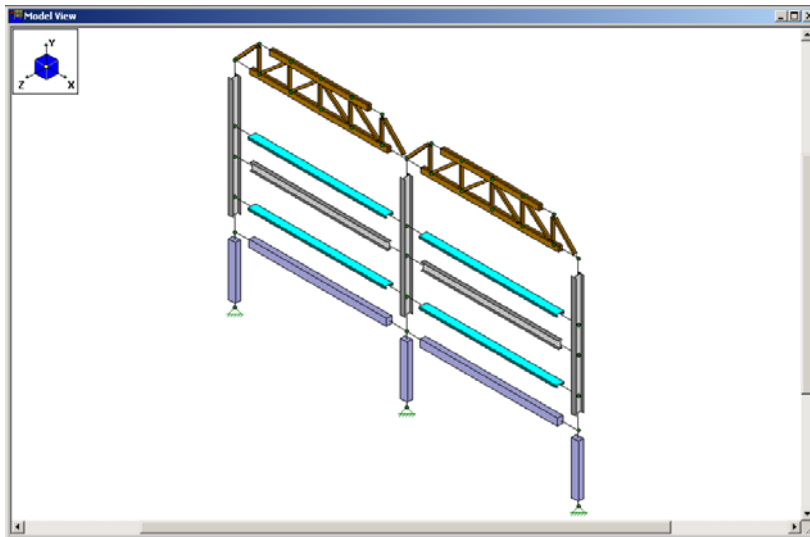
We'll make use of both the spreadsheet and graphic drawing capabilities of RISA-2D. In Tutorial 1 we will create a frame comprised of two steel columns supporting a wood truss. In Tutorial 2 we'll copy this into a 2 bay model and make changes. Tutorial 3 will show how to define loads in the model. Tutorial 4 will teach how to run static solutions and review the results.

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 you can *change* your model!

Tutorial 1 – Modeling

So we have started RISA-2D and are ready to define our model. We should note here that your picture might not look exactly like the screen below. Your computer 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.
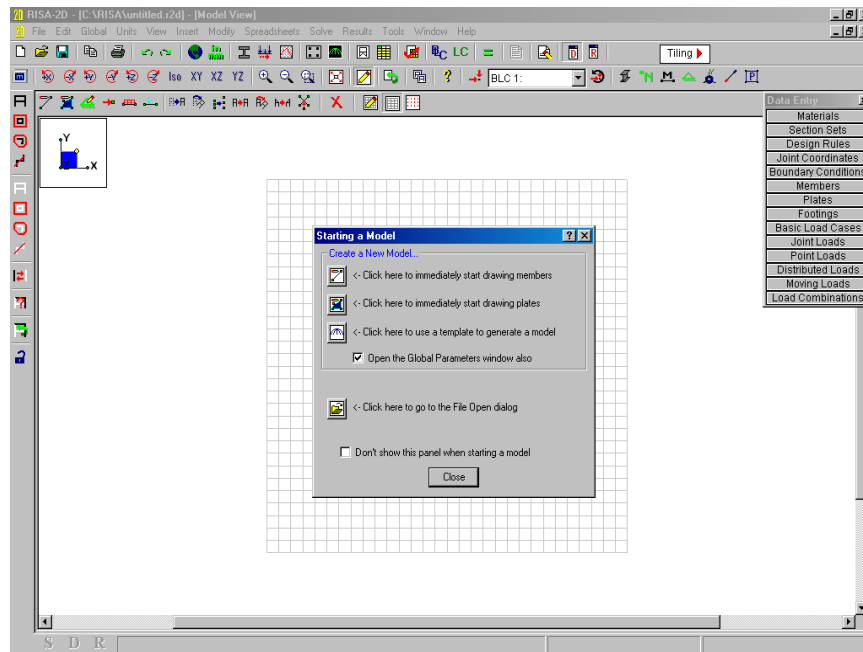
Starting a New File

Starting a new file presents 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 Model.** You can choose to start drawing your model or open an existing model. To start a new model you may begin by defining members, plates or use a template to generate it automatically, or click **Cancel** and work on your own.

Notice that a box is checked that reads "**Open the Global Parameters window also**".

Automatically Generating with Templates

RISA-2D has powerful templates that automate the generation of structures. You may access them by selecting **Structure** on the **Insert** menu or clicking ▧.

Some of the templates are very specific such as the **Parabolic Arc** template. Some are general such as the **Circular Disk With Plates**. This way you can generate just about any typical or repetitive model or model component. The tutorials do not use the templates in order to teach you how to model structures without them. Once you know this you will easily understand the templates and will be able to use the all of the power of RISA-2D.

Click the ▧ button to start drawing members.

Global Parameters

Global Parameters are settings that apply to the model as a whole. This window is opened automatically because of this checked box that we mentioned above:

☑ Open the Global Parmeters window also?

Use your mouse to select the **Description** tab and **click** in the first field, labeled **Model Title** so that a flashing cursor appears there. Now type:

Tutorial Problem, [TAB]

(Type your name)

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

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.

Use your mouse to select the **Solution** tab.

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 **Number of Internal Sections** controls how many places along each member the software calculates and stores results such as deflections and code checks. The member force diagrams displayed in the model view and the detail plot are also drawn from these results.

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 General Reference.

The **Merge Tolerance** is used as the maximum distance 2 joints can be apart and still be merged together. It is also used when scanning for crossing members and for unattached joints along the spans of members.

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

Gravity Acceleration and **Eigensolution Convergence Tolerance** are used for dynamics and are discussed in the Dynamics section of the General Reference.

The **Global Parameters** dialog should now look like this:
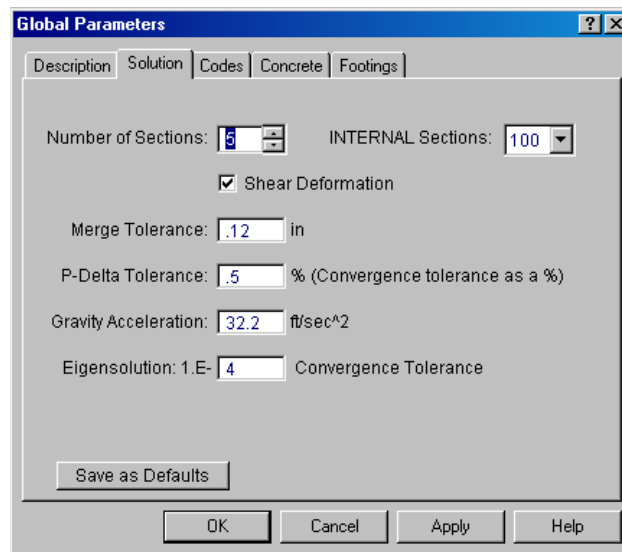
Use your mouse to select the **Codes** tab.
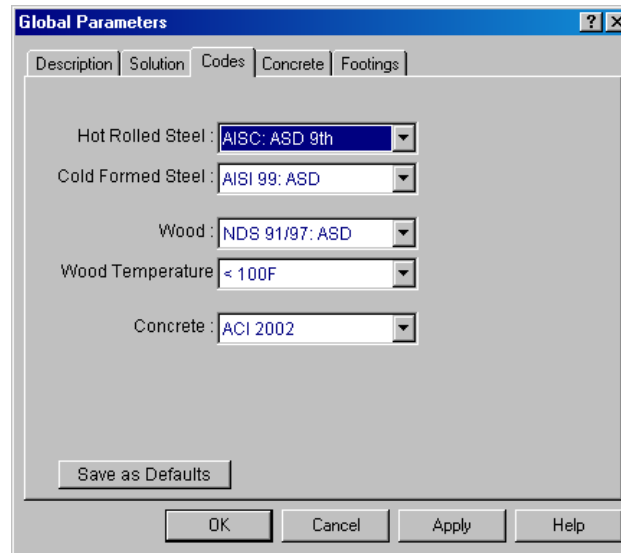
These items control how code checking is performed.

Currently the **Hot Rolled Steel Code** choices include **9th edition ASD, 2nd Edition LRFD, 3rd Edition LRFD,** and several international steel codes.

The **Cold Formed Steel Code** choices are **AISI 99: ASD & LRFD, AISI 01: ASD & LRFD,** and several international codes.

The choices for **Wood Design Code** and **Concrete Design Code** are also the standard US codes along with several international codes.

We will use the default specifications for the tutorial, which are already selected.

Use your mouse to select the **Concrete** tab.
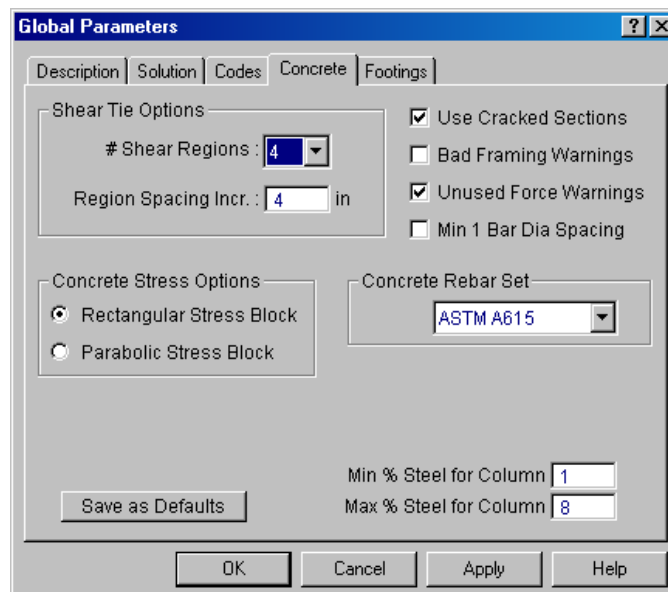
The entries under the **Concrete** tab contain options related to the analysis and design of concrete members.

The **Shear Tie Options** allow you to control the **Number of Shear Regions** that will be used when detailing a beam or column span. It also allows you to specify the increment that you'd like the program to use when increasing or reducing the spacing of the shear ties.

The **Concrete Stress Options** allow you to choose what type of stress block to consider in your analysis. The options are the constant **Rectangular Stress Block** and the **Parabolic Stress Block**. Since the parabolic block is more accurate, we will select this stress distribution for our design.

Click on ⊙ Parabolic Stress Block to select it for our design.

Check the **Used Cracked Sections** box if you want to modify the member stiffnesses by the **Icr Factor**. For more information refer to the General Reference Manual.

Checking the box for **Bad Framing Warnings** will produce a message in the error log whenever the program detects a member that is unsupported at either of its end joints. This can be common for models that contain cantilevers, rafters and such.

Checking the box for **Unused Force Warnings** will produce messages on the member detail report that informs you when weak axis and torsional moments have been neglected.

The **Concrete Rebar Set** allows you to specify the grade of rebar you wish to use. You can also specify a **Minimum % Steel** and **Maximum % Steel** for column reinforcement.

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 would become the default settings for future models.

Before we move on, note the **Help** button and the ⁇ button in the upper right corner. Both are there to assist you. The **Help** button will take you to a topic in the help file that explains all of the options before you. The ⁇ button allows you to get an explanation of any one option you click on. The help system in RISA-2D is second to none and is there only to help you.

Now close the **Global Parameters** window by clicking **OK**.

Members

You should now see a dialog box for **Member Properties**, which is shown below.
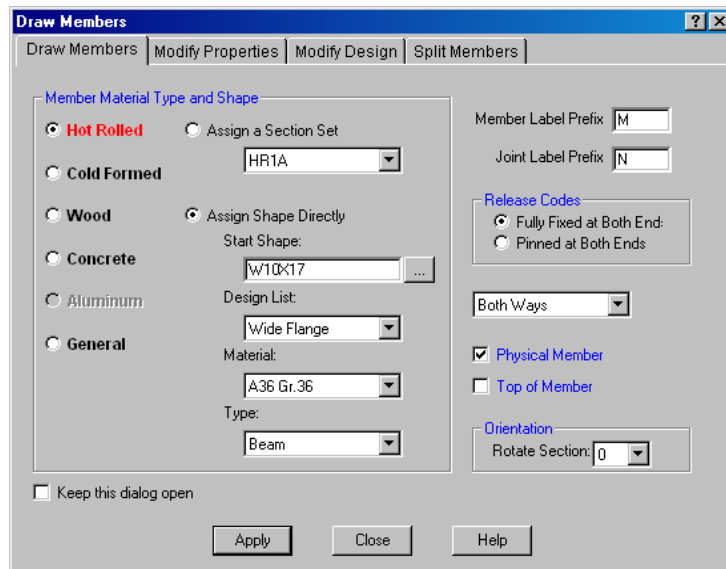
For the members we are going to draw we must first define what Material Type to use. The current options are **Hot Rolled**, **Cold Formed**, **Wood**, **Concrete** and **General**. Since the program knows what material a member is made from, it will sort your input spreadsheets and solution results based on that material. When performing a solution, the program will only analyze/design members for load combinations that are appropriate for that material. We will discuss this issue further in Tutorial 3 – Loading.

Click on Hot Rolled to indicate that the Material Type for our new members will be Hot Rolled Steel.

Notice that when you click them, the white circles ⊙ will become filled ⊙ to indicate that your choice has been accepted. These circles are known as radio buttons and are used when you must choose between two or more items that are mutually exclusive. To select a radio button, you may click on the circle itself or on the adjacent text.

Next we need to decide how we want to define the cross sectional properties of the member. We can assign a shape directly, or define our properties based on section sets. Since we know we want to start with W12x53 column we will start by assigning our column shape directly, but will later use Section Sets for our wood truss and concrete members.

Click on Assign Shape Directly to bypass the section sets and assign an explicit shape for these new members. Double click on the white box under **Start Shape** and type the following:

W12x53

We will discuss the Member Type, Design List and Material choice later in the tutorial. For now, just enter the following.

Click on the ▼ arrow just below the text that says, "Type" to open the drop down menu for member type. Select **Column**.

Click on the ▼ arrow just below the text that says, "Design List" to open that down menu, and select WF12.

Finally, click on the ▼ arrow just below the text that says, "Material" to open that and select A992.

When you are finished, the dialog should look like the following:
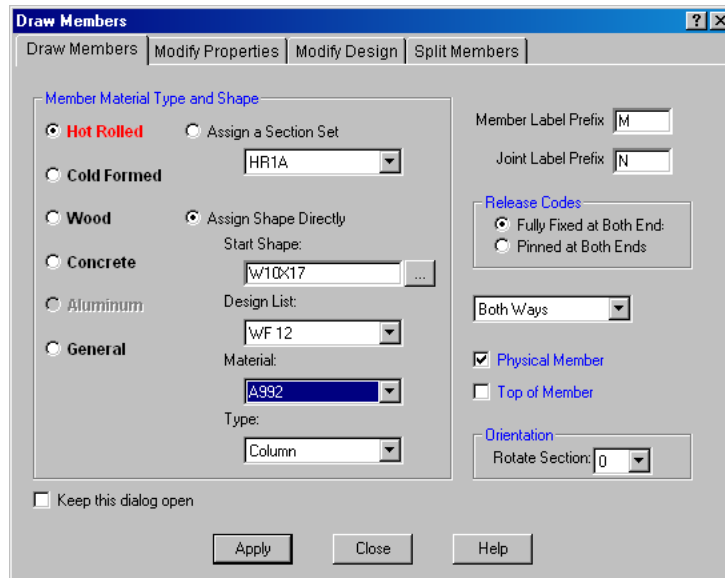
Member Releases

To the right of the Section Set are the member's **Release Codes**. The release codes 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 end joints).

Note

- 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.

We will use fixed ends, so there is nothing to change here. Let's start drawing the columns.

Click **Apply** and we are ready to start drawing**.**

Notice that your mouse cursor has changed to 🖉 and the **Draw/Modify Members** 📐 button on the **Drawing Toolbar** is "down".

Drawing Grid

RISA-2D will open with a 30 x 30 grid spaced at one foot. This would work with our model but let's change it to make it easier to find our column and beam locations.

Click 📐 on the Drawing Toolbar to redefine the grid. Use your mouse to select Drawing Grid tab if it is not already selected.

The drawing grid may be specified in plan or in either elevation. For our model we will leave the drawing grid in the XY plane.

To make our grid, 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 (including the commas):

> **3, 4@3.75, 3**

Now double-click on the current Y-Axis entry of "30@1" and type (including the commas):

> **10, 8.5, 4**

The dialog should now look like this:
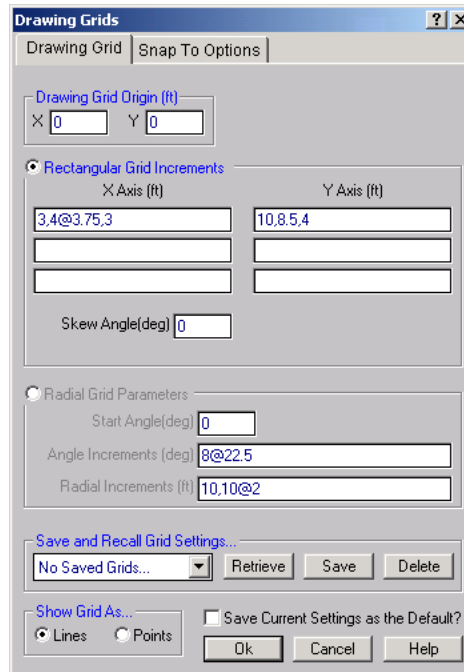
What we've done is define a drawing grid with an initial X-increment for our truss points of 3 ft followed by 4 equal increments of 3.75 ft each and a final increment of 3 ft. Note the use of the "@" symbol to define a series of equal increments. Our Y increments are set at the beam, the bottom chord and the top chord.

Other things to notice here are that you may specify a radial grid and you may save the grid layout to use in other models. You may also specify that the layout should be the default grid layout each time the program starts up.

Click **OK,** which closes the dialog. Also close the **Data Entry** toolbar in the upper right corner by clicking the **Close** ☒ button at the top of the toolbar to the right of the title "**Data Entry**".

Drawing Members

RISA-2D displays the coordinate location of the mouse in the lower right corner of the status bar (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.

We will provide sketches showing what is to be drawn, as well as dictating the required operations in terms of grid coordinates. For your model to be the same as that in this tutorial it is important to draw just as it is described in here. Watch the coordinates display at the bottom right to make sure you're moving to the right location.

We will start by drawing a line from point A to point B as shown here:
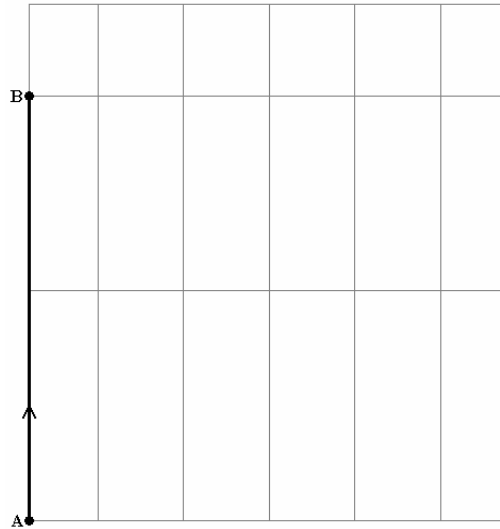
Move the mouse to the lower left corner of the grid (point A, location 0,0) and click once.

You should now have a line drawn from that point to the mouse, this line follows the mouse as you move it. Also a joint 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, so try again.

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

The line that follows the mouse should now be anchored to this new grid point, joint N2, and your screen should now look like the image on the following page:
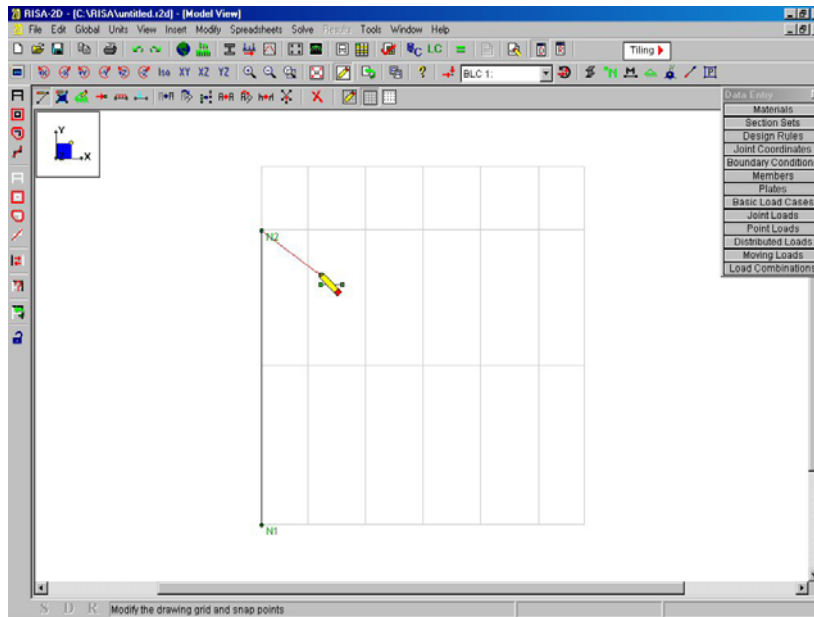
Now is a good time to point out the **Undo** button on the **RISA Toolbar**, which will undo any mistakes that you make.

Let's continue with our drawing. 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. The cursor indicates we are still in drawing mode.
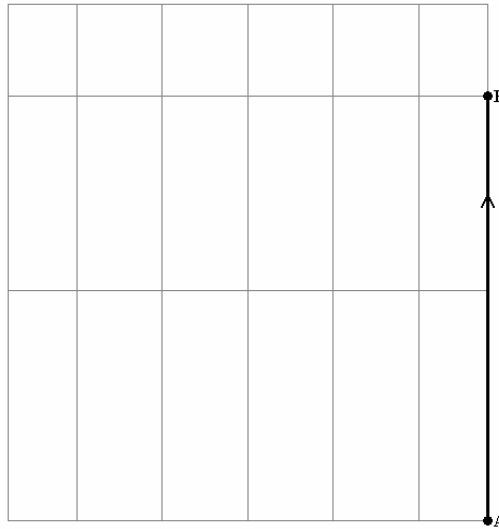
Move the mouse to the lower, right grid point (A) at coordinate (21,0) and click. Now move straight up to the grid point (B) at coordinate location (21,18.5) and click again, then click the RIGHT mouse button.

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

Before drawing the truss we will change the way that members are shown and color-coded by their section sets.

