RISA-3D

Rapid Interactive Structural Analysis – 3-Dimensional

Version 10.0- General Reference

RISA TECHNOLOGIES

26632 Towne Centre Drive, Suite 210

Foothill Ranch, California 92610

(949) 951-5815

(949) 951-5848 (FAX)

www.risatech.com

Table of Contents

Before You Begin

Welcome to the RISA-3D General Reference manual. Please read this topic prior to installing the program and pay particular attention to the License Agreement. If you agree to the terms of the license then read the Installation section and install the program. If you are a first time user of RISA-3D you should turn your attention to the User's Guide (a separate document) which is designed to get you up and running as fast as possible while still exposing you to the important features of the software.

The User's 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 quickly move through the tutorial. If you want more thorough explanations of the modeling process you may read all or some of the supporting text as you see fit.

After you have gone through the User's Guide, use this General Reference for detailed information on any topic. The topics are arranged in alphabetical order and are thoroughly indexed.

Overview

RISA-3D is a general-purpose 3-dimensional analysis and design program. This program has been developed to make the definition, solution and modification of 3D structural models as fast and easy as possible. Analysis, up to and including calculation of maximum deflections and stresses, may be done on structures constructed of any material or combination of materials. Complete steel and wood design are also included in the program.

RISA-3D has full graphical modeling capability. You can draw your model on the screen and also perform extensive graphical editing simultaneously in multiple views. To modify your model data directly, RISA-3D employs a powerful, proprietary spreadsheet. All this combined with flexible data generation algorithms makes modeling very easy. Graphic display of the model along with applied loads, boundary conditions and much more, is always available. The model can be rapidly edited, solved, viewed, modified, re-solved, etc. The truly interactive nature of RISA-3D is its primary strength. RISA-3D is also able to perform elaborate error checking as you define the model, and offers context sensitive help every step of the way.

RISA-3D is an *interactive* program as opposed to a *batch* mode program. With a batch mode program, you would edit a text file in one program (typically called a pre-processor), and then solve it with another program, and then view the solution results in yet another program (typically called a post-processor). With RISA-3D, all model editing, model solution, and results browsing is accomplished through the same interface and with the same program. The interactive approach offers several unique advantages over batch mode which include; the ability to do real time error checking of your model data, the ability to do rapid model editing, solution, editing, and re-solution without jumping from one program to another, and the need for the user to learn only one program interface.

You may access the features in RISA-3D by using the menu system, or the toolbars. The best way to learn RISA-3D is to go through the User's Guide. The advantage to this is that you are exposed to the tools RISA-3D provides **AND** the ways that you can take advantage of them.

Hardware Requirements

Minimum

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

Recommended

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

Note

- The amount of space needed by RISA-3D to solve a particular structural model depends on the size of the model. RISA-3D has been written such that it will use as much RAM as is available. If this isn't enough, RISA-3D will start using HD space until enough memory is obtained to solve the problem. Of course, if RISA-3D is required to use HD space, the solution will be much slower. So, the more memory you have available, the better. In general, 500 Megabytes (MB) of RAM is a good amount to solve most problems. However, if you will be regularly solving large problems, more memory will save you a lot of time in the long run. See Solving Large Models for more information.

Program Limits

100,000 Joints

32,000 Members

100,000 Plates

100,000 Solids

5,000 Section Sets

500 Materials

1,000 Custom Wood Species

500 Diaphragms

1,000 Basic Load Cases

200,000 Loads

500 Moving Loads

5,000 Load Combinations

500 Mode Shapes

Demonstration Version: While you can open and solve a larger model, the largest model that can be saved to disk with the demonstration version is limited to 40 Joints, 40 Members, 40 Plates and 4 wall panels.

License Agreement

END-USER LICENSE AGREEMENT FOR RISA Technologies, LLC® SOFTWARE

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

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

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

3. LIMITED WARRANTY. RISA Technologies, LLC warrants that the SOFTWARE PRODUCT will operate but does not warrant that the SOFTWARE PRODUCT will operate error free or without interruption. RISA Technologies sole obligation and your exclusive remedy under this warranty will be to receive software support from RISA Technologies via telephone, email 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. CHOICE OF LAW. By entering into this Agreement in accordance with Paragraph 1, above, you have agreed to the exclusive jurisdiction of the State and Federal courts of 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.

“BCI” is a registered trademark of BOISE CASCADE WOOD PRODUCTS, L.L.C.

“Georgia-Pacific” is a registered trademark of Georgia-Pacific Corporation

"GPI" is a registered trademark of Georgia-Pacific Wood Products South LLC.

“iLevel” is a registered trademark of Weyerhaeuser NR Company.

“iLevel Trus Joist Microllam LVL” is a registered trademark of Weyerhaeuser NR Company.

“iLevel Trus Joist Parallam PSL” is a registered trademark of Weyerhaeuser NR Company.

“International Beams” is a registered trademark of International Beams, Inc.

"LPI" is a registered trademark of Louisiana-Pacific Corporation.

“Nordic” is a registered trademark of Les Chantiers de Chibougamau ltée.

“Pacific Woodtech” is a registered trademark of Pacific Woodtech Corporation.

“Red” is a registered trademark of REDBUILT, LLC

“Rebuilt” is a registered trademark of REDBUILT, LLC

"RFPI" is a registered trademark of Roseburg Forest Products.

“TimberStrand Trus Joist” is a registered trademark of Weyerhaeuser NR Company.

“TJI” is a registered trademark of Weyerhaeuser NR Company..

Maintenance

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

When your maintenance expires, you will be given the opportunity to continue program maintenance on an annual basis. You are under no obligation to continue program maintenance, of course, but if you decide to discontinue maintenance you will no longer receive RISA-3D program upgrades and technical support.

Complete program support is available to registered owners of RISA-3D and is included in the purchase price. This support is provided for the life of the program. See Technical Support 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-3D is released, the life of the previous version is considered to be ended.

RISA Technologies will support only the current version of RISA-3D.

Installation

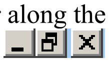
To install RISA-3D please follow these instructions:

1. Put the RISA-3D 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.

Application Interface

The User's Guide (a separate document) contains a tutorial that leads you through the RISA-3D interface with an actual model. Consider going through the tutorial if you have not done so already, as it is the fastest way to learn the program. Although it requires some time up front, the tutorial will save you time and brainpower in the long run.

The features that are available to you in RISA-3D may be accessed through the main menu, shortcut menus, toolbars and shortcut keystrokes. You may use any or all of these vehicles to interact with the software. The main menu has the advantage of containing all of the program options and features and may initially be the simplest to use, letting you learn just one system. The toolbars contain more common options and invoke with one click. The shortcut menus present options relevant to the task at hand. The shortcut keys provide a fast way to access features should you use the program often enough to make them familiar to you. All of these features are discussed in the sections below. There are many ways to access features and the method that you will use will simply be a matter of personal preference. The good news is that you have the options.

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

The actual work that you do will be in the main area on the screen, which is called the workspace. When you open a model view, a spreadsheet or a dialog it will be opened in the workspace and listed in the **Window** menu. You may have as many windows open as you like.

Main Menu

All of the program features may be accessed through the main menu system at the top of the screen beginning with **File** on the far left and ending with **Help** or possibly **Director** on the far right. Clicking on each of these menus (listed below) will display sub-menus that contain options that you may choose from. You may also select the main menus by using the ALT key along with the underlined letter in the menu you wish to choose. You may then continue to use the keyboard to choose from the menu options. In addition, some of the menu options will have hot key combinations listed to the right of the option. These hot keys allow you to use the keyboard to access features without using the menu system.

File Menu

New will close the current file, prompting for saving if necessary, and will open a new file.

Open will close the current file, prompting for saving if necessary, and will open an existing file.

Save will save the current file, prompting for a name if necessary.

Save As will save the current file, prompting for a name.

Append will insert another RISA-3D model into the current model.

Import will close the current file, prompting for saving if necessary, and will open an existing RISA-2D (R2D), DXF, or STD file.

Export will export the current file to a DXF, SDNF, or Pro-Steel exchange file. Also allows the export of connection forces to the Descon connection design program..

For more information on the interaction between RISA and other programs refer to Appendix E.

Print will access RISA-3D printing options.

Page Setup will present page setup options for printing.

Recent Files The five most recent files will be listed at the bottom of the menu. Selecting one of these files will close the current file, prompting for saving if necessary, and will open the selected file.

Exit will close RISA-3D, prompting for saving if necessary.

Edit Menu

Undo will undo the last edit that was applied to the model whether it was made graphically or in the spreadsheets. You may continue to apply Undo to remove up to 100 model edits.

Redo will reverse the last undo that was applied to the model. You may continue to apply Redo to remove up to 100 undo operations.

Copy will copy the selected spreadsheet cells or model view from the active window to the clipboard.

Paste will paste data from the clipboard to the spreadsheet cells.

Insert Line will insert a new line in the spreadsheet beneath the current line.

Delete Line will delete the current spreadsheet line.

Repeat Line will insert a new line in the spreadsheet beneath the current line and copy the data from the current line.

Mark All Lines will select all of the lines in the spreadsheet.

Unmark Lines will unmark any currently marked lines.

Delete Marked Lines will delete the marked lines in the spreadsheet.

Find will locate an item on the spreadsheet by its label.

Sort will sort the column containing the active cell.

Fill Block will fill the marked block of cells with a valid entry.

Math on Block allows you to add, subtract, multiply or divide the values in the marked block of cells.

Global

Global opens the Global Parameters for the model.

Units

Units opens the Units settings.

View Menu

New View will open a new model view window.

Save or Recall Views allows you to save a view or recall a view that has previously been saved.

Clone View makes a copy of the current view so you can modify one and maintain the other.

Refresh All will refresh all of the windows that are open in the workspace.

Select provides graphic select options that are also provided on the **Selection Toolbar**.

Unselect provides graphic unselect options that are also provided on the **Selection Toolbar**.

Save or Recall Selection States allows you to save a selection or recall a selection that has previously been saved.

Zoom provides options for zooming in and out of the current model view.

Rotate provides options to snap the model view to global planes or an isometric view.

Plot Options opens the Plot Options.

Render will turn rendering of the current model view on or off, depending on the current setting.

Drawing Grid will turn the display of the Drawing Grid on or off, depending on the current setting.

Project Grid will turn the display of the Project Grid on or off, depending on the current setting.

Axes will turn the display of the global axes in the model view on or off, depending on the current setting.

Boundaries will turn the display of the boundary conditions on or off, depending on the current setting.

Loads will turn the display of the model loads on or off, depending on the current setting.

Joint **Labels** will turn the display of the joint labels on or off, depending on the current setting. A third setting is also available where the joints themselves are not shown at all.

Member **Labels** will turn the display of the member labels on or off, depending on the current setting. However, if rendering is turned on, member labels will not be visible in the model view.

Insert Menu

The **Insert Menu** will help you insert new items into the model. Most of the options will provide a graphical method of insertion but some will open spreadsheets where appropriate. See Graphic Editing for specific information.

Modify Menu

The **Modify Menu** will help you modify existing items in the model. Most of the options will provide a graphical method of modification but some will open spreadsheets where appropriate. The **Delete Items Dialog** may also be accessed via this menu. See Graphic Editing for specific information.

Spreadsheets Menu

The **Spreadsheets Menu** provides access to any of the input spreadsheets. See Spreadsheet Operations to learn how to work within the spreadsheets.

Solve Menu

Clicking on the **Solve Menu** will immediately begin a solution to the model. See Solution for more information.

Results Menu

The Results Menu provides access to any of the results spreadsheets. See Results Spreadsheets for more information.

Tools Menu

Relabel Joints assigns new labels to the joints in their current order in the **Joint Coordinates** spreadsheet.

Relabel Members assigns new labels to the members in their current order in the **Members** spreadsheet.

Relabel Plates assigns new labels to the plates in their current order in the Plates spreadsheet.

Full Model Merge will merge the entire model. See Model Merge for more information.

Align Wall Panel will perform a merge of the wall panels to make sure they are lined up in the vertical direction. Use this utility if you are receiving wall panel errors at solution.

Round off joint coordinates will round off the coordinates.

Switch Vertical Axes allows you to switch your vertical axis while maintaining consistent member orientation.

Detach RISA3D from RISAFloor allows you to take a RISA-3D model that is linked to RISAFloor and “detach” it so that you can open / edit it without first going through RISAFloor.

Preferences contain settings that let you customize the program. See Customizing RISA for more information.

Customize Toolbar... allows you to modify the model view toolbar by adding, subtracting and re-ordering buttons. See the customizable toolbar section.

Reset All Program Defaults will reset all customized settings to the original factory settings.

Window Menu

In order to help you work with the model and the results, you are provided with many window arrangements to choose from. You may access them from the **Window Menu**. The best way to understand just what these 'tilings' do is to try them. Remember that once you choose a tiling you may adjust any of the windows as you wish. You may also use the **Tile** 

button on the **RISA Toolbar** to access a list of tilings.

Help Menu

Help Topics opens the RISA-3D Help File so that you may search the contents and the index. See Help Options to learn about getting help.

Check for Updates runs an internal check for possible program updates. If your program is up to date, you will receive a message saying you are up to date. If you are out of date, the check will offer you the option to email RISA Technologies for upgrade information if you are out of date for a major update. If you are out of date just a minor update, then we will send you to our website to upgrade.. This check is also offered during the installation process.

About provides RISA-3D version and hardware key information.

Director Menu

If you are working from within the RISA Building System (RBS), use this menu to switch between RISAFloor, RISA-3D and RISAFoundation. If you are not working within the RISA Building System, the **Director Menu** will not be shown.

The directory button is located at the far, far right hand side of the Main Menu Toolbar as shown in the image above.

Shortcut Menu

The **Shortcut Menu** is also referred to as the **Right-Click Menu**. This is because to access the shortcut menu you simply click the RIGHT mouse button where you are working to see options that are relevant to what you are doing. For example if you are working in a model view the right click menu will provide options to help you modify the view and edit the model graphically. If you are working in a spreadsheet the menu will provide editing tools for that spreadsheet.

This menu will appear wherever you RIGHT click the mouse. This way you do not need to move away from where you are working to select the features you want to use.

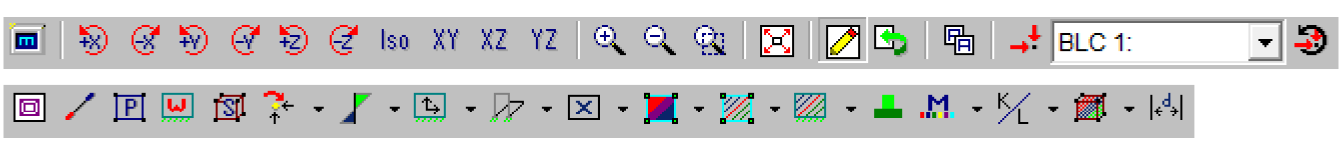
Toolbars

The **Toolbars** provide buttons to help you access common commands and popular options in the menu system discussed above. There are different toolbars that will appear as you work to build your model and browse your results. 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 first horizontal toolbar located just below the Main Menu is called the **RISA Toolbar.** The buttons on this bar facilitate file and window access. You may use these buttons to open files and windows and also to analyze the model.

Window Toolbar



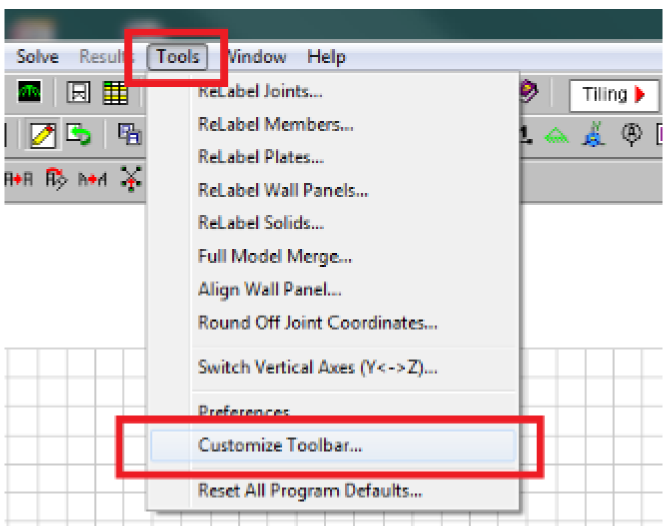
The **Window Toolbar** is the second horizontal toolbar located below the Main Menu. It gets its name because the buttons change as you move from window to window in order to help you with what you are currently doing. When you are working in a model view the buttons provide viewing tools, such as **Rotate** and **Zoom**, to assist you with that view. There are also many other results and information display toggles, including some icons with the drop down arrow next to them. Clicking the arrow will show you the different view options for that icon. Clicking the icon itself will bring you back to the default view. Note that this model view toolbar is now fully customizable. See below for more information.

Other model view windows that are open will not be affected so that each may show different information. When you are working in a spreadsheet, editing tools are provided that are appropriate to that particular spreadsheet. Note that not all tools are available with all spreadsheets. In fact there are many tools that are provided for one spreadsheet only. See Spreadsheet Operations for more information.

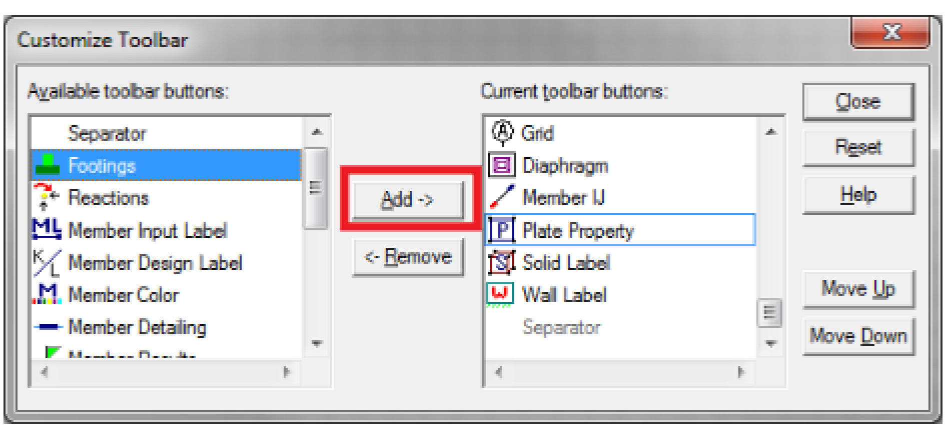
Customizable Model View Toolbar

The model view toolbar is fully customizable. By creating your personalized toolbar, you can quickly access your most frequently used buttons. This can be done quickly and easily in just a few steps.

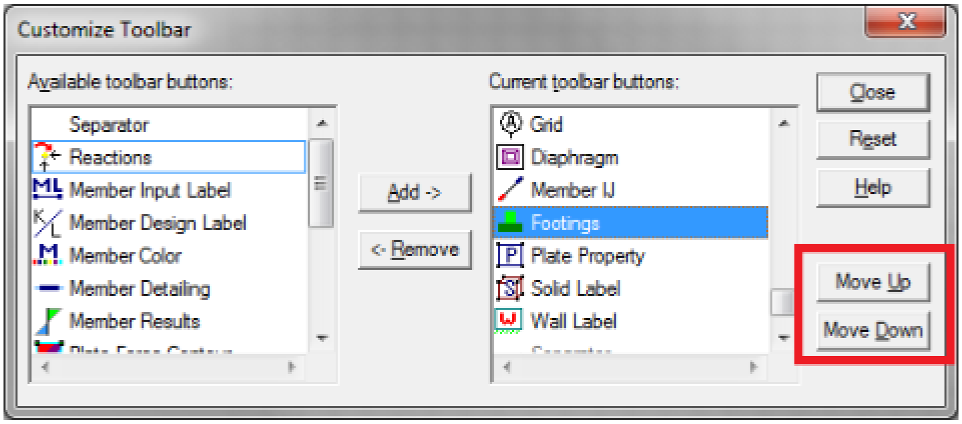
1. Go to Tools menu and select Customize Toolbar.



2. Select one of the toolbars by clicking in the box Available toolbar buttons, and click on Add to place them on the current toolbar.



3. Once you've moved the buttons to the Current Toolbar, you can rearrange them by clicking on Move Up or Move Down.



4. Click Close and you will see your selections on the model view toolbar.

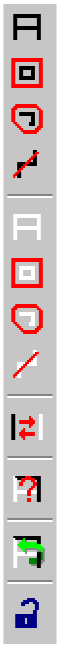
Note:

- You must have a model view as the current view to see this toolbar.
- If you add more buttons than will fit on the toolbar the buttons that are at the end of the "Current toolbar buttons" will be cut off.
- The changes you have made will automatically be saved on a per-user (Windows User) basis, such that next time you open the program the toolbar will be arranged per your preferences.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Customized Toolbar**.

Selection Toolbar

The vertical toolbar on the left side of the screen is the **Selection Toolbar.** This toolbar will only be available when the active window is a model view. The buttons on this toolbar help you select and unselect items in the model in order to help you build and modify the model or view results. See Graphic Selection for more information.

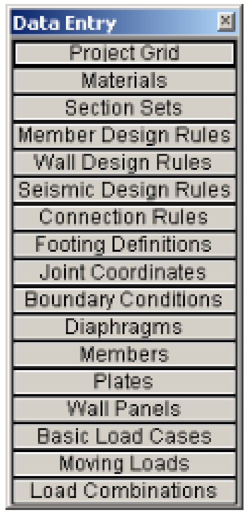


Drawing Toolbar

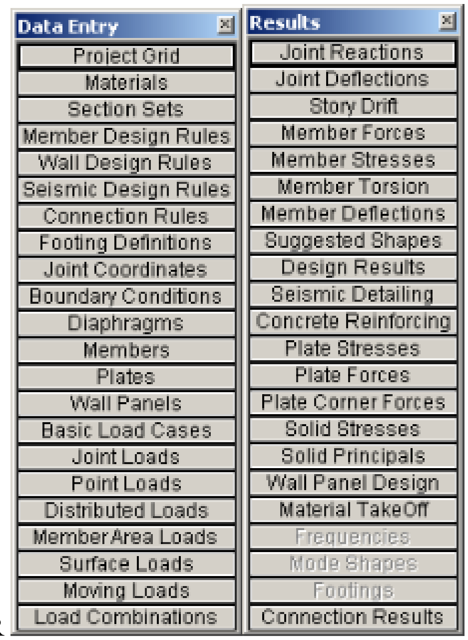
Another toolbar that is available is the **Drawing Toolbar**. Unlike those mentioned above, this toolbar is located in the model view windows rather than in the main application window. This way the drawing tools stay close to where you are working. This toolbar controls modeling features that help you draw, load, and modify your model graphically. You may have more than one view open and a Drawing Toolbar for each view. This way you can simultaneously draw plates in one window and members in another.

The **Drawing Toolbar** may be displayed in any model view window by clicking on the **Window Toolbar** while in the model view window. Some of the buttons on the toolbar are for one-time applications such as modifying the drawing grid. Other buttons place you in an editing mode, such as Draw Members, that remains active until you cancel it. The current mode is indicated by the mouse pointer and by the state of the button. While in an editing mode the button will stay down until you click it again or choose another button. See Graphic Editing for more information.

This brings us to an important point. Some of the toolbar buttons remain down when you press them to indicate that you are in a certain mode or that something is either on or off. For example the **Box Zoom** button will stay down to indicate that you are currently in the zooming mode. The **Show Drawing Toolbar** button will remain down when you turn on this toolbar for the active window. You may be in more than one mode at the same time as long as they are not mutually exclusive.



OR



The **Data Entry Toolbar** is the vertical toolbar on the right side of the application window. It contains buttons that facilitate data entry through the spreadsheets. The buttons on this toolbar provide quick access to the spreadsheets that are also listed in the **Spreadsheets Menu**. You may open and close the toolbar by clicking the button on the **RISA Toolbar.**

Note

- Some of the Loads buttons have been removed. However, there is now an option in the Tools-Preferences dialog that will allow you to add these buttons back. These spreadsheets can also be easily accessed from the Basic Load Cases spreadsheet.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Data Entry**.

The **Results Toolbar** is the vertical toolbar on the right side of the application window that is placed over the **Data Entry Toolbar** after the model has been solved. The buttons on this toolbar provide quick access to the results spreadsheets that are also listed in the **Results Menu**. You may open and close the toolbar by clicking the button on the **RISA Toolbar.**

Dynamic View Controls

When your current window is a graphical model view, you can use the mouse wheel to dynamically zoom, pan, or rotate the graphical image. These functions are only available to users who have a mouse with a wheel button and whose computers are running the Windows XP operating system.

Mouse Action	Model View Function
Rolling the Wheel Forward	Zoom In
Rolling the Wheel Backward	Zoom Out
Clicking and holding the Wheel Button	Grab the image and pan in the direction of mouse movement
Click and hold the Wheel button while pressing the	Dynamically rotate the structure in the direction of mouse movement

Shift key

Dynamic Pan: Clicking and holding the mouse wheel button triggers the tool and allows the user to pan or drag the view to the limit of scroll bars.

Dynamic Zoom: This tool uses the wheel button on the mouse. Rotating forward zooms in and rotating backward zooms out.

Dynamic Rotate: This tool is triggered by clicking and holding the mouse wheel button while holding the Shift key. The rotational movement will be based on the how the user drags the mouse cursor over the screen and the projection of global axis on the screen. For rotation about X axis, drag the cursor perpendicular to the projection of the global X axis. The same logic applies for Y or Z axis rotations. When rotation is initiated, the system locks for rotation about that axis until the user releases the middle mouse button.

Zoom Previous/Next: Function keys F3 and F4 are associated with Zoom Previous and Zoom Next respectively. The system holds a doubly linked list of zoom info. This list has 10 zoom-states in the list. The F3 or F4 keystroke moves the active pointer forward or backward on the list. Each window has its own zoom list.

Dynamic Distance Tool: This tool triggers by pressing the F5 key. The user has to pick up two points on the screen and the system gives back the total and partial distance between points on the status bar.

Shortcut Keys and Hot Keys

Shortcut Keys and **Hot Keys** allow you to use the keyboard to quickly access features. The difference between the two is simply that the shortcut keys are related to a specific window and will only work in that window while the hot keys will perform at most any time.

General Hot Keys

Key Combination	Function
F1	Help on the active window
F5	Activates the Dynamic Distance Tool
Ctrl-F1	Main Help topics
Ctrl-F2	Create New view
F7, Ctrl-F7	Opens solution choices
Ctrl-Alt-F7	Replace shapes with suggested shapes and re-solve the model
Ctrl-C	Copy to the clipboard
Ctrl-V	Paste from clipboard
Ctrl-N	Start a new file
Ctrl-O	Open an existing file
Ctrl-S	Save the current file
Ctrl-P	Print
Ctrl-Z	Undo
Alt-	Access the menus by combining the Alt key with the underlined letter in the menu

Shortcut Keys available for Specific Windows

Key Combination	Model View Window	Spreadsheet
Ctrl-D	Open last graphic editing dialog	Delete Marked Lines
Ctrl-G	Toggle Drawing Toolbar	
Ctrl-A	Select All	
Ctrl-U	Unselect all	
Ctrl-F		Block Fill
Ctrl-M		Block Math
Ctrl-I	Invert Selection	
Ctrl-L	Toggle Lock unselected	Unmark lines
Ctrl-Enter		Press cell  button
F2	Open Plot Options	Start/Stop Cell Edit
F3		Insert line
F4		Delete Line
F5	Initiates the "Distance" tool	Find
F8		Repeat Current Line
F9		Sort
+	Zoom In	
-	Zoom Out	
←→↑↓ PgUp PgDwn	Scrolling	Scrolling

Spreadsheet Hot Keys that open spreadsheets

Key Combination	Unsolved model	Solved Model
Ctrl-Alt-B	Basic Load cases	
Ctrl-Alt-C	Joint Coordinates	Corner Forces
Ctrl-Alt-D	Distributed Loads	Joint Deflections
Ctrl-Alt-E	Members – Primary Data	Member Deflections
Ctrl-Alt-F		Member Forces
Ctrl-Alt-G	Global Parameters	
Ctrl-Alt-H	Model Generation	Suggested Shapes
Ctrl-Alt-I	Diaphragms	Member Torsion
Ctrl-Alt-L	Load Combinations	Plate Forces
Ctrl-Alt-M	Materials	Material Take Off
Ctrl-Alt-N	Joint Loads	Concrete Reinforcing
Ctrl-Alt-O	Boundary Conditions	Mode Shapes

Ctrl-Alt-P	Member Point Loads	Plate Stresses
Ctrl-Alt-Q		Frequencies
Ctrl-Alt-R	Design Rules	Reactions
Ctrl-Alt-S	Section Sets	Member Stresses
Ctrl-Alt-T		Story Drift
Ctrl-Alt-U		Design Results
Ctrl-Alt-V	Moving Loads	
Ctrl-Alt-X	Surface Loads	
Ctrl-Alt-Y	Dynamics Settings	
Ctrl-Alt-Z	Area Loads	
Ctrl-Alt-4	Plates	

Status Bar

The **Status Bar** passes useful information to you as you work. It is divided into four parts located along the very bottom of the main application window, just beneath the workspace.

The left side of the status bar shows a solution flag to indicate the solved state of the model as follows:

Solution Type	Unsolved	Solved
Static		
Dynamic		
Response Spectra		

To the right of the solution flags there are three message boxes.

The first and largest box lets you know what you are currently doing. If you are in a spreadsheet, this box will contain the explanation of the current cell. If you are working in a model view and select a graphic editing option, look to this box for information on how to use the feature.

The second box is used to pass you units of the current spreadsheet cell.

The third box indicates the coordinates of the mouse when a model view is active. The mouse coordinates that are displayed are the coordinates of the grid point or joint that is nearest to the mouse.

Windows

Modeling the structure will take place within model views and spreadsheets, each in their own window that may be moved around the workspace and sized as you wish. The ability to have multiple model views and multiple spreadsheets open at one time is a powerful feature. The options in the **Window Menu** are provided to help you manage these windows.

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

Model Views

Model View windows show a graphic view of the model. Open a new view with the  button.

You may open as many model view windows as you like. This is especially helpful when working in close on large models. You might have one overall view and a few views zoomed in and rotated to where you are currently working. You may also have different information plotted in multiple views.

One thing to remember is that the toolbars that are displayed depends upon what window is active. The active window is the one with the blue 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

Spreadsheet windows 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. 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. There is always one and only one active cell, which is the cell that has the “attention” of the keyboard.

You may also select blocks of data to work on. You can select a block of data by clicking and holding the mouse button on the first cell in the block and then dragging the mouse to the opposite corner of the block and releasing the mouse.

Dialogs

A **Dialog** is a third type of window and is used to access a specific function within the program. Another powerful feature is that most of the dialogs may be left open while you edit the model, making it easy to make adjustments as you work. 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.

Window Tiling

Standard window tilings help you set up your workspace. Select the **Tile** Tiling button and then select a tiling or choose them from the **Window ▸ Special Tiling** menu.

The standard tilings include arrangements of spreadsheets and model view windows for creation of models and viewing results. Each of these groups have arrangements for working with joints, members, and plates and also loads. The best way to learn what these tilings do is to try them.

Modes

There are three basic program modes (**View, Select,** and **Edit**) and a mode hierarchy to allow you to move between them easily. While you are **editing** the model you may **select** items to edit. When you are finished **selecting** you will be returned to **editing**. Likewise, while you are **selecting** items you can adjust the **view** and then be returned to **selecting**.

Different mouse cursors are used with each mode to make it clear what the current mode is.

View Mode is the upper level mode that allows you to adjust the view by zooming in and out, rotating and setting plot options. This mode supersedes all other modes so that you may do these things at any time, and then be returned to the previous mode. This mode does not cancel other modes so that when you are finished adjusting the view you are returned to what you were doing. See Graphic Display for more information.

Select Mode is the middle level mode that allows you to make a graphic selection of joints, members and plates. This mode supersedes the **Edit Mode** but not the **View Mode**. This means that you can make a selection while in the middle of editing the view and when you are finished you are returned to the editing feature that you were using. It also means that you may adjust the view while remaining in the same **Select Mode**. See Graphic Selection for more information.

Edit Mode is the lower level mode that allows you to graphically edit the model. You may make selections and adjust the view while in the edit mode such that when you are finished selecting you will be returned to the **Edit Mode**. Some **Edit Mode** features have options on how you apply the edit. See Graphic Editing for more information.

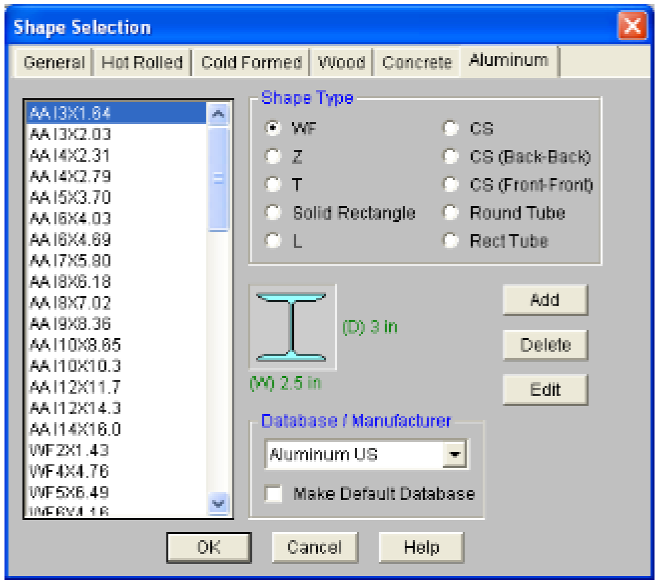
Note

- The default mode is the mode you are in if you are not in any other mode and is indicated by the standard mouse cursor. The default mode is a selection mode where you can select/unselect individual items by clicking on them. You may also double-click on an item to view information about the item.
- You may use the ESC key or the right mouse button to cancel a mode.

Aluminum - Databases

Shapes are organized in the database by country. The shapes available are from the ADM 2005 Section Properties section. You may type in the names directly, select shapes from these databases or add your own shapes.

RISA currently supports the following common Cold Formed steel databases: Aluminum US, and Aluminum CAN.

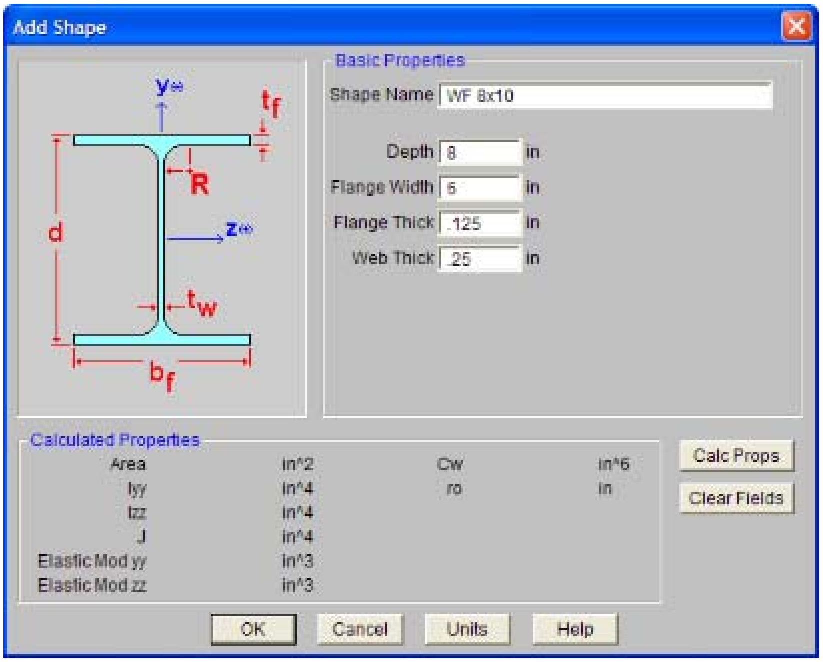


To Select a Cold Formed Database Shape

1. From the AluminumSection Sets tab on the **Spreadsheet**, or the **Primary** tab of the **Members Spreadsheet** move the cursor to the **Shape** field and click .
2. Specify the database and shape type you wish to use and then select from the list of available shapes by clicking on .

Database Files

The aluminum shape databases are stored in the files ADMdbUS32.fil and ADMdbCAN32.fil.



To Add a Database Shape

1. On the **RISA Toolbar** click the **Edit Shape Database** button.
2. Select the Aluminum tab, then select the shape type you wish to add and click the **Add** button.
3. Specify a name for the shape and fill in the **Basic Properties.**
4. Click **Calc Props** to determine the shape properties.

Note

- Alterations to the shape database are not permanent unless you agree to save them. Changes that are not saved only remain valid for the current session and will not be present the next time you start RISA.
- New shapes are added to the bottom of the database.
- To delete a shape, specify the database and shape type you wish to delete and then click the **Delete** button.
- To edit a shape, click the **Edit** button and edit the shape properties. Values can only be manually edited here, nothing will be recalculated. If you wish to have all the values for a shape recalculated, you will need to delete the shape and then add it again with the new properties.

Aluminum Shape Types

There are ten types of shapes. Names for each shape type follow the convention of the manufacturer for each shape. If you know the shape name, you can type the name directly into the **Shape** field on the spreadsheets. Alternately you may click the button to look up a shape and select it.

WF sections

The wide flange shapes are called out by the designation given them in the aluminum manual. For example, if you wanted to use a WF10x11.4 you would enter WF10X11.4 as the shape name in the database shape field. Aluminum Association Standard I-Beams(AA), American Standard(S), Army-Navy(A-N), Canadian(CAN) I-Beams and Wide Flange shapes are available.

Zee sections (Z)

The Z shapes are called out by the designation given them in the ADM manual.

Tee sections (T)

The T shapes are called out by the designation given them in the ADM manual. Army-Navy (A-N) shapes are also available.

Solid Rectangular

The Solid Rectangular sections or bar sections are defined by the user, there are no default shapes. The syntax is "htXbase", where "ht" is the rectangle height and "base" is the rectangle base (in inches or cm). For example, 10X4 would be a 10" deep, 4" width rectangular shape (assuming US Standard units). These shapes are also be defined in the Shape Editor.

Angles sections (L)

Angles are entered with an "L" prefix. The syntax is "LlongXshortXthick", where "long" is the long leg length, "short" is the short leg length, and "thick" is the thickness, in number of decimals. For example, L5X3X0.375 is a 5" by 3" angle 0.375" thick. Square End Angles (LS) shapes are also available.

Channel sections (CS)

The CS shapes are called out by the designation given them in the ADM manual. The Aluminum Association (AA), American Standard Channels (C) and Car and Ship Building Channels (CS), Canadian Channels (CAN) are available.

Double Sections

The CS shapes are also available Back-to-Back or Front-to-Front orientation.

Note

- The program currently only performs an analysis of double sections and does not perform a code check of any kind. This may be added into a future revision of the program.

Round Tube or Pipe (OD) or (NPS)

The Round Tube shapes are called out by the designation given them in the ADM manual. The Outer diameter call out is used as well as the Nominal Pipe Size.

Rect Tube sections (RT)

The RT shapes are called out by the designation given them in the ADM manual.

Aluminum - Design

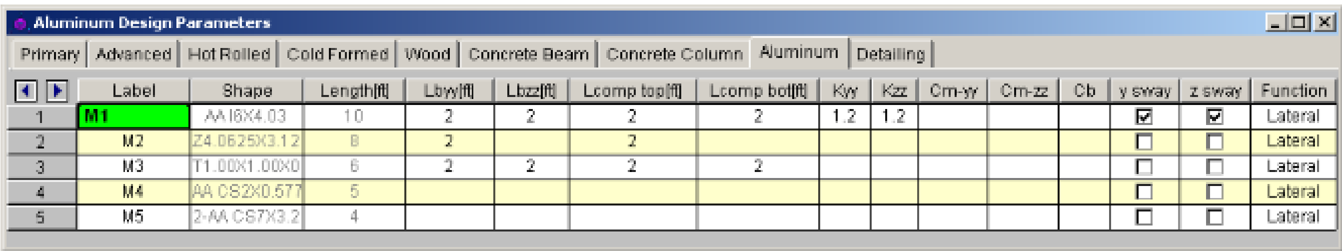
Full code checking can be performed on standard aluminum shapes, based on the following codes:

- Aluminum Design Manual 2005

Aluminum properties are available in the database and the values are based on the ADM values (See Aluminum Database). You may also input your own basic shapes and the properties will be calculated automatically.

Design Parameters

The **Aluminum** tab on the **MemberSpreadsheet** records the design parameters for the aluminum code checks. These parameters may also be assigned graphically. See Modifying Member Design Parameters to learn how to do this.


Aluminum Design Parameters

Primary | Advanced | Hot Rolled | Cold Formed | Wood | Concrete Beam | Concrete Column | Aluminum | Detailing

	Label	Shape	Length[ft]	Lbyy[ft]	Lbzz[ft]	Lcomp top[ft]	Lcomp bot[ft]	Kyy	Kzz	Cm-yy	Cm-zz	Cb	y sway	z sway	Function
1	M1	AA I6X4.03	10	2	2	2	2	1.2	1.2				☑	☑	Lateral
2	M2	Z4.0625X3.12	8	2		2							☐	☐	Lateral
3	M3	T1.00X1.00X0	6	2	2	2	2						☐	☐	Lateral
4	M4	AA CS2X0.577	5										☐	☐	Lateral
5	M5	2-AA CS7X3.2	4										☐	☐	Lateral

These parameters are defined for each aluminum member. The entries are explained below.

Label

You may assign a unique **Label** to all of the members. Each label must be unique, so if you try to enter the same label more than once you will get an error message. You may relabel at any time with the **Relabel** options on the **Tools** menu.

Shape

The member **Shape** or Section Set is reported in the second column. This value is listed for reference only and may not be edited as it is dictated by the entry in the Section/Shape column on the **Primary** tab.

Length

The member **Length** is reported in the third column. This value may not be edited as it is dependent on the member end coordinatelisted on the **Primary Data** tab. It is listed here as a reference for unbraced lengths which are discussed in the next section.

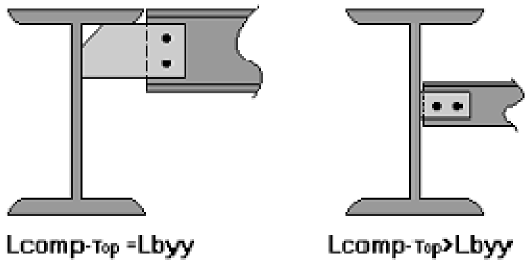
Unbraced Length

You may specify unbraced lengths or have RISA-3D calculate them for you. The unbraced lengths are **Lbyy**, **Lbzz**, **Lcomp-top** and **Lcomp-bot**.

The **Lb** values, **Lb**yy and **Lb**zz represent the unbraced length for the member with respect to column type buckling about the member's local y and z axes, respectively. These **Lb** values are used to calculate KL/r ratios for both directions, which in turn impact the calculation of the axial strength, Pn. The KL/r ratios gauge the vulnerability of the member to buckling. Refer to Section C4 in Part V of the AISI code for more information on this. Also, Section C4 lists some limiting values for KL/r. These limiting values are NOT enforced by the program.

The **Lcomp** values, **Lcomp-top** and **Lcomp-bot**, are the unbraced lengths of the compression flanges for flange buckling due to flexure. These may be the same as the Lbyy but not necessarily.

For continuous beams the moment will reverse such that the top and bottom flanges will be in compression for different portions of the beam span. **Lcomp-top** is the unbraced length of the top flange and **Lcomp-bot** is the unbraced length of the bottom flange.



If left blank these unbraced lengths all default to the member's full length. The exception to this is if **Lb**yy is entered and **Lcomp-top** is left blank, **Lcomp-top** will default to the entered value for **Lb**yy

For physical members, you can enter the code "**Segment**" in the unbraced length fields and the length of each segment will be used. A "segment" is the distance between the joints that are on the physical member. For example, suppose you have a physical member that is 20 feet in length, and there are two joints along the physical member, one 5 feet from the end and one at 15 feet. An unbraced length of feet will be used for the first segment then a value of 10 feet will be used in the middle segment, and again a value of 5 feet would be used in the last segment

Note

- If the intermediate framing members are considered to brace the bottom flange, then you can enter "segment" for Lcomp-bot. When the "segment" command is used ALL intermediate points along the beam are viewed as brace points. Therefore, you may have to delete unused or extraneous points.
- The Top Flange is defined as the flange corresponding to the positive local y axis for the member. For more information on setting local axes refer to the Members section.
- The calculated unbraced lengths are listed on the **Member Detail** report.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Unbraced Lengths**.

K Factors (Effective Length Factors)

The **K Factors** are also referred to as effective length factors. **Kyy** is for column type buckling about the member's local y-y axis and **Kzz** is for buckling about the local z-z axis.

If a value is entered for a **K Factor**, that value will be used for the entire length of the physical member. If an entry is not made (left blank), the value will internally default to '1' for that member.

RISA-3D is able to approximate the K values for a member based on the member's sway condition and end release configuration. The K-factor approximation is based on a rational method provided by other codes including AISC, AISI, etc. The following table gives the values used for various conditions.

Table Case	End Conditions	Sidesway?	K-Value
(a)	Fixed-Fixed	No	.65
(b)	Fixed-Pinned	No	.80
(c)	Fixed-Fixed	Yes	1.2
(d)	Pinned-Pinned	No	1.0
(e)	Fixed-Free	Yes	2.1
(f)	Pinned-Fixed	Yes	2.0

RISA-3D will recognize a pinned boundary condition for the K approximation for a full pin, i.e. if all the rotations in the boundary condition are released. If any of the rotations in a boundary condition are restrained, the boundary condition is considered “fixed” for the K approximation.

Any configuration not described here will be given the default value of 1.0.

If any value that influences these K values is changed, the K approximation should be redone. For instance, if you have RISA-3D approximate K values then change some end release designations, you should redo the K approximations.

Remember that the K-values are *approximations,* and you should check to make sure you agree with all K-values RISA-3D assigns. You can always override a K-value after an approximation by directly entering the value that you want in the appropriate field. Keep in mind that a subsequent approximation will overwrite any manually input values so you will need to override the approximation each time it is performed.

Limitation:

RISA-3D will currently neglect the influence of adjoining framing members when those members are connected at a joint that also has degrees of freedom restrained by boundary conditions. For example, suppose a column and beam member connect at a joint that is restrained for translation in all directions (i.e. the joint is "pinned"). The K factor approximation will neglect the beam member when it calculates the K factor for the column and visa-versa. The effect will be that the ends of the members at that joint will be seen as "pinned" and not "fixed" for the K factor approximation.

Cm - Interactive Bending Coefficient

Cm Coefficients are described in Section 4.1.1 of the ADM code. If these entries are left blank, they will be automatically calculated.

The **Cm** value is influenced by the sway condition of the member and is dependent on the member's end moments, which will change from one load combination to the next, so it may be a good idea to leave these entries blank.

Cb - Bending Coefficients

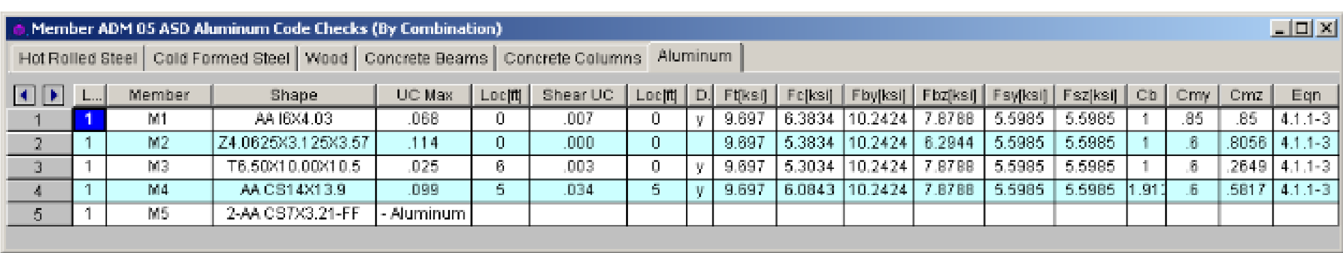
For the aluminum codes, **Cb Coefficients** depends on the moment variation over the unbraced length as described in section ADM 4.9.4. If this entry is left blank, it will be calculated automatically.

Sway Flags

The **Sway Flags** indicate whether the member is to be considered subject to sidesway for bending about its local y and z axes. The **y sway** field is for y-y axis bending and the **z sway** field is for z-z axis bending. Click on the field to check the box and indicate that the member is subject to sway for that particular direction, or leave the entry blank if the member is braced against sway. These sway flags influence the calculation of the K factors as well as the Cm and Cb factors.

Aluminum Design Results

Access the **Code Check** spreadsheet by selecting the **Results** menu and then selecting **Members** ▸ **Design Results** or by clicking on the **Design Results** button on the Results toolbar.


Member ADM 05 ASD Aluminum Code Checks (By Combination)

Hot Rolled Steel | Cold Formed Steel | Wood | Concrete Beams | Concrete Columns | Aluminum

	L...	Member	Shape	UC Max	Loc[ft]	Shear UC	Loc[ft]	D.	Ft[ksi]	Fc[ksi]	Fby[ksi]	Fbz[ksi]	Fsy[ksi]	Fsz[ksi]	Cb	Cmy	Cmz	Eqn
1	1	M1	AA I6X4.03	.068	0	.007	0	y	9.697	6.3834	10.2424	7.8788	5.5985	5.5985	1	.85	.85	4.1.1-3
2	1	M2	Z4.0625X3.125X3.57	.114	0	.000	0		9.697	5.3834	10.2424	6.2944	5.5985	5.5985	1	.6	.8056	4.1.1-3
3	1	M3	T6.50X10.00X10.5	.025	6	.003	0	y	9.697	5.3034	10.2424	7.8788	5.5985	5.5985	1	.6	.2649	4.1.1-3
4	1	M4	AA CS14X13.9	.099	5	.034	5	y	9.697	6.0843	10.2424	7.8788	5.5985	5.5985	1.91	.6	.5817	4.1.1-3
5	1	M5	2-AA CS7X3.21-FF	- Aluminum														

The final results of the code checking are the code check values **UC Max** and **Shear UC**. These values represents a factored ratio of actual to allowable load for ASD based on the provisions of ADM Section 4. So, if this value is less than 1.0, the member passes. If it is greater than 1.0, the member fails. If the value is greater than 9.999 it will be listed as "9.999". The Shear Check is based on fs/Fs. Note that torsional shear, if any, is also included in this check. The location for the shear

check is followed by "y" or "z" to indicate the direction of the shear. The **Loc** field tells at what location the maximum code check occurs measured from the I-joint location of the member. See Plot Options – Members to learn how to view the code check results graphically.

The remaining columns provide some of the values used in the code check with the equation number itself given in the last column. The Member Detail Report gives more values used to perform the code check.

The final field lists the controlling equation for the code check. This will be one of the equations from Section 4.

For enveloped results the combination that produced the listed code and shear checks is given in the column "lc". The other values are the corresponding values and are not necessarily the maximums across all the combinations.

Note

- The program currently only performs an analysis of double sections and does not perform a code check of any kind.
- The Member Detail Report gives more values used to perform the code check.
- See Spreadsheet Operations to learn how to use **Find**, **Sort** and other options.
- See Plot Options – Members to learn how to plot member results.

Aluminum Detail Report

The aluminum detail report has design information for the specific code checks:

AA ADM1-05: ASD Code Check

Max Bending Check	1.152	Max Shear Check	0.000 (s)
Location	0 ft	Location	0 ft
Equation	4.1.1-2	Max Defl Ratio	L/143

		Slender. Limit S1	S2	Slender. Ratio S	Gov Eqn
Ft	19.4872 ksi				3.4.1-2
Fc	18.8394 ksi	0	65.7	10.7	3.4.7-2
Fbz	24.8182 ksi	28.6	80.9	11.3	3.4.12-1
Fby	24.8182 ksi	28.6	80.9	11.3	3.4.12-1
Fsz	12.2468 ksi	35.6	64.5	21.9	3.4.20-1
Fsy	12.2468 ksi	35.6	64.5	21.9	3.4.20-1

	Y-Y	Z-Z
Cm	1	.6
Lb	2 ft	2 ft
KL/r	10.6949	10.6949
Sway	No	No

L Comp Top	2 ft
L Comp Bot	2 ft
Cb	1

The **Max Bending Check** is based on ADM Section 4, with the governing **Equation** and **Location** listed.

The **Max Shear Check** is not provided in the ADM specification, this represents fs/Fs with the governing **Location** listed.

The **Max Defl Ratio** is based on the entire length of the member.

The **Slender Limit S1** and **S2** are calculated based on the **Gov Eqn**. The Slenderness Ratio are given based on the Design Aids in Tables 2-2 thru 2-26 ADM Section VII. The **Slender. Ratio** is also based on the **Gov Eq** and is listed below for all code checks:

Sec. 3.4.	**Slend Ratio**
7	kL/r
8	kL/r
9	b/t
10	$(R_b/t)^{1/2}$

11	L_b/r_y
12	$(R_b/t)^{1/2}$
13	$(d/t)(L_b/d)^{1/2}$
14	$(2L_bS_c)/((I_yJ)1/2)$
15	b/t
16	b/t
17	b/t
18	h/t
19	h/t
20	h/t
21	a_e/t

Assumptions and Limitations

For all shape types, it is assumed that the transverse load on the member is occurring through the member's shear center. This means secondary torsional moments that may occur if the load is not applied through the shear center are not considered.

- **r_{ye}**, effective radius of gyration from Eq 4.9.1-1 is used for doubly symmetric sections beams.
- **Welded** regions are not checked in RISA-3D. You can use the welded material properties for the entire member, or create segments that are welded material in order to check the weld properties. For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Welded Aluminum**.
- **Connections** are not checked, including web crippling, fatigue or stiffeners.
- **Lt** is assumed to be the smaller of **Lb**yy or **Lb**zz. Torsional warping effects are not included. Torsion stiffness and stress are calculated as pure torsion only.
- **Kt** in is assumed to be the smaller of **Kyy** or **Kzz**.
- **Double Sections** - The program currently only performs an analysis of double sections and does not perform a code check of any kind.

Special Messages

Aluminum Code Check Not Calculated

This message is displayed when the member is not defined with a database shape, is defined as a double section, or an Aluminum code is not specified on the **Global Parameters**, or no units were specified.

Boundary Conditions

Boundary Conditions define how the model is externally constrained. All models must be attached to some external point or points of support. You may define these points of support as completely restrained or as partially restrained with a **Spring**. You can also define a spring support that has stiffness in only one direction with tension-only or compression-only springs.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Boundary Conditions**.

Creating and Modifying

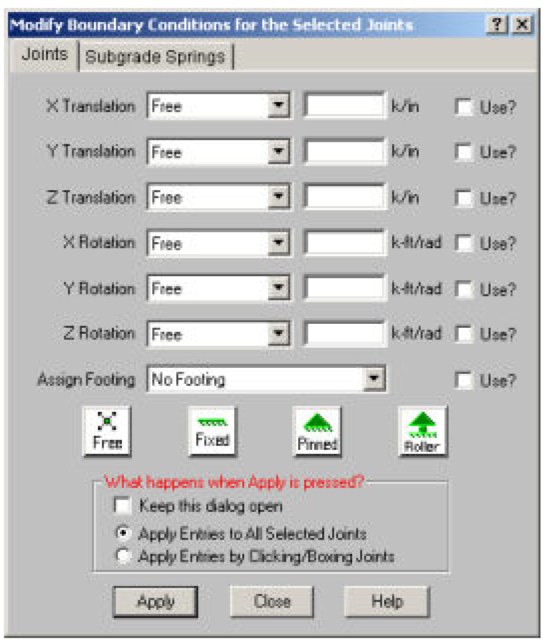
There are a number of ways to create or modify boundary conditions. You may view and edit the data in the **Boundary Conditions Spreadsheet**, you may double-click a joint to view and edit its properties, or you can use the **Modify Boundaries** tool to graphically assign or modify a possibly large selection of boundary conditions.

Modify Boundary Conditions for Joints

The graphical **Modify Boundary** tool discussed here lets you specify and modify boundary conditions graphically. To use this, you will typically specify the new boundary condition, then select the joints that you want to assign or modify.

You can modify or assign joints one at a time by selecting the **Apply by Clicking/Boxing** option and then click on the joints you wish to modify. You may also modify or assign entire selections of joints by selecting the joints first and then use the **Apply to All Selected** option.

The parameters shown are the same as those on the **Boundary Conditions Spreadsheet** and are described in Boundary Condition Options. Use the arrow buttons to select the boundary condition.



The **Use?** check boxes next to the data fields indicate whether the particular parameter will be used or not when the modification is applied. If the box next to a field is checked, that parameter will be applied to any selected joints If the box is NOT checked, the parameter will NOT be applied, even if a value is entered in the field. This lets you easily change one or two parameters on joints without affecting all the rest of the parameters.

To Apply Boundary Conditions

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Click the button and define the boundary condition. Check the **Use?** Box for the items to apply.
3. You may apply the boundary condition by choosing joints on the fly or apply it to a selection of joints. To choose joints on the fly, choose **Apply Entries by Clicking/Boxing Joints** and click **Apply**. Click/Box the joints with the left mouse button.

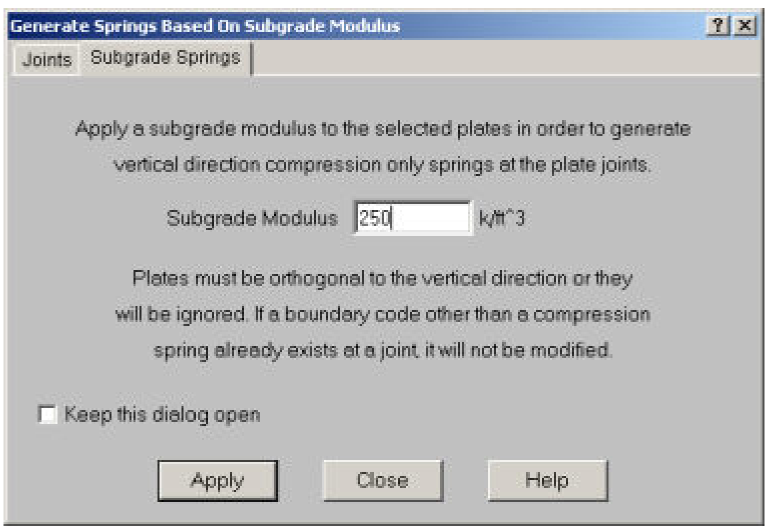
   To apply the boundary condition to a selection of joints, choose **Apply Entries to All Selected Joints** and click **Apply**.

Note

- To apply more boundaries with different conditions, press CTRL-D to recall the **Boundary Conditions Dialog**.
- You may also view and edit boundary conditions by double-clicking on a joint.
- You may also specify or edit boundary conditions in the **Boundary Conditions Spreadsheet**.
- To assign a Footing, you must have RISAFoot installed on the computer and you must first create a footing group in the **Footing** spreadsheet.
- You may undo any mistakes by clicking the **Undo** button.

Generate Soil Springs

A subgrade modulus may be automatically applied to horizontal plates in a model using the **Subgrade Springs** tool. The tool will generate compression-only springs in the vertical direction at all plate joints in the selected group of plates. Joints must be connected to plates which are perpendicular to the vertical direction or the joints will be ignored. If a boundary code other than a compression-only spring already exists in the vertical direction at that joint, the boundary code will NOT be modified.



RISA-3D calculates the tributary area for each plate joint individually and multiplies that area by the subgrade modulus to determine the spring stiffness for the compression-only spring at that joint. The boundary codes for all plate joints affected are automatically modified in the **Boundary Conditions Spreadsheet**.

Boundary Conditions Spreadsheet

The **Boundary Conditions Spreadsheet** records the boundaries for the joints and may be accessed by selecting **Boundary Conditions** on the **Spreadsheets Menu**.

Joint Boundary Conditions

	Joint Label	X [k/in]	Y [k/in]	Z [k/in]	X Rot [k-ft/r...	Y Rot [k-ft/r...	Z Rot [k-ft/r...	Footing
1	N1	Reaction	Reaction	Reaction				
2	N2	Reaction	Reaction	Reaction	Reaction	Reaction	Reaction	
3	N3	Reaction	Reaction	Reaction	Reaction	Reaction	Reaction	
4	N4	Reaction	Reaction	Reaction				
5	N5		S1000					
6	N6		S1000					
7	N7	Story 1						
8	N8			Story 1				
9	N21			SlaveN22				
10	N24			SlaveN23				

The **Joint Label** column contains the label of the joint that is restrained. The last column may contain the label for the spread footing that is assigned to the joint. The **Footing** column is only available if a current version of RISAFoot is also loaded on the computer.

The remaining columns record the boundary conditions that apply to the joint. There are six degrees of freedom for each joint (3 translation, 3 rotation), so there are six columns for degrees of freedom. The last column records the footing group, if any, applied to the joint. Footings may only be used if RISAFoot is also loaded on the computer. The boundary conditions are entered in these remaining columns by selecting the cell, clicking and choosing from the boundary options. You may also type them in directly.

Boundary Condition Options

Free joints have no restraint in any of the degrees of freedom and need not be listed on the **Boundary Conditions Spreadsheet**. The following are the valid boundary condition options that may be used for the six degrees of freedom.

Note

- Models that contain compression-only or tension-only springs must be iterated until the solution converges. Convergence is achieved when no more load reversals are detected in the springs. During the iteration process, each spring is checked, and if any springs are turned off (or back on), the stiffness matrix is rebuilt and model is resolved. This can take quite a bit longer than a regular static solution.
- You can enter the first letter of the option ("R" for Reaction, "S" for Spring, etc.) rather than typing out the entire code. RISA-3D fills in the rest automatically. The exceptions are the SLAVE and STORY entries, where the full word does have to be entered (since "S" denotes a spring).

Boundary Condition at ALL Joints

The entry "ALL" may be entered in the **Joint Label** field. The boundary conditions entered on this line will be applied to ALL the joints not otherwise listed. This is useful if you should want to lock certain directions of movement for all or most of the joints. For example, if you are solving a 2D frame defined in the XY plane and you're only interested in the planar action, you could enter "ALL" and put an "F" (for Fixed) for Z translation, X Rotation and Y Rotation. See the following figure:

Joint Boundary Conditions

	Joint Label	X [k/in]	Y [k/in]	Z [k/in]	X Rot [k-ft/r...	Y Rot [k-ft/r...	Z Rot [k-ft/r...
1	ALL			Fixed	Fixed	Fixed	
2	N1	Reaction	Reaction	Fixed	Fixed	Fixed	
3	N2	Reaction		Fixed	Fixed	Fixed	

Note

- If a joint is explicitly listed with boundary conditions, those boundary conditions override the "ALL" conditions for all 6 directions. The "ALL" specified boundary codes apply only to those joints NOT otherwise listed on the **Boundary Conditions Spreadsheet**. This is why joints 1 and 2 in the figure above also have the Fixed code in the Z translation, 2x Rotation and 2y Rotation fields.

Reaction Boundary Condition

The "R" code, for **Reaction**, specifies full restraint for the indicated direction. No movement will be allowed in the indicated direction for this joint. Furthermore, the reaction will be calculated at this joint, for this direction.

Fixed Boundary Condition

The "F" code, for **Fixed**, specifies full restraint for the joint in the indicated direction. The difference between "Fixed" and "Reaction" is that for the "Fixed" code, no reaction is calculated. The "Fixed" condition actually removes the degree of freedom from the solution, which is why the reaction value is not available. If you aren't interested in the reaction value, using the "Fixed" code will result in a slightly smaller model and less output.

Spring Boundary Condition

The "Snnn" code, for **Spring**, models a spring attached to the joint in the indicated direction. The "nnn" portion of the code is the numerical magnitude of the springs' stiffness. The units for the spring stiffness depend upon whether the spring is translational or rotational. The appropriate units are shown at the top of the column.

For example, if a spring of stiffness 1000 Kips per Inch were desired in the X direction at a particular joint, for that joint you would enter 'S1000' for the X direction boundary condition.

Compression-Only Springs

The "CSnnn" code, for **Compression-Only Springs**, models a one way "compression-only" spring attached to the joint in the indicated direction. This spring has stiffness for negative displacements and NO stiffness for positive displacements. The "nnn" portion of the code is the numerical magnitude of the springs' stiffness. The spring stiffness units are the same as those for a normal spring. Compression-only springs are useful as soil springs when analyzing foundations that may have uplift.

For example, if a compression-only (CS) spring with a stiffness of 500k/in were desired in the Y direction at a certain joint, you would enter 'CS500' for the Y direction boundary condition.

This means that all displacements at this joint in the negative Y direction will be resisted with a stiffness of 500k/in. However the joint is free to move in the positive Y direction.

- When a model contains T/C only springs, the program must iterate the solution until it converges. Convergence is achieved when no more load reversals are detected in the T/C only springs. During the iteration process, each T/C only boundary condition is checked. If any springs are turned off (or turned back on), the stiffness matrix is rebuilt and model is resolved. For models with lots of T/C only elements, this can take a bit longer than a regular static solution.

Tension-Only Springs

The "TSnnn" code, for **Tension-Only Springs**, models a one way "tension-only" spring attached to the joint in the indicated direction. This spring has stiffness for positive displacements and NO stiffness for negative displacements. The "nnn" portion of the code is the numerical magnitude of the springs' stiffness. The spring stiffness units are the same as for a normal spring.

For example, if a tension-only (TS) spring with a stiffness of 500k/in. were desired in the Y direction at a certain joint, you would enter 'TS500' for the Y direction boundary condition.

This means that all displacements at this joint in the positive Y direction will be resisted with a stiffness of 500k/in. However the joint is free to move in the negative Y direction.

- When a model contains T/C only springs, the program must iterate the solution until it converges. Convergence is achieved when no more load reversals are detected in the T/C only springs. During the iteration process, each T/C only boundary condition is checked. If any springs are turned off (or turned back on), the stiffness matrix is rebuilt and model is resolved. For models with lots of T/C only elements, this can take a bit longer than a regular static solution.

Slaved Joints

You may slave any or all of the joint degrees of freedom to another joint. See Slaving Joints for more information.

Story Drift Joints

- The **Boundary** spreadsheet is also used to record joints to be used for story drift calculation. For example, to indicate that a particular joint is to represent the fourth story level for X direction drift, you would enter “STORY 4” for the X direction boundary condition for the joint. These STORY entries may only be made in the translation degrees of freedom. See Drift for more information.

Footings at Boundary Conditions

If the current version of RISAFoot has been installed on your computer, then you can automatically integrate the Footing design directly into your RISA-3D results. For more information on this procedure, refer to the Footings Design section.

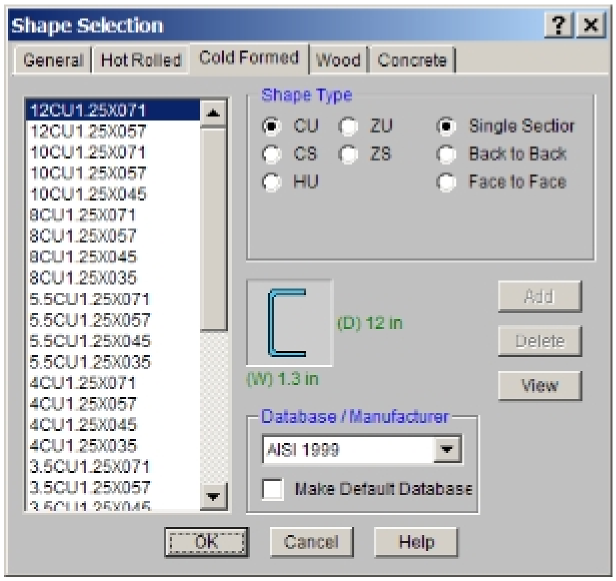
Boundary Conditions at Wall Panels

If the edge of a wall panel is to be viewed as continuously pinned or fixed, then the boundary condition for that wall must be set in the wall panel editor. Situations can arise where there is a difference between the wall panel edge boundarycondition and the boundary condition defined at a joint along that edge. In these situations the joint boundary condition will always govern for that joint. However, the rest of the edge will be based on the wall panel's boundary conditions.

Cold Formed Steel - Databases

Shapes are organized in the database by manufacturer. Common shapes are supported such as C sections with and without lips, Z sections with and without lips, and Hat sections without lips. Each of these shape types may be used as single section, a back to back section, or a face to face section. You may type in the names directly, select shapes from these databases or add your own shapes.

RISA currently supports the following common Cold Formed steel databases: AISI 2007, Dale-Incor, Dietrich, Marino-Ware, and SSMA.



To Select a Cold Formed Database Shape

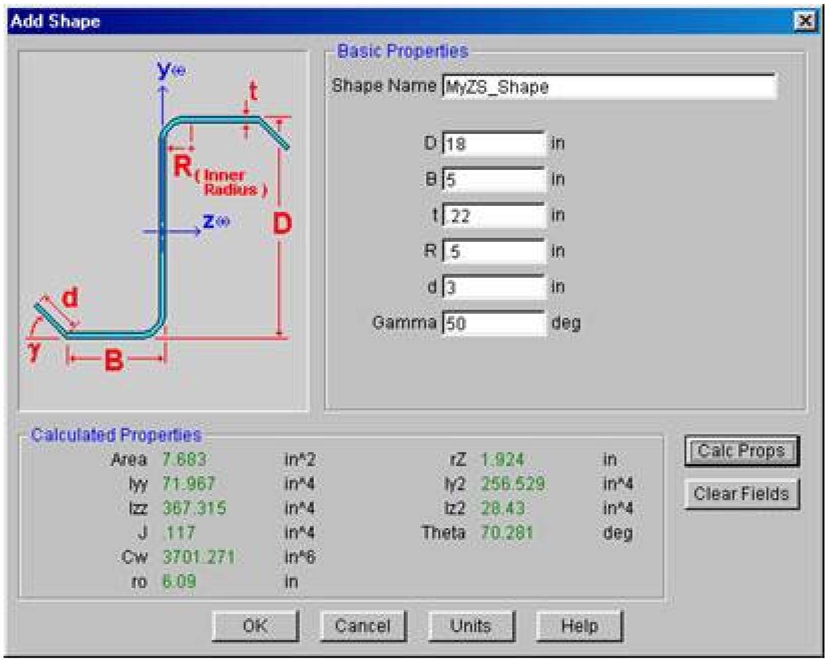
1. From the Cold Formed **Section Sets** tab on the **Spreadsheet**, or the **Primary** tab of the **Members Spreadsheet**, move the cursor to the **Shape** field and click .
2. Specify the database and shape type you wish to use and then select from the list of available shapes by clicking on .

Custom vs. Manufacturer Shapes

You can enter your own cold formed shapes as well as use those provided in the manufacturer database. When the cold formed database type is selected, you'll notice a "Manufacturer" list box that appears in the Shape Selection dialog. You can specify a manufacturer or choose “Custom” to select, add or edit your own custom shapes. New shape properties are calculated using the linear method described in Part I of the AISI code.

Database Files

The cold formed manufacturer shape databases are stored in the file aisidb32.fil, and the custom cold formed shapes are stored in the file aisicust.fil.



To Add a Database Shape

1. On the **RISA Toolbar** click the **Edit Shape Database** button.
2. Select the cold formed tab and set the Manufacturer type to "Custom", then select the shape type you wish to add and click the **Add** button.
3. Specify a name for the shape and fill in the **Basic Properties.**
4. Click **Calc Props** to determine the shape properties.

Note

- Alterations to the shape database are not permanent unless you agree to save them. Changes that are not saved only remain valid for the current session and will not be present the next time you start RISA.
- New shapes are added to the bottom of the database.
- To delete a shape, specify the database and shape type you wish to delete and then click the **Delete** button.
- To edit a shape, click the **Edit** button and edit the shape properties. Values can only be manually edited here, nothing will be recalculated. If you wish to have all the values for a shape recalculated, you will need to delete the shape and then add it again with the new properties. Manufacturer shapes cannot be edited, only custom shapes can be edited.

Cold Formed Shape Types

There are five types of shapes. Names for each shape type follow the convention of the manufacturer for each shape. If you know the shape name, you can type the name directly into the **Shape** field on the spreadsheets. Alternately you may click the button to look up a shape and select it.

C sections without lips (CU)

For the AISI database, CU shapes are called out by the designation given them in the AISI steel manual. For example, if you wanted a 12" deep unstiffened C section, you'd call it out as 12CU1.25x071. The '12' is the depth, the CU specifies a C section without lips, the '1.25' is the flange width, and the '071' is the decimal thickness. Other manufacturer databases generally follow similar conventions.

C sections with lips (CS)

For the AISI database, CS shapes are called out by the designation given them in the AISI steel manual. Other manufacturer databases generally follow similar conventions.

Z sections without lips (ZU)

For the AISI database, ZU shapes are called out by the designation given them in the AISI steel manual. Other manufacturer databases generally follow similar conventions.

Z sections with lips (ZS)

For the AISI database, ZS shapes are called out by the designation given them in the AISI steel manual. Other manufacturer databases generally follow similar conventions.

Hat sections without lips (HU)

For the AISI database, HU shapes are called out by the designation given them in the AISI steel manual. Other manufacturer databases generally follow similar conventions.

Double Sections

For each of the five shape types the selected shape may be used as a standard single section or as a double section. The choices for double sections are 'Back to Back' and 'Face to Face'. A typical double section is designated with a "2-" preceding the shape name and a "-BB" (Back to Back) or "-FF" (Face to Face) following the shape name. For example, a "2-12CU1.25x071-FF" section represents two 12" deep C sections with 1.25" wide flanges and a 0.071" thickness placed face to face.

Note

- The program currently only performs an analysis of double sections and does not perform a code check of any kind. This may be added into a future revision of the program.

Cold Formed Steel - Design

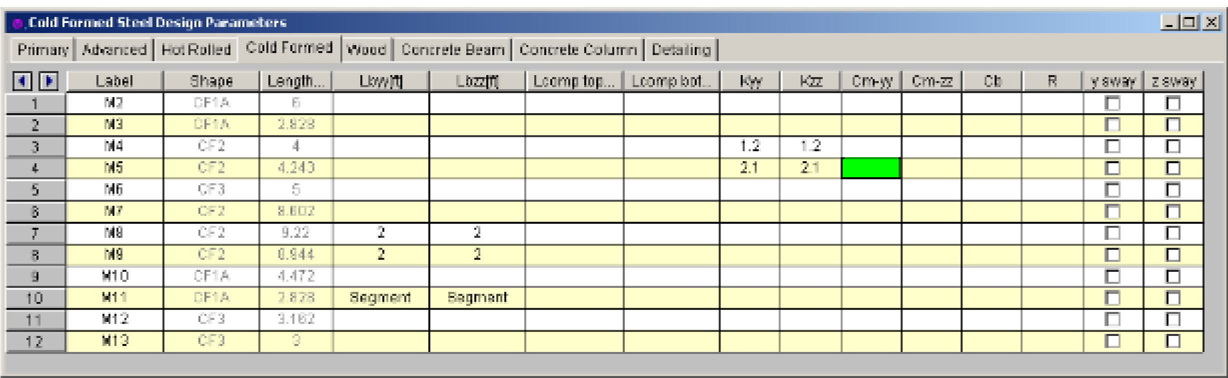
Full code checking can be performed on standard cold formed steel shapes, based on the following codes:

- The 1996 edition of the AISI code with 1999 Supplement (AISI-99 ASD and LRFD)
- The 2001 edition of the AISI code (AISI NAS-2001 ASD and LRFD)
- The 2001 edition of the Mexican code (CANACERO-2001 ASD and LRFD)
- The 2001 edition of the Canadian code (CSA S136-01 LSD)
- The 2004 Supplement of the AISI code (AISI NAS-2004 ASD and LRFD)
- The 2004 Supplement of the Mexican code (CANACERO-2004 ASD and LRFD)
- The 2004 edition of the Canadian code (CSA S136-04 LSD)
- The 2007 edition of the AISI code (AISI NAS-2007 ASD and LRFD) including Supplement No.1 (August 09)
- The 2007 Supplement of the Mexican code (CANACERO-2004 ASD and LRFD) including Supplement No.1 (August 09)
- The 2007 edition of the Canadian code (CSA S136-07 LSD)

Cold formed shape properties are available in the database and the values are based on the AISI or manufacturer values, whichever is selected (See Cold Formed Steel Database). You may also input your own basic shapes and the properties will be calculated automatically.