Click on the **Set Plot Options** button ⊞. It is the first button on the **Window Toolbar** (the second horizontal toolbar).
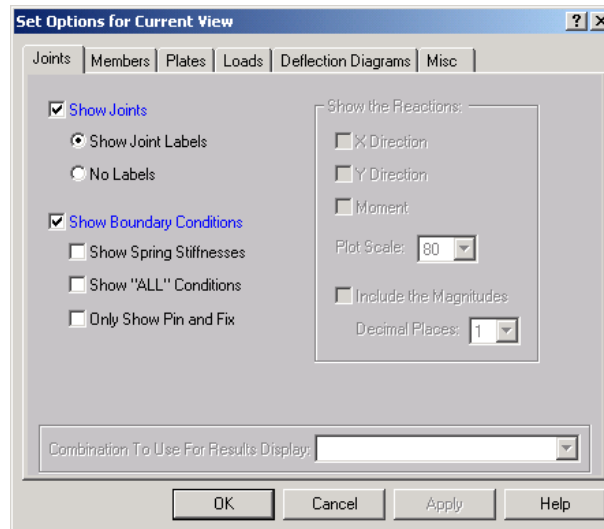
We'll discuss the plot options in more detail later.

For now, click on the second tab at the top, which is labeled **Members.** Now click on the 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-2D to color code the members by their section sets.

Now the dialog box looks like this:
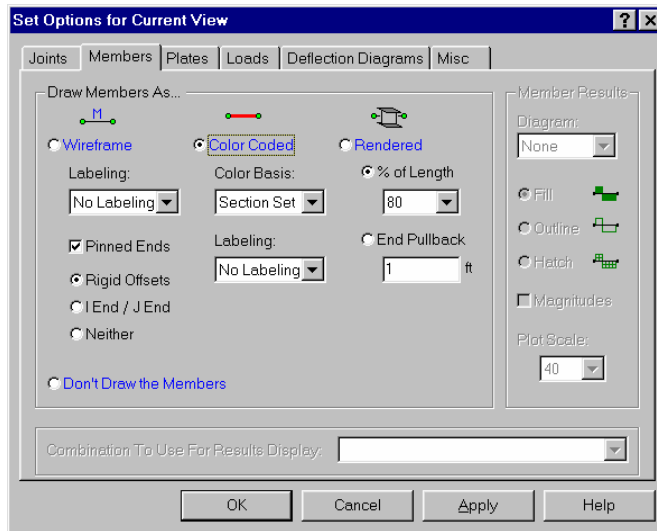
Now click on **OK** to close the **Set Plot Options** dialog and go back to the plot.

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 any section sets yet, but that is about to change. The "na" color used for our columns indicates that they were not defined with section sets.

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

Press the button to bring up the **Draw Members** dialog again.

The **Section Set** field is simply the label that refers to the cross-sectional properties to be used for the members. We will place the members in the sets as we build the model and define the properties later on.

Select **Wood** and then click on to assign a section set for our new members.

Notice how the currently selected Section Set now reads "WOOD1A". The drop down list for **Section Sets** and **Material** automatically changed when we selected **Wood** for our material type. The program knows that we have selected our **Material Type**. It has, therefore, adjusted the other menus and entries to be consistent with our material choice.

The dialog should look like this:
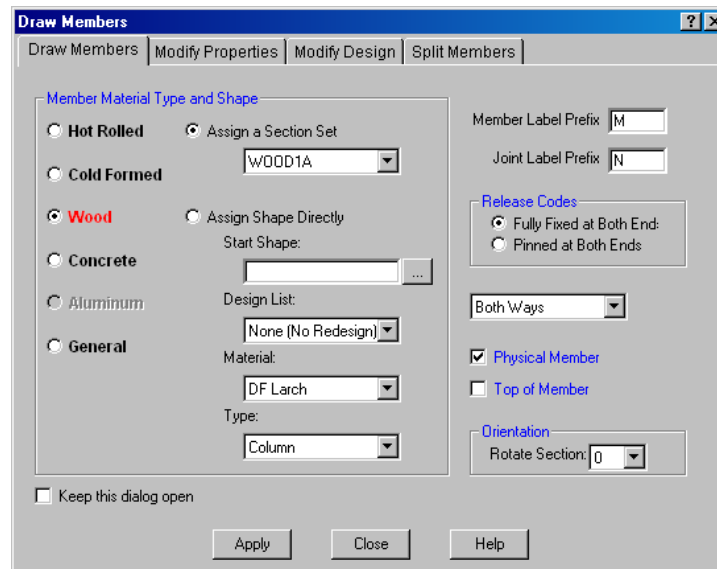
Click **Apply** to start drawing again and we are back at the model view.

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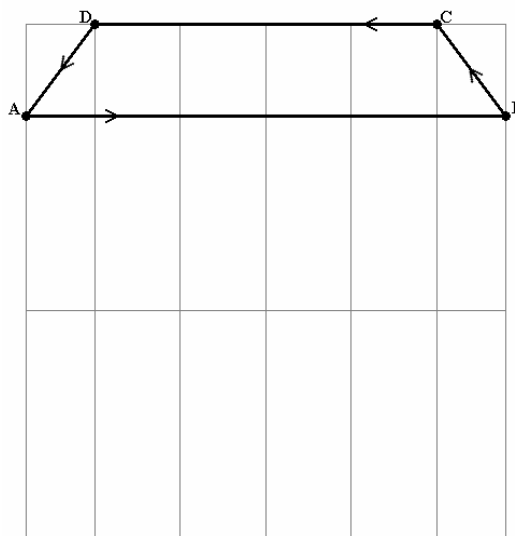
To begin drawing the truss chord, move the mouse to the top of the left side column at joint N2 (point A) at grid location (0,18.5) and click. Now move to the right to the top of the right side column at joint N4 (point B), grid location (21,18.5), and click again.

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 continue drawing new members. The truss chords are being drawn with a different color because of the different section set we are using.

Move up and to the left to grid location (18,22.5) (point C) and click again. Now move left to grid location (3,22.5) (point D) and click again. Finally, move down and to the left to grid location (0,18.5) (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 draw the web members. We will use the WOOD1A section set again, but will later change this assignment to give the web members different properties from the truss chords.

Another way to get back to the **Member Properties** dialog box is to use a specially tailored menu called the shortcut menu. This menu is accessed by clicking the RIGHT mouse button where you are working. RISA-2D 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:
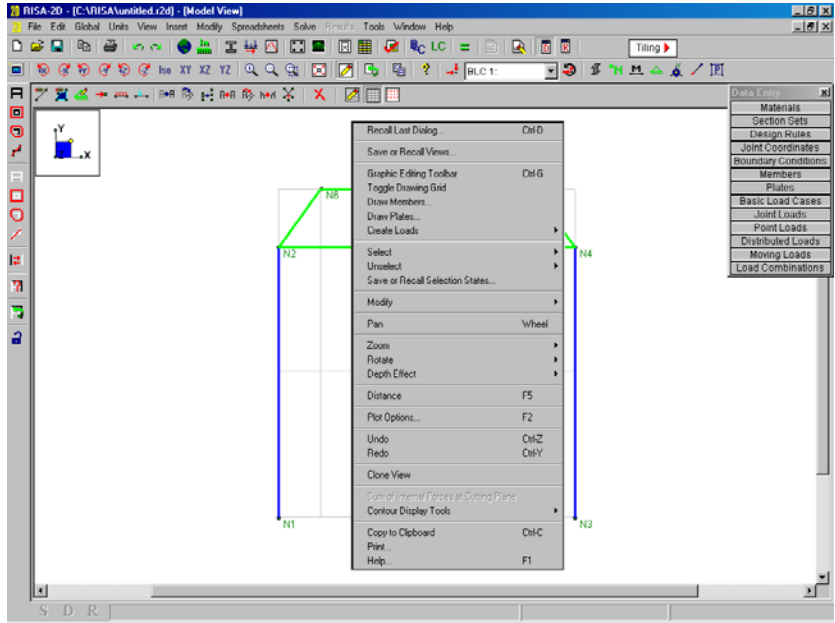
This menu brings relevant options right to where you are currently working. Notice that all of the options have to do with the view that we are currently working with. If we were working in a spreadsheet the choices would be relevant to that spreadsheet.

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

Now with the left mouse button click on **Recall Last Dialog** and the **Draw Members** dialog is opened. The web members will be pinned at both ends so click on "**Pinned at both ends**".

The dialog now looks like this:
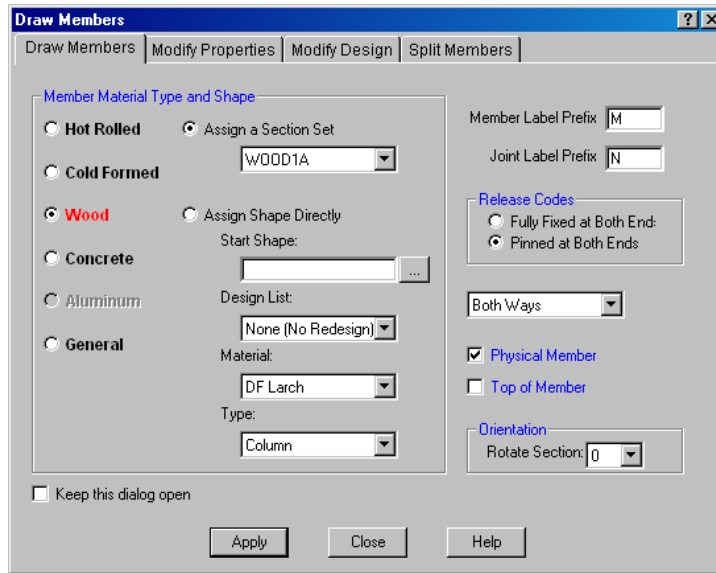
Now click **Apply**

This sketch shows what will be drawn next; lines from A to B, then to C, etc. ending at point F:
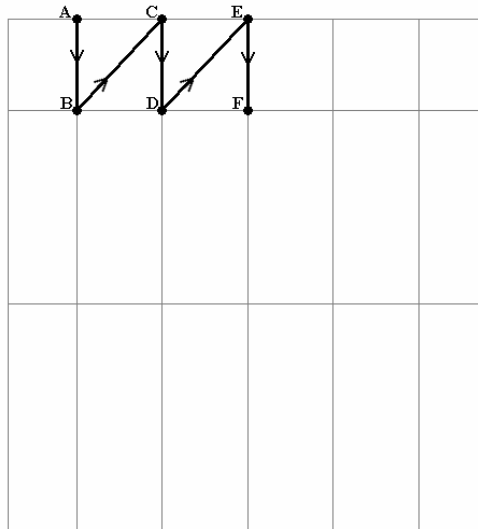
Move to point A, grid location (3, 22.5), and click.

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

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

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

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

Now move down to point F, grid location (10.5, 18.5), 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:
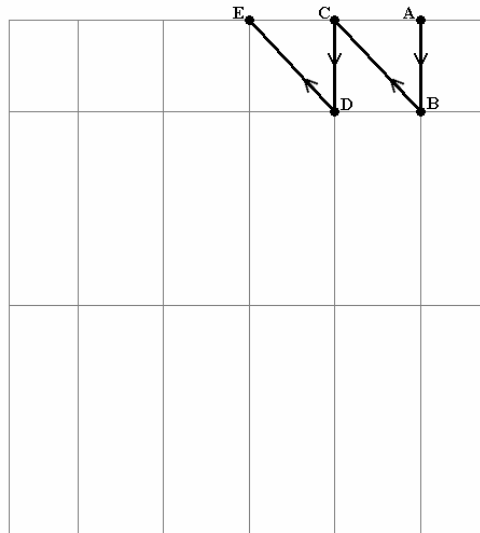
Move to point A, grid location (18, 22.5), and click.

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

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

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

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

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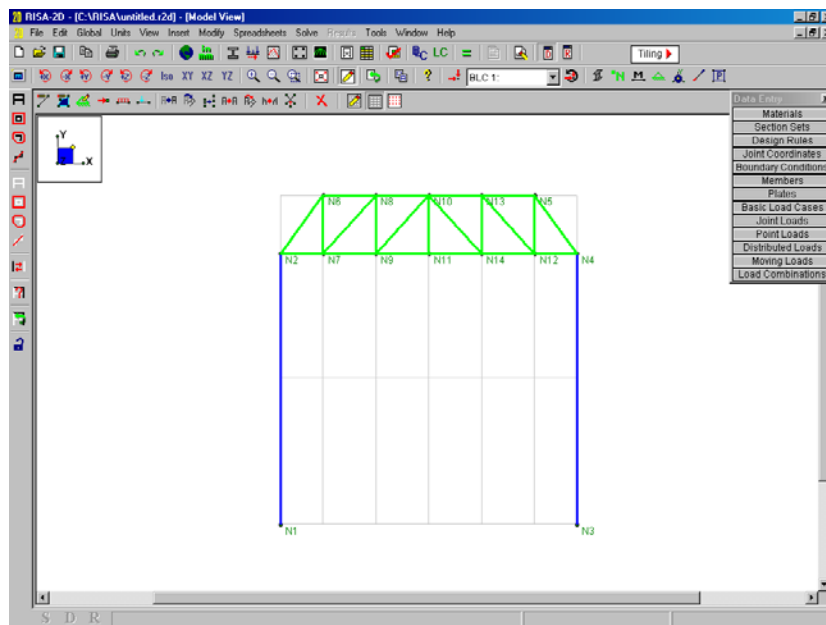
Physical Members

You may notice that we have defined our truss chords with one member, rather than multiple members between each panel point. RISA-2D's **Physical Member** feature allows us to do this and much more.
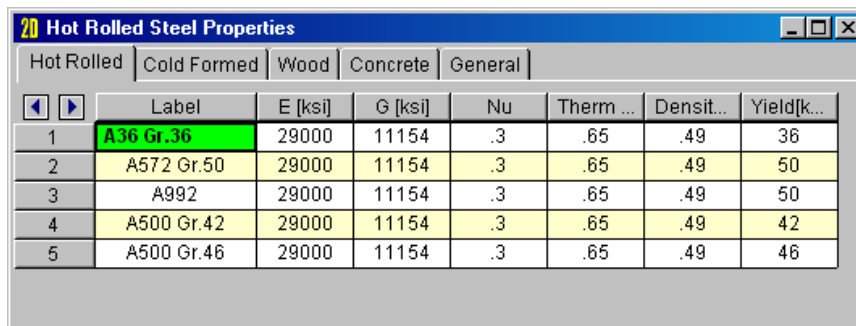
Physical Members automatically connect to the members that frame into them. This means that you don't have to use multiple members to model what is one Physical Member in the field. Later when changes are made to the model you don't have to work with multiple smaller members to make modifications. Most important, the results for these members will not be spread out over multiple members, making it difficult to find design values. RISA-2D does all of the bookkeeping behind the scenes letting you see your model as it really is.

RISA-2D starts you off with one section set for each basic material type. We will have to create new ones as we go along. Creating new sections is accomplished in the **Sections** spreadsheet**.** While we are there we will define the section properties for all the truss members that we have defined thus far. We have not reviewed our material properties either, which will be done in the **Material** spreadsheet.

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

Material Properties

On the **Data Entry Toolbar** click on ⬚ Materials ⬚ and this window will open:

Notice how the materials spreadsheet is organized based on material type with the detailed information for each type given on a separate, tabulated page. Take a moment to review the information shown on the Hot Rolled tab. Click the Cold Formed and Concrete Tabs as well. As you review this information you can see that RISA comes with standard properties already defined for the most common materials used in the United States.

Note

- You may save any changes, to this and other spreadsheets, as the default values. If, for instance, you're working on a Canadian job with steel that conforms to the G40.21 spec then you can add this material and save it by clicking the 🖫 button on the **Window Toolbar**. New files will then start with this steel material automatically.

- To increase or decrease the width of a column, drag the line to the right of the column heading. You may also look to the status bar for a full explanation of the current column.
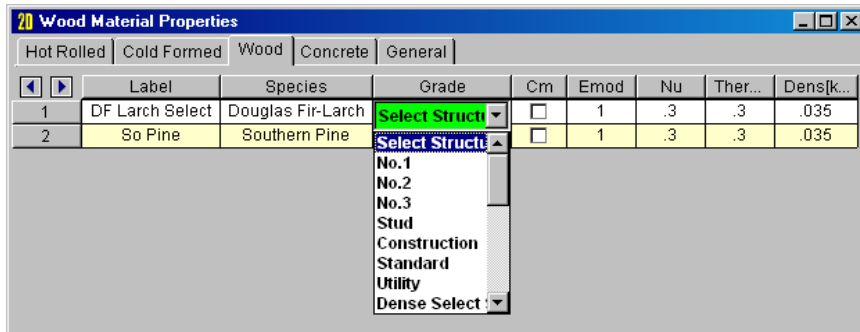
Wood Properties

For our model we will modify the wood material for the truss members. The complete NDS species database is built into the program. Therefore, you can select any valid species/grade combination and RISA-2D will provide the design values. We are going to change the grade from "No.1" to "Select Structural".

Select the Wood tab and type the following, using the tab button on the keyboard to advance between fields:

DFL Select, [TAB], [TAB]

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

Click the ▾ arrow to open the wood grade menu and then scroll up to pick "Select Structural" as shown in the following figure:

You could use the **Save as Defaults** button 🖫 to make our changes the default values for new models. Now that we've defined our material properties, we'll use them to define our members and section sets.

Design Rules – Size/UC

RISA-2D has the ability to optimize hot rolled, cold formed, wood and concrete members. This means RISA-2D will suggest a lighter, more efficient shape for each of our members. When suggesting an alternate shape, the program will only consider shapes that are included in the given Design List. For example, our Beam-Z section set defined with the "Wide Flange" redesign list. This list only contains wide flange sections from the AISC Manual of steel construction. This means that the program will never suggest that a channel or H pile would be a more efficient shape. For information about editing the existing Design Lists or creating your own custom design lists, refer to the General Reference manual.

Next we will specify some additional restrictions on the what types of sections the program will choose for redesign. We do this on the **Design Rules** spreadsheet.

On the **Data Entry** toolbar press [Design Rules] . The spreadsheet will open as shown below:
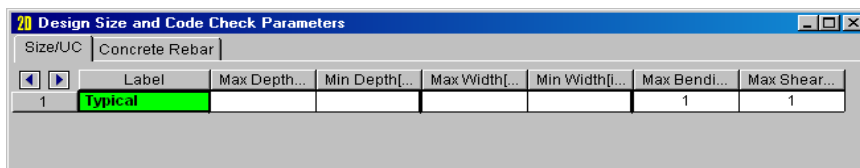
Type the following:

     Col-Rules, [TAB], [TAB], 12, [TAB], [TAB], 10, [ENTER]

     Steel-Bms, [TAB], [TAB], 12,[ENTER]

     Conc-Bms, [TAB], [TAB], 12 [TAB], [TAB], 10, [ENTER]

     Truss-Rules, [TAB], 10, [TAB], 3.5, [TAB], 6, [TAB], 1.5

This spreadsheet should look like the figure below:
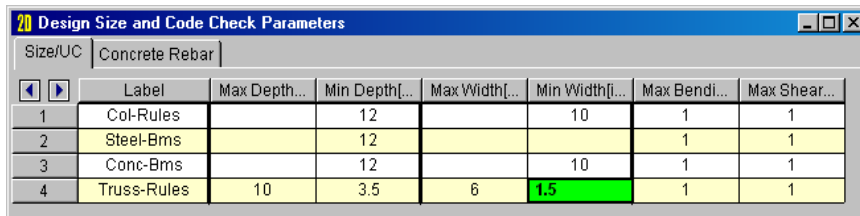
Notice there are several parameters available to control the suggested alternate shapes. You can define acceptable member depth and width ranges, and also code check ranges.

Design Rules – Concrete Rebar

In addition to controlling the size of your concrete members, RISA offers you the ability to control what reinforcing and cover are used.

Next click on the **Concrete Rebar** tab.

These entries shown here will control the size and placement of the flexural and shear reinforcement used in the design of our concrete beams and columns.

Click on the first cell in the **Min Flex Bar** column and use the drop down menu to change the minimum bar size for our columns to a #6. Then click on the 3rd cell in the **Max Flex Bar** column and use the drop down menu to change the maximum bar size for our beams to a #9.

	Label	Min Flex Bar	Max Flex Bar	Shear Bar	Flex Steel[ksi]	Shear Steel[ksi]	Legs per Stirrup	Top (Colu...	Bottom Co...	Side Cove...
1	Col-Rules	#6	#10	#4	60	60	2	1.5	1.5	1.5
2	Steel-Bms	#5	#10	#4	60	60	2	1.5	1.5	1.5
3	Conc-Bms	#5	#9	#4	60	60	2	1.5	1.5	1.5
4	Truss-Rules	#5	#10	#4	60	60	2	1.5	1.5	1.5

Although rules for Concrete Rebar are given for each Design Rule that we have created, these rebar entries will be ignored if they are applied to anything other than concrete members.

Section Sets Spreadsheet

On the **Data Entry Toolbar** click on [Section Sets] and this window will open:

	Label	Shape	Type	Design List	Material	Design Ru...	A [in2]	I (90,270...	I (0,180) ...
1	HR1A	W10X17	Beam	Wide Flange	A36 Gr.36	Col-Rules	9.71	36.6	170

Repetitive members are inherent in structural design and **Section Sets** provide an easy way to handle them. This will become even more apparent when we have RISA-2D pick new/optimized member sizes. Our truss has only nine web members but what if we had three trusses and we had to change the section, which we will do 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 member properties.

Hot Rolled Sections and Database

We already have one section set for each material type. Let's add some section sets and change the labels so that they are more meaningful. The active cell should already be the cell that contains the text "HR1". We haven't yet defined any members with this section, but will be using it for the beams in our steel moment frame.

Give the section a more descriptive label by typing the following:

Stl-Beam, [ENTER]

The spreadsheet should now look like this:

|	Label	Shape	Type	Design List	Material	Design Ru...	A [in2]	I (90,270...	I (0,180) ...
1	Stl-Beam	W10X17	Beam	Wide Flange	A36 Gr.36	Col-Rules	9.71	36.6	170
2	HR2	W10X17	Beam	Wide Flange	A36 Gr.36	Col-Rules	9.71	36.6	170

Note that 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.

Click the cell that says **A36 Gr.36**, then click the ▾ arrow next to open the drop down menu of hot rolled materials. Select A992 for the beam material as shown in the figure below:
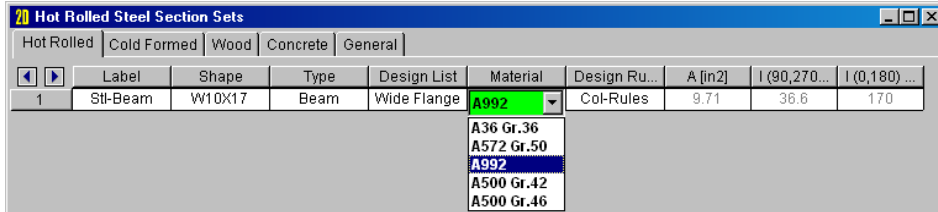
Notice that the Section Set data consists of a number of entries. The **Shape** entry indicates the current size that will be used for the member. The **Design List** and **Design Rules** entries tell the program what list of shapes to use and what design restrictions to impose when it tries to re-design/optimize the member. Refer to Appendix A of the General Reference Manual for more information on Design Lists.

Click the ▾ arrow next to the text that says "Col-Rules" to open the drop down menu of Design Rules. Select "Steel-Bms" for the Design Rule.

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

By default the current shape is a W10X17. If you know what shape you want to use, you could type it in directly. But, if you don't know the database by heart you may not know what shape you want until you see it. Therefore, we will open up the database and take a look at the full list of available shapes.

Click on the first cell in the Shape column and your screen will look like this:
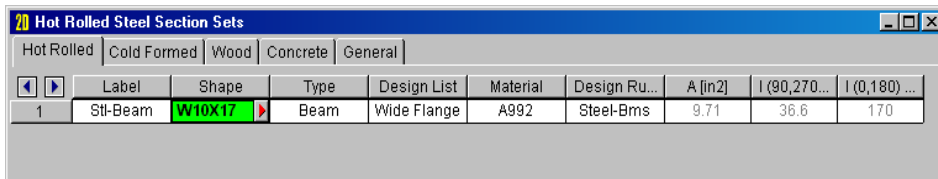
Click the ▶ arrow in the cell and a dialog is opened that lists the shapes in our hot rolled database.
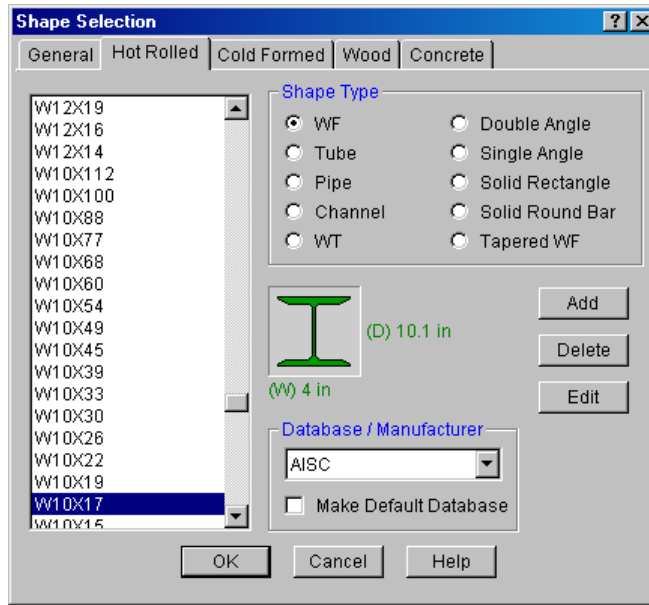
The left side of the dialog contains a list of available shapes. The shape highlighted in blue indicates the currently selected shape which should be a W10X17.

The rest of this dialog 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 many others.

Use the **Page Up** and arrow keys (↑,↓) on your keyboard to move through the list. Click **OK** when the highlighted shape is a W12X19.

You will see that W12X19 is now in the **Shape** column of the 2nd row in the Section Sets spreadsheet. The rest of the properties will be updated as soon as you leave the cell.

Cold Formed Sections and Database

Use your mouse to select the Cold Formed Tab. Type "Girt" to change the Section Label from "CF1A" to something more informative. Your screen should now look like this:

We will choose the shape from the shape database so click in the **Shape** column and click the ▶ arrow to open the Cold Formed Shape Database.

You will notice that the layout of this dialog is very similar to the Hot Rolled Steel Database, with the left side of the dialog containing a list of available shapes. The rest of this dialog contains selections for **Shape Type** and **Shape Database**. The current database is the AISI 1999 which is appropriate for our example.

Select **CS** as the shape type (in the middle). Finally choose **14CS3.75X135** as the selection at the top of the list of available shapes.
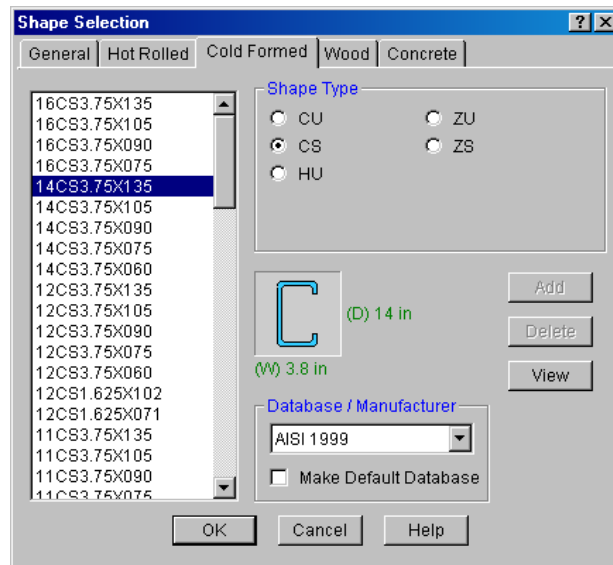
Click **OK** and you will return to the Sections Set spreadsheet.

Again, the **A** and **I** properties will be updated as soon as you leave the cell.

Click on the first cell in the **Design List** column to make this the active cell. Then click the ▼ arrow next to the text that says, "CU" to open the drop down menu of design list. Select **CS** as the design list.

This last selection will insure that the when the program performs member re-design/optimization it will suggest a new CS from the appropriate list of shapes.

Now, click on the first cell in the **Design Rules** column to make this the active cell. Then click the ▼ arrow next to the text that says, "Col-Rules" to open the drop down list. Select "Steel-Bms" for the design rules.

Wood Sections and NDS Database

Now we will focus on the Wood Tab of the **Section Sets** spreadsheet where we will define the sections used for the chords and webs of the wood truss.

Click on the Wood tab of the Section sets spreadsheet and type the following:

> **Chord, [ENTER]**

> **Web**

The spread sheet should now look like this:

The design group for both members is currently set to **Rectangular**. While this is sufficient for the web members, we would like to use doubled members for the chords.

Click on the **Shape** cell in the first row and then click the ▶ arrow to open the dialog with the NDS Wood Database. Select ⊙ Multiple to indicate that our top chords will consist of a multi-ply section. The number of plies already set to "2" which is what we want.
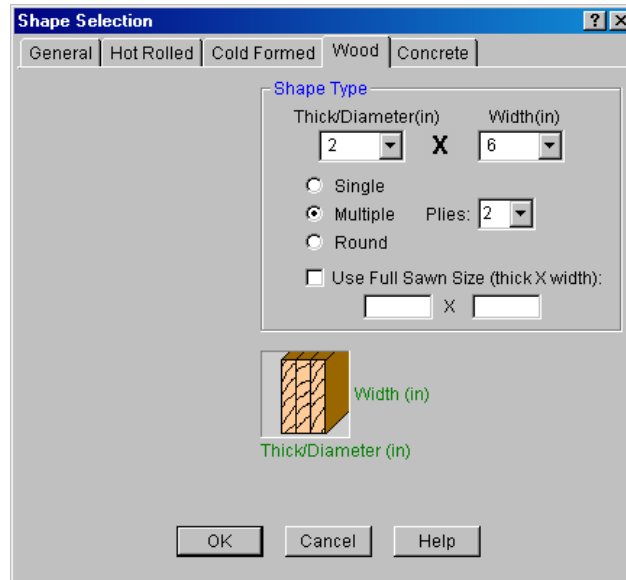
Click **OK** to return to the Section Sets spreadsheet.

We've now changed the current shape to a Double 2x6. If we want to program to suggest Double Rectangles when it performs member re-design/optimization, then we must also change the Design List.

Click on the first cell in the **Design List** column to make this the active cell. Then click the ▼ arrow next to the text that says, "Rectangular" to open the drop down menu for design lists. Select **Rectangular Double** for the Design List.

This last selection will insure that the when the program performs member re-design/optimization it will suggest another doubled up member for our chords. To control the depth and thickness of the suggested shapes, we will change our Design Rules.

Click on the first cell in the **Design Rules** column to make this the active cell. Then click the ▼ arrow next to the text that says, "Col-Rules" to open the drop down list. Select "Truss-Rules" for the design rules. Repeat these actions to change the Design Rules for the Web members as well.

Your spreadsheet should now look like this:

Concrete Sections

Click on the Concrete tab.

Take a moment to review the information given. Just like with the steel and wood tabs, the information consists of a section Label, Shape, Design List, member Type, Material and Design Rules.

Now type the following:

Conc-Col, [TAB], CRECT12x12, [ENTER]

Conc-Bm, [TAB], CRECT12x8

The spread sheet should now look like this:

Select the cell that says "Beam" in the first row. Then, click on the ▼ arrow next to the text to open the drop down menu for member type. Select **Column**.

Remember that our concrete reinforcing and cover information was specified as part of our Design Rules. This currently set to Col-Rules for both sections. While this is correct for Conc-Col, we would like to use a different rule for the beams.

Click on the second cell in the **Design Rules** column to make this the active cell. Then click the ▼ arrow next to the text that says, "Col-Rules" to open the drop down list. Select "Conc-Bms" for the design rules.

Your spreadsheet should now look like this:

Multiple Windows
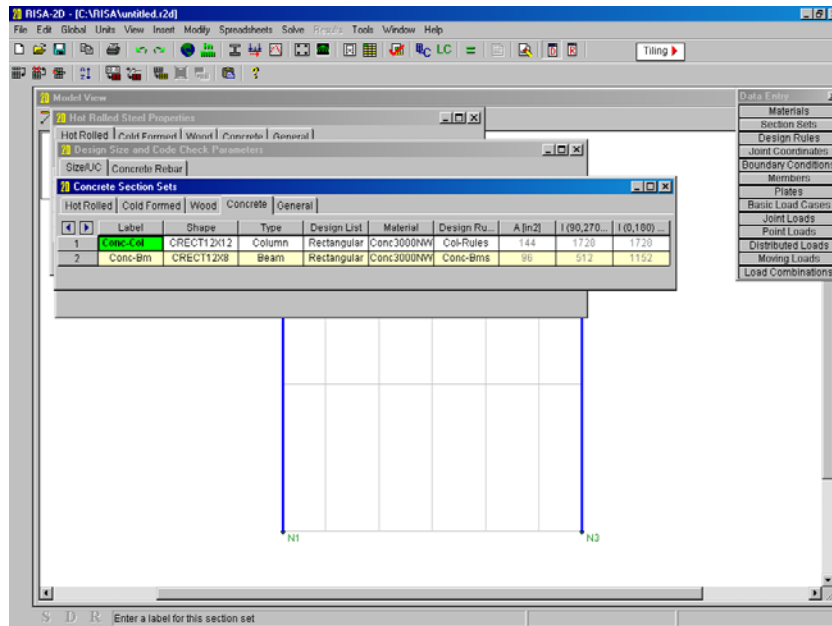
Now is a good time to talk about all of the windows that we have open on the screen. We have a model view and four spreadsheets. The "current window" is the one with the colored title bar (**Member Section Sets**) and the other title bars are gray. The windows are laid one on top of the other in the 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.
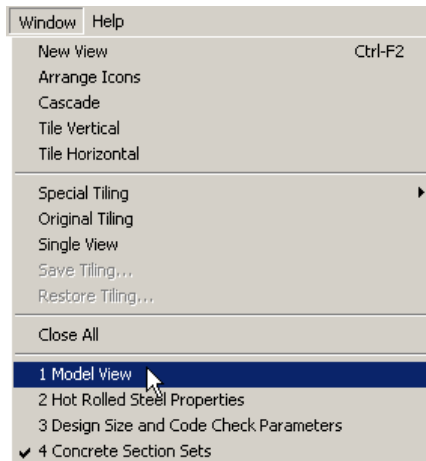
The top of the **Window** menu presents some arrangement and tiling options that we will get to later. At the bottom of the menu the open windows are listed. Notice that they are numbered in the order that we opened them starting with the **Model View** and ending with the **Member Section Sets** we 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 ⬚Section Sets button in the **Data Entry Toolbar.**

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

You may close windows either by clicking the close ✕ button or by pressing the Ctrl and F4 buttons simultaneously.

Press the key combination "Ctrl-F4" to close the **Sections** spreadsheet.

Back in the model view our cursor is still ✎ which indicates that we are still in drawing mode. RISA-2D remembers what mode you are in for all the open windows. We need to draw the crossbeam but we are still set up for drawing web members however because that is what we were doing when we left the model view.

Press the key combination "CTRL-D" to recall the **Draw Members** dialog. We want to draw the crossbeam, so click ⬚**Hot Rolled** to change the material back to hot rolled steel. Click on ⬚Assign a Section Set and you'll see that the Stl-Beam section sets becomes selected.
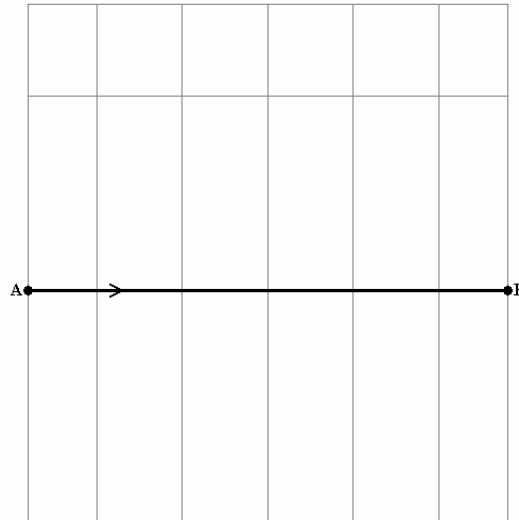
Finally, we want the beam to be fully fixed at both ends so select that option.

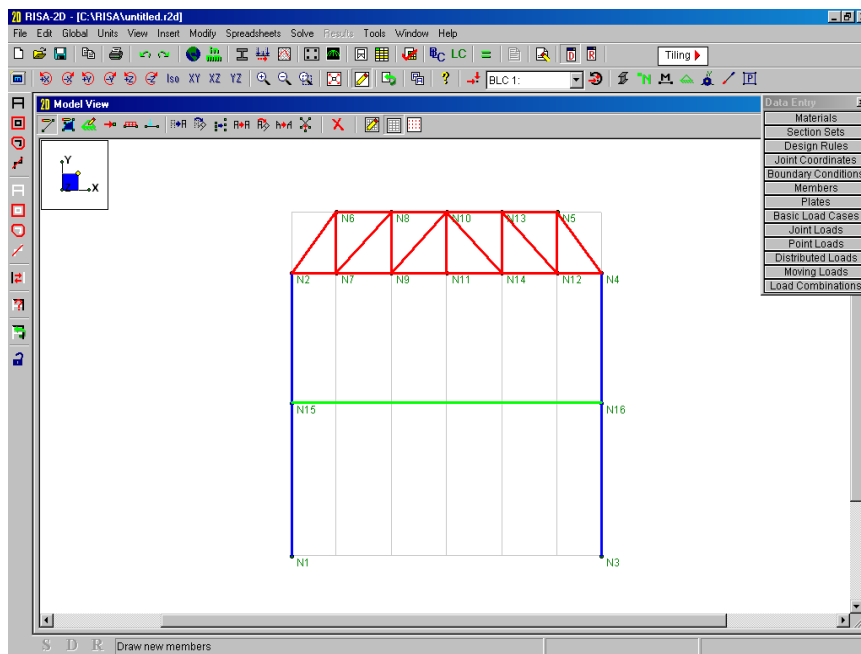When you are done, you screen should look like the following:
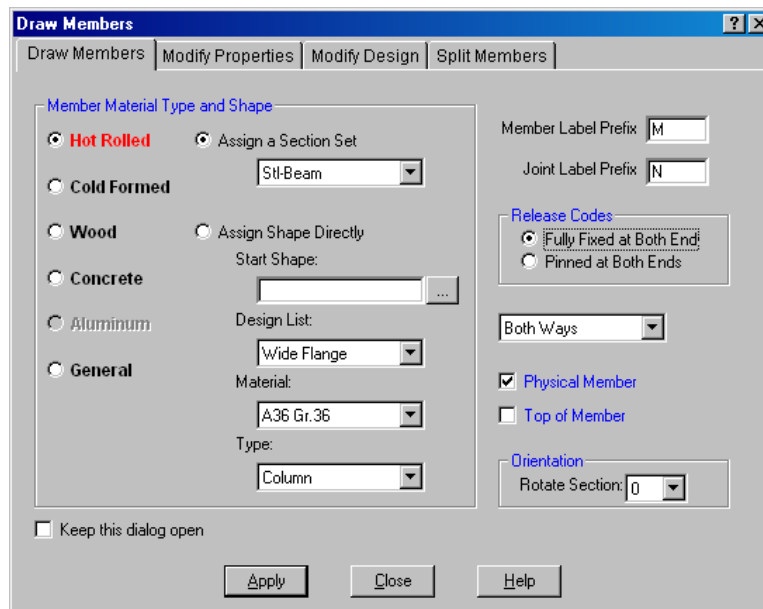
Click **Apply** and we go back to the model view.
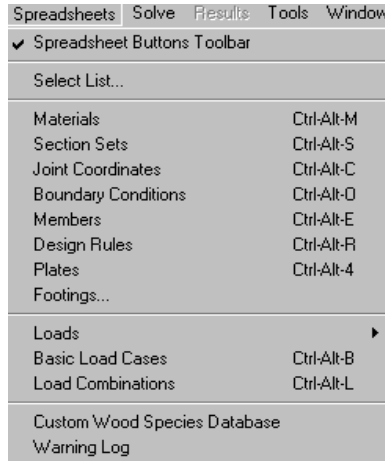
Here is what you'll be drawing; a line from A to B:

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

Entering Spreadsheet Data

In addition to its graphical editing features, RISA-2D allows you to add, organize or sort your data using a series of powerful spreadsheets.

Click the **Spreadsheets – Joint Coordinates** from the Main Menu toolbar.

This will bring up the Joint Coordinates spreadsheet as shown below:

Soon we will be adding lower concrete frame to support the rest of our structure. We could modify our grid to help us draw the members. Instead we will demonstrate some spreadsheet operations and create the additional joints that define the base of the frame.

Move to last line in the spreadsheet and type the following:

> **[ENTER]**
>
> **[TAB], [TAB], -10, [ENTER]**
>
> **[TAB], 21, [TAB], -10**

We'll spend some more time on the spreadsheets in Tutorials 2 and 3. For now we have all the joints we need, so we'll return to the Model View and continue drawing members.

Close the spreadsheet and return to the model view by clicking ☒ in the spreadsheet's title bar. Then click ⊡ to maximize the view plane.

You should now see the new joints we created below the existing columns.

Drawing Concrete Members

Press the key combination "Ctrl-D" to reopen the **Draw Members** Dialog.

Notice that the dialog has remembered the properties that we entered when we drew our steel cross beam. Let's change this information to be consistent with the new concrete member that we are going to enter.

Click ○ **Concrete** to change the material. If it is not already selected, click on ○ Assign a Section Set , and you'll see that the Conc-Col section sets becomes selected automatically.

When you are finished, the dialog should look like the figure below:
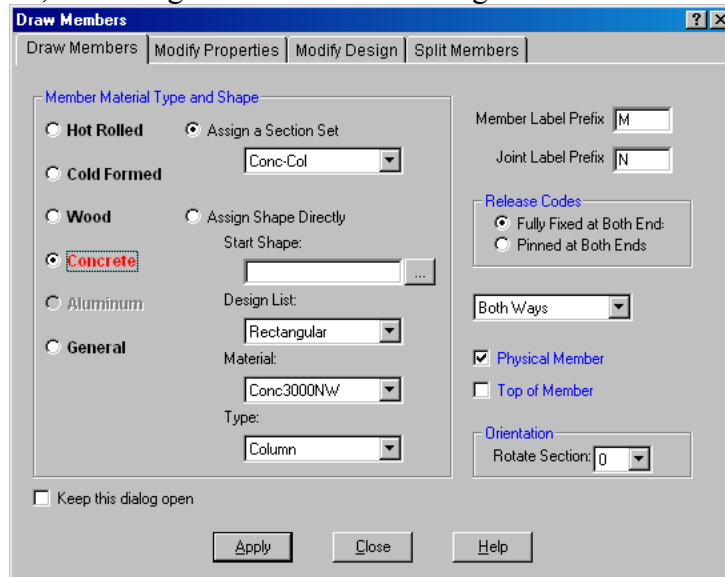
Click **Apply** and draw a member between Joints N17 and N1 which are circled in the image below:
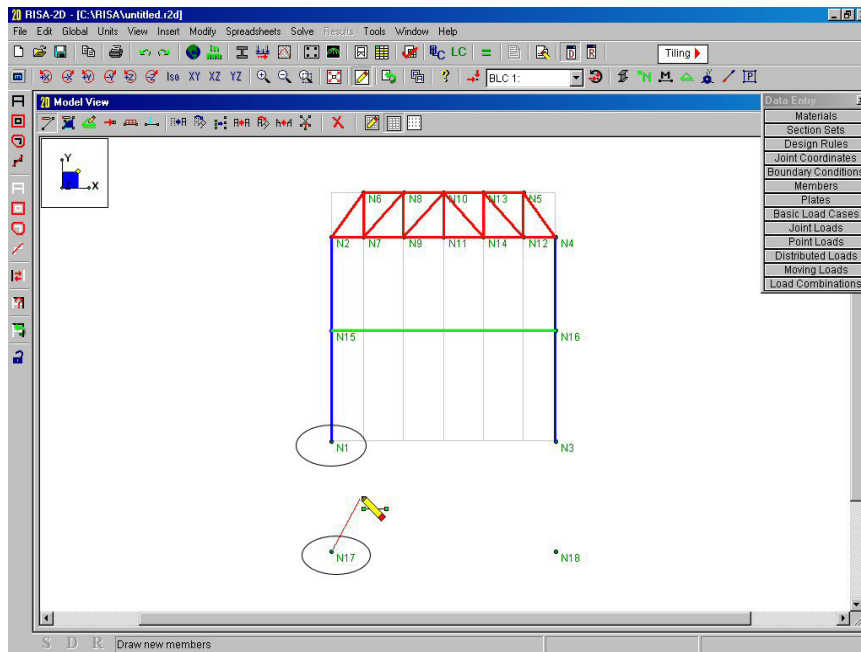
Continue drawing new members by clicking first on N3 and then N18. This will create a new member cross member from N1~N3 and a new vertical member from N3~N18. When you're finished, right-click your mouse.