Design Parameters

The **Cold Formed** tab on the **Members Spreadsheet** records the design parameters for the cold formed steel code checks. These parameters may also be assigned graphically. See Modifying Member Design Parameters to learn how to do this.



	Label	Shape	Length...	Lbyy[ft]	Lbzz[ft]	Lcomp top...	Lcomp bot...	Kyy	Kzz	Cm-yy	Cm-zz	Cb	R	y sway	z sway
1	M2	CF1A	6											☐	☐
2	M3	CF1A	2.828											☐	☐
3	M4	CF2	4					1.2	1.2					☐	☐
4	M5	CF2	4.243					2.1	2.1					☐	☐
5	M6	CF3	5											☐	☐
6	M7	CF2	8.602											☐	☐
7	M8	CF2	9.22	2	2									☐	☐
8	M9	CF2	0.944	2	2									☐	☐
9	M10	CF1A	4.472											☐	☐
10	M11	CF1A	2.828	Segment	Segment									☐	☐
11	M12	CF3	3.162											☐	☐
12	M13	CF3	3											☐	☐

These parameters are defined for each cold formed member. The entries are explained below.

Label

You may assign a unique **Label** to all of the members. Each label must be unique, so if you try to enter the same label more than once you will get an error message. You may relabel at any time with the **Relabel** options on the **Tools** menu.

Shape

The member **Shape** or Section Set is reported in the second column. This value is listed for reference only and may not be edited as it is dictated by the entry in the Section/Shape column on the **Primary** tab.

Length

The member **Length** is reported in the third column. This value may not be edited as it is dependent on the member end coordinates listed on the **Primary Data** tab. It is listed here as a reference for unbraced lengths which are discussed in the next section.

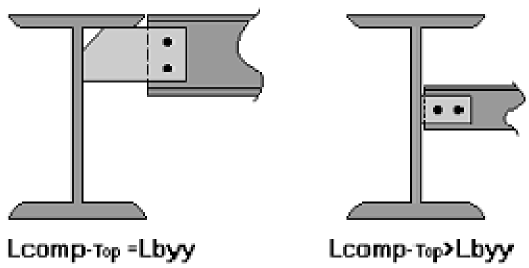
Unbraced Length

You may specify unbraced lengths or have RISA-3D calculate them for you. The unbraced lengths are **Lbyy**, **Lbzz**, **Lcomp-top** and **Lcomp-bot**.

The **Lb** values, **Lb**yy and **Lb**zz, represent the unbraced length for the member with respect to column type buckling about the member's local y and z axes, respectively. These **Lb** values are used to calculate KL/r ratios for both directions, which in turn impact the calculation of the axial strength, Pn. The KL/r ratios gauge the vulnerability of the member to buckling. Refer to Section C4 in Part V of the AISI code for more information on this. Also, Section C4 lists some limiting values for KL/r. These limiting values are NOT enforced by the program.

The **Lcomp** values, **Lcomp-top** and **Lcomp-bot**, are the unbraced lengths of the compression flanges for flange buckling due to flexure. These may be the same as the **Lb**yy value, but not necessarily. The **Lcomp** values are used in the calculation of bending strength, Mn. Refer to Section C3 in Part V of the AISI code for more information on this. In particular, Lcomp is used in equation C3.1.2.1.-8 as shown in Supplement 1 to the 1999 or the 2001 codes.

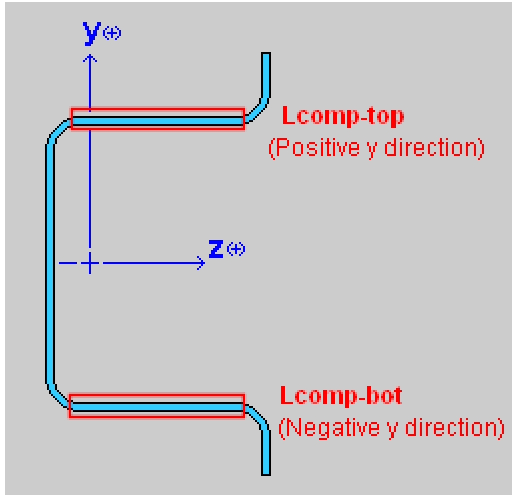
For continuous beams the moment will reverse such that the top and bottom flanges will be in compression for different portions of the beam span. **Lcomp-top** is the unbraced length of the top flange and **Lcomp-bot** is the unbraced length of the bottom flange.



If left blank these unbraced lengths all default to the member's full length. The exception to this is if **Lb**yy is entered and **Lcomp-top** is left blank, **Lcomp-top** will default to the entered value for **Lb**yy The unbraced torsional length “Lt” is always taken as the smaller of Lbyy or Lcomp-top. A future enhancement will be the addition of this parameter for the rare case when you’ll need to make it longer or shorter than the weak axis unbraced length.

Note:

- For Hat Channel (HU) shape types, the **Lcomp-top** and **Lcomp-bot** values only apply to the flanges perpendicular to the local y axis (please see image below). Therefore, if your loading is applied in the local z direction, these entries will not apply. This assumption was made in reference to section *C3.1.2.1 of the AISI Specification with Commentary* whose footnote tells us that the limit state of Lateral-Torsional Buckling does not apply to these shapes.



For physical members, you can enter the code "**Segment**" in the unbraced length fields and the length of each segment will be used. A "segment" is the distance between the joints that are on the physical member. For example, suppose you have a physical member that is 20 feet in length, and there are two joints along the physical member, one 5 feet from the end and one at 15 feet. An unbraced length of 5 feet will be used for the first segment, then a value of 10 feet will be used in the middle segment, and again a value of 5 feet would be used in the last segment.

Note

- If the intermediate framing members are considered to brace the bottom flange, then you can enter "segment" for Lcomp-bot. When the "segment" command is used ALL intermediate points along the beam are viewed as brace points. Therefore, you may have to delete unused or extraneous points.
- The Top Flange is defined as the flange corresponding to the positive local y axis for the member. For more information on setting local axes refer to the Members section.
- The calculated unbraced lengths are listed on the **Member Detail** report.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Unbraced Lengths**.

K Factors (Effective Length Factors)

The **K Factors** are also referred to as effective length factors. **Kyy** is for column type buckling about the member's local y-y axis and **Kzz** is for buckling about the local z-z axis.

If a value is entered for a **K Factor**, that value will be used for the entire length of the physical member. If an entry is not made (left blank), the value will internally default to '1' for that member. See the AISI code commentary for Section C4 for an explanation of how to calculate K Factors.

RISA-3D is able to approximate the K values for a member based on the member's sway condition and end release configuration. The K-factor approximation is based on Table C-C4.1, found on page VI-71 of the AISI code. The following table gives the values used for various conditions.

Table Case	End Conditions	Sidesway?	K-Value
(a)	Fixed-Fixed	No	.65
(b)	Fixed-Pinned	No	.80
(c)	Fixed-Fixed	Yes	1.2
(d)	Pinned-Pinned	No	1.0
(e)	Fixed-Free	Yes	2.1

(f)	Pinned-Fixed	Yes	2.0

RISA-3D will recognize a pinned boundary condition for the K approximation for a full pin, i.e. if all the rotations in the boundary condition are released. If any of the rotations in a boundary condition are restrained, the boundary condition is considered “fixed” for the K approximation.

Any configuration not described here will be given the default value of 1.0.

If any value that influences these K values is changed, the K approximation should be redone. For instance, if you have RISA-3D approximate K values then change some end release designations, you should redo the K approximations.

Remember that the K-values are *approximations,* and you should check to make sure you agree with all K-values RISA-3D assigns. You can always override a K-value after an approximation by directly entering the value that you want in the appropriate field. Keep in mind that a subsequent approximation will overwrite any manually input values so you will need to override the approximation each time it is performed.

Limitation:

RISA-3D will currently neglect the influence of adjoining framing members when those members are connected at a joint that also has degrees of freedom restrained by boundary conditions. For example, suppose a column and beam member connect at a joint that is restrained for translation in all directions (i.e. the joint is "pinned"). The K factor approximation will neglect the beam member when it calculates the K factor for the column and visa-versa. The effect will be that the ends of the members at that joint will be seen as "pinned" and not "fixed" for the K factor approximation.

Sway Flags

The **Sway Flags** indicate whether the member is to be considered subject to sidesway for bending about its local y and z axes. The **y sway** field is for y-y axis bending and the **z sway** field is for z-z axis bending. Click on the field to check the box and indicate that the member is subject to sway for that particular direction, or leave the entry blank if the member is braced against sway. These sway flags influence the calculation of the K factors as well as the Cm and Cb factors.

Cm - Interactive Bending Coefficient

Cm Coefficients are described in Section C5 of the AISI code. If these entries are left blank, they will be automatically calculated.

The **Cm** value is influenced by the sway condition of the member and is dependent on the member's end moments, which will change from one load combination to the next, so it may be a good idea to leave these entries blank.

Cb - Bending Coefficients

For the cold formed codes, **Cb Coefficients** are used in the calculation of the nominal flexural strength, Mn. If this entry is left blank, it will be calculated automatically.

R Value

The **R Value** for cold formed steel design is described in section C3.1.3 of the AISI code and is used to calculate the moment capacity of beams that have one flange fastened to deck or sheathing. This value only applies to C or Z members and can vary from 0.4 to 0.7 based on the depth of the member (See table C3.1.3-1 in the AISI Supplement for the actual values).

If a value is entered by the user, that value will be used by the program in the moment capacity calculation of the member. There are a number of restrictions that must be met to use this section of the code for moment capacity and the user is responsible to check that these restrictions are satisfied.

Note

- If the R value is entered, the program will use section C3.1.3 when performing moment capacity calculations and will ignore the standard LTB checks from section 3.1.2.

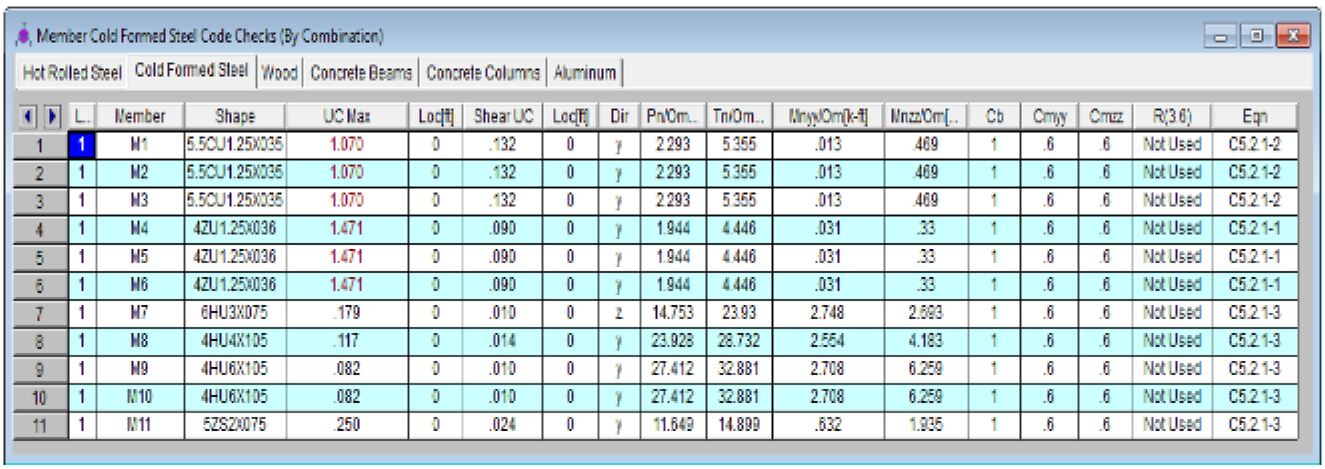
Phi Factors

The following table provides a list of safety factors (ASD) and resistance factors (LRFD and LSD) being used for different codes.

Code	Ft	Fc	Fb	Fv	Wt	Wc	Wb	Wv
AISI ASD 99					1.67	1.8	1.67	1.5/1.67
AISI LRFD 99	0.95	0.85	0.95/0.9	1.0/0.9				
AISI/Canacero ASD 01					1.67	1.8	1.67	1.6
AISI/Canacero LRFD 01	0.9	0.85	0.95/0.9	0.95				
CSA S136-01 LSD	0.9	0.8	0.9	0.8				
AISI/Canacero ASD 04					1.67	1.8	1.67	1.6
AISI/Canacero LRFD 04	0.9	0.85	0.95/0.9	0.95				
CSA S136-04 LSD	0.9	0.8	0.9	0.8				
AISI/Canacero ASD 07					1.67	1.8	1.67	1.6
AISI/Canacero LRFD 07	0.95	0.85	0.9	0.95				
CSA S136-07 LSD	0.9	0.8	0.9	0.8				

AISI Steel Code Check Results

Access the **Code Check** spreadsheet by selecting the **Results** menu and then selecting **Members** ▸ **Design Results** or by clicking on the **Design Results** button on the Results toolbar.


Member Cold Formed Steel Code Checks (By Combination)

Hot Rolled Steel | Cold Formed Steel | Wood | Concrete Beams | Concrete Columns | Aluminum

	L...	Member	Shape	UC Max	Loc[ft]	Shear UC	Loc[ft]	Dir	Pn/Om...	Tn/Om...	Mnyy/Om[k-ft]	Mnzz/Om[...	Cb	Cmyy	Cmzz	R(3.6)	Eqn
1	1	M1	5.5CU1.25X035	1.070	0	.132	0	y	2.293	5.355	.013	.469	1	.6	.6	Not Used	C5.2.1-2
2	1	M2	5.5CU1.25X035	1.070	0	.132	0	y	2.293	5.355	.013	.469	1	.6	.6	Not Used	C5.2.1-2
3	1	M3	5.5CU1.25X035	1.070	0	.132	0	y	2.293	5.355	.013	.469	1	.6	.6	Not Used	C5.2.1-2
4	1	M4	4ZU1.25X036	1.471	0	.090	0	y	1.944	4.446	.031	.33	1	.6	.6	Not Used	C5.2.1-1
5	1	M5	4ZU1.25X036	1.471	0	.090	0	y	1.944	4.446	.031	.33	1	.6	.6	Not Used	C5.2.1-1
6	1	M6	4ZU1.25X036	1.471	0	.090	0	y	1.944	4.446	.031	.33	1	.6	.6	Not Used	C5.2.1-1
7	1	M7	6HU3X075	.179	0	.010	0	z	14.753	23.93	2.748	2.693	1	.6	.6	Not Used	C5.2.1-3
8	1	M8	4HU4X105	.117	0	.014	0	y	23.928	28.732	2.554	4.183	1	.6	.6	Not Used	C5.2.1-3
9	1	M9	4HU6X105	.082	0	.010	0	y	27.412	32.881	2.708	6.259	1	.6	.6	Not Used	C5.2.1-3
10	1	M10	4HU6X105	.082	0	.010	0	y	27.412	32.881	2.708	6.259	1	.6	.6	Not Used	C5.2.1-3
11	1	M11	5ZS2X075	.250	0	.024	0	y	11.649	14.899	.632	1.935	1	.6	.6	Not Used	C5.2.1-3

The final results of the code checking are the code check values **UC Max** and **Shear UC**. These values represents a factored ratio of actual to allowable load for ASD or ultimate load to design strength for LRFD or LSD, based on the provisions of Section C5. Section 3.3.1 and 3.3.2 are also used to check combined bending and shear. So, if this value is less than 1.0, the member passes. If it is greater than 1.0, the member fails. If the value is greater than 9.999 it will be listed as "9.999". The Shear Check is based on section C3.2.1. Note that torsional shear, if any, is also included in this check. The location for the shear check is followed by "y" or "z" to indicate the direction of the shear. The **Loc** field tells at what location the maximum code check occurs measured from the I-joint location of the member. See Plot Options – Members to learn how to view the code check results graphically.

The remaining columns, discussed below, provide some of the values used in the code check with the equation number itself given in the last column. The Member Detail Report gives more values used to perform the code check.

For ASD code checking, Pn, Tn, and Mnyy and Mnzz are the member capacities calculated for the member. Pn is calculated according to the provisions of AISI 1999 / 2001, Section C4. Tn is based on Section C2. The Mn values are calculated based on Section C3. Note that for RISA-3D, "zz" corresponds to "xx" in the AISI code, i.e. RISA-3D substitutes Mnzz for Mnx, to maintain consistency with the member local axis system.

For LRFD or LSD, the factored compression Phi*Pn, factored tension, Phi*Tn, and factored moment strengths Phi*Mnyy and Phi*Mnzz values are displayed. For tension Tn, the value is fy * area, per Section C2. Compression Pn is calculated per Section C4. The Mn values are calculated per Section C3.

Cb is set to 1.0 if not specifically entered by the user, which is conservative. The Cm coefficients, described in Section C5 are also listed. These also are influenced by the sway flag settings.

The final field lists the controlling equation for the code check. This will be one of the equations from Section C5 or Section C3.3.

For enveloped results the combination that produced the listed code and shear checks is given in the column "lc". The other values are the corresponding values and are not necessarily the maximums across all the combinations.

Note

- The program currently only performs an analysis of double sections and does not perform a code check of any kind.
- The Member Detail Report gives more values used to perform the code check.
- See Spreadsheet Operations to learn how to use **Find**, **Sort** and other options.
- See Plot Options – Members to learn how to plot member results.

Assumptions and Limitations

For all shape types, it is assumed that the transverse load on the member is occurring through the member's shear center. This means secondary torsional moments that may occur if the load is not applied through the shear center are not considered.

Iterations for the effective section modulus (Se and Sc) are ended when a difference less than 1% is achieved in the neutral axis distance calculation with a maximum of 5 iterations. Holes in sections are not considered in the shear strength calculations or for effective width calculations. Deflections are based on the full section properties, not the effective section properties.

Lt is assumed to be the smaller of **Lb**yy or **Lcomp**. Torsional warping effects are not included. Torsion stiffness and stress are calculated as pure torsion only. Web crippling is not considered.

Kt in section C3.1.2.1 is assumed to be 1.0. All conditions listed for the use of C3.1.3-1 are assumed satisfied. Section C3.1.4 is not considered in the calculation of Mn. Effects of shear stiffeners for section C3.2.1 are not considered. Only strong axis bending and strong axis shear are considered for equation C3.3.1 (combined bending and shear).

Section C4.4 is not considered in the calculation of the axial strength at this time.

Z Shapes – The bracing length in Lbyy is assumed to brace the minor principal axis. Z sections in compression are assumed to buckle in Euler buckling about their weakest principal axis. The value of r_{min} is used rather than the geometric r_x and r_y values.

H Shapes – Hat sections in bending about the y-y axis such that the brims are in compression are assumed braced such that the brims cannot each fail in lateral torsional buckling independently.

Double Sections - The program currently only performs an analysis of double sections and does not perform a code check of any kind.

Slenderness Limitations - The w/t limits of Section B1.1 are enforced. However, the shear lag effects (section B1.1c) are not enforced.

Special Messages

AISI Code Check Not Calculated

This message is displayed when the member is not defined with a database shape, or is defined as a double section, or a steel code is not specified on the **Global Parameters**, or no units were specified.

Can't do code check, stiffener D/w > 0.8 (Sect. B4.2)

The ratio D/w exceeds the limiting criteria listed in Section B4.2 for simple lip stiffeners. ("D" and "w" are length of the stiffener and the flat length of the flange as defined in B4.2)

Stiffener angle gamma is < 40 or > 140 (Sect. B4.2)

The angle (gamma) for a simple lip stiffener must be greater or equal to 40 degrees or less than or equal to 140 degrees per the criteria in section B4.2. The angle gamma for this shape is outside this range.

Can't do code check, flange w/t > limit (Sect. B1.1)

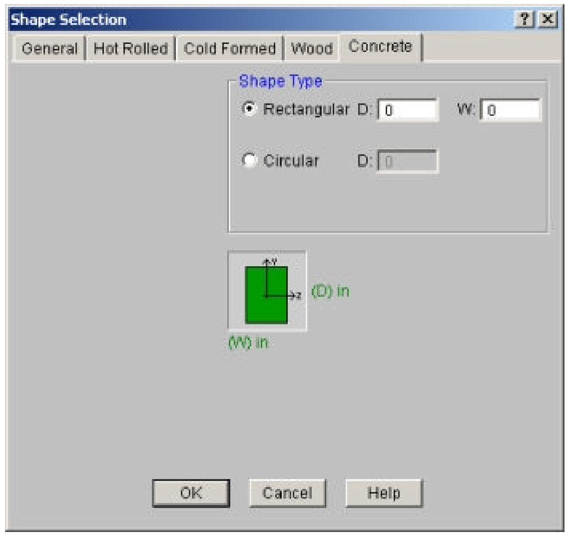
The ratio w/t exceeds the limiting criteria listed in section B1.1 for flanges. A value of 60 is used per the AISI code for unstiffened elements and elements stiffened with simple lips.

Can't do code check, web h/t > 200 (Sect. B1.2)

The ratio h/t exceeds the limiting criteria listed in section B1.2 for webs. The program currently considers all webs as unreinforced, so a value of 200 is used as the limit.

Concrete - Database

There are two types of shapes currently supported, Rectangular and Circular. If you're familiar with the shape definitions, you can type the name directly into the appropriate field. Alternately you may click the button to have the program generate the desired shape definition for you.



Rectangular Sections

Rectangular sections are defined using a parametric shape code since a rectangular shape may be any depth or width. The code is CRECT'depth'X'width', where 'depth' and 'width' are the values in the current dimension units. For example, if you wanted a beam that was 18" deep and 12" wide, you would enter "CRECT18X12". Note that the dimensions can also be decimal values like "18.25".

Circular Sections

Circular/Round sections are also defined using a parametric shape code since a round shape may have any diameter. The code is CRND'diameter', where 'diameter' is the value in the current dimension units. For example, if you wanted a column that was 14" in diameter, you would enter "CRND14". Note that the dimension can also be a decimal value like "14.5".

Rebar Layout Database

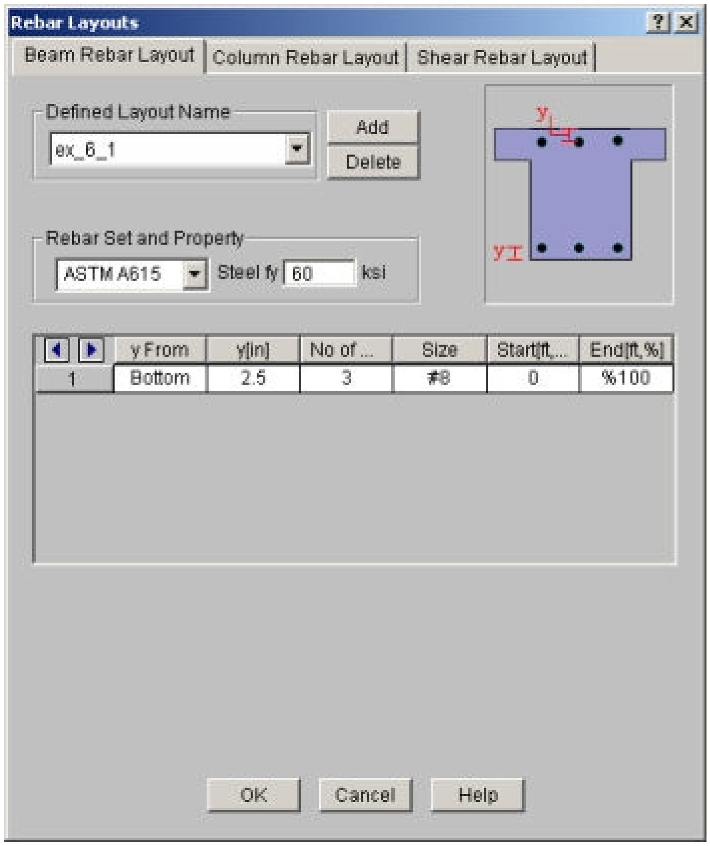
Pressing the button on the **RISA Toolbar** will open the database that is used for creating and storing custom rebar layouts. This allows the user to create multiple layers of bars and add in compression reinforcement or unusual bar arrangements.

These reinforcement layouts may be assigned to beam or column members in the same way as the other concrete design parameters are assigned. This can be done on the **Concrete** tabs of the **Members Spreadsheet**, from the **Design** tab of the **Member Information Dialog**, or from the **Modify Members Dialog**.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Custom Rebar Layouts**.

Beam Rebar Layouts

Since beams are only designed for uniaxial bending, the only requirements for the beam layouts are that you specify the depth at which the bars are located and the size & number of the bars that are present at that depth. You can specify the depth with respect to the top surface of the beam or the bottom surface.



The **Start** and **End** locations dictate the location along the length of the beam where these bars will be present. You can use these entries to specify partial length bars that will only be present in locations with a higher moment demand. If the bar should be present for the entire length of the beam, the start location should be '0' and the end location should be '%100' as shown in the dialog above.

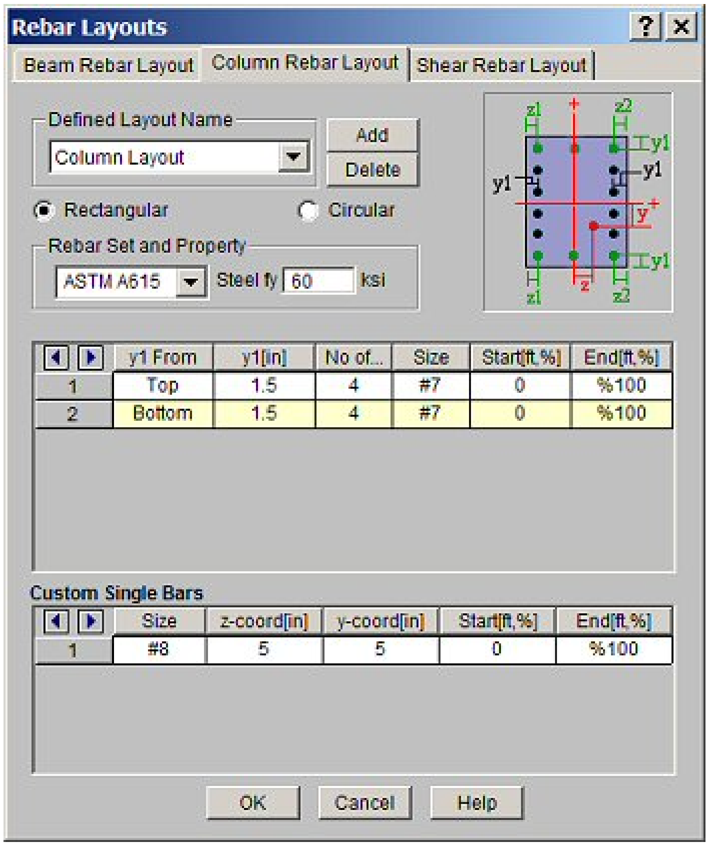
Note

- While the rebar layout sheet resembles one of RISA's spreadsheets in appearance it is NOT a spreadsheet and standard TAB controls will not work. Instead, the arrow keys or the new arrow buttons can be used to advance from cell to cell.

Rectangular Column Rebar Layouts

Since columns are designed for biaxial bending, they require more information about the location and arrangement of the bars.

Normally, column bars are arranged in **layers**. One 'top' and one 'bottom' horizontal layer must **always** be defined, each containing at least two bars. These layers, as well as any additional horizontal layers, will be specified by entering a **y1** value to specify the depth from the top or bottom fiber to the centerline of the reinforcing steel. The number and size of the bars must then be entered. The **z1** and **z2** values dictate where the first and last bar in that layer are located. Additional bars in that layer will be placed so that they are evenly spaced in that layer.



Vertical layers can be specified by entering a **y1** value specifying the depth from the right or left most fiber to the centerline of the reinforcing steel. The number and size of the bars must then be entered. The **z1** and **z2** values are ignored for vertical layers because the bars will be assumed to be evenly spaced between the required top and bottom layers referred to previously. If this is not desired, then the side bars should be entered individually as **custom single bars**.

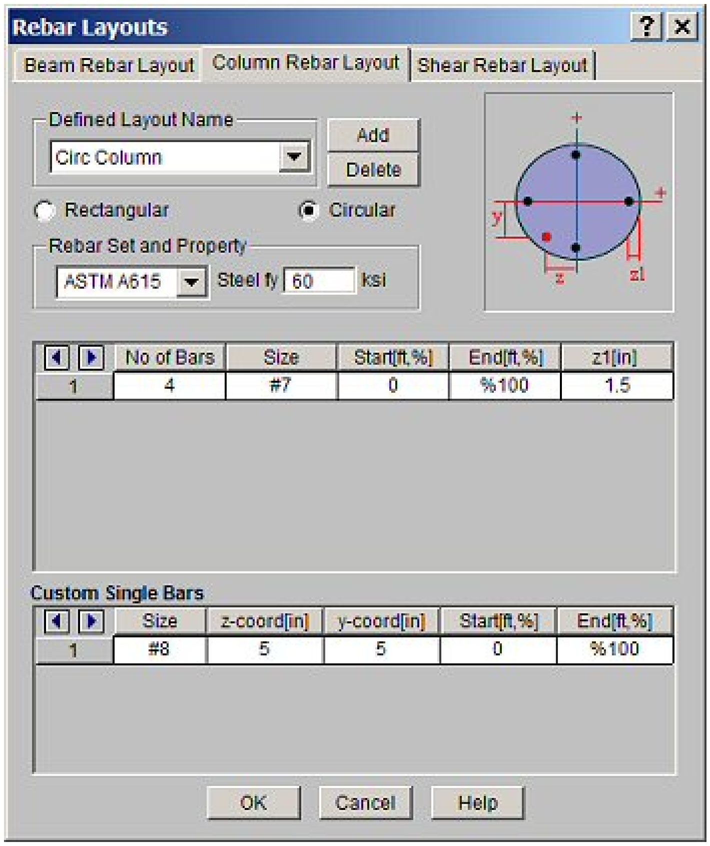
Custom single bars are specified by their y and z coordinates measured from the local y and z-axis respectively. A positive y coordinate would place the bar closer to the top fiber and a negative y coordinate would place the bar closer to the bottom fiber. Similarly, a positive z coordinate would place the bar closer to the right side and a negative z coordinate would place the bar closer to the left side.

The **Start** and **End** locations dictate the location along the length of the member where these bars will be present. You can use these entries to specify partial length bars that will only be present in locations with a higher moment demand. If the bar should be present for the entire length of the member, the start location should be '0' and the end location should be '*%100*' as shown in the dialog above.

Circular Column Rebar Layouts

For circular columns, you may specify equally spaced concentric *rings* of bars at given depths, **z1**, measured from the exterior fiber of the column. You may also specify custom single bars.

Custom single bars are specified by their y and z coordinates measured from the local y and z axis respectively. A positive y coordinate would place the bar closer to the top fiber and a negative y coordinate would place the bar closer to the bottom fiber. Similarly, a positive z coordinate would place the bar closer to the right side and a negative z coordinate would place the bar closer to the left side.



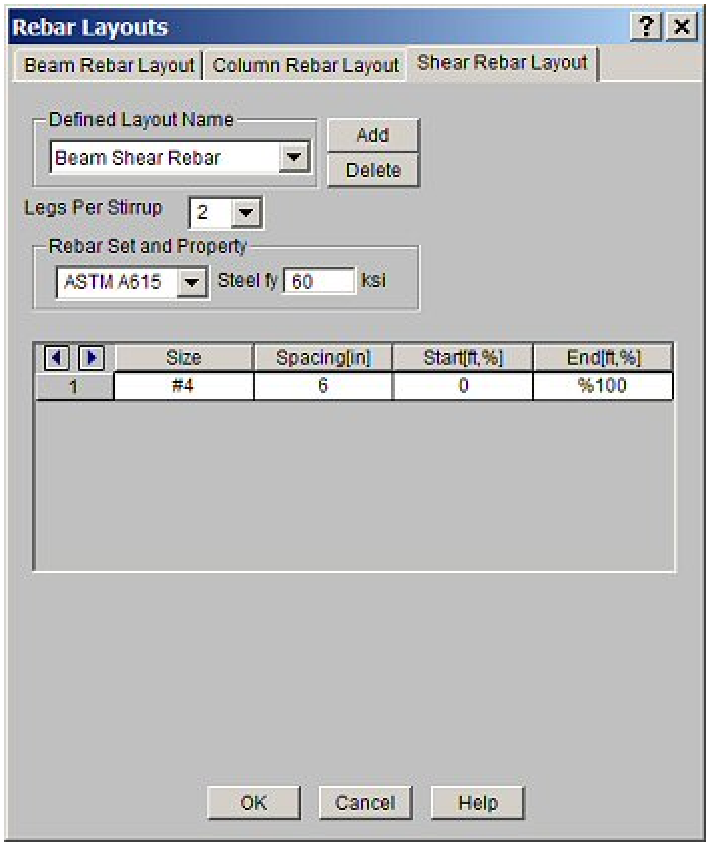
The **Start** and **End** locations dictate the location along the length of the member where these bars will be present. You can use these entries to specify partial length bars that will only be present in locations with a higher moment demand. If the bar should be present for the entire length of the member, the start location should be '0' and the end location should be '*%100*' as shown in the dialog above.

Note

- While the rebar layout sheet resembles one of RISA's spreadsheets in appearance it is NOT a spreadsheet and standard TAB controls will not work. Instead, the arrow keys or the new arrow buttons can be used to advance from cell to cell.

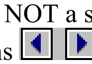
Shear Rebar Layouts

These rebar layouts may be assigned to either columns or beams. They are specified by giving the size and spacing of the bars as well as the location on the member where that reinforcement will be present.



The **Start** and **End** locations dictate the location along the length of the member where these bars will be present. You can use these entries to specify partial length reinforcement that will only be present in locations with a higher shear demand. If the reinforcement should be present for the entire length of the member, the start location should be '0' and the end location should be '%100' as shown in the dialog above.

Note

- While the rebar layout sheet resembles one of RISA's spreadsheets in appearance it is NOT a spreadsheet and standard TAB controls will not work. Instead, the arrow keys or the new arrow buttons  can be used to advance from cell to cell.

Concrete - Design

Concrete design and optimization can be performed for standard concrete shapes based on the following codes:

- The 1999, 2002, 2005, 2008, and 2011 Editions of ACI 318
- The 1997 Edition of the British code (BS 8110)
- The 1992 EuroCode (EC2) and the British publication of the 2004 Eurocode (BSEN)
- The 1994 and 2004 Editions of the Canadian code (CSA A23.3)
- The 2000 Edition of the Indian code (IS 456)
- The 2001 Edition of the Australian code (AS 3600)
- The 1995 Edition of the New Zealand code (NZS 3101)
- The 2004 Edition of the Mexican code (NTC-DF)
- The 2007 Edition of the Saudi Building Code (SBC 304)

Note:

- Unless otherwise specified, all code references below are to ACI 318-11.

The program will design the longitudinal and shear reinforcement for rectangular beams and rectangular or circular columns. These calculations encompass all the code requirements except those noted in the Limitations section of this document. The program also provides reinforcement detailing information for concrete beams and interaction diagrams for concrete columns in the member detail reports.

To Apply a Concrete Design Code

1. On the **Code** tab of **Global Parameters Dialog**, select the concrete code from the drop down list.
2. Click **Apply** or **OK**.

Concrete Spans

RISA-3D will automatically break a concrete physical member into spans based on the number of internal supports. Each internal joint is NOT automatically treated as a support. Instead, we go through the whole model geometry to determine where a beam or column is supported. Note that for a physical member to see a support, there must be a joint at that support point. If a physical column and a physical beam cross each other without a joint at their intersection, then no support / span will be detected and they will not be connected.

Beam member types are supported by the following: Vertical Boundary Conditions (Fixed, Reaction), Column Members, Near Vertical Plate Elements, and other Beam Members that are supporting that member.

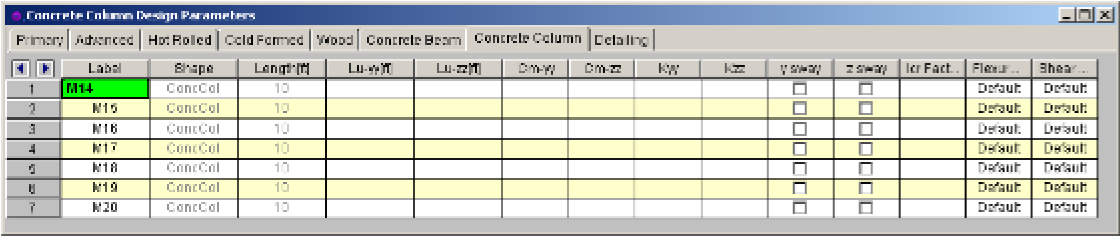
Column member types are supported by the following: Horizontal Boundary Conditions (Fixed, Reaction, Spring), Beam Members, Near Horizontal Plate Elements, and Rigid Diaphragms.

Note

- The quickest way to create new joints at beam / column intersections is to run a Model Merge.
- The program's ability to recognize spans is important because it will give you more relevant span to span information without overwhelming you with independent design results for each finite element segment that comprises your physical member.
- For continuous beam members, the program will evaluate the framing to determine which beams elements are supporting other beam elements so that only supporting members are treated as supports and not visa versa.
- Currently, members of type HBrace, VBrace, and None do not affect the span distances. Nor do any arbitrary joints within each span along a member.

Concrete Design Parameters - Columns

The Concrete Column tab on the **Members Spreadsheet** records the design parameters for the code checks of concrete columns. These parameters may also be assigned graphically. See Modifying Member Design Parameters to learn how to do this.



	Label	Shape	Length[ft]	Lu-yy[ft]	Lu-zz[ft]	Cm-yy	Cm-zz	Kyy	Kzz	y sway	z sway	Icr Fact...	Flexur...	Shear...
1	M14	ConcCol	10							☐	☐		Default	Default
2	M15	ConcCol	10							☐	☐		Default	Default
3	M16	ConcCol	10							☐	☐		Default	Default
4	M17	ConcCol	10							☐	☐		Default	Default
5	M18	ConcCol	10							☐	☐		Default	Default
6	M19	ConcCol	10							☐	☐		Default	Default
7	M20	ConcCol	10							☐	☐		Default	Default

The following parameters can be defined for each concrete column.

Label

You may assign a unique **Label** to all of the members. Each label must be unique, so if you try to enter the same label more than once you will get an error message. You may relabel at any time with the **Relabel** options on the **Tools** menu.

Shape

The member **Shape** or Section Set is reported in the second column. This value is listed for reference only and may not be edited as it is dictated by the entry in the Section/Shape column on the **Primary** tab.

Length

The member **Length** is reported in the third column. This value may not be edited as it is dependent on the member end coordinates listed on the **Primary Data** tab. It is listed here as a reference for unbraced lengths which are discussed in the next section.

Unbraced Length

You may specify unbraced lengths or have RISA-3D calculate them for you. The unbraced lengths are **Lu-yy** and **Lu-zz**.

The **Lu** values, **Lu-yy** and **Lu-zz**, represent the unbraced length of column members with respect to column type buckling about the member's local y and z axes, respectively. These **Lu** values are used to check the column for Euler buckling, and for the Moment Magnification Procedure in older editions of the ACI code.

If left blank these unbraced lengths all default to the member's full length.

For physical members, you can enter the code “**Segment**” in the unbraced length fields and the length of each segment will be used. A “segment” is the distance between the joints that are on the physical member. For example, suppose you have a physical member that is 20 feet in length, and there are two joints along the physical member, one 5 feet from the end and one at 15 feet. An unbraced length of 5 feet will be used for the first segment, then a value of 10 feet will be used in the middle segment, and again a value of 5 feet would be used in the last segment.

Note

- When the "segment" code is used, ALL joints on a column will be considered to brace the column for that type of buckling, even if a joint is associated with a member that would actually only brace the column against buckling in the other local axis. Therefore, the "segment" code should only be used for columns that are truly braced in that direction at each interior joint.
- The calculated unbraced lengths are listed on the **Member Detail** report.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Unbraced Lengths**.

K Factors (Effective Length Factors)

The **K Factors** are also referred to as effective length factors. **Kyy** is for column type buckling about the member's local y-y axis and **Kzz** is for buckling about the local z-z axis.

If a value is entered for a **K Factor**, that value will be used for the entire length of the physical member. If an entry is not made (left blank), the value will internally default to '1' for that member. See Section R10.10.1 of the ACI commentary for an explanation of how to calculate K Factors.

RISA-3D is able to approximate the K-values for a column based on the member's sway condition and end release configuration. The K-factor approximation is based on the idealized tables given in the AISC steel specification. The following table gives the values used for various conditions.

Table Case	End Conditions	Sidesway?	K-Value
(a)	Fixed-Fixed	No	.65
(b)	Fixed-Pinned	No	.80
(c)	Fixed-Fixed	Yes	1.2
(d)	Pinned-Pinned	No	1.0
(e)	Fixed-Free	Yes	2.1
(f)	Pinned-Fixed	Yes	2.0

Note

- This is an approximation of K-values and is NOT based on the Jackson and Moreland Alignment Charts presented in Section R10.10.1 of the ACI commentary.

RISA-3D will recognize a pinned boundary condition for the K approximation for a full pin, i.e. if all the rotations in the boundary condition are released. If any of the rotations in a boundary condition are restrained, the boundary condition is considered “fixed” for the K approximation.

Any configuration not described here will be given the default value of 1.0.

If any value that influences these K values is changed, the K approximation should be redone. For instance, if you have RISA-3D approximate K-values then change some end release designations, you should redo the K approximations.

Remember that the K-values are *approximations,* and you should check to make sure you agree with all K-values RISA-3D assigns. You can always override a K-value after an approximation by directly entering the value that you want in the appropriate field. Keep in mind that a subsequent approximation will overwrite any manually input values so you will need to override the approximation each time it is performed.

Limitation:

RISA-3D will currently neglect the influence of adjoining framing members when those members are connected at a joint that also has degrees of freedom restrained by boundary conditions. For example, suppose a column and beam member connect at a joint that is restrained for translation in all directions (I.e. the joint is “pinned”). The K factor approximation will neglect the beam member when it calculates the K factor for the column and visa-versa. The effect will be that the ends of the members at that joint will be seen as “pinned” and not “fixed” for the K-factor approximation.

Sway Flags

The **Sway Flags** indicate whether the member is to be considered subject to sidesway for bending about its local y and z axes. The **y sway** field is for y-y axis bending and the **z sway** field is for z-z axis bending. Click on the field to check the box and indicate that the member is subject to sway for that particular direction, or leave the entry blank if the member is braced against sway. These sway flags influence the calculation of the K Factors as well as the Cm.

Cm – Equivalent Moment Correction Factor

The **Cm Coefficients** are used to check the column for Euler buckling, and for the Moment Magnification Procedure in older editions of the ACI code. **Cm-yy** is for bending about the columns's local y-y axis and **Cm-zz** is for bending about the local z-z axis. If these entries are left blank they will be automatically calculated.

In the ACI design code, the Cm values are only applicable for non-sway frames. Therefore, this value will be ignored unless the corresponding sway flag is checked.

Flexural and Shear Rebar Layout

The user may choose to manually create the reinforcement layout for the column. This must be done if the user wishes to take advantage of bundled bars, multiple layers of reinforcement, or an unequal number of bars per face. See the section on the Concrete Database and Rebar Layouts for more information. If 'Default' is specified, then the program will design for an equal number of bars in each face of the rectangular column and may vary that reinforcing based on ACI minimums, maximums and the moment and shear demand at each section along the span.

Icr Factors (Cracked Moment of Inertia Factors)

The **Icr Factor** is used to reduce the bending stiffness of concrete columns per section 10.10.4 of the ACI code. If this entry is left blank, default values of 0.35 for beams and 0.70 for columns will be used.

Note

- The **Icr Factor** will be ignored if the **"Use Cracked Stiffness"** box is not checked on the **Concrete** tab of the **Global Parameters** dialog.
- The alternative calculations in ACI eqn's 10-8 and 10-9 are not considered.
- The sustained load reduction of ACI 318 Section 10.10.4.2 is not considered.

Service Level Stiffness

Due to cracking and material non-linearity, modeling the stiffness of concrete members is more complex than it is for steel or wood members.

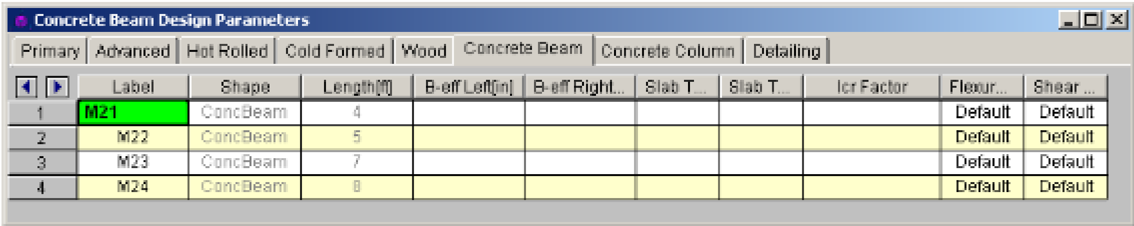
For typical applications, ACI section 10.10.4 requires that member stiffness be reduced to account for the cracking that occurs when a member is subjected to ultimate level loads. As described in the previous section, RISA uses the **Icr Factor** to account for this stiffness reduction. However, for service level analysis, the level of cracking will be significantly less. Therefore, the stiffness used in your analysis should be representative of the reduced loading and reduced cracking. Per the ACI commentary (R10.10.4.1), the program will account for this increased stiffness by applying a factor of 1.43 to the cracked section properties for any load combination that has the **"Service Load"** flag checked on the **Design** tab of the **Load Combinations Spreadsheet**.

Note

- When the **"Use Cracked Stiffness"** box is not checked on the **Concrete** tab of the **Global Parameters** settings, the program will use the un-cracked section for both service level and ultimate level member stiffness.

Concrete Design Parameters - Beams

The **Concrete Beam** tab on the **Members Spreadsheet** records the design parameters for the code checks of concrete beams. These parameters may also be assigned graphically. See Modifying Member Design Parameters to learn how to do this.



The following parameters can be defined for each concrete member.

Label

You may assign a unique **Label** to all of the members. Each label must be unique, so if you try to enter the same label more than once you will get an error message. You may relabel at any time with the **Relabel** options on the **Tools** menu.

Shape

The member **Shape** or Section Set is reported in the second column. This value is listed for reference only and may not be edited as it is dictated by the entry in the Section/Shape column on the **Primary** tab.

Length

The member **Length** is reported in the third column. This value may not be edited as it is dependent on the member end coordinates listed on the **Primary Data** tab. It is listed here as a reference only.

Effective Widths (B-eff Left and B-eff Right)

B-eff Left and **B-eff Right** are the effective widths of the slab for T-beam and L-beam design. See the section on T-beam & L-beam Sections below for more information on Effective widths.

Flexural and Shear Rebar Layout

The user may choose to manually create the reinforcement layout for the beam. This must be done if the user wishes to take advantage of compression steel, or multiple layers of reinforcement. See Concrete Database - Rebar Layouts for more information. If *Optimize* is specified, then the program will design for one layer of reinforcing and may vary that reinforcing based on ACI minimums, maximums, and the moment and shear demand at each section along the span. If you define your own rebar layout, and compression reinforcement is defined, then the program will consider the compression reinforcement in the analysis.

Icr Factors (Cracked Moment of Inertia Factors)

The **Icr Factor** is used to reduce the bending stiffness of concrete beams per section 10.10.4 of the ACI code. If this entry is left blank, default values of 0.35 for beams and 0.70 for columns will be used.

Note

- The **Icr Factor** will be ignored if the **"Use Cracked Stiffness"** box is not checked on the **Concrete** tab of the **Global Parameters** dialog.

Service Level Stiffness

Due to cracking and material non-linearity, modeling the stiffness of concrete members is more complex than it is for steel or wood members.

For typical applications, ACI section 10.10.4 requires that member stiffness be reduced to account for the cracking that occurs when a member is subjected to ultimate level loads. As described in the previous section, RISA uses the **Icr Factor** to account for this stiffness reduction. However, for service level analysis, the level of cracking will be significantly less.

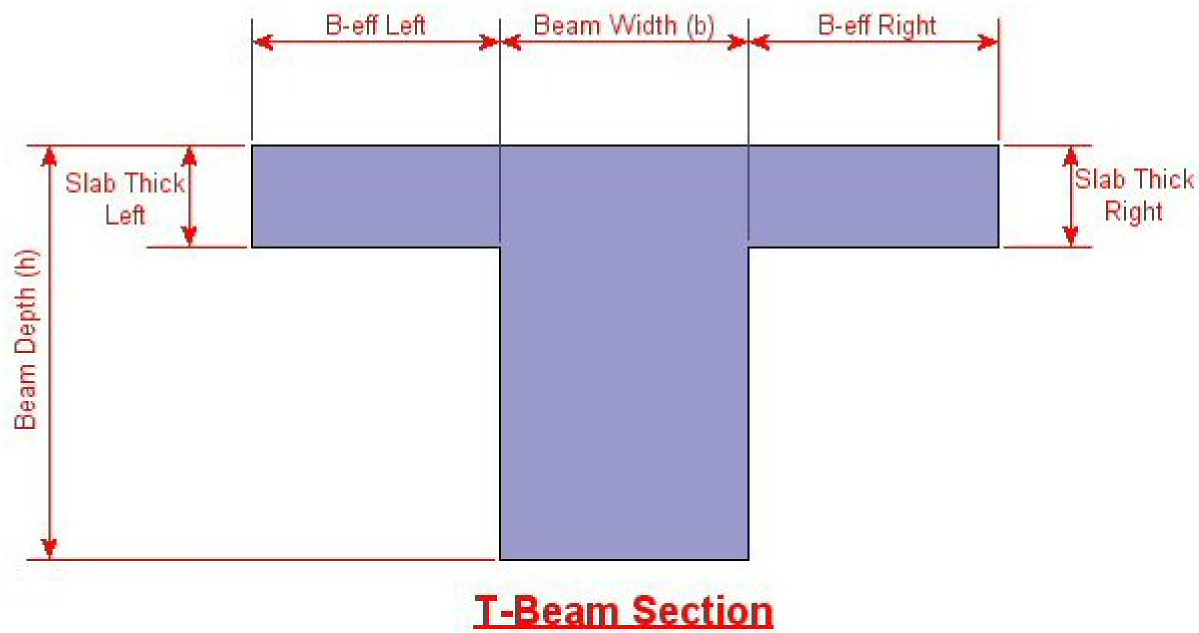
Therefore, the stiffness used in your analysis should be representative of the reduced loading and reduced cracking. Per the ACI commentary (R10.10.4.1), the program will account for this increased stiffness by applying a factor of 1.43 to the cracked section properties for any load combination that has the **"Service Load"** flag checked on the **Design** tab of the **Load Combinations Spreadsheet**.

Note

- When the **"Use Cracked Stiffness"** box is not checked on the **Concrete** tab of the **Global Parameters** settings, the program will use the un-cracked section for both service level and ultimate level member stiffness.

T-beam & L-beam Sections

T-beams and **L-beams** may be specified by assigning effective slab widths and slab thicknesses for the left and right side of the beam on the **Concrete Beam** tab of the **Members Spreadsheet**. These modifications may also be made graphically via the **Modify Properties** tab of the **Draw Members** tool.



RISA-3D will automatically trim the effective slab widths, **B-eff Left** and **B-eff Right**, to the maximum values indicated in sections 8.12.2(a) and 8.12.3(a) & (b) of ACI 318 if the value entered by the user is greater than that allowed by the code. It should be noted that RISA-3D does not check sections 8.12.2(b) and 8.12.3(c) of ACI 318 because no adjacent framing checks are performed.

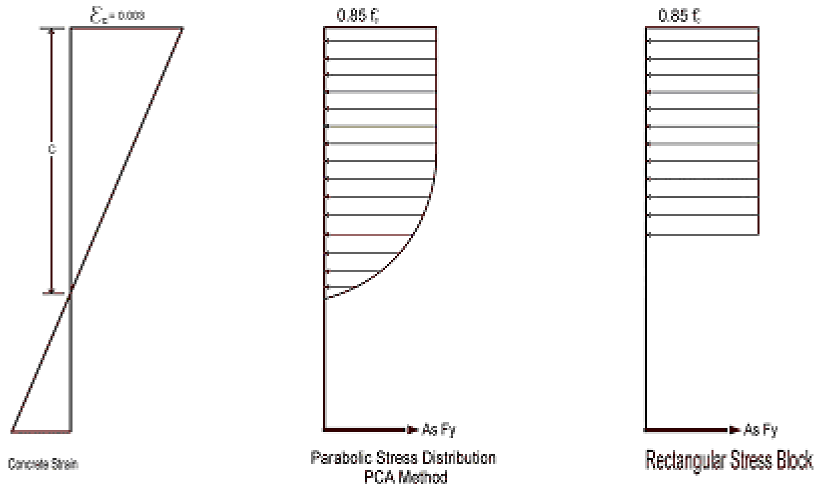
If the values of either **B-eff Left** or **B-eff Right** are left blank, a value of zero will be assumed, indicating no additional slab width beyond 1/2 the beam width on that side.

Note

- B-eff Right corresponds to the positive local z-axis of the beam. Subsequently, B-eff Left corresponds to the negative local z-axis.

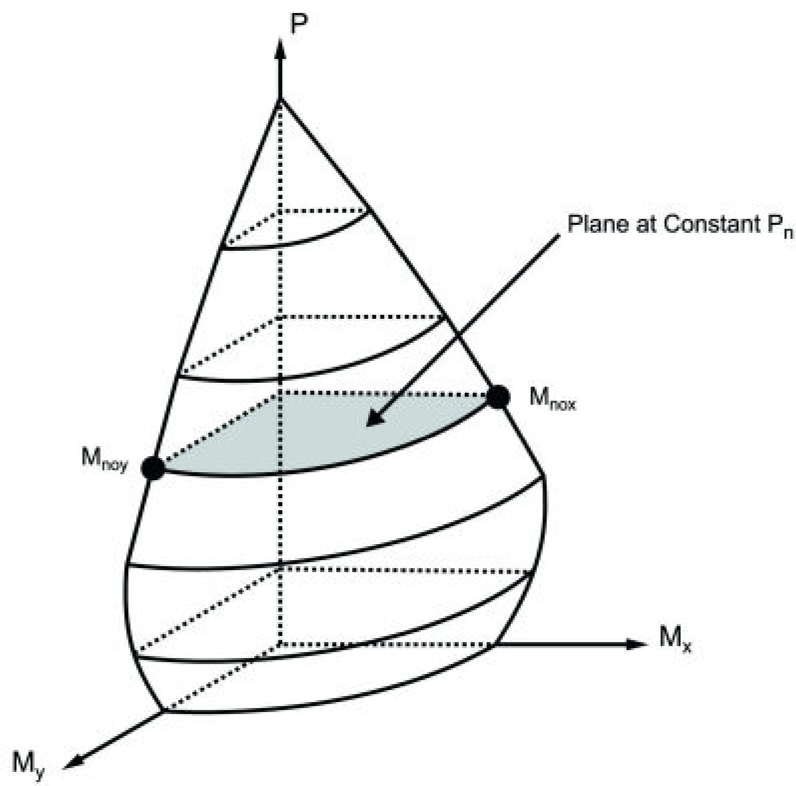
Parabolic vs. Rectangular Stress Blocks

You can specify whether you want your concrete design to be performed with a rectangular stress block, or with a more accurate parabolic stress block. While most hand calculations are performed using a rectangular stress block, the parabolic stress block is more accurate. In fact, most of the PCA design aids are based upon the parabolic stress distribution. A good reference on the parabolic stress block is the PCA Notes on ACI 318-99.



Biaxial Bending of Columns

You can specify whether you want your column design to be performed by using **Exact Integration**, or by using the **PCA Load Contour Method**. While most hand calculations are performed using the Load Contour Method, this method is merely an approximation based on the uniaxial failure conditions and the Parme Beta factor. In contrast, the Exact Integration method uses the true biaxial strain state to design the member. A good reference on the Load Contour Method is chapter 12 of the PCA Notes on ACI 318-99.



Biaxial Failure Surface
PCA Load Contour Method

Limitations - General

Torsion – Beams and columnsignore torsion with respect to the design of shear reinforcement. A message is shown in the detail report to remind you of this. You can turn the warning messages off on the **Concrete** tab of the **Global Parameters Dialog**. However, when using the 2002 and newer ACI 318 code the program does check the torsion on the member against the Threshold Torsion value (see ACI 318-11 Section 11.5.1). A warning is produced, implying that the shear reinforcement will have to be designed by the engineer for torsion.

Beam Design – Beams are not designed for weak axis y-y bending, weak axis shear, or axial forces. A message is shown in the detail report to remind you of this. You can turn the warning messages off on the **Concrete** tab of the **Global Parameters Dialog**. Beams currently do not consider any compression steel in the calculation of the moment capacity. Beam "skin reinforcement" per the requirements of ACI 10.6.7 for beams with "d" greater than 36" is currently not specified by the program. The provisions in ACI 10.7 for deep beams are not considered.

Column Design – Columns with biaxial moment and no axial load will currently be designed using the **PCA Load Contour Method** even if Exact Integration is selected on the Global Parameters dialog. This is shown on the detail report.

Limitations - ACI

Shear Design – The shear strength of the concrete alone is limited to the standard 2*sqrt (f'c) equation from ACI 318 section 11.2.1.1 and does not use the more detailed calculations of section 11.2.2. Also note that we use provision 11.2.1.3 which states "For members subject to significant axial tension, shear reinforcement shall be designed to carry total shear unless a more detailed analysis is made using 11.2.2.3." The program does not use this more detailed analysis.

Deep Beam Design – The program does not design deep beams as defined in ACI 318 Section 10.7.

Threshold Torsion - The program does not adjust the threshold torsion value for the presence of axial force in a member.

Limitations - Canadian Code

Concrete Stress Profile – Concrete stress strain curve (parabolic) is assumed same as PCA method for the Canadian codes.

Bi-Axial Bending - The program uses the simplified uniaxial solution provided in the Canadian specification rather than performing a complete biaxial condition.

Mid-Depth Flexural Strain for Shear Design - The program uses the code equation (per the General Method) to calculate $\varepsilon_x$with the moment and shear at the section taken from the envelope diagrams. The maximum ε_x for each span is conservatively assumed for the entire span. Currently the program has no option for pre-stressing, so V_p and A_p are both taken as zero.

Shear Design - The shear strength of concrete is calculated using β and θ, which are both calculated per the General Method (Clause 11.3.6.4 from the *2004 CSA A23.3*). S_{ze} is calculated per equation 11-10 and a_g is always assumed to equal 20 mm (maximum aggregate size).

Limitations – Aus / NZ Codes

Concrete Stress Profile – Concrete stress strain curve (parabolic) is assumed same as ACI for the New Zealand and Australian codes.

Cracked Sections – Icracked is only considered for US and Canadian codes. Icracked for the Australian and New Zealand codes is ignored and the program uses the full gross properties.

Neutral Axis Parameter – Ku in AS code is always assumed to be less than 0.4.

Rebar Spacing – NZS and AS codes: max spacing of rebar (beam) is 300 mm and minimum spacing is one bar diameter or 25mm whichever is bigger.

Shear Strength in Beams – In AS code, when calculating the shear strength of a beam β2, β3 are always assumed to be unity. This is always conservative for beams will little axial load, or beams in compression. But, may be unconservative for members subjected to significant net tension.

Bi-Axial Bending – The New Zealand code does not appear to give a simplified method for solving biaxial column design. Therefore, the PCA load contour method is being used instead.

Shear Tie Spacing – Column/beam shear tie spacing is based on (a) and (c) of NZS 9.3.5.4 :1995.

Development Length – Development length in NZS is based on NZS 7.3.7.2 where αa is conservative assumed to be 1.3 (top bars) for all cases. For the AS code, it is assumed that K1=1 and K2=2.4 in clause 13.1.2.1 of AS 3600:2001.

Slender Column Calculations – EI is assumed to be equal to 0.25EcIg (with βd =0.6) in slender column calculations in AS and NZS codes (like in ACI).

Limitations - British

Concrete Stress Profile – Concrete stress strain curve (parabolic) is taken from the British specification.

Cracked Sections – Icracked is only considered for US and Canadian codes. Icracked for the British code is ignored and the program uses the full gross properties.

Bi-Axial Bending – The program uses the simplified uniaxial solution provided in the British specification rather than performing a complete biaxial condition.

Limitations - Euro

Concrete Stress Profile – Concrete stress strain curve (parabolic) is taken from the EuroCode specification.

Cracked Sections – Icracked is only considered for US and Canadian codes. Icracked for the EuroCode is ignored and the program uses the full gross properties.

Bi-Axial Bending – The program uses the simplified uniaxial solution provided in the EuroCode rather than performing a complete biaxial condition.

Limitations - Indian

Concrete Stress Profile – Concrete stress strain curve (parabolic) is taken from the Indian specification.

Cracked Sections – Icracked is only considered for US and Canadian codes. Icracked for the Indian code is ignored and the program uses the full gross properties.

Bi-Axial Bending – The program uses the simplified uniaxial solution provided in the Indian specification rather than performing a complete biaxial condition.

Limitations - Saudi Code

Concrete Stress Profile – Concrete stress strain curve (parabolic) is assumed to be the same as the ACI code.

Shear Strength– The shear strength is based on 11.3.1.1 and does not include the more detailed provisions of section 11.3.1.2.

Yield Strength of Shear Ties - The yield strength of shear ties is not allowed to exceed 420MPa.

Shear Tie Spacing - Minimum spacing of shear ties is set to 50mm

Bi-Axial Bending – Both the Exact Integration and the PCA Load Contour methods for bi-axial bending are supported in the Saudi code.

Special Messages

In some instances code checks are not performed for a particular member. A message is usually shown in the **Warning Log** and **Detail Report** explaining why the code check was not done. There are also instances where a code check is performed, but the results may be suspect as a provision of the design code was violated. In these cases, results are provided so that they can be examined to find the cause of the problem. Following are the messages that may be seen.

No Load Combinations for Concrete Design have been run.

None of the load combinations that were run had the **Concrete Design** box checked on the **Design** tab of the **Load Combinations Spreadsheet**. Since there are no concrete design specific load combinations, there are no results or force diagrams to show.

Warning: No design for spans with less than 5 sections.

Certain very short spans in physical members can end up with less than 5 design sections. No design is attempted without at least 5 sections because maximum values may be missed and an un-conservative design may result.

Warning: No design for spans less than 1 ft.

Certain very short spans in physical members can end up with lengths less than 1 foot. No design is attempted for these sections.

Warning: Member is slender and can sway, but P-Delta Analysis was NOT run.

Under older ACI codes slender sway members need to be run with the P-Delta option turned on to account for secondary forces and moments. In some situations, a preliminary design without P-Delta is useful and so a design is performed and this warning is shown to remind you to run the final analysis including P-Delta effects. Alternately, if you're using the redesign feature, the next suggested column may resolve this issue if it's not slender.

Warning: Slender Compression Failure (Pu > .75Pc). No Slender calculations done.

Since RISA-3D allows you to specify a starting column size, it's possible that for slender columns under substantial axial load you'll exceed the critical buckling load used in the slenderness equations in ACI 10.12.3. Design results are still shown so the suggested shapes can be used to pick a new suggested column size that will not have this problem. Note that the design results shown are NOT valued because the slender moment effects have NOT been considered.

Warning: KL/r > 100 for this compression member. See ACI99 10.10.1

Members that violate the KL/r limit still have design results calculated and shown. If you're using the redesign feature, the next suggested shape should resolve this problem.

Warning: Exact Integration selected but PCA method used

This message is shown when you've requested the **Exact Integration** option on the **Global Parameters Dialog**, but we weren't able to converge a solution for the column in question. When Exact Integration does not converge, the **PCA Method** is used instead to give an idea of the demand vs. the capacity.

Warning: PCA Method Failed. Axial Load > Axial Capacity.

One of the limitations of the **PCA Method** is that it requires the column being checked to have a greater axial capacity than the axial demand. Since RISA-3D allows you to set a starting size, it's possible that the demand may be greater than the capacity. In this case a very rough estimate of the capacity is calculated by using the independent moment capacity about each axis considering the axial load. The resulting code check value is then based on the combined demand vector over the combined capacity vector and will always be greater than 1.0. The purpose of the results in this case is to show the column failed, not to give an accurate estimate of the over-demand. The redesign feature will suggest a larger shape to resolve this issue.

Warning: The shear tie spacing does not meet the code Minimum Requirement

This warning is stating that either minimum spacing or strength requirements are not being met for the shear reinforcement in the concrete member.

P-Delta analysis required for all ACI 318-08/11 Load Combinations

A second order analysis is required as of the 2008 edition of the ACI 318 code. A code check will only be given if P-Delta is turned on in the Load Combinations spreadsheet, or if this requirement is intentionally waived in the Preferences.

*Compression Pu exceeds 0.75*Pc (Euler buckling)*

As of the 2008 edition of the ACI 318 a loophole exists which allows the design of columns without considering the Euler buckling failure mode. The program will not give design results for columns which have more than 75% of Euler Buckling load, as is the intention (but not the outright statement) of ACI eqn 10-12.

Concrete - Design Results

You can access the **Concrete Results Spreadsheets** by selecting the **Results Menu** and then selecting **Members** ▸ **Design Results** or **Concrete Reinforcing**. Unlike wood and steel, concrete results are different for beams and columns so they each get their own results spreadsheet. Note also that concrete results are always based on envelope results, even if you've run a batch solution.

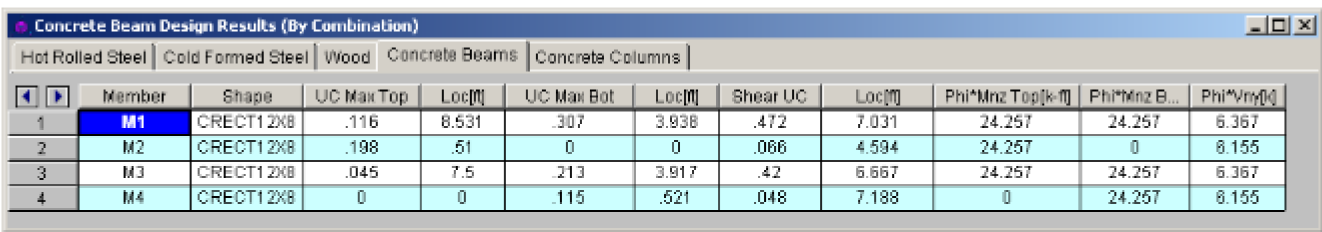
For beam flexural design, the required bars are based on the envelope moment diagrams. For column flexural design, the required bars for each load combination are calculated at various sections for the moments and axial forces at those sections. The required bars for all load combinations are then enveloped. For both beam and column shear steel design, the required bars are based on the enveloped shear force diagrams.

Beam Results

Beam results are shown in the three following spreadsheets: **Design Results**, **Beam Bending Reinforcement**, and **Beam Shear Reinforcement**.

Design Results Spreadsheet

The **Design Results Spreadsheet** shows the governing maximum code check for the top and bottom of the beam for all spans.

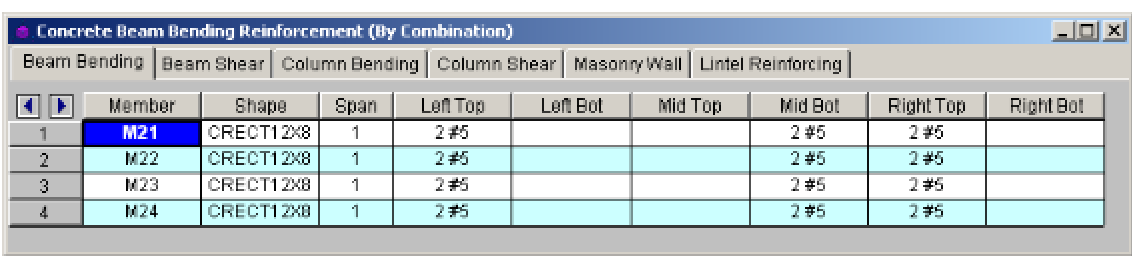


	Member	Shape	UC Max Top	Loc[ft]	UC Max Bot	Loc[ft]	Shear UC	Loc[ft]	Phi*Mnz Top[k-ft]	Phi*Mnz B...	Phi*Vny[k]
1	M1	CRECT12X8	.116	8.531	.307	3.938	.472	7.031	24.257	24.257	6.367
2	M2	CRECT12X8	.198	.51	0	0	.066	4.594	24.257	0	8.155
3	M3	CRECT12X8	.045	7.5	.213	3.917	.42	6.667	24.257	24.257	6.367
4	M4	CRECT12X8	0	0	.115	.521	.048	7.188	0	24.257	8.155

These top and bottom code checks, **UC Max Top** and **UC Max Bot**, are based on the top/bottom moment capacities and maximum top/bottom moment. Currently the moment capacity is based only on the tension steel (NO compression steel is considered in the capacity calculation). The governing maximum shear check for all spans, **Shear UC**, is also shown. The capacities shown are only for the governing section. Capacities for each span, as well as beam reinforcement detailing diagrams, may be viewed on the Detail Report.

Beam Bending Reinforcement Spreadsheet

The **Beam Bending Reinforcement Spreadsheet** records the top and bottom flexural reinforcement steel required for the left, middle, and right locations of each beam. This spreadsheet may be accessed by selecting **Members** ▸ **Concrete Reinforcing** on the **Results Menu** and the results are listed on the **Beam Bending** tab.



	Member	Shape	Span	Left Top	Left Bot	Mid Top	Mid Bot	Right Top	Right Bot
1	M21	CRECT12X8	1	2 #5			2 #5	2 #5	
2	M22	CRECT12X8	1	2 #5			2 #5	2 #5	
3	M23	CRECT12X8	1	2 #5			2 #5	2 #5	
4	M24	CRECT12X8	1	2 #5			2 #5	2 #5	

The **Member** column lists the beam label.

The **Shape** column displays the beam size. When no adequate member could be found from the available shapes list, this field will display the text "not designed". Consider re-framing, relaxing the design requirements (see Design Optimization), or adding more shapes to the available Redesign List (see Appendix A – Redesign Lists).

The **Span** column displays the span number corresponding to the reinforcement sections listed. Span '1' is the span beginning at the "start" of the beam and subsequent spans are numbered '2', '3', '4', and so forth moving from the "start" to the "end" of the beam.

The program assumes that the moment diagrams for all beam spans have two or fewer points of inflection. Therefore, each span is broken into **Left, Middle, and Right Reinforcement Sections** for flexural steel layout. Each section is further broken into **Top and Bottom Reinforcement Sections**. Note that a beam may have only two or even one reinforcement section. In this case, the other reinforcement sections would be left blank in this spreadsheet.

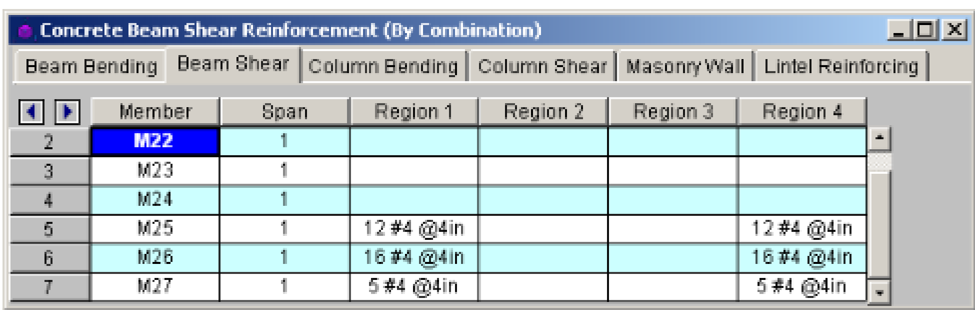
The **Left Top**, **Left Bot**, **Mid Top**, **Mid Bot**, **Right Top**, and **Right Bot** entries record the number and size of flexural reinforcement bars that are required in each of the six **Reinforcement Sections**. The first number indicates the number of parallel reinforcement bars in that section. The second number, preceded by the '#' sign, indicates the size of reinforcement bars used.

Note

- Only reinforcement bars selected by the program are listed in this spreadsheet. If a custom rebar layout is used for a particular beam, all six reinforcement section entries will be left blank.
- Longitudinal reinforcement bars are assumed to be in a **single layer** at the top and/or bottom of the member.
- Longitudinal reinforcement bars for the left and right sides of adjacent spans have been "smoothed" such that the larger steel area is used for both sides.

Beam Shear Reinforcement Spreadsheet

The **Beam Shear Reinforcement Spreadsheet** records the shear reinforcement ties required in each shear region of each beam. This spreadsheet may be accessed by selecting **Members** ▸ **Concrete Reinforcing** on the **Results Menu** and the results are listed on the **Beam Shear** tab.



	Member	Span	Region 1	Region 2	Region 3	Region 4
2	M22	1				
3	M23	1				
4	M24	1				
5	M25	1	12 #4 @4in			12 #4 @4in
6	M26	1	16 #4 @4in			16 #4 @4in
7	M27	1	5 #4 @4in			5 #4 @4in

The **Member** column lists the beam label.

The **Span** column displays the span number corresponding to the shear regions listed. Span '1' is the span beginning at the "start" of the beam and subsequent spans are numbered '2', '3', '4', and so forth moving from the "start" to the "end" of the beam.

Each beam's shear reinforcement layout is broken into either two or four **Shear Reinforcement Regions**. The user can control whether the program uses '2' or '4' regions from the **Concrete** tab of the **Global Parameters Dialog**. The program will try to group the required shear ties/stirrups into two or four regions and will allow for a middle region to have no shear reinforcement if the shear force is lower than that for which the code requires shear reinforcement.

The **Region 1**, **Region 2**, **Region 3**, and **Region 4** entries record the number, size, and spacing of shear reinforcement ties/stirrups that are required in each of the **Reinforcement Regions**. The first number of each entry indicates the total number of ties/stirrups that are required in that region of the beam span. The second number, proceeded by the '#' sign, indicates the size of reinforcement bars used. The third number, proceeded by the "@" symbol, indicates the spacing of the ties/stirrups in that region of the beam span.

Note

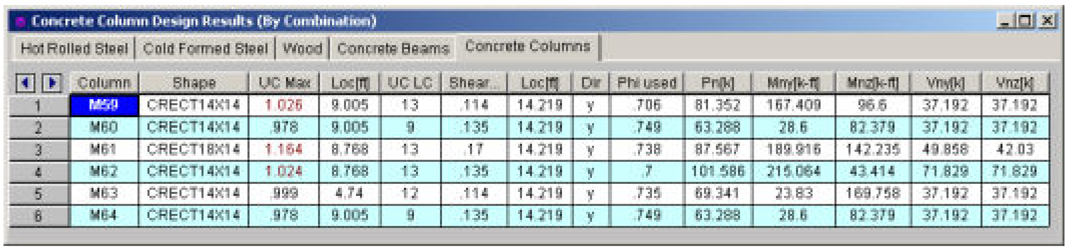
- If '2' shear regions are selected on the **Concrete** tab of the **Global Parameters Dialog**, the Region 2 and Region 3 entries in this spreadsheet will be left blank.
- The concrete code checks are only performed at the sections where the internal forces are calculated. The number of internal force calculations is based on the setting in the Global Parameters dialog. Normally, this is acceptable for design and analysis. However, it is possible for the design locations (face of support for moment and "d" from the face of support for shear) to be located far enough away from the nearest internal force location that it could affect the code check results. If this happens, it may be advisable to use a larger number of internal sections. Or, the user may be forced to calculate the maximum Vu and Mu themselves.

Column Results

Column results are shown in the three following spreadsheets: **Design Results**, **Column Bending Reinforcement**, and **Column Shear Reinforcement**.

Design Results Spreadsheet

The **Design Results Spreadsheet** shows the governing maximum code check for the column for all spans.



Concrete Column Design Results (By Combination)

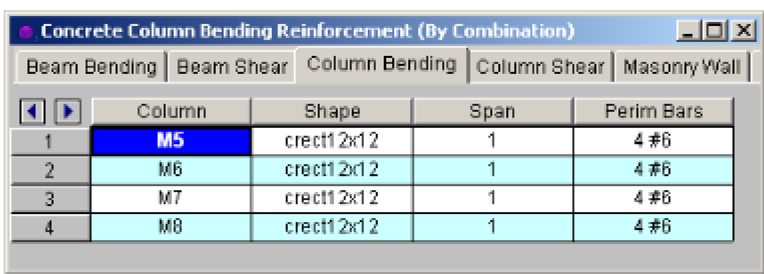
Hot Rolled Steel | Cold Formed Steel | Wood | Concrete Beams | Concrete Columns

	Column	Shape	UC Max	Loc[ft]	UC LC	Shear...	Loc[ft]	Dir	Phi used	Pn[k]	Mny[k-ft]	Mnz[k-ft]	Vny[k]	Vnz[k]
1	M59	CRECT14X14	1.026	9.005	13	.114	14.219	y	.706	81.352	167.409	96.6	37.192	37.192
2	M60	CRECT14X14	.978	9.005	9	.135	14.219	y	.749	63.288	28.6	82.379	37.192	37.192
3	M61	CRECT18X14	1.164	8.768	13	.17	14.219	y	.738	87.567	189.916	142.235	49.858	42.03
4	M62	CRECT14X14	1.024	8.768	13	.135	14.219	y	.7	101.586	215.064	43.414	71.829	71.829
5	M63	CRECT14X14	.999	4.74	12	.114	14.219	y	.735	69.341	23.83	169.758	37.192	37.192
6	M64	CRECT14X14	.978	9.005	9	.135	14.219	y	.749	63.288	28.6	82.379	37.192	37.192

The governing maximum shear check for all spans is also shown. The governing load combination for the governing code check is shown because the column capacity is based upon the actual moments and axial forces for that load combination. The capacities shown are only for the governing section. Capacities for each span, as well as beam reinforcement detailing diagrams, may be viewed on the Detail Report.

Column Bending Reinforcement Spreadsheet

The **Column Bending Reinforcement Spreadsheet** shows the perimeter flexural reinforcement steel required in each span of each column. This spreadsheet may be accessed by selecting **Members** ‣ **Concrete Reinforcing** on the **Results Menu** and the results are listed on the **Column Bending** tab.



Concrete Column Bending Reinforcement (By Combination)

Beam Bending | Beam Shear | Column Bending | Column Shear | Masonry Wall

	Column	Shape	Span	Perim Bars
1	M5	crect12x12	1	4 #6
2	M6	crect12x12	1	4 #6
3	M7	crect12x12	1	4 #6
4	M8	crect12x12	1	4 #6

The **Column** field displays the column label.

The **Shape** column displays the physical column or lift size. When no adequate member could be found from the available shapes, this field will display the text “not designed”. Consider re-framing, relaxing the design or deflection requirements (see Design Optimization), or adding more shapes to the available Redesign List (see Appendix A – Redesign Lists).

The **Span** column displays the span number corresponding to the perimeter reinforcement listed. Span '1' is the span beginning at the "start" of the column and subsequent spans are numbered '2', '3', '4', and so forth moving from the "start" to the "end" of the column.

The **Perim Bars** column records the number and size of perimeter longitudinal reinforcing bars. The first number indicates the total number of longitudinal bars in that span. The second number, preceded by the '#' sign, indicates the size of the reinforcement bars used.

Note

- Only reinforcement bars selected by the program are listed in this spreadsheet. If a custom rebar layout is used for a particular column, the **Perim Bars** entry will be left blank.
- Longitudinal reinforcement bars are assumed to be uniformly arranged around the perimeter of the column for both rectangular and round column sections.
- A minimum of 6 bars will be used in round column sections.
- Longitudinal reinforcement bars for the bottom and top sides of adjacent spans have been "smoothed" such that the larger steel area is used for both sides.

Column Shear Reinforcement Spreadsheet

The **Column Shear Reinforcement Spreadsheet** shows the shear reinforcement ties required in each shear region of each column. This spreadsheet may be accessed by selecting **Members** ▸ **Concrete Reinforcing** on the **Results Menu** and the results are listed on the **Column Shear** tab.

Concrete Column Shear Reinforcement (By Combination)

Beam Bending | Beam Shear | Column Bending | Column Shear | Masonry Wall | Lintel Reinforc

	Column	Span	Region 1	Region 2	Region 3	Region 4
1	M5	1	9 #4 @12in			
2	M6	1	11 #4 @12in			
3	M7	1	8 #4 @12in			
4	M8	1	10 #4 @12in			

The **Column** field displays the column label.

The **Span** column displays the span number corresponding to the shear regions listed. Span '1' is the span beginning at the "start" of the column and subsequent spans are numbered '2', '3', '4', and so forth moving from the "start" to the "end" of the column.

Each column's shear reinforcement layout is broken into either two or four **Shear Reinforcement Regions**. The user can control whether the program uses '2' or '4' regions from the **Concrete** tab of the **Global Parameters** dialog. The program will try to group the required shear ties into 2 or 4 regions. Unlike beams, columns cannot have a zero shear steel region. Note also that columns in tension receive NO shear capacity from the concrete.

The **Region 1**, **Region 2**, **Region 3**, and **Region 4** entries record the number, size, and spacing of shear reinforcement ties/stirrups that are required in each of the **Reinforcement Regions**. The first number of each entry indicates the total number of ties/stirrups that are required in that region of the column span. The second number, proceeded by the '#' sign, indicates the size of reinforcement bars used. The third number, proceeded by the "@" symbol, indicates the spacing of the ties/stirrups in that region of the column span.

Note

- If '2' shear regions are selected on the Concrete tab of the Global Parameters dialog, the Region 2 and Region 3 entries in this spreadsheet will be left blank.

Concrete Detail Reports

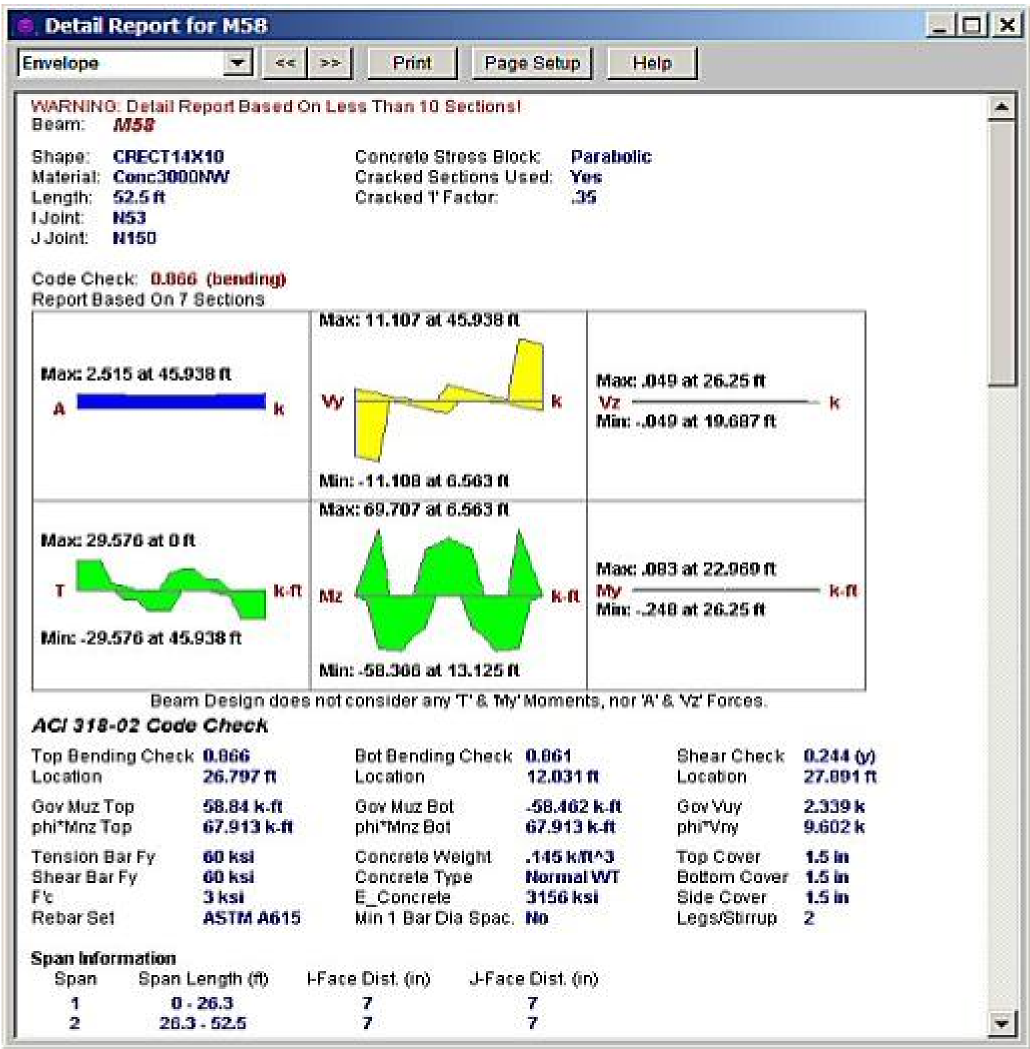
The **Concrete Detail Reports** allow you to see the overall force, stress, and deflection state for any particular member. Detail reports for concrete members are not based on individual load combinations as they are for steel or wood members. Instead, they are based on an envelope of the solved load combinations. Concrete columns are the exception to this in that the

columns are solved for all load combinations and then the resulting required steel is enveloped. The detail reports for concrete Column member types are also different than those for concrete Beam member types in terms of the design information that is shown below the force diagrams.

Detail reports for concrete members can, and often do, go more than one page in length due to the large amount of information that must be displayed for concrete design. One reason for this is that RISA-3D figures out the number of spans for concrete beams and columns based on the number of internal supports, thus one physical member may have several spans that all must be reported.

Beam Detail Reports

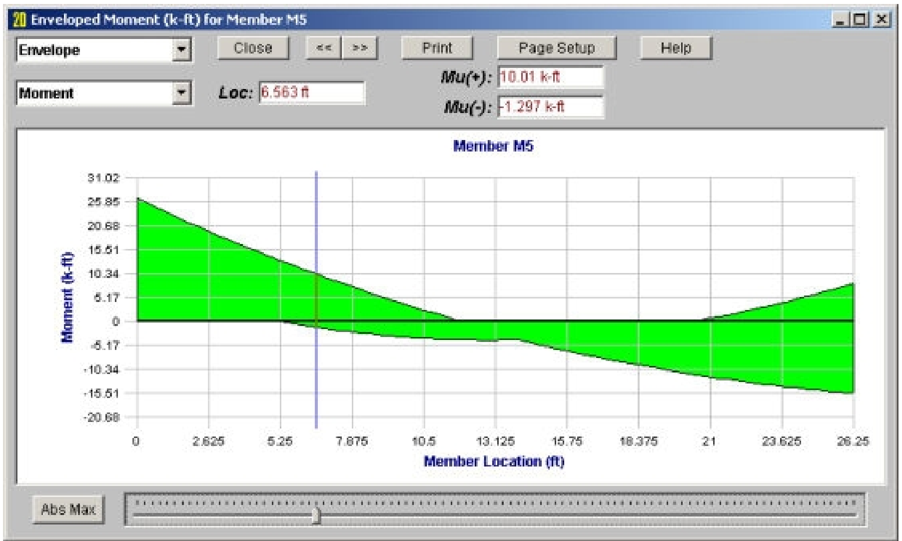
The image below is the first portion of a detail report for a concrete beam member showing the member information, warnings, force diagrams, code checks, and span information.



You can tell the **Member Type** by looking at the black title in the upper left corner next to the red member label. This title will always show the member type (Beam, Column, HBrace, VBrace). If the member type is 'None', this title will be displayed as 'Member'.

The **Member Information** in the text above the force diagrams shows basic member information as well as the **Concrete Stress Block** type used in the solution, whether **Cracked Sections** were used for the nominal design, and the **Cracked 'I' Factor** that was used for that member.

The next section of the detail report contains the **Member Force Diagrams**. The diagrams shown are envelope diagrams of all solved load combinations. Any **Unused Force Warnings** or critical **Design Warnings** will be shown directly below the force diagrams in the detail report. An enlarged interactive member force diagram can be accessed by clicking on the desired diagram.



Each enlarged diagram will also have a slider bar at the bottom of the window for checking forces at all locations along the member. There is also an **Abs Max** button that will jump the slider bar to the absolute maximum value in the diagram. Note that once an enlarged diagram is opened, diagrams for other forces may be accessed via the pull down menu on the left.

The **Code Check Information** directly below the force diagrams is a summary of the governing checks for bending and shear, their location, and the section capacities at those locations. Separate bending checks for the most critical top and most critical bottom condition are given. **Gov Muz Top** and **Gov Muz Bot** represent the governing ultimate moment in the top and bottom of the beam respectively. **Gov Vuy** represents the governing ultimate shear along the local y axis of the beam.

The values **phi*Mnz Top** and **phi*Mnz Bot** represent the nominal moment strength in the top and bottom of the beam respectively, reduced by the appropriate **Strength Reduction Factor**, **Phi**, as indicated in the code. Likewise, the value **phi*Vuy** represents the nominal shear strength in the beam, reduced by the appropriate Phi Factor.

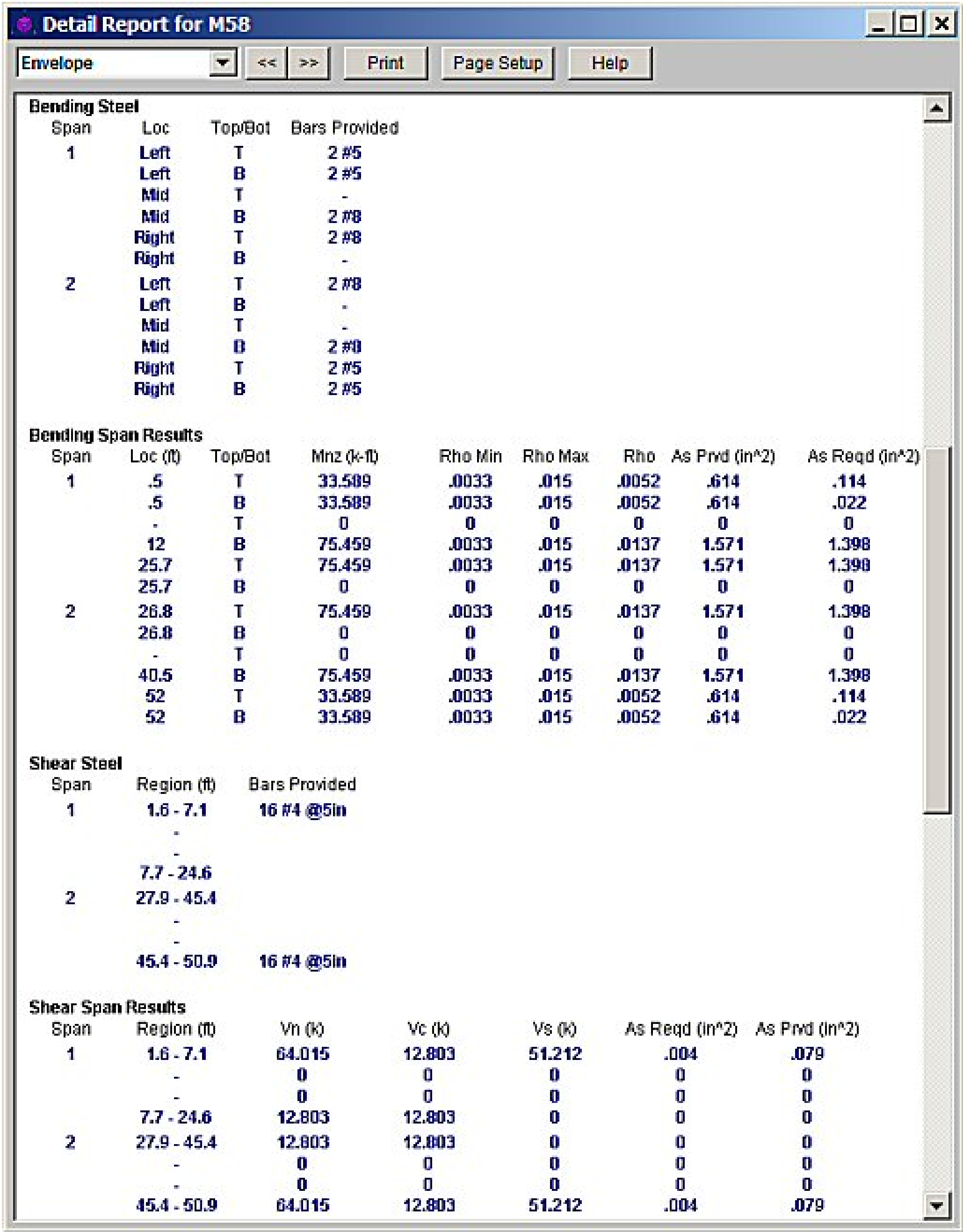
There is also general concrete, reinforcement, and bar cover information about the section provided which you would need if you were doing a hand check. **Concrete Type** (Normal Weight vs Light Weight) is automatically determined from the **Concrete Weight** density per the ACI code. λ is taken from the Materials spreadsheet. The **E_Concrete** value shown here is either the value entered on the **Concrete** tab of the **Materials Spreadsheet** or is the calculated value based on the given f'c and weight density (if the 'E' value was left blank on the Materials Spreadsheet).

The **Span Information** gives the start and end of each span centerline within the member, as well as the distance from the column centerline to the face of the column for each end of the span.

The next portion of the detail report shown below contains detailed information for the placement of the **Bending Steel** and the **Bending Span Results** for each span. The bending capacity for the governing section in each span is shown as **Mnz**, the nominal moment strength. **Rho Min** and **Rho Max** are the minimum and maximum required reinforcement ratios at each location. These values are based on the minimum and maximum reinforcing requirements for flexural members as described in ACI 318-11 sections 10.5.1 and 10.3.3/10.3.5 respectively. **Rho** is the ratio of reinforcement corresponding to the area of steel provided at each location, **As Prvd**. The **As Req** value is the area of steel required at each location.

Note:

- Per ACI 318-11 Section 10.5.3, the reinforcement ratio (ρ) chosen by the program can be less than ρ_{min} when A_s Provided exceeds A_s Required by more than 33%



Detail Report for M58

Envelope | << | >> | Print | Page Setup | Help

Bending Steel

Span	Loc	Top/Bot	Bars Provided
1	Left	T	2 #5
	Left	B	2 #5
	Mid	T	-
	Mid	B	2 #8
	Right	T	2 #8
	Right	B	-
2	Left	T	2 #8
	Left	B	-
	Mid	T	-
	Mid	B	2 #8
	Right	T	2 #5
	Right	B	2 #5

Bending Span Results

Span	Loc (ft)	Top/Bot	Mnz (k-ft)	Rho Min	Rho Max	Rho	As Prvd (in^2)	As Reqd (in^2)
1	.5	T	33.589	.0033	.015	.0052	.614	.114
	.5	B	33.589	.0033	.015	.0052	.614	.022
	-	T	0	0	0	0	0	0
	12	B	75.459	.0033	.015	.0137	1.571	1.398
	25.7	T	75.459	.0033	.015	.0137	1.571	1.398
	25.7	B	0	0	0	0	0	0
2	26.8	T	75.459	.0033	.015	.0137	1.571	1.398
	26.8	B	0	0	0	0	0	0
	-	T	0	0	0	0	0	0
	40.5	B	75.459	.0033	.015	.0137	1.571	1.398
	52	T	33.589	.0033	.015	.0052	.614	.114
	52	B	33.589	.0033	.015	.0052	.614	.022

Shear Steel

Span	Region (ft)	Bars Provided
1	1.6 - 7.1	16 #4 @5in
	-	
	-	
	7.7 - 24.6	
2	27.9 - 45.4	
	-	
	-	
	45.4 - 50.9	16 #4 @5in

Shear Span Results

Span	Region (ft)	Vn (k)	Vc (k)	Vs (k)	As Reqd (in^2)	As Prvd (in^2)
1	1.6 - 7.1	64.015	12.803	51.212	.004	.079
	-	0	0	0	0	0
	-	0	0	0	0	0
	7.7 - 24.6	12.803	12.803	0	0	0
2	27.9 - 45.4	12.803	12.803	0	0	0
	-	0	0	0	0	0
	-	0	0	0	0	0
	45.4 - 50.9	64.015	12.803	51.212	.004	.079

The next portion of the detail report, shown above, contains detailed information for the placement of the **Shear Steel** and the **Shear Span Results** for each span. Shear results are shown by region within each span. The number of regions used is a function of the shear diagram, with the maximum number of regions being taken from the **Shear Regions** setting on the **Concrete** tab of the **Global Parameters Dialog**. The number, size, and spacing of reinforcing bars is given for each region. Also indicated is the nominal shear strength, **Vn**, in each region. The portion of the nominal shear strength provided by the concrete and the steel, **Vc** and **Vs** respectively, is given for each region. The area of steel required, **As Reqd**, and the area of steel provided, **As Prvd**, are also given for each shear region and are reported as 'area of steel per unit dimension', i.e. in^2/in or mm^2/mm.

It should be noted that the values for **Mn** and **Vn** given in this section of the detail report are the UNREDUCED nominal capacities of the member at each span/region. The actual design capacities would be obtained by multiplying these values by their respective **Phi Factors** indicated in the code.

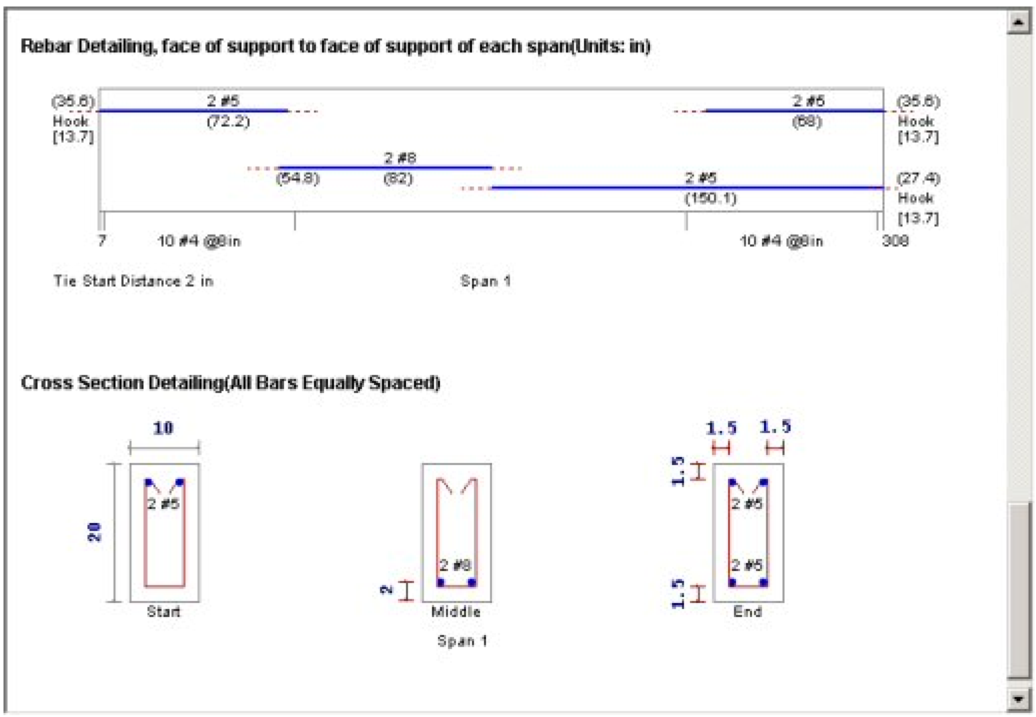
The last section of the detail report shows the **Beam Reinforcement Detailing Diagrams**. The **Rebar Detailing** portion of the report shows elevation views of the beam complete with top and bottom flexural reinforcement indicated for the left, middle, and right portions of each span. The number and size of bars required in each section is indicated on the top middle of each drawn bar. The required length of each bar is indicated on the bottom middle of each drawn bar in parenthesis. Development lengths are shown in parenthesis at one end of each bar and is represented by a dashed line. For bars at the ends of the beam, hook lengths are given in addition to the development lengths and are shown in brackets.

Note:

- Development lengths are calculated per ACI 318-11 Section 12.2.2 and 12.5.2. No additional factors are used, aside from the lightweight modification factor.

The values shown at the bottom corner of each span indicate the distance from the start of the beam to the face of a support. Flexural bars at the ends of the beam are measured beginning at the face of the support and bars at intermediate supports are measured to the center of the support.

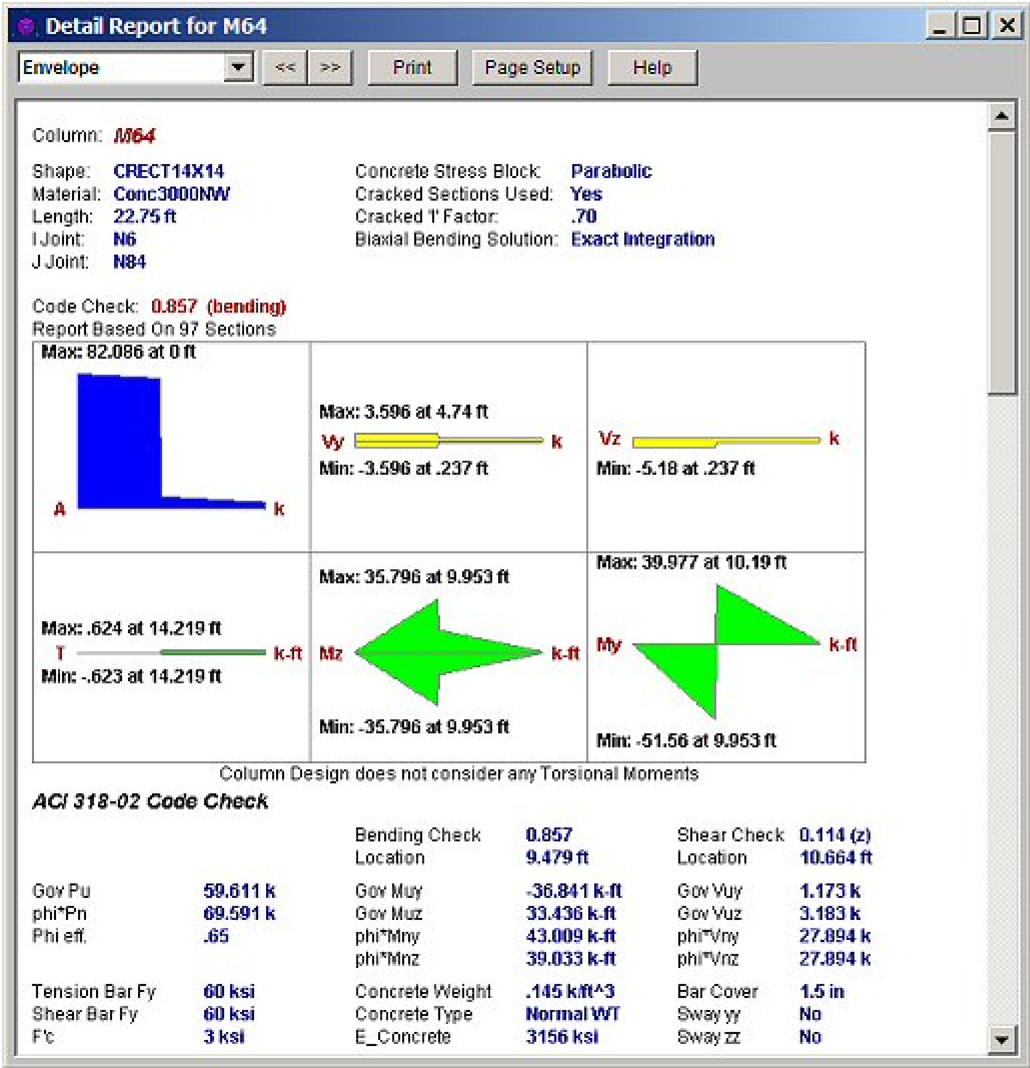
The number, size, and spacing of shear reinforcement is also indicated below each span in the corresponding shear region. Each shear region is indicated by vertical lines at the bottom of the beam.



The **Cross Section Detailing** portion of the report shows cross sectional views for the start, middle, and end of each beam span. The number and size of flexural bars for each cross section are shown as well as the orientation of the shear ties/stirrups. The clear cover to each stirrup for the top and sides is shown. The overall beam dimensions for each span are indicated on the 'Start' cross section.

Column Detail Reports

The image below is the first portion of a detail report for a concrete column member showing the member information, warnings, force diagrams, code checks, and span information. As can be seen, the concrete column results are very similar to the beam results with just a few additions and differences.



You can tell the **Member Type** by looking at the black title in the upper left corner next to the red member label. This title will always show the member type (Beam, Column, HBrace, VBrace). If the member type is "None", this title will be displayed as "Member".

The **Member Information** in the text above the force diagrams shows basic member information as well as the **Concrete Stress Block** type used in the solution, whether **Cracked Sections** were used for the nominal design, and the **Cracked 'I' Factor** that was used for that member. The **Biaxial Bending Solution** method that was used is also reported, and if applicable, the **Parme Beta Factor**.

The next section of the detail report contains the **Member Force Diagrams**. The diagrams shown are envelope diagrams of all solved load combinations. Any **Unused Force Warnings** or critical **Design Warnings** will be shown directly below the force diagrams in the detail report. An enlarged interactive member force diagram can be accessed by clicking on the desired diagram. For more information, see Beam Detail Reports.

The **Code Check Information** below the force diagrams is a summary of the governing checks for bending and shear, their location, and the section capacities at those locations. **Gov Pu** represents the governing ultimate axial load in the column. **Gov Muy** and **Muz** represent the governing ultimate moment about each local axis of the column. **Gov Vuy** and **Vuz** represent the governing ultimate shear along each local axis of the column.

There is also general concrete, reinforcement, and bar cover information about the section provided which are useful for hand calculation verification.**Concrete Type** (Normal Weight vs Light Weight) is automatically determined from the **Concrete Weight** density per the ACI code. The **E_Concrete** value shown here is either the value entered on the **Concrete** tab of the

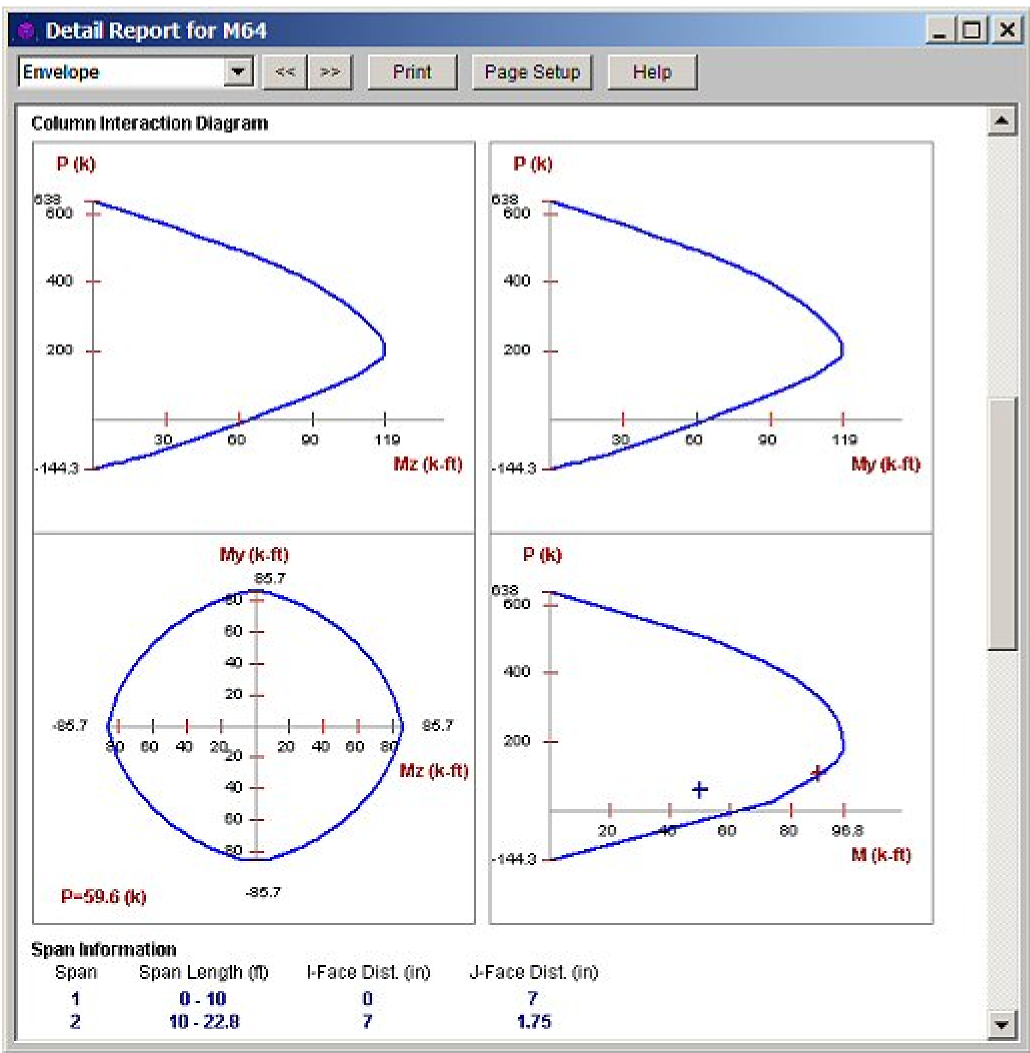
Materials Spreadsheet or is the calculated value based on the given f'c and weight density (if the 'E' value was left blank on the Materials Spreadsheet).

Note:

- When solving using the PCA Load Contour method, P_u will always equal P_n. This represents the axial value at which the controlling slice of interaction diagram was taken. The bending check is taken as the following equation, which is derived from the PCA Notes on ACI 318-99, Chapter 12.

$$UC = \left(\frac{M_{uy}}{\varphi M_{noy}}\right)^{\frac{\log 0.5}{\log \beta}} + \left(\frac{M_{uz}}{\varphi M_{noz}}\right)^{\frac{\log 0.5}{\log \beta}}$$

- When solving using the Exact Integration method, a worst-case combination of P_u, M_{uy}, and M_{uz} is determined. A straight line is essentially drawn between the origin of the interaction diagram, and this coordinate within the 3D interaction diagram. The bending check is taken as the length of that line, divided by the distance from the origin to the intersection of that line and the interaction diagram. For this reason the ratios $(P_u/\varphi P_n)$, $(M_u/\varphi M_n)$ are all equal to the bending check.

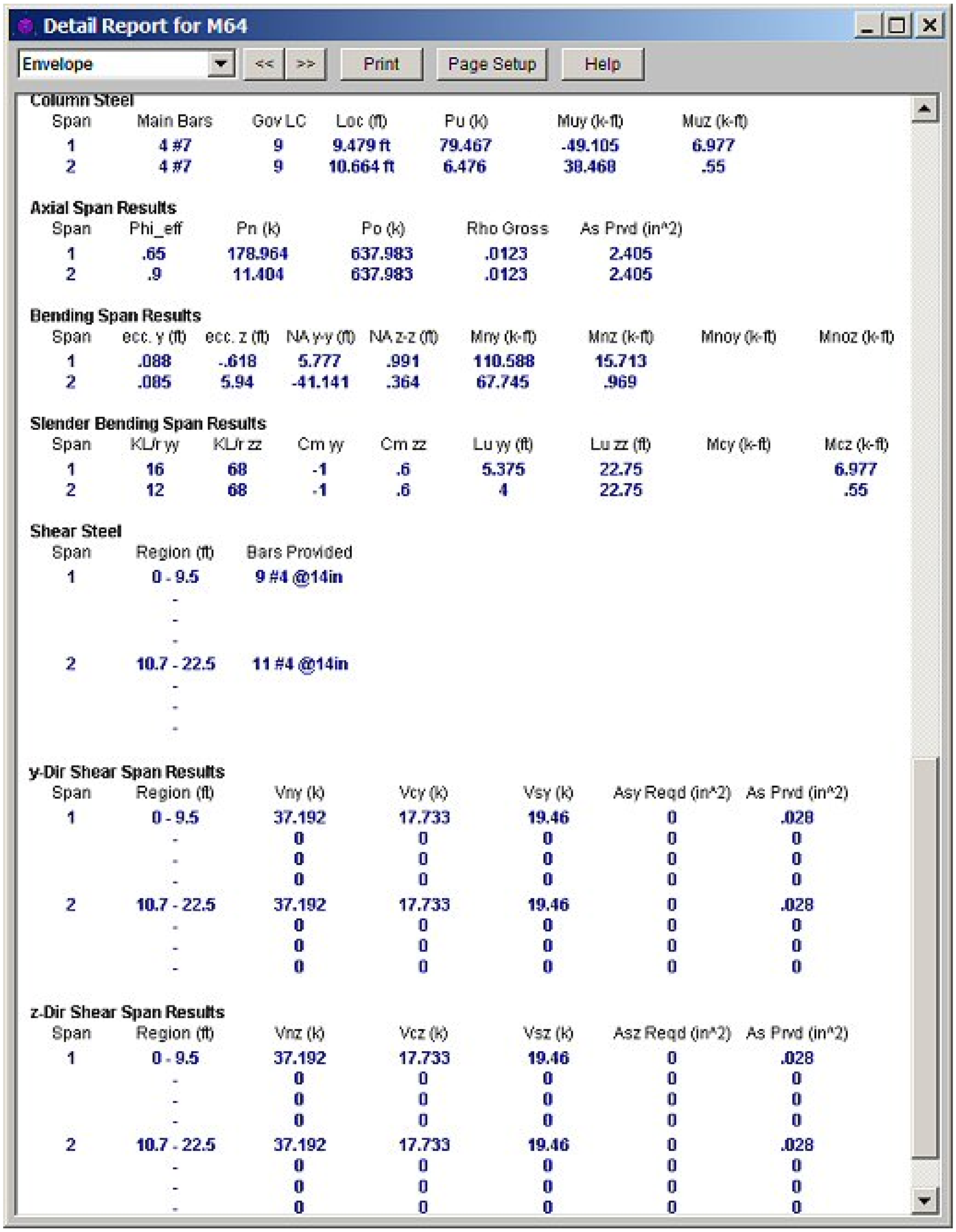


The next portions of the detail report shown above contain the **Column Interaction Diagrams** for the column member and the **Span Information**.

A **Column Interaction Diagram** for uniaxial bending is shown for each axis of the column. These diagrams plot the unreduced nominal strengths **P vs. M** for the corresponding column local axis. If the column only has bending about one axis there will be only one interaction diagram shown.

For columns under biaxial bending there is also a diagram which plots the unreduced nominal moments strengths **Mz vs. My** at the governing ultimate axial load, **P**. The last diagram is for the biaxial bending condition where the exact integration method is used and shows the interaction surface plotted at the angle of applied load (Pu, Muy, Muz). This last diagram is only shown when the **Exact Integration Method** is used.

The **Span Information** section shows the length of each span and the distances from the centerline of each support to the face of each support.


Detail Report for M64

Envelope | << | >> | Print | Page Setup | Help

Column Steel

Span	Main Bars	Gov LC	Loc (ft)	Pu (k)	Muy (k-ft)	Muz (k-ft)
1	4 #7	9	9.479 ft	79.467	-49.105	6.977
2	4 #7	9	10.664 ft	6.476	38.468	.55

Axial Span Results

Span	Phi_eff	Pn (k)	Po (k)	Rho Gross	As Prvd (in^2)
1	.65	178.964	637.983	.0123	2.405
2	.9	11.404	637.983	.0123	2.405

Bending Span Results

Span	ecc. y (ft)	ecc. z (ft)	NA y-y (ft)	NA z-z (ft)	Mny (k-ft)	Mnz (k-ft)	Mnoy (k-ft)	Mnoz (k-ft)
1	.088	-.618	5.777	.991	110.588	15.713		
2	.085	5.94	-41.141	.364	67.745	.969		

Slender Bending Span Results

Span	KL/r yy	KL/r zz	Cm yy	Cm zz	Lu yy (ft)	Lu zz (ft)	Mcy (k-ft)	Mcz (k-ft)
1	16	68	-1	.6	5.375	22.75		6.977
2	12	68	-1	.6	4	22.75		.55

Shear Steel

Span	Region (ft)	Bars Provided
1	0 - 9.5	9 #4 @14in
	-	
	-	
	-	
2	10.7 - 22.5	11 #4 @14in
	-	
	-	
	-	

y-Dir Shear Span Results

Span	Region (ft)	Vny (k)	Vcy (k)	Vsy (k)	Asy Reqd (in^2)	As Prvd (in^2)
1	0 - 9.5	37.192	17.733	19.46	0	.028
	-	0	0	0	0	0
	-	0	0	0	0	0
	-	0	0	0	0	0
2	10.7 - 22.5	37.192	17.733	19.46	0	.028
	-	0	0	0	0	0
	-	0	0	0	0	0
	-	0	0	0	0	0

z-Dir Shear Span Results

Span	Region (ft)	Vnz (k)	Vcz (k)	Vsz (k)	Asz Reqd (in^2)	As Prvd (in^2)
1	0 - 9.5	37.192	17.733	19.46	0	.028
	-	0	0	0	0	0
	-	0	0	0	0	0
	-	0	0	0	0	0
2	10.7 - 22.5	37.192	17.733	19.46	0	.028
	-	0	0	0	0	0
	-	0	0	0	0	0
	-	0	0	0	0	0

This portion of the report shown above contains the sections pertaining to the axial, bending, and shear results as well as the longitudinal and shear reinforcement.

The **Column Steel** section indicates the longitudinal reinforcement in each span as well as the governing load combination and location. The ultimate axial load, **Pu**, and the ultimate moments, **Muy** and Muz, are also given for each span.

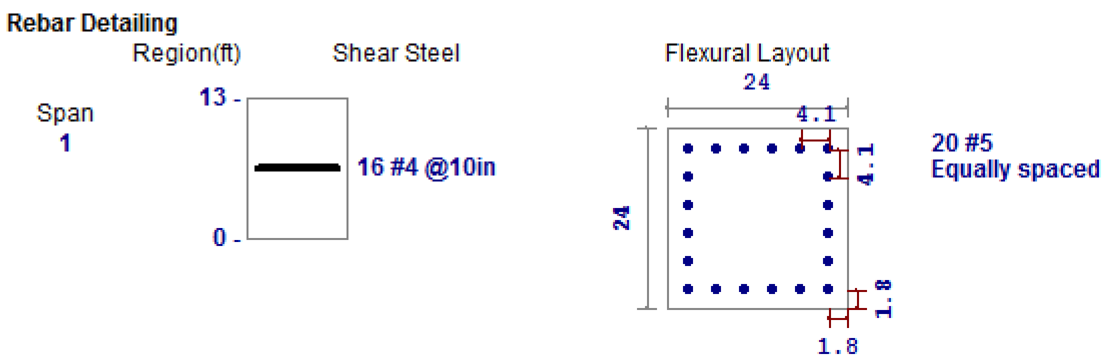
The **Axial Span Results** show the strength reduction factor, **Phi_eff**, used for each span. The axial capacities for each span are shown as **Pn**, the nominal axial strength, and **Po**, the nominal axial strength with zero eccentricity. The area of longitudinal reinforcement provided in the column is listed as **As Prvd** and the ratio of As Prvd to the gross cross sectional area of the column is listed as **Rho Gross**.

The **Bending Span Results** show the calculated eccentricities, **ecc. y** and **ecc. z**, due to the ultimate moment about each axis divided by the ultimate axial load. The neutral axis depth for bending about the local y-axis and z-axis are listed as **NA y-y**

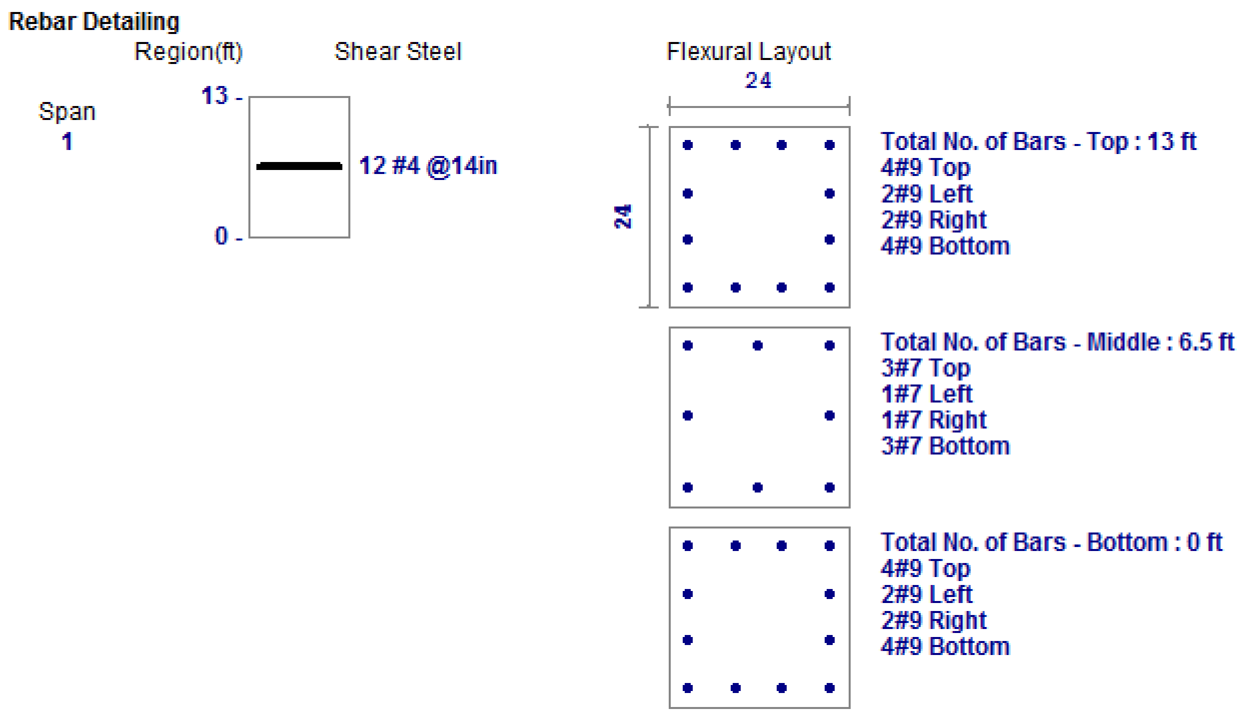
and **NA z-z** respectively. These neutral axis locations are always given with respect to the geometric center of the column. Also shown in this section are the unreduced nominal moment capacities, **Mny** and **Mnz**, for each span of the column. If the **PCA Load Contour Method** is used, **Mnoy** and **Mnoz** are given, representing the maximum allowable moment for uniaxial bending at the nominal axial strength, **Pn** (see Biaxial Bending of Columns). If the **Exact Integration Method** is used, these values will be left blank.

The **Sheer Steel** section of the report shows each span of the column broken into one or more shear regions and the number, size, and spacing of shear stirrups required in each of those regions is given. The shear design for columns is the envelope of all the shears for both directions.

The **y-Dir and z-Dir Shear Span Results** show the nominal shear strength, **Vny** and **Vnz**, in each shear region of the column followed by the nominal shear strengths of the concrete, **Vcy** and **Vcz**, and the nominal shear strengths of the steel, **Vsy** and **Vsz**. The area of shear reinforcement required in each shear region of the column is shown as **Asy Reqd** and **Asz Reqd**. The area of shear reinforcement provided in each shear region of the column is shown as **As Prvd**. Shear demand and concrete capacity are shown for both directions, but only one design of shear ties is used. Thus the As_reqd may vary for each side, but the As_prvd will always be the same.



The **Rebar Detailing**portion of the report shows cross sectional view of the shear ties and flexural bars for each span. When the program designs the flexural reinforcement, the spacing will be equally spaced and the cover is dimensioned to the centerline of the rebar.



The**Custom Rebar Detailing**portion of the report shows the cross sectional views of the shear ties and flexural bars for each span for the top, middle and bottom sectionsof each column span. The number and size of flexural bars for each cross section are shown drawn to scale and labeled next to the column. The Start and End locations are measured from the I-End of the column. The I-End of the column corresponds with the 0ft or 0% in the Rebar Layout. Therefore to orient the column correctly in the drawing, you need to define your column from the lower elevation upward to the higher elevation..

Magnified Moments / Slenderness Effects

Span	KL/r yy	KL/r zz	Cm yy	Cm zz	Lu yy (ft)	Lu zz (ft)	Mcy (k-ft)	Mcz (k-ft)
1	16	134	-1	.6	5.375	45		142.6104
2	12	134	.6157	.6157	4	45		49.0923

The **Slender Bending Span Results** give the ultimate moments for each axis amplified for the effects of member curvature, **Mcy** and **Mcz**. These values will be left blank for spans that do not meet the criteria for slender columns in the specific direction. Also shown in this section are the values **KL/r** for the y and z-axis, followed by the equivalent moment correction factors **Cm yy** and **Cm zz**. The unbraced lengths of the column for each span and each direction, **Lu yy** and **Lu zz**, are given as well.

For Non-Sway frames, the assumption is that EI = 0.25*Ec*Ig. This is equivalent to setting Bd to 0.6 in ACI 318-05 equation 10-12. For sway frame columns with a KL/r value greater than 22, the moment amplification is applied to the total moment rather than the "non-sway" portion of the moment.

Warning Log Messages will be produced when the following occurs:

- If the KL/r for the column exceeds 100 per Section 10.11.5 of ACI 318-05.
- If a slender member is classified as being part of a Sway frame, but a P-Delta analysis was NOT performed. For sway frames this P-Delta requirement applies anytime the slenderness ratio KL/r exceeds 22.

RISAConnection Integration

RISAConnection can be used in tandem with both RISAFloor and RISA-3D to design hot-rolled steel connections. The integration sends the geometry, loads, shape type and connection types automatically into RISAConnection. This allows you to design your connections in RISAConnection and then bring the results back into RISAFloor and RISA-3D for results presentation.

Here we will walk through the steps required to design connections using this integration.

1. Completing the Model

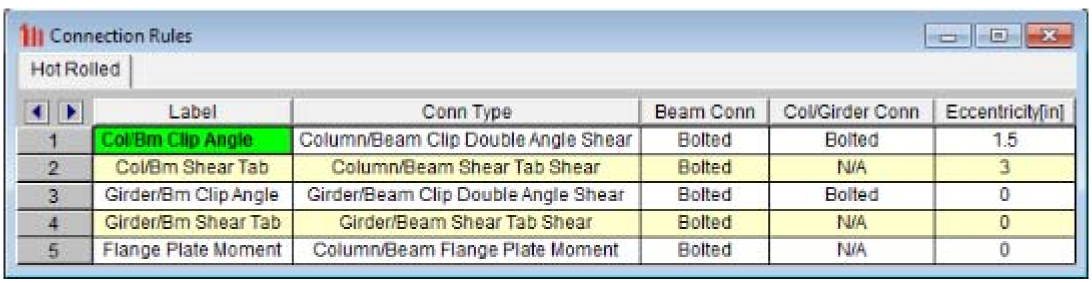
You must first draw your model. It is important that you use the Member Type (Column, Beam, VBrace, HBrace) designation properly. Otherwise your connections will not be designed. Keep in mind that connection design will only work for hot rolled connections at this point. Future versions will add to the RISAConnection connection type capabilities.

Note:

- See the Shear Connections section for more information on these types of connections.
- See the Moment Connections section for more information on these types of connections.
- See the Vertical Braces section for more information on these types of connections.
- See the Splices section for more information on these types of connections.

2. Defining Connection Rules

Next, connection rules must be created. These rules allow you to define which types of connections you want to design in your model. You must have a rule for each type of connection you want considered. Also, if you have lightly loaded connections and heavily loaded connections of the same Type you may want to create two separate rules for these, so that you can manipulate these groups separately.



	Label	Conn Type	Beam Conn	Col/Girder Conn	Eccentricity[in]
1	Col/Bm Clip Angle	Column/Beam Clip Double Angle Shear	Bolted	Bolted	1.5
2	Col/Bm Shear Tab	Column/Beam Shear Tab Shear	Bolted	N/A	3
3	Girder/Bm Clip Angle	Girder/Beam Clip Double Angle Shear	Bolted	Bolted	0
4	Girder/Bm Shear Tab	Girder/Beam Shear Tab Shear	Bolted	N/A	0
5	Flange Plate Moment	Column/Beam Flange Plate Moment	Bolted	N/A	0

To open the **Connection Rules** spreadsheet click the Connection Rules button on the **Data Entry** toolbar. Then create rules for each type of connection that you want to be designed with RISAConnection.

Label

This is how you will identify your Connection Rule in other areas within the program. Each Connection Rule must have a unique **Label**.

Conn Type

The **Connection Type** refers to the different types of connections currently considered in RISAConnection. You will need to set up at least one connection rule for each connection type in your project. When you solve the connections in RISAConnection the member ends assigned to an individual Connection Rule will be grouped together.

Note:

- You may have many double angle shear connections in your project. However, some are bolted with A325 bolts and some use A490 bolts. You should define these as separate Connection Rules.

Beam Conn

This option defines how the beam is connected to the supporting pieces (plate, clip angle, etc). When you go to RISAConnection all connections grouped in this Connection Rule will default to this type of connection.

Note:

- Many connections do not have options for these, so N/A will be shown

Col/Girder Conn

This option defines how the supporting member (column or girder) is connected to the supporting pieces (plate, clip angle, etc.). When you go to RISAConnection all connections grouped in this Connection Rule will default to this type of connection.

Note:

- Many connections do not have options for these, so N/A will be shown

Eccentricity

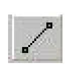
This value has no effect on the RISAConnection integration. Instead it is used for analysis purposes. In models integrated with RISAFloor, beam/column shear connections add an automatic eccentricity equal to half the column's depth automatically (see the Global Parameters-Solution tab for more information on this). However an *additional* eccentricity away from the column face may be entered in the Connection Rules spreadsheet. The value entered here should be equal to the distance between the face of column and the resultant beam end reaction (i.e. centroid of bolt/weld group on beam web).

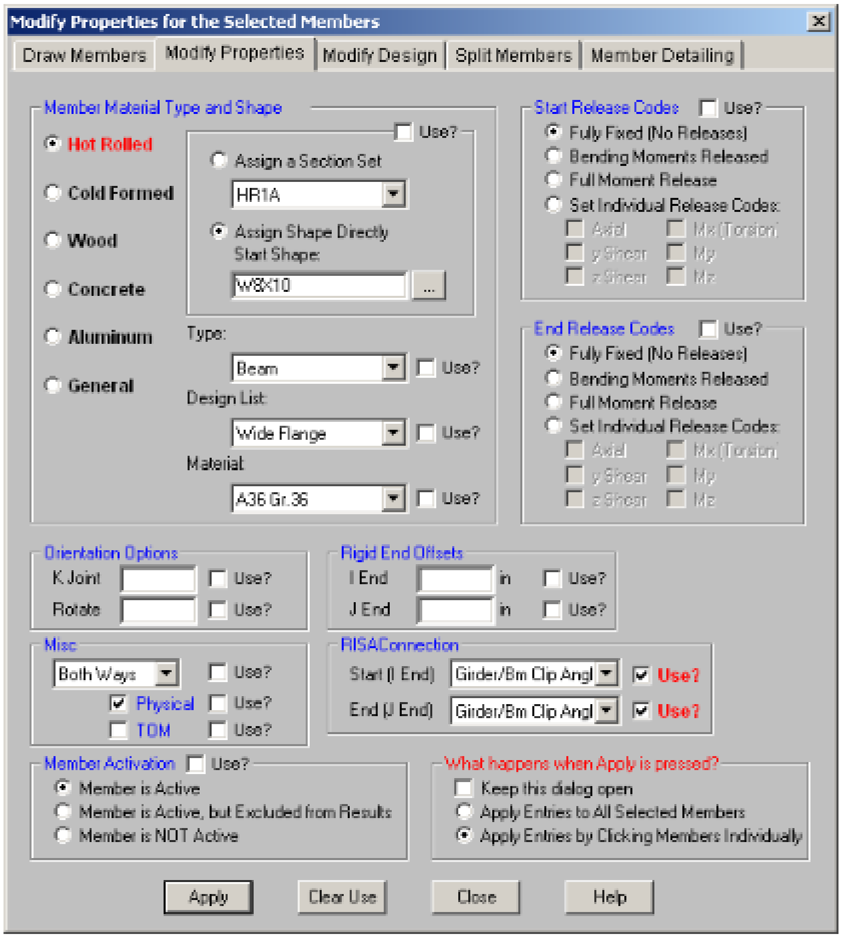
Note:

- This column is only available in RISAFloor and in RISA-3D if you came in from RISAFloor.
- This eccentricity is only applied to beams which connect to columns and which have the Connection Rule specified in the Connections tab of the Beams spreadsheet.

3. Assigning Connection Rules

Once the **Connection Rules** are defined they must be applied to the member ends in your model. This can be done graphically.

- To assign **Connection Rules** graphically click  on the **Drawing Toolbar** and click on the **Modify Properties** tab.

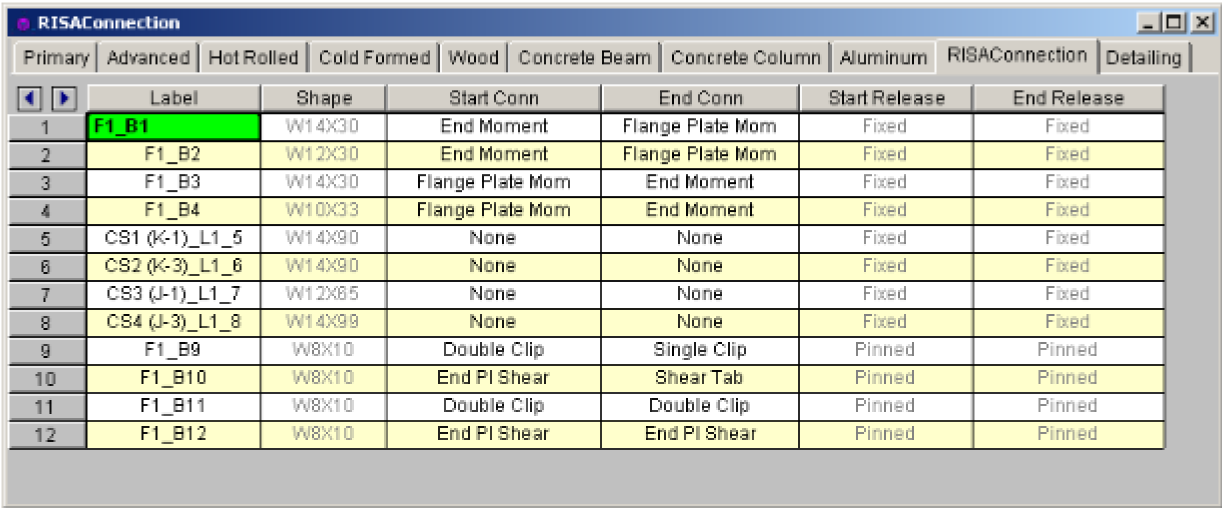


Here you can select the proper Connection Rule for both the **Start** and **End** of the member and check the **"Use"** checkboxes. You can then either apply the entries to all selected members or by clicking members individually.

- It is also possible to modify and view connections from the **RISAConnection** tab of the **Members** spreadsheet. See the next section.
- Currently column splices can only be applied in RISA-3D. This will also be available in a later version of RISAFloor.

RISAConnection Spreadsheet

The **RISAConnection** tab of the **Members** spreadsheet provides a place to assign, edit and view Connection Rules as they apply to individual members.



RISAConnection

Primary | Advanced | Hot Rolled | Cold Formed | Wood | Concrete Beam | Concrete Column | Aluminum | RISAConnection | Detailing

	Label	Shape	Start Conn	End Conn	Start Release	End Release
1	F1_B1	W14X30	End Moment	Flange Plate Mom	Fixed	Fixed
2	F1_B2	W12X30	End Moment	Flange Plate Mom	Fixed	Fixed
3	F1_B3	W14X30	Flange Plate Mom	End Moment	Fixed	Fixed
4	F1_B4	W10X33	Flange Plate Mom	End Moment	Fixed	Fixed
5	CS1 (K-1)_L1_5	W14X90	None	None	Fixed	Fixed
6	CS2 (K-3)_L1_6	W14X90	None	None	Fixed	Fixed
7	CS3 (J-1)_L1_7	W12X65	None	None	Fixed	Fixed
8	CS4 (J-3)_L1_8	W14X99	None	None	Fixed	Fixed
9	F1_B9	W8X10	Double Clip	Single Clip	Pinned	Pinned
10	F1_B10	W8X10	End Pl Shear	Shear Tab	Pinned	Pinned
11	F1_B11	W8X10	Double Clip	Double Clip	Pinned	Pinned
12	F1_B12	W8X10	End Pl Shear	End Pl Shear	Pinned	Pinned

Note:

- Connections can only be designed for hot-rolled members, so only those members are shown in this spreadsheet.

Label

These are the member labels for all hot-rolled members in the model.

Shape

This allows you to view what shape is assigned to the member.

Note:

- RISAConnection currently designs shear and moment connections for only wide flange shapes.

Start Conn/End Conn

This allows you to select a rule from the **Connection Rules** spreadsheet. You will need to know which end of the member is the Start (I end) and the End (J end), which you can view graphically.

Start Release/End Release

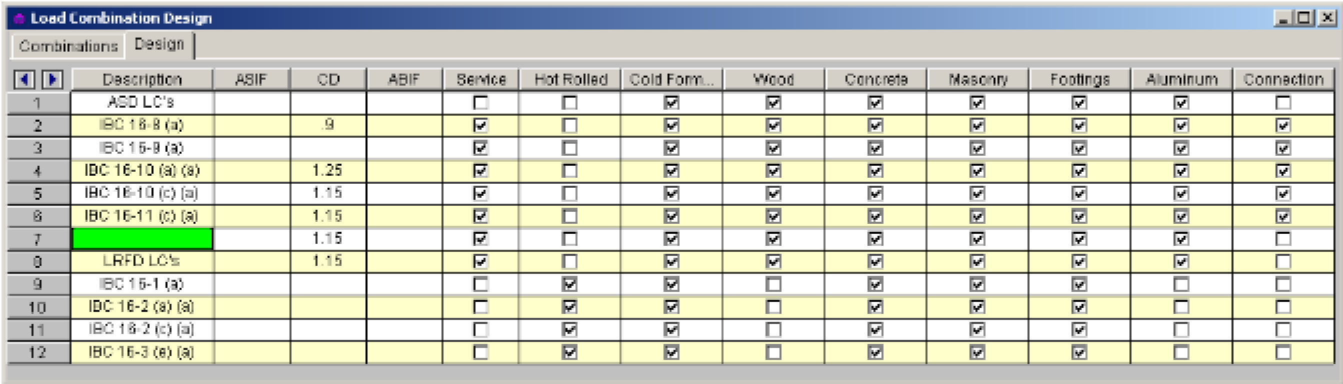
The end release for the beam is reported here. This is useful in verifying the that the chosen connection for the beam is valid for its end fixity (i.e. shear connections for pinned-end beams)

4. Assigning Load Combinations

Once you have all of your connection rules assigned properly you must define the load combinations that you wish to use for connection design.

Load Combinations - Design Tab

In the **Load Combinations** spreadsheet there is a checkbox for **Connection**. This checkbox defines whether you want your connections designed for that LC or not.

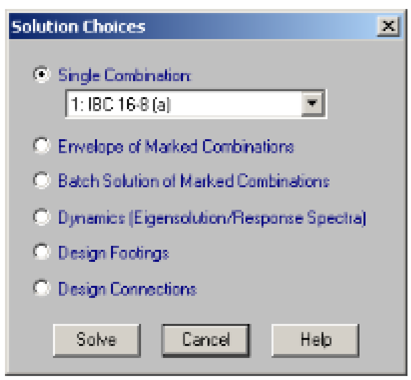


The image above illustrates why this checkbox is necessary. There are times where the member design may be designed by LRFD methods and the connections designed by ASD methods. This checkbox allows for that flexibility. The **Hot Rolled** column, which governs member design, has only the LRFD combinations selected. The **Connection** column only has the ASD combinations selected. If we run a batch solution of these LC's we will get the appropriate results.

5. Designing Connections

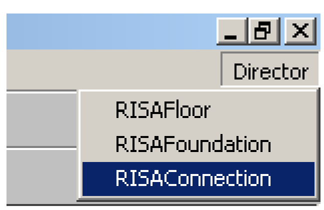
To design connections we must first have either a Single LC or Batch solution present. Once we have a solution, we have two ways to design connections.

1. Choose **Solve - Design Connections**.



This option will not automatically open RISAConnection. Instead, it will run in the background, using whatever default or previous settings are in RISAConnection. Once you've done this the **Connection Results** browser will be populated.

2. Choose **Director - RISAConnection**.



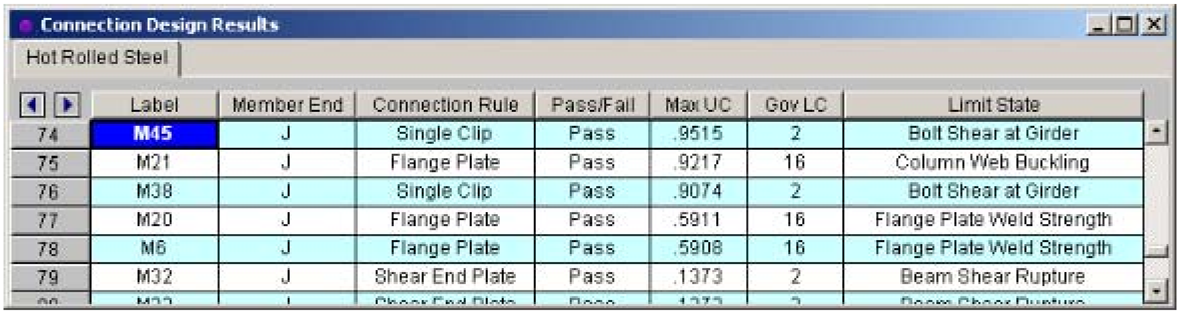
This option will automatically launch RISAConnection without designing your connections. This will allow you go modify connection parameters, such as number of bolts, clip angle size, clearances, etc., for all the valid connections in your project. You can then solve your model in RISAConnection and then send the information back to RISA-3D to populate the **Connection Results** browser.

See the RISAConnection help file for more information on how to design connections using RISAConnection.

6. Connection Results Viewing

Once the connections have been designed in RISAConnection from either of the methods above the results will be available in RISA-3D. You can view the results browser or see color-coded results graphically.

Connection Results Browser



Connection Design Results

Hot Rolled Steel

	Label	Member End	Connection Rule	Pass/Fail	Max UC	Gov LC	Limit State
74	M45	J	Single Clip	Pass	.9515	2	Bolt Shear at Girder
75	M21	J	Flange Plate	Pass	.9217	16	Column Web Buckling
76	M38	J	Single Clip	Pass	.9074	2	Bolt Shear at Girder
77	M20	J	Flange Plate	Pass	.5911	16	Flange Plate Weld Strength
78	M6	J	Flange Plate	Pass	.5908	16	Flange Plate Weld Strength
79	M32	J	Shear End Plate	Pass	.1373	2	Beam Shear Rupture

Label/Member End

These fields give the location in the model that this connection result corresponds to.

Connection Rule

Shows the Connection Rule assigned to this location.

Pass/Fail

Tells whether the connection passes ALL connection checks or not.

Max UC

Gives the maximum unity check for the worst case LC.

Gov LC

Gives which LC provided the Max UC.

Limit State

Gives the governing Limit State which produced the worst case code check or failing criteria.

When viewing this browser there is a button at the top of the screen, **Detail Results for Current Connection**.

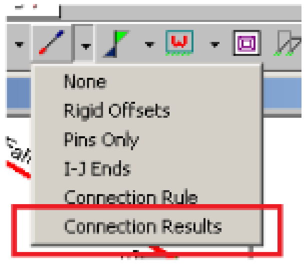
Clicking this button will open up RISAConnection to this specific connection, allowing you to take a further look at the connection results and allow you to edit connection properties in a quick, efficient manner.

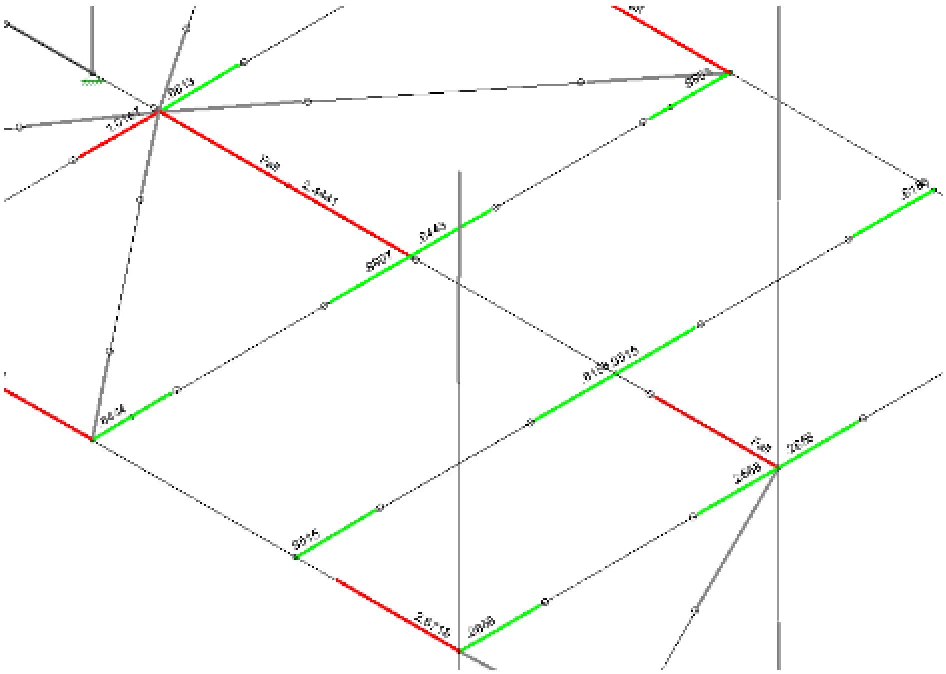
Note

- The Limit State field will also give notes for any connections that were not able to be designed in RISAConnection. See the Integration Error Messages section below.

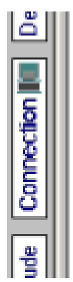
Viewing Results Graphically

Click on the Member End Display button from the Windows toolbar to view the Connection Results directly on your model, in color-coded view. The labeling will display the governing unity check value.





If you are in a graphic view and have valid connection results you can also press the Connection button on the left-hand side of the screen.



This will give a Connection cursor and allow you to click on a member end that has a Connection Rule defined and will open RISAConnection to the specific connection that you clicked on.

Other Considerations

Shear Connections

The program can handle a variety of shear connections with or without axial forces present. These connections include:

- Single or double clip-angle connections for wide flanges
- Shear tab connections for wide flanges
- End plate connections for wide flanges

Moment Connections

The program can handle a variety of moment connections with or without axial forces present. These connections include:

- Flange plate moment connections to the for wide flanges for flange connections
- Extended end plate moment connections for wide flanges with an 8-bolt configuration

Vertical Brace Connections

RISAConnection can design Vertical Brace Connections (braces in the vertical plane) when the rules below are observed.

Diagonal Brace Connections

In order to properly export a diagonal brace connection to RISAConnection you must obey the rules below:

- A Diagonal Brace connection rule must exist in the **Connection Rules** spreadsheet.
- That connection rule must be applied to the end of the VBrace which connects to the gusset
- That connection rule must also be applied to the end of the beam which connects to the gusset and to the column.
- There may be one or two VBrace members per connection. If there are two VBrace members, they must frame in from above and below the beam respectively
- There must be one beam member on the same side as the VBrace per connection.
- The VBrace(s) and the beam must share a common node, and that node must fall along the length of a column.
- The beam and VBrace(s) must exist in the same plane.
- The beam must be horizontal, and the VBrace cannot have an angle of less than 10 degrees with respect to the Beam or the column.
- The column and beam must be wide flanges, and the VBrace(s) must be single or double angles.

Chevron Brace connections

In order to properly export a Chevron Brace connection to RISAConnection you must obey the rules below:

- A Chevron Brace connection rule must exist in the **Connection Rules** spreadsheet.
- That connection rule must be applied to the ends of both VBraces which connect to the gusset
- There must be exactly two VBrace members per connection. They must frame in from the same side (above or below) of the beam.
- The VBraces must share a common node, and that node must fall along the length of a beam.
- The beam and VBraces must exist in the same plane.
- The beam must be horizontal, and the VBraces cannot have an angle of less than 10 degree with respect to the beams or the vertical plane.
- The beam must be a wide flange and the VBraces must be single or double angles.

Splice Connections

RISAConnection can design splice connections for beam and column splices. These can be either shear or moment splices. Keep in mind these items when creating your model:

- For beam to beam or column to column splices, both members must be oriented in the same direction.
- You must apply your splice Connection Rule to both sides of the splice.
- Currently column splices can only be applied in RISA-3D. This will also be available in a later version of RISAFloor.

RISAConnection File Creation and Workflow

When using RISAConnection integration, a RISAConnection file will be created automatically. The file will have the same naming convention as the RISA-3D file and will be created in the same directory as the RISA-3D file. It will have a .rcn file extension.