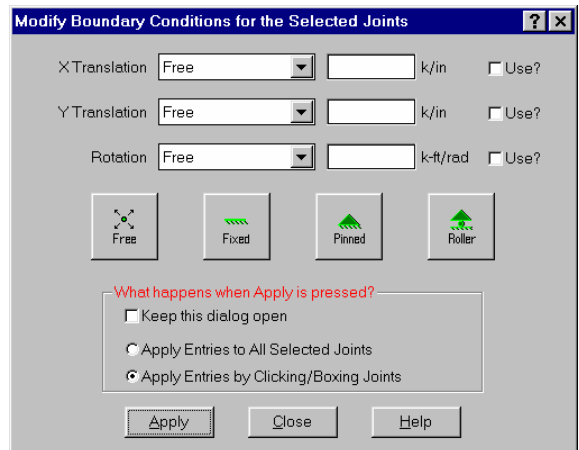At this point your screen should look similar to the one shown in the following image:
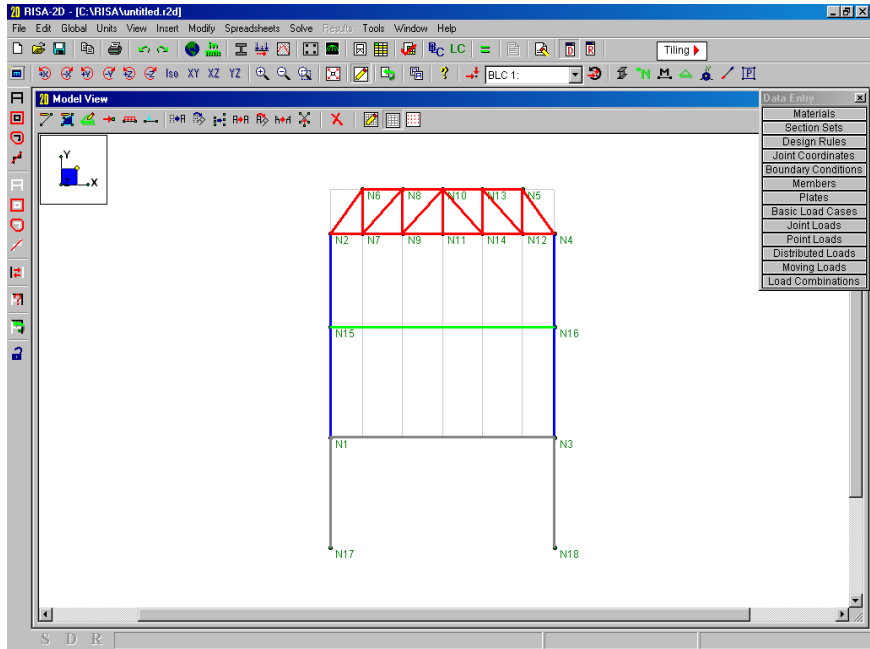
Boundary Conditions

We will now add boundary conditions at the base of the columns.

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

You will be presented with the following dialog:

Boundary conditions define how the model is constrained. All models must be attached to some external point or points of support. Boundary conditions are applied to particular joints. There are 3 degrees of freedom for each joint so there are 3 parameters provided one for each direction.

Click the first down [▼] arrow in the **X Translation** field and you will see all the options available for the boundary conditions:
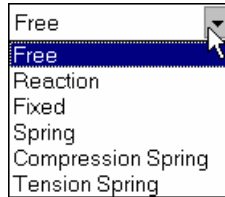
Reaction – The "Reaction" code specifies full restraint for the indicated direction. No movement will be allowed in the indicated direction for this joint and a "reaction" will be calculated and reported.

Fixed – The "Fixed" code also specifies full restraint for the joint in the indicated direction however 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 option models a spring attached to the joint in the indicated direction and must be accompanied by a spring stiffness. The units for the spring stiffness depend upon whether the spring is translational or rotational and are displayed next to the edit box.

For example, 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 and Y translations restrained, but we want the rotation unrestrained. This is a standard boundary condition that RISA-2D has built in.

Click the **Pinned** [image] 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-2D selection tools yet so just be aware that this is an important feature that we will come back to.

The dialog now looks like this:
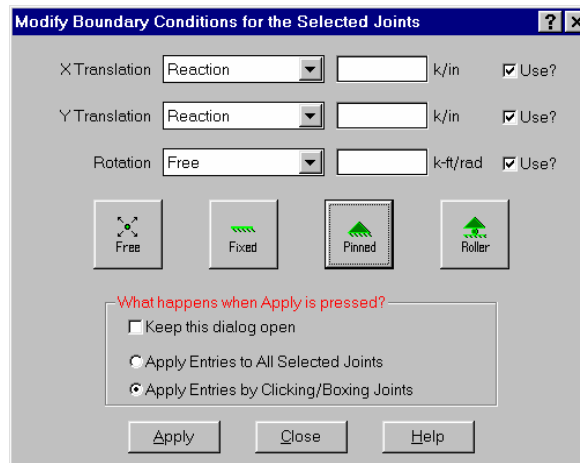
Click **Apply**.

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

Click on joint N17 and then on joint N18.

We are finished with the boundary conditions and your screen should look like this:
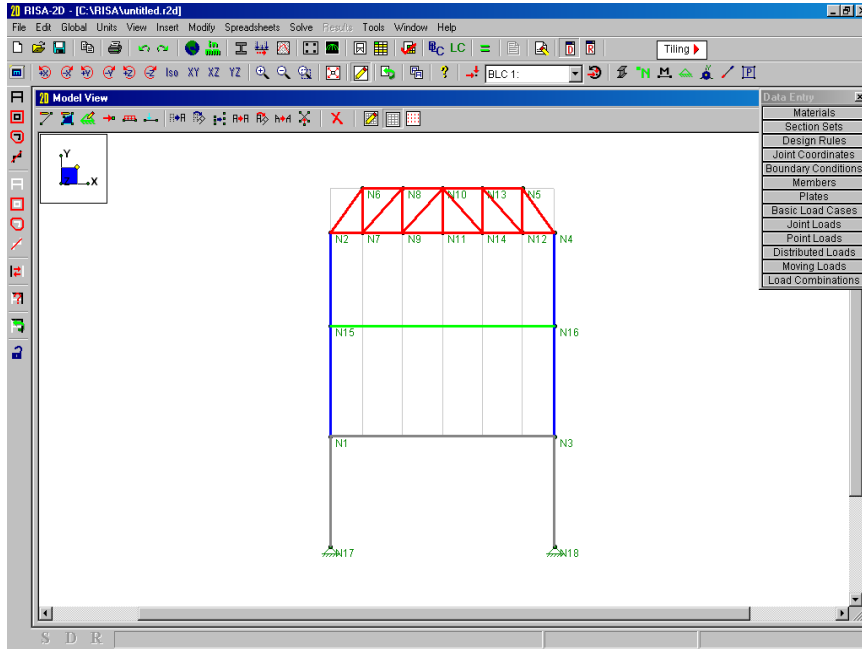
To see how these boundary conditions were recorded click [Boundary Conditions] to open the **Boundary Conditions** spreadsheet:

We could edit the boundary conditions here, or add new ones. 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 [icon] arrow to view the available options.

Saving Your Data

This concludes Tutorial 1. The next tutorial teaches you how to make modifications to the model.

If you wish to keep this file select **Save** from the **File** menu and name the file.

If you wish to stop and do Tutorial 2 later, select **Exit** from the **File** menu. Otherwise continue on and open the file at the beginning of Tutorial 2.

Tutorial 2 – Modifying

In this tutorial we will modify the 2D frame we defined in the last tutorial and use RISA-2D's graphic editing features to expand it into a two bay frame. Make sure you start off correctly by opening the file mentioned next.

After starting RISA-2D click the **Open File** ![button] button. In the **File Open** dialog open the "Tutorials" folder and select the file named "RISA-2D Tutorial 2 Starter." Click **Open** and close the **Global Parameters** window by clicking **OK** and we will begin working in the model view. Click ![D button] to close the **Data Entry Toolbar** if it is open. Your screen should look like this:
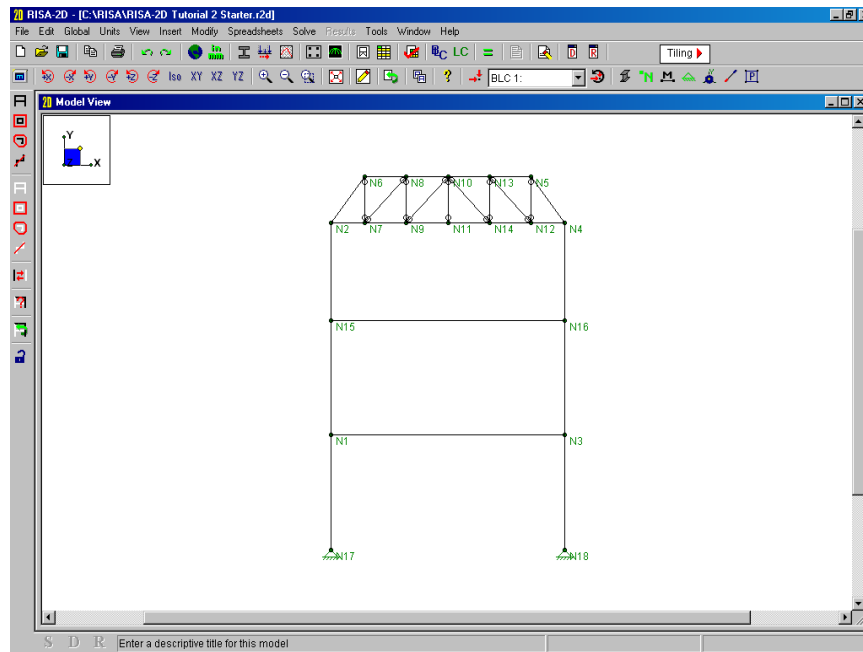

The first thing that we will modify is the end conditions at the tops of the steel columns. You can see in the model view that the web members of the truss are pinned at the ends. The circle at the end of each member indicates this. All other members are fixed at the end. We will change the column top ends so that they are pinned as well.

Making Changes to an Item

RISA-2D provides a quick and easy way to view and edit properties of any item in the model. You may simply double-click on any joint, member or plate to view and edit all of the information for that item.

Double-click the steel column on the left to obtain the following dialog:
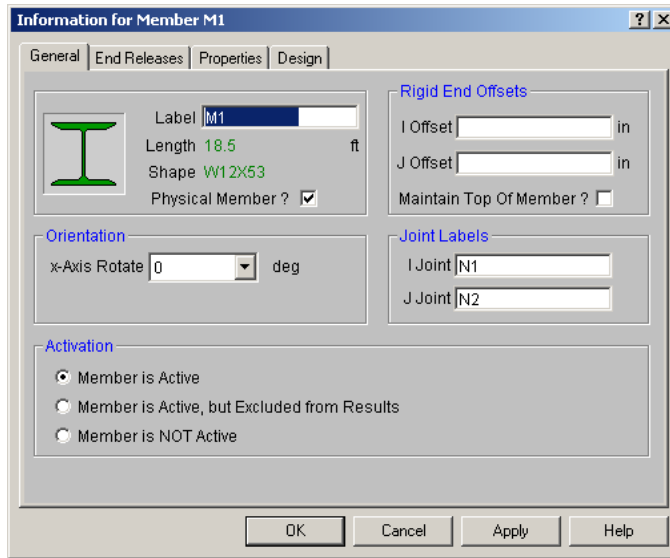
Click the tabs at the top to view the information on each page, some of which we have not covered yet.

After scanning the pages click the **End Releases** tab. Change the **J End Release** to **Pinned**. Click **OK** to close the dialog.

Now double-click the steel column on the right to change that one. Click the **End Releases** tab. Change the **J End Release** to **Pinned.** Click **OK** to close the dialog. You should now see pins at the tops of the columns.
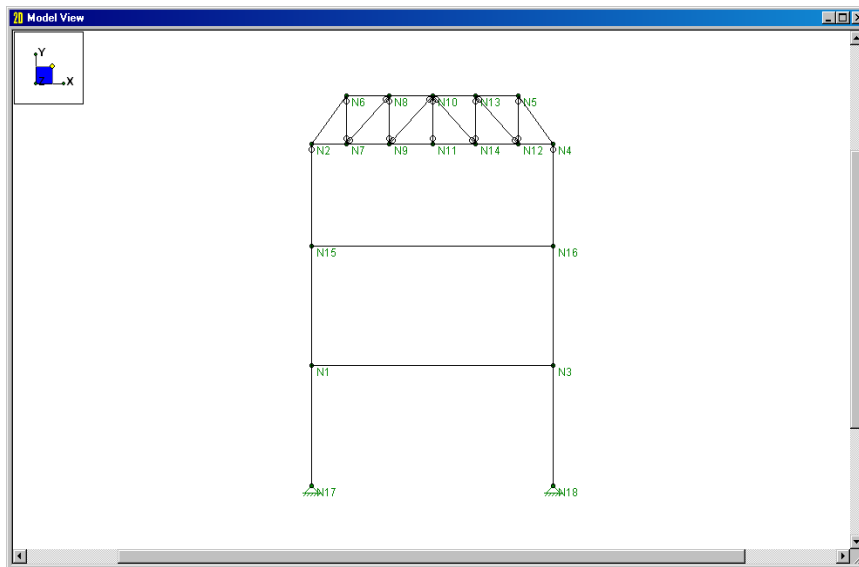
Remember you can make modifications to any joint, member or plate simply by double-clicking on it and making the adjustments. We will take a look at a joint before moving on.

Double-click on the bottom left joint to see information for joint N17.

Changing one or two items in the model is best handled in this manner. However, if you had to change a lot of members there is an easier way. This is where the graphical selection and editing tools in RISA-2D 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.**

Model View

Next we will learn how we can modify the model view. 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:
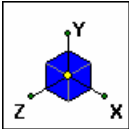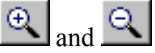
When you're finished experimenting place the model in an isometric view by clicking .

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.

If your mouse has a center wheel, RISA-2D also offers a feature that allows you to zoom in on a particular point. Simply place the cursor on the point you wish to zoom in on and rotate the mouse wheel forward. Rotate the wheel backwards to zoom back out.

Also, if you have a mouse wheel, RISA-2D supports a "grab and move" feature. Simply press down on the wheel (like a button) and move the mouse. RISA-2D "grabs" the image at the current cursor location and moves the image with the mouse.

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

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 into view by dragging the title bar. You are now working in a new view and the old one is preserved.

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, , are provided to help you view the loads. We will come back to this once we have defined our loads.

Toggling Items On and Off

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 joint labels.

Click to toggle the member labels.

Click [icon] to toggle the boundaries.

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

Click [icon] to toggle the I-J end flags.

Press Ctrl-F4 to close the view we created by cloning and return to the original view.

Plot Options

Now click the first button on the **Window Toolbar**: [icon].

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.

Choose the **Members** tab if it is not selected. Click on the down arrow next to the lists to view the options they give you.

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.

Options that are drawn in gray 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.

When you're done reviewing this dialog choose the Color Coded option and select Section Set as the color basis. Click OK to close the dialog and return to our model view.

Note

- You may make any plot options your default options by clicking the **Save as Defaults** button on the **Misc** tab.

Multiple Views

We demonstrated that you could 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** [icon] button and you will be presented with an original (not cloned) model view. Feel free to use the buttons on the **Window Toolbar**. You will see that you can adjust this view without affecting our original view.
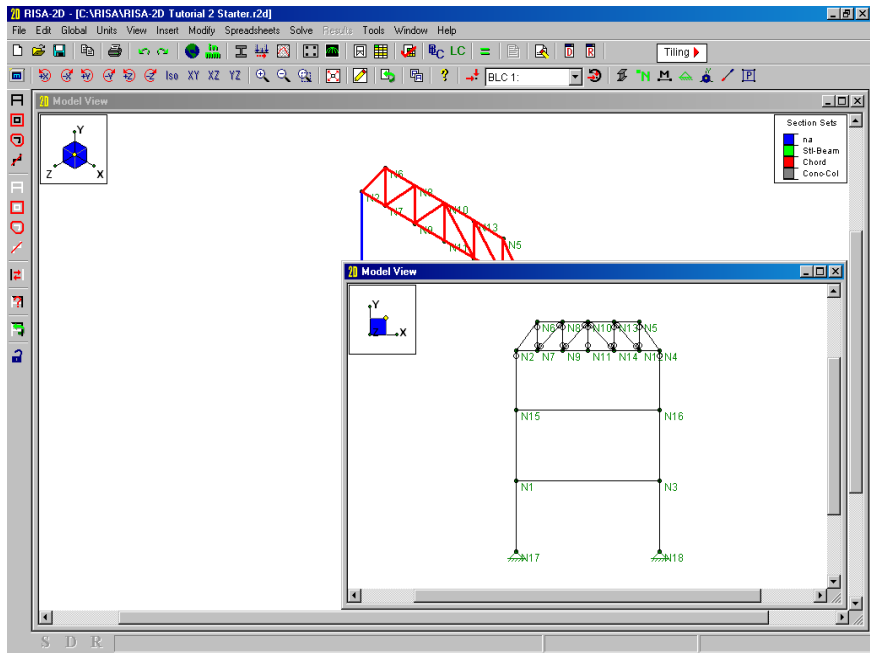
The importance of multiple views cannot be stressed enough. If you don't want to change your existing view but need to view the other side of the model, simply open a new window to view the other side. You can have a different drawing grid open in each view. For example you can draw beams on one grid in one view and columns in another. As you move between each view RISA-2D 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.

Selection Tools

Selections are important to modify and view only parts of the model.

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. All of our model elements are "selected" at this time.

Click on the steel crossbeam (the upper beam) that we just created and you will see that the colored line disappears and that a gray ghost line takes it's place.

Your model should now look like this:
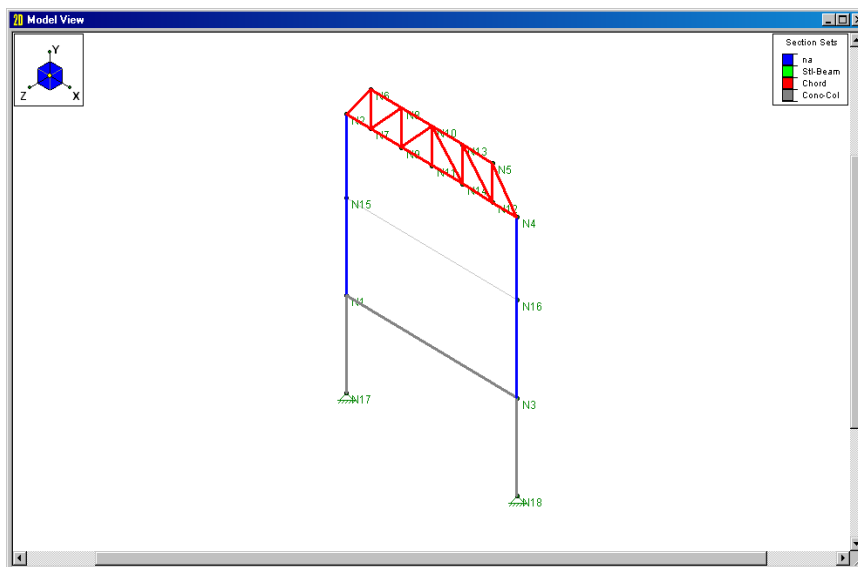
The crossbeam is now "unselected".

Currently your mouse cursor is the standard arrow ⬚ when in the model view. 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 joints so try this as well.

Rather than clicking one item at a time you may use RISA-2D'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 cannot affect 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 that we know about the selection tools we can use them to help define the model.

Making Changes to Selections

Click the **Unselect All** ⬜ button, to unselect the entire model. Then click the 🔲 button to change the model view to an XY. Then click 🔲 to maximize the view plane.

Click the **Line Select** 🔲 button.

Your cursor changes ⫯╟╟ to indicate that you are now in the line selection mode.

Click and hold your mouse button to create the start of your selection line near the left most truss web member. Move your cursor past the right most web member and release the mouse button.
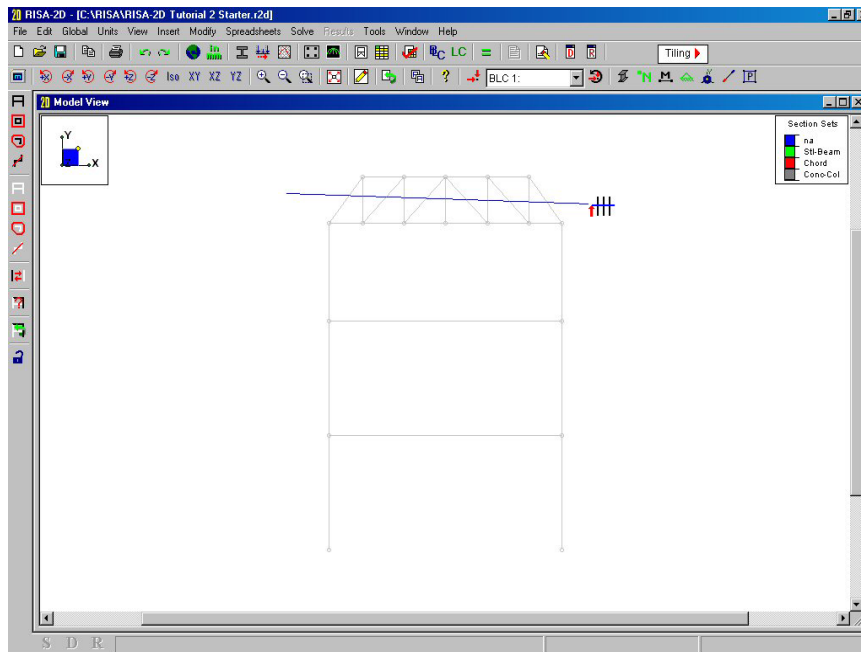
All of the truss web members should now be selected. We will use our current selection state to easily modify the members. Notice that the section set for all of our web members is listed as "Chord". This is because we only had one wood section set available to us when we first drew our truss. In the next section we will use the current selection state to easily modify these members.

Make sure that the 🔲 button has been pressed and that the **Drawing Toolbar** is active.

The **Drawing Toolbar** has different graphic editing features, some "**insert**" new data such as new beams or distributed loads. Other options help "**modify**" items that already exist in the model. Likewise the menu at the top of the screen has an **Insert** menu and a **Modify** menu. We will use the **Modify** tools to help us rotate our columns.
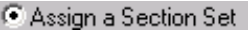
For these "modify" options a "**Use?**" checkbox is available for each item that may be modified.

☐ Use?
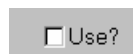
This indicates what parameters are applied. We saw this when we defined the boundary conditions. If the little box is not checked, the parameter *won't* be used.

Modify Members

Now click on the **Draw/Modify Members** button on the **Drawing Toolbar.** Select the **Modify Properties** tab to view the modification options.

Click on , and in the upper-left portion of this dialog. Then click on and change the **Section Set** from "Chord" to "Web".