Once this RISAConnection file is created then you can use this file separately to make connection changes. The file can be transferred to another machine and worked on separately. Items related to the connections can be modified (bolt criteria, weld criteria, connector sizes, clearances, edge distances, etc.). You can then modify/design your connections so that they now work. Then, simply move that file back to the location where the RISA-3D model is located and then redesign connections in RISA-3D. These changes will then be considered.

Note that any items defined by the RISA-3D models can not be modified (beam/column sizes, bolted vs welded connections, design code, etc). Any of these changes must be taken care of in RISA-3D and then sent back over to RISAConnection.

Integration Error Messages

If you try to send a connection to RISAConnection from RISAFloor or RISA-3D, the program will test whether that is a valid connection. If the program finds the connection or the **Connection Rules** to be incorrect or unsupported, then the program will not design the connection and will give the error in the **Connection Results spreadsheet - Limit State** column.

The possible error messages are:

- ***Connection not supported*** : This message will occur when there is a connection configuration that RISAConnection can not design. An example would be a horizontal brace connection where brace members frame into a beam web.
- ***Invalid or missing supporting connection members***: This message will occur if the member(s) that a member is framing into is not of the proper material, shape, member type, etc.
- ***Invalid connection member material***: This message will occur if the member's material is not hot-rolled steel.
- ***Invalid member/beam slope (more than 15 degrees)***: This message will occur if the member being connected has an invalid slope. Currently, beam/column connection design is only for orthogonal connections. If the angle between members is greater than 15 degrees from orthogonal then this message is given.
- ***Invalid member/brace rotation (more than 15 degrees)***: This message will occur if the member being connected has an invalid rotation. Currently, members can only be designed at certain orientations. For example, weak axis beam to column design is not currently supported and will instead give this message.
- ***Invalid member skew (more than 15 degrees)***: This message will occur if the member being connected has an invalid skew. Currently, beam/column connection design is only for orthogonal connections. If the angle skew between members is greater than 15 degrees from orthogonal then this message is given.
- ***Connections on skewed members are not supported***: This is a similar message to the invalid member skew message.
- ***Invalid vertical brace connection***: This message will occur if there was not a valid configuration. If there is not a valid column, beam and vertical brace intersection (at proper orientation, member type, member material, etc) then you may see this message. Note that for integration you must assign BOTH the beam and the brace to the vertical brace Connection Rule.
- ***Brace angle is invalid***: For a corner brace or chevron vertical brace connection the brace angle with the column and beam must be > 10 degrees.
- ***Members have different/missing connection rules***: If there is a vertical brace connection and the beam and brace have different Connection Rules, you will get this message.

RISAFloor and RISA-3D Interaction

RISAFloor and RISA-3D both have the ability to have their connections designed with RISAConnection.

Thus, there are three possible scenarios for connections between RISAFloor, RISA-3D and RISAConnection:

1. Connections that are only in RISAFloor (Gravity connections in a combined RISAFloor/RISA-3D model).
2. Connections that are in both RISAFloor and RISA-3D (Lateral connections in a combined RISAFloor/RISA-3D model).
3. Connections that are only in RISA-3D.

The third option above does not involve RISAFloor, thus it will not be discussed here.

RISAFloor Gravity Member Connections (Scenario 1)

Connections that are gravity in RISAFloor (option 1) are only RISAFloor members, which makes this more straightforward. When designing connections from RISAFloor you get a very similar behavior to what is shown in the behavior above.

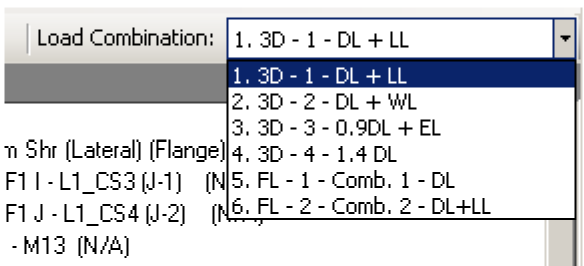
The difference is when you use the Director to go into RISA-3D and then into RISAConnection, the Gravity connections are still in the RISAConnection project. The .rcn file contains BOTH RISAFloor AND RISA-3D connections. Therefore, when entering RISAConnection from RISA-3D, you will see more connections in RISAConnection then just those sent over from RISA-3D.

RISAFloor Lateral Member Connections (Scenario 2)

Additional items to consider are:

1. **Load Combinations**

Lateral connections in RISAFloor will also come into RISA-3D. Thus, two sets of load combinations will come into RISAConnection. Below is the **Load Combination** drop-down list from RISAConnection after coming in from a RISAFloor/RISA-3D model.

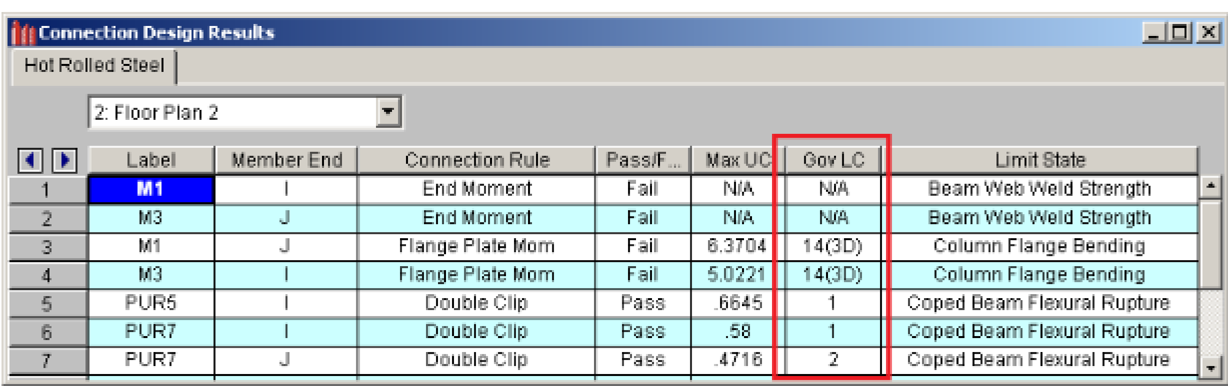


Here we can see that the RISA-3D LC's come in first followed by those from RISAFloor.

Note:

- This also occurs in scenario 1 above, except that all of the RISA-3D load combinations will have zero loading.

When reporting results in the **Connection Results** spreadsheets, both programs may list load combinations from the other program if those LC's happened to be governing. In RISAFloor it would look similar to the image below:


Connection Design Results

Hot Rolled Steel

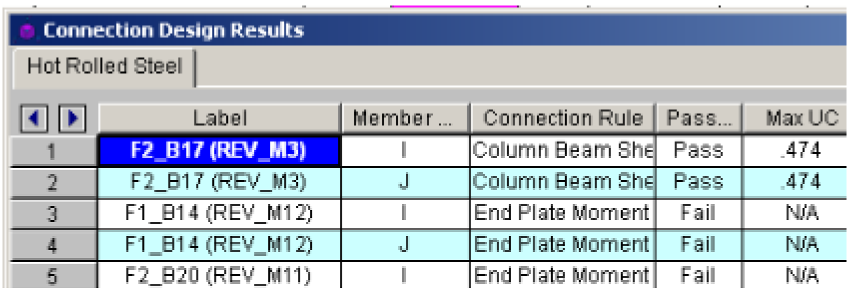
2: Floor Plan 2

	Label	Member End	Connection Rule	Pass/F...	Max UC	Gov LC	Limit State
1	M1	I	End Moment	Fail	N/A	N/A	Beam Web Weld Strength
2	M3	J	End Moment	Fail	N/A	N/A	Beam Web Weld Strength
3	M1	J	Flange Plate Mom	Fail	6.3704	14(3D)	Column Flange Bending
4	M3	I	Flange Plate Mom	Fail	5.0221	14(3D)	Column Flange Bending
5	PUR5	I	Double Clip	Pass	.6645	1	Coped Beam Flexural Rupture
6	PUR7	I	Double Clip	Pass	.58	1	Coped Beam Flexural Rupture
7	PUR7	J	Double Clip	Pass	.4716	2	Coped Beam Flexural Rupture

2. **Naming Convention**

RISAFloor and RISA-3D have different naming conventions for members. Thus, members taken from RISAFloor to RISA-3D will NOT have the same name between both programs.

RISAConnection will ALWAYS use the RISAFloor naming convention. This may be a little confusing. One way we have made this easier to comprehend is that in the Connection Results spreadsheet in RISA-3D we give BOTH the RISAFloor and RISA-3D label.


Connection Design Results

Hot Rolled Steel

	Label	Member ...	Connection Rule	Pass...	Max UC
1	F2_B17 (REV_M3)	I	Column Beam She	Pass	.474
2	F2_B17 (REV_M3)	J	Column Beam She	Pass	.474
3	F1_B14 (REV_M12)	I	End Plate Moment	Fail	N/A
4	F1_B14 (REV_M12)	J	End Plate Moment	Fail	N/A
5	F2_B20 (REV_M11)	I	End Plate Moment	Fail	N/A

In this image the F1_B?? labels are the RISA-3D labels. The labels in parentheses are the RISAFloor labels.

Multiple Round Trips Between RISAFloor/RISA-3D and RISAConnection

When round tripping between RISAConnection and RISAFloor/RISA-3D multiple times there are a few items to consider. After the first pass into RISAConnection, connection property changes in the RISAConnection file will be saved. This means that if the RISAFloor/RISA-3D model is modified and the connections are re-designed you will not lose the connection information that you have modified.

One exception to this is if you make a change to the **Connection Rule** for that member end. If that occurs then the blueprint of the connection is modified and your connection changes will be deleted and replaced with the new connection's default settings.

Thus, you are able to work on a RISAConnection file separately from the RISAFloor/RISA-3D file. You can then re-locate them back in the same directory and they will be able to work together, as long as none of the Connection Rules changed in RISAFloor/RISA-3D. Any change of an individual member end's Connection Rule will wipe out any changes made in the RISAConnection file.

Connection Types Not Currently Supported

- Non wide flange shapes (column, girder or beam)
- Moment connections into column webs
- Skewed or sloped connections. If the skew or slope is less than 15 degrees, we will transfer the connection to RISAConnection as an orthogonal connection. If the skew or slope is greater than 15 degrees than the connection will not be transferred.

Customizing RISA-3D

You may customize many of the default parameters, design and analysis options in RISA-3D. In this way you can modify the program so that it best suits you and your work processes. All customization may be defined or redefined at any time. The **Preferences** option on the **Tools Menu** provides you control over the behavior of the software. The **Save as Defaults** feature allows you to specify the default settings for new model files. These features are discussed below. Custom reports may also be defined and saved for future use. See Printing to learn how to build a custom report.

Save as Defaults

You may use the **Save as Defaults** feature in the following dialog boxes by entering the default information in the dialog and clicking the **Save as Defaults** button: **Global Parameters, Units**, and **Drawing Grids**. This will cause the program to use these settings with any new files that are then created.

Many of the spreadsheets also provide the option to save the current data as the default and every subsequent new file will already have that data. Simply enter the data you want then save it as the default by clicking on the button. This way the office standards that you might use in most of your models are already entered and available in new models. This feature is available in the following spreadsheets: **Materials**, **Custom Wood Species**, **Design Rules**, **Footing Definitions**, and **Load Combinations**.

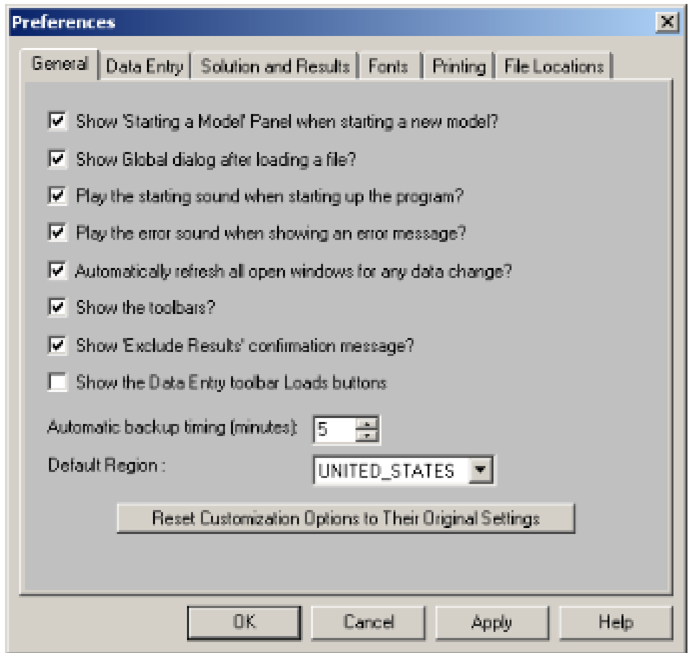
Once you create a new file you may redefine any of the default data and settings for that particular file so the **Save as Defaults** feature may be used to give you a good starting point for new files but won't hold you to those settings.

Preferences

Program options may be accessed by selecting **Preferences** from the **Tools Menu** and are divided into the five sections described below. Many of the preferences themselves are self-explanatory.

General Preferences

The general preferences are straightforward. For help on an item click and then click that item. It may be a good idea to disable the **Automatically refresh…** option when working with large files or slower computers. You may also set the backup timing. See Automatic Backup to learn about the backup capabilities of RISA-3D. The **Reset Customization Options** button will clear all of the preferences that you have set on any of the tabs.



Show "Starting a Model" Panel when starting a new model – The New Model Dialog will be displayed when opening the program or selecting 'New File' from the File Menu.

Show Global dialog after loading a file – Displays the Global Parameters settings automatically after loading a file.

Play the starting sound when starting up the program – A startup sound will be played when the program opens.

Play the error sound when showing error messages – An error sound will be played when an error is displayed.

Automatically refresh any open windows for any data change – Changes to the model will automatically be reflected in all windows – spreadsheets and model views. For large models you may want to limit the number of open windows or disable this feature altogether.

Show Toolbars – All toolbar commands may also be found in the menu system so if you want more work space you may disable the toolbars.

Show Exclude Results confirmation message – After solving the model you may use the Exclude Feature to graphically select items that you wish to see in the results spreadsheets, "excluding" other results. This enables a confirmation message warning you that some results are not shown.

Show the Data Entry toolbar Loads buttons – This provides an option as to whether you wish to see the Joint, Point, Distributed, Member Area, and Surface Loads buttons on the Data Entry toolbar.

Automatic backup timing – Automatic backup occurs at the specified interval. No backup occurs if the interval is set to zero.

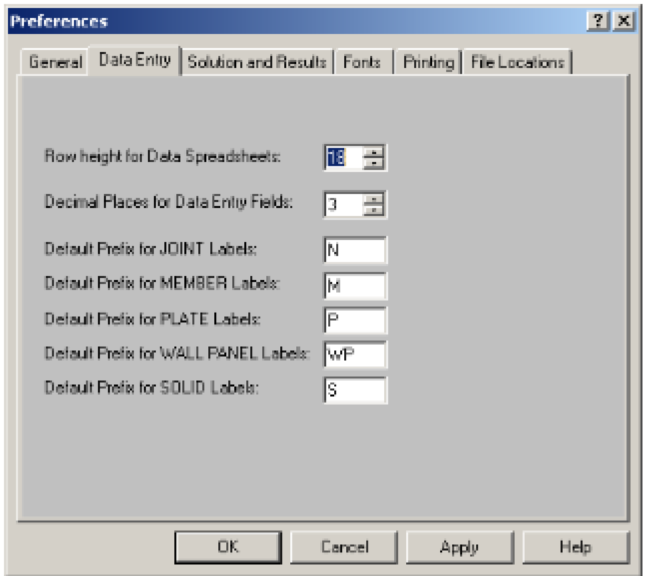
Reset Customization Options – Choose this to return to the program defaults.

Default Region - Choose the default region for your projects.

Data Entry Preferences

To use bigger or smaller fonts in the spreadsheets you may adjust the row heights. You may also specify the number of decimal places that are displayed. The one exception is the **Joint Coordinates**. RISA-3D maintains the coordinates to 15 significant figures and the exact value is always displayed. You may use the **Tools Menu** to round off joint coordinates.

If you wish to use a prefix with your joint, member, and plate labels, such as “J” with joints, you can specify the default prefix. These prefixes may be changed as you build your model.



Row height for data spreadsheets – Sets the row height and font size for data spreadsheets.

Decimal places for data entry fields – Sets the number of decimal places to display in the data spreadsheets with a maximum of four places.

Default prefix for JOINT labels – Sets the default prefix to be used in joint labels.

Default prefix for MEMBER labels – Sets the default prefix to be used in member labels.

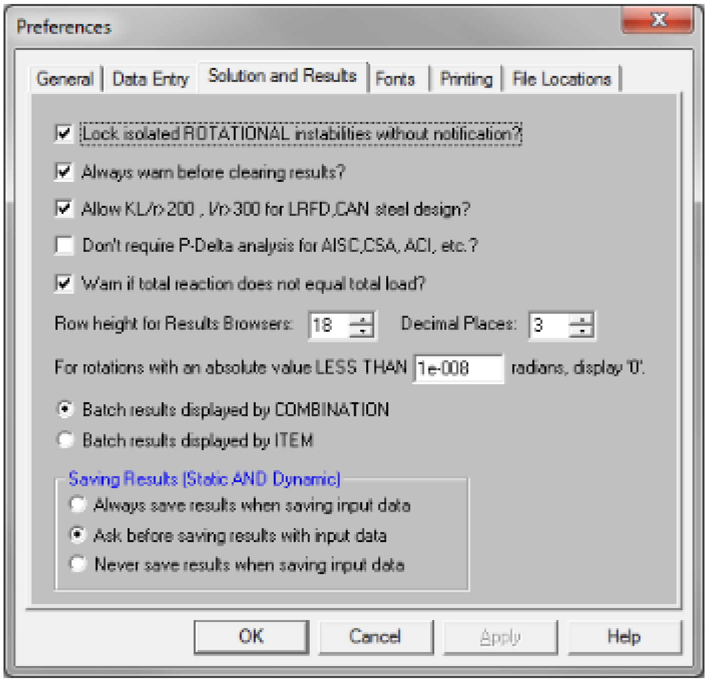
Default prefix for PLATE labels – Sets the default prefix to be used in plate labels.

Default prefix for WALL PANEL labels - Sets the default prefix to be used in wall panel labels.

Solution and Results Preferences

At solution time RISA-3D finds and locks any instabilities to allow the solution to occur. See Stability to learn more about this. Rotational instabilities are commonly inconsequential and RISA-3D allows these instabilities to be locked without any warning.

RISA-3D can provide a warning when clearing results. To use bigger or smaller fonts in the results spreadsheets you may adjust the row heights. You may also specify the number of decimal places that are displayed. The number of figures displayed may not be the actual number. Behind the scenes RISA-3D maintains numbers to numerous decimal places.



Lock isolated ROTATIONAL instabilities without notification? – Locks insignificant rotational instabilities at solution time without warning. This will cut down on joint instabilities that are technically unstable but in practice are unnecessary.

Note:

- If you are getting a Warning Log message that states "Sum of reaction is not equal to the sum of the loads", you may need to uncheck this box to look for instabilities in your model.

Always warn before clearing results? – Verifies that results are to be cleared to edit the model.

Allow KL/r>200, l/r>300 for LRFD,CAN steel design? – Waives the slender check for slender members.

Don't require P-Delta analysis for AISC, CSA, ACI, etc.? - Waives the P-Delta analysis requirement for certain design codes.

Warn if Total Reaction does not equal total load? - Enables a check in the program to confirm that the total applied load equals the sum of the joint reactions. See the Warning Log topic for more information on this.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Sum of Reactions**.

Row height for Results Browsers – Sets the row height and font size for results spreadsheets.

Decimal Places – Sets the number of decimal places to display in the results spreadsheets with a maximum of four places.

Rotation Limits – Shows 0 for the rotation when smaller than this value.

Batch Results Display - The results of a batch solution may be grouped by load combination or by item. For example you can group results for all members under each particular load combination or you can group results from each combination under a particular member. The setting here is merely a preference. Once you have solved a model you can switch back and forth using the **Results Menu**.

Saving Results – These options let you control what is done with the results when saving a file.

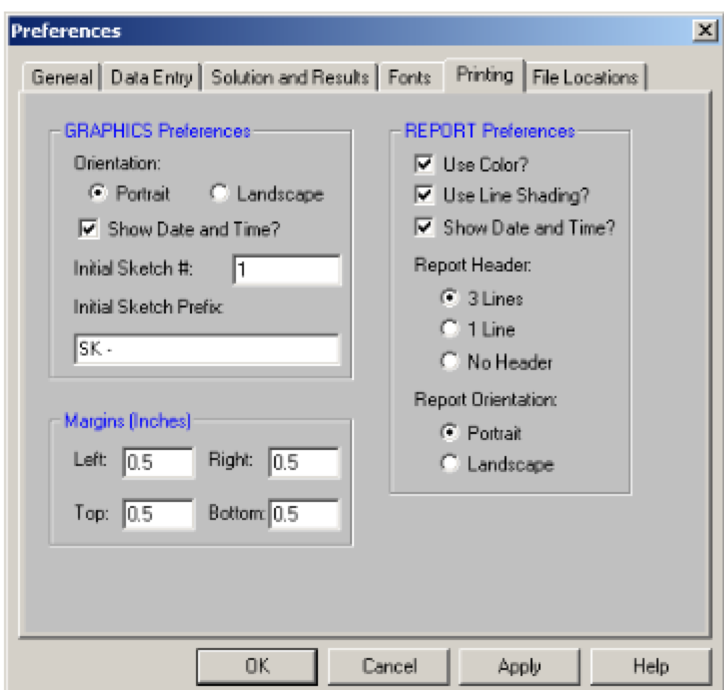
Font Preferences

The font preferences are straightforward. They can be used to adjust the fonts used by the spreadsheets, results browsers, and graphics. The font changes will affect both the on-screen displayed data and the printed data. The exceptions to this are the spreadsheet and browser fonts which may be changed for on-screen display but are hard-wired for printing purposes.

If the font data has be set to some unusual settings, then the user can click the **Reset All Fonts to Program Defaults** button to restore the fonts to what is normally expected for the RISA program.

Printing Preferences

The printing preferences are straightforward. See Printing for more information.



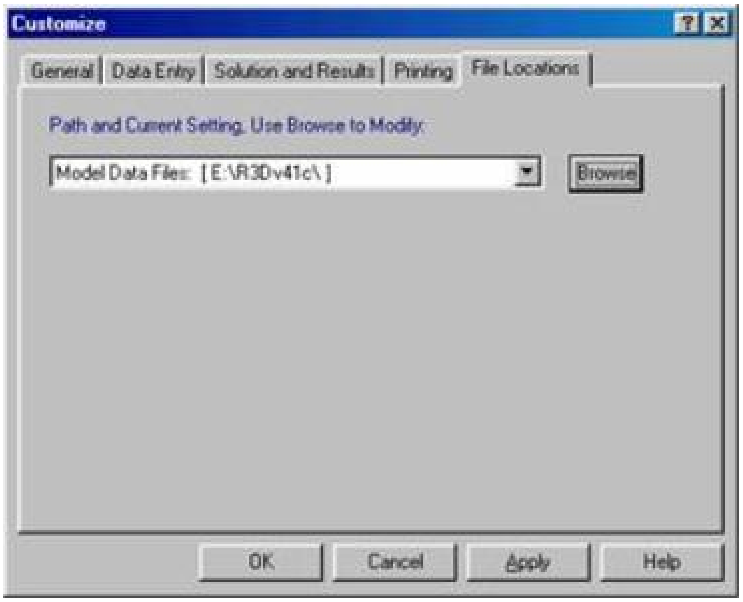
Preferred orientation for graphics – Sets the default paper orientation for graphic printing.

Margins – Sets the default printing margins.

Report Preferences – Sets color or black and white options and header options.

File Preferences

The locations for data files, databases, temporary space, importing, and backing up may be specified separately by choosing from the list.



Path and current setting – For each file type in the list the current setting is displayed. Click the drop down list to view different file types. Click the browse button to choose a different location.

Member Design Optimization

RISA-3D will optimize Hot Rolled Steel, Cold Formed Steel, Wood, and Concrete members. The criteria used for this optimization are the selected design code and the **Design Rules** assigned to the member. The sizes are chosen from a Redesign List assigned to the member.

The **Member Design Rules Spreadsheet** records the parameters for the optimization. Optimization is performed for minimum weight, taking into consideration any depth, width, rebar limitations, etc. Note that the design rules input is one large spreadsheet, thus all of your design rules will be in the same place. Note that the dimensional rules and the reinforcement rules are all a separate entity. They have no interaction with each other in the program. They are simply all input into the same location. For example, your minimum member width rules will not be influenced by concrete beam reinforcement rules.

You can assign the design rules graphically as you draw members or later as a modification. . See Modifying Member Properties for more information.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keyword: **Redesign**.

Wall Design Optimization

The program will also optimize concrete, masonry, and wood walls. For Wall Design Rule information see these topics:

- Concrete: Concrete Wall - Design Rules
- Masonry: Masonry Wall - Design Rules
- Wood: Wood Wall - Design Rules

Note:

- For the optimization procedure on concrete walls see Concrete Wall - Design.
- For the optimization procedure on masonry walls see Masonry Wall - Design.
- For the optimization procedure on wood walls see Wood Wall - Design.

Member Design Lists

A **Design List** defines a set of members that will be used in the design and optimization of a member. RISA comes with the following Pre-Defined Design Lists.

Pre-Defined Design Lists

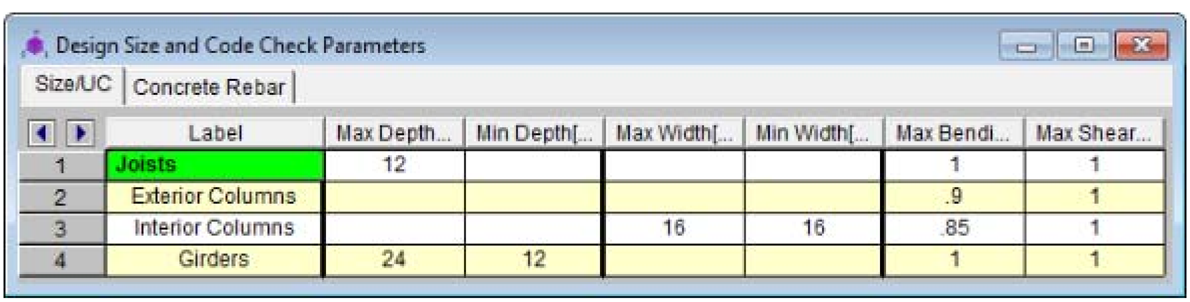
Material Type	Member Type	Shape Groups
Hot Rolled Steel	Beams	Wide Flange
		Channel
		Tube
	Columns	W14
		W12
		W10
		Tube
		Pipe

Cold Formed Steel	Beams	CU shapes
		CS Shapes
		ZU Shapes
		ZS Shapes
		HU Shapes
	Columns	CU shapes
		CS Shapes
		ZU Shapes
		ZS Shapes
		HU Shapes
NDS Wood	Beams	Rectangular – Single
		Rectangular - Double
		Rectangular – Triple
		Round
	Columns	Rectangular – Single
		Rectangular - Double
		Rectangular – Triple
		Round
Concrete	Beams	Rectangular
	Columns	Rectangular
		Round

You may edit these lists or create additional custom lists of your own. For more information on these redesign lists, including file format, editing procedure, and user defined lists refer to Appendix A - Redesign Lists.

Member Design Rules – Size / U.C.

The **Design Rules Spreadsheet** records the limitations for the design and may be accessed by selecting **Design Rules** on the **Spreadsheets Menu**. You may create and name any number of design rules and assign different rules to various members.


Design Size and Code Check Parameters

Size/UC | Concrete Rebar

	Label	Max Depth...	Min Depth[...	Max Width[...	Min Width[...	Max Bendi...	Max Shear...
1	Joists	12				1	1
2	Exterior Columns					.9	1
3	Interior Columns			16	16	.85	1
4	Girders	24	12			1	1

The spreadsheet has two tabs: **Size/UC** and **Concrete Rebar**. When a RISA-3D model is linked with RISAFloor an additional **Wood Diaphragms** tab is also available. The entries for the **Size / UC** tab are explained below:

Design Rule Labels

You must assign a unique label to the design rules. You then refer to the design rule by its label when assigning it to members. The label column is displayed on all tabs of the spreadsheet.

Max/Min Depth

You may enforce depth restrictions by setting either a maximum and/or minimum depth.

Max/Min Width

You may enforce width restrictions by setting either a maximum and/or minimum width.

Max Bending and Shear Check

Enter the maximum bending and shear unity checks. This should usually be specified as "1". If you desire a larger factor of safety, provide a lower factor (i.e. ".95").

Member Design Rules – Concrete Rebar

Concrete Rebar Parameters

Size/UC | Concrete Rebar

	Label	Min Flex Bar	Max Flex B...	Shear Bar	Legs per Stirrup	Top (Column...	Bottom Cover[...	Side Cover[in]
1	Joists	#8	#10	#4	2	1.5	1.5	1.5
2	Exterior Columns	#9	#9	#4	2	1.5	1.5	1.5
3	Interior Columns	#5	#10	#4	2	1.5	1.5	1.5
4	Girders	#5	#10	#4	2	1.5	1.5	1.5

The entries for the **Concrete Rebar** tab are explained below.

Note

- If you would like to define specific flexural and shear rebar layouts for beams and columns, see Rebar Layout Database.

Flexural Bars

Use the **Min Flex Bar** and **Max Flex Bar** columns to restrict bar sizes for your flexural reinforcing. Currently we support the ASTM A615 (imperial), ASTM A615M ("hard" metric, i.e. #8M is an 8mm bar), BS 4449 (British), prENV 10080 (Euro), CSA G30.18 (Canadian), and IS 1786 (Indian) reinforcement standards. You may specify your rebar set in the Global Parameters - Concrete. You can force the program to analyze one bar size by setting the Min and Max values to be the same bar.

Shear Ties

Use the **Shear Bar** column to enter the size of your shear ties. Currently we support the ASTM A615 (imperial), ASTM A615M ("hard" metric, i.e. #8M is an 8mm bar), BS 4449 (British), prENV 10080 (Euro), CSA G30.18 (Canadian), and IS 1786 (Indian) reinforcement standards. You may specify your rebar set in the Global Parameters - Concrete.

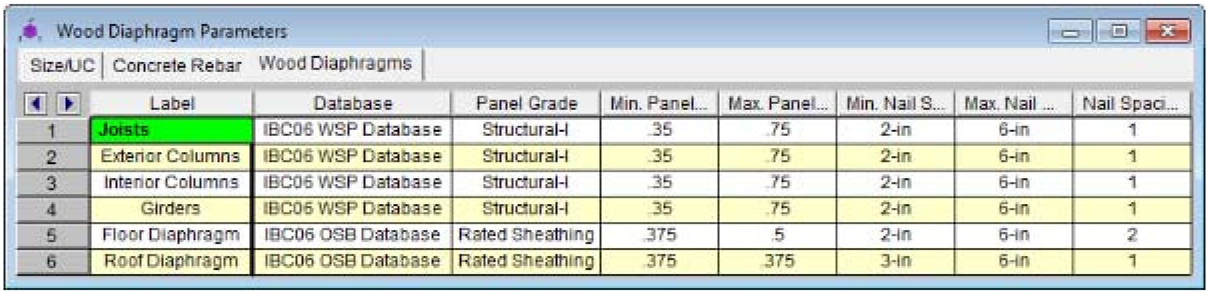
Legs per Stirrup

Use the **Legs per Stirrup** column to enter the specific information about how may legs (1 to 6) each of your shear ties is expected to have.

Concrete Cover

The last three columns are used to specify the clear cover measured to the shear reinforcing. Note that the Top Cover is used for all sides of Column members.

Design Rules - Diaphragms



	Label	Database	Panel Grade	Min. Panel...	Max. Panel...	Min. Nail S...	Max. Nail ...	Nail Spaci...
1	Joists	IBC06 WSP Database	Structural-I	.35	.75	2-in	6-in	1
2	Exterior Columns	IBC06 WSP Database	Structural-I	.35	.75	2-in	6-in	1
3	Interior Columns	IBC06 WSP Database	Structural-I	.35	.75	2-in	6-in	1
4	Girders	IBC06 WSP Database	Structural-I	.35	.75	2-in	6-in	1
5	Floor Diaphragm	IBC06 OSB Database	Rated Sheathing	.375	.5	2-in	6-in	2
6	Roof Diaphragm	IBC06 OSB Database	Rated Sheathing	.375	.375	3-in	6-in	1

Note:

- In RISA-3D the wood diaphragms tab of the Design Rules spreadsheet is only available if the model is being transferred from RISAFloor.

Database

Within the Wood Diaphragm Schedule dialog, you can select between oriented strand board (OSB) or wood structural panels (WSP) databases (plywood or OSB), which have values that are pulled directly from the 2006 IBC. You are also allowed to select between design groups based on cases and whether the diaphragm is blocked or not. You may also specify exact diaphragm parameters by unchecking the **Use Entire Case** checkbox. For more information on this database, as well as information on how to edit or create your own custom database, see Appendix F-Wood Design Databases

Panel Grade

This column allows you to specify the sheathing grade for your diaphragm. The program will then choose a diaphragm selection from the database that has this grade.

Max/Min Panel Thickness

These values set minimums and maximums for the thickness of the sheathing that will be designed. If the same value is input for both max and min, then that will be the thickness used.

Max/Min Nail Spacing

These values set minimums and maximums for the edge nailing spacing. If the same value is input for both max and min then that nail spacing will be used. These values will only control the outer-most zones of a multi-zone diaphragm region.

Nail Spacing Increment

The nail spacing increment defines how much the spacing must differ between adjacent zones. The program will design the diaphragm panel with the highest demand for the outer zone, and then step-up the nail spacing by the increment until it finds the next nail spacing that matches all other criteria. This nail spacing will be used to create the next zone.

Member Optimization Procedure

Member optimization is performed both on explicitly defined members and on members defined through the use of Section Sets. Members defined as part of a Section Set are checked to determine which member has the highest code check value and

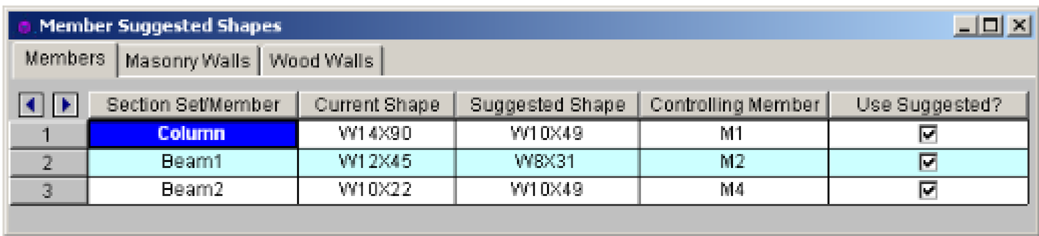
which member has the highest shear check value. These members are considered to be the controlling members for that section set.

The controlling forces on a member or a section set are then applied to new shapes satisfying the redesign parameters and a code check is calculated. If the calculated code check and shear check falls within the specified range the shape is considered to be an acceptable alternate.

Optimization Results

Suggested Shapes Spreadsheet

Access the **Member Suggested Shapes** by selecting the **Results Menu** and then selecting **Members** ▸ **Suggested Designs**. Alternatively you can click the **Suggested Design** button on the **Results** floating toolbar.



	Section Set/Member	Current Shape	Suggested Shape	Controlling Member	Use Suggested?
1	Column	W14X90	W10X49	M1	☑
2	Beam1	W12X45	W8X31	M2	☑
3	Beam2	W10X22	W10X49	M4	☑

These are the suggested shapes resulting from the optimization calculations. They are chosen from each member's assigned Design List. The suggested shape is *estimated* to most closely meet the criteria specified in the Design Rules without exceeding them.

Note:

- The suggested shape may be larger or smaller than the current shape, except for the case of members brought over from RISAFloor, for which the program never recommends downsizing.

To confirm that these alternate shapes are acceptable you **MUST** adopt any changes into the model then re-solve and check the results. The suggested shapes are based on the forces for the current model. Keep in mind that the current results are based on the stiffness of the current shapes. Changing the shapes will change the stiffness, which is why the model needs to be resolved. It may be necessary to cycle through this process a few times to achieve the best shapes.

You may try the new shapes by clicking the **Replace and Resolve**  button. The shapes listed in the **Suggested Shapes** column will only be used to update the model if the "Use Suggested?" box is checked for that particular member or section set.

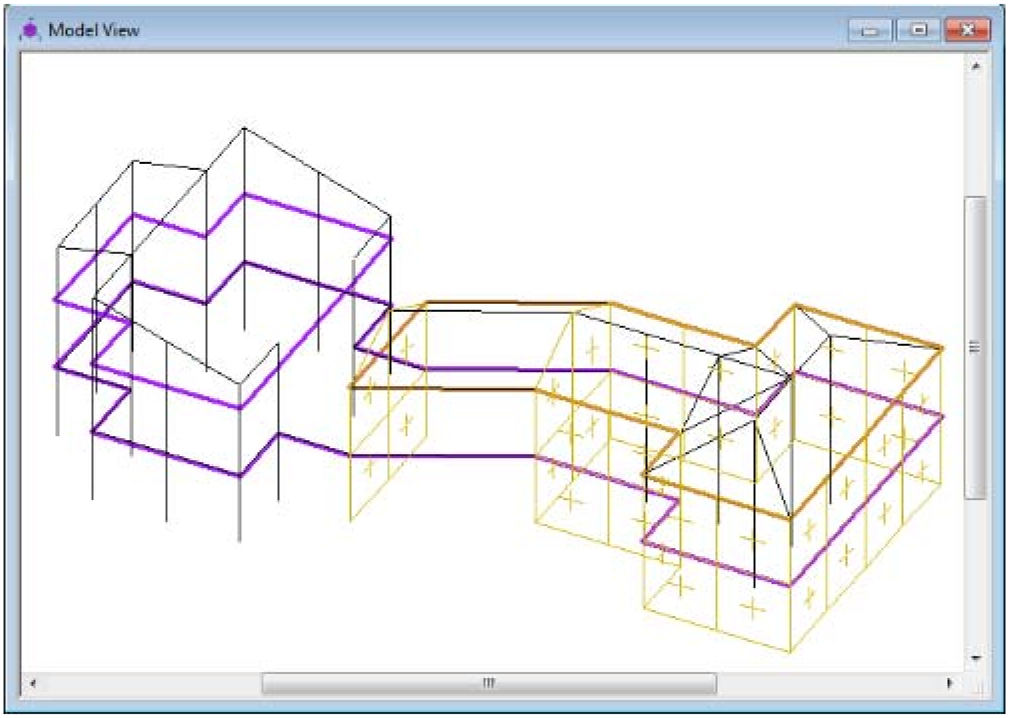
If the message "No Shapes Found" is given, then no satisfactory shapes could be found in the Design List specified for that member or section set. This can occur for a number of reasons. Common reasons are:

- The loads applied are too large for the shapes in the redesign list.
- No load combinations were checked for the design of this material type. See Load Combinations - Design Tab for more information.
- A code check could not be performed for a member in the section set. See the Design Results or the Warning Log for these members.
- The member has not been assigned an initial redesign list. Check the Members and Section Sets spreadsheets to be sure they are defined with a redesign list.
- On-line shapes (RE, PI and BAR) cannot be redesigned.
- If you've entered a minimum code check value and the members assigned to this section set are lightly loaded, it is possible that no shape generates a code check value high enough to exceed the minimum.
- A code was not specified for that material on Global Parameters - Codes.

Note:

- For masonry wall design optimization see the Masonry Wall - Design topic.
- For wood wall design optimization see the Wood Wall - Design topic.

Diaphragms



Diaphragms provide lateral load distribution functions, and are necessary for automatic Wind and Seismic load generation. There are three fundamental types of diaphragms in structural modeling:

- Rigid Diaphragms
- Semi-Rigid Diaphragms
- Flexible Diaphragms

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keyword: **Diaphragms**.

Rigid Diaphragms

Rigid diaphragms represent a plane of infinite rigidity. Rigid diaphragms distribute load to elements which connect to them solely based on the stiffness of the elements. There are two fundamental types of rigid diaphragms:

Planar Diaphragms tie all of the joints within the diaphragm plane together for both translation and rotation within the plane, as well as outside of the plane.

In a floor application, this would imply that placing a vertical load at any point on the floor would result in vertical reactions at every single node connected to the floor. Since most floor construction does not possess this infinite out-of-plane rigidity, planar diaphragms have no useful function for building analysis.

Membrane Diaphragms tie all of the joints within the diaphragm plane together for both translation and rotation, but only within the plane of the diaphragm.

This is typical behavior for most slabs and decks, which attribute vertical loads based on the tributary area of their supporting members.

Load Attribution

Loads applied within the plane of a diaphragm will be attributed to all elements which connect to the diaphragm. The amount of load which each element takes is proportional to its stiffness. Diaphragms are capable of both translation and rotation, so the torsional effects of the moment arm between the center of load and the center of rigidity are accounted for. This is also true for a dynamic mass which is offset from the center of rigidity.

Because a rigid diaphragm is part of the stiffness matrix, an explicit **Center of Rigidity** is not calculated or reported. Internally, the program creates a hidden set of Rigid Links which interconnect all of the joints in the diaphragm, therefore preserving the diaphragm's rotational degree of freedom (something which nodal slaving is incapable of).

Connectivity

All joints which fall within the plane of the diaphragm automatically become connected to it. Joints may be intentionally disconnected from the diaphragm by checking the Detach From Diaphragm box in the Joint Coordinates spreadsheet. If a boundary condition exists within the plane of a diaphragm it will act as a restraint for all of the joints connected to the diaphragm.

Rigid Diaphragms must be defined along the Global Axes, therefore they can only exist in the XY, XZ, or YZ planes. If rigid behavior is desired along a plane other than these, a semi-rigid diaphragm with a large stiffness value can be used instead.

Rigid Diaphragm Stiffness

The stiffness of the rigid diaphragm is set to a unitless value of 1×10^7 by default. This value has been calibrated as providing the best behavior for most models. It can be adjusted from within the diaphragms spreadsheet, however adjusting this value is only recommended in the following circumstances:

1. The lateral stiffness of elements which pass through the diaphragm is sufficiently large to cause the rigid diaphragm to behave as a semi-rigid diaphragm (i.e. the translations of the joints within the diaphragm do not correspond to a uniform rotation about one point). If this is the case, try a diaphragm stiffness of 1×10^8.
2. The dynamics solver is not converging. In this case try reducing the diaphragm stiffness to 1×10^6, however be sure to confirm that the diaphragm is not behaving semi-rigidly (see #1)
3. The program has generated a warning that the sum of the reactions does not equal the total applied load. In the case of points on the diaphragm which have a very close proximity to each other, the stiffness of the internally generated rigid link between them may approach the stiffness of a boundary condition. If this happens the model can have Ghost Reactions, which are points which act as boundary conditions (dumping load out of the model) without any notification. In this case try reducing the diaphragm stiffness to 1×10^6, however be sure to confirm that the diaphragm is not behaving semi-rigidly (see #1)

To adjust the diaphragm stiffness either right-click on the diaphragms spreadsheet and select Set Diaphragm Stiffness, or click on the  button.

Semi-Rigid Diaphragms

A semi-rigid diaphragm is one which distributes loads based on both the stiffness of elements which connect to it, and based off of the tributary area of these elements within the plane of the diaphragm. Neither RISA-3D nor RISAFloor have the ability to explicitly define a Semi-Rigid diaphragm explicitly. However they can be modeled using plates.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Semi-Rigid Diaphragm**.

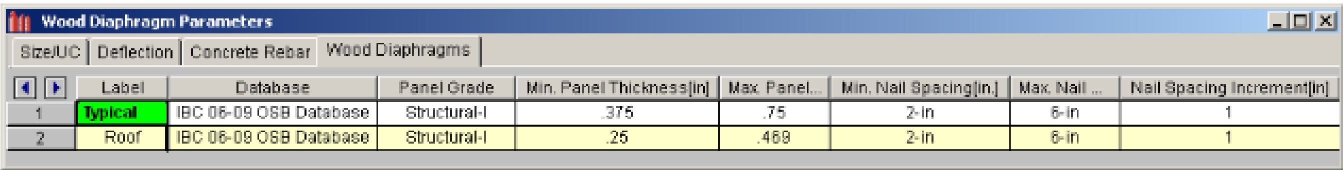
Flexible Diaphragms

Flexible diaphragms distribute loads to elements which connect to them solely based on the tributary area of the element within the plane of the diaphragm. Flexible diaphragms can be modeled in RISAFloor and can be analyzed by RISA-3D when integrated with RISAFloor.

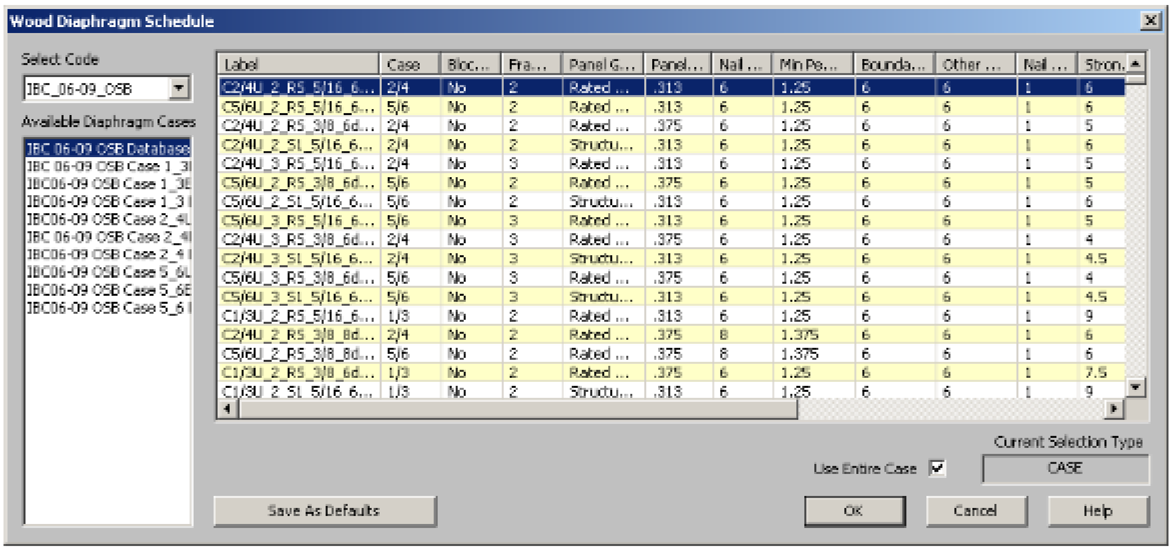
Flexible diaphragms within RISAFloor/RISA-3D are used solely as load-attribution devices, and do not exist as elements within the stiffness matrix, unlike rigid or semi-rigid diaphragms. Flexible diaphragms have no stiffness, and are incapable of transferring load from one element within them to another.

Diaphragm Design Rules

Note that these are only available in a combined RISAFloor/RISA-3D model. Here you can input the parameters for your diaphragm design. See the Design Optimization topic for more information.



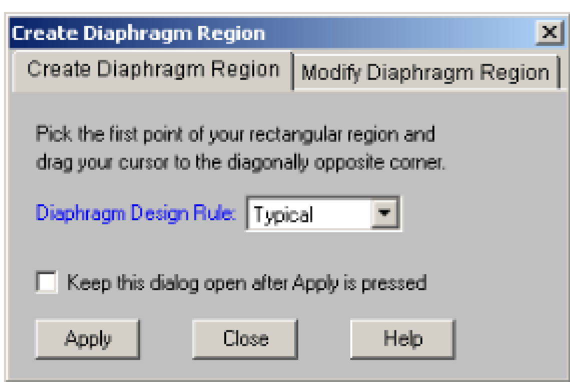
The program will use your min/max panel thickness and nail spacing to choose an appropriate nailing and panel layout from the **Database** chosen. The database options can be viewed by clicking the red arrow in the **Database** column.



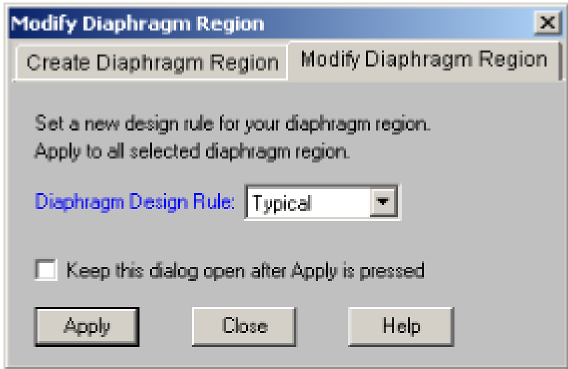
For more information on this, see the Appendix F - Wood Shear Wall Database Files topic.

Diaphragm Regions

Diaphragm Regions are used for explicitly defining load attribution, and for wood diaphragm design. They are not *required* for flexible diaphragm analysis. To draw a diaphragm region, click on the  button on the graphic editing toolbar.



A diaphragm is drawn by clicking the opposite corners of the rectangular area. For regions that have already been created it is possible to update the design rule by clicking the **Modify Diaphragm Region** tab:



Below are some limitations for drawing diaphragm regions:

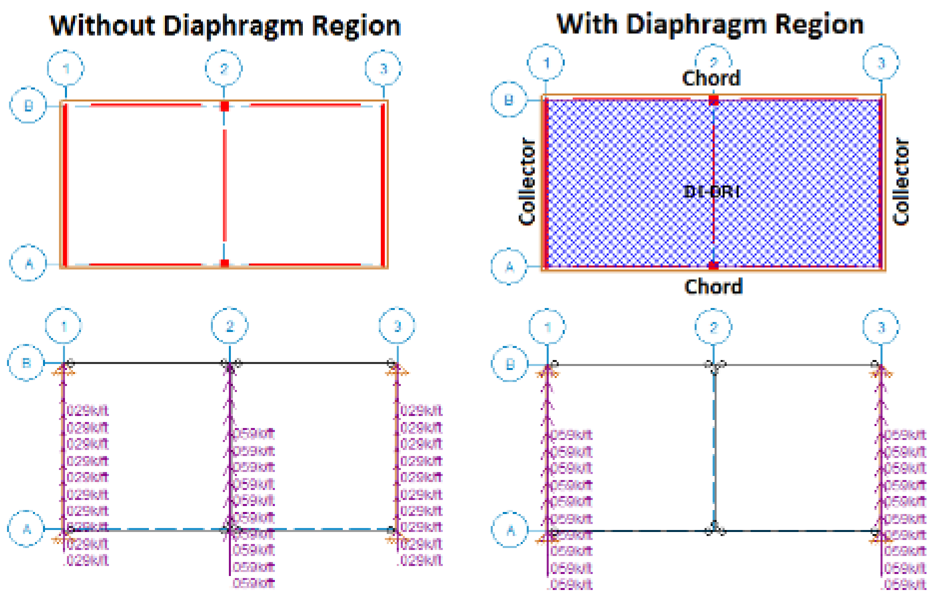
- Diaphragm Regions can only be drawn on flexible diaphragms. Therefore a Flexible slab edge must be present around the portion of the floor where the diaphragm region is drawn.
- Diaphragm Regions must be rectangular.
- Diaphragm Regions must have a closed-circuit of lateral members around their perimeter. This ensures that there will be chords and collectors for diaphragm design.
- Diaphragm Regions must be oriented along the Global X and Z axes.

Load Attribution

Only automatically generated lateral loads (wind, seismic, and notional) are considered for flexible diaphragms. Any user-defined loads are ignored.

If Diaphragm Regions have not been drawn then loads are attributed to all horizontal members within the floor plane (except members perpendicular to load direction).

If Diaphragm Regions have been drawn then loads are attributed to the perimeter of the diaphragm region, and all members within the region are ignored.



If any diaphragm regions have been drawn on a Floor then the total generated lateral load on that Floor will be attributed to the diaphragm regions, and no other members on that Floor will receive lateral loads. This allows you to use diaphragm regions to explicitly define load paths in the building.

Note:

- Openings which fall within diaphragm regions are ignored.
- When a flexible diaphragm has been defined on a sloped roof, the generated lateral loads (wall wind and roof seismic) are applied at the base of the roof. This assumes that a 'ceiling diaphragm' is present to flexibly distribute the loads. This assumption is not valid for structures which have no ceiling or horizontal bridging/bracing, such as log cabins.

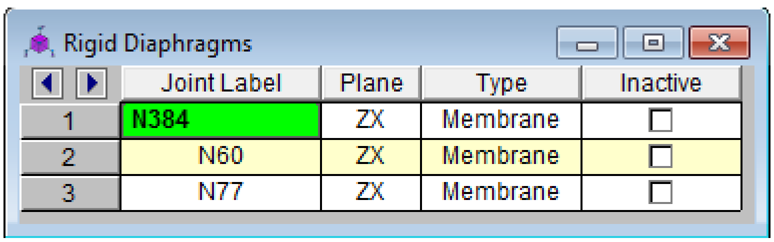
For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Flexible Diaphragm Design**.

Diaphragms Spreadsheet

The **Diaphragms** Spreadsheet contains diaphragm information and may be accessed by selecting **Diaphragms** on the **Spreadsheets** menu. Alternatively, it can be accessed by clicking the Diaphragms button on the Data Entry Toolbar.

For RISA-3D models there are two different versions of the Diaphragms spreadsheet. One version is for models which have been created in RISA-3D, the other is for models which are linked to RISAFloor.

RISA-3D Only


Rigid Diaphragms

	Joint Label	Plane	Type	Inactive
1	N384	ZX	Membrane	☐
2	N60	ZX	Membrane	☐
3	N77	ZX	Membrane	☐

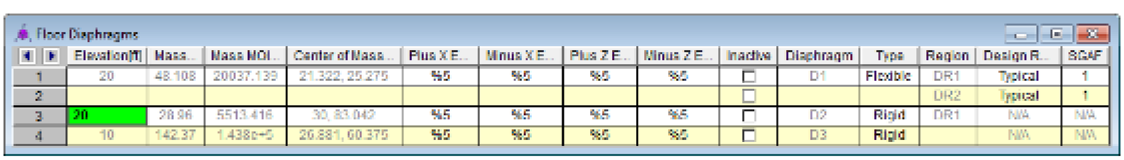
Joint Label defines the master joint for the diaphragm. For diaphragms in the ZX plane, the diaphragm will be created at the Y-coordinate of the specified joint. Similarly, XY diaphragms will use the joint's Z coordinate, and YZ diaphragms will use the joint's X coordinate.

Plane defines the plane in which the diaphragm exists. There are only three options, as a diaphragm must always fall in the plane of the Global Axes.

Type sets whether the diaphragm is to be Rigid Membrane, or Rigid Planar. There are not semi-rigid or flexible options for RISA-3D.

When the **Inactive** box is checked the diaphragm will be ignored by the program. This is a convenient way to disable diaphragms without deleting them.

RISA-3D Linked With RISAFloor


Floor Diaphragms

	Elevation[ft]	Mass...	Mass MOI...	Center of Mass...	Plus X E...	Minus X E...	Plus Z E...	Minus Z E...	Inactive	Diaphragm	Type	Region	Design R...	SGAF
1	20	48.108	20037.139	21.322, 25.275	%5	%5	%5	%5	☐	D1	Flexible	DR1	Typical	1
2									☐			DR2	Typical	1
3	20	28.96	5513.416	30, 83.042	%5	%5	%5	%5	☐	D2	Rigid	DR1	N/A	N/A
4	10	142.37	1.438e+5	26.881, 60.375	%5	%5	%5	%5	☐	D3	Rigid		N/A	N/A

Elevation displays the elevation of the diaphragm. This is the same elevation as the floor which the diaphragm was created on.

Mass displays the dynamic mass tributary to the diaphragm. This mass is used to calculate seismic forces for both static and response spectra methods.

Mass MOI displays the dynamic mass moment of inertia of the diaphragm. This moment of inertia is used to calculate seismic forces for response spectra analysis.

Center of Mass displays the X and Z coordinates of the center of mass of each diaphragm. This is the location at which static (equivalent lateral force) seismic loads are applied for each diaphragm.

Eccentricities are the percent of length/width of the diaphragm which are used to place "accidental eccentric" seismic loads for each diaphragm. This only applies to the static (equivalent lateral force) procedure. See ASCE 7-10, Section 12.8.4.2 for more information.

When the **Inactive** box is checked the diaphragm will be ignored by the program. This is a convenient way to disable diaphragms without deleting them.

Diaphragm displays the diaphragm label. This label is used for the naming of Diaphragm Regions.

Type specifies whether the diaphragm is Rigid (Membrane) or Flexible. If a diaphragm has been defined as Flexible within RISAFloor it can be toggled between Rigid and Flexible in RISA-3D.

Region lists the diaphragm regions for each diaphragm. Diaphragm regions are used for the design of wood flexible diaphragms, and are also useful for explicitly defining how flexible load attribution is to be performed.

Design Rule specifies the Design Rule which is assigned to each region. Only the information on the Diaphragms tab of the Design Rules Spreadsheet is considered.

SGAF is the specific gravity adjustment factor for the design of wood flexible diaphragms. For more information see AF&PA NDS SDPWS, Table A4.2, Note 2. This value defaults to 1, however it should be manually changed if the framing supporting the wood flexible diaphragm is not Douglas Fir-Larch or Southern Pine.

Diaphragm Modeling Tips

Partial Diaphragms (RISA-3D Only)

There may be times when you want to model a partial diaphragm, i.e., a diaphragm that extends over only a portion of a floor or plane. For example, let’s say you are trying to model a floor that is composed of a relatively rigid section (thick concrete slab) and a relatively flexible section (corrugated steel decking). You would like a way to model a rigid diaphragm for only the rigid portion of the floor.

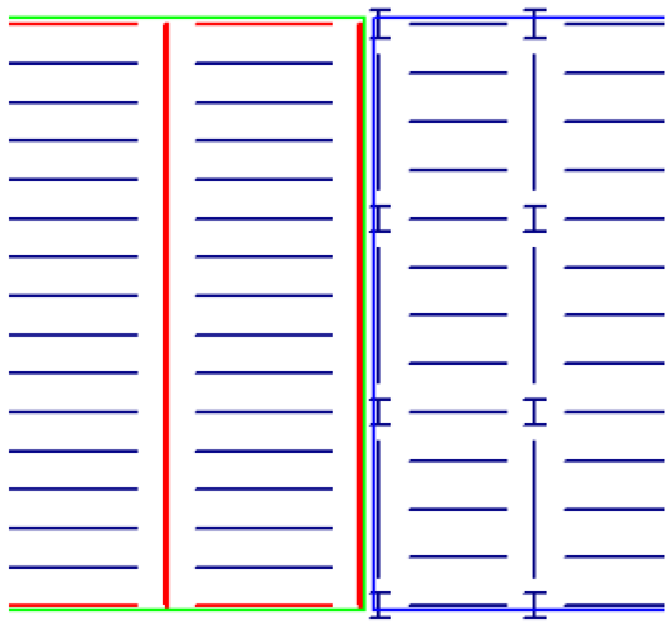
To accomplish this you may specified that a joint or group of joints be detached from the diaphragm. This may be accomplished by selecting the **Detach from Diaphragm** option in the **Joints** spreadsheet or double-click a joint and specify it in the **Information** dialog.

Another way this can be done is to offset the elevations of the joints that comprise the rigid floor section so that they are all a little higher or a little lower than the surrounding floor. (The offset only needs to be slightly larger than 0.01ft. since this is the tolerance for other joints to be on the same plane as the master joint) This works because the rigid diaphragm feature will only rigidly connect joints that are at the same elevation as the master joint. The other joints, which are on the flexible portion of the floor and are now at a different elevation than the master joint, will not be incorporated into the diaphragm.

Partial Diaphragms (RISAFloor/RISA-3D Integration)

For buildings where a flexible diaphragm and rigid diaphragm occur on the same floor you can model the diaphragms using separate slab edges. This will require a gap between the framing of the two diaphragms however, such that load will not automatically be shared between the diaphragms.

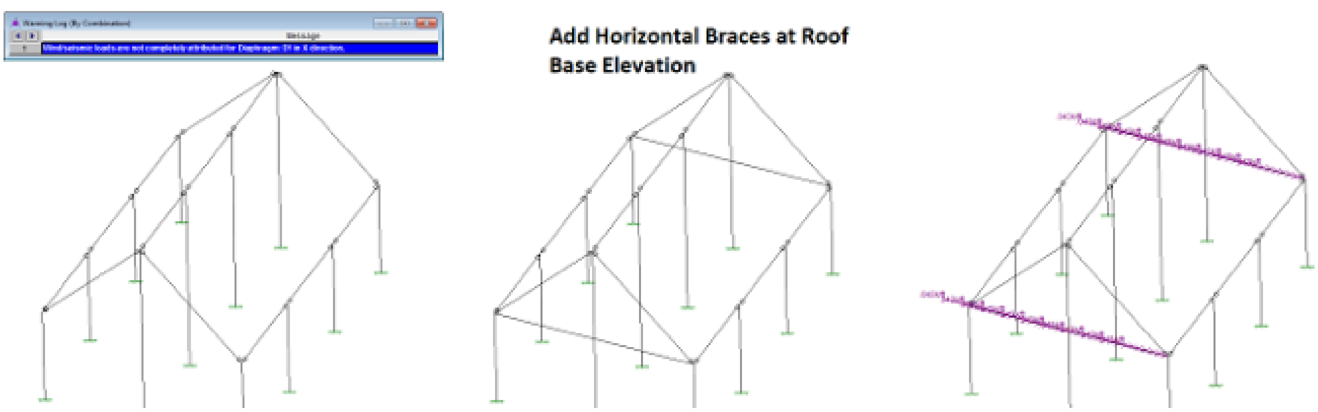
Below is a screenshot of the gap for example:



Sloped Roof Flexible Diaphragms (RISAFloor/RISA-3D Integration)

The flexible diaphragms at sloped roofs require members that are in the horizontal plane to attribute load to. These members must exist at the base roof elevation. For that reason in the example below the program reports "Loads are not attributed for Diaphragm". In the direction perpendicular to the ridge there are no members for the program to attribute the wall wind loads to, so no loads are attributed to the diaphragm at all in that direction.

To correct this issue, simply draw horizontal bracing in the structure which can pick up the load and transmit it to the main lateral force resisting system.



For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Sloped Roofs**.

Diaphragms - Analysis and Results

Flexible diaphragms can only be designed in RISA-3D if they have come in from an integrated RISAFloor / RISA-3D model. This is because RISAFloor contains information about slab edges, deck direction and geometry that cannot be entered in RISA-3D. To get this design, however, you must create diaphragm regions in RISAFloor. See the **Diaphragms** topic in the Flexible Diaphragms section for more information.

Analysis / Loading

Seismic Load

The program only applies seismic loads to flexible diaphragms that were generated using the seismic load generator. The load is internally converted into a one way member area load where the direction of the load attribution is perpendicular to the direction of the applied load. This internal conversion is done at solution time. This area load then gets broken down into a series of "transient" distributed loads that are applied to the members which support the diaphragm. These transient loads can be viewed as Basic Load Cases. Refer to Member Area Loads for more information about how these transient loads are generated.

Wind Loads

The program only applies wind loads to flexible diaphragms that were generated using the wind load generator. The load is internally converted into a distributed load at the front face of the diaphragm. Then it is broken into area loads of varying magnitudes depending on the depth of the diaphragm. In this way, an L shaped building will have less load per foot applied at the deep section of the L than the skinny section, since an equal amount of wind from each side is spread over more floor area in the deeper section.

These area loads then gets broken down into a series of "transient" distributed loads that are applied to the members which support the diaphragm. These transient load can be viewed as Basic Load Cases. Refer the section on Member Area Loads for more information about how these transient loads are generated.

Note:

The shear capacities of the diaphragm panels will automatically be multiplied by 1.4 for wind forces if theWind ASIF function is enabled.

Diaphragm Results - Detail Reports

Diaphragm results can be accessed by clicking “Diaphragms” within the Results menu. This opens the Diaphragm Detail Report dialog, from which a diaphragm can be chosen to review its results.

The detail report gives detailed information about the diaphragm design. It is split into four main sections: input echo, diagrams and design, design details, and chord forces.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Flexible Diaphragm Design**.

Input Echo

Below is the input echo portion of the detail report:

CRITERIA		**GEOMETRY**		
Code	: NDS 2005 ASD	Total Length	: 50	ft
Design Rule	: Typical	Total Width	: 60	ft
Panel Grade	: Other	L/W Ratio	: .833	
Panel Schedule	: IBC06 WSP Database	Elevation	: 22	ft
SGAF	: 1			

Code – Reports the code used to design the diaphragm.

Design Rule - Lists the Design Rule which has been assigned to this diaphragm region.

Panel Grade – Reports the grade of the panels used in the design (Structural-I, Rated Sheathing, or Other)

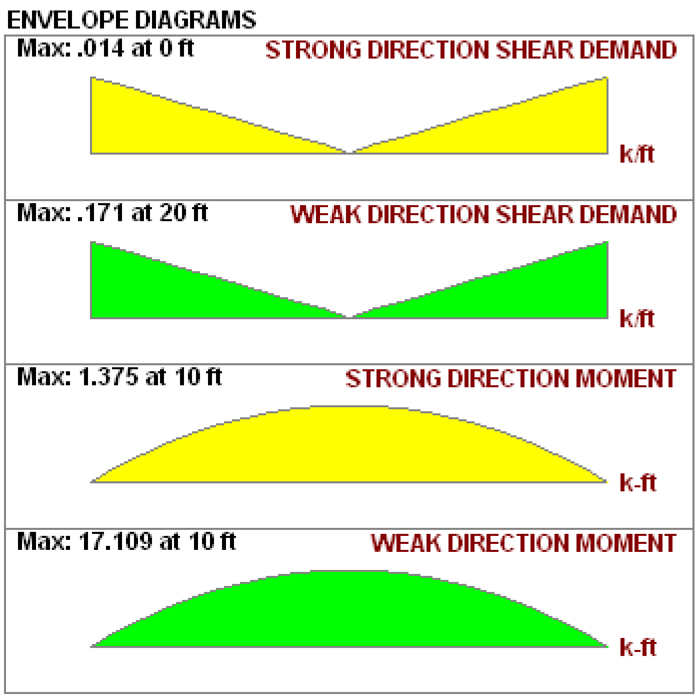
Panel Schedule – Reports the nailing schedule database used to optimize panel selection

SGAF - The specific gravity adjustment factor, input in the Diaphragms spreadsheet, reduces the capacity of the diaphragm. Note 2 from Tables A.4.2A-A.4.2C of the NDS 2005 SDPWS gives the calculation of this value as: SGAF = (1 - (0.5 - G)), where G = Specific gravity of the floor framing members.

In addition the basic geometry information for the diaphragm (**Total Length**, **Total Width**, **Elevation**, and **L/W Ratio**) are also reported here.

Envelope Diagrams

The detail report presents envelope diagrams for the shear and moment demand seen on the flexible diaphragm for both the strong and weak directions. The diagrams are color coded yellow for strong direction and green for weak direction for quick reference.



Design Summary

The results for design summary are displayed for both strong and weak directions as shown below.

DESIGN SUMMARY

STRONG DIRECTION		
Panel Required Capacity	: .355	k/ft
Panel Provided Capacity	: .36	k/ft
Ratio	: .986	
Governing LC	: 1 (Seismic)	

WEAK DIRECTION		
Panel Required Capacity	: .082	k/ft
Panel Provided Capacity	: .504	k/ft
Ratio	: .163	
Governing LC	: 2 (Wind)	

UNSCALED DEFLECTIONS (Without Fp)

STRONG DIRECTION		
Maximum Shear	: .341	k/ft
Flexure Component	: .004	in
Shear Component	: .655	in
Total	: .659	in
Governing LC	: 1	

WEAK DIRECTION		
Maximum Shear	: .059	k/ft
Flexure Component	: .001	in
Shear Component	: .135	in
Total	: .136	in
Governing LC	: 2	

Panel Required Capacity is the scaled maximum design shear, also reported on the envelope diagram. The maximum shear as determined from analysis is compared against Fp for seismic loads. If Fp exceeds the shear as determined by analysis then that shear is scaled up to match Fp. For more information see ASCE7-10, Section 12.10.1.1

Panel Provided Capacity is the capacity of the designed diaphragm. For diaphragms with multiple nailing zones, this will be reported as capacity at the point of maximum shear. For more information on how the program selects between the various nailing schedules provided in the database, please refer to Appendix F.

The **Ratio** provides the code check for the diaphragm based on shear demand versus shear capacity.

Governing LC gives the load combination that controls the design. This is based on whichever load combination resulted in the highest ratio for shear capacity. In parentheses it is stated whether the governing load combination was seismic or wind based. If the panel was governed by wind, then you are allowed to use a 40% increase in panel strength. The program will automatically consider this, as long as you have checked the **Increase Nailing Capacity for Wind** checkbox in the Global Parameters-Solution tab.

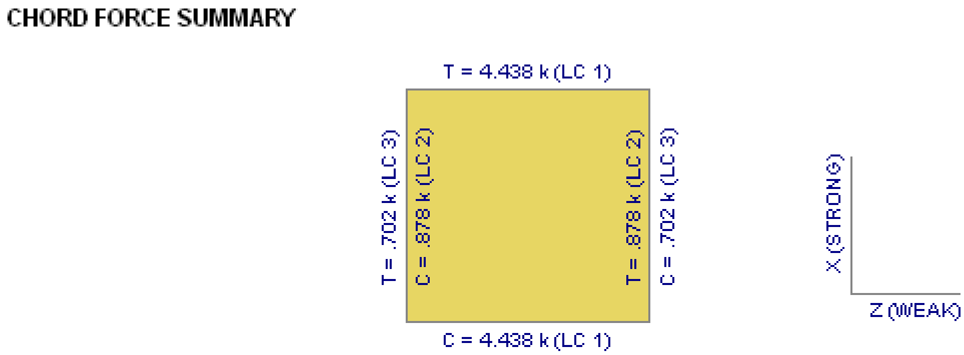
The **deflection** of a diaphragm is comprised of two main terms, depending on the equation used. The terms are as follows:

The **Flexure Component** accounts for the deflection of the diaphragm based on bending stresses. Refer to Diaphragm Deflections for more information.

The **Shear Component** accounts for the deflection of the diaphragm based on shear stresses. Refer to Diaphragm Deflections for more information.

This deflection is based on the actual maximum shears in the diaphragm, which are given. It is not based on the scaled Fp forces, thus we report the unscaled shears.

Chord Force Summary



The chord force summary gives an overall view of the chords on all sides of the diaphragm design region. The chord force calculation is simply the calculated moment divided by the length of the diaphragm (M/L). As the diagram above shows, we will give both tension and compression maximum values, if there are both tension and compression forces for the solved load combinations. The global axes are shown as a reference.

Design Details

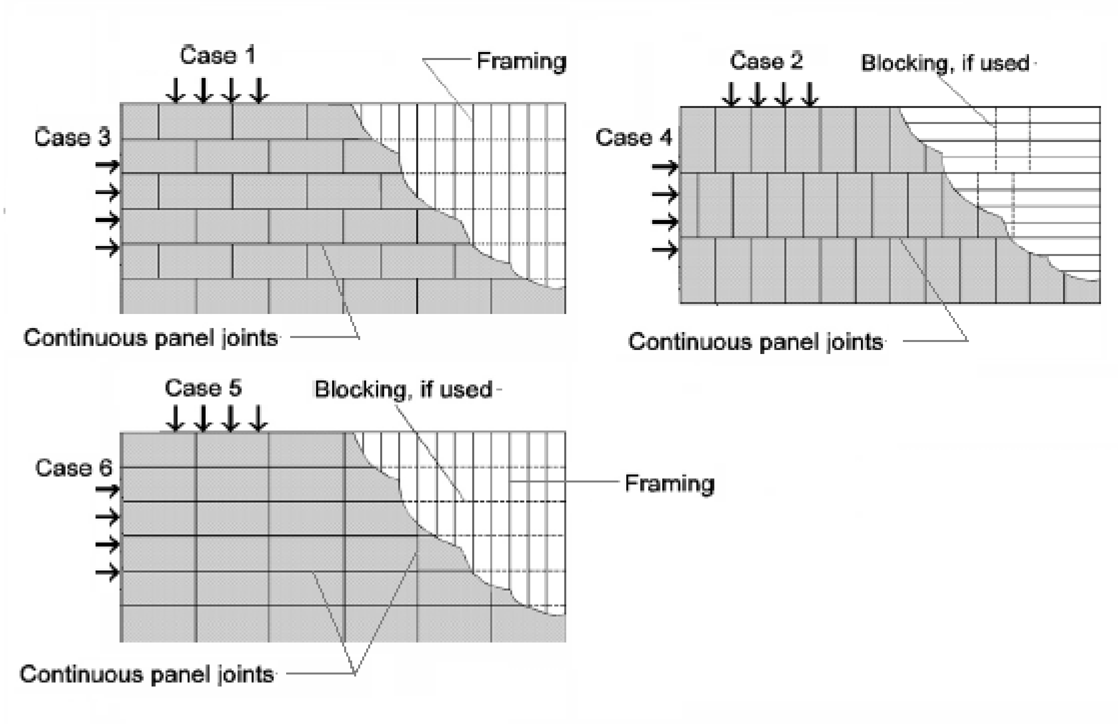
The details of the designed panel are reported here:

DESIGN DETAIL

Panel Label	: C1/3B_2_0t_15/32_8d@4/6/1	
Layout Case	: 1/3	
Blocked	: Yes	
Panel Thickness	: .469	in
Required Nail Penetration	: 1.375	in
Nail Size	: 8 d	

Panel Label - This is the call-out that the program uses to describe the specific panel and nailing. This information is expanded upon afterward.

Layout Case- This reports the case (panel layout) used for the diaphragm design. Because each diaphragm has a strong and weak direction, the program will actually report two cases associated with each diaphragm.



Blocked – This reports whether the diaphragm was designed as blocked or unblocked

Panel Thickness – This reports the decimal thickness of the wood panels. For convenience the following table lists the decimals that correspond with common panel thicknesses:

Panel Thickness	Decimal Equivalent
5/16	0.3125
3/8	0.375
7/16	0.4375
15/32	0.4688
19/32	0.5938

Required Nail Penetration – This is the minimum nail penetration as required by the IBC/NDS. This value is taken from the diaphragm nailing schedule/database. For more information on the database refer to Appendix F.

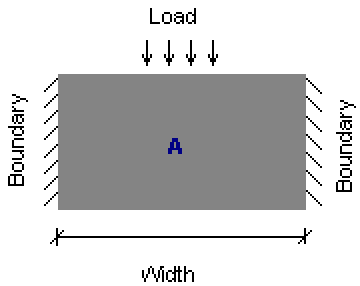
Nail Size - This is the size of the nail required for this call-out.

Nail Spacing Schedule

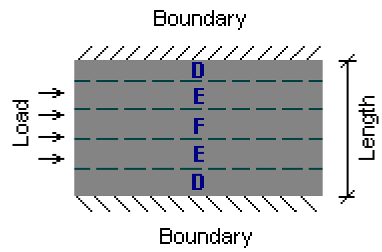
A diaphragm region is split into various nailing zones. The optimum nailing is chosen from the diaphragm nailing database for the highest point of shear of the diaphragm region in each direction. The program considers the Nail Spacing Increment as defined in the Design Rules. If a nailing arrangement is available that has spacing wider than the sum of the nailing increment and the spacing in the "higher" zone then a new nailing zone can be created. The arrangement of the nailing zones is such that they line up with the threshold shears of the capacities of the zones.

NAIL SPACING SCHEDULE							
Zone	Location (ft)	Label	Lines	Framing Width (in)	Boundary (in)	Cont Edge (in)	Other Edge (in)
A	0	C1/3B_2_Ot_15/32_8d@2/3/1	1	2	2	2	3
D	0	C1/3B_2_Ot_15/32_8d@2/3/1	1	2	2	2	3
E	9.496	C1/3B_3_Ot_15/32_8d@4/6/1	1	3	4	4	6
F	14.622	C1/3B_3_Ot_15/32_8d@6/6/1	1	3	6	6	6

LAYOUTS



Strong Direction



Weak Direction

The nail spacing schedule reports the nail spacing for each zone. The zones are shown on a legend below.

Note:

- If there is only one zone defined in each direction, their default names will be A and D.

The following information is shown in the nail spacing schedule:

The **Zone** gives the zone label so you can compare it to the zone layout below.

The **Location** defines the distance to that zone from the edge of the diaphragm. Note that this is a symmetric distance from either side of the diaphragm.

The **Label** is giving the call-out for the diaphragm panel.

The **Lines** entry gives the number of lines of nails on each panel edge. Normally, this value is greater than one only for High-Load diaphragms. This entry is taken directly from the diaphragm nailing schedule database and is not otherwise used in the diaphragm design.

Note:

- If there is only one zone, the location value will just be zero.

The **Framing Width**displays the nominal width required of the framing members that support the diaphragm panels. This entry is taken directly from the diaphragm nailing schedule database and it not otherwise used in the diaphragm design.

The **Boundary** column gives the required boundary edge nail spacing.

The **Cont Edge** column gives the required nail spacing at continuous boundary edges that are parallel to the direction of load.

The **Other Edge** column gives the required nail spacing at all other edges. i.e. edges that are not considered to be boundaries or continuous edges parallel to load.

Collector Forces

The collector forces are not currently reported in the Detail Report. However, the applied seismic forces are considered in the normal member design of the collectors. The only issue here is that the seismic forces have not been amplified by the seismic overstrength factor (Ω).

Deflection Calculations

Flexure Component

If you think of the diaphragm as a beam with the chord members acting as the flanges, then this is the deflection due to the tension and compression forces that develop in the chords.

$$\delta_{flex} = \frac{5vL^3}{8EAW}$$

v = Maximum shear load in diaphragm region

L = Diaphragm dimension perpendicular to the direction of applied load

E = Chord modulus of elasticity

A = Chord cross-sectional area

W = Diaphragm dimension parallel to the direction of applied load

Shear Component

If you think of the diaphragm as a beam with the diaphragm sheathing acting as the web of the beam, then this component represents the deflection due to the shear forces in the sheathing. This term is based on the apparent shear stiffness (Ga) as described in the NDS *Special Design Provisions for Wind and Seismic*. As such, this term includes the effects of both elastic shear deformation of the sheathing and nail slip of the panels.

$$\delta_{shear} = \frac{0.25vL}{1000G_a}$$

v = Maximum shear load in diaphragm region

L = Diaphragm dimension perpendicular to the direction of applied load

Ga = Apparent diaphragm shear stiffness

For diaphragm regions that have only one zone in each direction the value of Ga is taken directly from the nailing database. For multi-zone diaphragm regions the program internally calculates an equivalent Ga using the formula below:

$$G_a = \frac{1.4v_s}{\frac{1.4v_s}{Gt} + 0.75\alpha e_n}$$

vs = Diaphragm shear capacity (ASD value taken from nailing database)

Gt = Shear stiffness of panel depth (taken from nailing database)

en = Nail slip (taken from nailing diaphragm nailing schedule / database)

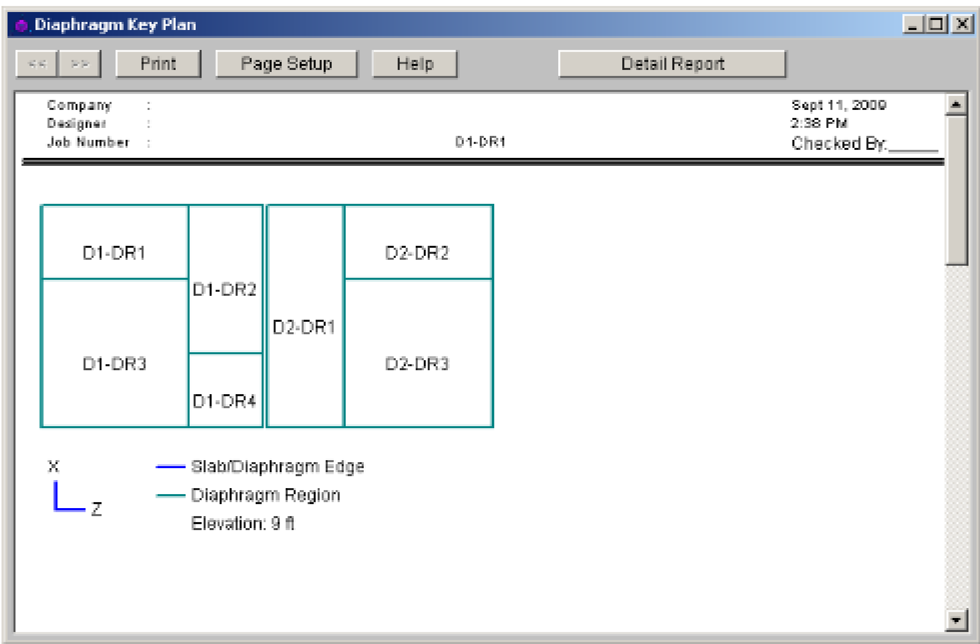
α = Nail slip adjustment factor. This factor accounts for non-uniform nail slip across multiple zones, and is derived from the process outlined in the ATC-7 and further explained in the APA design / Construction Guide on Diaphragms and Shears Walls.

Diaphragm Key Plan

The diaphragm key plan is just a plan view of the diaphragms at a given level. The key plan can be accessed by clicking “Diaphragms” within the Results menu. This opens the Diaphragm Detail Report dialog, from which a diaphragm can be chosen to review its results.

Once you have chosen a specific diaphragm you can view the Key Plan for all of the diaphragm regions at that level by pressing the "Diaphragm Key Plan" button 

.



The key plan will list each diaphragm (D1, D2, etc.) and the diaphragm regions within (DR1, DR2. etc.). This key plan is to be used as a summary in RISA-3D for where the diaphragm regions are located within the floor plan. The elevation is also given.

In the future this key plan will give a summary of the diaphragm designs at this level.

Diaphragm Design Limitations

Rigid Diaphragms - Currently, the program does not provide any design or code check information for Rigid diaphragms. The rigid diaphragms are solely used to attribute load to the supporting frames or walls.

Unblocked Diaphragm - The program will not optimize multiple nailing zones for unblocked diaphragms.

Chord Slip - The deformation due to chord slip is not considered in the deflections calculations.

Required Framing Size - The detail report lists a required framing nominal width, however no check is performed to ensure that the framing modeled actually meets this requirement.

Diaphragm Loading - The only load seen by flexible diaphragms is the load coming directly from the automatically generated loading. Any loading added in RISAFloor or RISA-3D will not be considered in the design of the diaphragms and will not be attributed to the shear walls.

Sloped Diaphragms - Currently the program will not design sloped flexible diaphragms. At solution in RISAFloor the program will force you to change the diaphragm to rigid. This feature will be added in an upcoming release.

Moisture Content - The program is not considering a reduction in strength due to moisture content.

Orientation - No diaphragm design will be done for regions placed over deck that is not parallel to the Global X and Z axes.

Drift

You may calculate and report inter-story drift based on calculated joint displacements. Simply specify which joints represent which stories. You can have the drift calculations done for any or all of the three global translation directions (X, Y and Z). Once the solution is performed you may view the drift results in the **Story Drift** spreadsheet.

Joint Boundary Conditions

	Joint Label	X [k/in]	Y [k/in]	Z [k/in]	X Rot.[k-ft/r...	Y Rot.[k-ft/r...	Z Rot.[k-ft/r...
1	N14	Story 1					
2	N34			Story 1			
3	N39	Story 2					
4	N59			Story 2			
5	N64	Story 3					
6	N86			Story 3			

To Define a Story for Drift Calculation

- On the Boundary spreadsheet specify a joint label and, for the translational boundary condition for the desired direction, enter **Story nn** , where nn is the story number.

Note

- If a story 0 is NOT defined, the base height and displacement values are assumed to be zero (0.).
- If a story is skipped (not defined), then there will be no calculations for both that story and the following story.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Story Drift**.

Drift Results

Access the **Story Drift** spreadsheet by selecting it from the **Results** Menu. This report lists the drift for all defined stories.

Story Drift for LC 6:X + Z Lateral

	Story	Joint(X)	X Drif...	% of Ht	Joint(Y)	Y Drif...	% of Ht	Joint(Z)	Z Drif...	% of Ht
1	1	N14	1.872	.867				N34	.485	.225
2	2	N39	1.735	.803				N59	2.905	1.345
3	3	N64	.844	.391				N86	1.957	.906

To calculate inter-story drift for a particular direction, the previous story displacement is subtracted from the current story displacement. For example, to calculate X direction drift for story 2, the X displacement for the joint representing story 1 is subtracted from the X displacement for the joint representing story 2.

As for story heights the vertical axis is used to determine the distance. For example when the **Y-**axis is specified as the vertical axis, the story joint Y coordinate values are used to calculate heights for X and Z direction drift. If Y direction drift is being calculated, the Z coordinate values for the story joints are used to calculate heights.

If you wish, you may define a "STORY 0" joint. If defined, this story 0 joint's displacement and coordinate values will be used for the story 1 calculations. No drift calculations will be performed for story 0. If a story 0 is NOT defined, the base height and displacement values are assumed to be zero (0.). In this case, the coordinate value for the story 1 joint is used as the story 1 height, and the displacement is used as the story drift.

If a story is skipped (not defined), then there will be no calculations for both that story and the following story. For example, say we define stories 1, 2, 4, 5 and 6. We don't define a story 3 joint. When we view the drift report, there will be no results for story 3, and there also will be no results for story 4, since story 4 depends on the story 3 values.

DXF Files

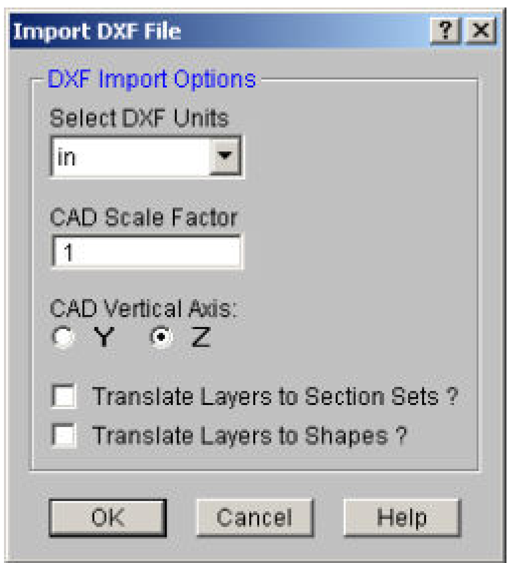
You may read and writeexport DXF files. Generally, you would read in a DXF file to create the geometry for a new structural model, or you could write out a DXF file from an existing model to form the basis for a model or a CAD drawing. This feature provides two-way compatibility with any other program that can read and write DXF files. This includes most major CAD programs and many analysis programs.

Note

- Perform a Model Merge on any model created from a DXF file. See Model Merge for more information.
- You may want to round off the joint coordinates after importing a DXF file. You may do this from the Tools menu.
- When importing a DXF file it is essential to specify a column layer. Only beams that are fully supported will be imported.

Importing DXF Files

You may translate POINT's, LINE elements and 3DFACE's. POINT's are converted into joints, LINE's are converted into members and 3DFACE's are converted into plates. Circles, arcs, polylines, text, etc. may be present in the DXF file, but these will be ignored. At this time, only the basic geometry will be translated via DXF files. You have several options available for controlling how DXF files are imported. They are as follows:



DXF Units

Select the units you used in the CAD model from which you produced the DXF file. The supported DXF units are none, inches, feet, mm, cm, and meters.

CAD Scale

Enter the scale factor that will cause the DXF file to be scaled up or down to full scale. For instance, if you had created a scaled model in AutoCAD at a scale of 1/4"=12", then the appropriate scale factor to produce a full size RISA-3D model would be 48. The default is 1.0.

DXF File Vertical Axis

Although it is not specifically noted in the AutoCAD documentation, the implied default vertical axis is the positive Z-axis of the current User Coordinate System.

The default vertical axis in RISA is usually the positive Y-axis and may be specified on the **Global Parameters**. When you import your model from a DXF file, you can have the program automatically rotate your geometry so that the Y axis is now the vertical axis for your RISA model.

Translate Layer Names to Section Sets

This is a Yes/No check box. If you check the box "Yes", the program will translate the DXF file's layer names into RISA Section Sets Labels. The program requires that you add a prefix to each Layer Name to designate what type of material that section set is defined for. The prefixes are as follows:

Material Type	Layer Prefix
Cold Formed Steel	CF_
Concrete	CN_
Hot Rolled Steel	HR_
General Materials	GN_
Wood	WD_

For example, let's say you have designed a structure with Hot Rolled steel section sets that you want to call "Column"and " Girder", as well as a Wood section set called "Joist". If you do not prefix your section sets then they will all be imported as General Material section sets. To have them imported into the proper Material type the column layer would have to be named "HR_Column", the girder a layer "HR_Girder", and the joists layer "WD_Joist".

Translate Shapes to Layer Names

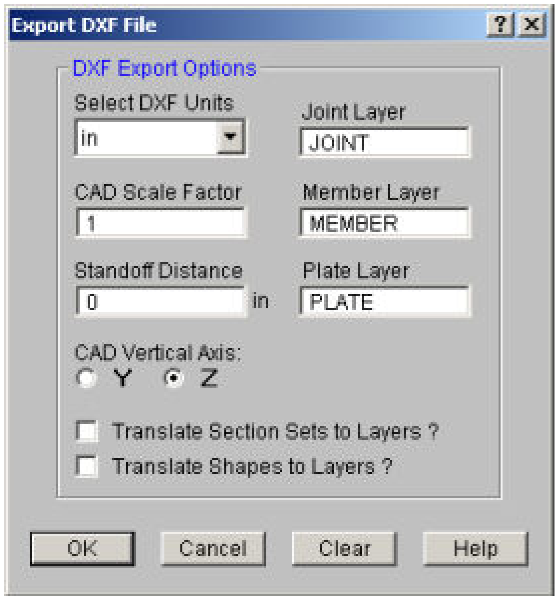
This is a Yes/No choice. If you choose "Yes", the program will take members and assign them to a shape based on their shape label. If there is not a database shape that corresponds to the DXF Layer Name, then these member will be assigned a general RE4x4 shape or a BAR2.

Note

- When assigning layer names in AutoCAD, remember to use an underscore character ("_") in place of a period (".") where a period would normally occur. For instance a C10X15.3 should be entered as C10X15_3. RISA-3D will automatically convert the "_" to a "." when the DXF file is read in.

Exporting DXF Files

Only the joint, member, wall and element geometry will be translated and used to create an ASCII DXF file. Any other information such as the boundary conditions, loads, member end releases, etc. will not be translated at this time. You have several options available for controlling how DXF files are exported as follows:



Joint Layer

Type the name of the layer for the joint point entities. If you don't enter anything, the default layer name will be "MODEL".

Member Layer

Type the name of the layer for the line entities. If you don't enter anything, the default layer name will be "MODEL". Note that this entry will be ignored if you select the option below to translate section set database shape names into layer names.

Plate Layer

Type the name of the layer for the plate elements, which will be represented as 3DFACE entities. If you don't enter anything, the default layer name will be "MODEL".

DXF Units

Select the units you desire the CAD model to be created in. The options for the DXF units are none, inches, feet, mm, cm, and meters.

CAD Scale

Enter the scale factor that will cause the full scale RISA model to be scaled up or down to the desired drawing scale. For example, if you created a full scale model that you wanted scaled down to 1/4"=12", the factor would be 0.020833, which is (.25/12).

Line End Standoff Distance

Enter the distance you wish to have the line entities "stand off" from the joints to which they are attached. The standoff distance is measured along the axis of the line. The distance will be in the DXF units, which is defined below. The distance will be used as entered and will not be scaled by the CAD Scale factor.

Note that if you create a DXF file with a non-zero standoff distance, it will be difficult to use the file for model geometry if you read the file back into RISA. (If you read such a file back in, you will end up with multiple joints at each member endpoint which will separated by the standoff distance)

DXF File Vertical Axis

Although it is not specifically noted in the AutoCAD documentation, the implied default vertical axis is the positive Z-axis of the current User Coordinate System.

The default vertical axis in RISA is usually the positive Y-axis and may be specified on the **Global Parameters**. When you export your model to a DXF file, you can have the program automatically rotate your geometry to match the vertical axis of your CAD program.

Translate Section Sets to Layer Names

This is a Yes/No check box. If you check the box “Yes”, the program will translate Section Set Labels to layer names. Layers will be created in the DXF file corresponding to the section set labels in the RISA model. A “Yes” choice here overrides any layer name entered for the member layer.

The program will add a prefix to each section set layer to designate what type of material that section is. The prefixes are as follows:

Material Type	**Layer Prefix**
Cold Formed Steel	CF_
Concrete	CN_
Hot Rolled Steel	HR_
General Materials	GN_
Wood	WD_

For example, let’s say you have designed a structure with Hot Rolled steel section sets called "Column"and " Girder", as well as a Wood section set called "Joist". If you type in a member layer name such as "STEEL" then all members, regardless of size, will appear on a layer named "STEEL". However, if you choose “Yes” for the *translate section sets to layers* option, then all the member that are assigned to the Column section set will appear on a layer named "HR_Column", the girders on a layer named "HR_Girder", and the joists on a layer called "WD_Joist".

Translate Shapes to Layer Names

This is a Yes/No choice. If you choose “Yes”, the program will take members and assign them to a layer which uses their shape label as the layer name. Layers will be created in the DXF file corresponding to the shape labels in the RISA model. A “Yes” choice here overrides any layer name entered for the member layer.

Note

- If you check BOTH the *Translate Section Sets* and *Translate Shapes* boxes, then only the explicitly defined shapes will be put placed on a layers according to their shape labels. All members defined with section sets will still be placed on layers according to their section set label.
- Please note that if the section set database shape designation includes one or more decimal point (".") characters, the export will translate each occurrence of a decimal point character into an underscore (“_”) character. For instance, a section set or shape label such as a C10X15.3 will translate into a layer name of "C10X15_3" or "HR_C10X15_3".
- The DXF format does not properly recognize certain ASCII text characters for layer names (< > / \ “ : ; ? * | = ‘). Therefore, these characters should be avoided for shape or section sets when using the "translate to layer names" options.

Merge After Importing a DXF File

It's always a good idea to do a Model Merge on any model created from a DXF file! In the process of creating a wire frame model in your CAD software, certain events may take place that cause end-points of LINE elements that were once matched to become mismatched by very small amounts. This most often happens as a result the following:

- Use of mirroring or rotating operations.
- Improper use or lack of use of point snaps.
- Trimming or breaking operations.
- Inconsistent precision when inputting point coordinates from the keyboard.

Model Merge combines joints that are within the “merge tolerance” distance of one another. The default distance for the merge tolerance is 0.01 ft. for all unit types.

You can also deal with several other possible problems by performing a **Model Merge**. This feature will also deal with intermediate joints along member spans, a common problem in models created from DXF drawings and members that cross, but do not have joints at their intersection point. See Model Merge for more information.

DXF Element Numbers

Different CAD packages handle ordering of geometric data in their DXF files in two basic ways.

Method 1:

Entities are written out into the DXF file based on the order in which they were created within the CAD program itself regardless of the order in which they were selected at the time the DXF file was made. Different operations such as copying, mirroring, arraying, etc. can produce unexpected results and it therefore becomes necessary to consult your CAD program documentation to understand how it stores and orders the geometry that you create via these various operations.

Method 2:

Entities are written out into the DXF file based on the order in which they were selected at the time the DXF file was made. AutoCAD is such a program. In order to control the ordering of the LINE entities, you must select the "Entities" option under the DXFOUT command and then select the lines in the order that you want them to appear in the RISA model.

Note

- Another option to help improve the ordering of the joints, members and elements in a model obtained from reading in a DXF file is to sort and relabel them once in RISA.

DXF File Format

The specific DXF file that you may read and write is the ASCII Drawing eXchange Files (DXF) file. Please note that AutoCAD has several different forms of DXF files available. ASCII is the default form and is the only form currently supported. The DXF read/write feature was written based on the DXF documentation for AutoCAD release 14. The feature has been tested with AutoCAD Versions 13 and 14.

The following is a short excerpt of the AutoCAD ASCII DXF format. This information is provided to help you debug any problems you may be having with DXF files that you are trying to read. For more complete information, consult your CAD documentation.

General

A DXF file is composed of sections of data. Each section of data is composed of records. Each record is stored on it’s own line. Each particular item is stored as two records, the first record is a group code and the second record is the data or a keyword. RISA only reads the ENTITIES section.

Group Codes

Each 2 record item start with an integer group code. RISA recognizes the following group codes:

Group Code	Description

0	Identifies the following overall keywords: SECTION, ENDSEC, and EOF. Within the ENTITIES section it also identifies POINT, LINE, and 3DFACE.
2	Identifies a section name (I.e., ENTITIES)
8	Identifies a layer name.
10, 11, 12, 13	Identifies the X coordinate of the 1st, 2nd, 3rd and 4th points of an item.
20, 21, 22, 23	Identifies the Y coordinate of the 1st, 2nd, 3rd and 4th points of an item.
30, 31, 32, 33	Identifies the Z coordinate of the 1st, 2nd, 3rd and 4th points of an item.

First and Last Records for a DXF file

Each DXF file must start with the first record as the group code “0”. The 2nd record must be the keyword “SECTION”. Each DXF file must have the 2nd to last record as the group code “0". The last record must be the keyword “EOF”.

Entities Section

The ENTITIES section will be identified by a group code of “0”, followed in the next record by the keyword “SECTION”. The next record will be the group code 2, followed in the next record by the keyword “ENTITIES”.

Item Formats within the ENTITIES Section

The POINT format is started by a group code of “0” followed by the keyword “POINT”. The layer name for the POINT will start with a group code record of 8, followed by a record with the actual layer name.

The coordinates for the point will be started by the 10, 20, and 30 group codes respectively for the X, Y, and Z coordinates. Other group codes and data may be present within the POINT data but these will be ignored.

The LINE format is started by a group code of “0” followed by the keyword “LINE”. The layer name for the LINE will start with a group code record of 8, followed by a record with the actual layer name.

The coordinates for the first point will be started by the 10, 20, and 30 group codes respectively for the X, Y, and Z coordinates. The coordinates for the second point will be started by the 11, 21, and 31 group codes respectively for the X, Y, and Z coordinates. Other group codes and data may be present within the LINE data but these will be ignored by RISA-3D.

The 3DFACE format is started by a group code of “0” followed by the keyword “3DFACE”. The layer name for the 3DFACE will start with a group code record of 8, followed by a record with the actual layer name.

The X, Y, and Z coordinates for the 1st through 4th points will be started by the 10, 20, and 30 through 14, 24, and 34 group codes respectively. Other group codes and data may be present within the 3DFACE data but these will be ignored.

AutoCAD Layer Names

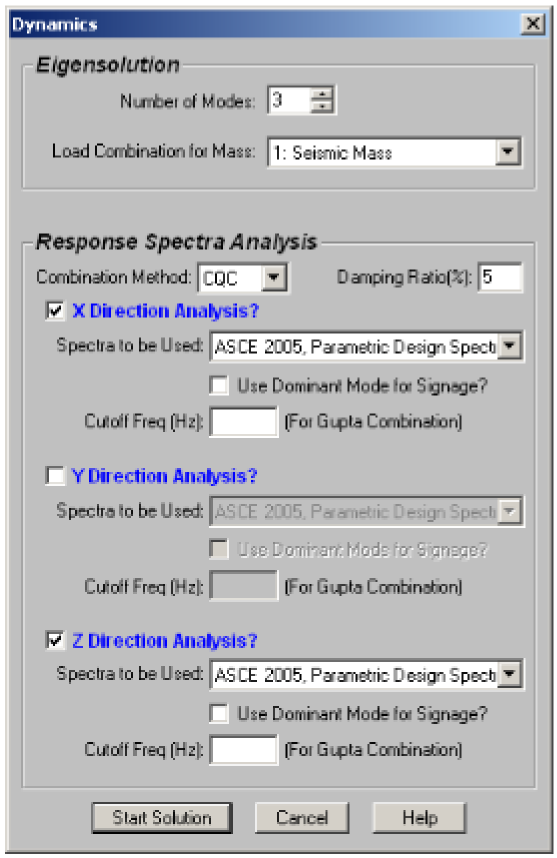
The only valid characters in an AutoCAD layer name are the letters A to Z, the numbers 0 to 9, and the three following characters: the dollar sign “$”, the underscore “_”, and the dash “-”.

Dynamic Analysis - Eigensolution

The dynamic analysis calculates the modes and frequencies of vibration for the model. This is a prerequisite to the response spectra analysis, which uses these frequencies to calculate forces, stresses and deflections in the model. For more information, see "Dynamic Analysis - Response Spectra".

You may calculate up to 500 modes for a model. The process used to calculate the modes is called an eigensolution. The frequencies and mode shapes are referred to as eigenvalues and eigenvectors.

The dynamic analysis uses a lumped mass matrix with inertial terms. Any vertical loads that exist in the **Load Combination for Mass** will be automatically converted to masses based on the acceleration of gravity entry on the Solution tab of the Global Parameters. However, you must always enter the *inertial* terms as Joint Masses.



To Perform a Dynamic Analysis / Eigensolution

1. You may wish to solve a static analysis first to verify that there are no instabilities.
2. Click **Solve** on the main menu and select **Dynamics** from the solution options.
3. Specify the load combination to use as the mass and the number of modes to solve.

Note

- You may view the mode shapes graphically by choosing this option in the Plot Options.
- Refer to "Dynamic Analysis - Response Spectra" for more information on that type of dynamic analysis.
- The Dynamic Solver option has been moved to the Global Parameters - Solution tab Advanced options.

Required Number of Modes

You may specify how many of the model's modes (and frequencies) are to be calculated. The typical requirement is that when you perform the response spectra analysis (RSA), at least 90% of the model's mass must participate in the solution. Mass participation is discussed in the Response Spectra Analysis section.

The catch is you first have to do a dynamic analysis in order to know how much mass is participating so this becomes a trial and error process. First pick an arbitrary number of modes (5 to 10 is usually a good starting point) and solve the RSA. If you have less than 90% mass, you'll need to increase the number of modes and try again. Keep in mind that the more modes you request, the longer the dynamic solution will take.

Note

- If you are obtaining many modes with little or no mass they are probably local modes. Rather than asking for even more modes and increasing the solution time see "Dynamics Troubleshooting – Local Modes" to learn how to treat the unwanted modes.

Dynamic Mass

The eigensolution is based on the stiffness characteristics of your model and also on the mass distribution in your model. There must be mass assigned to be able to perform the dynamic analysis. Mass may be calculated automatically from your loads or defined directly.

In order to calculate the amount and location of the mass contained in your model, RISA takes the vertical loads contained in the load combination you specify for mass and converts them using the acceleration of gravity defined in the Global Parameters. The masses are lumped at the joints and applied in all three global directions (X, Y and Z translation).

You may also specify mass directly. This option allows you to restrict the mass to a direction. You can also apply a mass moment of inertia to account for the rotational inertia effects for distributed masses. See "Loads - Joint Load / Displacement" to learn more about this.

Note

- Only the VERTICAL loads (including vertical components of inclined loads) contained in the load combination are converted to mass! Remember, you can designate which of the three global axes is to be considered the vertical axis via the Global Parameters.
- The self-weight of the model is NOT automatically included in the mass calculation. If you wish to have self-weight included, you must have it defined as part of the load combination.
- You may want to move the mass to account for accidental torsion, which is discussed in the next section.

Floor Diaphragm Mass

If your model was generated from within RISAFloor then the program will automatically assign dynamic mass and mass moments of inertia (MMOI) to each of the RISAFloor diaphragms. This mass and MMOI is based on the self weight and loading information defined in RISAFloor.

Modeling Accidental Torsion

Most design codes require an assumed accidental torsion that is in addition to the natural torsion created by the location of mass with respect to rigidity. While RISA-3D calculates the natural torsion you may want to model additional *accidental* torsion. This can be done easily by taking advantage of the rigid diaphragm feature.

Note:

- Review the specific requirements of the building code to confirm this. But, most codes will allow you to neglect accidental torsion for dynamic analysis and response spectra.

If you have modeled the dynamic mass at the center of mass only, then you may simply move the joints that specify the center of mass. For example, if the required accidental eccentricity is 5% of the building dimension then move the joints that distance, perpendicular to the applied spectra. You may then run the dynamic/rsa solution and combine the results with a static solution to check your members and plates for adequate capacity. You will have to do this by running solutions for all four directions to capture the controlling effects on all frames. You will not be able to envelope your results since you are changing the dynamic results each time you move the mass, so you will probably want to check all your load conditions one additional time after all your member sizes work to make sure that any force redistribution in your frames hasn't caused other members to fail.

Note that when you lump all your floor mass to the center of mass, you should also enter a Mass Moment of Inertia for your diaphragm as well by applying it as a Joint Mass to the center of mass joint. The rotational inertial effects of the diaphragm mass will contribute to your torsion shears and should not be ignored in most cases. See "Loads - Joint Load / Displacement" to learn more about this.

If you have not modeled the mass at discrete points that can be easily moved then you will have to apply the accidental torsion as a static load that will be part of a static analysis solution that includes the response spectra or equivalent lateral force procedure results. The magnitude of the torque will be the product of the story force and the accidental offset distance.

The accidental offset distance is usually a percentage of the building dimension perpendicular to the assumed earthquake direction. The story force is the story mass times the acceleration at that story level. If you are using an equivalent lateral static force procedure, you will have already calculated your story forces. If you are performing a response spectrum analysis, you can get the story forces exactly as the difference between the sum of the shears below and above the floor. Alternately, you could more simply use the full weight of the floor as the story force and then apply the scaling factor for your normalized spectra to this value as well. This simplified method may be unconservative if your floor accelerations have large amplifications as compared to your base acceleration due to the dynamic characteristics of your building.

The torsion can be applied as a point torque that you can apply to a joint on the diaphragm. The torque can also be applied as a force couple, with the magnitude of the forces determined by the distance between them to make the needed torque value. Often it is convenient to apply the forces for the couple at the ends of the building. One advantage of applying the accidental torsion as a static force is that you can set up all your required load combinations and let RISA-3D envelope them for you in one solution run.

Eigensolution Convergence

The eigensolution procedure for dynamic analysis is iterative, i.e. a guess is made at the answer and then improved upon until the guess from one iteration closely matches the guess from the previous iteration. The tolerance value is specified in the "Global Parameters" and indicates how close a guess needs to be to consider the solution to be converged. The default value of .001 means the frequencies from the previous cycle have to be within .001 Hz of the next guess frequencies for the solution to be converged. You should not have to change this value unless you require a more accurate solution (more accurate than .001?). Also, if you're doing a preliminary analysis, you may wish to relax this tolerance to speed up the eigensolution. If you get warning 2019 (missed frequencies) try using a more stringent convergence tolerance (increase the exponent value for the tolerance).

Saving Dynamic Solutions

After you’ve done the dynamic solution, you can save that solution to file to be recalled and used later.

Note

- This solution is saved in a .__R file and will be deleted when the Save or Save As options are used to overwrite the file. You may also delete this file yourself.

Work Vectors

When you request a certain number of modes for dynamic analysis (let's call that number N), RISA tries to solve for just a few extra modes. Once the solution is complete, RISA goes back to check that the modes it solved for are indeed the N lowest modes. If they aren't, one or more modes were missed and an error is reported.

Dynamics Modeling Tips

Dynamics modeling can be quite a bit different than static modeling. A static analysis will almost always give you some sort of solution, whereas you are not guaranteed that a dynamics analysis will converge to a solution. This is due in part to the iterative nature of the dynamics solution method, as well as the fact that dynamics solutions are far less forgiving of modeling sloppiness than are static solutions. In particular, the way you model your loads for a static analysis can be very different than the way you model your mass for a dynamic analysis.

The term "dynamics solution" is used to mean the solution of the free vibration problem of a structure, where we hope to obtain frequencies and mode shapes as the results.

In general, the trick to a "good" dynamics solution is to model the structure stiffness and mass with "enough" accuracy to get good overall results, but not to include so much detail that it take hours of computer run time and pages of extra output to get those results. Frame problems are simpler to model than those that include plate elements. "Building type" problems, where the mass is considered lumped at the stories are much easier to successfully model than say a cylindrical water tank with distributed mass. It is often helpful to define a load combination just for your dynamic mass case, separate from your "Dead Load" static case (You can call it "Seismic Mass"). Your seismic mass load combination will often be modeled very differently from your "Dead Load" static case.

If you apply your dynamic mass with distributed loads or surface loads on members/plates that are adjacent to supports, remember that the some of the load will go directly into the support and be lost to the dynamic solution. The mass that can actually vibrate freely is your "active mass", as opposed to your "static mass" which includes the mass lost into the supports. If you are having trouble getting 90% mass participation, you should roughly calculate the amount of mass that is being lost into your supports. You may need to reapply some of your mass as joint loads to your free joints. Or you may want to add more free joints to your model, by splitting up your plates or beams.

Modes for discretized mass models with very few degrees of freedom may not be found by the solver, even if you know you are asking for fewer modes than actually exist. In this case it may be helpful to include the self weight of the model with a very small factor (i.e. 0.001) to help the solver identify the modes.

Distributed mass models with plate elements, like water tanks, often require special consideration. You will want to use a fine enough mesh of finite elements to get good stiffness results. Often though, the mesh required to obtain an accurate stiffness will be too dense to simply model the mass with self-weight or surface loads. You will want to calculate the water weight and tank self-weight and apply it in a more discrete pattern than you would get using surface loads or self-weight. This method of using fewer joints to model the mass than to model the stiffness is often referred to as "discretizing" the mass. You want to lump the mass at fewer points to help the solution converge faster, however you have to be careful to still capture the essence of the dynamic behavior of the structure.

Whenever you perform a dynamic analysis of a shear wall structure, and the walls are connected to a floor, you must be careful to use a fine mesh of finite elements for each wall. Each wall should be at least 4 elements high between floors. This will give you at least 3 free joints between them.

When you perform a dynamic analysis of beam structures, such that you are trying to capture the flexural vibrations, (i.e., the beams are vibrating vertically or in the transverse direction), you must make sure that you have at least 3 free joints along the member between the points of support. If you use a distributed load as the mass, you must remember that some of the load will automatically go into the supports and be "lost" to the dynamic solution. In general, you will get the best results by applying your mass as joint loads to the free joints.

If you are trying to model dynamic effects on a 2D frame, you will want to make sure that you restrain the out-of-plane degrees of freedom. See "Boundary Condition at ALL Joints" to learn how to do this.

Modal Frequency Results

Access the **Modal Frequency** spreadsheet by selecting it from the **Results** Menu.

Frequencies and Participation

	Mode	Frequency (Hz)	Period (Sec)	SX Par...	SY Par...	SZ Par...
1	1	2.68	.373			
2	2	2.703	.37			
3	3	4.006	.25			
4	4	6.507	.154			
5	5	7.574	.132			
6	6	9.642	.104			
7	Totals:					

These are the calculated model frequencies and periods. The period is simply the reciprocal of the frequency. These values will be used along with the mode shapes when a response spectra analysis is performed. The first frequency is sometimes referred to as the model's natural or fundamental frequency. These frequency values, as well as the mode shapes, will be saved and remain valid unless you change the model data, at which time they will be cleared and you need to re-solve the dynamics to get them back.

Also listed on this spreadsheet are the participation factors for each mode for each global direction, along with the total participation. If no participation factors are listed, the response spectra analysis (RSA) has not been performed for that direction. If the RSA has been done but a particular mode has no participation factor listed, that mode shape is not participating in that direction. This usually is because the mode shape represents movement in a direction orthogonal to the direction of application of the spectra. See "Dynamic Analysis - Response Spectra" for more information.

Mode Shape Results

Access the **Mode Shape** spreadsheet by selecting it from the **Results** Menu.

Mode Shape 1 Period .373

	Joint	X	Y	Z	X Rotat...	Y Rotat...	Z Rotat...
1	N1	0	0	0	0	0	0
2	N2	0	0	0	0	0	0
3	N3	0	0	0	0	0	0
4	N4	0	0	0	0	0	0
5	N5	0	0	0	0	0	0
6	N6	0	0	0	0	0	0
7	N7	0	0	0	0	0	0
8	N8	0	0	0	0	0	0
9	N9	0	0	0	0	0	0
10	N10	1.611e-4	-9.843e-3	-1.474e-4	0	1.552e-5	-4.035e-3
11	N11	1.611e-4	9.843e-3	1.474e-4	0	1.552e-5	-4.035e-3
12	N12	1.611e-4	9.843e-3	-1.474e-4	0	-1.552e-5	-4.035e-3
13	N13	1.611e-4	-9.843e-3	1.474e-4	0	-1.552e-5	-4.035e-3
14	N14	.948	-4.407e-2	4.716e-4	2.417e-5	1.154e-4	-1.302e-3
15	N15	.948	-4.351e-2	6.673e-5	-3.741e-4	0	1.066e-3
16	N16	.948	0	0	0	8.481e-5	1.320e-5
17	N17	.948	4.352e-2	-6.675e-5	3.741e-4	0	1.066e-3
18	N18	.948	4.407e-2	-4.716e-4	-2.417e-5	1.154e-4	-1.302e-3
19	N19	.948	-4.491e-2	2.244e-4	-1.243e-5	-3.134e-5	3.816e-4
20	N20	.948	-3.461e-2	6.927e-5	1.558e-4	0	4.770e-5
21	N21	.948	0	0	0	-2.177e-5	1.122e-3
22	N22	.948	3.461e-2	-6.929e-5	-1.558e-4	0	4.770e-5

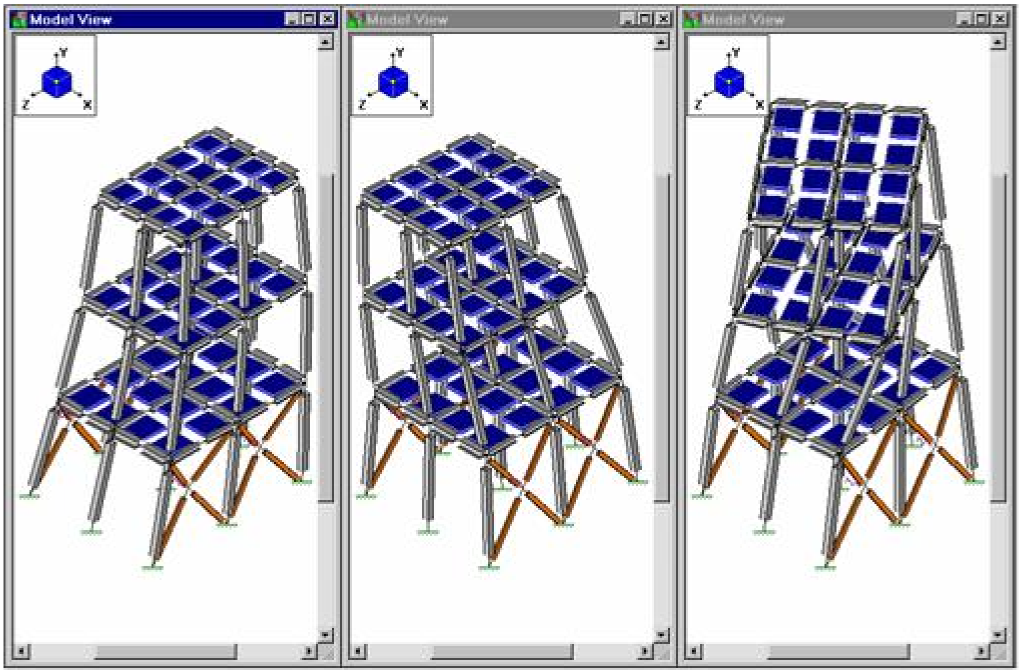
These are the model's mode shapes. Mode shapes have no units and represent only the movement of the joints relative to each other. The mode shape values can be multiplied or divided by any value and still be valid, so long as they retain their value relative to each other. To view higher or lower modes you may select them from the drop-down list of modes on the Window Toolbar.

Note

- Keep in mind that these mode shapes do not, in and of themselves, represent model deflections. They only represent how the joints move relative to each other. You could multiply all the values in any mode shape by any constant value and that mode shape would still be valid. Thus, no units are listed for these mode shape values.

These mode shapes are used with the frequencies to perform a Response Spectra Analysis. The first mode is sometimes referred to as the natural or fundamental mode of the model. The frequency and mode shape values will be saved until you

change your model data. When the model is modified, these results are cleared and you will need to re-solve the model to get them back.



You can plot and animate the mode shape of the model by using the Plot Options. This allows you to verify the mode shapes that were obtained and highlights local modes making them easy to troubleshoot. See "Graphic Display" for more information.

Dynamics Troubleshooting – Local Modes

A common problem you may encounter are “localized modes”. These are modes where only a small part of the model is vibrating and the rest of the model is not. A good example of this is an X brace that is vibrating out of plane. Localized modes are not immediately obvious from looking at the frequency or numeric mode shape results, but they can be spotted pretty easily using the mode shape animation feature. Just plot the mode shape and animate it. If only a small part of the model is moving, this is probably a localized mode.

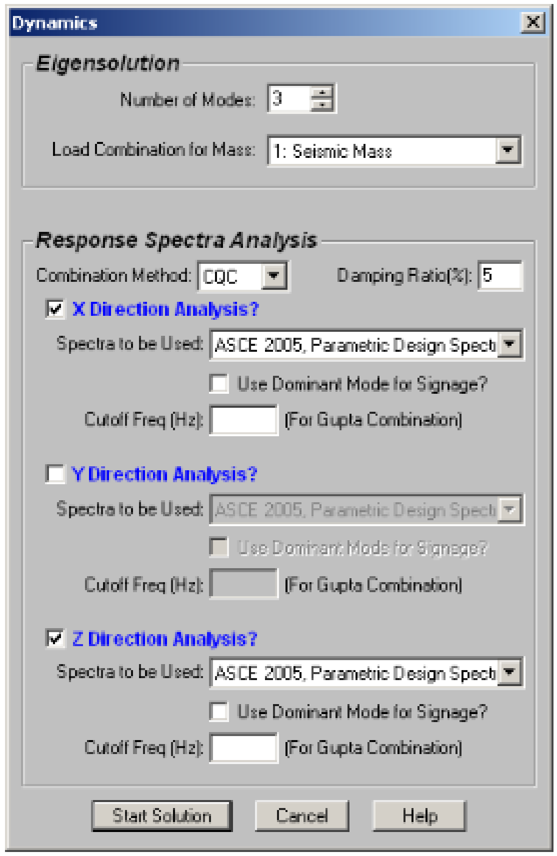
The problem with localized modes is that they can make it difficult to get enough mass participation in the response spectra analysis (RSA), since these local modes don’t usually have much mass associated with them. This will show up if you do an RSA with a substantial number of modes but get very little or no mass participation. This would indicate that the modes being used in the RSA are localized modes.

Quite often, localized modes are due to modeling errors (erroneous boundary conditions, members not attached to plates correctly, etc.). If you have localized modes in your model, always try a Model Merge before you do anything else. See "Model Merge" for more information.

To get rid of localized modes that are not the result of modeling errors, you can sometimes use boundary conditions to restrain the mode shape. For example, if your localized mode is at a weak X brace (as mentioned before), you could attach a spring to the center of the X brace to restrain the mode shape. Another cause of local modes is including the self-weight in models with walls or horizontal diaphragms modeled with plate/shell elements. These walls and floors can have many modes that will tend to vibrate out-of-plane like drums, but will have very little effect on the overall structure response. One way to reduce these out-of-plane modes is to make the plate material weightless by setting the material density to zero. You would then need to “lump” the plate weight at just a few joints on the wall or floor using joint loads. If you have many sets of X-braces that are vibrating out of plane, you can make all your braces weightless as well. This takes a little mass out of the model, but is much faster than trying to put in springs for all the out of plane directions.

Dynamic Analysis - Response Spectra

A response spectra analysis may be performed after the dynamic analysis to obtain forces, stresses and deflections. In general, the response spectra analysis procedure is based on the assumption that the dynamic response of a structural model can be approximated as a summation of the responses of the independent dynamic modes of the model.



To Perform a Response Spectra Analysis

1. Select **Dynamics (Eigensolution / Response Spectra)** from the **Solve** menu.
2. Set the Eigensolution parameters.
3. Select the desired **Combination Method**. Then use the checkboxes to indicate which directions you want to perform your response spectra analysis.
4. Select the spectra to be used for each direction. Then specify the other parameters. For help on an item, click  and then click the item.

Note

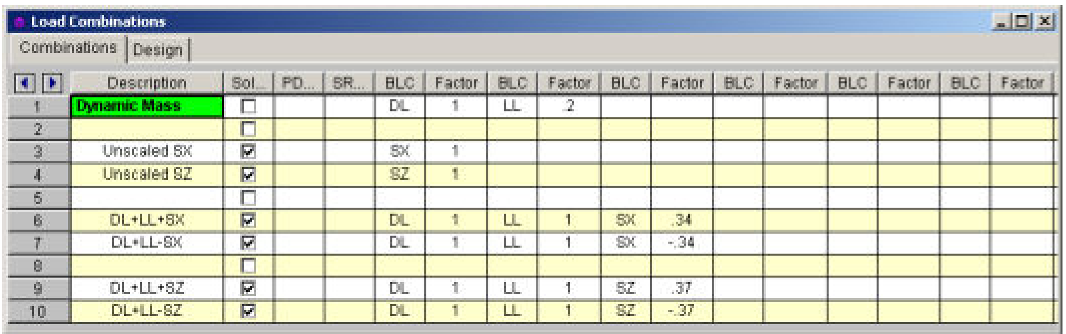
- For a more thorough explanation of the Eigensolution options refer to Dynamic Analysis - Eigensolution.
- Upon the completion of the solution you are returned to the **Frequencies** and **Participation** spreadsheet and the participation yielded by the RSA is listed. To view model results such as forces/deflections/reactions you will need to create a load combination on the **Load Combination** spreadsheet that includes the spectra results. See below.
- The Dynamic Solver option has been moved to the Global Parameters - Solution tab Advanced options.

To Include Response Spectra Analysis Results in a Load Combination

1. After running the response spectra analysis go to the desired combination on the **Load Combination** spreadsheet.
2. In the **BLC** column enter "**SX**", "**SY**", or "**SZ**" as the BLC entry (**SX** for the X direction RSA results, **SY** for the Y direction RSA results, etc.).
3. To scale the spectral results enter the spectra-scaling factor in the **Factor** column.

Note

- You can include more than one spectra solution in a single load combination. If you do this you can also have RISA-3D combine the multiple RSA results using an SRSS summation. To do this, set the "RSA SRSS" flag for the combination to "+" or “-”. Use “+” if you want the summed RSA results (which will be all positive) added to the other loads in the load combination. Use “-” if you want the summed results subtracted.



	Description	Sol...	PD...	SR...	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor
1	Dynamic Mass	☐			DL	1	LL	.2								
2		☐														
3	Unscaled SX	☑			SX	1										
4	Unscaled SZ	☑			SZ	1										
5		☐														
6	DL+LL+SX	☑			DL	1	LL	1	SX	.34						
7	DL+LL-SX	☑			DL	1	LL	1	SX	-.34						
8		☐														
9	DL+LL+SZ	☑			DL	1	LL	1	SZ	.37						
10	DL+LL-SZ	☑			DL	1	LL	1	SZ	-.37						

Response Spectra

The response spectra represent the maximum response of any single degree of freedom (SDOF) system to a dynamic base excitation. The usual application of this method is in seismic (earthquake) analysis. Earthquake time history data is converted into a "response spectrum". With this response spectrum, it is possible to predict the maximum response for any SDOF system. By "any SDOF system", it is meant a SDOF system with any natural frequency. "Maximum response" means the maximum deflections, and thus, the maximum stresses for the system.

Response Spectra Analysis Procedure

In the response spectra analysis procedure, each of the model's modes is considered to be an independent SDOF system. The maximum responses for each mode are calculated independently. These modal responses are then combined to obtain the model's overall response to the applied spectra.

The response spectra method enjoys wide acceptance as an accurate method for predicting the response of any structural model to any arbitrary base excitation, particularly earthquakes. Building codes require a dynamics based procedure for some structures. The response spectra method satisfies this dynamics requirement. The response spectra method is easier, faster and more accurate than the static procedure so there really isn't any reason to use the static procedure.

If you wish to learn more about this method, an excellent reference is *Structural Dynamics, Theory and Computation* by Dr. Mario Paz (1991, Van Nostrand Reinhold).

Frequencies Outside the Spectra

If a response spectra analysis is solved using modal frequency values that fall outside the range of the selected spectra, RISA will extrapolate to obtain spectral values for the out-of-bounds frequency. If the modal frequency is below the smallest defined spectral frequency, a spectral velocity will be used for the modal frequency that will result in a constant Spectra Displacement from the smallest defined spectral frequency value. A constant spectral displacement is used because modes in the “low” frequency range will tend to converge to the maximum ground displacement. If the modal frequency is above the largest defined spectral frequency, a spectral velocity will be used for the modal frequency that will result in a constant Spectra Acceleration from the largest defined spectral frequency value. A constant spectral acceleration is used because modes in the “high” frequency range tend to converge to the maximum ground acceleration (zero period acceleration).

Mass Participation

Frequencies and Participation

	Mode	Frequency (Hz)	Period (Sec)	SX Par...	SY Par...	SZ Par...
1	1	1.4	.714			90.353
2	2	1.815	.551	94.517		
3	3	1.999	.5			
4	Totals:			94.517		90.353

The mass participation factors reported on the **Frequencies Spreadsheet** reflect how much each mode participated in the Response Spectra Analysis solution. Remember that the RSA involves calculating separately the response for each mode to the applied base excitation represented by the spectra. Here is where you can tell which modes are important in which directions. Higher participation factors indicate more important modes. The participation factor itself is the percent of the model's total dynamic mass that is deflecting in the shape described by the particular mode. Thus, the sum of all the participation factors in a given direction can not exceed 100%.

The amount of participation for the mode may also reflect how much the mode moves in the direction of the spectra application. For example, if the 1st mode represents movement in the global Y direction it won't participate much, if at all, if the spectra is applied in the global X direction. You can isolate which modes are important in which directions by examining the mass participation.

Note

- Usually for the RSA to be considered valid, the sum of the modal participation factors must equal or exceed **90%**. If you do an RSA and the total participation is less than 90%, you need to return to the dynamic solution and redo the dynamic analysis with more modes. If you are getting a lot of modes with little or no participation see Dynamics Troubleshooting – Local Modes.
- You may also want to account for accidental torsion. See Modeling Accidental Torsion to learn how to do this.
- Models with a large amount of mass lost into boundary conditions may have difficulty achieving 90% mass participation. See Dynamics Modeling for more information.

Modal Combination Option

There are three choices for combining your modal results: CQC, SRSS, or Gupta. In general you will want to use either CQC or Gupta. For models where you don't expect much rigid response, you should use CQC. For models where the rigid response could be important, you should use Gupta. An example of one type of model where rigid response would be important is the analysis of shear wall structures. The SRSS method is offered in case you need to compare results with the results from some older program that does not offer CQC or Gupta.

CQC stands for "Complete Quadratic Combination". A complete discussion of this method will not be offered here, but if you are interested, a good reference on this method is *Recommended Lateral Force Requirements and Commentary, 1999*, published by SEAOC (Structural Engineers Assoc. of Calif.). In general, the CQC is a superior combination method because it accounts for modal coupling quite well.

The Gupta method is similar to the CQC method in that it also accounts for closely spaced modes. In addition, this method also accounts for modal response that has "rigid content". For structures with rigid elements, the modal responses can have both rigid and periodic content. The rigid content from all modes is summed algebraically and then combined via an SRSS combination with the periodic part which is combined with the CQC method. The Gupta method is fully documented in the reference, *Response Spectrum Method*, by Ajaya Kumar Gupta (Published by CRC Press, Inc., 1992).

The Gupta method defines lower (f 1) and upper (f 2) frequency bounds for modes containing both periodic and rigid content. Modes that are below the lower bound are assumed to be 100% periodic. Modes that are above the upper bound are assumed to be 100% rigid.

Unsigned (All Positive) Results

A response spectra analysis involves calculating forces and displacements for each mode individually and then combining these results. The problem is both combination methods offered (SRSS and CQC) use a summation of squares approach that

loses the sign. This means that all the results are positive, except reactions, which are all negative as a result of the positive displacements.

Because the RSA results are unsigned you cannot directly add the results to other static loads in you model. One way around this is to treat the RSA results as both positive and negative by manually providing the sign. Using two combinations for each RSA result, one with a positive factor and the other with a negative factor you can capture the maximum deflections, stresses and forces when combining with other loads. See Load Combinations with RSA Results for an example.

The mass participation may indicate that a model is dominated by a single mode in a direction. You may base the signs for the final combined RSA results on the signs for the RSA for this single dominant mode by checking the box that says "**Use Dominant Mode for Signage?**". When that option is selected then the Mode that that the highest mass participation in that direction will be considered to be the dominant mode.

Other Options

Cut Off Frequency

This is the frequency used by the Gupta method to calculate the upper bound for modes having periodic and rigid content. The "rigid frequency" is defined as "The minimum frequency at which the spectral acceleration becomes approximately equal to the zero period acceleration (ZPA), and remains equal to the ZPA". If nothing is entered in this field, the last (highest) frequency in the selected response spectra will be used.

Damping Ratio

The damping ratio entered here is used in conjunction with the CQC and Gupta combination methods. This single entry is used for all the modes included in the RSA, an accepted practice. A value of 5% is generally a good number to use. Typical damping values are:

2% to 5% for welded steel

3% to 5% for concrete

5% to 7% for bolted steel, wood

Localized Modes

A common problem you may encounter in dynamic analysis is localized modes. These are modes where only a small part of the model is vibrating and the rest of the model is not. Localized modes may not be obvious from looking at the numeric mode shape results, but they can usually be spotted by animating the mode shape. See Plot Options - Deflections to learn how to do this. If only a small part of the model is moving, this is probably a localized mode.

The problem with localized modes is that they can make it difficult to get enough mass participation in the response spectra analysis, since these local modes don't usually have much mass associated with them. Solving for a substantial number of modes but getting very little or no mass participation would indicate that the modes being found are localized modes.

If you have localized modes in your model, always try a Model Merge before you do anything else. See Model Merge for more information. To get rid of localized modes not omitted with a merge the options are to adjust either the mass or the stiffness of the model. For example, if your localized mode is an X-brace vibrating out of plane, you could attach a spring (adjusting the stiffness) to the center of the X brace and restrain the brace. Alternatively you could make the brace weightless and lump the mass at the end joints (adjusting the mass).

If you try the model merge and are still having trouble, see Dynamics Modeling for more help resolving local modes or getting 90% mass participation for your model.

RSA Scaling Factor (Manual Scaling)

The most difficult part of the entire RSA procedure is normally calculating the scaling factor to be used when including the RSA results in a load combination.

The ASCE 7 uses a particular "shape" for it's spectra (See Figure 11.4-1), but the parameters S_{DS} and S_{D1} make it specific to a particular site. However, the ASCE 7 imposes several requirements regarding the minimum design values. ASCE 7-10 Section 12.9.4.1 specifies a modification factor, $0.85*V/V_t$, that may be used to scale the response spectra results to something less than or equal the base shear calculated using the static procedure (ASCE 7 Sect. 12.8).

The static base shear (V) is calculated using the equations in ASCE 7-10 Sect. 12.8.1

Note that there are limiting values for the static base shear in ASCE 7-10 equations 12.8-3 through 12.8-6.

Therefore, in order to calculate the proper scaling factor, we need to know what the *unscaled* RSA base shear (this is called the Elastic Response Base Shear in the IBC) is, and we also need to calculate the value of "V" (static base shear). The calculation of V isn't particularly difficult because the two values that present the biggest problem in this calculation (T and W) are provided by RISA. To calculate the value of W, simply solve a load combination comprised of the model seismic dead weight. This almost certainly will be the same load combination you used in the **Dynamics** settings for the **Load Combination for Mass**. The vertical reaction total is your "W" value.

The T value is simply the period associated with the dominant mode for the direction of interest. For example, if you're calculating the scaling factor for a Z-direction spectra, determine which mode gives you the highest participation for the Z direction RSA. The period associated with that mode is your T value. Note that there are limiting values for T, see ASCE 7-10 section 12.8.2.

Calculating the unscaled RSA base shear also is very straightforward. Just solve a load combination comprised of only that RSA, with a factor of 1.

Example:

- Assuming we're looking at a Z direction RSA, enter "SZ" in the **BLC** field of the **Load Combinations** spreadsheet and for the **Factor** enter "1". Leave all the other BLC fields blank.
- Solve the load combination and look at the Z direction reaction total. This total value is the unscaled RSA base shear.
- To get the correct scaling factor, solve this equation:

Scale Factor = (V / Unscaled RSA base shear)

You would do this calculation to obtain the scaling factors for all the directions of interest (X, Y and/or Z). Unless the model is symmetric the fundamental period for each direction is probably different. Be sure to use the proper value for "T" for the direction being considered.

Note: The ASCE 7 has additional requirements for vertical seismic components. (See ASCE 7-10 section 12.4.2.2).

RSA Scaling Factor (Automatic Scaling)

The most difficult part of the entire RSA procedure is normally calculating the scaling factor to be used when including the RSA results in a load combination. The Scaling Factor tool can be used to automatically calculate the RSA scaling factor for each direction within your model.

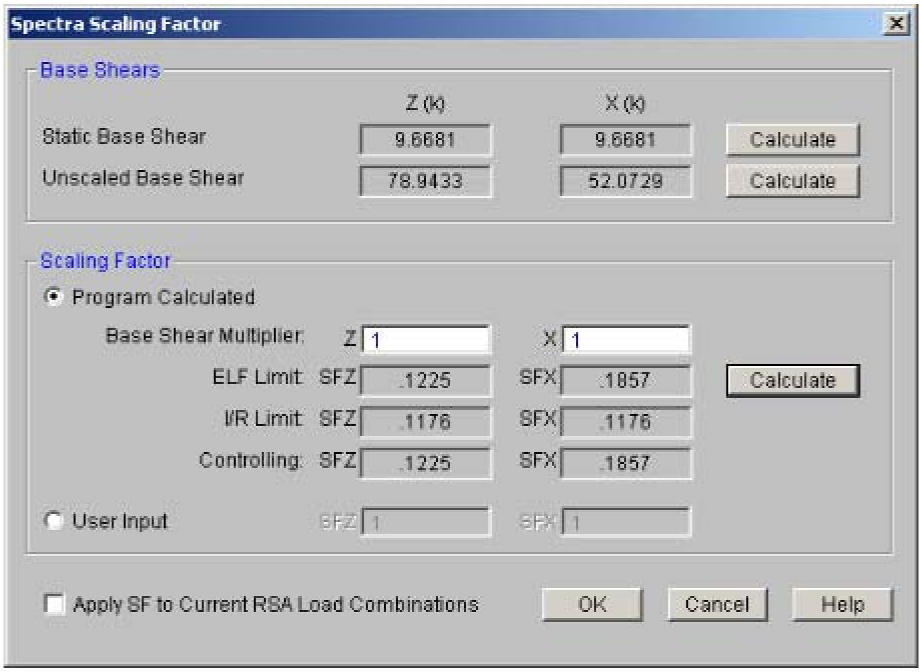
The RSA scaling factors automatically default to 1.0 and can only be changed from within the Scaling Factor dialog. The dialog may be opened by clicking the scale factor button, **SF**, on the Spreadsheet Toolbar. This button is only available when the Load Combinations Spreadsheet is the active window.

Note

- Currently these factors are only calculated when you open this dialog and click the **Calculate** button. Therefore, if you have previously calculated these and then made changes to your model, you will need to come back into this dialog and re-click the **Calculate** buttons to update the scaling factors.

Base Shear

The **Base Shears** section of the Spectra Scaling Factor dialog box reports to you the static base shear and the un-scaled RSA base shear for each direction. If these values have not been calculated yet, then you may press the **Calculate** button. This will launch the Seismic Load Generation window for the calculation of the **Static Base Shear**, or the Dynamic Solution dialog for the calculation of the **Unscaled Base Shear** for your Response Spectra results.



Scaling Factor

Program Calculated

This section compares the ELF (Equivalent Lateral Force) Limit, as reduced by the Base Shear Multiplier, and the I/R Limit. These values are explained in detail below.

- The **Base Shear Multiplier** is used in the calculation of the ELF Limit. This multiplier can be used to reduce the calculated (unscaled) base shear below the base shear value obtained by the Equivalent Lateral Force Method. For the 1997 UBC, this is covered in section 1631.5.4. For the 2000 IBC, this is covered in section 1618.7. For the 2010 NBC, it is discussed in clause 4.1.8.12 (8). In the ASCE 7-10, it is discussed in section 12.9.4.1.
- The **ELF Limit** is the limiting scaling factor calculated per the Equivalent Lateral Force procedure:

$$ELF\ Limit = \frac{(Base\ Shear\ Multiplier * Static\ Base\ Shear)}{Unscaled\ Base\ Shear}$$

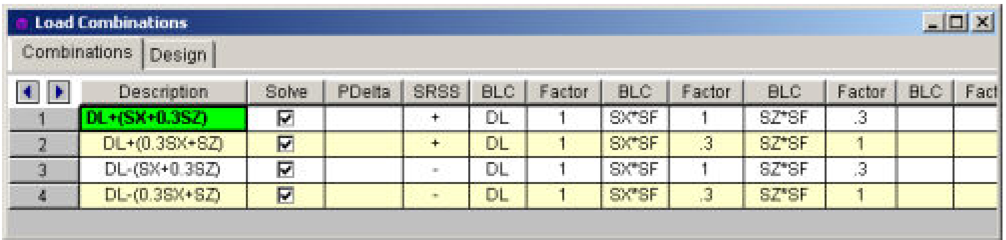
- The **I/R Limit** simply takes the elastic dynamic base shear and reduces it to an inelastic base shear. This value is calculated per the I (Importance Factor) and R (Response Modification Coefficient). You may edit these values in the Seismic tab of Global Parameters.
- The **Controlling Limit** is the governing (larger) value of the ELF limit and I/R limit.

User Input

You may manually override the program calculated factor by selecting the **User Input** option and entering in an alternate value.

Apply Checkbox

The **Apply SF to all RSA Load Combinations** checkbox will apply the SF scale factors to all load combinations that currently reference response spectra results. The final Load combinations will appear similar to those shown below:



	Description	Solve	PDelta	SRSS	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Fact
1	DL+(SX+0.3SZ)	☑		+	DL	1	SX*SF	1	SZ*SF	.3		
2	DL+(0.3SX+SZ)	☑		+	DL	1	SX*SF	.3	SZ*SF	1		
3	DL-(SX+0.3SZ)	☑		-	DL	1	SX*SF	1	SZ*SF	.3		
4	DL-(0.3SX+SZ)	☑		-	DL	1	SX*SF	.3	SZ*SF	1		

Automatic Response Spectra Generation

1997 UBC

You may have the 1997 UBC spectra generated automatically by selecting "UBC 97, Parametric Design Spectra" for your RSA. The Ca and Cv seismic coefficients are needed to calculate the values for the UBC ’97 spectra. See Figure 16-3 in the UBC for the equations used to build the spectra. See Tables 16-Q and 16-R to obtain the Ca and Cv values. The default values listed are for Seismic Zone 3, Soil Type “Se” (Soft Soil Profile). These values can be edited in the Seismic tab of Global Parameters.

2000 IBC

You may have the 2000 IBC spectra generated automatically by selecting "IBC 2000, Parametric Design Spectra" for your RSA. The S_{DS} and S_{D1} seismic coefficients are needed to calculate the values for the IBC 2000 spectra. See Figure 1615.1.4 in the IBC for the equations used to build the spectra. See section 1615.1.3 to obtain the S_{DS} and S_{D1} values.These values can be edited in the Seismic tab of Global Parameters.

2005 ASCE

You may have the 2005 ASCE spectra generated automatically by selecting "ASCE 2005, Parametric Design Spectra" for your RSA. The S_{DS}, S_{D1}, and TL seismic coefficients are needed to calculate the values for the ASCE 2005 spectra. See Figure 11.4-1 in ASCE-7 2005 for the equations used to build the spectra. See section 11.4.4 to obtain the S_{DS} and S_{D1} values and Figures 22-15 thru 22-20 for the TL value. These values can be edited in the Seismic tab of Global Parameters.

2010 ASCE

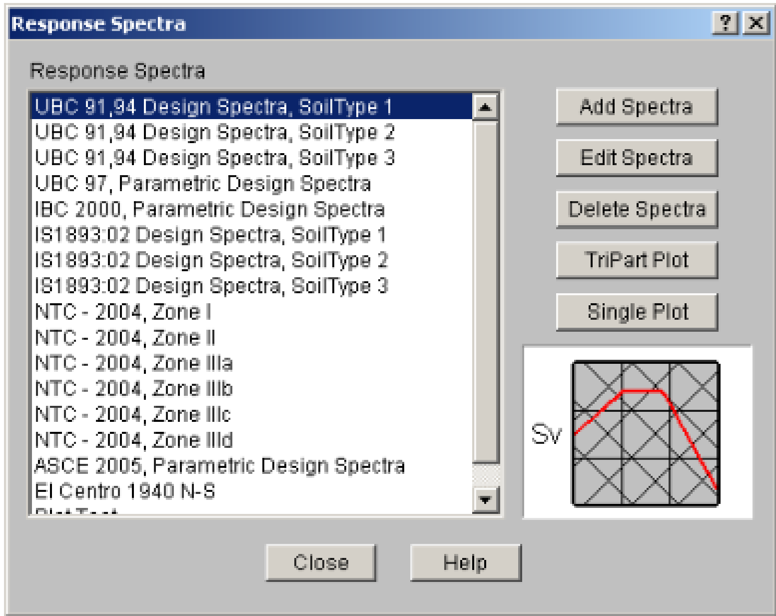
You may have the 2010 ASCE spectra generated automatically by selecting "ASCE 2010, Parametric Design Spectra" for your RSA. The S_{DS}, S_{D1}, and TL seismic coefficients are needed to calculate the values for the ASCE 2010 spectra. See Figure 11.4-1 in ASCE-7 2010 for the equations used to build the spectra. See section 11.4.4 to obtain the S_{DS} and S_{D1} values and Figures 22-12 thru 22-16 for the TL value. These values can be edited in the Seismic tab of Global Parameters.

2005 NBC

You may have the 2005 NBC spectra generated automatically by selecting "NBC 2005 Parametric Design Spectra" for your RSA. The Site Class and the $S_a$values are needed to calculate the values for the NBC 2005 spectra. Please see section 4.1.8.47 to obtain the S_a values and Table 4.1.8.4.A for the Site Class. Please see section 4.1.8.4.7 for the equations used to build the spectra.These values can be edited in the **Seismic** tab of Global Parameters.

Adding and Editing Spectra

You may add your own spectra to the database and edit and delete them once they are created. You can add/edit spectra data pairs in any configuration by choosing between Frequency or Period and between the three spectral values. You may also choose to convert the configuration during editing. At least two data points must be defined. Log interpolation is used to calculate spectra values that fall between entered points. Make sure that all of the modal frequencies in your model are included within your spectra.



To Add or Edit a Spectra

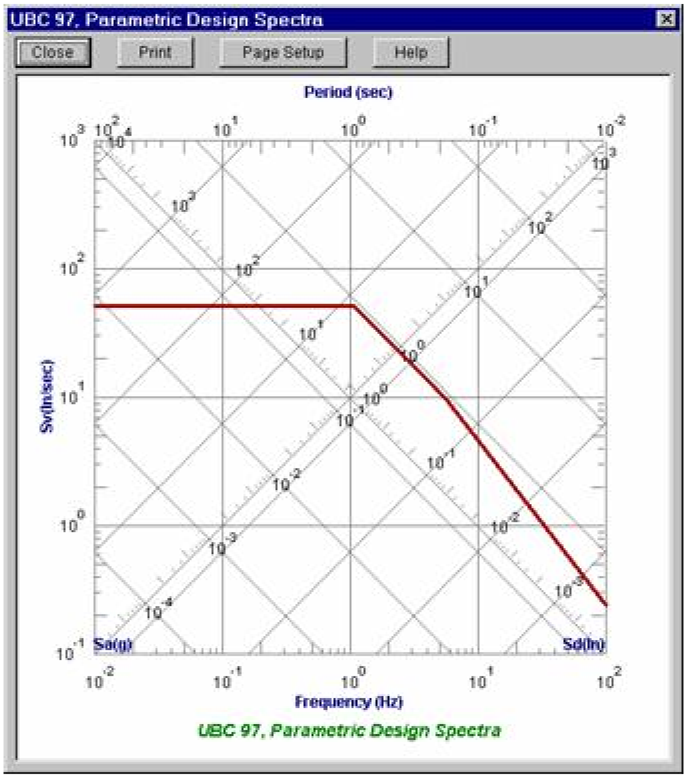
1. On the **Modify** menu choose Response Spectra Library.
2. Select **Add** or **Edit**.
3. Select the format to be used and specify the parameters. For help on an item, click ? and then click the item.

Note

- Zero values are not allowed in the data.
- The spectra data is not currently stored with the RISA model file. Instead it is stored in the RSPECT32.FIL database file located in the directory set using Tools - Preference - File Locations. If a file with a custom spectra needs to be transferred to another computer, then this file must also be transferred to the new computer.

Tripartite Response Spectra Plot

This plot is a convenient logarithmic representation of all the values of interest in the response spectra definition.



These values are as follows:

Frequency (f)

Period (T)

Pseudo Velocity (Sv)

Pseudo Acceleration (Sa)

Pseudo Displacement (Sd)

The relationships between these values (for the undamped case) is as follows:

$T = 1. / f$

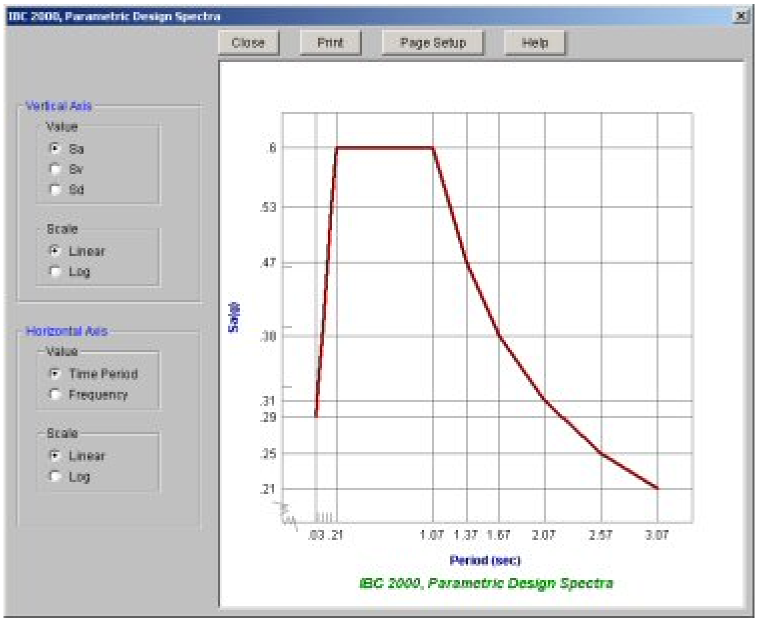
$Sv = Sd * 2\pi f = Sa / 2\pi f$

For the tripartite plot, the frequency values are plotted along the bottom with the reciprocal period values displayed along the top. The ordinate axis plots the Sv values (labeled on the left side) and the diagonal axes plot the Sa (lower left to upper right) and Sd (upper left to lower right) values.

The spectra data itself is represented with the thick red line. Therefore, to determine the Sv, Sa or Sd value for a particular frequency or period, locate the desired period or frequency value along the abscissa axis and locate the corresponding point on the spectra line. Use this point to read off the Sv, Sa and Sd values from their respective axes. Remember, all the axes are logarithmic!

Single Spectra Plot

This plot is a Response spectra representation that is given in the building codes. The vertical axis can plot the spectra using Pseudo-Acceleration, Pseudo-Velocity, or Pseudo Displacement on a vertical or logarithmic scale. The horizontal axis will plot the Period or Frequency using a Linear or Logarithmic scale.



These values are as follows:

Frequency (f)

Period (T)

Pseudo Velocity (Sv)

Pseudo Acceleration (Sa)

Pseudo Displacement (Sd)

The relationships between these values (for the undamped case) is as follows:

$T = 1. / f$

$Sv = Sd * 2\pi f = Sa / 2\pi f$

The spectra data itself is represented with the thick red line. Therefore, to determine the Sv, Sa or Sd value for a particular frequency or period, locate the desired period or frequency value along the horizontal axis and locate the corresponding point on the spectra line. Use this point to read off the Sv, Sa and Sd values from their vertical axis.

File Operations

You may create, save, and recall files from the **File Menu** or from the toolbar buttons just beneath the menu on the **RISA Toolbar**. Importing and exporting other file types may also be accomplished from the **File Menu**.

Starting Off

When you open the program or start a new model you are presented with the following options: Draw Members, Draw Plates, Generate a Model, and Go to the File Open Dialog

The first group of options is to help you to create a new model by drawing members or plates or by generating with a template. The second group of options helps you open existing files by showing you the most recent files or letting you find the model you want in the Open File Dialog.

Creating a New Model

- On the RISA toolbar click the New Model button. You will be prompted to 'Save' if you have made changes to the current model that have not yet been saved.

Opening an Existing Model

1. On the RISA toolbar click the **Open File** button.
2. In the **Look In** field click the drive, folder, or location that contains the file.
3. In the folder list, double-click the folders until you open the folder that contains the file you want.
4. Double-click the file you want to open.

Tip

- To open a model you have used recently, click the **File Menu** and select from the file names at the bottom of the menu.

Saving a New Untitled Model

1. On the RISA toolbar click the **Save Model** button.
2. To save the model in a different folder, click a different drive in the **Save in** field, or double-click a different folder in the folder list. To save the model in a new folder click the **Create New Folder** button.
3. In the **File name** field type a name for the document. You can use long, descriptive filenames if you want.
4. Click **Save**.

Saving an Existing Model

1. On the RISA toolbar click the **Save Model** button.

Appending Models

You may append other models into the current model by choosing **Append** from the **File** menu. This lets you have a library of common model parts that you can use later or to divide projects into smaller parts and later combine them. This feature does not allow you to combine two unopened models. You must first open one of the models, and then append the second model into the first model. If you want to have an unmodified copy of the first model, you will need to save the new combined model under a new filename.