Now click the box just to the right of the "Material Type and Shape" field that is labeled "**Use?**" so that the box is checked. Finally in the lower right choose **Apply Entries to All Selected Items.**

The dialog is now set and should look like this:
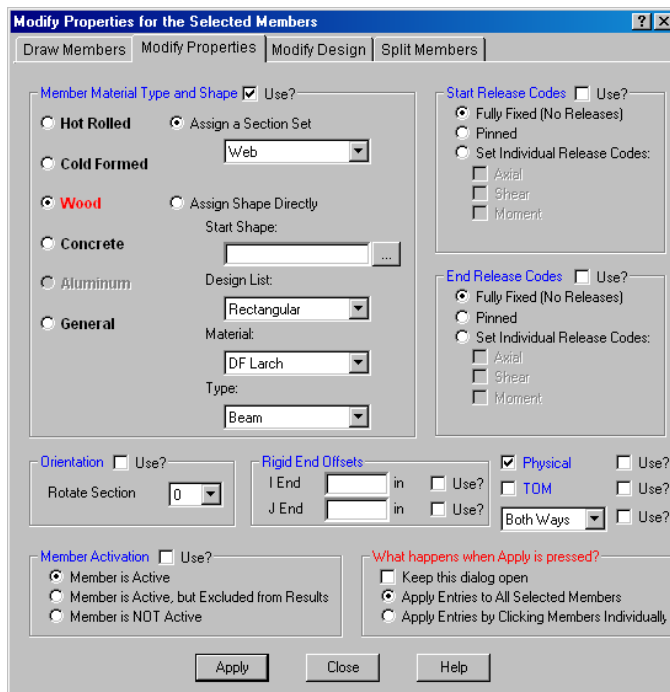
We made our selection prior to opening this dialog but we could have waited until now to select the members. Regardless, clicking Apply will modify the section set for all of the selected members.

Click **Apply** to apply these changes and close this dialog. Click **Select All** to select the entire model. Then click to see an isometric view of the entire structure.

Your model should now look like this:
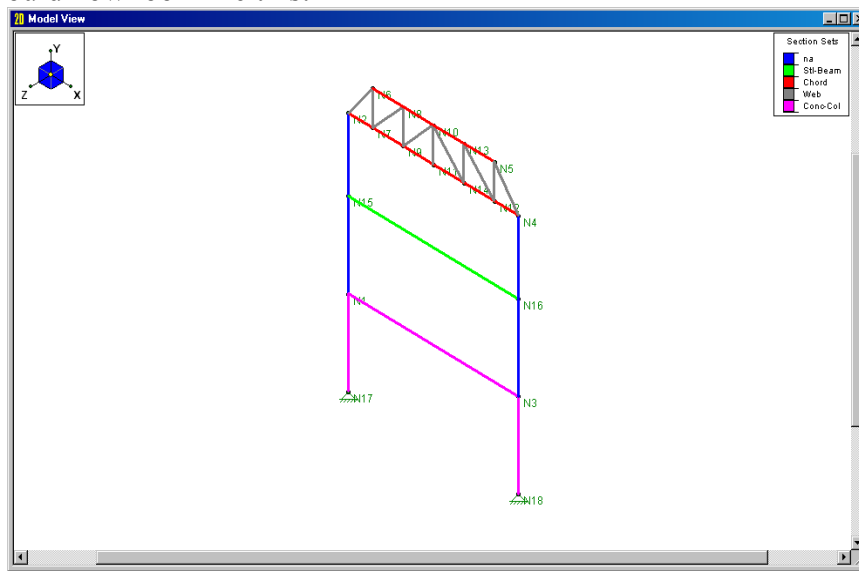
Notice that our concrete cross beam is defined with the same section set as our columns. We'll have to change this to the "Conc-Bm" section that we created in tutorial 1.

Double-click the concrete beam (the lower beam) and select the **Properties** tab to obtain the following dialog:
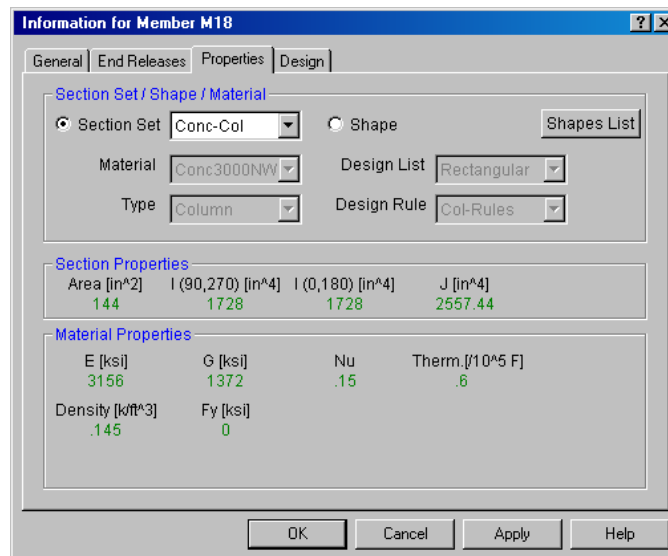
Use the ▼ arrow next to the text that says, "Conc-Col" to open the drop down menu of section sets, and change this setting to "Conc-Bm". Click **OK** to close this dialog.

Model Rendering

A powerful graphics option offered by RISA-2D is the ability to display a rendered image of the model. A rendered image shows each member plotted using a true scale representation of the shape assigned to the member. A rendered view offers some big benefits; besides producing a very nice image, we can also very easily review member orientations and member connectivity.

Turn on the model rendering by clicking the ⚡ button.

Copying Model Elements

RISA-2D allows you to copy selected portions of the model and we will use this feature to turn our model into a two bay structure.

Click the **Copy** [A→A] 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 four fields each for increments in the X and Y directions. You can use the "@" symbol to specify multiple equal increments.

In the section marked **"Axis Increments"** click in the top right field, which is labeled "X inc", and type **"21".**
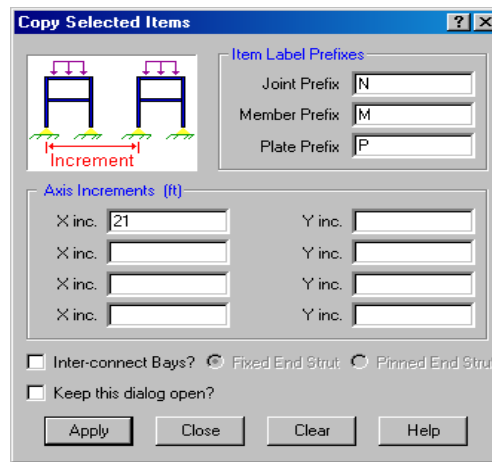
Click **Apply**.
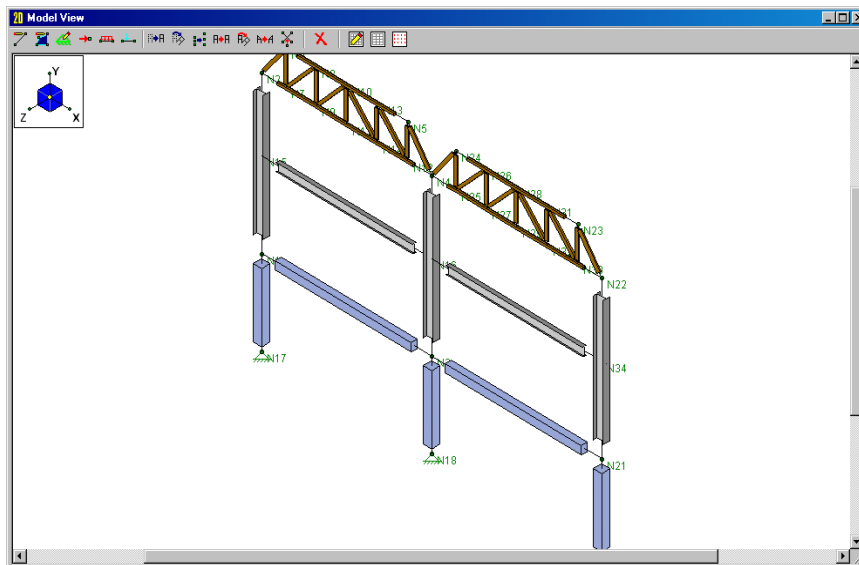
Our entire frame was "selected" so all of the frames were be copied. In addition to copying the joints and members, if we had any loads on the frames they would have been copied as well. We will now add the connecting beams.

Connecting Along Members

We will add steel girts to the columns between the beams. There are no joints at these locations. We could place a drawing grid there as we did in Tutorial 1, or we can insert joints on the columns. We'll learn how to insert joints.

Click [icon] to re-open the **Draw/Modify Members** dialog and select the **Split Members** tab. In the second field, marked **Specific Location**, specify "14". Below that select the radio buttons for **Just Add Joints to Physical Members** and **Apply entries by Clicking Individually.**

[image: Split Members dialog]
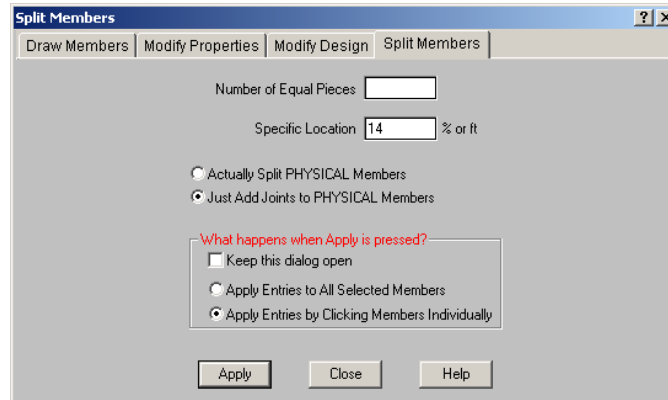
Click **Apply** and then click on the three steel columns. New nodes will appear on the upper half of the columns between the two beams.

Press the key combination "Ctrl-D" to recall the last dialog, then click on the **Draw Members** tab. Choose **Cold Formed** for your material type and click on **Assign a Section Set** if it is not already selected. Then select "Girt" for your Section Set. Finally, change the end releases to **Pinned at Both Ends.**

We also want the cold formed members to be turned so that their strong axis resists wind load. We will specify the rotation before we draw them.

[image: Draw Members dialog]
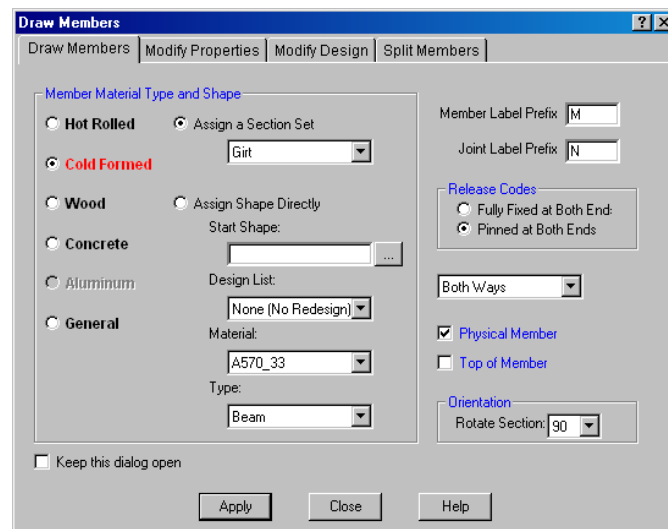
Specify "90" as the **Rotate Angle** on the bottom right as shown in the above figure. Click **Apply** and we are ready to draw.

Click from left to right on the three new nodes circled in the figure below:
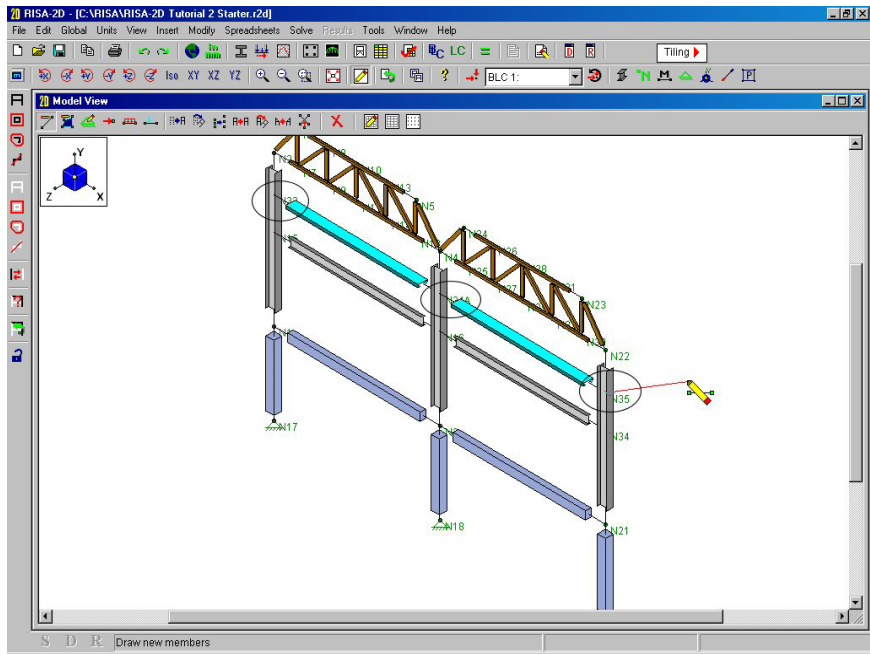
Now RIGHT click to pick up the pencil. Then click on to bring up the Drawing Grid options and select the **Snap To Options** tab.

The settings on this dialog allow you to set up a grid of graphical editing points along any orthogonal plane. They also allow you to automatically snap to the quarter or third points of existing members.

Make sure the **Quarter Points** box is checked. Also make sure that the **Third Points** and **Use Universal Increments** boxes are *un*checked as shown below:
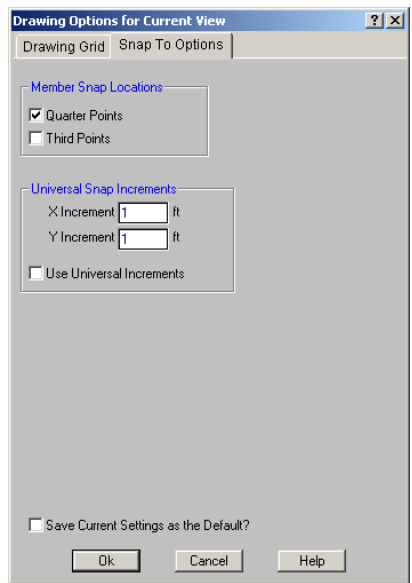
Click **Ok** to return to the model view:

We will draw in our lower girts using the drawing points defined at the quarter points of the columns. Remember that when you are in a drawing mode, graphical editing points are highlighted by a red dot ✳ whenever the tip of your mouse cursor passes over them. Back in the model view our cursor is still 🖊 which indicates that we are still in drawing mode.

Move the pencil tip from the bottom of the front left column up the length of the column. When the tip of your cursor reaches the quarter point of the member a red dot will appear indicating the presence of a graphical editing point. When this point appears, the status bar will display the coordinate location of the drawing point. This should be listed as 0, 5.6875 as highlighted in the figure below:
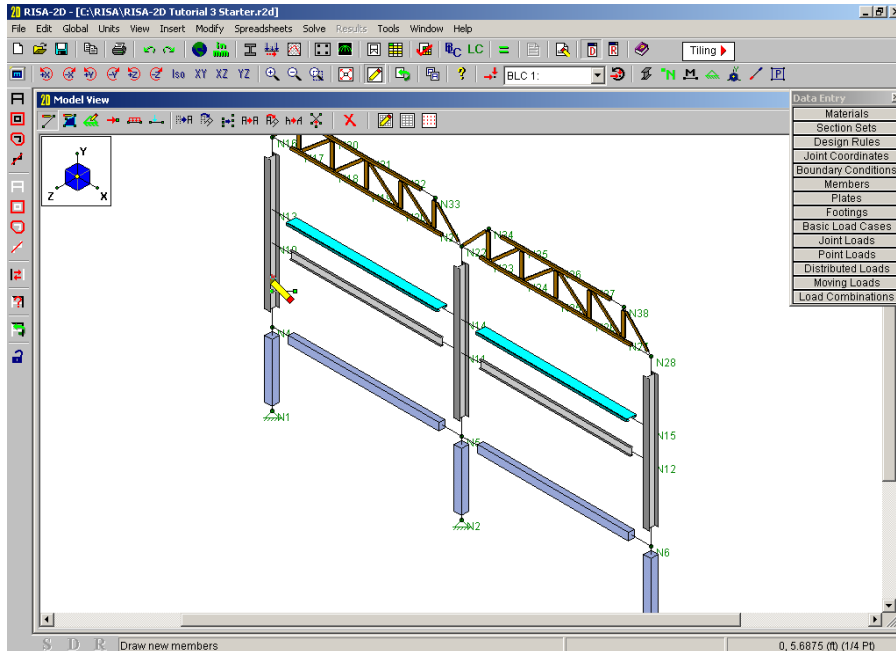
Click on this drawing point (0, 5.6875). Then do the same for the middle column (15, 5.6875) and the right column (30, 5.6875). When you are finished your screen should look like the following:
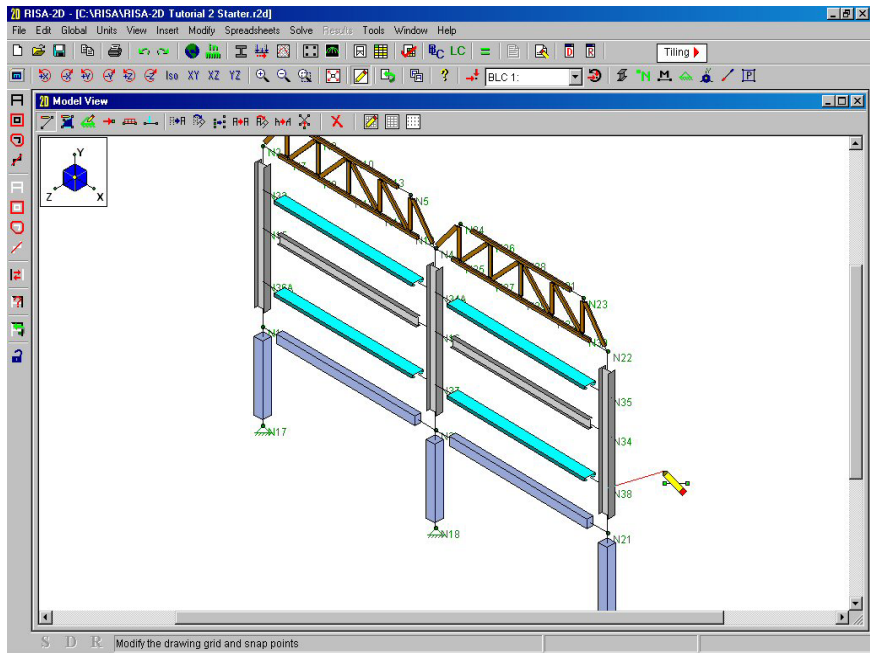
We are finished drawing, so right click to pick up the pencil and RIGHT click again to stop drawing altogether.

Model Merge

Click on the **Model Merge** button on the **Drawing Toolbar** in the model view to see this dialog:
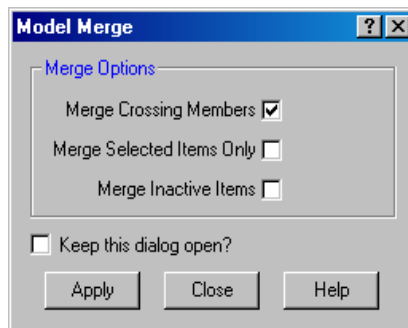
In a nutshell, model merge looks for duplicate nodes and members (with the same coordinates) and merges them into one. If the "**Merge crossing members?**" option is checked in the model merge panel, RISA-2D will look for members crossing each other, and connect them. A typical case might be X-bracing.

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

Click on **Apply** and you will be presented with the results of the merge. Click **Ok** to close the results.

Selecting By Properties

One way to select these members quickly would be to snap to an elevation view and use the **Line Select** tool to draw a line through all of the columns at one time. Another way, which we will use here, is to select all members with the column properties.

Click the **Unselect All** button and then click the **Criteria Selection** button to open the following dialog.
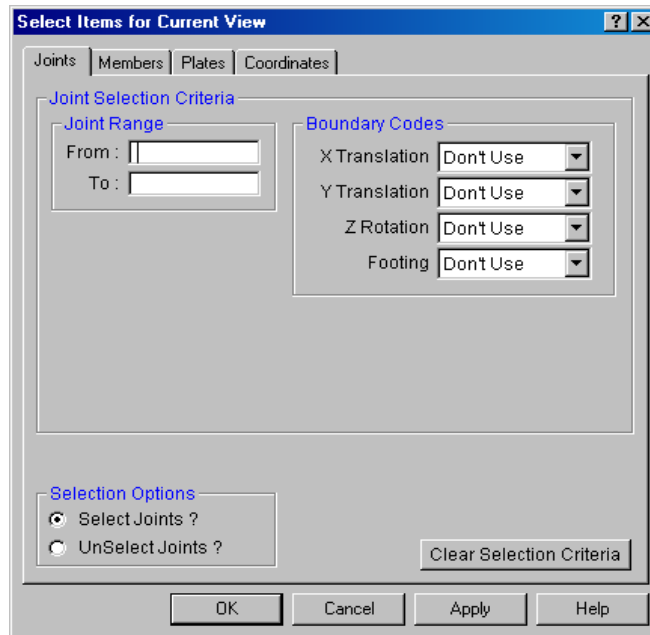
Here you can select joints, members and plates by the properties that define them. The tabs at the top divide the options into four groups.

Click the **Members** tab, and select Y for **Parallel to Axis** entry. Choose your steel material from the drop down list for Material Type as shown here:
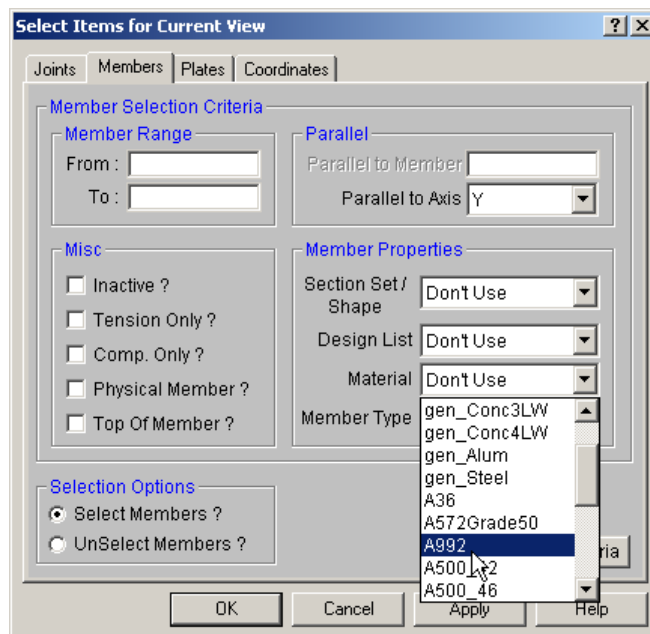
Click **OK**. All of the steel columns should now be selected.