You can only append files from version 6.0.x files. If you've got an older file that you want to append, you will first need to open that file by itself and save it.

Your appended model will be placed in the current model using the original coordinate data that was in the appended model. The appended model is NOT moved to the origin of the current model. So, if the appended model had an X coordinate of 9.0 for all joints, then it will be placed at a X coordinate of 9.0 in the current model.

The **Drawing Toolbar** will already be opened in the model view so that you can easily adjust the placement of your appended model using the graphics editing tools. The entire original model will automatically be unselected and your appended model will be selected. If you're going to make several adjustments to the position of your appended model, you should probably click the **Lock Unselected** button. Then you can use the graphical editing tools to put your appended model where you want it in the current model.

You can use the labels for the Joints, Members, and Plates to help identify the parts as you append them into your current model. Use the **Relabel Joints, Relabel Members**, and **Relabel Plates** options on the **Tools** menu to make the label prefix unique for each of your model parts before you append them.

Note

- The appended model properties will be set to the current model properties in cases where there is a conflict. For example, if both models use the same section set label, the current model properties will be used to define the section set. The same is true for material properties, moving loads, and so on.
- Loads remain in the same basic load case. For example if the appended model has loads in BLC 3 these loads will be placed in BLC 3 of the current model.
- Load combinations are not appended.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keyword: **Appending**.

Importing and Exporting Files

You may import RISA-2D files, STAAD files, DXF drawing files and BIM Exchange files. See DXF Files, Appendix C, and Appendix E for more on these file types.

When importing from RISA-2D, the model will be put in the 3D X-Y plane, i.e. the Z coordinates for all the joints will be '0'. The local y-axes are defined differently for the two programs and thus some members may "flip" and loads are automatically adjusted. The model should be reviewed to confirm that the members and loads are still headed in the correct direction.

To Import Other File Types

1. On the **File** menu click **Import**.
2. Choose the **File Type.**
3. Select the file you wish to import.
4. Choose from the options associated with the type of file. For help on an item, click and then click the item.
5. Perform a Model Merge. See Model Merge for more information.

To Export Other File Types

1. On the **File** menu click **Export**.
2. Choose the **File Type** in the drop down list.
3. Select the file you wish to export.
4. Choose from the options associated with the type of file. For help on an item, click and then click the item.

Importing RISA-2D version 4.0 and Higher Files

RISA-2D models may only be imported if they are from version 4.0 or higher and will be put in the 3D X-Y plane. "PIN" member releases will be converted to "BENPIN" in RISA-3D. If the 2D members are not assigned a database shape, the torsional stiffness, J, is set to '1'. The out of plane translational degree of freedom (Z), and the out of plane rotational degrees of freedom (2y and 2x), are fixed in the 3D model for all joints.

Automatic Backup

RISA-3D provides an automatic backup system for systematically backing up your model data. The purpose of this save file is to provide a means for you to recover your data in the event RISA-3D terminates abnormally. As you work within RISA-3D data is stored in a file called "__3DAUTO.TMP". When you exit the program normally, this file is renamed to "__3DAUTO.BAK".

By default, this automatic save operation takes place every 15 minutes, but you can alter the timing and location with the **Preferences** option on the **Tools Menu**.

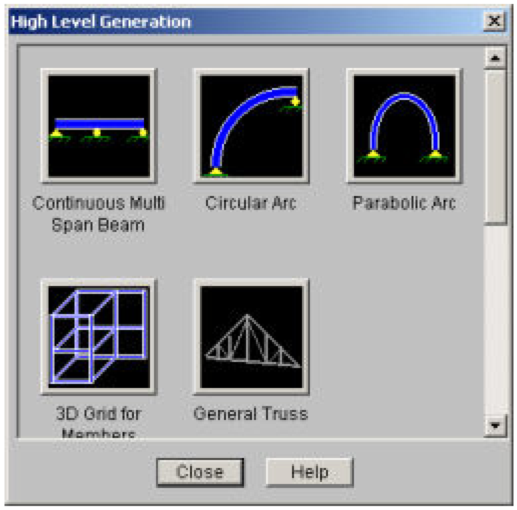
If RISA-3D does terminate abnormally, the next time you start the program this message will appear:

If this message appears and you answer "Yes", the data from the last backup of the previous session will be reloaded. If you answer "No", the backup data will be deleted and a new backup file will be started.

Generation

You may automatically generate regular structures or portions of structures to start a new model or add to an existing model.

Access the Generation types by clicking  on the RISA toolbar. You will see the following options, which require you to specify basic parameters to generate structures with beam elements, plates, or a combination of both.

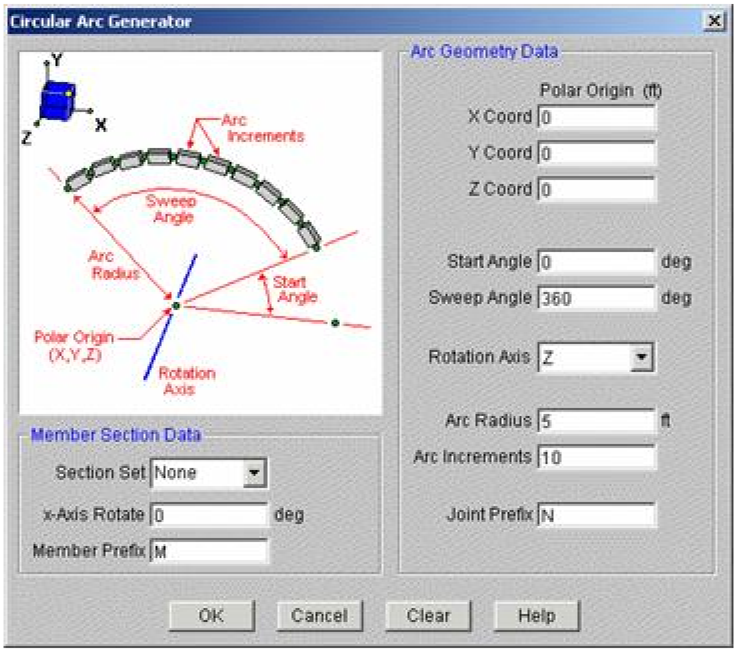


Note

- To generate items that are not coincident to the global axes first define them in the global directions and then rotate them into the correct position.
- The plate and member generation is optional for each generation. For example the cone generator can generate a cone of joints, members and plates or just a cone of joints.
- The generations are powerful but as a result of this some also have many options. Remember that any generation may be undone by clicking **Undo** so don't hesitate to try something just to see what it does.

Circular Arc Generation

The circular arc generation enables you to generate a full circle or partial arc comprised of beam elements.



The polar origin is the center point of the arc. A global axis (X, Y or Z) is entered as the axis of rotation and the arc will be in the plane normal to this axis. You may generate an arc the full 360 degrees around the axis of rotation or generate a partial arc by specifying the start and sweep angles.

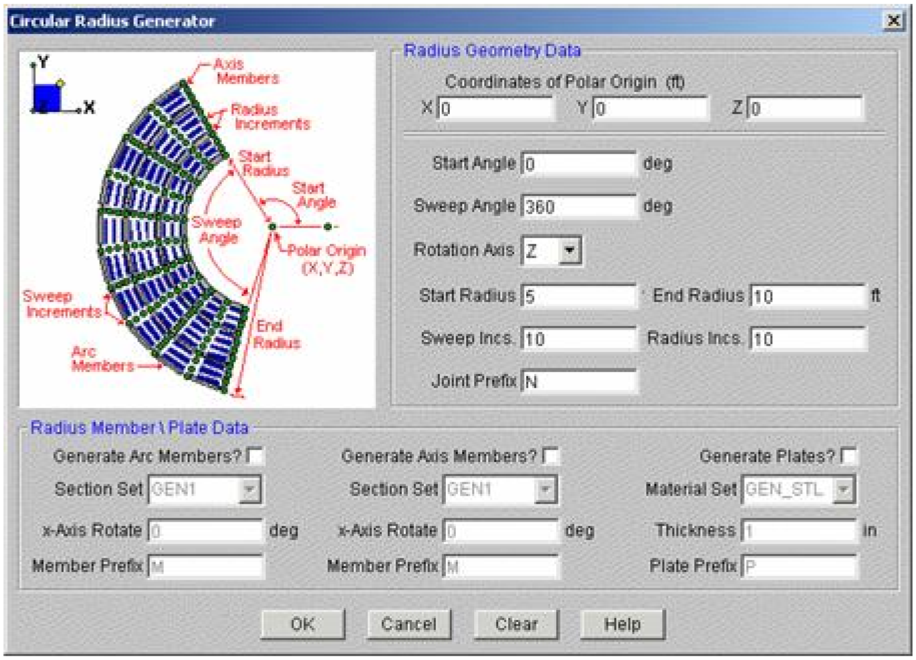
The arc radius is length from the polar origin to the arc. The arc increment determines how many piecewise straight segments are used to model the arc.

To generate members for the arc, you must select a valid section set. This section set will be used for all parts of the arc. If you don't select a section set, you will generate an arc of joints without members. The "x-axis rotate" may be used to rotate the local axes to a desired orientation, however you may find that a K-joint better serves this purpose in some instances. You may also have unique labels assigned to the generated members, by entering a start label.

For additional advice on this topic, and how to use the Circular Arc to model a Dome structure, please see the RISA News website: www.risanews.com. Type in Search keywords: **3D Dome**.

Circular Radius Generation

The circular radius generation enables you to generate a full or partial circular grid comprised of joints and optional members and plates.



The polar origin is the center point of the disk. A global axis is entered as the axis of rotation (X, Y or Z) and the disk will be in the plane normal to the axis of rotation. You may generate a grid the full 360 degrees around the axis of rotation or generate a partial grid by specifying the start and sweep angles.

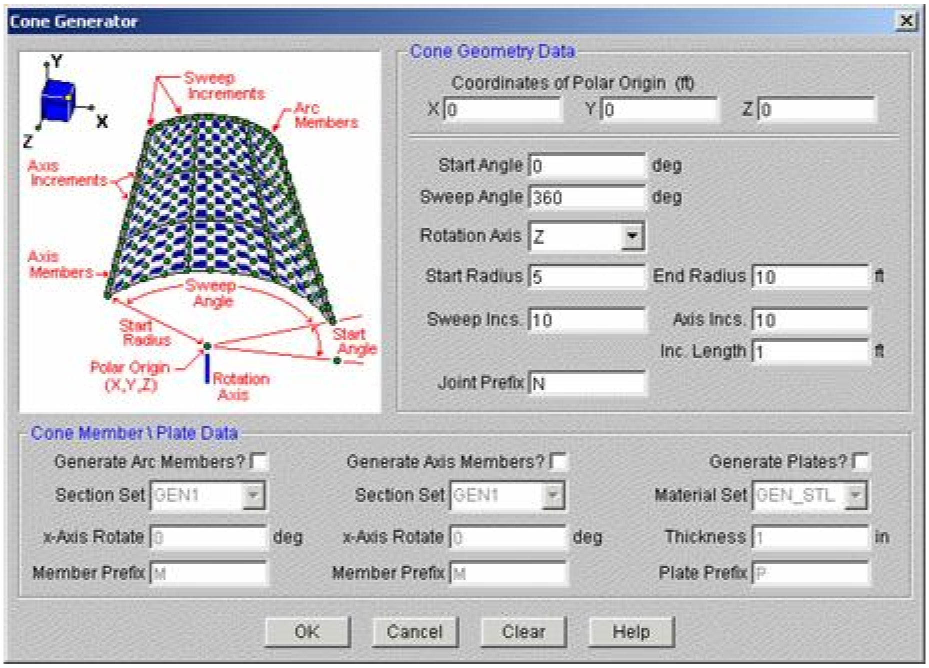
The start radius is the radius of the "hole" in the center. This distance must be greater than zero and less than the end radius entry. The end radius is total length from the polar origin out to the edge of the grid. The number of increments along the radius tells RISA-3D how many "rings" to use to create the grid. The sweep increment determines how many piecewise straight segments are used to model the arc.

There are entries for a section set for both the arc members and the radius members so these two sets of members can be different sizes. To generate these members you must select a valid section set. If you don't select a section set, you won't generate these members. You may also have unique labels assigned to the generated members by entering a start label.

Finally, you enter a material set and thickness to have plates defined. Only quadrilateral plates are generated. You may also have unique labels assigned to the generated plates by entering a start label.

Cone Generation

The cone generation is used to make cones comprised of joints and optional members and plates.



Specify the polar origin as the point about which the arc rotates and a global axis as the axis of rotation. You may generate a cone the full 360 degrees around the axis of rotation or generate a partial cone by specifying the start and sweep angles.

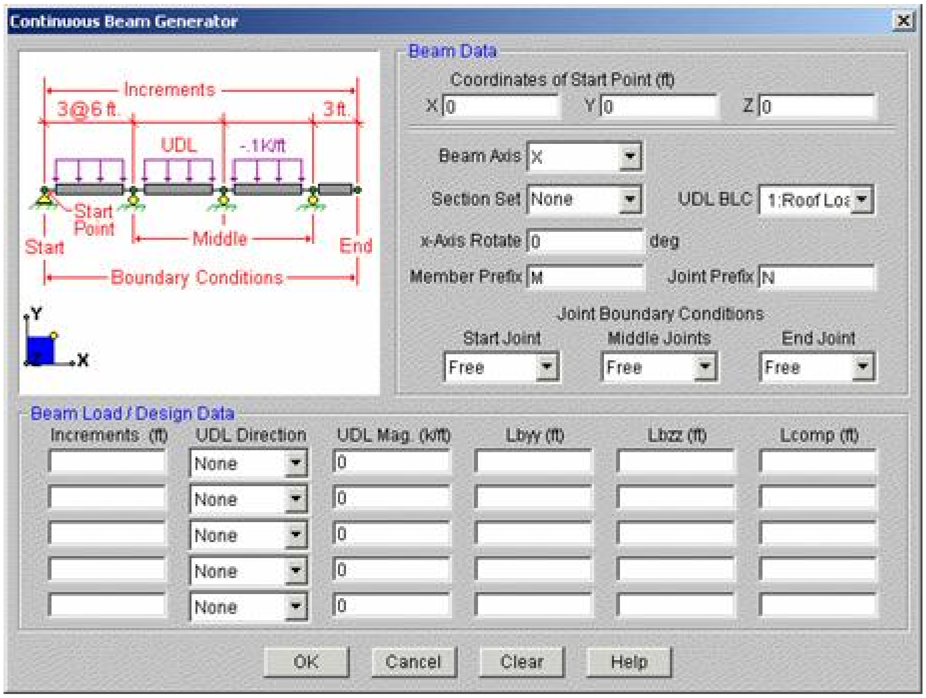
The start radius and end radius are the distances from the axis of rotation to the cone. The number of increments along the sweep determines how many piecewise straight segments are used to model the arc of the cone. The number and length of increments along the axis of rotation is used to "extrude" the cone in the direction of the axis of rotation.

There are entries for a section set for both the arc members and the axis members so these two sets of members can be different sizes. To generate these members you must select a valid section set. If you don't select a section set, you won't generate these members. Different labels may be assigned to the two member types. The “x-axis rotate “ may be used to rotate the local axes in to place however you may find that a K-joint better serves this purpose.

Finally, you may enter a material set, thickness and label for plates to be defined.

Continuous Beam Generation

The Continuous Beam generation will produce a complete model of various types of beams. You can simultaneously generate the beam geometry, section properties, boundary conditions, uniform distributed loads, and unbraced lengths for code checking.



Specify the start point for the beam generation. This defines the first point for beam and is also the "Start Joint" for any boundary conditions.

Choose which axis you want to be the longitudinal axis of the beam. (I.e., the beam will run parallel to this axis).

To generate members for the beam, you must select a valid section set. This section set will be used for all parts of the beam. If you don't select a section set, you will just get a sequence of joints.

If you are generating loads, select a Basic Load Case for any distributed loads. All the loads will put into this basic load case.

For any generated members, you may assign a "x-axis rotate angle", and member label prefix. The "x-axis rotate" may be used to rotate the local axes to a desired orientation, however you may find that a K-joint better serves this purpose in some instances.

You may optionally enter a Joint label prefix which will be used for all the joints generated as part of the beam. This is useful if you want a way to track individual parts of your model by looking at joint prefixes.

You can also generate boundary conditions for the joints along the continuous beam. You can specify different boundary conditions for the Start Joint, all the middle joints, and the End Joint. You choices for the boundary conditions are Free, Roller, Pinned, and Fixed. These are just like the regular boundary conditions that you would assign elsewhere in the program. The "Free" option is useful for the first or last joint if you want a cantilever overhang. The "Roller" option will only restrain the vertical translation. The "Pinned" option will restrain all the translations. The "Fixed" option restrains all the translations and rotations. See Boundary Conditions for more information.

Special symbols may be used to define multiple equal increments:

The "@" entry may be used to specify multiple, equally spaced, grid increments. For example, if you wanted 7 increments at 10 units each, you would type "7@10".

The "/" entry subdivides a larger increment into smaller equal increments. For example, the entry "12/4" would create 4 increments of 3 units each.

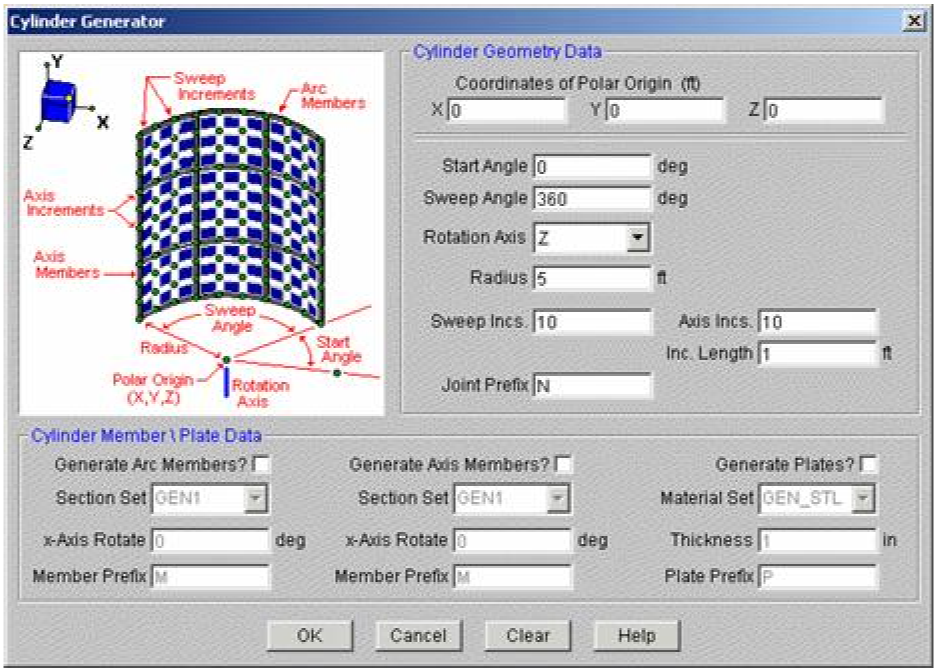
Use commas (",") to enter multiple increments in a single edit field. For example, if you wanted to define increments of 3, 4, 7 and 2 units, you could enter each increment in a separate field, or you could enter "3,4,7,2" in a single field to get the same result.

With each increment, you can also generate distributed loads by selecting a valid distributed load category and direction, and assigning a load magnitude.

Lastly, you can also generate unbraced lengths for your increments. These are used by the code checking to calculate axial and flexural stresses or strengths depending on what design code you are using. See Hot Rolled Steel Design and Wood / Timber Design for information on member design.

Cylinder Generation

The cylinder generation is used to make cylinders comprised of joints and optional members and plates.



The polar origin is the point about which the arc rotates and a global axis is the axis of rotation. You may generate a cone the full 360 degrees around the axis of rotation or generate a partial cylinder by specifying the start and sweep angles.

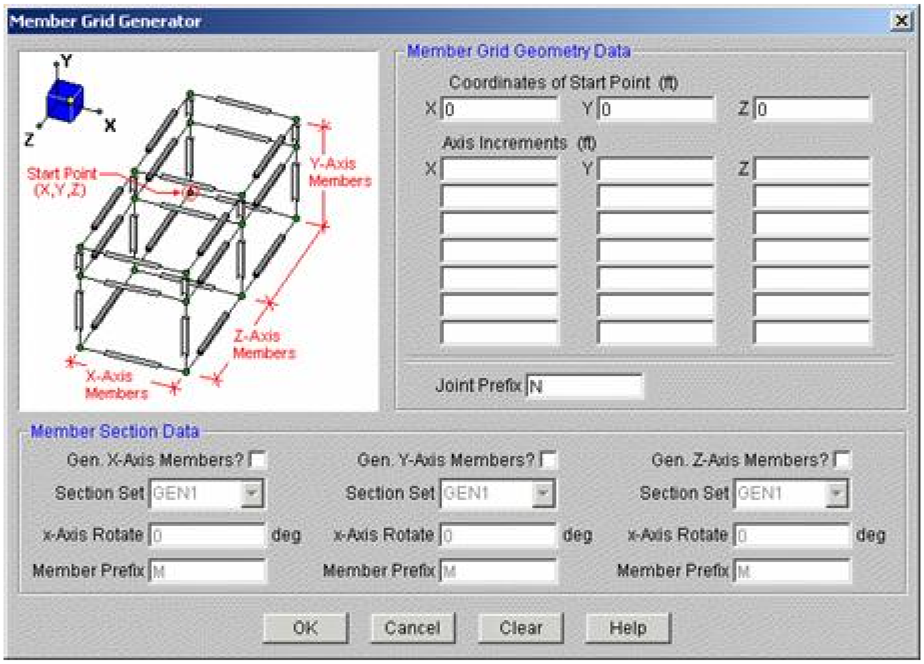
The radius is the distance from the axis of rotation to the cylinder. The number of increments along the sweep determines how many piecewise straight segments are used to model the arc of the cone. The number and length of increments along the axis of rotation is used to "extrude" the cylinder in the direction of the axis of rotation.

There are entries for a section set for both the arc members and the axis members so these two sets of members can be different sizes. To generate these members you must select a valid section set. If you don't select a section set, you won't generate these members. Different labels may be assigned to the two member types. The “x-axis rotate “ may be used to rotate the local axes in to place however you may find that a K-joint better serves this purpose.

You must enter a material set and thickness for plates to be defined and the label prefix may be specified as well.

Grid Member Generation

This generation enables you to generate 2D or 3D grids (or even 1D (line) grids) of joints and possibly members with equal or unequal grid increments.



Define the X, Y, and Z coordinates of the starting point and the increments for spacing in any or all three global directions. If you don't define any increments parallel to a particular axis no grids will be generated in that direction. Negative increments are OK. Joints are created at all the grid intersection points.

Special symbols may be used to define multiple equal increments:

The "@" entry may be used to specify multiple, equally spaced, grid increments. For example, if you wanted 7 increments at 10 units each, you would type "7@10".

The "/" entry subdivides a larger increment into smaller equal increments. For example, the entry "12/4" would create 4 increments of 3 units each.

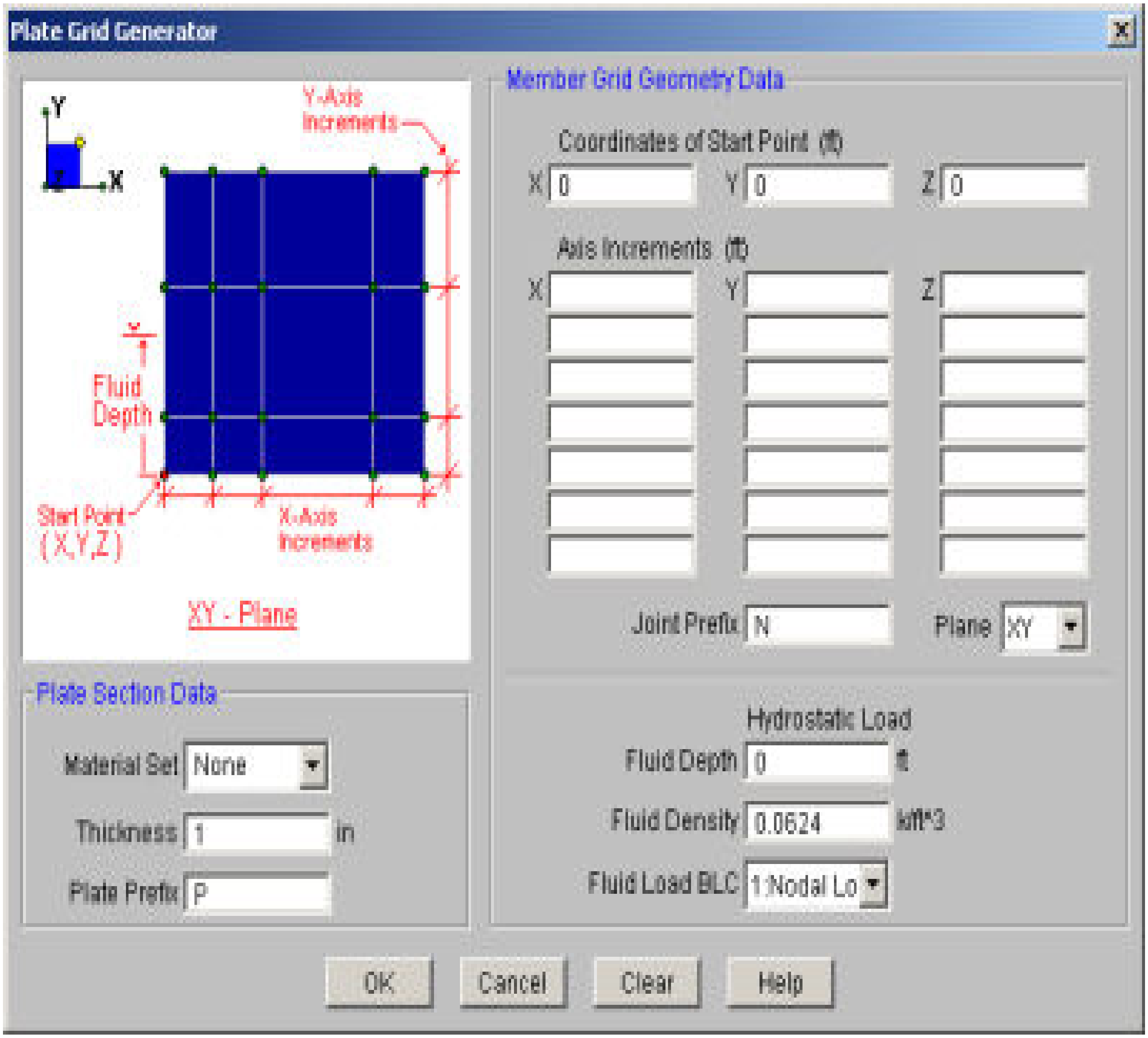
Use commas (",") to enter multiple increments in a single edit field. For example, if you wanted to define increments of 3, 4, 7 and 2 units, you could enter each increment in a separate field, or you could enter "3,4,7,2" in a single field to get the same result.

You may also have members created in the global directions between the grid points. You may use different section sets in the three global directions. For example if the Y-axis is the vertical axis you can specify a column shape for the Y-axis members and different or similar beam shapes for the X-axis and Z-axis members.

You may have unique labels assigned to the generated members, by entering a start label, and you may also use different labels for the three member directions.

Grid Plate Generation

The Grid Plate generation is used to make 2D and 3D grids comprised of joints and optional plates.



Simply define a starting point (enter the X, Y and Z coordinates of that point) and then define the increments for grid spacing in any or all three global directions. If you don't define any increments parallel to a particular axis no grids will be generated in that direction resulting in a 2D grid. Negative increments are OK. Joints are created at all the grid intersection points.

Special symbols may be used to define multiple equal increments:

The “@” entry may be used to specify multiple, equally spaced, grid increments. For example, if you wanted 7 increments at 10 units each, you would type “7@10”.

The “/” entry subdivides a larger increment into smaller equal increments. For example, the entry “12/4" would create 4 increments of 3 units each.

Use commas (“,”) to enter multiple increments in a single edit field. For example, if you wanted to define increments of 3, 4, 7 and 2 units, you could enter each increment in a separate field, or you could enter “3,4,7,2" in a single field to get the same result.

You may also have plates created in a global plane between the grid points and have unique labels assigned to the generated plates, by entering a start label. Keep in mind that the grid increments you define and the global plane you select must be

consistent for the plates to be generated. For example, if you defined increments only along the X and Y axes, you've defined an XY plane of joints. If you then enter "ZX" for the plane of the plates, no plates will be generated; only the joints in the XY plane would be generated. You would need to specify "XY" as the plane to have the plates generated.

A valid material set must be selected in order for plates to be generated. The default is "None", so you will need to select a material from the list of currently defined material labels to generate your plates.

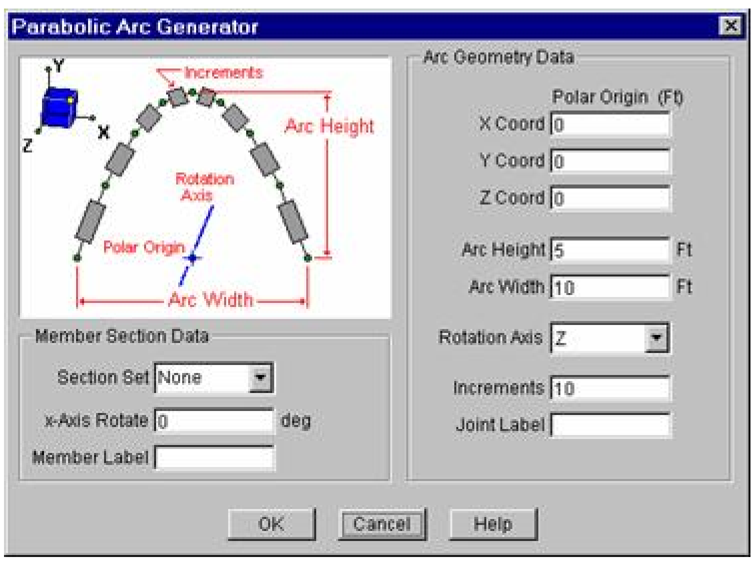
A hydrostatic load may also be generated along the grid of plates as you generate it. The hydrostatic load will be generated using a series of uniform surface loads on your plate elements. If the fluid depth is constant along the grid, you will just get uniform loads on your plates. If the fluid depth varies along the grid, you will get a series of uniform surface loads that increase in a "stair step" fashion , as you move down the fluid depth. The value of each uniform surface load is equal to the value of the actual hydrostatic load at the mid-point of the plate. The load direction will be perpendicular to the plates.

The fluid depth is measured in the positive vertical direction from the starting point for the plate grid.

The default fluid density is for water, but any density can be entered. The Fluid Load BLC provides a drop down list of all the Basic Load Cases. If you've defined in any descriptions for your BLC's, these will be shown in the drop down list. The surface loads generated will be placed in the selected Basic Load Case.

Parabolic Arc Generation

The Parabolic Arc generation is used to make arcs that are parabolic. The arc model will be comprised of joints and optional members.



Specify the polar origin about which the arc will be generated.

You must enter the arc height and the arc width and choose a rotation axis.

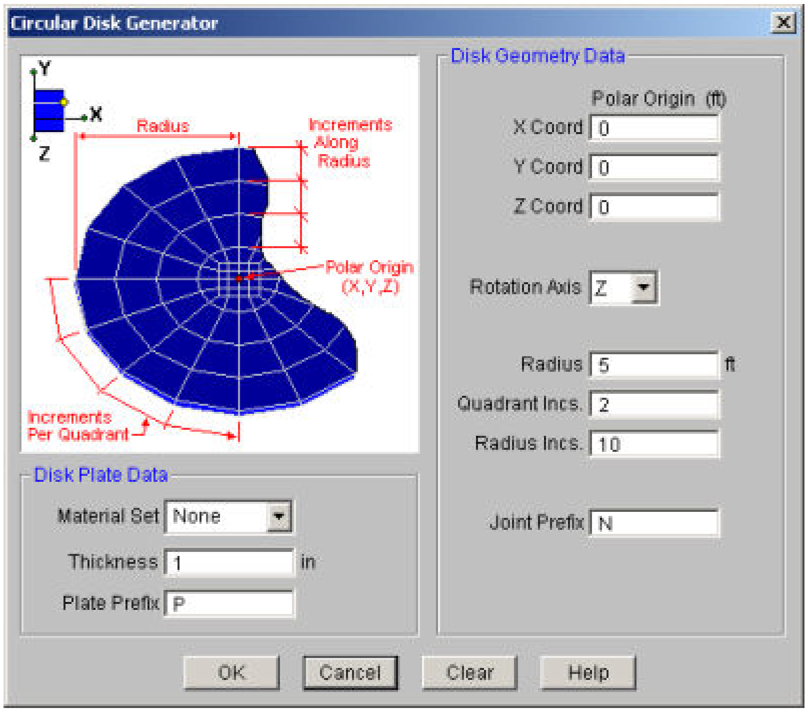
Enter the number of increment to be used to model the arc. The more increments used, the more closely the final geometry will follow the desired parabola. The minimum increments are two, which will give you a triangular shape.

You may optionally enter a Joint Label prefix that will be used for all the joints generated as part of the arc. This is useful if you want a way to track individual parts of your model by looking at joint prefixes.

To generate members for the arc, you must select a valid section set. This section set will be used for all parts of the arc. If you don't select a section set, you will generate an arc of joints without members. The "x-axis rotate" may be used to rotate the local axes to a desired orientation, however you may find that a K-joint better serves this purpose in some instances. You may also have unique labels assigned to the generated members, by entering a start label.

Circular Disk Generation

The Circular Disk generation enables you to generate a circular disk comprised of plate elements.



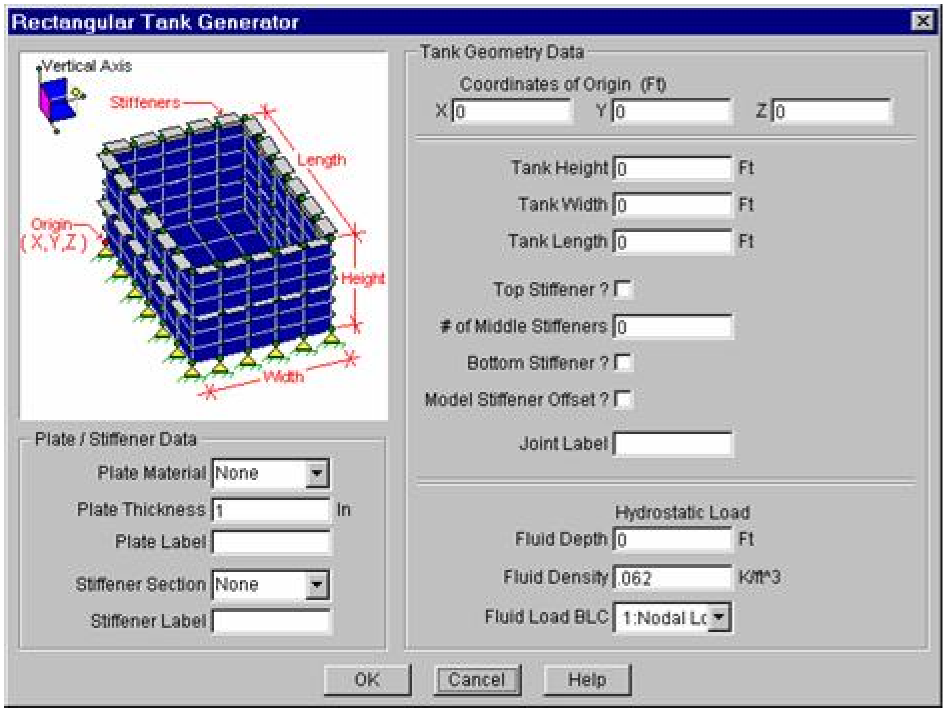
Specify the polar origin as the point about which the grid rotates and a global axis as the axis of rotation. The grid is generated the full 360 degrees around the axis of rotation.

The radius is total length from the polar origin to the outer edge of the grid. The quadrant increment determines how many piecewise straight segments are used to model the sweep for each quadrant of the arc. Note this entry must be a multiple of 2 (2,4,6, etc.) due to modeling constraints at the center of the disk. The radius increment specifies how many “rings” to use to create the grid.

You must enter a material set and thickness to have plates defined. Only quadrilateral plates are generated. You may also specify labels assigned to the plates.

Rectangular Tank Generation

The Rectangular Tank generation is used to make square or rectangular tanks out of plate finite elements. Stiffeners and hydrostatic loads may also be modeled. The tank will be pin supported around the bottom perimeter. No supports will be generated in the interior of the tank floor.



Specify the origin for the tank generation. This defines the first bottom corner of tank floor. The tank height will always be towards the positive vertical axis. The width and length will be along the other 2 axes as shown in the figure. For example, if the vertical axis is Y, then the width would be along the X-axis, and the length would be along the Z-axis.

Horizontal stiffeners may optionally be generated around the top, bottom, or at intermediate locations on the tank walls. The middle stiffeners are evenly spaced from the top and bottom of the tank. For example, specifying 2 middle stiffeners would divide the tank wall horizontally in thirds, with the first stiffener at one-third the tank height, and the second stiffener at two thirds of the tank height.

You may offset the stiffeners with rigid links in order to model the composite action of the stiffener and the tank wall due to the center of the stiffener being offset from the tank wall. When this option is not used, the stiffener and plate centerlines are at the same location and share the same joints. If this option is used, the generator will automatically create a material set for the rigid links called "TANK_RIGID_RISAMAT", and a section called "TANK_RIGID_RISASEC". The properties for these are preset and should only be modified if the rigid links are acting "flexible" relative to the rest of your model. Each time the generator is run, the properties are reset to the default values if these material/section sets have already been created.

Note

- You cannot use this option for Arbitrary shapes, Tapered WF shapes, and shapes that are defined on the section screen by typing in their area, moment of inertia's and torsional stiffness. These shapes can still be used as stiffeners; you just can't have the composite offset automatically calculated and modeled.

A joint label prefix may be entered, which will be used for all joints generated for the tank. A valid material set must be selected in order for plates to be generated. A plate label prefix may be entered, which will be used for all the plates generated for the tank.

A valid section set must be selected in order for stiffeners to be generated. A stiffener label prefix may be entered, which will be used for all the members generated for the tank stiffeners. Any rigid links generated to model the offsets will have a prefix of "RIGIDTANK"

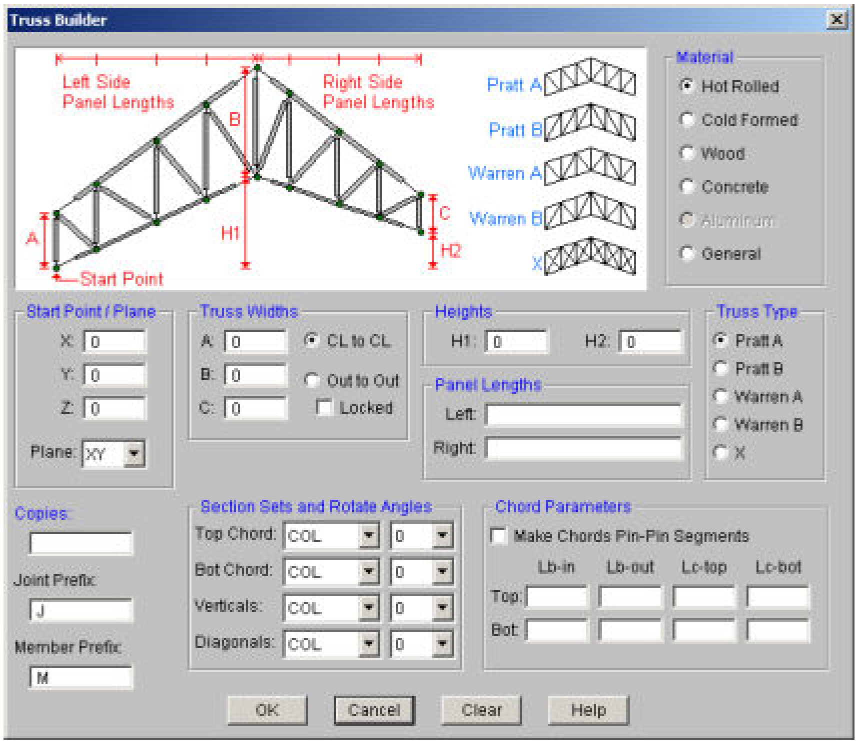
A hydrostatic load may also be generated. This load will be generated using a series of uniform surface loads on the plate elements. The fluid depth is measured in the positive vertical direction from the bottom of the tank and is constant along the floor giving a uniform surface load on the floor plates. The fluid depth will vary along the walls of the tank and these will increase in a "stair step" fashion, as you move down the fluid depth. The value of each uniform surface load is equal to the

value of the actual hydrostatic load at the mid-point of the plate. The load direction will be perpendicular to the plates and outward.

The default fluid density is for water, but any density can be entered. The Fluid Load BLC provides a drop down list of all the Basic Load Cases. If you've defined in any descriptions for your BLC's, these will be shown in the drop down list. The surface loads generated will be placed in the selected Basic Load Case.

General Truss Generation

The general Truss Builder generation is used to make various trusses and various forms of truss joists. The truss will be comprised of joints and an optional top chord, bottom chord, vertical strut member, and diagonal brace member. You may make the truss out of any material type (General, Hot Rolled, Wood, et cetera) by choosing your Material from the radio buttons provided.



The **start point** for the truss generation defines the first point of the bottom chord.

You must enter the **truss height** and the **panel lengths** to the left and right of truss "peak". You may use symbols such as "@", "/" and "," when entering the panel lengths.

The "@" entry may be used to specify multiple, equally spaced panels. For example, if you wanted 7 panels at 10 units each, you would type "7@10" in the panel lengths as shown in the image below.

The "/" entry subdivides a larger panel length into smaller, equal increments. For example, the entry "12/4" would create 4 panels of 3 units each as shown in the image above.

Use commas (",") to enter multiple, varied panel lengths. For example, if you wanted to define panels of 3, 4, and 7 units, you could enter "3,4,7" in the increment field as shown in the image above.

The **Truss Type** selects what configuration the truss will use. The truss plane defines the plane of the truss.

You can vary the elevation of the bottom chord by setting the Heights **(H1** and **H2**). You may adjust the elevation of the top chords by adjusting the truss Widths (**A, B, C**). In addition, you can specify whether these truss widths represent the centerline to centerline widths of the truss or whether they represent the total out to out dimension of the truss. If you check the "Locked" box, then that total out to out dimension will be maintained as the truss members are automatically optimized and resized.

You may optionally enter a Joint label prefix that will be used for all the joints generated as part of the truss. Similarly, you may enter a Member label prefix that will be used for all the members generated as part of the truss. This is useful if you want a way to track individual parts of your model by looking at member or Joint prefixes.

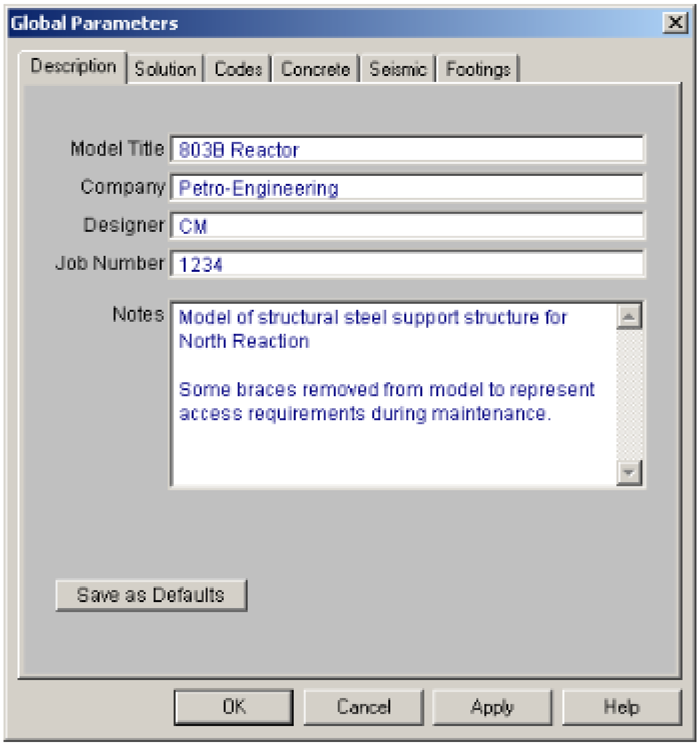
You may optionally create members for the bottom chord, top chord, vertical strut, and diagonal brace. For each one of these members you must assign a valid independent section set, and "x-axis rotate angle"**.** Note that you must mark the checkbox for a member type before you can set any of the values to indicate that you want generation performed for that member type.

The "x-axis rotate" may be used to rotate the local axes to a desired orientation, however you may find that a K-joint better serves this purpose in some instances.

Global Parameters

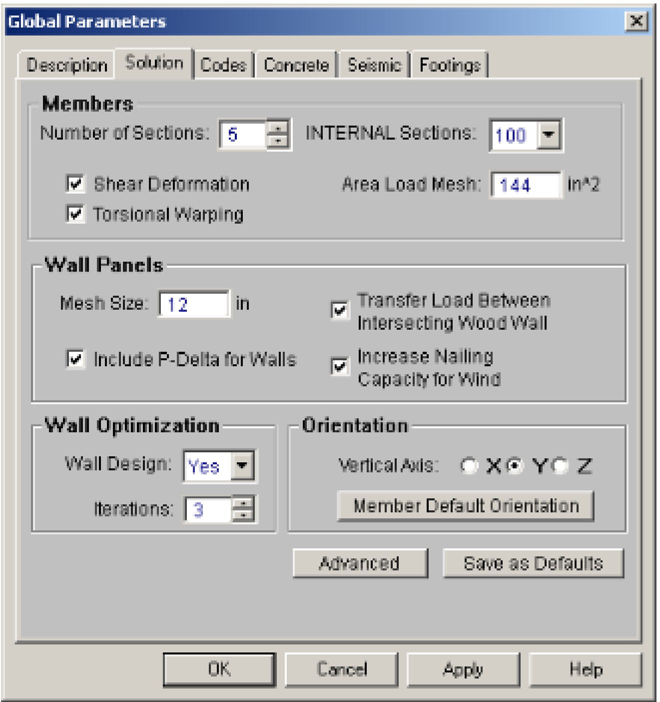
The **Global Parameters Dialog**, accessed through the **Options Menu**, is used to define information that influences the model and its solution in an overall (global) manner. You may save any of the information as the default setting so that when you start a new model that information is already there. To do this, simply enter the information that you want to save and click the **Save as Defaults** button.

Description



The entries under the **Description** tab are used to enter descriptive information such as a title for the particular model being defined, the name of the designer and a job number. The title may then be printed at the top of each sheet of the output, and on the graphic plot of the model. Note that for any of these tabs, you can click on **Save as Defaults** and then your settings will be saved the next time you open the program.

Solution



The entries under the **Solution** tab are used to control settings that affect the general solution of the model.

Members

Number of Sections controls how many places you receive **reported** member force, stress, torsion, and deflection results. This only affects the amount of data displayed in the results spreadsheets, and has no effect on the solution of the model or the code checks. See Printing and Member Results for more information.

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. Increasing this value means that the program will make more "cuts" along the beam's length, which means it is more likely to hit the theoretical maximum and minimum values for code checks.

Note:

- Number of Sections cannot exceed 20. Also, Number of Internal Sections cannot be less than twice the Number of Sections. If unacceptable values are entered for either of these fields the program will automatically reset them to acceptable values.
- In the embedded version of RISAFoundation in RISAFloor or RISA-3D, the Beam Section Options will not appear. In these cases, the Global Parameters from RISA-3D will control here.
- The Number of Sections and INTERNAL sections are remembered between RISAFloor and RISA-3D. If either of these is changed in RISA-3D, the RISAFloor results will be cleared as soon as RISAFloor is entered again. This is to keep results consistency between the programs.

Check the **Shear Deformation** box if shear deformation considerations are to be included in the model solution. See Member Shear Deformations for more information.

The **Torsional Warping** option considers torsional warping effects when calculating stiffness and stress values for shape types that warp. See Member Warping for more information.

The **Area Load Mesh** is used to determine the maximum size when meshing an area load and attributing the load to members. See Area Loads to learn more about this.

Wall Panels

The **Mesh Size** is the base mesh size that is used when wall panels solved. If there is a constraint smaller than this mesh size, the mesh will be refined to accommodate this constraint. You can see the mesh size graphically on the wall panels when you solve your model.

Include P-Delta for Walls is used to enable P-Delta analysis for wall panels. Even if this box is checked the P-Delta analysis will only be performed on load combinations which have P-Delta enabled.

The **Transfer Load Between Intersecting Wood Wall** allows the user to decide whether wall panels framing into each other will transfer loads. Wall panels that are parallel to and touching each other will always transfer loads. This was added because there are times when intersecting perpendicular walls are not actually attached to each other, thus may separate if loading conditions cause this.

The **Increase Nailing Capacity for Wind** option will automatically increase the shear capacity of wood wall and diaphragm panels by 1.4 for all load combinations that contain wind loads. Load combinations that have both wind and seismic loads acting simultaneously will not receive this increase.

Wall Optimization

The **Wall Design** option defines whether you want to automatically iterate the solution for masonry and wood walls.

The **Iterations** option defines how many automatic iterations will occur.

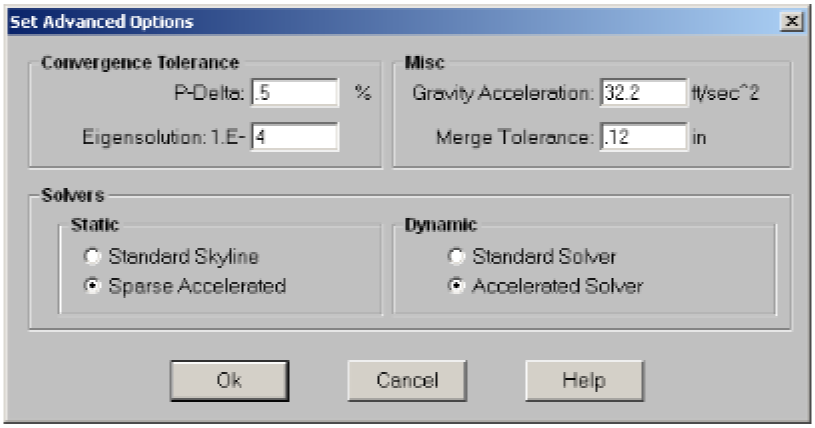
Note:

- This automatic iteration procedure will only happen for Batch or Envelope solutions. This is because we do not want to update your wall panel thicknesses if you are only solving a DL combination. In that case the program would then downsize your panel unnecessarily.
- For more information on masonry wall optimization see the Masonry Wall - Design topic.
- For more information on wood wall optimization see the Wood Wall - Design topic.
- Concrete wall design will be done automatically and the selections here will not affect this design. This is because concrete wall reinforcement changes will have very little effect on the stiffness of the walls.
- The program will stop the iteration procedure if the results from the previous solution match the results from the current solution.

Orientation

The **Vertical Axis** may also be set here as well. Setting the vertical axis may also require you to set the default member orientation also shown. As you specify new members RISA-3D will try to orient them correctly. The member local z-axis is typically the strong axis for a member and RISA-3D will orient members such that when you draw non-vertical members (beams) they will automatically be oriented such that loads in the vertical direction are resisted by the strong axis bending of the member. See Member Local Axes and Defining Member Orientation for more information.

Advanced



The **P-Delta Tolerance** is used to adjust the tolerance used to determine convergence of the P-Delta analysis. Be sure to enter this value as a *percentage*! The default for this is ½ of 1 percent (.5%). See P-Delta Analysis to learn more about this.

The **Convergence Tolerance** is used to set how close a subsequent solution must be to the previous solution for a mode to be considered converged. See Dynamic Analysis for more information.

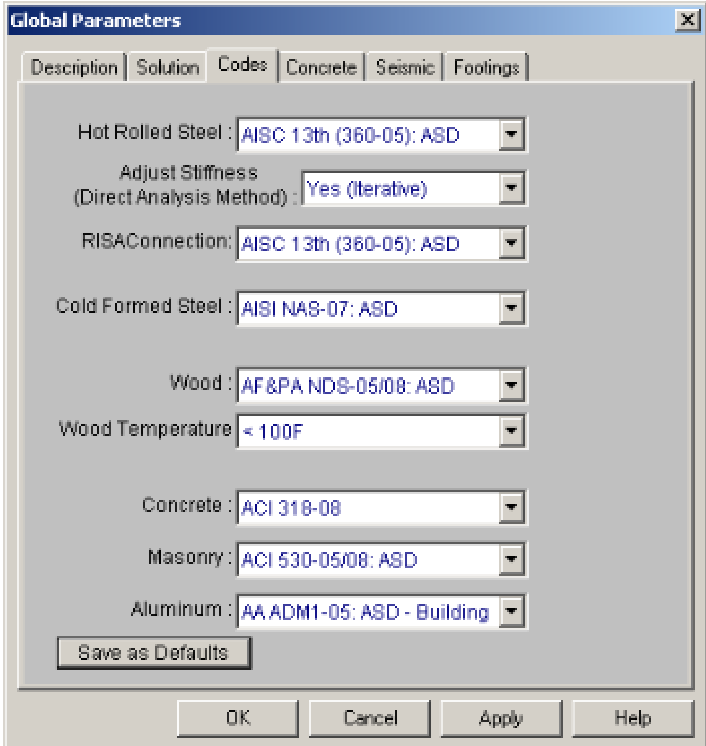
The **Gravity Acceleration** is used to convert loads into masses for the purpose of a dynamic analysis.

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.

The **Solver** to be used during solution may be selected by clicking the radial button next to **Standard Skyline** or **Sparse Accelerated**. See Solution for more information on these two options.

The **Dynamics Solver** options allow you to choose between the Standard Solver and the Accelerated Solver. The accelerated solver uses an accelerated sub-space iteration with a Lanzcos starting vector. The accelerated solver is the default and should produce solution in a fraction of the time that the standard solver would take to produce them. The Standard Solver uses a simple sub-space iteration to solve for the natural frequencies. This solver has been used for years and the accuracy of the results is very well established. It has been included only for comparative / verification purposes.

Codes



The entries under the **Codes** tab contain options related to the steel, wood, concrete code, or masonry checking.

The **Hot Rolled Steel** entry indicates which steel code is to be used in the design of steel shapes. Currently, the choices are:

- The 13th Edition of AISC's Steel Construction Manual (AISC 360-05 LRFD & ASD)
- The 2nd and 3rd Editions of AISC’s Steel Construction Manual for Load & Resistance Factor Design (LRFD)
- The 9th Edition of AISC's Steel Construction Manual for Allowable Stress Design (ASD)
- The 2000 Edition of the British code (BS 5950-1)
- The 2005, 2001 and 1994 Editions of the Canadian code (CSA-S16)
- The 2005 and 1992 EuroCode (ENV 1993-1-1)
- The 1998 Edition of the Indian code (IS 800)
- The 1997 Edition of the New Zealand code (NZS 3404)
- The 1998 Edition of the Australian code (AS 4100)

Note

- If the AISC 13th Edition code is selected (either ASD or LRFD), the **Adjust Stiffness** Menu will appear. The options in this menu control the stiffness reduction provisions listed in Appendix 7 (Direct Analysis Method) of the code. See Hot Rolled Steel - Design for more information on how the reduction is calculated and applied.

The **RISAConnection** entry indicates which code is to be used for connection design. This entry is only considered if you are also using RISAConnection. See the RISAConnection Integration topic for more information. Currently the choices are:

- The 13th Edition of AISC's Steel Construction Manual (AISC 360-05 LRFD & ASD)

The **Cold Formed Steel** entry indicates which steel code is to be used in the design of cold formed steel shapes. Currently, the choices are:

- The 2007 Edition of the AISI code (AISI-NAS-07 ASD and LRFD)
- The 2001 Edition of the AISI code with the 2004 Supplement (AISI-NAS-01/04 ASD and LRFD)
- The 1996 Edition of the AISI code with the 1999 Supplement (AISI-99 ASD and LRFD)
- The 2001 Edition of the Canadian code (CSA S136-01 LSD)
- The CANACERO-2001 (ASD and LRFD)

The **NDS Wood** entry indicates which wood code is to be used in the design of wood members. Currently, the choices are the 1991/97, 2001 and 2005/08 Editions of the NDS allowable stress design specification (AF&PA NDS). The 2005 and 2008 specifications had no differences regarding any design checks in the program.

The **NDS Temperature** is used to calculate the temperature factor (Ct) used in the design of wood members. See Section 2.3.3 of the NDS for more information.

The **Concrete** entry indicates which concrete code is to be used in the design of concrete members. Currently, the choices are:

- The 2011, 2008, 2005, 2002 and 1999 Editions of ACI 318
- The 2004 and 1994 Editions of the Canadian code (CSA-A23.3)
- The 2004 Edition of the Mexican code (NTC-DF)
- The 1997 Edition of the British code (BS 8110-1)
- The British publication of the 2004 Eurocode (BS EN 1992-1-1)
- The 1992 EuroCode (EN 1992-1-1)
- The 2000 Edition of the Indian code (IS 456)
- The 2001 Edition of the Australian code (AS 3600)
- The 1995 Edition of the New Zealand code (NZS 3101)
- The 2007 Edition of the Saudi Building Code (SBC 304)

Note:

- The 2011 and 2008 versions of the ACI code had no differences between them regarding any checks in the program.

The **Masonry** entry indicates which masonry code is to be used in the design of masonry walls. Currently the choices are:

- The 2008 edition of the ACI 530 (ASD & Strength)
- The 2005 edition of the ACI 530 (ASD & Strength)
- The 2002 edition of the ACI 530 (ASD & Strength)
- The 1999 edition of the ACI 530 (ASD)
- The UBC 97 (ASD and Strength)

Note:

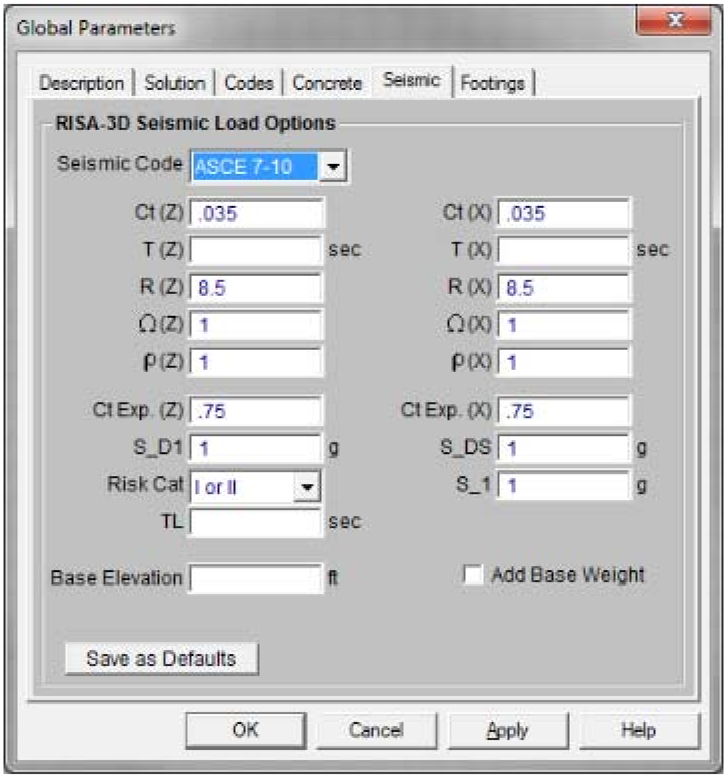
- The 2005 and 2008 versions of the masonry specification had no differences between them regarding any checks in the program.
- When you choose UBC 97 ASD, there is an option for whether you will have special inspection during construction. This will affect some values in design.

The **Aluminum** entry indicates which aluminum code is to be used in the design of aluminum members. The Building and Bridge options refer to the Safety factors that are used in the model. Currently the choices are:

- The 2005 edition of the AA ADM1 ASD Building
- The 2005 edition of the AA ADM1 ASD Bridge

Seismic

The options on the **Seismic** tab present the global parameters that are specifically related to calculation of code prescribed Seismic Loads. This information can be used by RISA-3D to automatically generate the Lateral Loading on your structure. Depending on which Seismic Code you select will depend on the input options that you have.

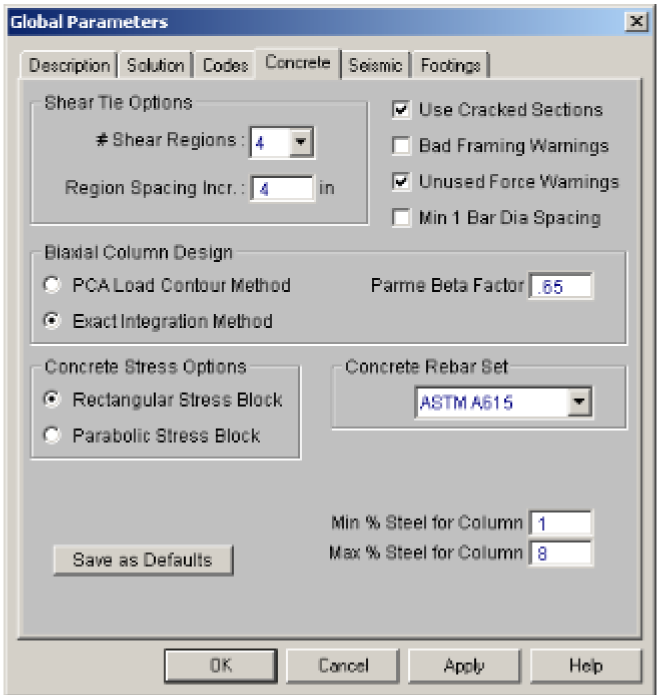


Most of the input fields on this tab are discussed in the Seismic Load topic. Below are a few inputs which are specific to this tab.

(Ω) is the **Overstrength Factor**. This can be included in the seismic load combinations generated by the LC Generator.

The **Redundancy Factor** (ρ) is based on the extent of redundancy in your structure. This can be included in the seismic load combinations generated by the LC Generator.

Concrete



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 **Biaxial Column Design Options** controls which method will be used to determine the biaxial column capacity. The options are the **PCA Load Contour Method** and the **Exact Integration Method**. The **Parme Beta Factor** is used to approximate the column's 3D interaction surface when using the PCA Load Contour Method. See Biaxial Bending of Columns for more information.

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**. See Parabolic vs. Rectangular Stress Blocks for more information.

Check the **Used Cracked Sections** box if you want to modify the member stiffnesses by the **Icr Factor** as described in Concrete - Design.

Check the **Bad Framing** or **Unused Force Warnings** check box to see force warnings on the detail report or bad framing warnings in the Warning Log.

Checking the **Minimum 1 Bar Dia Spacing** box will allow a minimum spacing of one bar diameter between parallel bars. Otherwise, RISA will default to a two bar diameter or one inch clear spacing, whichever is greater, to allow for lap splices and continue to maintain adequate spacing between parallel bars.

The **Concrete Rebar Set** allows you to choose from the standard ASTM A615 (imperial), ASTM A615M (metric), BS 4449 (British), prENV 10080 (Euro), CSA G30.18 (Canadian), and IS 1786 (Indian) reinforcement standards.

The **Minimum % Steel for Columns** allows you to choose the minimum percentage of reinforcement steel to be used in a concrete column section. The percentage entered will be multiplied times the gross area of the column section to determine the minimum amount of reinforcement required in each column. It should be noted that the minimum percentage allowed by section 10.9.1 in ACI 318-11 is 1%.

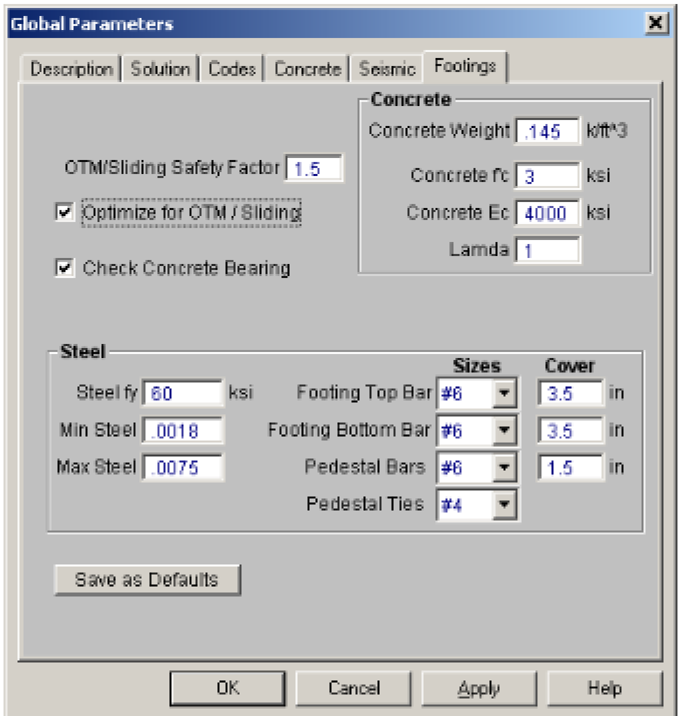
The **Maximum % Steel for Columns** allows you to choose the maximum percentage of reinforcement steel to be used in a concrete column section. The percentage entered will be multiplied times the gross area of the column section to determine

the maximum amount of reinforcement allowed in each column. It should be noted that the maximum percentage allowed by section 10.9.1 in ACI 318-11 is 8%.

Note:

Concrete cover for beams and columns is specified under Design Rules under the Concrete Rebar tab.

Footings



The entries under the **Footing** tab are used to define design criteria and material properties for the footings. These fields are only available if you have installed the latest version of RISAFoot.

RISAFoot checks bearing stresses for axial loads only. Under some circumstances (high moments/shears with low vertical loads) the concrete bearing check done by RISAFoot could be misleading. You have the option of excluding it from the analysis by clearing the "**Check Concrete Bearing**" check box.

The Optimize for OTM/Sliding checkbox will dictate whether or not the footing should be optimized to conform to the safety factors specified on the Load Cobminations spreadsheet. For new design this box would normally be checked. However, to match results produced by older versions of RISAFoot (which did not have the ability to optimize for OTM/Sliding) this box should be unchecked.

Concrete Properties

The concrete properties required by RISAFoot are the weight, strength (f'c) and modulus of elasticity (Ec). **Lambda**, **λ** is the lightweight concrete modification factor. This factor only applies to the *ACI 318-08*, *ACI 318-11*, and *CSA A23.3-04* codes.

Steel Properties

The reinforcing steel properties are the strength (fy) and the minimum and maximum steel ratios.

Acceptable steel ratios are controlled by ACI 318-11 section 7.12.2.1 which stipulates that the minimum steel ratio for Grade 40 or 50 steel is .0020. For Grade 60 the ratio is 0.0018. The ratio cannot be less than 0.0014 for any grade of steel. Please keep in mind that section 7.12 explicitly states "gross concrete area" is to be used.

In addition, the codes limit the maximum steel that can be used. The 1995 code and 1999 code limit the steel ratio to 75% of the balanced condition. The 2002 and newer codes instead limit the minimum strain on the reinforcing steel to 0.004. RISAFoot enforces these conditions depending on the code you specify.

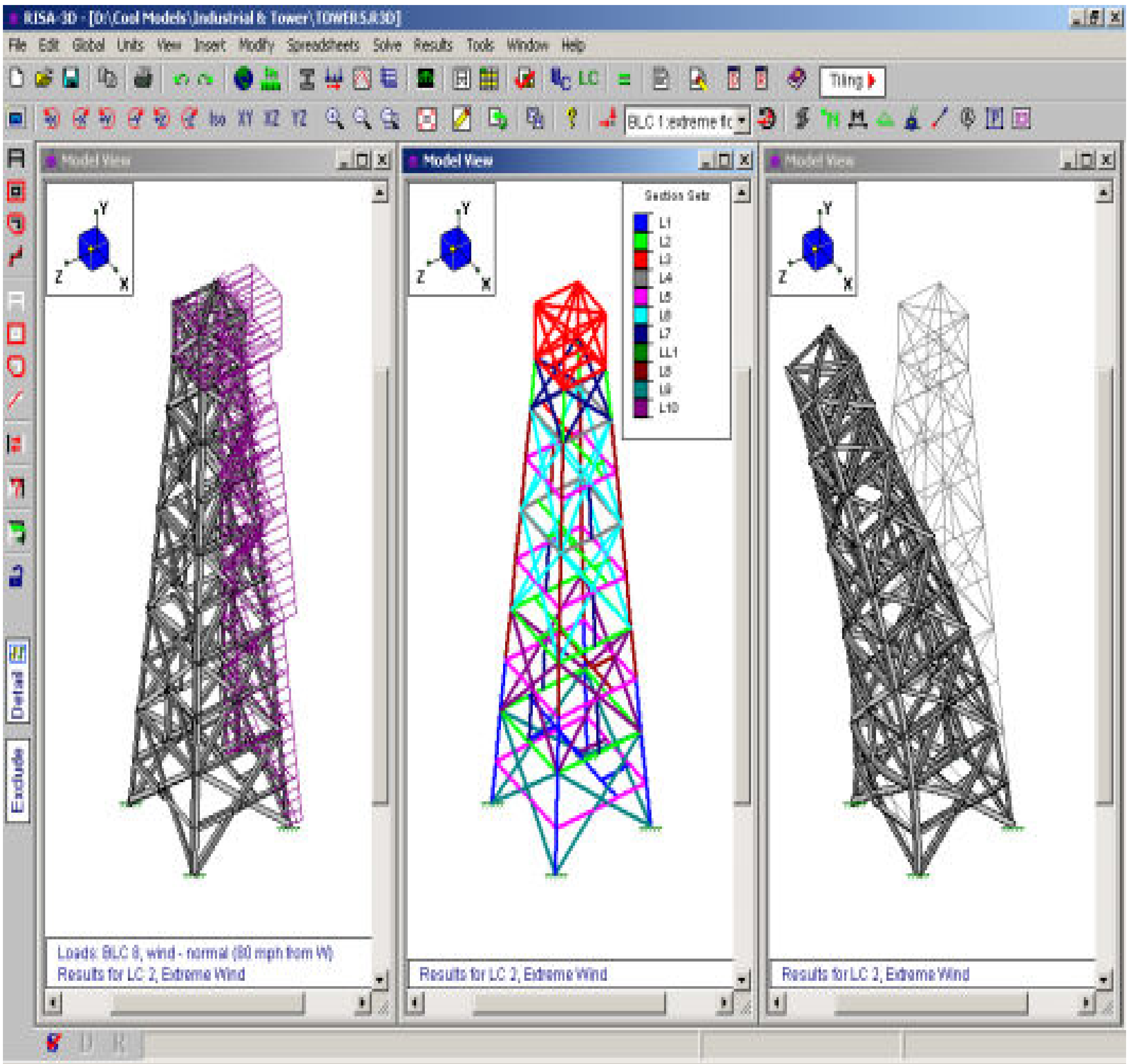
The **Sizes** section of this dialog allows the user to set different bar sizes for the top and bottom bars as well as the pedestal reinforcement and ties. The sizes are selected from the corresponding drop down list.

The **Cover** section of this dialog allows the user to set concrete cover for top and bottom bars as well as the pedestal reinforcement. The cover defines the distance from the concrete face to the outside face of the reinforcing steel.

Graphic Display

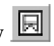
RISA-3D's robust graphics make building and understanding your model much easier. You may have numerous independent views open and active at the same time. Fast redraw times, responsive panning, true to scale rendering, animation capabilities, and graphic editing tools all help you build and understand your model. You may draw the model and specify loads and boundary conditions graphically. These items may be modified graphically as well. Verification of your model is made simple with true-to-scale rendering and color coded code checks. Results such as member force diagrams, color-coded stress levels, deflected shapes, and animations may also be viewed for solved models.

Multiple Windows



You may have multiple views of the model open at the same time, each with it's own characteristics. For example you can simultaneously view and work with members in one window and plates in another. You can have multiple spreadsheets open as well.

To Open a New View or Clone an Existing View

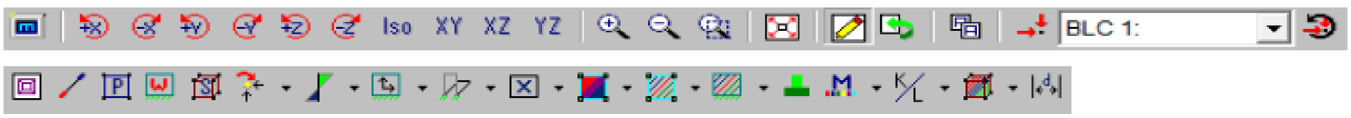
- To open a new view of the model click the **New View**  button. Alternately, you can open a new view that is an exact replica of the current view by clicking the **Clone View**  button.

Note

- With the model view active you may then use the **Window Toolbar** to adjust the view of the model.

Controlling the Model View

The two main tools to help you manipulate the views of the model are the **Window Toolbar** and the **Plot Options**.



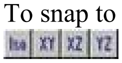
The **Window Toolbar,** discussed in this section, is the second toolbar from the top when a model view is active. You may use these buttons to rotate the model, zoom in and out, render the model, and display common items such as joint labels, boundary conditions, and loads.

The **Plot Options Dialog** (See Plot Options for more information) is opened by the first toolbar button  and provides various options not otherwise available on the toolbar. It allows you to specify how to plot joints, members, plates, and loads and the information that is to accompany them. The tabs across the top of the dialog break things up into basic elements to help you find the options.

Adjusting the Viewpoint

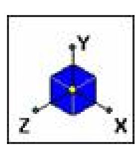
The remaining buttons on the **Window Toolbar** help you change the viewpoint by rotating, snapping, and zooming the view. You may use the scroll bars and the mouse wheel (if you have one) to pan the view.

To Rotate a View

- When holding down the Shift key, the mouse wheel button can be used to dynamically rotate the structure. See Application Interface - Dynamic Controls for more information.
- To rotate a view of the model about a global axis, click any of the **Rotate** buttons on the **Window Toolbar:** 
- To snap to an isometric or planar view of the model click any of the **Snap** buttons on the **Window Toolbar:** 

Note

- As you rotate the view, the global X-Y-Z axis orientation is shown in the view.



To Zoom In and Out

- Use the wheel button on the mouse to dynamically zoom in or out the model view. Rotating the wheel forward zooms in and rotating the wheel backward zooms out.
- To zoom in towards the center of the model view click the **Zoom In** button or press the + key.
- To zoom out from the center of the model view click the **Zoom Out** button or press the **-** key.
- To zoom in on a specific area of the model view click the **Zoom Box** button and then draw a box around the zoom area by clicking and dragging with the left mouse button.
- To snap the model to a full model view click the **Full Model View** button.
- The Function Keys, F3 and F4, are for the Zoom Previous and Zoom Next functions respectively. The system holds a total of 10 zoom-states. The F3 or F4 keystroke moves the active pointer forward or backward on the list of zoom states. If multiple model views are present, each window will have its own zoom list.

To Pan the View

- Clicking and holding the mouse wheel button triggers a dynamic pan and allows the user to drag the current model view to the limits of scroll bars.
- If you do not have a mouse wheel you may drag the scroll bars and click the arrows to pan the view.