Modify Members

Now click on the **Draw/Modify Members** 🖉 button on the **Drawing Toolbar** and select the **Modify Properties** tab. Uncheck any "**Use?**" boxes that are currently checked.

You should now be presented with various modification options.

Click in the **"Rotate"** field half way down under the blue header that reads Orientation Options. Select the following rotation angle:

> **90**

Now click the box just to the right of that field that is labeled "**Use?**" so that the box is checked. Finally in the lower right choose **Apply Entries to All Selected Items.**

The dialog is now set and should look like this:
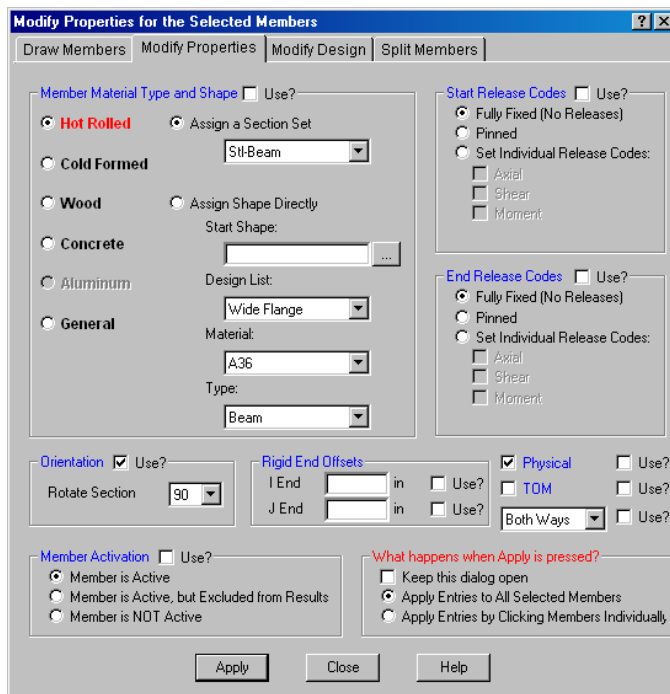
We made our selection prior to opening this dialog but we could have waited unit now to select the members. Regardless, clicking Apply will rotate the selected members 90 degrees.

Click **Apply**.
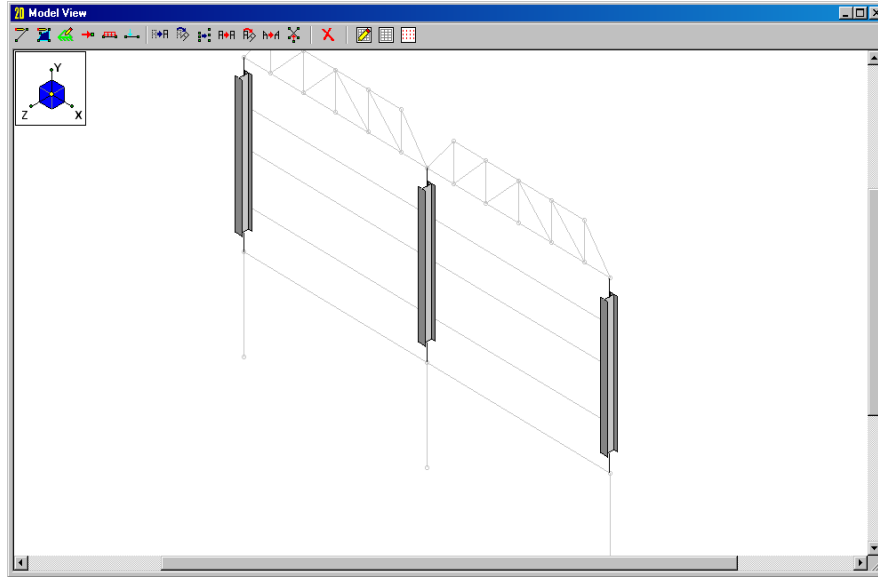
Click **Select All** [icon] to select the entire model.

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. 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.

A major strength of RISA-2D 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 explain 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.

Scaling Elements

Click on the **Modify** menu and select **Scale**. (There is a **Scale** button on the toolbar but we have ignored the menus up until now so it is time to use them.)

Looking at the scaling options the **scale origin** is 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:
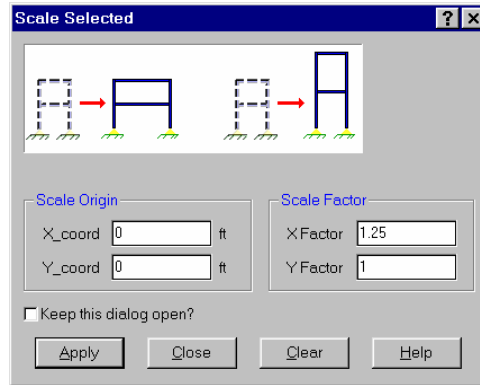
Click **Apply**.

The model view displays the change and we are finished widening the truss. We cannot use **Scale** feature to raise the truss because it would also lift the crossbeam. 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 [XY] and then [⊡] 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 joints 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:
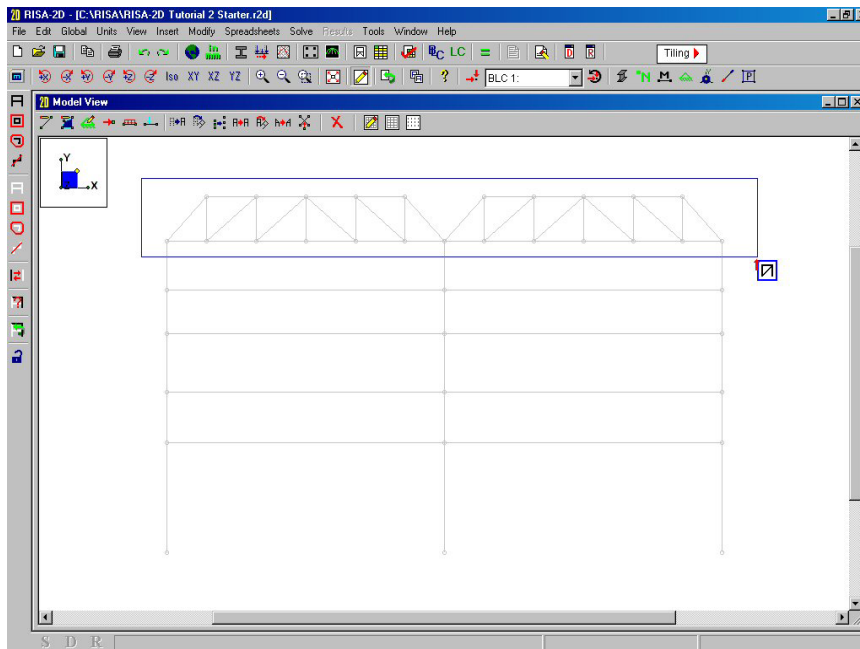
Release the mouse button and the entire truss, and nothing else, should be selected.

Now click the **Move** [icon] 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".
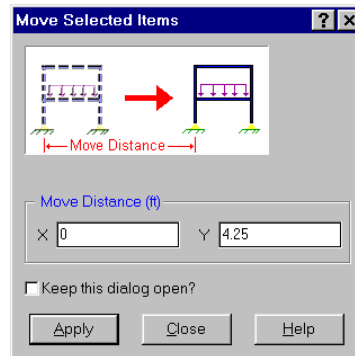
Now click **Apply**.

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 joint 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** [icon]. Also click on the **Isometric** [icon] button and then the **Full View** [icon] button.

Design Parameters

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

Select **Design Parameters** from the **Modify** menu. Type "Segment" (or "Seg") in the first field and check the **Use** checkbox. Also select **Apply Entries to All Selected Members** as shown here:
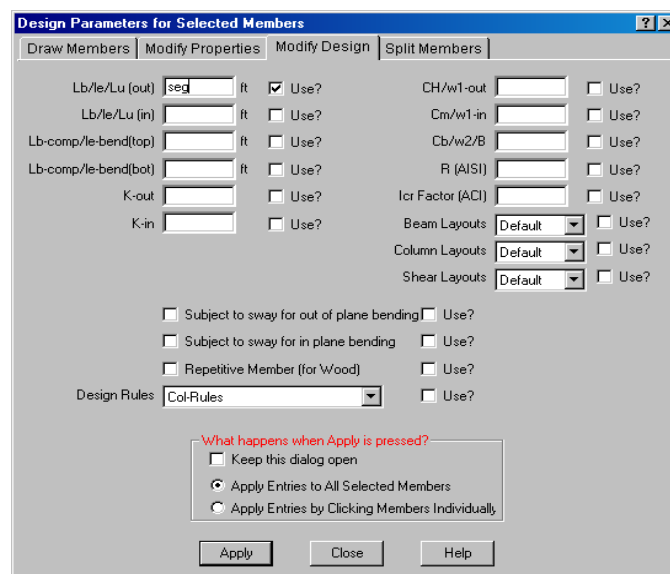
We will now explain what you have in front of you, and what we have done. RISA-2D calculates most of these parameters for you although you may specify your own values as well. It is important to understand how each parameter is handled so you can take advantage of the capabilities of RISA-2D.

Most fields have three labels separated by a slash (/), the first label is for steel, the second for timber, and the third for concrete. Each field is described below. Refer to the RISA-2D General Reference or On-line Help file for more information.

The first three fields are for specifying the unbraced length. The following options are available:

- When nothing is specified RISA-2D will use the full length of the member for unbraced length. This is the I-joint to J-joint length shown in the third column.

- You may specify **Segment** (or type **Seg**) and RISA-2D will automatically use the distance between nodes on physical members. This way members and plates framing into physical members are treated as adequate bracing.

- You may specify a value. This value may be less than or greater than the I-joint to J-joint length.

The **Lb/le** fields are unbraced lengths for in plane and out of plane buckling of the member. **Lb** is the steel symbol and **le** is the wood symbol. These are used in KL/r ratio calculations for steel & concrete, and le/d calculations for wood. We have specified that all members be braced in the out of plane direction at every node along their length.

The **Lcomp** values (used for steel) are the laterally unbraced length of the compression flanges used in the calculation of allowable bending stress for ASD design, or member strength for LRFD design. The **le-bend** values (used for wood) are used in the calculation of R_B.

The **K** factors are the effective length factors, which RISA-2D is capable of approximating for you. A value of 1.0 will be used when nothing is specified. Refer to the RISA-2D General Reference or On-line Help file for more information.

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-2D. **CH** is the NDS shear stress factor.

If you leave the **Cb** factor blank RISA-2D will calculate it for you if you're doing ASD design. For LRFD design, the **Cb** value is set to "1.0" if not specified.

The final two fields are the sway check boxes to indicate whether the member is to be considered subject to sidesway for in plane or out of plane bending. Sway comes into play in the calculation of **Cm** and **Cb** and in the approximation of **K** factors.

Currently, all the entries are blank except for the first. For the fields we left blank RISA-2D will use the default values when the code check calculations are performed. For weak axis bending each node is treated as a brace point as a result of the use of the "**Segment**" code.

Click **Apply**.

To actually see the settings for the unbraced lengths we could double-click any member or click **Plot Options** to show the unbraced lengths in the view.

Sorting

Next up we will see how we can sort information in our model. 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** Tiling ▶ button on the **RISA Toolbar** and the following list of window tiling options is presented.
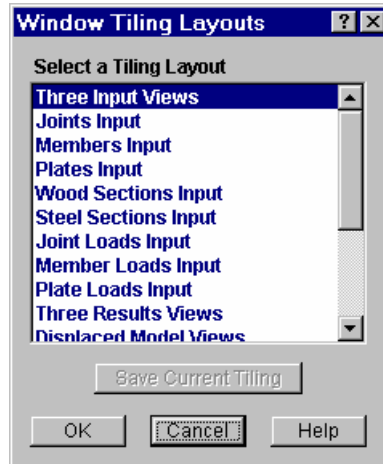
Later on we will experiment with these options. For now, select the second option which is "**Joints Input**" and click **OK**.

Your screen should look like this:
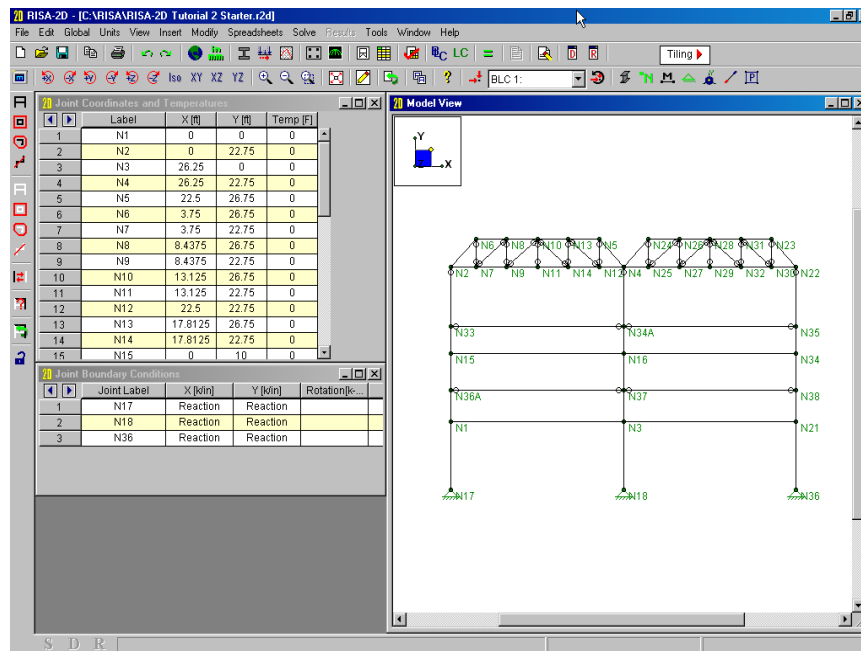
This tiling option has opened the **Joint Coordinates** spreadsheet in the upper left corner and placed the **Joint Boundary Conditions** spreadsheet below it. The joints are listed in the order in which they were created as we defined our model. It would be more useful to list them based on their coordinate locations.

Click anywhere in the X-coordinate column.

Click (A-Z icon) on the **Window Toolbar,** select "**Sort Low to High**" and click **OK**.

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

Click in the Y coordinate column. Click (A-Z icon) again and 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 joints and the model plot looks exactly the same.

Now select **Relabel Joints** from the **Tools** menu**.** Click **OK**.

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

Now let's sort our members as well.

Click ![Tiling] 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-Joint and then the section set.

In the **Members** spreadsheet click anywhere in the **I-Joint** column and click ![icon] With "**Low to High**" as the sort option click **OK**.

Click anywhere in the **Section** column and click ![icon] 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 arranged in the spreadsheet.

Now select **Relabel Members** from the **Tools** menu**.** Click **OK**.

This concludes Tutorial 2. The next tutorial teaches you how to load the model.

If you wish to stop and do Tutorial 3 later, select **Exit** from the **File** menu. Otherwise continue on and open the file at the beginning of Tutorial 3.

Tutorial 3 – Loading

In this tutorial we will load the model we defined in the last tutorial. Make sure you start off correctly by opening the file mentioned next.

After starting RISA-2D click the **Open File** ![icon] button. In the **File Open** dialog open the "Tutorials" folder and select the file named "RISA-2D Tutorial 3 Starter." Click **Open** and close the **Global Parameters** window by clicking **OK** and we will begin working in the model view.

Click ![icon] to close the **Data Entry Toolbar** if it is open. Your screen should look like this:
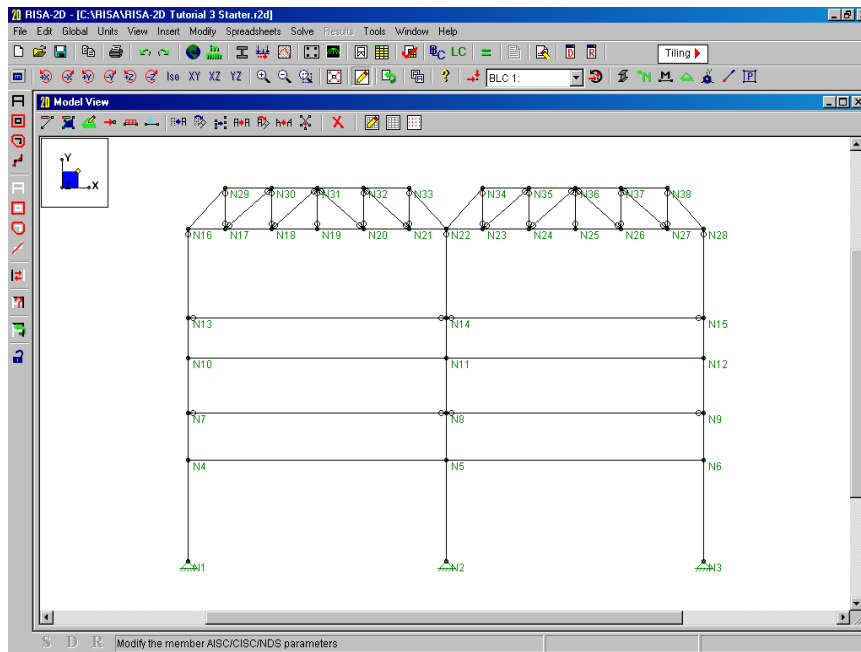
Load Cases, Categories and Combinations

Before we load the model it is important to understand just how RISA-2D lets you manage loads. When loads are defined in RISA-2D they must be assigned to a "Basic Load Case" which are simply sets of loads. 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.

Open the **Basic Load Cases** spreadsheet by selecting it on the **Spreadsheet** menu.

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]**
>
> **Dead Load, [ENTER]**

So we've given descriptions to three BLC's. These descriptions are strictly for your benefit. RISA-2D doesn't require a description.

The rest of the spreadsheet is blank because we have no loads applied. We will apply the loads and then come back to this spreadsheet to discuss it further.

Click ☒ to close the spreadsheet.

Load Direction Options

RISA-2D allows you to specify the direction of your loads in different ways. The three basic choices are:

1.) Apply loads aligned with global axes, designated with upper case labels such as **X** and **Y**.

2.) Apply loads aligned with member or plate local axes, designated with lower case labels **x** and **y**.

3.) Apply projected loads along global axes, designated with upper case labels such as **V** and **H**.

You will be presented with a list of options each time you define the loads so don't worry about remembering what they are. A click of the **Help** button will explain the options each time you need it.

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.

On the **Insert** menu select **Joint Load**.

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, an enforced displacement, or a mass 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.

Notice the last option specifies **"Apply Entry by Clicking/Boxing Joints"**:
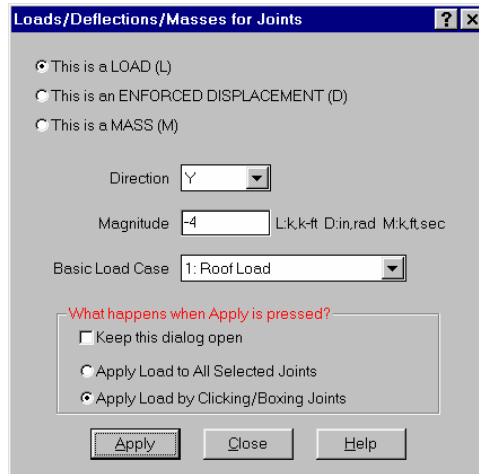
We will use this to apply loads to the interior joints on the top of the trusses and apply loads of different magnitudes to the outer joints.

Click **Apply,** the mouse cursor changes to .

Now click and drag to create a box that contains the interior top chord joints like this:
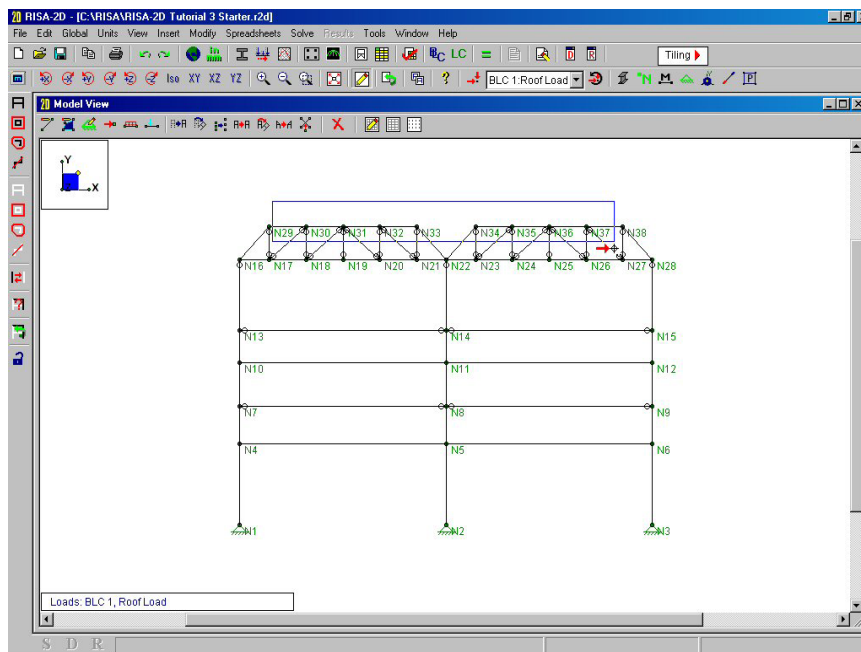
The loads are applied to the interior joints. Don't forget about the **Undo** button should you make any mistakes.

If you cannot see the entire load, hit the button to zoom out.

Now press [Ctrl-D] to go back to the **Joint Loads** dialog and change the magnitude to "-2" and click **Apply** again.

Return to the model and click on the two exterior panel points on the top chord which are Nodes N29 and N38.