Enabling Graphic Editing

The **Graphic Editing Toolbar** or **Drawing Toolbar** may be turned on to graphically edit the model at any time and it may be turned off to provide a larger view. See Graphic Editing for more information.

To Toggle the Graphic Editing Toolbar

- To toggle the graphic editing toolbar on or off, click the button on the **Window Toolbar**.

Saving Views

Click the button to save and recall views. If you have a view that you like to work with or have created a view that took some time to set up, save it with this feature. All of the plot options are saved with the model for later recall. See Saving and Retrieving Model Views for more information.

Cloning Views

You can clone a view. This allows you to preserve the original view and modify the clone. This is useful in achieving two views that have similarities in some way.

To Clone the Model View

- Click the **Clone View** button on the **Window Toolbar.**

Help with the Model Views

If you click the **Help** button while in a model view, the help file will open in the Controlling the Model View topic.

Displaying Loads

Use these buttons to control the display of loads. You may display one basic load case or an entire load category or load combination. The first button turns the display of loads on and off in the current view. The pull down list controls which load case, category or combination is shown in the view. Click the last button to choose between plotting load cases, load categories or load combinations. More control over the display of loads is provided in the **Plot Options Dialog** discussed in the next section.

Toggle Buttons

The last buttons on the **Window Toolbar** are toggles that plot information in the current view.

Click to toggle model rendering. See Plot Options for more information on rendering.

Click to toggle the joint labels.

Click to toggle the member labels on and off. If rendering is on, member labels will not be visible.

Click to toggle the boundaries.

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

Click to toggle the I-J end flags.

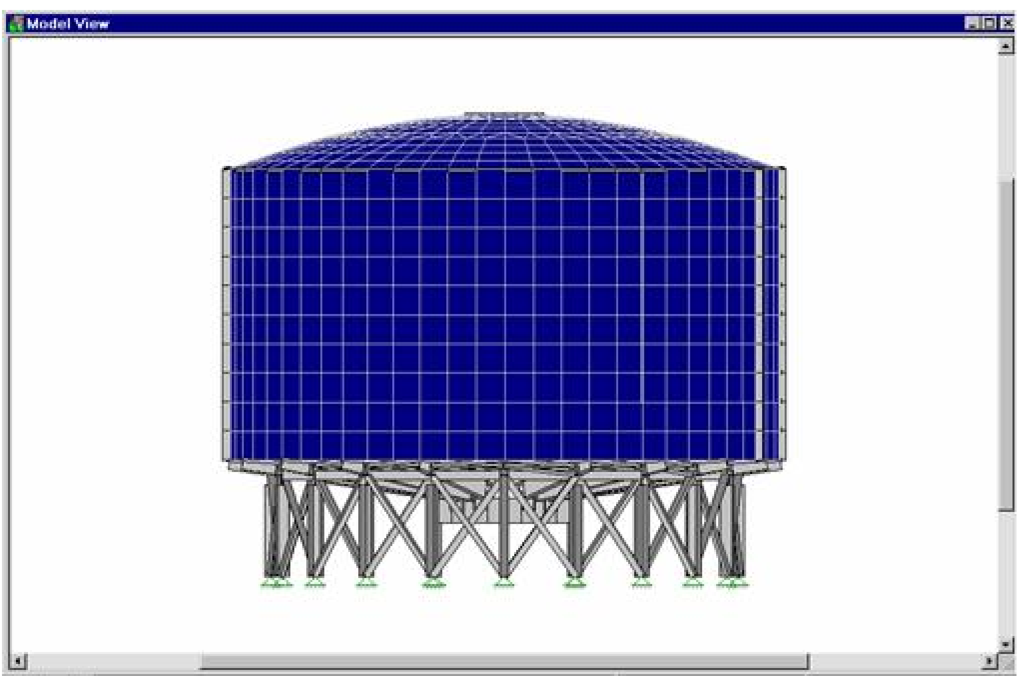
Click to turn the project grid on and off in the current view.

Click to toggle the display of diaphragms on and off.

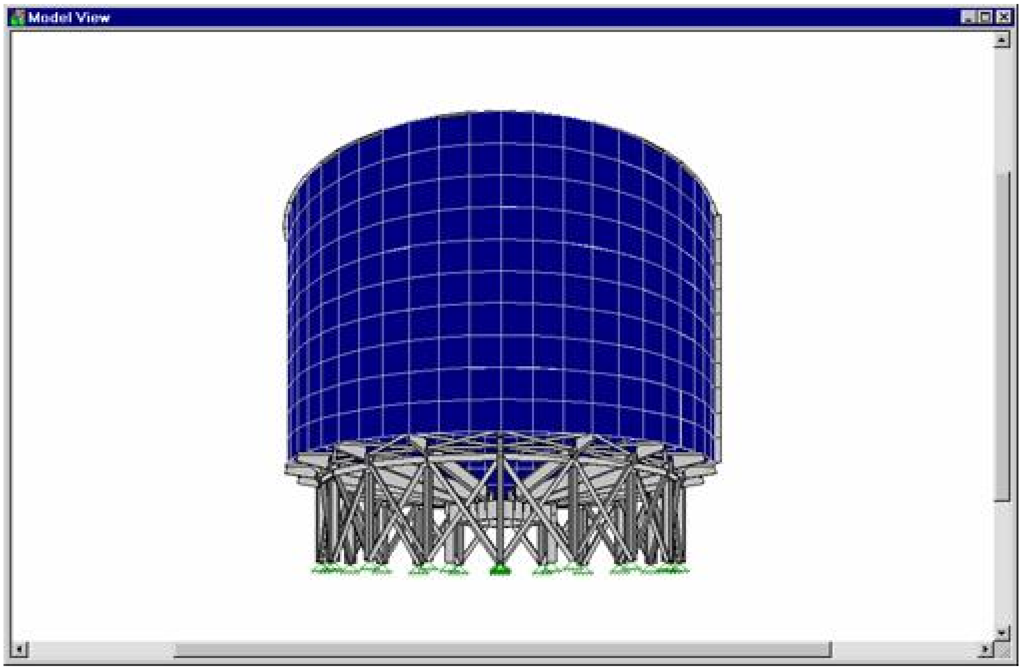
Click to toggle the display of solid labels on and off.

Depth Effect

Right click in the model view and click **Depth Effect** on the **Shortcut Menu** to access the depth effect options. Depth effect is used to increase or decrease the "depth" effect of the plot. Increasing the depth effect causes points farther away from the screen to converge towards a vanishing point, giving the image a more realistic three-dimensional appearance. Removing the depth effect causes the vanishing point to become infinity (no vanishing point). The button resets the view to have no depth effect.



The image above has no depth effect and the image below has depth effect.



Viewing Part of the Model

You may want to view only part of your model, especially if it is a big one. You may use the selection tools to give you the view that you want. You can graphically unselect parts of the model that you don't wish to see, or you can use a range of coordinates or other criteria to specify what to view. See Graphic Selection for more information.

Saving and Retrieving Model Views

You can save and recall views for a model. Saved views are model dependent so any views you save will stay with the model. A saved view includes information such as the current view angle, zoom state, pan location, plot option settings, etc. Saved views do NOT include the selection state for the model. You can save selection states separately. See Saving Selections for more information.

To Save the Model View

- Click the **Save View**  button on the **Window Toolbar.**
- Click the **Save** button and enter a name and then click **OK.**

To Retrieve a Saved Model View

- Click the **Save View**  button on the **Window Toolbar.**
- From the drop down list, choose the view you wish to recall and then click the **Retrieve** button.

To Delete a Saved Model View

- Click the **Save View**  button on the **Window Toolbar.**
- From the drop down list, choose the view you wish to delete and then click the **Delete** button.

Graphic Display - Plot Options

The **Window Toolbar** (See Graphic Display for more information) offers some common plotting choices, but these are just a few of the options available. Many more options are located in the **Plot Options Dialog** which may be accessed by clicking the button on the Window Toolbar or pressing the F2 function key.

The options are organized into groups of items. Access each group by clicking on the tabs along the top of the dialog box. Some of the options are mutually exclusive and others are conditional. Typically, only one option in a vertical row may be chosen at a time and indented options will not be applied if the heading option is not also applied.

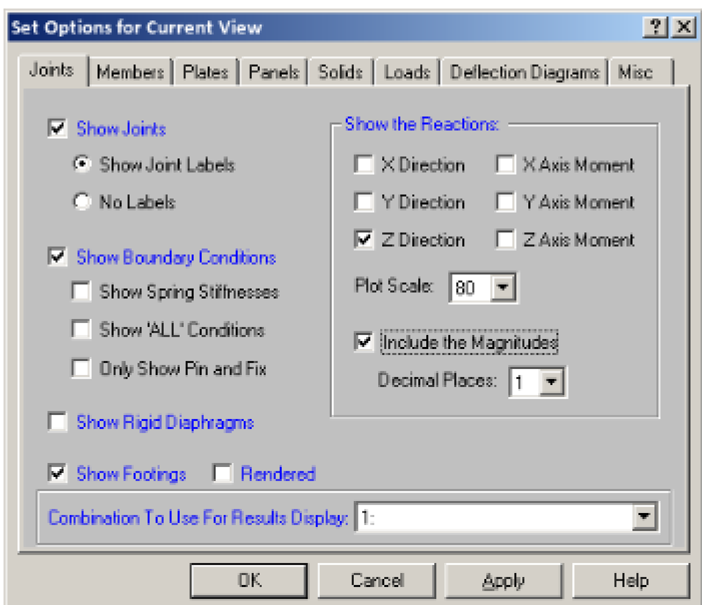
Each of the groups presents the option to turn off those particular items in the view. This is not the same as unselecting the items. Turning items on and off is independent from selecting and unselecting. For example if you make a selection of members and then turn off members all together, the selection still stands for any modification applied to members.

Note

- You may make any plot option settings the default startup settings. To do this, go through all the plot options tabs and set all the options to what you want as the startup default. Once that is done, select the **Misc** tab and press the button labeled **Make Current Plot Settings the Default**. Remember, this button applies to ALL tabs in the plot options dialog.

Joints

Access the plotting options for joints by clicking the **Plot Options** button on the **Window Toolbar** and selecting the **Joints** tab shown below.



You may specify that joints are, or are not, displayed and you may include the joint label.

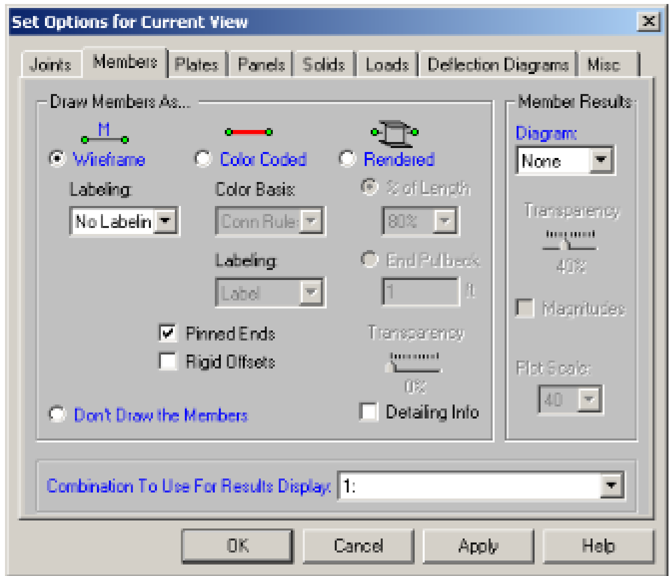
You may display boundary conditions and as part of this option you may display any spring stiffnesses that you have specified. The “All” boundary condition will not be shown unless it is also specified.

Rigid diaphragms will be plotted if its box is checked.

If a single or a batch solution has been performed you may choose to show the reactions in any of the six degrees of freedom and include the magnitudes. For batch solutions you must choose which combination you want to view.

Members

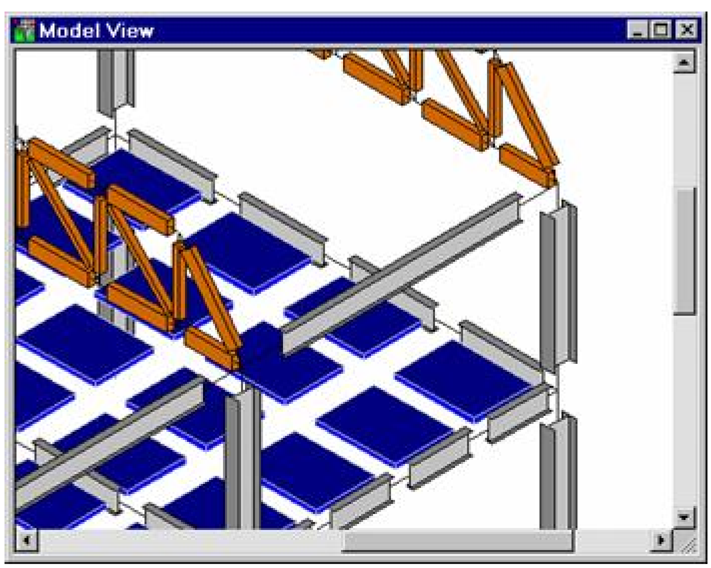
Access the plot options for members by clicking the **Plot Options**  button on the **Window Toolbar** and selecting the **Members** tab shown below.



You may specify that no members are drawn, or you may draw them as wireframe, color coded, or rendered.

With the **wireframe** and **color-coded** options you may include textual member information alongside each member. The **Labeling** drop down list provides the choices you have as to what information will be displayed alongside each member. The wireframe option also allows you to display member pinned end conditions and either rigid end offsets or I/J End representations at the member ends.

Color-coding of members plots the members using various colors to represent particular information such as the section set assigned to each member, stress levels, etc. The **Color Basis** drop down list provides the choices you have as to how the colors are to be assigned to each member. If a single or batch solution has been performed you may color the members by the code check or the stress magnitudes. For batch solutions you must choose which combination you want to view. The key that defines these colors is shown in the upper right corner of the model view.

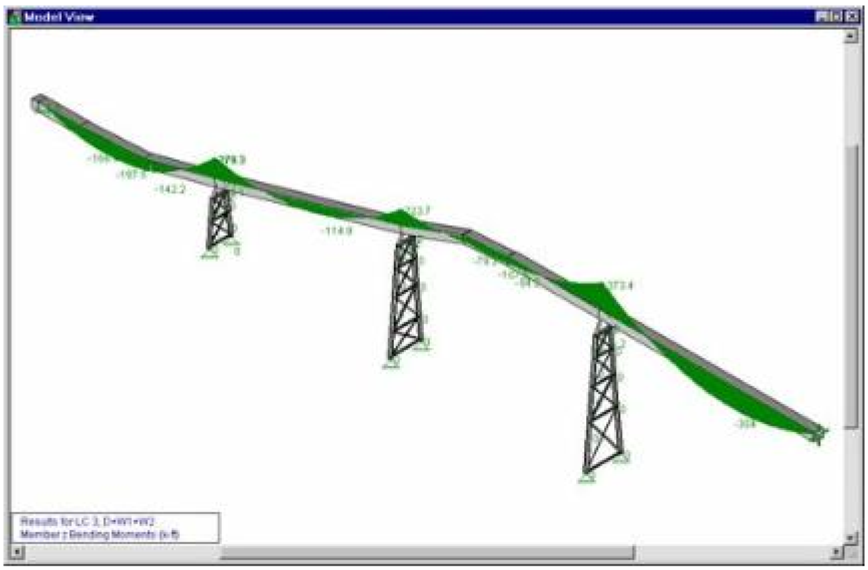


The **rendering** option will produce true-to-scale renderings of the members. These may be rendered full length or pulled back by a specified distance or a percentage of the member length for a better understanding of complicated intersection areas and a better view of the member cross-section. This option is also very useful for verifying member orientations.

If the model has been solved you may also plot the **member forces** along each member and include the magnitudes. You can also control the graphic scaling of these force diagrams. The diagrams themselves may be presented with varying degrees of **transparency** with 100% transparency meaning an outline of the force diagram and 0% indicating a solid fill. If a batch solution has been performed you may also choose which combination you want to view.

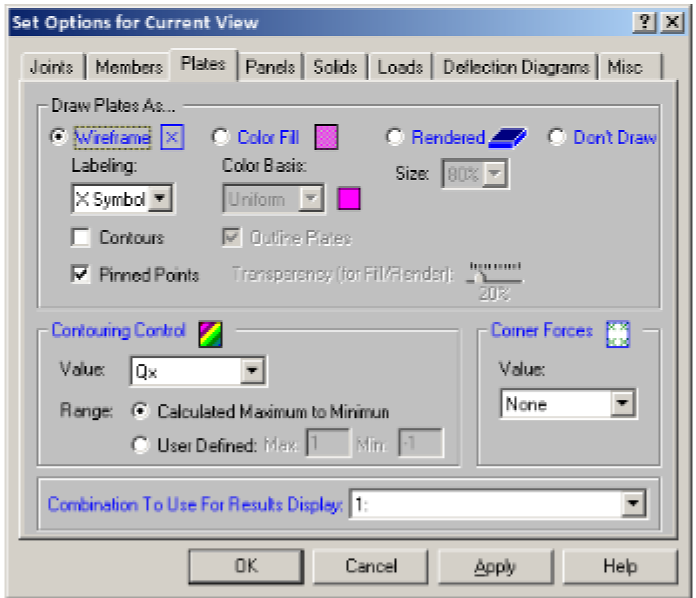
For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keyword: **Graphics**.

The detailing information checkbox pertains to information that is for export through the CIS\2 translator. See the Cardinal Points topic for more information.



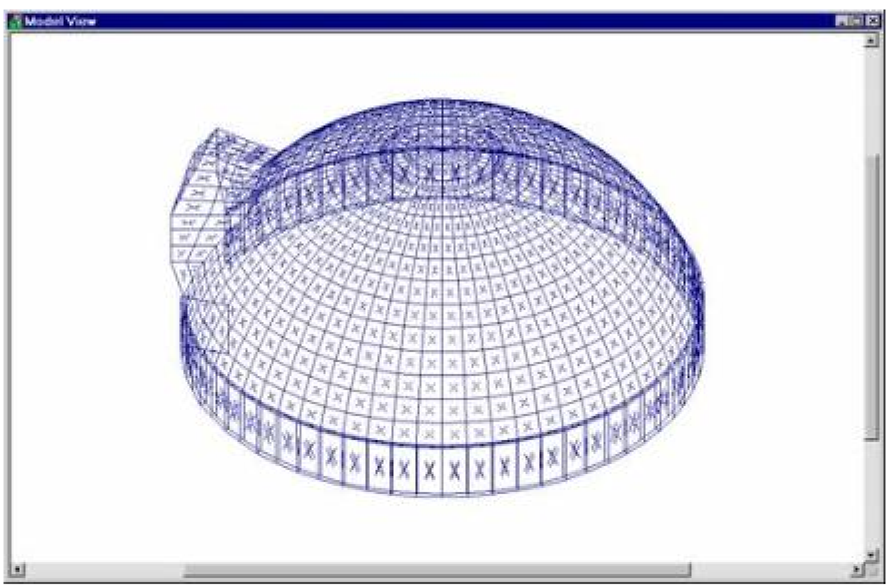
Plates

Access the plot options for plates by clicking the **Plot Options**  button on the **Window Toolbar** and selecting the **Plates** tab shown below.



The **Draw Plates As...** section of this tab allows you to specify that plates are to be drawn as wireframe, color filled, or rendered elements, or that they are not to be shown at all.

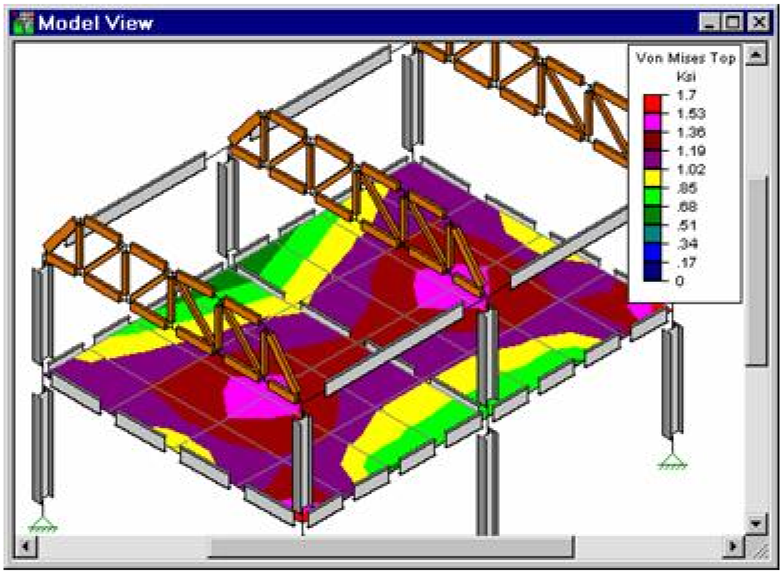
When the **Wireframe** option is selected, you may include information on the plates when plotted. The **Labeling** drop down list provides the choices you have as to what information will be displayed on each plate, such as the plate label, material, thickness, or the plate local axes. If a single or batch solution has been performed, the wireframe option also allows for the display of line contours representing the force or stress results for the plates. Each color line represents a specific value. For batch solutions, you must choose which load combination you want to view in the **Combination To Use For Results Display** drop down list.



When the **Color Filled** option is selected, plates are plotted with different colors that are mapped in a key shown in the upper right corner of the model view. You may color the plates by material set or specify a uniform color for all plates. If a single or batch solution has been performed, you also have the option to plot the stress and force results using filled color contours where each fill color represents a range of values. The color filled plates themselves may be presented with varying **Transparency** with 100% transparency meaning completely see through and 0% indicating completely solid. For batch solutions you must choose which combination you want to view.

Note

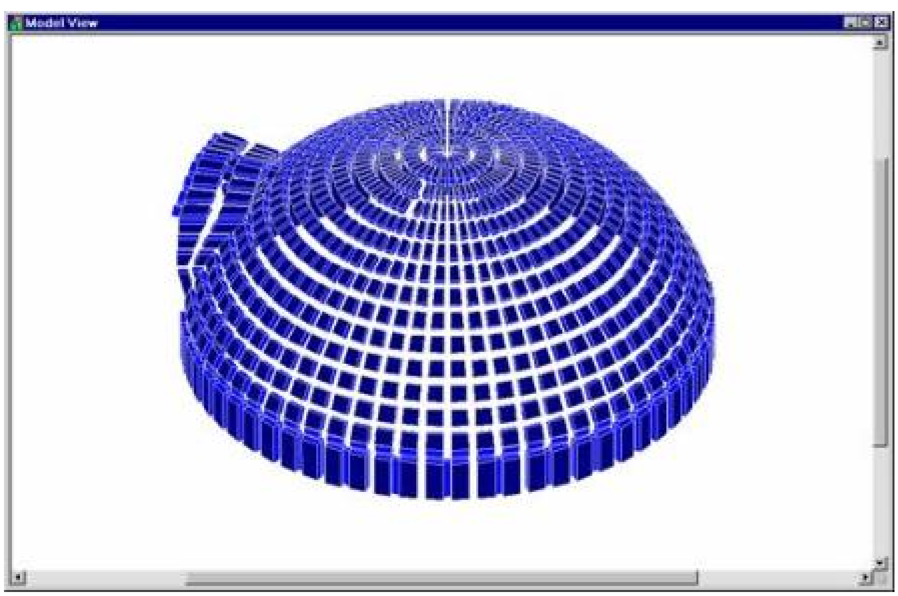
- The transparency setting does not apply when "Contour" is selected as the color basis from the drop down list.



The **Color Contours** are plotted with a global smoothing algorithm that allows the contour to vary across the plates. Because of this, the plotted contour results will differ slightly from the tabulated results in the spreadsheets. This effect will be heightened in regions of a high rate of change such as loads or boundaries. The contours may be drawn as either lines or as color filled areas.

The **Contouring Control** section determines what force or stress result is to be contoured and how the contour colors are to be assigned. The **Value** drop down list is where you select the specific result to be contoured (Qx, Fx, Mx, Von Mises, etc.). The **Range** controls determine how the contour colors are assigned. You can either contour the full range of the results by choosing **Calculated Maximum to Minimum**, or, if you are only interested in a specific range of values, you can choose **User Defined** and enter your own max and min values.

When the **Rendered** option is selected, true to scale representations of the plates will be drawn with thickness. You may also display the rendered plates at a percentage of their size. This is useful in understanding orientation and connectivity in complex views as shown below.

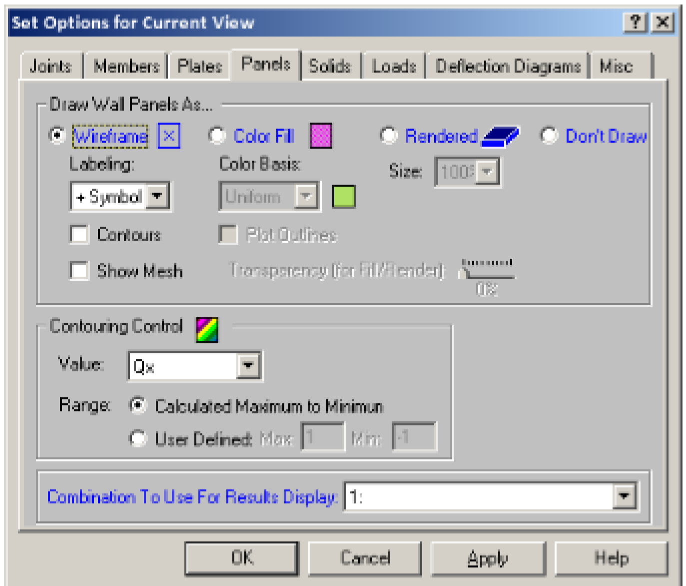


Along with any of the above options you may also plot the plate **Corner Forces**. These are the global direction forces applied to the corners of the plates that hold them in equilibrium. The **Value** drop down list is used to pick which corner force (or moment) is to be displayed. For batch solutions you must choose which combination you want to view.

The **Transparency** setting described above for Color Filled plates also applies to Rendered plates.

Panels

Access the plot options for panels by clicking the **Plot Options** button on the **Window Toolbar** and selecting the **Panels** tab shown below.



The **Panels** section of this tab allows you to specify that panels are to be drawn as wireframe, color filled, or rendered elements, or that they are not to be shown at all.

When the **Wireframe** option is selected, you may include information on the panels when plotted. The **Labeling** drop down list provides the choices you have as to what information will be displayed on each panel, such as the panel label, material, or the panel number. If a single or batch solution has been performed, the wireframe option also allows for the display of line contours representing the force or stress results for the panels. Each color line represents a specific value. For batch solutions you must choose which load combination you want to view in the **Combination To Use For Results Display** drop down list. You also have the option to see the mesh that the program automatically creates internally for all Wall Panels.

When the **Show Mesh** option is selected, panels are plotted showing the internal plate mesh automatically generated during a solution. This is useful for verifying and understanding how the forces/stresses are distributed within a panel.

When the **Color Filled** option is selected, panels are plotted with different colors that are mapped in a key shown in the upper right corner of the model view. You may color the panels by material set or specify a uniform color for all panels. If a single or batch solution has been performed then you also have the option to plot the stress and force results using filled color contours where each fill color represents a range of values. The color filled panels themselves may be presented with varying **Transparency** with 100% transparency meaning completely see-through and 0% indicating completely solid. For batch solutions you must choose which combination you want to view.

Note

- The transparency setting does not apply when "Contour" is selected as the color basis from the drop down list.

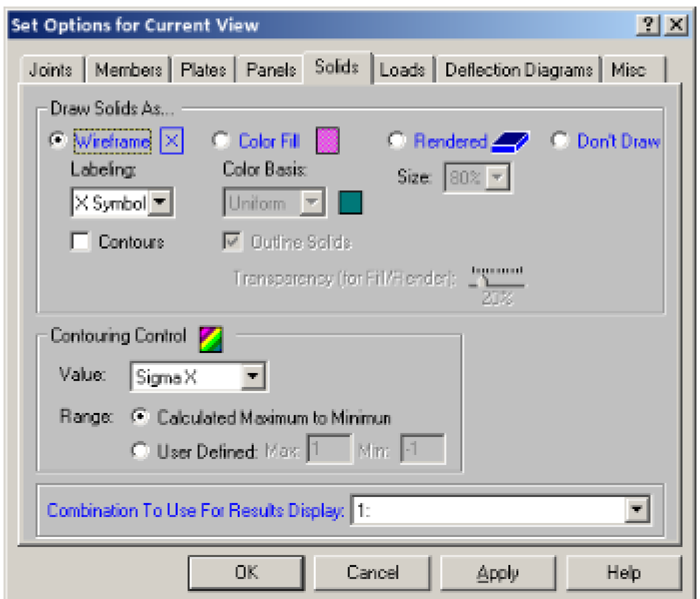
The **Color Contours** are plotted with a global smoothing algorithm that allows the contour to vary across the panels. Because of this, the plotted contour results will differ slightly from the tabulated results in the spreadsheets. This effect will be heightened in regions of a high rate of change such as loads or boundaries. The contours may be drawn as either lines or as color filled areas.

The **Contouring Control** section determines what force or stress result is to be contoured and how the contour colors are to be assigned. The **Value** drop down list is where you select the specific result to be contoured (Qx, Fx, Mx, Von Mises, etc.). The **Range** controls determine how the contour colors are assigned. You can either contour the full range of the results by choosing **Calculated Maximum to Minimum**, or, if you are only interested in a specific range of values, you can choose **User Defined** and enter your own max and min values.

When the **Rendered** option is selected, true to scale representations of the panels will be drawn with thickness. You may also display the rendered panels at a percentage of their size. This is useful in understanding orientation and connectivity.

Solids

Access the plot options for solids by clicking the **Plot Options**  button on the **Window Toolbar** and selecting the **Solids** tab shown below.



The **Draw Solids As...**section of this tab allows you to specify that solids are to be drawn as wireframe, color filled, or rendered elements, or that they are not to be shown at all.

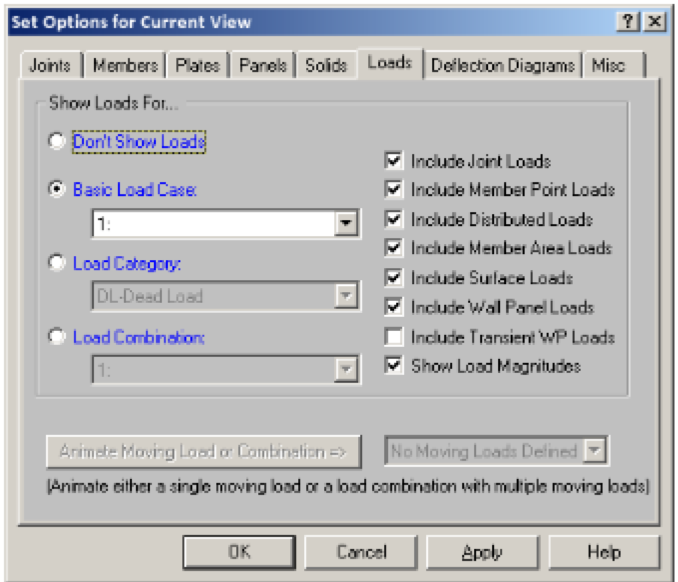
When the **Wireframe** option is selected, you may include information on the solid elements when plotted. The **Labeling** drop down list provides the choices you have as to what information will be displayed on each solid, such as the label, material, or number. If a single or batch solution has been performed, the wireframe option also allows for the display of line contours representing the force or stress results for the solids. Each color line represents a specific value. For batch solutions, you must choose which load combination you want to view in the **Combination To Use For Results Display** drop down list.

The **Color Contours** are plotted with a global smoothing algorithm that allows the contour to vary across the solid elements. Because of this, the plotted contour results will differ slightly from the tabulated results in the spreadsheets. This effect will be heightened in regions of a high rate of change such as applied loads or boundary conditions. The contours may be drawn as either lines or as color filled areas.

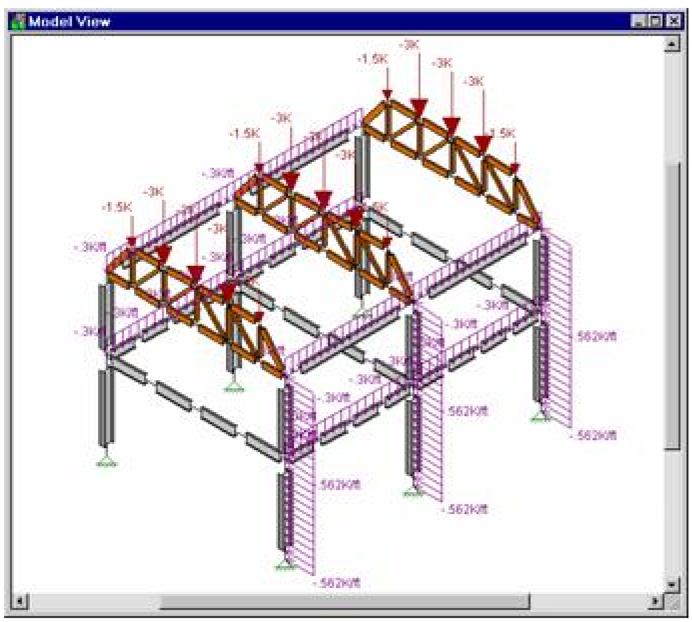
The **Contouring Control** section determines what force or stress result is to be contoured and how the contour colors are to be assigned. The **Value** drop down list is where you select the specific result to be contoured (SigmaX, Y or Z, Sigma1, 2 or 3, Von Mises, etc.). The **Range** controls determine how the contour colors are assigned. You can either contour the full range of the results by choosing **Calculated Maximum to Minimum**, or, if you are only interested in a specific range of values, you can choose **User Defined** and enter your own max and min values.

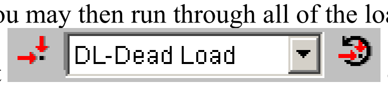
Loads

Access the plot options for loads by clicking the **Plot Options**  button on the **Window Toolbar** and selecting the **Loads** tab shown below.

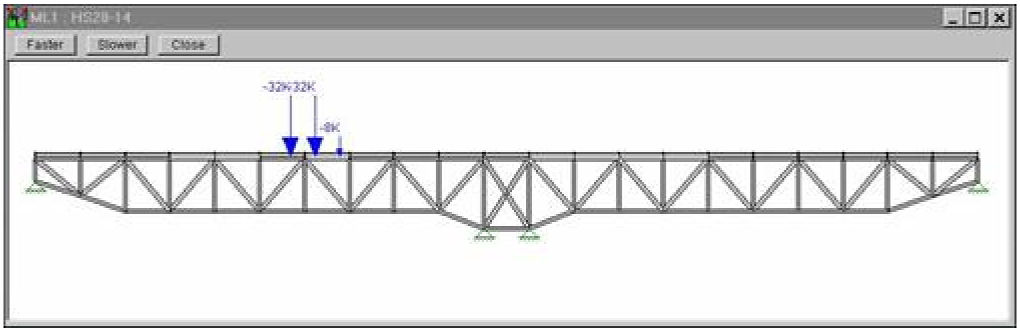


You may specify that the loads be drawn as Load Cases, Load Categories or Load Combinations by selecting the corresponding radio button. The load types to be displayed can be selected by checking the corresponding boxes.



Once you set the plot options for loads, you may then run through all of the load cases, load categories or load combinations by choosing them from the drop down list 

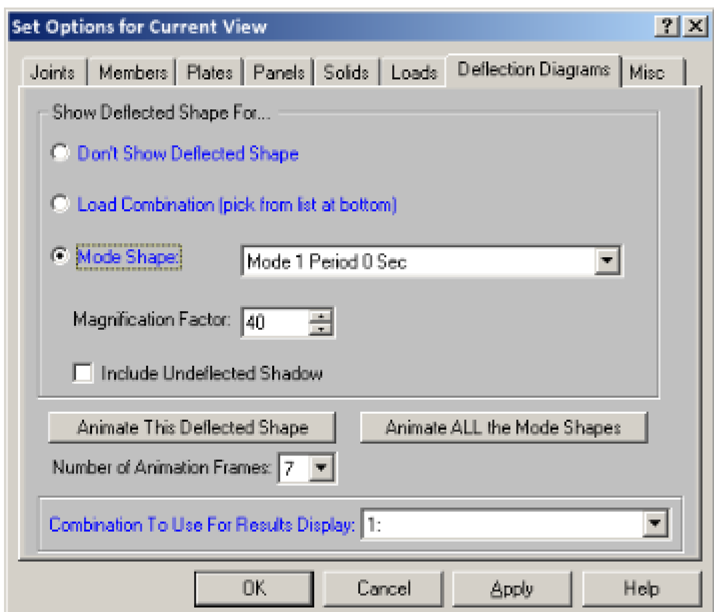
on the **Window Toolbar**.



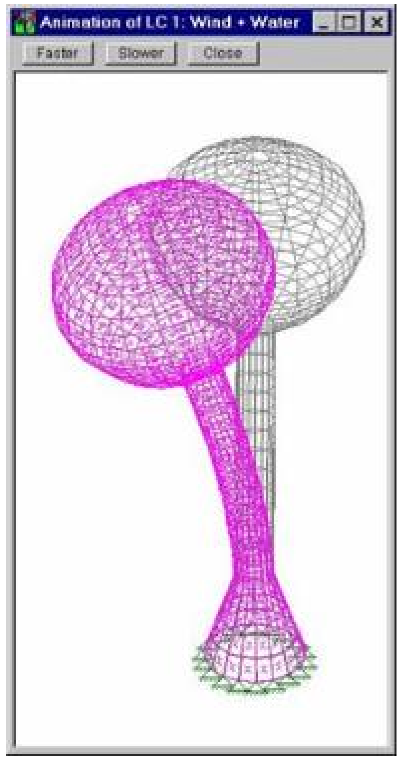
You may also animate any **Moving Loads** by selecting them from the drop-down list and then clicking on the Animate button.

Deflection Diagrams

Access the plot options for deflections by clicking the **Plot Options** button on the **Window Toolbar** and selecting the **Deflection Diagrams** tab shown below.



In the **Show Deflected Shape For...** section of the tab you may specify that a deflected shape be drawn in the current model view based on either a load combination (if a single or batch static solution has been done) or a mode shape (if a dynamic solution has been done). You can control the magnification of the deflections with the **Magnification Factor** text box and specify whether an undeflected shadow is to be shown by checking the **Include Undeflected Shadow** box. For batch solutions you must choose which combination you want to view in the **Combination To Use For Results Display** drop down list.



To animate a particular deflected shape, first select the deflected shape (as described above) and then simply click the **Animate This Deflected Shape** button. A new model view will be created with the animation. The **Number of Animation Frames** drop down list allows you to specify how many frames are used in the animation to move from the undeflected state of the model to the fully deflected state.

Note

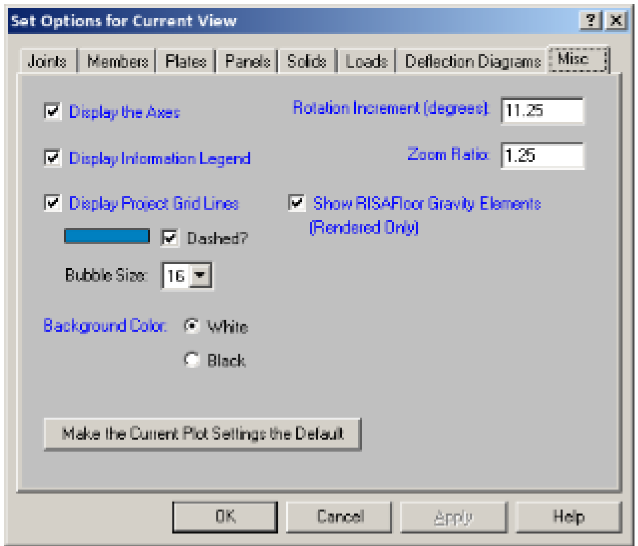
- The animation that is created is placed behind the original view. You may shrink the original view or use the **Window Menu** to bring the animation into view.

A special option allows you to **Animate ALL the Mode Shapes**. This button is only active if a dynamic solution has been performed. Clicking this button will cause a group of animation views to be created, one for each mode shape currently solved. Keep in mind that it may take a few minutes for all the animations to be created.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keyword: **Deflected**.

Miscellaneous

Access miscellaneous plotting options by clicking the **Plot Options** button on the **Window Toolbar** and selecting the **Misc** tab shown below.



You may indicate whether the global axes icon is to be displayed in the upper left corner of the model view by checking the **Display the Axes** box. You also have the option of displaying the informational legend drawn in the bottom left corner of the model view by checking the **Display Information Legend** box. This legend has information such as which combination has been solved, which loads are being displayed, etc.

Checking the **Display Project Grid Lines** box will turn on the display of the project grid in the current model view. The color and line type for the grid may also be customized. Click the colored rectangle to choose a custom color. Check the **Dashed?** box to display grid lines as dashed lines. The **Bubble Size** drop down list allows you to choose a font size for the grid labels and the bubbles that encompass them.

You may specify whether the **Background Color** for the model view is to be black or white by clicking the corresponding radial button. A black background shows colors more vividly, but a white background shows a more realistic representation of what you'll get if you print the graphic view.

By entering values in the text boxes, you may control the **Rotation Increment** and **Zoom Ratio** that will be used with each click of the corresponding toolbar buttons on the **Window Toolbar**.

Check the **Show RISAFloor Gravity Elements** box to display your entire Floor / 3D model whenever you select a rendered view. This is only available when you are working with a combined RISA-3D / RISAFloor model.

Finally, you may make any plot option settings the default startup settings. To do this, go through all the plot options tabs and set all the options to what you want as the startup default. Once that is done, press the **Make Current Plot Settings the Default** button. This button applies to ALL the plot options tabs.

Graphic Editing

You may draw, edit, and load your model directly in the model views. You can draw members and plates between existing joints or draw to grid intersections and have the joints created automatically. You may graphically select joints to be restrained, members to be loaded, and plates to be submeshed. All or selected parts of the model may be moved, copied, and/or modified allowing you to quickly model and edit your structure. See "Graphic Selection" to learn how to make selections.

Drawing and Modification Features

There are several graphic editing features that make the creation and modification of models quite easy. Use the **Insert** and **Modify Menus** or the **Drawing Toolbar** to utilize these features in the model view. To create new members, plates, etc., you can draw them using a drawing grid or draw "dot to dot" from existing joints. Once you have created these items you may use other graphic features to load the model and set boundary conditions.

- See Members for more information on drawing and modifying **Members**.
- See Plates/Shells for information on drawing and modifying **Plates/Shells**.
- See Boundary Conditions for information on creating and modifying **Boundary Conditions**.
- See Loads - Point Loads for information on drawing **Point Loads**.
- See Loads - Joint Force / Displacement for information on drawing **Joint Loads**.
- See Loads - Distributed Loads for information on drawing **Distributed Loads**.
- See Loads - Surface Loads for information on drawing **Surface Loads**.
- See Loads - Area Loads for information on drawing **Area Loads**.

All model data is automatically recorded in spreadsheets and the spreadsheets and model views are always in tune (unless you turn this feature off in the **Preferences** on the **Tools Menu**). As you edit a model graphically the spreadsheets are automatically updated and as you make changes in the spreadsheets the model views reflect these changes immediately.

All of the graphical modeling tools may be found on the **Drawing Toolbar** shown below. This toolbar may be turned on and off by clicking the  button on the **Window Toolbar**.

Where to Start

The Project Grid and Drawing Grid are often useful when you are starting a new model from scratch or adding a new section to a model. They allow you to set up grid lines which may then be used to define the members, plates, and boundaries. The differences between the two grids are simple. The Project Grid is part of the model and may be allowed to move model elements as grid lines are moved. The Project Grid is limited to the model 'plan'. The Drawing Grid is independent of the model so you may change the grid and place it anywhere, without affecting the model and whenever it is convenient. The Drawing Grid may be placed in any global plane so you may draw in 'plan' or in 'elevation'.

There are also times when it is simpler to define joints in the spreadsheet and then draw the members or plates between them. This might be the case if you are working with just a few joints, or if the structure is irregular and does not lend itself to a grid.

Apply Options

Some of the graphic editing options offer more than one way to apply a modification. This is because there are times when each option is useful. For example, changing the material of a beam from A36 steel to A572 steel can be accomplished using the **Members Spreadsheet**. If you had to apply this change to 100 beams however you would not want to do that for each of them. A better way to do this would be to graphically select all of the beams and then apply the changes all at once.

Use the following options to specify how you want to choose the items to modify. Choosing **Apply Entries to All Selected Items** allows you to use the tools on the **Selection Toolbar** to choose the items you want and then apply the modifications to all selected items at once. Choose **Apply Entries by Clicking Items Individually** to then click on the items you wish to modify individually. See "Graphic Selection" for more information on the selection tools.

Note

- The selection and viewing tools override the graphic editing modes so that as you are editing the model you can alter the selected state. You will be returned to the current editing mode when you exit a selection mode.
- You may also double click any joint, member, or plate to view and edit it's properties.
- To correct any modeling errors it is a good idea to run **Model Merge** before performing a solution. Becoming familiar with this feature will also allow you to take shortcuts while modeling. See "Model Merge" for more information.

Undo Operations

RISA-3D provides you with unlimited 'Undo' capability so that you may easily correct mistakes or just back up to try different possibilities. Simply click the  button on the **RISA Toolbar** as many times as you wish to undo your previous actions. The model view and the spreadsheets will visually display the "undoing". Remember that spreadsheet edits are undone as well.

Note

- Changes made to the selection state of the model, the zoom level or rotation of the model view, or to the Plot Options settings are NOT undone. Only a change to the model data can be undone.

Redo Operations

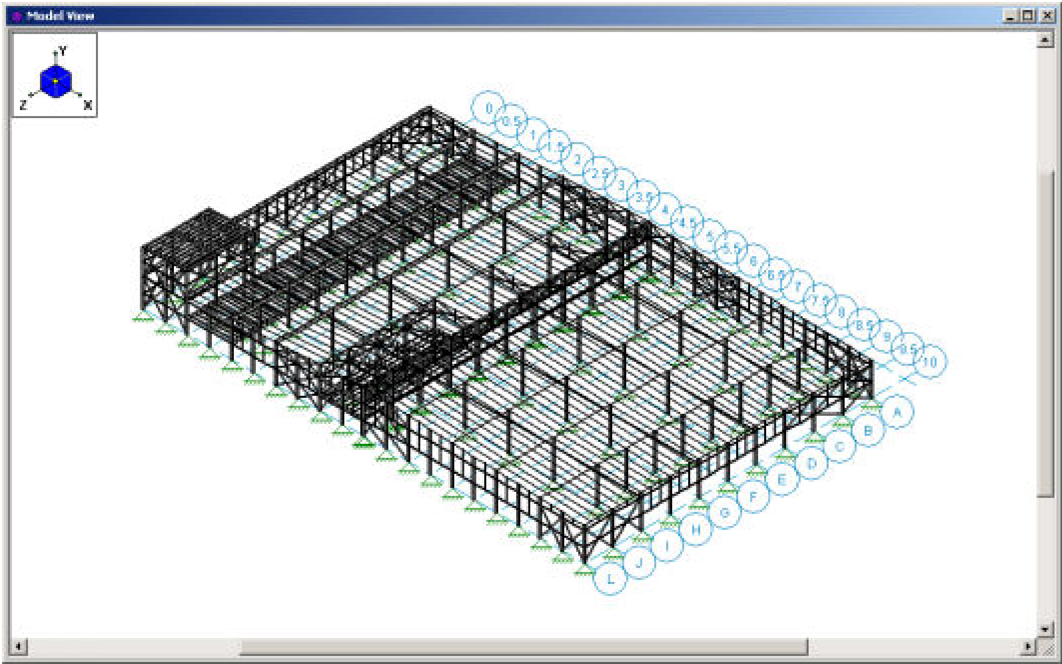
RISA-3D provides you with unlimited 'Redo' capability so that you may easily redo any actions that were previously undone using the 'Undo' button. Simply click the  button on the **RISA Toolbar** as many times as you wish to redo actions that were previously undone. The model view and the spreadsheets will visually display the "redoing". Remember that spreadsheet edits are redone as well.

Note

- Changes made to the selection state of the model, the zoom level or rotation of the model view, or to the Plot Options settings are NOT redone, since they cannot be undone to begin with. Only a change to the model data can be redone.

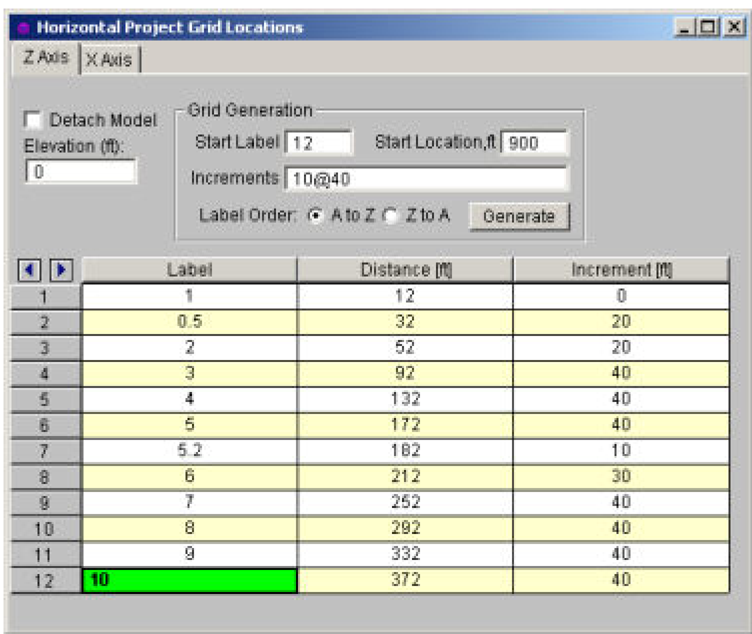
Project Grid

The **Project Grid** is a tool that lets you draw new members and plates in the model view window. You may specify the project grid lines in the 'plan view' of the model to aid in defining or editing the model and detailing printed views. This grid is saved with the model and may be used to move elements of the model simply by moving a grid line.



Project Grid Spreadsheet

Click the button on the **RISA Toolbar** to open the **Project Grid Spreadsheet** shown below. The grid lines may be specified for both plan directions, which are determined from the vertical axis specified in the **Global Parameters Dialog**. For example a vertical Y-axis leaves X and Z as the plan axes used for the project grid. You may turn the project grid on and off in the current model view by clicking the button on the **Window Toolbar**.



	Label	Distance [ft]	Increment [ft]
1	1	12	0
2	0.5	32	20
3	2	52	20
4	3	92	40
5	4	132	40
6	5	172	40
7	5.2	182	10
8	6	212	30
9	7	252	40
10	8	292	40
11	9	332	40
12	10	372	40

The project grid lines consist of a label, a distance from the origin of the grid, and the increment from the previous grid line. The distance and the increment are related such that defining one automatically updates the other. You may enter and edit the grids one at a time in the spreadsheet or you may generate grid lines using the **Grid Generation** section.

Project Grid Generation

The generation requires a start label, start location (origin), and increment information. You may use symbols such as "@", "/" and "," when entering the increments.

The "@" entry may be used to specify multiple, equally spaced, grid increments. For example, if you wanted 7 increments at 10 units each, you would type "7@10" in the increment field.

The "/" entry subdivides a larger increment into smaller equal increments. For example, the entry "12/4" would create 4 increments of 3 units each.

Use commas (",") to enter multiple increments in the increment field. For example, if you wanted to define increments of 3, 4, 7 and 2 units, you could enter "3,4,7,2" in the increment field.

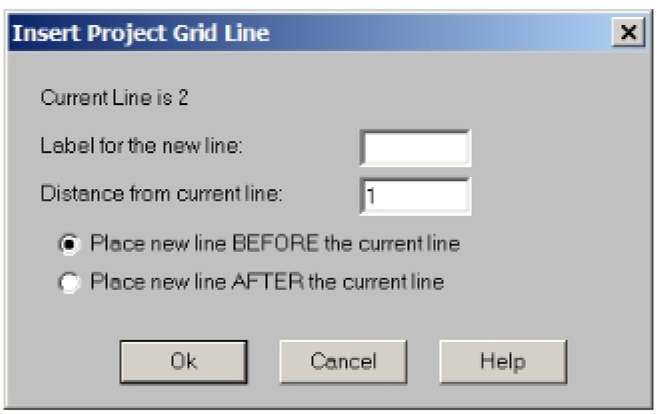
Once the project grid is specified and displayed in the model view, it will provide snap points while drawing your model. The grid exists at an elevation specified in the **Elevation** field.

Editing the Model with the Project Grid

You may also modify the grid and choose to have this affect your model as well. Changing the distance or increment value of a grid line will adjust that grid line and the lines beyond. The distances of the lines beyond are adjusted in order to maintain their increments. If you do NOT check the **Detach Model** box, then the model joints on or beyond the modified grid line are moved as well. This will subsequently move any members or plates that are attached to those joints.

Editing the Project Grid (Adding Intermediate Grid Lines)

After you have generated your project grid, you may add intermediate grid lines. In the **Project Grid Spreadsheet**, click in the row where you wish to add this intermediate grid line. Click the **Insert New Line** button on the **Window Toolbar** or right-click the mouse and choose **Insert New Line** from the **Right-Click Menu**. The **Insert Project Grid Line Dialog** box will appear as shown below.



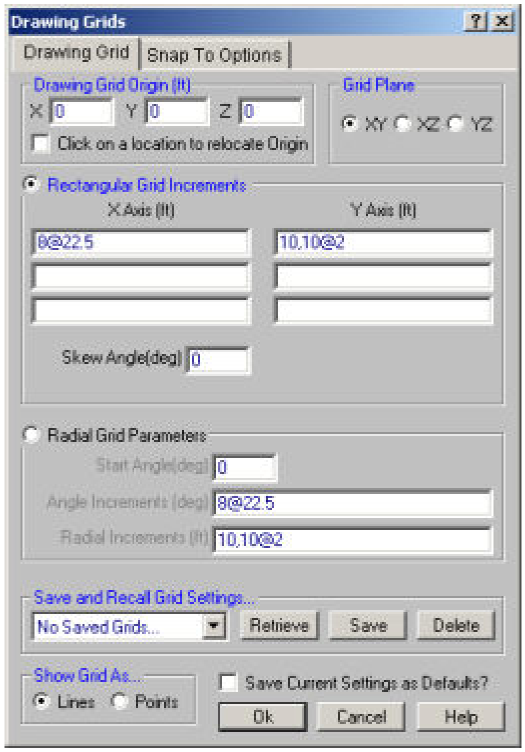
You may specify a label for the new line as well as a relative distance from the existing grid line. You can also specify the location of this new grid line by clicking the appropriate radial button next to **Place new line BEFORE the current line** or **Place new line AFTER the current line**. The Increment values for the existing grid lines will be adjusted automatically to accommodate this new grid line.

Drawing Grid

The **Drawing Grid** is a tool that lets you draw new members and plates in the model view. This grid is independent of the model, so you may change the grid as you build your model without changing any modeling that you have completed. This is because as you draw the members and plates the joints used to define them are created automatically. These joints remain in their locations if the grid is relocated.

Drawing Grid Dialog

Click the button on the **Drawing Toolbar** to open the **Drawing Grid Dialog** shown above. The display of the **Drawing Grid** in the current model view may be turned on and off by clicking the button on the **Drawing Toolbar**. If the Drawing Toolbar is not visible in the current model view, click the button on the **Window Toolbar**.



You may choose between a rectangular drawing grid or a radial drawing grid and may place the grid in any of the three global planes. Also, rectangular drawing grids may be skewed from perpendicular to the global axis plane.

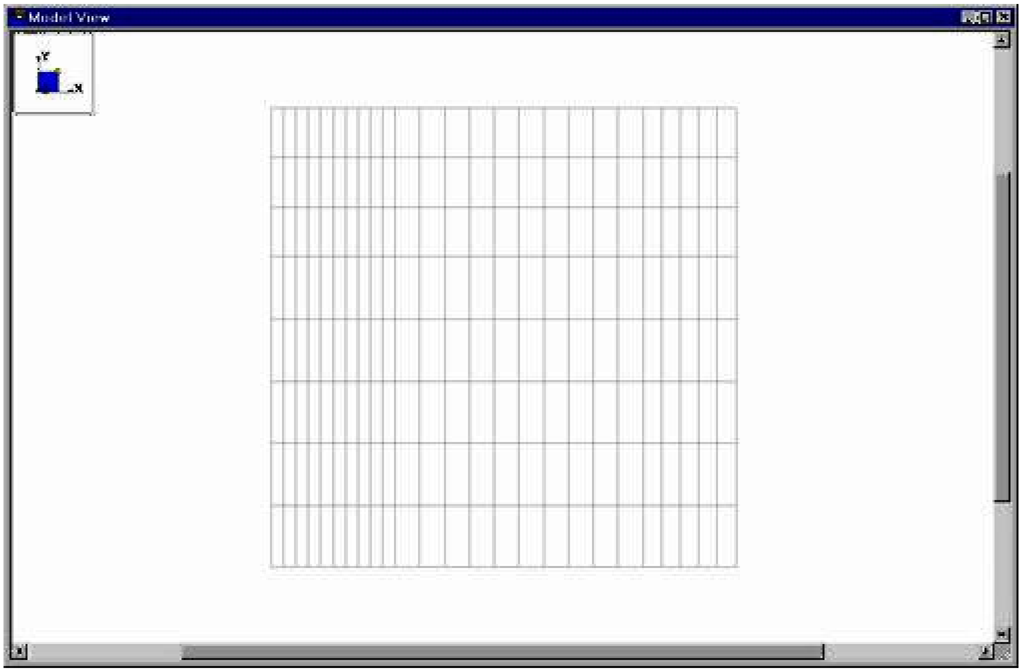
The **Save and Recall Grid Settings...** section allows for drawing grids may be saved and recalled for later use. Saved drawing grids are model independent, i.e. when you save a grid, you can reuse it in any other model you are working with in the future. To save a drawing grid, click the **Save** button after defining the grid. You will be prompted for a name for the drawing grid and it will then be saved and added to the list of grids available for recall. To recall a previously defined drawing grid, select the grid name from the drop down list and click the **Retrieve** button. If you wish to delete a previously saved drawing grid, select the grid name you want to delete from the drop down list and click the **Delete** button.

The **Show Grid As...** section gives the option of displaying the drawing grid as lines from grid point to grid point or simply as a grid of points.

You may save any of the grid information as the default setting so that when you start a new model that information is already there. To do this, simply enter the information that you want to save in the **Drawing Grids Dialog**, check the **Save Current Settings as Defaults** box, and then click the **OK** button.

Rectangular Drawing Grid

The rectangular drawing grid is defined by increments in two directions. The **Drawing Grid Origin** is where you want the grid increments to start. The **Rectangular Grid Increments** are the distances between the grid points or lines in two global directions.



You may use symbols such as "@", "/" and "," when entering the drawing grid increments.

The "@" entry may be used to specify multiple, equally spaced, grid increments. For example, if you wanted 7 increments at 10 units each, you would type "7@10" in the increment field.

The "/" entry subdivides a larger increment into smaller equal increments. For example, the entry "12/4" would create 4 increments of 3 units each.

Use commas (",") to enter multiple increments in the increment field. For example, if you wanted to define increments of 3, 4, 7 and 2 units, you could enter "3,4,7,2" in the increment field.

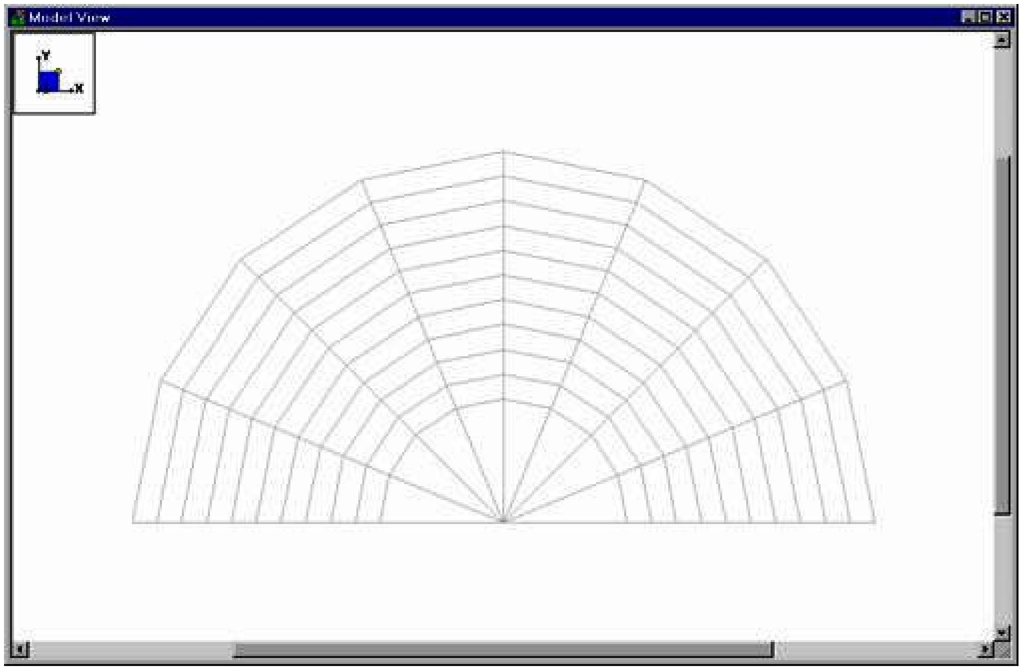
Once the drawing grid is specified and displayed in the model view, it will provide snap points while drawing your model.

Skewed Drawing Grid

The rectangular drawing grid can be skewed by specifying a skew angle. This option is available in the **Rectangular Grid Increments** section of the **Drawing Grids Dialog**. This skew angle will allow the creation of a regular rectangular drawing grid, but displayed in the model view at the specified angle, inclined from the global axis.

Radial Drawing Grid

Increments in two polar directions define a radial drawing grid. The **Drawing Grid Origin** is the point about which the grid will rotate. The default is at the global origin (0,0). The **Start Angle** defines the angle from the global axis that the first spoke will be drawn. The **Angle Increment** controls the number and angular spacing of the spokes in the grid. The **Radial Increments** controls the number and location of the rings in the grid.

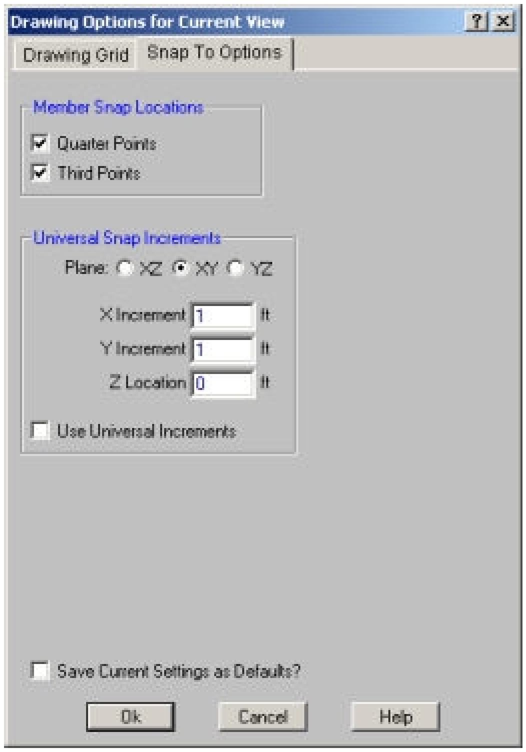


Relocating the Drawing Grid

The origin of the drawing grid may be specified in two ways. The first is to enter the exact global coordinates for the origin. This can be done by entering the values in the **X**, **Y**, and **Z** fields of the **Drawing Grid Origin** section. The second option is to specify the origin by clicking on an existing point in the model. This option is available by checking the **Click on a location to relocate Origin** box, clicking the **OK** button, and then clicking on the specific location in the model view where you wish the drawing grid origin to be located. The drawing grid origin will then be moved to this point.

Snap Points

Snap Points let you draw in the model view without the use of grids. To view or modify the snap point settings, click the button on the **Drawing Toolbar** and select the **Snap To Options** tab shown below.



In the **Member Snap Locations** section, you can set the program to automatically snap to the **Quarter Points** and/or **Third Points** of a member by checking the appropriate boxes.

The **Universal Snap Increments** section is used to define a snap grid for "free" drawing to any incremental location on a plane. To activate this feature, check the **Use Universal Increments** box. Or, the feature may be toggled on and off in the current model view by clicking the button on the **Drawing Toolbar**. If the Drawing Toolbar is not visible in the current model view, click the button on the **Window Toolbar**.

When snap points are activated, a red dot or asterisk will appear on your screen as you move your drawing cursor over one of these points. The exact coordinates of this point, and whether it is a 1/3 or 1/4 point of a member, are reported in the status bar at the very bottom of the main application window just beneath the workspace.

Copying Model Elements

To Copy Selected Items

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Select the items you want to copy.
3. To copy the selection *linearly* click the **Copy** button and select one of the following options:
   - **Copy by Point Location**
   - **Copy by Increment**
4. To *mirror* the selection click the **Mirror Copy** button and specify the mirror plane.

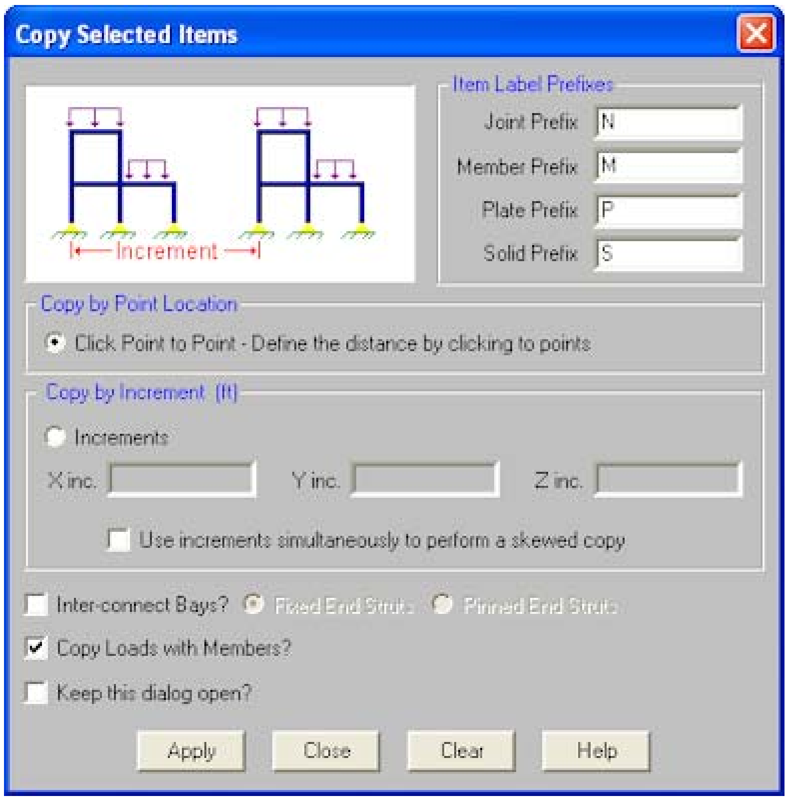
Note:

- Be sure to check your member orientations after performing the copy. RISA-3D will apply the default member orientation if you do not explicitly define it. You may use a **K joint** to help maintain orientations. See [Defining Member Orientation](#) for more information.

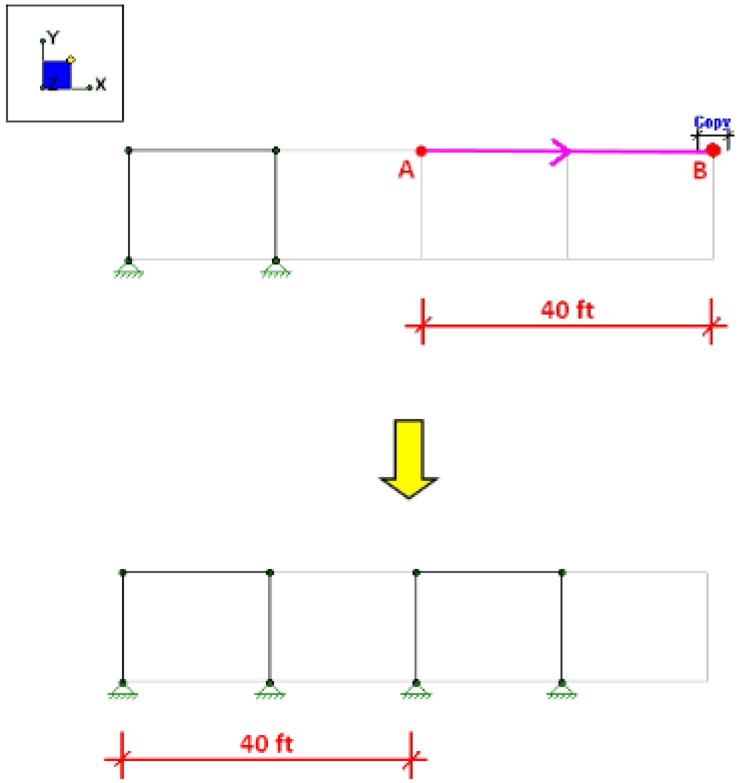
- Use the "@" symbol to specify multiple equal increments. For example specifying "3@10" will give you 3 copies at 10 units apart.
- You may undo any mistakes by clicking the **Undo** button.

Copy by Point Location

This allows you to copy the selected items by clicking on any two points. The selected items will copy the distance between the two points in the direction of the first to second click.



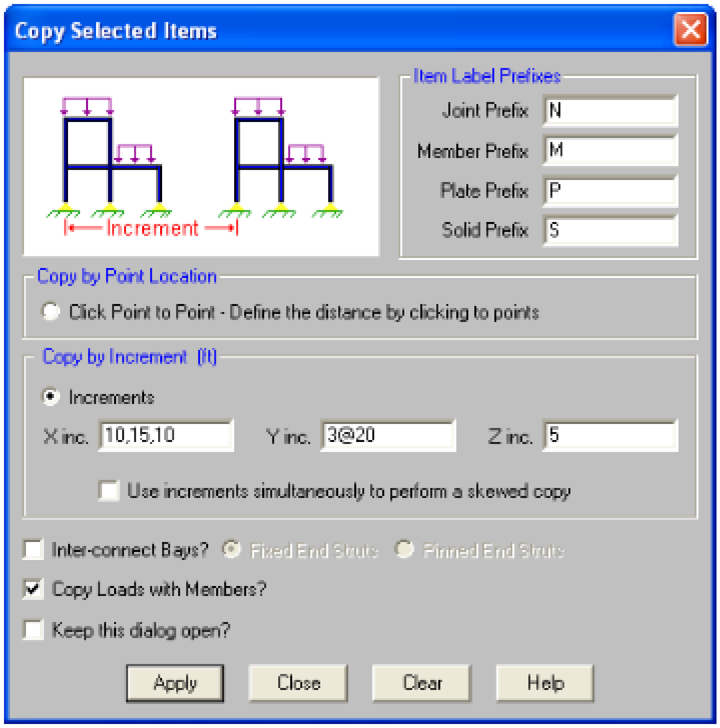
For example, if you click two points: A then B in the model below, the frame will copy 40 feet in the positive X direction.



Copy by Increments

This allows you to copy the selected items by entering in increments in any or all of the global directions. The selected items will copy the increment distance(s) that you have entered.

Use the "@" symbol to specify multiple equal increments. For example specifying "3@10" will give you 3 copies at 10 units apart.



Select the **Use increments simultaneously to perform a skewed copy** checkbox to combine the orthogonal increments into a single resultant increment vector to copy the elements in a direction other than the three global orthogonal directions.

Checking the **Copy Loads with Members?** box will cause ALL loads associated with the original selected model elements to be copied to the corresponding newly created model elements.

Checking the **Inter-connect Bays?** box will cause new members to be generated that connect all member end joints of the originally selected model elements with the corresponding member end joints in the newly created model elements. You may also indicate whether the new member end releases are to be **Fixed** or **Pinned** by selecting the corresponding radial button.

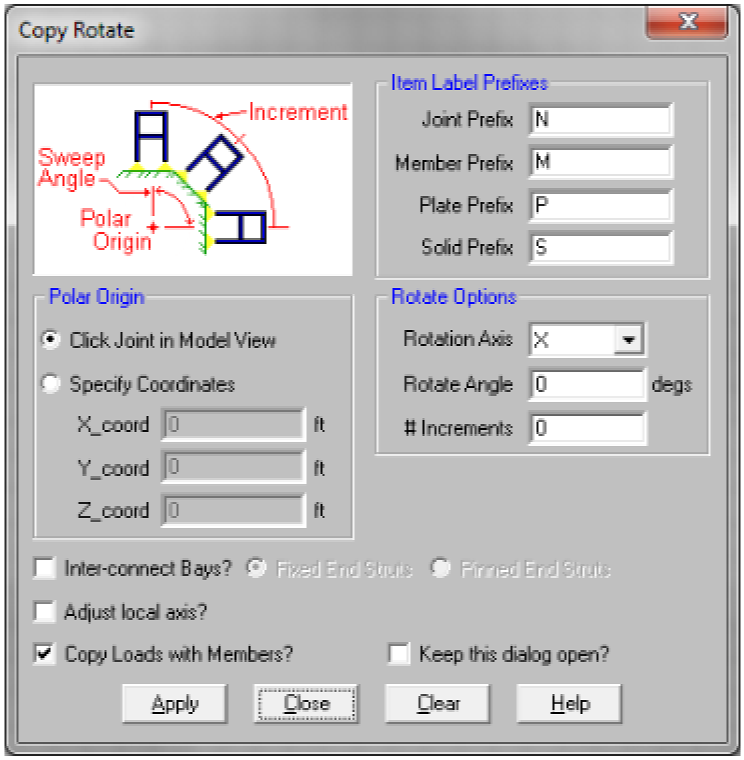
Note:

- Only member end joints will be inter-connected. The corner joints of Plates and Solids will NOT be inter-connected.
- Member end joints that have a boundary condition associated with them will NOT be inter-connected.

Rotate Copy

You can copy the selected part of the model by rotating the copies about an axis. Simply enterwhich axis is to be rotated about, the sweep angle, the number of increments along the sweep (rotation) angle, and the location of the polar origin (the point rotated about). Only two of the three polar coordinate values are used, the value corresponding to the axis of rotation is not used.

Instead of specifying coordinates for a polar origin, it may instead be chosen by clicking in the model view. Select **Click Joint in Model View**, specify the appropriate **Rotation Axis** and **Rotation Angle**, then click **Apply**. The cursor will show a cross-hair, with which a polar origin may be specified by left-clicking on a Joint or snap point.



For example, suppose you wish to copy the selected part of your model in six 30° increments (for a total of 180°) about the Y-axis. You would enter"Y" as the axis of rotation, "180" as the sweep (rotation) angle and "6" as the number of increments. For the polar origin, enter the X and Z coordinates of where you want the Y-axis of rotation to pass through the XZ plane.

Checking the **Inter-connect Bays?** box will cause new members to be generated that connect all member end joints of the originally selected model elements with the corresponding member end joints in the newly created model elements. You may also indicate whether the new member end releases are to be **Fixed** or **Pinned** by selecting the corresponding radial button.