When you are finished your model should look like this:
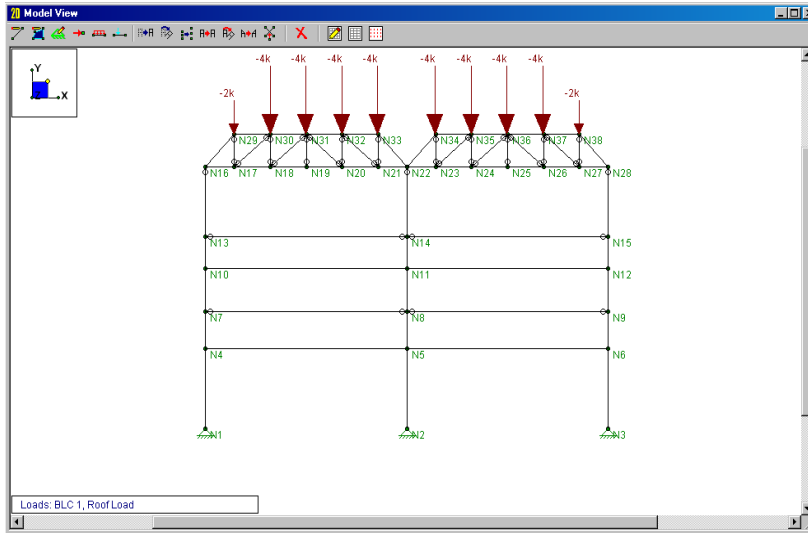
Distributed Loads

The wind load will be applied as a 250plf distributed load.

Click the **Distributed Loads** ![button] button. Specify the direction to be **"X".** Change the value for the **Start** and **End Magnitudes** so that they are both set to "-0.25". Now click on the down ![arrow] arrow in the **Basic Load Case** field and select **"2: Wind Load".**
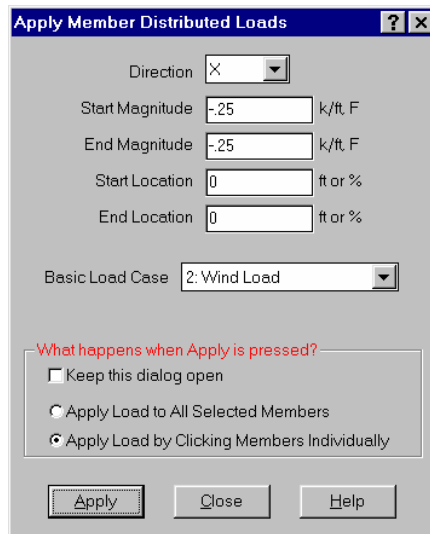
Make sure the **Apply Load By Clicking Members Individually** option is selected. Click **Apply**. Now click on the steel and concrete columns on the right side.
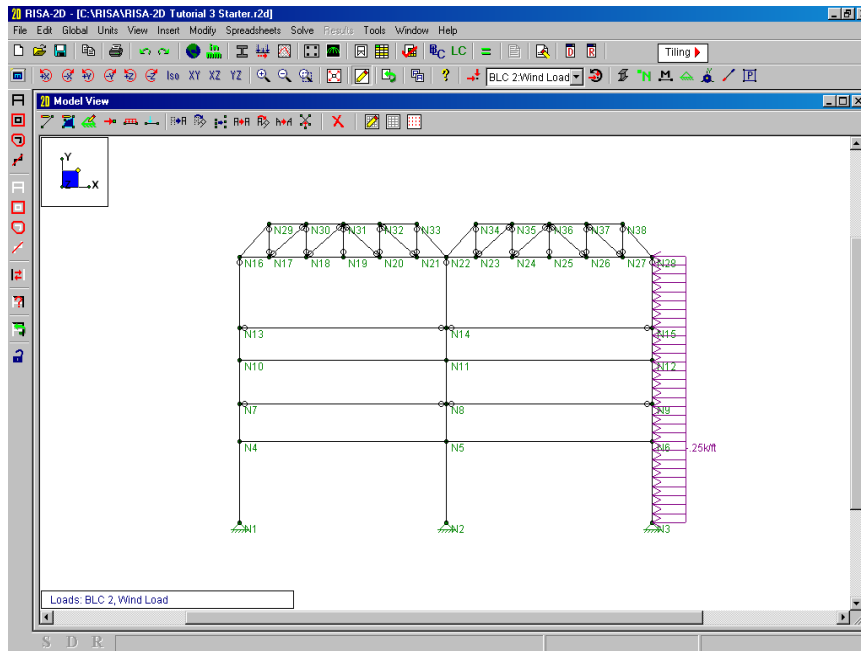
Now lets look at the loads in the spreadsheets.

Basic Load Case Spreadsheet

Open the **Basic Load Cases** spreadsheet by selecting it on the **Spreadsheet** menu.

This is where we named our basic cases earlier. Notice that there are now some numbers on the right side of the spreadsheet. These numbers summarize the loads that we just defined. The "10" on the first line represents the "10" joint loads that we placed in the Roof Load case. The "2" represents the distributed wind loads.

RISA-2D 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 in the 1st row on the cell that contains the number "10".

The **Joint Loads** spreadsheet opens with the loads that were assigned to BLC 1. Note that the drop-down list on the right side of the **Window Toolbar** now contains "BLC 1:Roof Load". 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 dead load listed on the **Basic Load Case** spreadsheet. Also we haven't discussed the **Category** and the **Gravity** columns.

Press [Ctrl-F4] to close the joint loads and return to the **Basic Load Cases**.

Copying Loads

The spreadsheet now shows the roof live load as 10 joint loads. We will now add dead load at the roof level. The dead load at the roof occurs at the same locations and only differs from the live load by a factor. In this case it would be easiest to copy the roof live load into the dead load case and modify it. We will use the copy loads and block math features to do this.

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

The dialog should look like this:
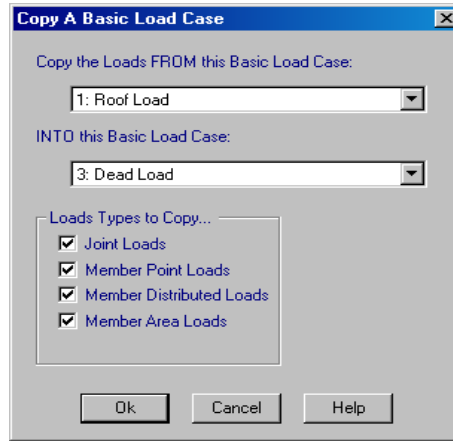
Click **OK** and there is now a "10" in the **Joint** column of the Dead Load case. Click on the "10" to open the **Joint Loads** spreadsheet for the dead load (BLC 3).

We now need to change the magnitude of the load so that it represents the dead load. We will use the **Block Math** feature to do this with a factor. First we will select all of the cells in the **Magnitude** column.

Click on the column button 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 25% of the copied live load.

Enter "0.25" in the field and select the **Multiply** option.
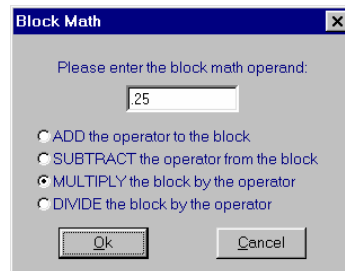
Click **OK** and the joint loads now have magnitudes of -1K and -0.5K or one quarter of the roof live load. Press [Ctrl-F4] to close the **Joint Loads** spreadsheet.

Self-weight

We will include the self-weight of the structure and be finished defining loads. We will add it to the same load case that we just 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 third row and type:

-1

This is how you tell RISA-2D 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 or Y direction.

Load Categories
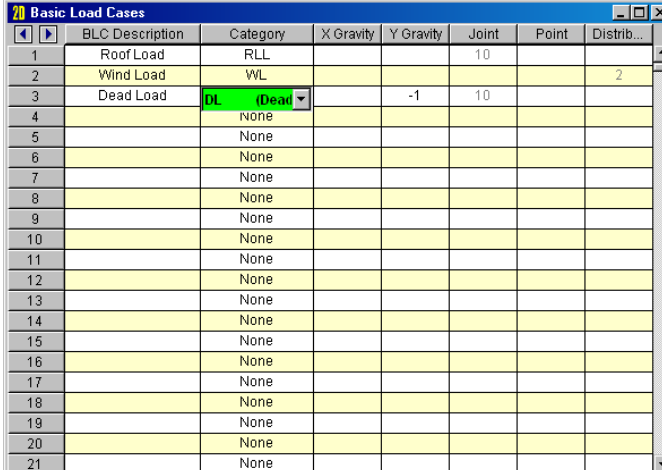
The last to do on the **Basic Load Case** spreadsheet is assign each load case to a category. Categories are especially helpful in bringing all of your loads together. In this model the loads are quite manageable and we could just refer to them by their BLC number. Many structures, however, will be modeled 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 **"DL (Dead Load)"** category.

The spreadsheet is complete and looks like this:

	BLC Description	Category	X Gravity	Y Gravity	Joint	Point	Distrib...
1	Roof Load	RLL			10		
2	Wind Load	WL					2
3	Dead Load	DL (Dead ▼)		-1	10		
4		None					
5		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					

Close it by clicking the ☒ in the title bar.

Load Combinations

The last step in applying loads is to define how the loads are to be combined. It is these combinations that are analyzed when we solve the model and view results. Most standard code combinations are already built into RISA-2D. We will apply the ASCE minimum design loads and then modify them.

From the **Spreadsheet** menu choose **Load Combinations.** On right side of the **Window Toolbar** click the **LC Generator** button to bring up the Load Combination Generator dialog.
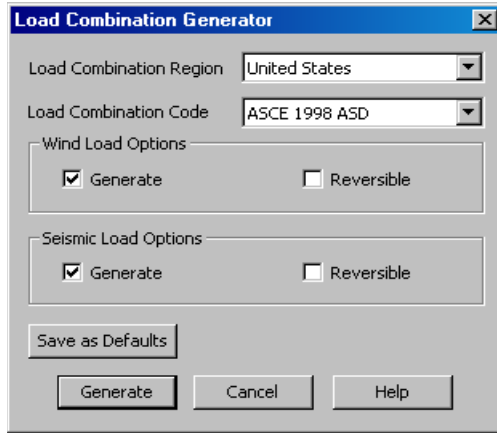
Choose "United States" as the **Load Combination Region** and "ASCE 1998 ASD" as the **Load Combination Code.**

Choose "Generate" for the **Wind Load Options** and "Generate" for the **Seismic Load Options**. For more information on the **Wind Load Options** and **Seismic Load Options** refer to the Generating Building Code Combinations section of the General Reference Manual.

Now click the Generate button at the bottom of the **Load Combination Generator** dialogue box.

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 three 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 General Reference or the help file for more information about these options.

The **Solve** checkbox indicates whether this combination is to be included in the **Batch** or **Envelope** solution. RISA-2D allows you to solve a single load combination, a batch of combinations or an envelope of multiple combinations. The **Batch** solution allows you to solve multiple combinations and review all the results together. The **Envelope** solution solves multiple combinations keeping only the maximum and minimum values in the results. All of our combinations are currently checked to be included.

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 General Reference or 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 through 10 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 10 before releasing.

These lines should now be shaded yellow. If they are magenta you selected cells in the spreadsheet, not the button on the very left. If you have selected the rows correctly the line delete button ![icon] on the **Window Toolbar** should be active.

Once you have the lines selected, click the **Delete Lines** ![icon] button on the **Window Toolbar**.

The lines are deleted leaving you with 8 combinations. Next we will delete combinations 3 through 4. You may also delete lines in a spreadsheet one at a time.

Click on the 3rd row in the Description column. Hit the F4 key on your keyboard twice to delete load combinations 3 and 4.

Click on the 4th row in the Description column. Hit the F4 key on your keyboard once to delete load combination 4

These load combinations are sufficient for our steel and wood members. However, we will have to add more load combinations for the design of our concrete members.

On right side of the **Window Toolbar** click the ![LC Generator] button.

Choose "United States" as the **Load Combination Region** and "ACI Strength 99" as the **Load Combination Code.**

Choose "Generate" for the **Wind Load Options** and uncheck "Generate" for the **Seismic Load Options**. Now click the ![Generate] at the bottom of the **Load Combination Generator** dialogue box.

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

Look over your load combinations to make sure they are the same as above and then click the **Design** tab to view more load combination data.

The field labeled **ASIF** is used to enter the allowable stress increase factor for the design of hot rolled steel per AISC 9th Edition specifications. Since we are using the ASCE load combinations, this increase factor (usually

1.333 for wind or seismic loads)will not be used. An entry in this field greater than 1.0 is also used to indicate whether the seismic provisions of the LRFD code are to be used.

CD is the load duration factor used only in timber design. This factor is dependent on the load with the shortest duration that is present in the load combination.

The **Service** checkbox indicates whether the load combination is for service level loads. This will affect the stiffness used for concrete members during the solution.

The remaining fields define which material types will be designed/checked for that load combination. This way we will never get an NDS wood code check for the strength level load combinations used for concrete design.

	Description	Sol...	PD...	SR...	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Fac
1	ASCE 1	☑			DL	1										
2	ASCE 2 (a)	☑			DL	1	LL	1	RLL	1						
3	ASCE 3 (a)	☑			DL	1	LL	.75	RLL	.75	WL	.75				
4	ASCE 4	☑			DL	.6	WL	1								
5	ASCE 5	☑			DL	.6	EL	.7								
6	ACI 9-1	☑			DL	1.4	LL	1.7								
7	ACI 9-2	☑			DL	1.05	LL	1.275	WL	1.275						
8	ACI 9-3	☑			DL	.9	WL	1.3								

Review the information in this screen to make sure the design data is the same as above, then close the spreadsheet by clicking the ☒ button.

Displaying Loads

RISA-2D provides easy ways to view your loads. They may be displayed as Basic Cases, Categories or Combinations. First we will go through the basic cases.

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". You may go through all of the Basic Load Cases in this manner if you like.

Now we will look at the load combinations.

Click the **Switch Loads** ↺ button one time and you will see the loads by combination.

The drop-down list box on the **Window Toolbar** now lists 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 3:ASCE 3 (a)" from the drop down list.
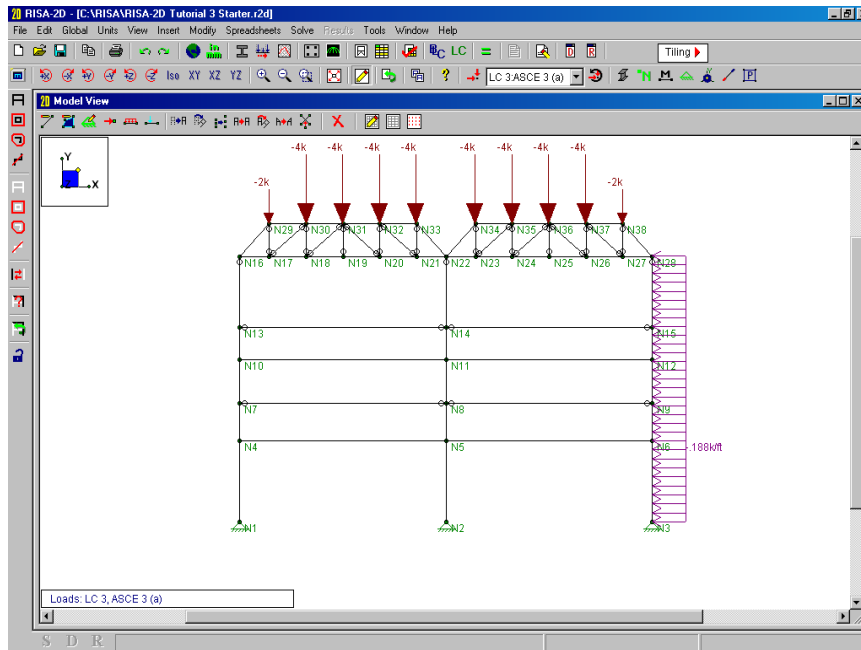
This is a great way to verify that our model is solving the problem that we want it to solve. One thing to note here is that the magnitudes of the loads represent the factored combination. For example the displayed roof load magnitude is the combination of the dead and live loads multiplied by their factors.

When you are finished viewing the load combinations click the **Switch Loads** button once more to see loads by category.

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

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

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

If you wish to stop and do Tutorial 4 later, select **Exit** from the **File** menu. Otherwise continue on and open the file at the beginning of Tutorial 4.

Tutorial 4 – Solving & Results

In this tutorial we will solve the model we defined in the last tutorial. Make sure you start off correctly by opening the file mentioned next.

After starting RISA-2D click the **Open File** ![Open File icon] button. In the **File Open** dialog open the "Tutorials" folder and select the file named "RISA-2D Tutorial 4 Starter." Click **Open** and close the **Global Parameters** window by clicking **OK**. Click ![icon] to close the **Data Entry Toolbar** if it is open. Your screen should look like this:
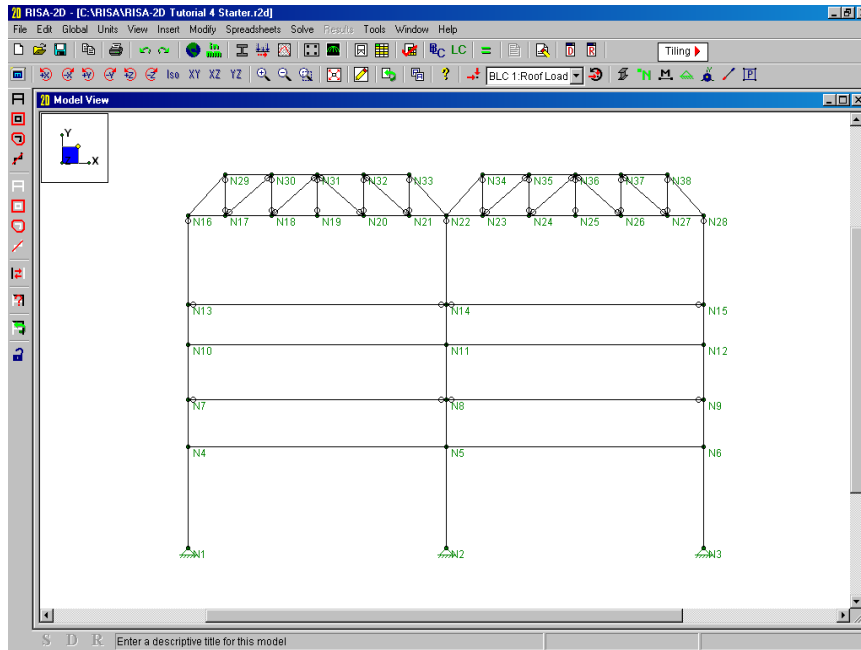
Solving the Model

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** ![Solve icon] button and the **Solution Choices** dialog is presented. The first option is to solve a single combination. Click on the down ![down arrow] arrow and select "4: ASCE 4".
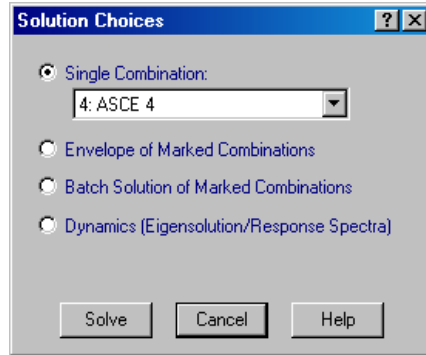
Click **Solve**.

RISA-2D 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. The **Results** toolbar is also opened for access to other results. You may view results in the spreadsheets, plotted on the model in the view, or in detail for each member. We will cover these three options next.

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.

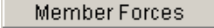To review the results, you can page up and down or scroll up and down a line at a time. [Home] and [End] take you to the top and bottom of the results. You can use the **Find** feature to locate specific results and the **Sort** feature to sort the results. The **Exclude** feature allows you to hide results that are not of interest. For now just note what results are available, in the next section we'll discuss how to quickly manipulate them.
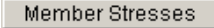
Select [Joint Deflections] from the **Results Toolbar** to review these results.

Now select [Member Forces] to view the forces which are listed for 7 locations on each member. This is because we specified 7 sections back on the **Global** settings.

Remember you can sort the spreadsheets. We will use the **Sort** feature to sort the strong axis moment results.

Click in any cell in the last column, which is the **Moment** column. Now click the **Sort** [A↕Z] button on the **Window Toolbar** to access the sorting options. Select "**Abs Max to AbsMin**" and click **OK**.

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

Select [Member Stresses] to view the member stresses.

Select [Member Deflections] to view the member 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 [Suggested Shapes] to view the suggested redesign shapes (we'll use these later).

Select [Design Results] and choose the **Hot Rolled** tab**.**

Here we have the AISC code checks, i.e. ratio of actual to allowable stresses, per ASD 9th Edition criteria. Failing members would be 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 the **Cold Formed** tab to view the cold formed steel code checks.

The design strengths Pn, Mny, & Mnz are listed along with the code check values and controlling equation. Factored design stresses and controlling equation are also listed.

Select the **Wood** tab to view the NDS code checks for our wood truss members.

The factored design stresses and controlling equation are also listed. Notice that, because our chord members are relatively slender, the program reports that they exceed the maximum Le/d value that the NDS gives for compression members.

Select the **Concrete Beams** tab.

Notice that there are not any concrete results for this load combination. This is because when we defined this load combination, the checkbox for concrete design was not checked. Later, when we solve batch and envelope solutions we'll see how this prevents us from getting unnecessary steel or wood code checks for load combinations that are specifically intended for concrete design.

Select Material TakeOff to view the material take-off.

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 joint labels by clicking 'N twice. Finally, close the **Results toolbar** by clicking the close ✕ button.

Plotting Deflected Shapes

Click 🔲 to open the **Plot Options** and select the **Deflection Diagrams** tab. Now select the option labeled "**Load Combination**" which will plot the combination we have solved. Select **Include Undeflected Shadow** and click **OK.**
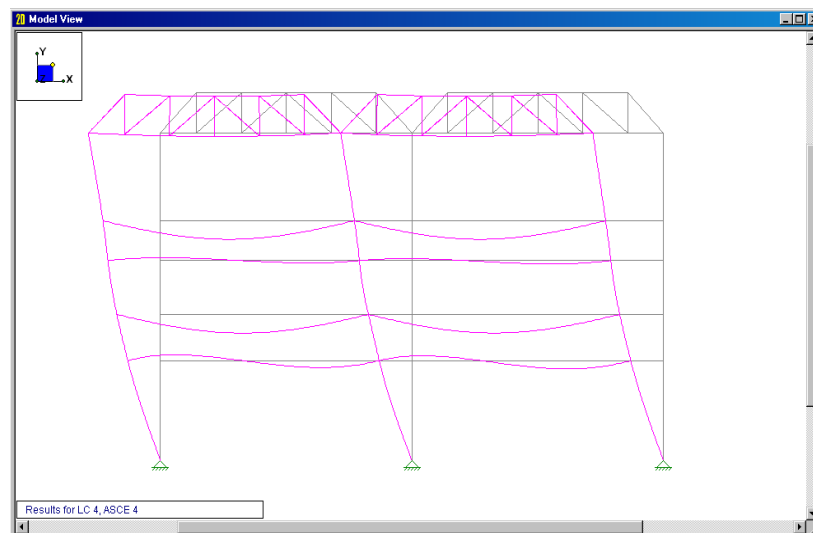
This is a fairly nice representation of how the model deflects under these loads. Now let's use some of RISA-2D's more advanced graphics features.

Click on the [Iso] button to get an isometric view. Then click [⊡] to redraw a full model view. Now, click on the **Render** [↯] button on the **Window** toolbar.

As you can see, a rendered isometric view can really show you what's happening with the model.

Animation

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

Click [m] 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:4 ASCE 4" and the animation is brought back to the front. You may drag the window up to get a better view.

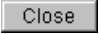Once the animation begins, you can speed it up or slow it down. This provides a very vivid presentation of how the model is deflecting.

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-Coding Member Results

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 [m] 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**.

Let's return to an elevation view to better see these color coded results.

Click on the [XY] button to get an view of the XY plane. Then click [⊡] to redraw a full model view.

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 [m] 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 **Moment** and then click the **Magnitudes** box so that it has a checkmark in it.

Click the **Joints** tab and in the **Reactions** section select the **Y Direction** and "**Include the Magnitudes"** options.

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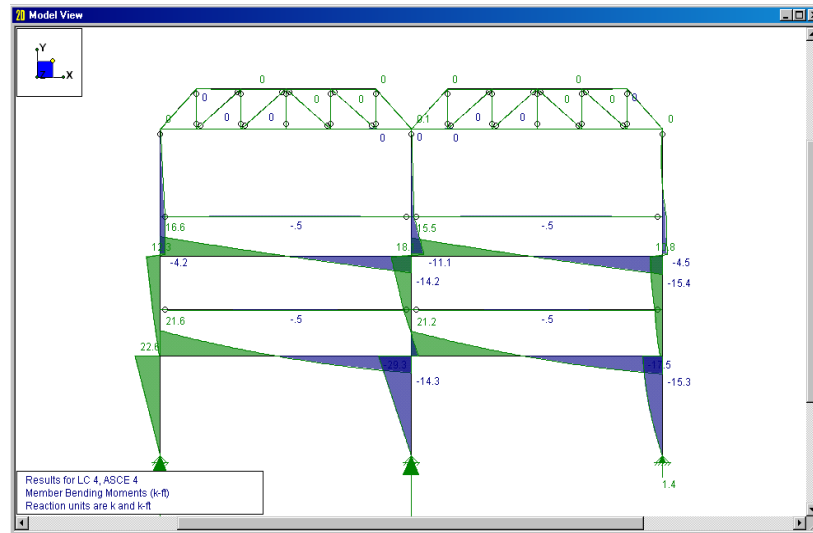
Member Detail Report

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

From the **Results** menu, select **Members ▸ Member Detail**. Type in M17 and click **OK** and you will get this report on member M17 which is our middle steel column.
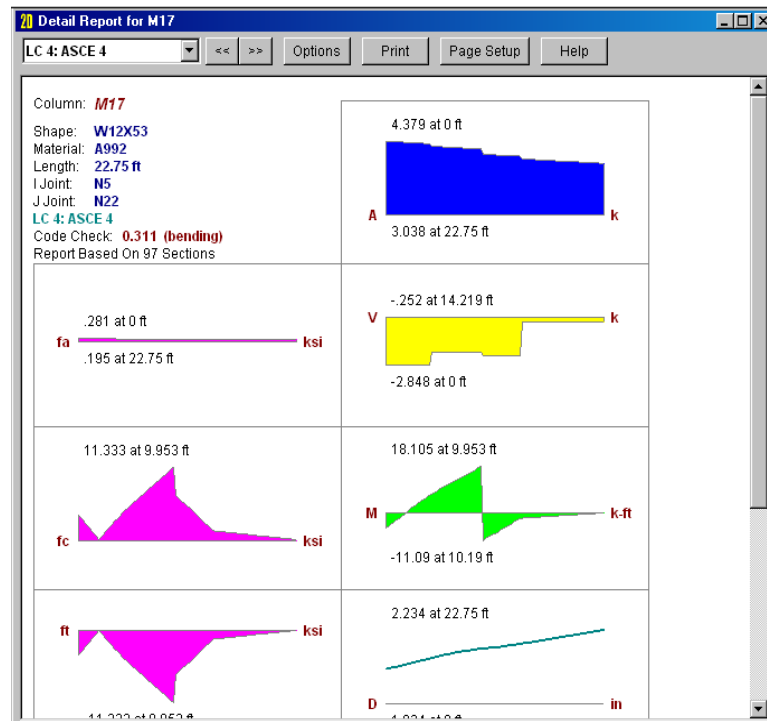
This report gives you the ability to see a detailed report showing all the forces, stresses and deflections for any individual member. You can expand any of these diagrams for closer inspection by clicking on them.

Click on the deflection diagram, the bottom diagram on the right. Then hit the ⟨⟨ button twice to get the deflection diagram for Member M15.
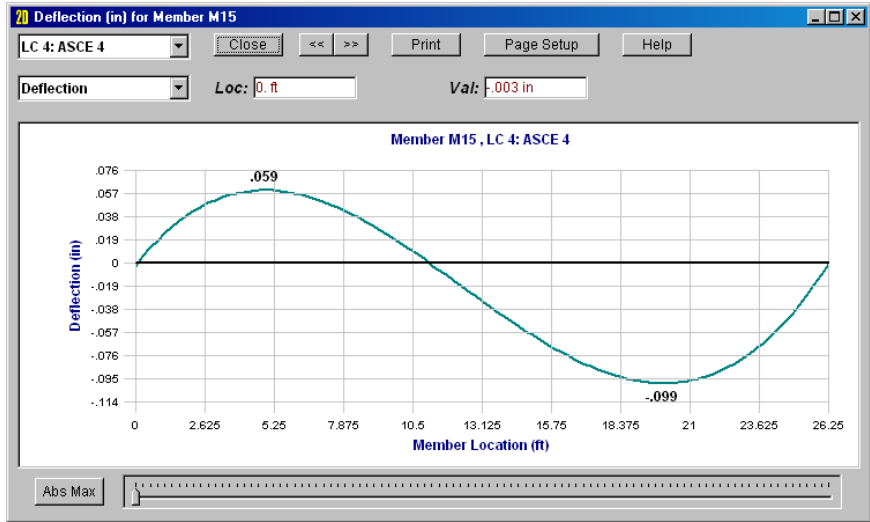
You should now be looking at the deflection diagram for member M15. You may click the **Abs Max** button to retrieve the absolute maximum value and it's location. The slider control to the right of the **Abs Max** button allows you to retrieve values at approximately 100 section locations.

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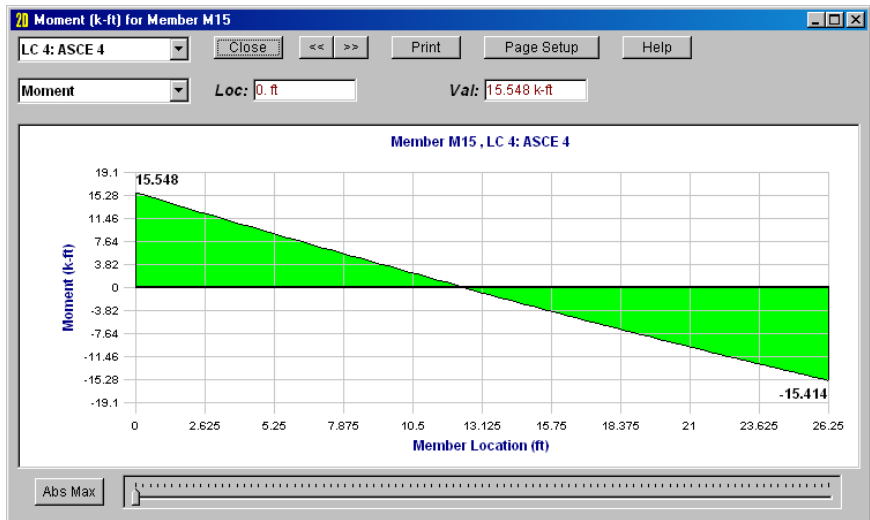
Feel free to try any of the options or view another diagram from the list. When you are finished click **Close** and you will be returned to the member detail report.

Scroll down to view code check information and you will see the code check value along with the controlling equation and all of the design values. Finally, close the detail report by clicking the button.

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** button.

The mouse cursor will change to [icon] 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 ▸ Design Results**. Click on the Hot Rolled, Cold Formed or Wood tabs 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 look at plate results.

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

Printing Graphic Results

Click on the **Plot Options** [icon] button and on the **Members** tab select the **Rendered** option and click **OK**. Now click the **Print** [icon] 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.

Now that we have covered all of the result spreadsheets and graphics we will take a minute to look at the window tiling options that RISA-2D provides.

Tiling Results Windows

To start, click on the **Tiling** [Tiling ▸] button, scroll down, and select **Member Analysis Results.** Click **OK** to accept your tiling selection. You may look at as many of the tiling options as you wish. Before you move on to the next section select the **Three Results Views** tiling.

Batch Solution

The **Batch** solution solves multiple combinations, keeping the results for all of them. The results spreadsheets may then be organized either by load combination or by item.

Click on the **Solve** [icon] button again and this time choose **Batch** solution. Click **Solve**. You will be warned that the results will be cleared. Select **Yes**.

RISA-2D 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-2D *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. You may override the warning in the **Preferences.**

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

Click in the lower left view to make it active and click on the **Set Plot Options** button [image] . At the bottom change the **Combination to Use for Results Display** to "4: ASCE 4" and click OK.

The lower left view now shows results for load combination 8, with the wind load in the opposite direction.

On the **Results** menu choose **Reactions** to display them once again.

For the **Batch** solution the spreadsheets display all of the results for each load combination. Currently the results are grouped by combination; all of the reactions for Load Combination 1 are listed first and then for each combination thereafter. The combination number is listed in the first column. You can also group the results so that all the reactions for joint N1 are together, then N2, and so on…

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

Now let's take a closer look at our concrete results.

On the **Results** menu choose **Members ▸ Design Results**. Then select the **Concrete Beams** tab.

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

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

Click on the **Concrete Columns** tab.

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

Click on the Detail Report for Current Member button located on the window toolbar.

This will bring up an envelope detail report for the concrete column that looks similar to the one shown below:
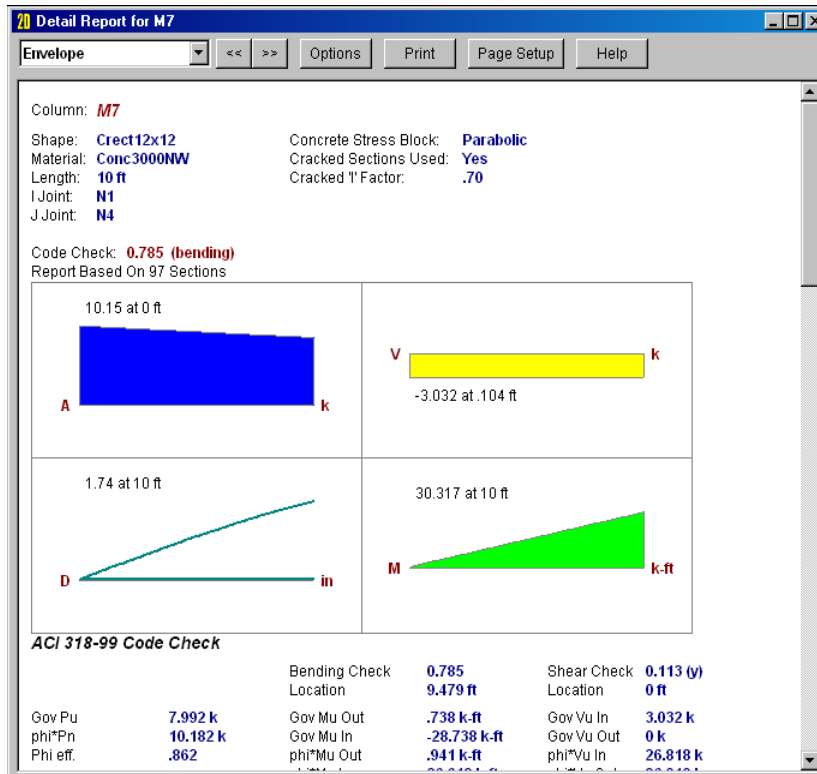
Because this detail report is based on moment and shear envelopes, it is somewhat different that the one we looked at for a steel member.

Scroll down to take a look at the concrete span and reinforcing information.

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

After this brief review of our concrete results, it should be clear that we will need to increase the size of our members.

Envelope Solution

Click on the **Solve** $\boxed{=}$ button. Select **Envelope of Marked Combinations** and click **Solve.** Again, agree to clear the results.

Code Check Results

Click on the **Tiling** $\boxed{\text{Tiling} \blacktriangleright}$ button, select **Member Steel Results** and click **OK.**

The first things you will notice are the red members in the model view indicating that we have members that are failing.

Notice that the spreadsheet results are now reporting the "worst case" results as a result of the envelope solution we ran. The columns labeled "**lc**" report the load combination that generated the controlling values. Other results not shown here such as the member forces display **Maximum** and **Minimum** results in order to capture the worst of both negative and positive results. When you perform an envelope solution the spreadsheet results will contain this additional information.

Optimizing Member Sizes

The fourth spreadsheet from the top reports what RISA-2D is suggesting for alternate shape sizes along with the member that is controlling the design.

	Section Set/M...	Current Shape	Suggested Sh...	Controlling Me...	Use Suggeste...
1	**Stl-Beam**	W12X19	W12X26	M15	☑
2	Girt	14CS3.75X135	12CS3.75X105	M10	☑
3	Chord	2-2X4	2-2X10	M3	☑
4	Web	2X4	3X4	M19	☑
5	Conc-Col	Crect12x12	CRECT14X14	M9	☑
6	Conc-Bm	crect12x8	CRECT12X10	M5	☑
7	M16	W12X53	W12X58		☑
8	M17	W12X53	W12X58		☑
9	M18	W12X53	W12X58		☑

On this spreadsheet RISA-2D suggests an alternative shape for each section set or member. If we are already using the optimum shape for a section set or member, then it will not be listed on this spreadsheet.

Note that if you have modified your design lists by adding or removing shapes RISA-2D may pick different shapes than are shown here.

Click on the **Suggest Shapes** spreadsheet to make it active. The **Window Toolbar** presents the **Replace and Resolve** $\boxed{\text{⊕}}$ button. Click this button and agree to resolve by clicking **Yes.**

Now only a couple of members are in trouble. Let's look at another way to substitute suggested shapes.

Press Ctrl-Alt-F7 to substitute the shapes and resolve the model.

Now, the only suggested modification is to replace a W14x22 beam with a W12x22 beam and a CRECT12x10 column with a CRECT14x14 column. Since these don't result in any real cost savings, we'll ignore these last suggestions and move on.

Excluding Results

We will use the **Wood Code Checks** spreadsheet to sort the members and then hide results for members that have a code check value lower than "0.6".

From the Main Menu Toolbar, select **Window – Single View**. Then select **Results** and **Members ‣ Design Results** from the Main Menu Toolbar.

If it is not already selected, click on the **Wood** tab. If necessary, stretch the bottom of the window so that you can view more of the results.

Now, we'd like to sort our results based on the maximum code check to see which members are closest to failure.

Click anywhere in the code check column. Then click on the **Sort** ⬛ button and select **Sort Max to Min** and click **OK**. Now look for the last member with a code check value that is 0.5 or higher which is the member on the 10th line.

Click anywhere in this row to place the active cursor like this:

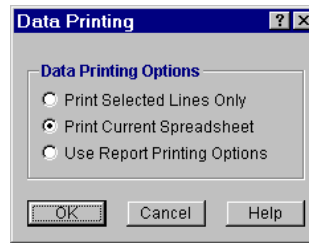|	Member	Shape	Code ...	Lo...	Ic	Shea...	Lo...	Ic	Fc'[ksi]	Ft'[ksi]	Fb'[ksi]	Fv'[ksi]	Eqn
1	M1	2-3X10	.736	26.25	2	.671	26.25	2	.465	1.375	2.063	.119	3.9-1
2	M2	2-3X10	.736	0	2	.671	0	2	.465	1.375	2.063	.119	3.9-1
3	M22	4X6	.569	0	2	.009	0	1	1.088	1.625	2.438	.119	3.6.3
4	M24	4X6	.569	0	2	.009	0	1	1.088	1.625	2.438	.119	3.6.3
5	M23	4X6	.566	0	2	.080	0	2	1.293	1.625	2.438	.119	3.6.3
6	M35	4X6	.566	5.483	2	.080	5.483	2	1.293	1.625	2.438	.119	3.6.3
7	M28	4X6	.564	0	2	.082	5.483	2	1.293	1.625	2.438	.119	3.6.3
8	M29	4X6	.564	5.483	2	.082	0	2	1.293	1.625	2.438	.119	3.6.3
9	M19	4X6	.546	0	2	.009	6.162	1	1.088	1.625	2.438	.119	3.6.3
10	M27	4X6	.546	0	2	.009	6.162	1	1.088	1.625	2.438	.119	3.6.3
11	M3	2-3X10	.473	4.688	2	.092	18.75	2	.85	1.375	2.063	.119	3.6.3
12	M4	2-3X10	.473	9.375	2	.092	0	2	.85	1.375	2.063	.119	3.6.3
13	M30	4X6	.234	0	2	.000	0	1	1.805	1.625	2.438	.119	3.9-1
14	M40	4X6	.234	0	2	.000	0	1	1.805	1.625	2.438	.119	3.9-1
15	M20	4X6	.196	0	2	.009	0	1	1.088	1.625	2.438	.119	3.6.3

Now click on the **Exclude After** ⬛ button (it has a red "X" on it) which is on the **Window Toolbar**.

You will see that the results for members after line 10 are no longer displayed. (Don't despair, you can always click **Unexclude** ⬛ to get them back.)

Printing Desired Results

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.

```
Data Printing                        ? ✕
┌─ Data Printing Options ──────────────┐
│  ○ Print Selected Lines Only         │
│  ● Print Current Spreadsheet         │
│  ○ Use Report Printing Options       │
└──────────────────────────────────────┘
   [  OK  ]    [ Cancel ]    [ Help ]
```

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 one concrete member, one steel member, one wood member, one cold formed steel member, and two joints so your screen looks something like this:
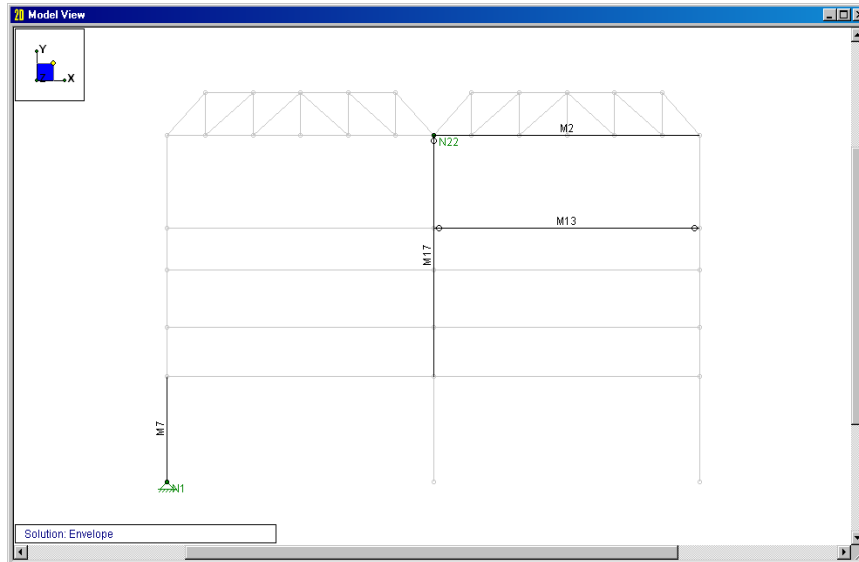
Now click on the **Exclude** Exclude button located on the **Select Toolbar.** Select **Yes** to exclude the results.

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] 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 **Story 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.
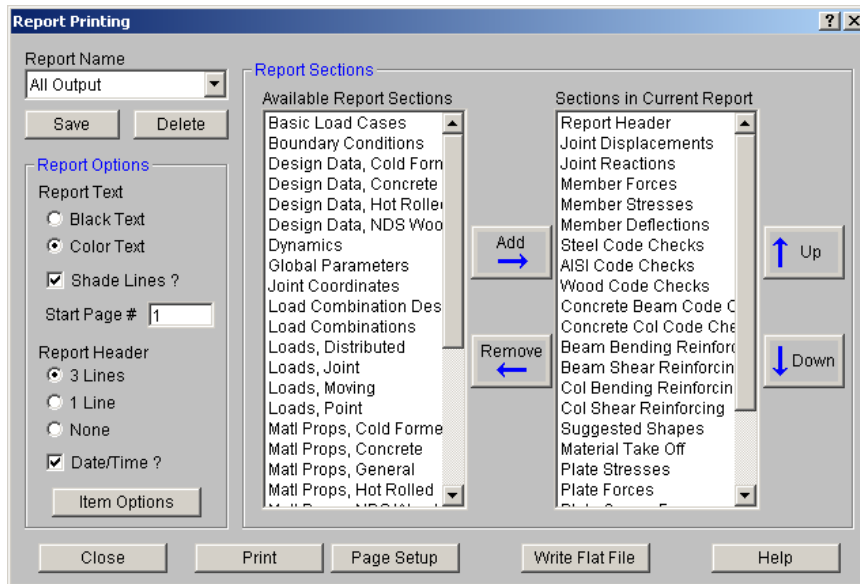
Click **Print** to get to your systems print dialog where you can select your printer and click **OK**. Click **Close** on the **Report Printing Options** dialog box.

At this point, we're finished with the tutorial. If you like, you can go back and increase member sizes, solve different Load Combinations, experiment with the plot; whatever you like.

To leave RISA-2D at this point, select **Exit** from the **File** menu.

Conclusion

If you've completed the tutorials you should have a very good feel for how to use RISA-2D. Some of the topics we have not covered are listed below. You may not need all of these features but it is good idea to know they are available. Of course, if you wish to know more about specific features, you can refer to the RISA-2D General Reference or the On-line help system.

Model Generation

RISA-2D has an expanding library of automatic generators that can quickly generate continuous beams, arcs, trusses, cylinders and numerous mesh layouts of our plate shell element. Click the button to access this feature.

Moving Loads

The standard AASHTO loads are built into the moving loads database and you can add custom moving loads as well.

Thermal Loads

You can model temperature differentials causing axial expansion or contraction of members.

Dynamics

You can run an eigenvalue solution to determine natural frequencies, and mode shapes. You can run a or Response Spectra analysis for earthquake loading.

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.

Member Offsets

You may rigidly offset the member ends to model rigid end zones and you may offset the tops of horizontal members to align member tops.

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 choosing RISA!