Note

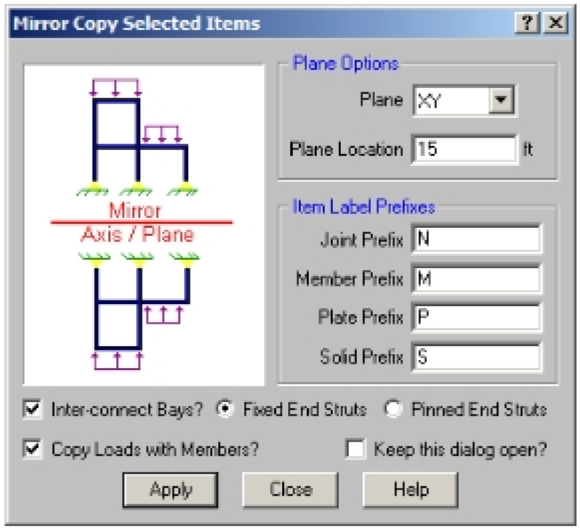
- Only member end joints will be inter-connected. The corner joints of Plates and Solids will NOT be inter-connected.
- Member end joints that have a boundary condition associated with them will NOT be inter-connected.

Checking the **Adjust Local Axis?** box will cause the local axis of each member to be rotated with the newly created model elements. Otherwise, the member local axes will retain their original orientation with respect to the global axis. The local axes of Plates and Solids will ALWAYS be rotated with the newly created model elements.

Checking the **Copy Loads with Members?** box will cause ALL loads associated with the original selected model elements to be copied to the corresponding newly created model elements.

Mirror Copy

You can mirror selected parts of your model. Enter the global planethat you want to mirror about and enter a location along the normal axis to move the mirror plane location. If the mirror plane location is left blank, the mirror plane is placed at the origin.



For example, suppose you want to mirror part of your model about the XY plane at a location +3.0 feet from the origin along the Z-axis. You would enter "XY" as the Mirror Plane and enter a "3.0" for the mirror plane location. (If the mirror plane is XY, the normal axis is the Z-axis)

Checking the **Inter-connect Bays?** box will cause new members to be generated that connect all member end joints of the originally selected model elements with the corresponding member end joints in the newly created model elements. You may also indicate whether the new member end releases are to be **Fixed** or **Pinned** by selecting the corresponding radial button.

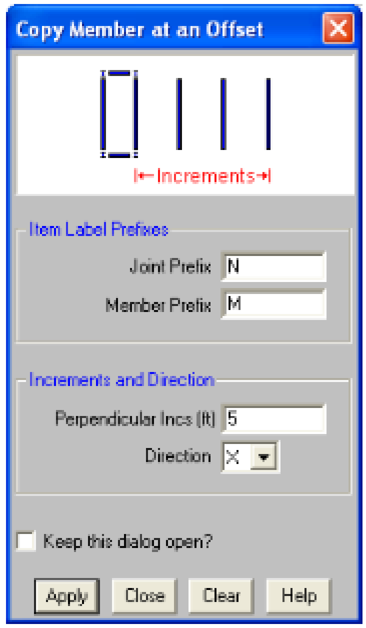
Note

- Only member end joints will be inter-connected. The corner joints of Plates and Solids will NOT be inter-connected.
- Member end joints that have a boundary condition associated with them will NOT be inter-connected.

Checking the **Copy Loads with Members?** box will cause ALL loads associated with the original selected model elements to be copied to the corresponding newly created model elements.

Copy Offset

This tool allows you to copy selected members at an offset simply by entering the offset distance and then clicking on t the member to be offset.



Moving and Rotating Model Elements

To Move or Rotate Selected Items

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Select the items you want to move.
3. To move the selection linearly click the **Move Linear** button and specify the offset distances.

   To move the selection in a polar fashion click the **Rotate** button and specify the axis and angle.

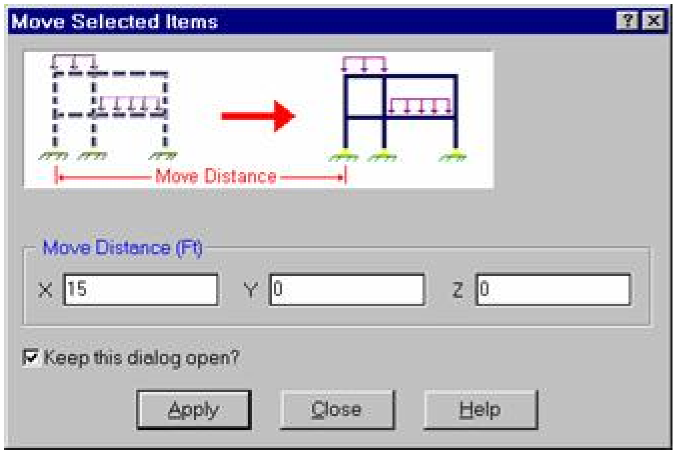
   For help on an item, click and then click the item.

Note

- Be sure to check your member orientations after performing the copy. RISA-3D will apply the default member orientation if you do not explicitly define it. You may use a **K joint** to help maintain orientations. See Defining Member Orientation for more information.
- You may undo any mistakes by clicking the Undo button.

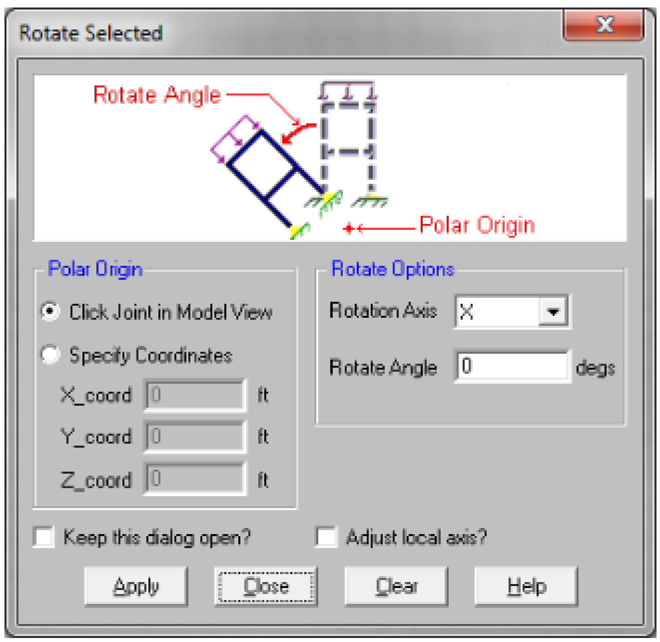
Linear Move

You can move the selected part of the model. Just enter the desired translation distances in the global axes directions.



Rotate Move

To rotate the selected parts of the model about an axis enter which axis is to be rotated about, the desired rotation angle, and the coordinates of the polar origin (the point rotated about). Only two of the three polar coordinate values are used, the value corresponding to the axis of rotation is ignored. Instead of specifying coordinates for a polar origin, it may instead be chosen by clicking in the model view. Select **Click Joint in Model View**, specify the appropriate Rotation Axis and Rotation Angle, then click **Apply**. The cursor will show a crosshair, with which a polar origin may be specified by left-clicking on a Joint or snap point.

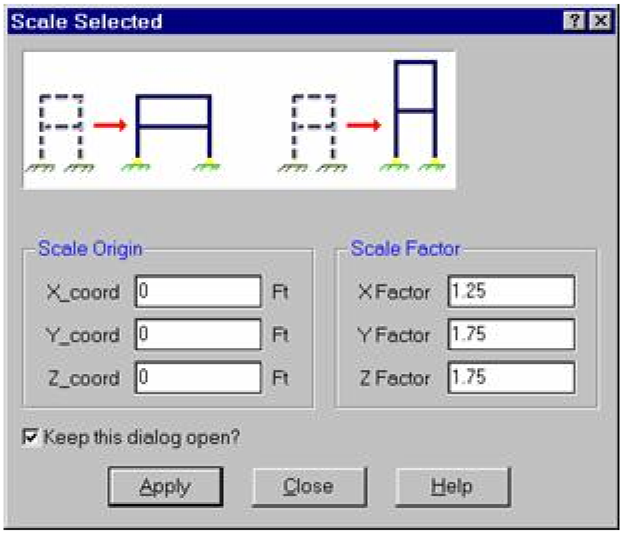


For example, say you wish to rotate the selected part of your model 90° about the Z-axis. You would enter "Z" as the axis of rotation and "90" as the rotate angle. For the polar origin, enter the X and Y coordinates of where you want the Z-axis of rotation to pass through the XY plane.

Checking the **Adjust Local Axis?** box will cause the local axis of each member to be rotated along with the model elements. Otherwise, the member local axes will retain their original orientation with respect to the global axis. The local axes of Plates and Solids will ALWAYS be rotated along with the model elements.

Scaling Elements

The scale feature allows you to change the size of selected items. Selected joints, members and plates are all affected when the scaling is applied. Loads are not scaled except distributed and surface loads that are a function of the size of the element.



Enter an origin about which to scale and a scale factor to apply to each global direction. The origin is the point that remains stationary during the scaling. The factors are applied in each direction to adjust the joint coordinates, member lengths and plate sizing. The factors work so that a factor of one has no affect, a factor of two will double the size of the item, and so on.

What is actually happening when you scale items is that the joint coordinates that define these items are being moved. For this reason the scaling is applied to all selected items plus unselected joints of selected items. For example if you have selected members to scale but have unselected some of the end joints, the joints are scaled anyway since this is the only way to scale the member.

Note

- If you wish to scale in a direction other than a global direction you can rotate the model to a global direction, scale it, and rotate it back.

To Scale Selected Items

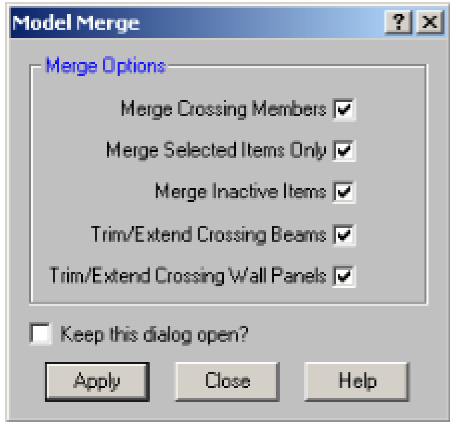
1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Select the items you want to copy.
3. To scale the selection click the Scale and specify the scale factors for each direction. For help on an item, click and then click the item.

Note

- You may undo any mistakes by clicking the Undo button.

Merging Model Elements

As you build your model, you may find that you will need to perform a model merge from time to time. In fact, if you count on doing this, you can generally build your models faster and let the model merge feature do a lot of the work for you. See Merge Tolerance to learn about inputting the merge tolerance.



To Perform a Model Merge

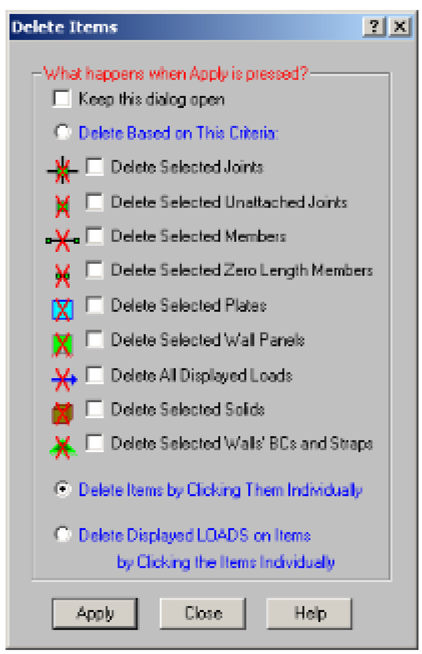
1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Select the items you wish to merge. Typically you will want everything to be selected.
3. Click the **Merge** button and set the parameters for the new merge. Click the Help button for more information.

Deleting Elements

You can delete parts of the model based on the current selection state, or you can click on the items you wish to delete individually. If you accidentally delete something you didn't want deleted, you can use the Undo feature to repair the damage.

If you wish to delete based on the current selection state, you must use the check boxes to define the criteria the program will use to perform the deletion. Only items that are selected and that have their check boxes "checked" will be deleted. The choices let you delete joints, members, plates, wall panels, solids and/or loads.

Keep in mind that if you delete joints, any members, plates, etc. attached to the deleted joints MUST also be deleted, regardless of whether those elements are selected or not.



Two noteworthy features are the ability to delete unattached joints and zero length members. Sometimes in the process of modeling you accidentally create unwanted unattached joints or zero length members. These two parameters give you a convenient way to remove these unwanted items.

If you request deletion of displayed loads, you'll get exactly that. Any load currently displayed will be deleted. By controlling what loads are displayed via the **Loads** tab in the **Plot Options Dialog**, you can easily delete specific types of loads for particular basic load cases, load categories or load combinations.

Alternatively, you can choose to delete items by clicking on them individually. Select the radial button next to **Delete Items by Clicking Them Individually** and use the mouse to click on any individual item you wish deleted.

A final choice is to delete loads individually by clicking on the item the load is applied to. Select **Delete Displayed LOADS on Items** and then click the items (joints, members or plates) whose displayed loads are to be deleted. For example, if you wish to delete certain distributed loads, first use the **Loads** tab in the **Plot Options** to display the loads to be deleted, then choose the **Delete Displayed LOADS on Items** option and proceed to click on the items to which the loads to be deleted are applied.

Remember to click the **Apply** button to make your choices active.

Note

- Master joints for rigid diaphragms are not treated as unattached joints.

To Delete Selected Items

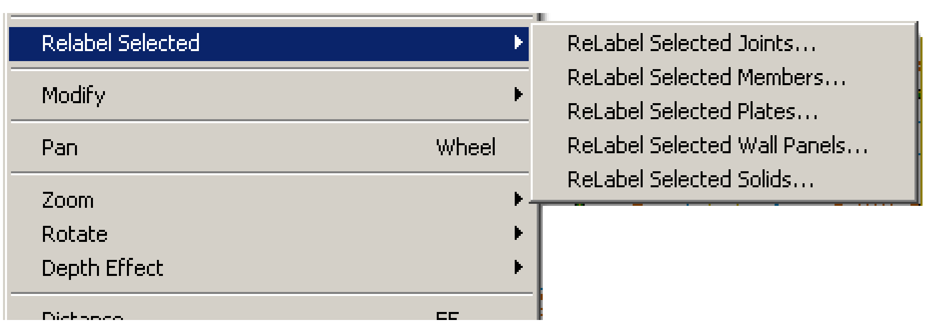
1. If there is not a model view already open then click the button on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Select the items you want to delete.
3. To delete the items click the **Delete** and specify the types of items. For help on an item, click and then click the item.

Note

- You may undo any mistakes by clicking the Undo button.

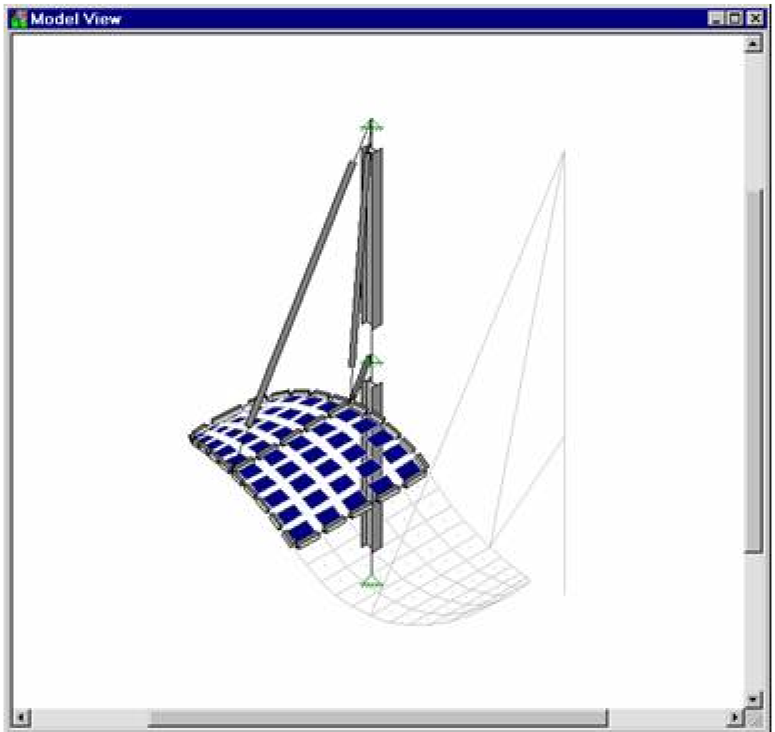
Re-Labeling Selected Elements

You can use the right-click menu in the program to re-label only the selected items. This is useful when you want all members at a certain elevation to have a prefix which denotes the floor on which it is located.



Graphic Selection

You may graphically select items in order to view or modify part of the model or to print part of the results. When used in conjunction with the graphic editing features it allows you to quickly model and make modifications to the model. If used with results it allows you to view and print only the input or results that you want.



The elements that you see in the model views have two possible states; selected and unselected. By default, all items are selected, and therefore fully displayed. If you unselect any items, they are drawn in a gray or “ghosted” shade. To select or unselect an item simply click on it with the left mouse button. You may also use one of the selection tools below to select multiple items.

Note

- As an alternative to using the selection tools, some operations offer a **Click to Apply** option, which allows you to modify items by clicking or boxing them with the mouse. This is useful when working with a few items.
- Inactive items can only be selected with **Criteria Selection**.
- Unselected items remain as part of the model for the solution. To remove items from the analysis you need to make them inactive.
- If you choose not to display items by turning them off in the **Plot Options Dialog**, they remain in the current selected state and can still be selected/unselected.

Selection Modes

The **Selection Toolbar** is the vertical toolbar located on the left side of the screen. This toolbar is for selecting joints, members, and plates in model views. This toolbar will not be available when a model view is not active. You may also access the selection tools from the **View Menu**.

Some of the tools are for one-time applications such as **Select All**. Other tools, such as **Box Select**, place you in a selection mode that remains active until you cancel it. The current mode is indicated by the mouse pointer and by the state of the button. While in a selection mode the button will stay depressed until you click it again, choose another button, or press the ESC key. You may have more than one model view open and may be in different modes in each view.

Note

- There are other types of graphic modes such as **editing** and **viewing** (zooming) modes. The viewing mode overrides the selection mode, which overrides the editing mode. This is so that while you are editing the model you can alter the selected state. You will be returned to the current editing mode when you terminate a selection mode.
- To cancel a selection mode re-click the same button so that it is in the "up" position, or press the ESC key, or click the right mouse button.
- You can cancel a box, line, or polygon selection that is underway by dragging the mouse off the model view while still holding the mouse button down.

Select All and Unselect All

Clicking **Select All** and **Unselect All** tools will select or unselect all of the active joints, members, and plates in the model.

Box Select and Unselect Modes

The **Box Select** and **Box Unselect** tools allow you to draw a box around the items that you wish to select or unselect. Members and plates must be entirely within the box in order for them to be affected.

Polygon Select and Unselect

The **Polygon Select** and **Polygon Unselect** tools allow you to draw a polygon around the items that you wish to select or unselect. Members and platesslabs must be entirely within the polygon in order for them to be affected.

Line Select and Unselect

The **Line Select** and **Line Unselect** tools allow you to draw a line through the items that you wish to select or unselect. Any element the line crosses will be affected. This is useful when choosing items between other items such as columns between floors.

Inverting Selections

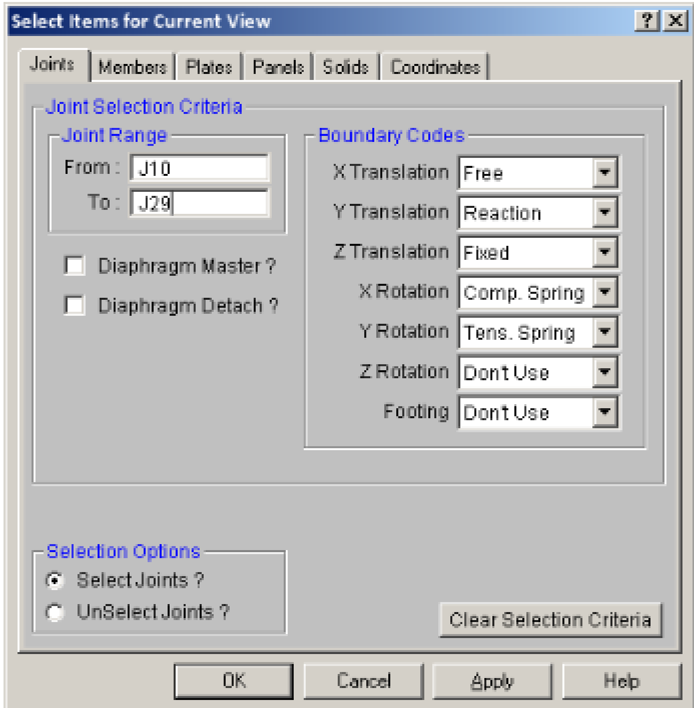
The **Invert Selection** button is used to invert the selected state of the model. When clicked, all selected items are made unselected and all previously unselected items are made selected. For example, this can be very useful when just a few items are to be selected in a large model. Simply click on the desired items to unselect them, then click the **Invert Selection** button to make them selected and everything else unselected.

Criteria Selections

The **Criteria Selection** button allows you to select items based on a wide range of criteria such as coordinates, labels and many other conditions. When the **Criteria Select** button is clicked, you will be presented with the Criteria Selection Dialog with options grouped by tabs across the top. Each tab represents groups of criteria that you may use to refine your selection.

The options are numerous making it easy to quickly achieve complicated selections. This is a powerful tool so it is worth taking the time to experiment with the options so that you will know how to use it to your advantage. The various tabs in the dialog are described below.

Joints



You may **Select** or **Unselect** joints as follows.

Joint Range – You may specify a range of joint labels. Specifying only one label selects just that one joint.

Diaphragm Master – Checking this box selects the joints that currently define diaphragms.

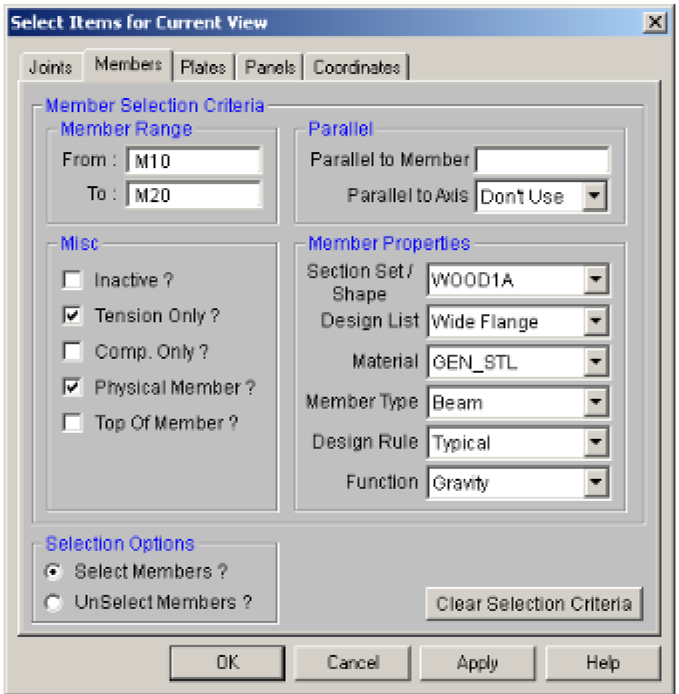
Diaphragm Detach – Checking this box selects the joints that currently have their “Detach From Diaphragms” flag set. This would let you see which joints were not being included on any defined diaphragms.

Boundary Codes – You may specify boundary condition criteria in all six degrees of freedom.

Note

- If no criteria are specified, all the joints will be selected/unselected.

Members



You may **Select** or **Unselect** members based on the following criteria.

Member Range – You may specify a range of member labels. If only one label is entered, only that one member will be selected.

Parallel – You may select members that are parallel to a certain member, whose label you enter, or parallel to a global axis.

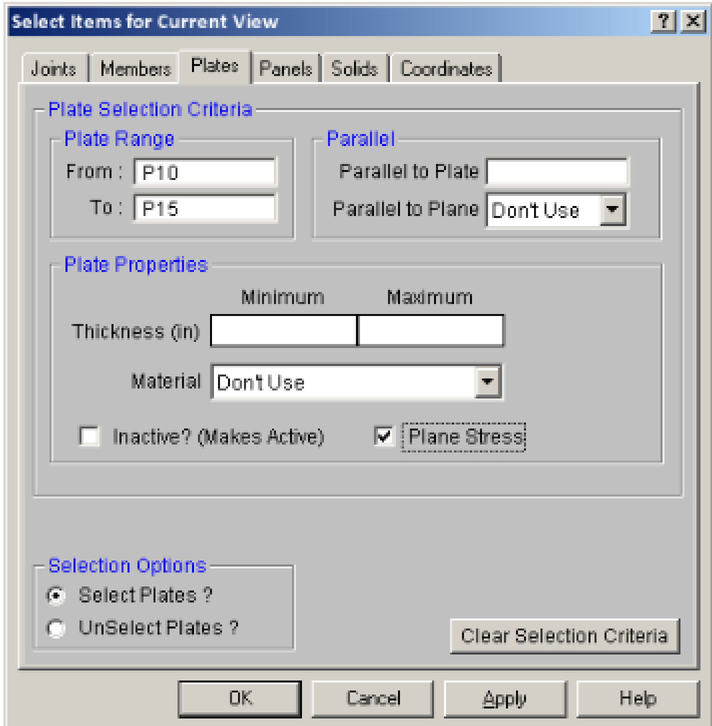
Member Section Properties – You may specify a Section Set, Database Shape and Material to be applied when selecting members.

Misc – You may select members that are currently inactive. Using this criteria will make these members active. To make them inactive once again you can then use **Modify Members**. You may select members that are tension only and compression only members as well as Physical members. Members with the Top of Member flag set can also be quickly found.

Note

- If no criteria are specified, all the members will be selected/unselected.

Plates



You may **Select** or **Unselect** plates based on the following criteria.

Plate Range – You may specify a range of plate labels. If only one label is entered, only that one plate will be selected.

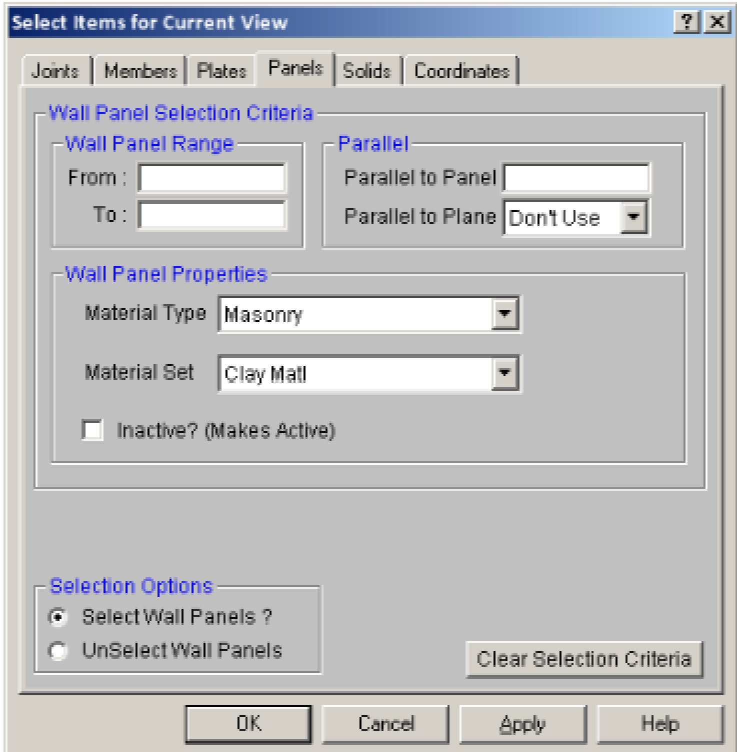
Parallel – You may select plates that are parallel to certain plate, whose label you enter, or parallel to a global plane.

Plate Properties – You may specify a minimum and maximum Thickness and Material to be applied when selecting plates. You may also select only the plane stress plates. You also may select plates that are currently inactive. Using this criterion will make these plates active. To make them inactive once again you can then use **Modify Plates**.

Note

- If no criteria are specified, all the plates will be selected/unselected.

Panels



You may **Select** or **Unselect** panels based on the following criteria.

Wall Panel Range – You may specify a range of panel labels. If only one label is entered, only that one panel will be selected.

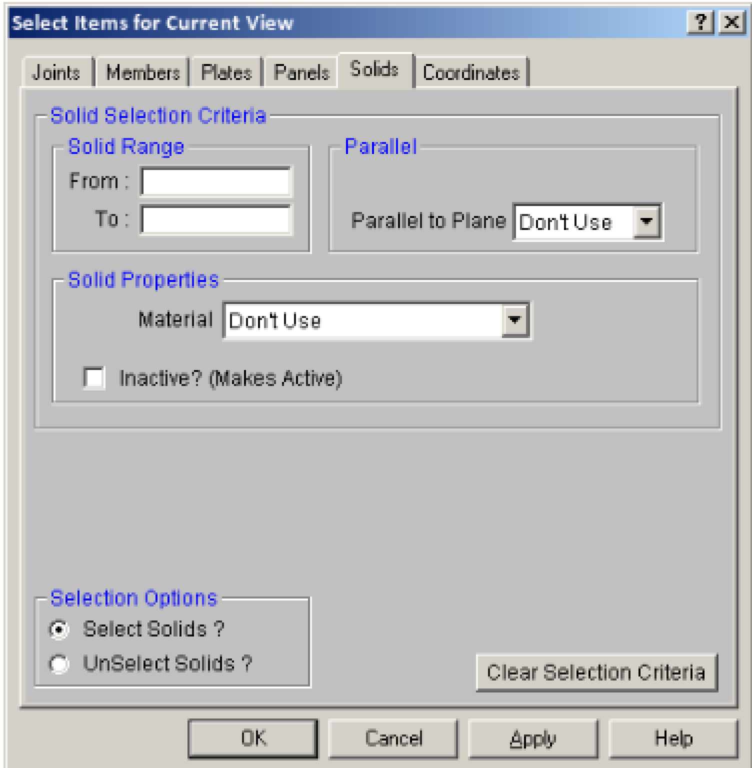
Parallel – You may select panels that are parallel to certain panel, whose label you enter, or parallel to a global plane.

Wall Panel Properties – You may specify a Material Type (Wood, Masonry, etc) and/or Material Set (Larch, Clay, etc). You also may select panels that are currently inactive. Using this criterion will make these panels active. To make them inactive once again you can then use **ModifyPanels**.

Note

- If no criteria are specified, all the panels will be selected/unselected.

Solids



You may **Select** or **Unselect** solids based on the following criteria.

Solid Range – You may specify a range of Solid labels. If only one label is entered, only that one Solid will be selected.

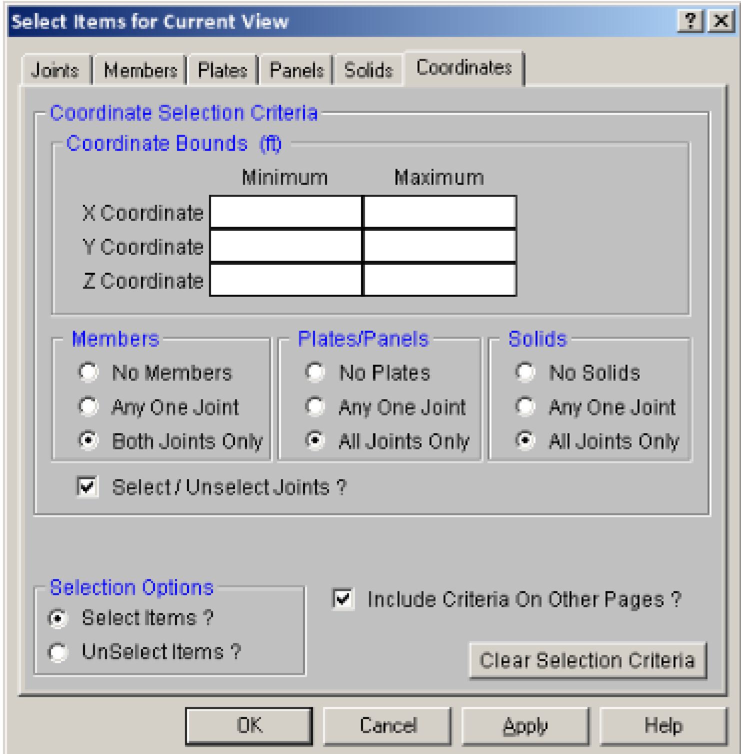
Parallel – You may select Solid Elements that have a plane which is parallel to one of the global planes. If any of the 6 surfaces of the plate are parallel to the plane, then the plate will be selected.

Material Properties – You may select or unselect solid elements base on their material. You also may select solids that are currently inactive. Using this criterion will make these solids active. To make them inactive once again you can then use **ModifySolids**.

Note

- If no criteria are specified, all the solids will be selected/unselected.
- The ONLY way to make an inactive solid active again is to use this criteria select dialog.

Coordinates



You may select or unselect elements based on coordinate criteria and combine this with the criteria on the other tabs.

Coordinate Bounds – You may specify minimum and maximum bounds in the global directions. Items within these bounds AND meeting the criteria in the other groups will be selected/unselected.

Member Criteria – You may specify that one or both member end joints be within the bounds. You may also specify that no members be selected within the range.

Plate/Panel Criteria – You may specify that one or all plate corner joints be within the bounds. You may also specify that no plates be selected within the range.

Select/Unselect Joints – You may specify that joints in the coordinate range be selected or not.

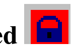
Include Criteria On Other Pages – This option allows you to include selection criteria from the other pages (Joints, Members, Plates) and combine it with the coordinate range specified on this page.

Note

- If the "Include Criteria On Other Pages" option is selected, all criteria will be applied together so that the affected items will meet all of the criteria. For example if you specify a range of joint labels and coordinates, only the joints within the coordinate bounds and within the label range will be selected/unselected.

Locking Selections

Click the **Lock Unselected**  button to cause all currently unselected items to stay unselected and be visually removed from the current model view. This is useful when you are editing or printing a portion of a model and need to clear the model view of all items not involved. For example in a multistory building when working on one floor you can lock the other floors. Just unselect the entire model, then use box select to select the floor you would like to work on, then press the Lock Unselected button and the rest of the model will stay unselected until you press the button again to turn it off. This tool is also helpful when trying to select an item that is behind other items.

To Unlock the unselected members in the model view, click the **Unlock Unselected**  button and the unselected items that were previously "removed" from the model view will be returned to the view in the unselected state.

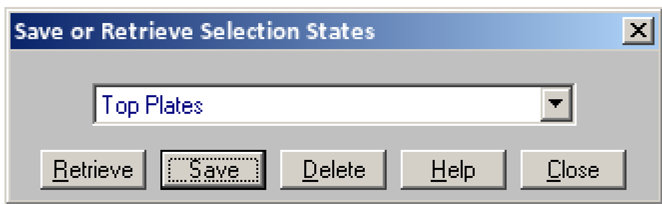
Graphic Selection from Spreadsheets

In many spreadsheets, both Data Entry and Results, you can select rows on the spreadsheet and use the right-click menu to select the S**elect Marked Lines in Current View** or **Unselect Marked Lines in the Current View** options. This will graphically select or unselect the items corresponding to the selected rows in the top most model view window. This is very useful for highlighting failing members in a spreadsheet and then having them graphically selected in a model view to see where the problems are.

Saving Selections

You can save and recall various selection states for a model. If the model is altered after a selection state has been saved, the saved selection state will also be altered. Any new items (joints, members…) will be set to "selected" in any selection states saved prior to the creation of the new item.

To save a selection state, click the **Save Selection State**  button on the **Selection Toolbar**.



Click the **Save** button and provide a name for the saved selection. You can have up to 16 different saved selections in a model. To retrieve a saved selection, choose the selection state from the drop down list and click the **Retrieve** button. To delete a saved selection, choose the selection state from the drop down list and click the **Delete** button.

Help Options

RISA Technologies has, and will, put a great deal of effort into assisting you in getting your work done as quickly as possible. This includes providing many ways that you can get help in understanding the software.

Electronic Help File

The **Help File** is designed to help you get your work done as quickly as possible and is intended to provide:

- Procedures that lead users through the steps of completing tasks
- Context-sensitive Help topics that provide users with quick descriptions of items on their screens
- Troubleshooting topics that guide users through solutions to common problems
- Extensive discussions for a thorough understanding of engineering and modeling topics
- Easy access to related topics

The electronic help file can be accessed by clicking the **Help File** button on the **RISA Toolbar**. A new window containing a Table of Contents will be opened. Click on any item in the Table of Contents for extensive information on the topic.

Context Sensitive Help

Context Sensitive Help is help that a user can access in context while working in a program. It provides you with the information you need where and when you want it. This type of help is provided in two different ways:

The first method provides specific information on every item that you see on the screen. For help on an item such as an entry field in a dialog, click and then click the item. This feature is referred to as "**What's This?**" help. This feature may not function properly on Vista computers.

You may also get more detailed help when working in some windows by clicking on the **Help** button at the bottom of the window or dialog. This will launch a Help File window displaying the topic that is related to the window in which you are working. The topic will be explained and links to related topics may also be provided.

RISA Technologies Online

Our website, **www.risatech.com**, provides various support information and documents.

Visit RISA Technologies on the web for:

- Frequently Asked Questions
- Download program Manuals (General Reference or Tutorial)
- Check our website for the latest updates - When a bug is discovered it is posted on the web site along with possible work-around procedures and/or service releases to update your software.
- Software Verification Problems

Tool-tips

Are you uncertain what a toolbar button is for? Simply hold your mouse pointer over that button without clicking. **Tool-tips** are displayed that will explain what the button will do should you decide to press it.

Tutorial

The comprehensive **Tutorial** (part of the User's Guide - a separate document) guides you through using most features. It is a real-world example of building and solving a model, making changes, and optimizing. This is the best way to quickly get up and running. The User's 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 quickly move through the tutorial. If you want more thorough explanations of the modeling process you may read all or some of the supporting text as you see fit.

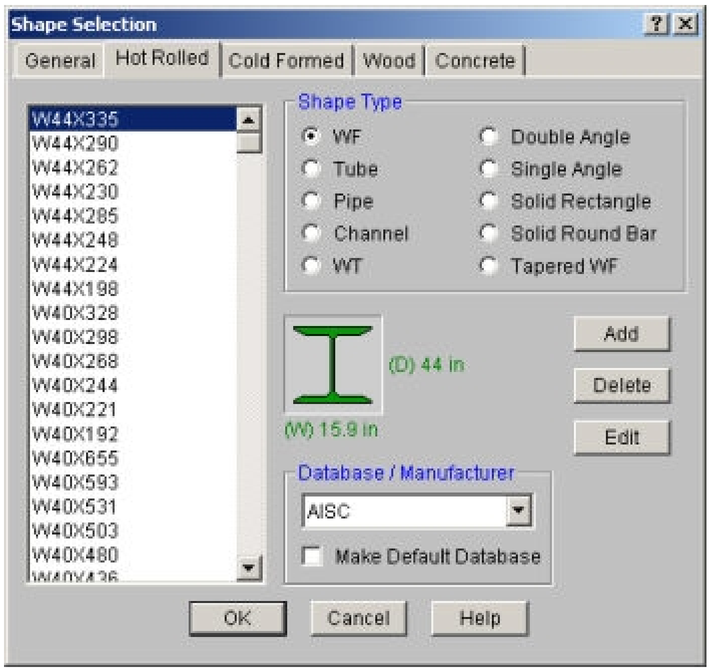
Hot Rolled Steel - Databases

Shapes are organized by **Shape Type** and **Database / Manufacturer**. Common shapes are supported such as wide flanges, tubes, pipes, channels, etc. You may type in the names directly, select shapes from these databases or add your own shapes.

RISA currently supports the following common Hot Rolled steel databases: AISC (United States), Australian, British, Canadian, Chilean, Chinese, European, Indian, Korean, Mexican and Trade Arbed.

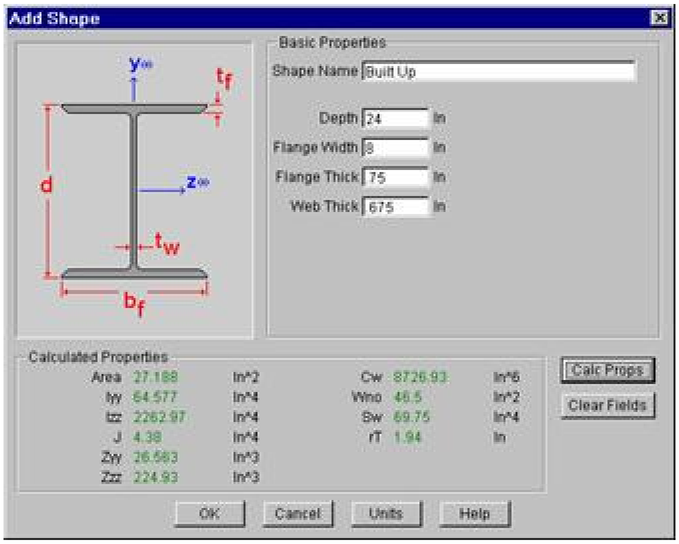
Note:

- Older AISC shapes which are no longer part of the AISC database will be automatially moved / exported to a new AISC_backup database during installation or update.



To Select a Hot Rolled Database Shape

1. From the Hot Rolled tab on the **Section Sets Spreadsheet**, or the **Primary** tab of the **Members Spreadsheet,** move the cursor to the **Shape** field and click .
2. Specify the database and shape type you wish to use and then select from the list of available shapes by clicking on .



To Add a Database Shape

1. On the **RISA Toolbar** click the **Edit Shape Database**  button.
2. Specify the database and shape type you wish to add and then click the **Add** button.
3. Specify a name for the shape and fill in the **Basic Properties.**
4. Click **Calc Props** to determine the shape properties.

Wide Flange Database Shapes

For the AISC database, wide flange shapes are called out by the designation given them in the steel manuals. For example, if you wanted to use a W10x33 you would enter W10X33 as the shape name in the database shape field. M, S and HP shapes are also available. Trade Arbed shapes are called out similar to AISC shapes but with a "_ARB" suffix. I.e. to call a Trade Arbed W12X96 would enter W12X96_ARB as the shape name in the database shape field. Canadian and British shapes use the same format as the AISC shapes, but there values are metric. The depth is called out in millimeters and the mass per length is kg/meter.

Tube Database Shapes(Hollow Rectangular shapes)

The HSS tube properties are also available in the AISC database. The prefix for these tube shapes is "HSS". The syntax is "HSSdepthXwidthXthick", where "depth" is the tube depth, "width" is the tube width and "thick" is the tube thickness in number of 1/16ths. The nominal wall thickness is always used to call out a HSS tube, even though the design wall thickness will vary based on the manufacturing process for the tube. Tubes manufactured using the ERW process will use .93 times the nominal wall thickness as their design thickness. Tubes manufactured using the SAW process will use the full nominal thickness as their design thickness. For example, an HSS12X10X8 would be a 12" deep, 10" wide tube, and a have a design wall thickness of .465" = .93*1/2" (8/16ths) . A HSS32X24X10 would be 32" deep by 24" wide, and have a design wall thickness of 5/8"(10/16ths)

For the Canadian database, tubes also have a "HSS" prefix and the dimensions are all called out in millimeters. British shapes use the prefix "SHS" for square tubes and "RHS" for rectangular tubes.

Note:

- The prefix for older AISC tube shapes is "TU". These shapes reflect the properties that were published with the older AISC 9th edition. As such, these shapes may only exist in the AISC_backup database.
- Tubes using the TU prefix will have a design wall thickness of the nominal wall thickness. The syntax is "TUdepthXwidthXthick", where "depth" is the tube depth, "width" is the tube width and "thick" is the tube thickness in number of 1/16ths. For example, TU16X12X8 would be a 16" deep, 12" wide tube with a thickness of 1/2" (8/16ths).

Pipe Database Shapes

Pipe shapes, which are hollow circular shapes, are entered as on-line shapes. The syntax for these shapes is "PIdiaXthick", where "dia" is the pipe outside diameter and "thick" is the pipe thickness (in inches or centimeters). For example (assuming US Standard units), PI10X.5 would be a 10" diameter pipe with a wall thickness of 1/2".

Channel Database Shapes

Channel shapes are entered with the "C" or “MC” prefix. For example C15X50 would be a valid entry. For Canadian and British shapes, the depth is called out in millimeters and the mass per length is in kg/meter.

Tee Database Shapes

The Tee shapes are entered with the "WT", “MT” or “ST” prefix. For example WT15X74 would be a valid entry. For Canadian and British shapes, the depth is called out in millimeters and the mass per length is in kg/meters.

Tapered Wide Flange Shapes

Tapered Wide Flange shapes are called out by referring to the shape name that was given when it was defined in the database shape editor. Tapered WF shapes can only be defined as database shapes using the “ADD” shape function in the database editor.

Tapered WF shapes are special in that the cross sectional properties change along the length of the member. This is as opposed to prismatic members, which have the same cross sectional properties along their length. (All other shapes are prismatic members). Keep this in mind when defining the I and J joints for tapered shapes. To obtain alternate tapered shape suggestions it is best to define all Tapered WF members consistently in the shape database and handle orientation with the I and J joints.

The Tapered WF shape can also be used to define a prismatic WF with unequal flanges. Tapered wide flange shapes can taper all cross section properties independently and can also have unequal top and bottom flanges. Each basic property is assumed to taper linearly from the Start value to the End value. Shape properties like the area and the moments of inertia will be computed at any required intermediate point from the linearly interpolated basic properties. (This means that the area and moments of inertia will probably NOT vary linearly along the member length). Intermediate shape properties are used to calculate the member stiffness and stresses. The member stiffness is computed internally from a series of piecewise prismatic sections. The error in the member stiffness computed in this manner, as opposed to the theoretically “correct” stiffness, is always less than 10%.

Note that tapered members are always treated as physical members – not finite members. All the rules and behaviors described for physical members always apply to tapered members even if the physical member flag is not set for those members. See Physical Members to learn more about this feature.

To Add a Tapered Wide Flange to the Database

- To enter a tapered shape in the database click  , select the Tapered Shape Type and click **Add**. Enter the basic shape properties at the Start and End locations and to have all the necessary parameters calculated for analysis and design at the member end points and at all the required intermediate locations.

Note

- To enter a prismatic WF member with unequal top and bottom flanges, just make sure that the shape properties are the same at the Start and End points. The top and bottom flange information is entered independently.

Double Angle Database Shapes

These shapes are entered with the prefix "LL". The syntax is "LLbackXflangeXthickXspace" where "back" is the back to back leg length, "flange" is the single angle flange leg length, "thick" is the angle thickness in number of 1/16ths and "space"

is the space between the angles in 1/8ths. For example, **LL6X3.5X5X3** would be L6X3.5 angles 5/16" thick, long legs back to back with a spacing of 3/8". For the Canadian and British shapes, all the dimensions are called out in millimeters.

Single Angle Database Shapes

Angles are entered with an "L" prefix. The syntax is "LlongXshortXthick", where "long" is the long leg length, "short" is the short leg length, and "thick" is the thickness, in number of 1/16ths. For example, L9X4X8 is a 9" by 4" angle 1/2" (8/16ths) thick. The thickness is entered as 8, because the number of 1/16ths in 1/2 is 8. For the Canadian and British shapes, all the dimensions are called out in millimeters.

Solid Rectangular Shapes

These shapes can be defined as on-line shapes. The syntax is "REhtXbase", where "ht" is the rectangle height and "base" is the rectangle base (in inches or cm). For example, RE10X4 would be a 10" deep, 4" width rectangular shape (assuming US Standard units). These shapes can also be defined in the Shape Editor. When defined in the Shape Editor the depth of the solid rectangular section must always be greater than or equal to the width.

Solid Circular Shapes

These shapes are defined as on-line shapes. The syntax is "BARdia", where "dia" is the circle diameter. For example (assuming metric units), BAR2 would be a circular bar with a diameter of 2 cm.

Hot Rolled Steel - Design

Full code checking member optimization can be applied to standard steel shapes based on the following codes:

- United States:
    - AISC 360-10 (14th Edition) ASD & LRFD
    - AISC 360-05 (13th Edition) ASD & LRFD
    - AISC LRFD (2nd and 3rd Editions)
    - AISC ASD (9th Edition)
- Canada:
    - CSA S16-05
    - CSA S16-01
    - CSA S16.1-94
- Europe/Great Britian:
    - ENV 1993-1-1: 2005
    - ENV 1993-1-1: 1992
    - BS 5950-1: 2000
- India:
    - IS 800: 2007
    - IS 800: 1998
- Australia:
    - AS 4100-1998
- New Zealand:
    - NZS 3404: 1997

The calculations performed encompass all the code requirements (including the local buckling criteria for slender compression elements in Appendix B of the 2nd, 3rd, and 9th Edition AISC codes) except those noted in the *Limitations* section of this document.

To Apply a Steel Design Code

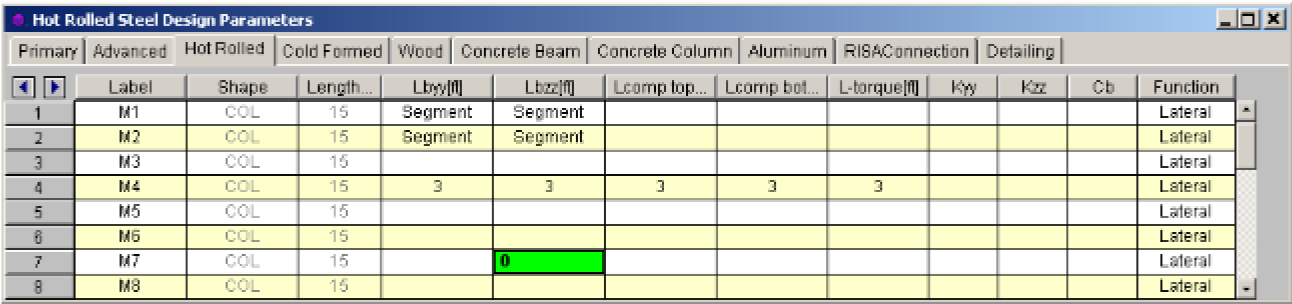
1. On the **Global Parameters**, select the steel code from the drop down list.
2. On the **Hot Rolled** tab of the **Members Spreadsheet**, enter the appropriate bracing information and factors.

Note

- For code checking to be performed on a member, the member must be defined with a database shape on the **Section Sets** or **Members Spreadsheet**.

Design Parameters Spreadsheet

The **Hot Rolled Steel Design Parameters Spreadsheet** records the design parameters for the steel code checks and may be accessed by selecting **Members** on the **Spreadsheets** menu and then selecting the **Hot Rolled** tab. These parameters are defined for each individual member and may also be assigned graphically. See Modifying Member Design to learn how to do this.



	Label	Shape	Length...	Lbyy[ft]	Lbzz[ft]	Lcomp top...	Lcomp bot...	L-torque[ft]	Kyy	Kzz	Cb	Function
1	M1	COL	15	Segment	Segment							Lateral
2	M2	COL	15	Segment	Segment							Lateral
3	M3	COL	15									Lateral
4	M4	COL	15	3	3	3	3	3				Lateral
5	M5	COL	15									Lateral
6	M6	COL	15									Lateral
7	M7	COL	15		0							Lateral
8	M8	COL	15									Lateral

The following topics will discuss the steel design parameters by first discussing how it applies to regular prismatic steel sections. If the parameter is treated differently for Tapered WF shapes, that will be discussed at the end of each topic.

Label

You may assign a unique **Label** to all of the members. Each label must be unique, so if you try to enter the same label more than once you will get an error message. You may relabel at any time with the **Relabel** options on the **Tools** menu.

Shape

The member **Shape** or Section Set is reported in the second column. This value is listed for reference only and may not be edited as it is dictated by the entry in the Section/Shape column on the **Primary** tab.

Length

The member **Length** is reported in the third column. This value may not be edited as it is dependent on the member end coordinates listed on the **Primary Data** tab. It is listed here as a reference for unbraced lengths which are discussed in the next section.

Member Design Parameters - General

Unbraced Lengths

You may specify unbraced lengths or have RISA-3D calculate them for you. The unbraced lengths are:

- **Lbyy**
- **Lbzz**
- **Lcomp-top**
- **Lcomp-bot**
- **L-torque**

The calculated unbraced lengths are listed on the **Member Detail** report after solution.

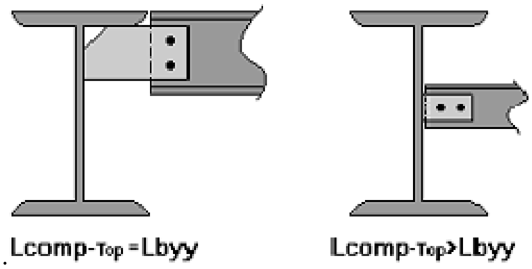
Lb Values

The **Lb** values: **Lb**yy and **Lb**zz, represent the unbraced length for the member with respect to column type buckling about the member's local y and z axes, respectively. These **Lb** values are used to calculate KL/r ratios for both directions, which in turn impact the calculation of allowable axial stress or the axial strength. The KL/r ratios gauge the vulnerability of the member to buckling. Refer to Chapter E of the ASD or LRFD code for more information on this. Also, Section B7 (for both codes) lists the limiting values for KL/r.

The **Lcomp** values, **Lcomp-top** and **Lcomp-bot**, are the unbraced lengths of the compression flanges for flange buckling due to flexure. These may be the same as the **Lb**yy value, but not necessarily. The **Lcomp** values are used in the calculation of allowable bending stress, or bending strength. Refer to Chapter F of the ASD or LRFD code for more information on this, specifically the definition of "*l*" on page 5-47 of the ASD code or the definition of Lb on page 6-53 of the LRFD code.

Continuous Beams

For continuous beams the moment will reverse such that the top and bottom flanges will be in compression for different portions of the beam span. **Lcomp-top** is the unbraced length of the top flange and **Lcomp-bot** is the unbraced length of the bottom flange.



Default Values

If left blank these unbraced lengths all default to the member's full length. The exception to this is if **Lb**yy is entered and **Lcomp-top** is left blank, **Lcomp-top** will default to the entered value for **Lb**yy

L-torque

The **L-torque** value represents the unbraced torque length of the member. This value is used in the Torsional and Flexural-Torsional Buckling calculations per chapter E4 of the AISC 14th edition. Setting this entry to zero will disable all the chapter E4 checks for your members when the AISC 14th edition code it selected. This entry will be ignored for all other design codes.

Note

- The L-torque value is NOT used for calculation stiffness or stress for members subjected to warping. This is done using an internal "warping length" set by the program.
- The Wide Flange check per equation (E4-4) will only be checked if L-torque > Lbzz AND Lbyy.
- Single Angles will only be checked if b/t>20, per the code provision in chapter E4, for principal axis buckling.

Default Values

If left blank, this unbraced length defaults to the member's full length.

Segment Command

For physical members, you can enter the code '**Segment**' in the unbraced length fields and the length of each segment will be used. A "segment" is the distance between the joints that are on the physical member. For example, suppose you have a physical member that is 20 feet in length, and there are two joints along the physical member, one 5 feet from the end and one at 15 feet. An unbraced length of 5 feet will be used for the first segment, then a value of 10 feet will be used in the middle segment, and again a value of 5 feet would be used in the last segment.

Notes

- If the intermediate framing members are considered to brace the bottom flange, then you can enter “segment” for Lcomp-bot. When the “segment” command is used ALL intermediate Joints along the beam are viewed as brace points. Therefore, you may have to delete unused or extraneous Joints.
- The Top Flange is defined as the flange corresponding to the positive local y axis for the member. For more information on setting local axes refer to the Members section.

- Chapter C3 of the 3rd Edition of AISC's LRFD specification gives additional criteria regarding strength and stiffness requirements for column and bracing.
- For ASD 9th edition calculation of Kl/r for WTs and Double Angles, there is an effective Kl/r ratio that is used. This can be found in the commentary Section E3.
- Specifying LcompTop and LcompBot both as zero constrains a single angle to bend/buckle about its geometric axes. Otherwise it will behave about its principal axes. Alternatively, specifying L-torque as zero also constrains single angles to bend/buckle about their geometric axes.
- For single angles behaving about their principal axes, Lbyy specifies bracing against buckling about the minor principal axis. Lbzz specifies bracing against buckling about the major principal axis.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Unbraced Lengths**.

K Factors (Effective Length Factors)

The **K Factors** are also referred to as effective length factors. **Kyy** is for column type buckling about the member's local y-y axis and **Kzz** is for buckling about the local z-z axis.

If a value is entered for a **K Factor**, that value will be used for the entire length of the physical member. If an entry is not made (left blank), the value will internally default to '1' for that member. See the ASD or LRFD code commentary for Chapter C for an explanation of how to calculate K Factors.

Note

- **Kt** for the Torsional or Flexural-Torsional Buckling checks of chapter E4 of the AISC 14th edition will always be assumed to be 1.0.
- RISA-3D is able to approximate the K values for a member based on the member's sway condition and end release configuration. The K-factor approximation is based on Table C-C2.2, found on page 16.1-240 of the AISC 13th Edition code. The following table gives the values used for various conditions.

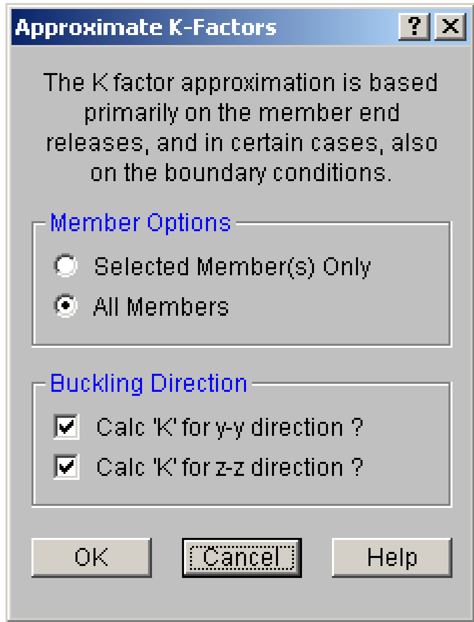
Table Case	End Conditions	Sidesway?	K-Value
(a)	Fixed-Fixed	No	.65
(b)	Fixed-Pinned	No	.80
(c)	Fixed-Fixed	Yes	1.2
(d)	Pinned-Pinned	No	1.0
(e)	Fixed-Free	Yes	2.1
(f)	Pinned-Fixed	Yes	2.0

RISA-3D will recognize a pinned boundary condition for the K approximation for a full pin, i.e. if all the rotations in the boundary condition are released. If any of the rotations in a boundary condition are restrained, the boundary condition is considered "fixed" for the K factor approximation.

Any configuration not described here will be given the default value of '1.0'.

If any value that influences these K values is changed, the K factor approximation should be redone. For instance, if you have RISA-3D approximate K factors, then change some of the member end release designations, you should redo the K factor approximations.

Remember that the K factors are *approximations* and you should check to make sure you agree with all K factors RISA-3D assigns. You can always override a K factor after an approximation by directly entering the value that you want in the appropriate field. Keep in mind that a subsequent approximation will overwrite any manually input values so you will need to override the approximation each time it is performed.



To Auto-Calc K-Factors

1. Open the Members spreadsheet from the Data Entry toolbar and click on the any of the Hot Rolled, Cold Formed, Wood, Concrete Beam or Concrete Column tabs.
2. Select the members you wish to have K-factors auto calculated by highlighting rows in yellow. If you would like all members under an individual tab to be calculated you don't need to highlight anything.
3. Click the **Calculate Approximate K Values** button or right-click in the specific member spreadsheet and choose **Approximate K**.
4. Choose the appropriate options.

Limitation:

RISA-3D will currently neglect the influence of adjoining framing members when those members are connected at a joint that also has degrees of freedom restrained by boundary conditions. For example, suppose a column and beam member connect at a joint that is restrained for translation in all directions (i.e. the joint is "pinned"). The K factor approximation will neglect the beam member when it calculates the K factor for the column and visa-versa. The effect will be that the ends of the members at that joint will be seen as "pinned" and not "fixed" for the K factor approximation.

Tapered Wide Flanges - The **Kyy** factor for Tapered WF shapes can be approximated by RISA-3D, and is the same as for a regular prismatic member. The **Kzz** value cannot be approximated by RISA-3D and should be entered by the user. The default value for **Kzz** will be '1.0' if not entered by the user. See the ASD or LRFD Commentary on Appendix F for an explanation of how to calculate the **Kzz** factor for Tapered members.

Sway Flags

The **Sway Flags** indicate whether the member is to be considered subject to sidesway for bending about its local y and z axes. The **y sway** field is for y-y axis bending and the **z sway** field is for z-z axis bending. Click on the field to check the box and indicate that the member is subject to sway for that particular direction, or leave the entry blank if the member is braced against sway. These sway flags influence the calculation of the K factors as well as the Cm and Cb factors.

Member Design Parameters - AISC Codes

Cm - Interactive Bending Coefficient

Cm Coefficients are described in Chapter H of the 9th Edition AISC (ASD) code. If these entries are left blank, they will be automatically calculated.

The **Cm** value is influenced by the sway condition of the member and is dependent on the member's end moments, which will change from one load combination to the next, so it may be a good idea to leave these entries blank.

Tapered Wide Flanges - For Tapered WF members, the Cm values will be used for C'm. The C'm values are described in Appendix F7.6 of the ASD and LRFD codes. These terms are used in the interaction equations in Appendix F. If these entries are left blank they will be calculated automatically.

Note

- The Cm factor is not used for LRFD code checking because the Chapter C requirement that P-Delta effects be considered is met with a direct P-Delta analysis. LRFD code checks will not be performed without P-Delta analysis.

Cb - Lateral-Torsional Buckling Modification Factor

Cb Factors are described in Chapter F of the AISC code and are used in the calculation of the nominal flexural strength, Mn. If this entry is left blank, it will be calculated automatically for AISC code checks.

The calculation of **Cb** is based on the unbraced length of the compression flange and the moment diagram for the unbraced segment in question. If a specific unbraced length is entered by the user, the program cannot interpret the location of brace points and the Cb value will default to '1.0'. In some cases, it may be better to enter 'segment' as the unbraced length for a physical member. When 'segment' is entered, the Cb value will be calculated individually for each segment of the beam based on that segment's moment diagram.

Tapered Wide Flanges - For Tapered WF members, the Cb field is actually used to enter the "B" value. B is described in Appendix F7.4 of the ASD (9th Edition) and LRFD (2nd & 3rd Edition) codes. This value is not calculated automatically and if it is left blank, a value of '1.0' will be used per the commentary for Appendix F7. The Cb term used in the Chapter F equations is calculated internally for Tapered WF members and will be shown on the Member Detail report and in the Code Check Spreadsheet.

Function for Stiffness Reduction

The **Function** entry may be set to either 'Lateral' or 'Gravity' using the drop down list in the spreadsheet. If an AISC 13th Edition code (ASD or LRFD) is selected on the Codes tab of the Global Parameters Dialog, then all members with a 'Lateral' Function will be considered for the stiffness reduction required in Appendix 7 (Direct Analysis Method) of the code.

The **Flexural Stiffness Reduction** of section 7.3.(3) will be applied to all 'Lateral' members whose member type is set to either 'Beam' or 'Column' on the Primary Tab of the Members Spreadsheet.

The program can perform an iterative analysis during the solution depending on the value of $\boldsymbol{\tau_b}$. In this case, the stiffness matrix is recomputed for each iteration until the value of τ_b converges within 1 percent for all 'Lateral' members in compression. In the unlikely event that τ_b is less than zero, the value of τ_b is considered to be 1.e-5. When used in the analysis, the value for $\boldsymbol{\tau_b}$will be listed in the Detail Report for that member.

When the users sets the Adjust Stiffness flag on the Global Parameters to **Yes (Tau =1.0)**, then the program will use a Tau of 1.0 in the stiffness analysis and no iteration of the stiffness matrix is necessary. This option is a good feature for models which take a long time to solve or which have not yet been proportioned to control drift.

The **Axial Stiffness Reduction** of section 7.3.(4) will be applied to all 'Lateral' members whose member type is set to either 'Column' or 'VBrace' on the Primary Tab of the Members Spreadsheet.

Note

- The stiffness reduction required by the Direct Analysis Method will be ignored if the Adjust Stiffness option is not selected on the codes tab of the **Global Parameters**, or if the design code chosen does not have an option for stiffness reduction.
- When the Adjust Stiffness flag is set to **Yes (Tau=1.0),** then the code would requires a higher value for the applied Notional Loads.

Allowable Stress Increase Factor (ASIF)

AISC 9th Edition

Increasing of allowable stresses may be allowed when forces are transient. You may enter an **ASIF** factor on the **Load Combinations Spreadsheet** to allow the increase for a specific load combination. The ASIF factor is then applied to the allowable stresses in accordance with section A5. The ASIF factor also is applied to the Euler values (F'e).

Note

- If the allowable stress increase is being used, the final code check value should still be compared to '1.0'.

All Other US Codes

Setting the **ASIF** factor to a value greater than '1.0' will not cause the capacities to be increased by that factor. However, setting the **ASIF** factor to a value greater than '1.0' is used as a flag to use of the seismic compactness criteria of Table I-8-1 of AISC 341-05 Seismic Provisions of Steel Buildings. Specifically we will use the limiting width-thickness ratios from this table for capacity calculations (for compression flange local buckling for example). In these cases, we use Table I-8-1 of AISC 341-05 as opposed to Table B4.1 of AISC 360-05.

Hot Rolled Design Parameters - Canadian

Parameters controlling the steel design are entered on the **Member Design Parameters** spreadsheet. These parameters are entered on a per member basis, and control the code checking on a per member basis.

w1 Coefficients (Interactive Bending Coefficients)

Section 13.8.4 of the Canadian code describes the w1 coefficient. If these entries are left blank, RISA-3D will calculate them. The w1 value is dependent on the member's end moments, which will change from one Load Comb to the next. It's a good idea to leave these entries blank and let RISA-3D calculate them.

Tapered Wide Flanges - For Tapered WF members, the LRFD code will be used and the Cmyy(w1yy) value will be used for C'myy, and the Cmzz(w1zz) value will be used for C'mzz. The C'm values are described in Appendix F7.6 of the LRFD codes. These terms are used in the interaction equations in Appendix F for the LRFD code. If these entries are left blank RISA-3D will calculate them.

w2 Coefficients (Bending Coefficients)

This coefficient is discussed in Section 13.6 of the CAN/CSA S16.1-94 code and is used in the calculation of Mr, the flexural strength. If this entry is left blank it will be calculated automatically. This value also is dependent on the member's end moments so it may be a good idea to let it be calculated internally. An exception to this would be for cantilever members, where this value should be 1.75, and it will be automatically calculated as 1.0. This will be addressed in a future program version.

Tapered Wide Flanges - For Tapered WF members the LRFD code will be used and the w2 field is used for the “B” value. B is described in Appendix F7.4 of the LRFD codes. The Cb term used in the Chapter F equations in the LRFD code is always calculated internally for Tapered WF members and will be shown on the Member Detail report and in the Unity check spreadsheet. If the w2 entry is left blank a value of 1.0 will be used for B, per the commentary for Appendix F7. (The value of “B” is not calculated at this time.)

Hot Rolled Design Parameters - British

Parameters controlling the steel design are entered on the **Member Design Parameters** spreadsheet. These parameters are entered on a per member basis, and control the code checking on a per member basis.

m Coefficients (Interactive Bending Coefficients)

Section 4.8.3.3.4 of the British code describes the m coefficient. If these entries are left blank, RISA will calculate them. The m value is influenced by the sway condition of the member and is dependent on the member's end moments, which will change from one Load Comb to the next. It's a good idea to leave these entries blank and let RISA calculate them.

Tapered Wide Flanges - For Tapered WF members, the LRFD code will be used and the Cmyy(m_yy) value will be used for C'myy, and the Cmzz(m_zz) value will be used for C'mzz. The C'm values are described in Appendix F7.6 of the LRFD codes. These terms are used in the interaction equations in Appendix F for the LRFD code. If these entries are left blank RISA will calculate them.

m-LT Coefficients (Bending Coefficients)

This coefficient is discussed in Section 4.3.6.6 of the BS5950-1:2000 code and is used in the calculation of the flexural strength. If this entry is left blank it will be calculated automatically. This value also is impacted by the member's sway condition and is dependent on the member's end moments so it may be a good idea to let it be calculated internally. An exception to this would be for cantilever members in sway frames, this value should be 1.75, and it will be automatically calculated as 1.0. This will be addressed in a future program version.

Tapered Wide Flanges - For Tapered WF members the LRFD code will be used and the m-LT field is used for the “B” value. B is described in Appendix F7.4 of the LRFD codes. The Cb term used in the Chapter F equations in the LRFD code is always calculated internally for Tapered WF members and will be shown on the Member Detail report and in the Unity check spreadsheet. If the m-LT entry is left blank a value of 1.0 will be used for B, per the commentary for Appendix F7. (The value of “B” is not calculated at this time.)

Hot Rolled Design Parameters - EuroCode

Parameters controlling the steel design are entered on the **Member Design Parameters** spreadsheet. These parameters are entered on a per member basis, and control the code checking on a per member basis. Note that most fields are used for more than one parameter (see the labels at the top of each column). The value you enter will be interpreted based on the type of shape that the current member references. The three possible shape types with regard to code checking are: regular prismatic steel section, Tapered WF steel shape, and wood section.

The following topics will discuss the steel design parameters by first discussing how it applies to regular prismatic steel sections. If the parameter is treated differently for Tapered WF shapes, that will be discussed at the end of each topic. (Note that if the Canadian Code is the current design code, all Tapered WF shapes will be designed using the LRFD 2nd code.) For NDS design see the Wood - Design section.

Bm Coefficients (Interactive Bending Coefficients)

Section 5.5.4 (7) of the EuroCode describes the Bm coefficients. If these entries are left blank, RISA will calculate them. The Bm value is influenced by the sway condition of the member and is dependent on the member's end moments, which will change from one Load Comb to the next. It's a good idea to leave these entries blank and let RISA calculate them.

Note:

- In the 2005 EuroCode, the nomenclature for these factors changed to Cm and Cm.LT. Please see Annex A of the 2005 EuroCode for more information on how these are calculated.

Tapered Wide Flanges - For Tapered WF members, the LRFD code will be used and the Cmyy(m_yy) value will be used for C'myy, and the Cmzz(m_zz) value will be used for C'mzz. The C'm values are described in Appendix F7.6 of the LRFD

codes. These terms are used in the interaction equations in Appendix F for the LRFD code. If these entries are left blank RISA will calculate them.

Bm-LT Coefficients (Bending Coefficients)

This coefficient is discussed in Section 5.5.4 (7) of the 1992 EuroCode code and is used in the calculation of the flexural strength. If this entry is left blank it will be calculated automatically. This value also is impacted by the member's sway condition and is dependent on the member's end moments so it may be a good idea to let it be calculated internally. An exception to this would be for cantilever members in sway frames, this value should be 1.75, and it will be automatically calculated as 1.0. This will be addressed in a future program version.

Note:

- In the 2005 EuroCode, the nomenclature for these factors changed to Cm and Cm.LT. Please see Annex A of the 2005 EuroCode for more information on how these are calculated.

Tapered Wide Flanges - For Tapered WF members the LRFD code will be used and the m-LT field is used for the “B” value. B is described in Appendix F7.4 of the LRFD codes. The Cb term used in the Chapter F equations in the LRFD code is always calculated internally for Tapered WF members and will be shown on the Member Detail report and in the Unity check spreadsheet. If the m-LT entry is left blank a value of 1.0 will be used for B, per the commentary for Appendix F7. (The value of “B” is not calculated at this time.)

Hot Rolled Design Parameters - Indian

Parameters controlling the steel design are entered on the **Member Design Parameters** spreadsheet. These parameters are entered on a per member basis, and control the code checking on a per member basis. Note that most fields are used for more than one parameter (see the labels at the top of each column). The value you enter will be interpreted based on the type of shape that the current member references. The three possible shape types with regard to code checking are: regular prismatic steel section, Tapered WF steel shape, and wood section.

The following topics will discuss the steel design parameters by first discussing how it applies to regular prismatic steel sections. If the parameter is treated differently for Tapered WF shapes, that will be discussed at the end of each topic. (Note that if the Canadian Code is the current design code, all Tapered WF shapes will be designed using the LRFD 2nd code.)

Cm Coefficients (Interactive Bending Coefficients)

If these entries are left blank, RISA will calculate them. The m value is influenced by the sway condition of the member and is dependent on the member's end moments, which will change from one Load Comb to the next. It's a good idea to leave these entries blank and let RISA calculate them.

Tapered Wide Flanges - For Tapered WF members, the LRFD code will be used and the Cmyy(m_yy) value will be used for C'myy, and the Cmzz(m_zz) value will be used for C'mzz. The C'm values are described in Appendix F7.6 of the LRFD codes. These terms are used in the interaction equations in Appendix F for the LRFD code. If these entries are left blank RISA will calculate them.

General Limitations

Welded Sections - There is a basic assumption in the program that the steel sections are hot rolled not welded. Therefore, any code provisions that assume additional stresses due to welding a built-up cross section are not specifically accounted for. For the AISC-LRFD codes this generally means that the flange residual stress (Fr) is always taken as 10ksi, as for a rolled shape. The only exception is for Tapered WF shapes where it is always taken as 16.5 ksi, as for a welded shape.

Load Location - For all shape types, it is assumed that the transverse load on the member is occurring through the member's shear center. This means secondary torsional moments that may occur if the load is not applied through the shear center are not considered.

Single Angles - In all codes other than AISC 360-05 and 360-10, single angles are only checked for axial loading. Flexural effects are not considered in the code check calculation. Under AISC 360-05 and 360-10 single angles are checked for

combined bending and axial about either their principal or geometric axes depending on how their unbraced length has been defined.

- Lateral-Torsional buckling does not apply to Geometric bending angles, and is therefore not checked for them.
- Lateral-Torsional buckling does not apply to minor-axis bending on Principal bending angles, and is therefore not checked for them.
- The provisions of Section E5 of the specification are not considered. The angle is considered to be loaded concentrically, and on both legs.
- Interaction equation H2-1 is used for all the code check on all angles (equal and unequal leg).

Double Angles - For y axis buckling (where stitch connections would be experiencing shear), the program only considers KL/r of the overall built up shape and does NOT attempt to reduce the KL/r value based on the spacing between connectors. Therefore the program assumes that there are pretensioned bolts or welds spaced closely enough to allow the double-angles to act as one unit per AISC 360-10 Eqn E6-2a.

Net Section for Tension Capacity - The tension capacity is calculated using the Gross area. A later release will include a "net area factor" that the user can enter to indicate what the effective area should be for the tension capacity calculation.

Torsional and Flexural Torsional Buckling of Doubly Symmetric Shapes - Flexural Torsional Buckling and Torsional Buckling are checked for the following shape types for AISC 360-10 (14th Edition Steel Code):

- Non Slender WF
- Non Slender Channels
- Non Slender WTs
- Non Slender Double Angles
- Non Slender Single Angles

Flexural Torsional Buckling and Torsional Buckling are also checked for Non Slender WTs and Double Angles for the AISC 360-05 (13th Edition Steel Code).

For WTs $(KL/r)_m$ per section E4 (a) is calculated per equation E6-1 with '**a**' always assumed to be 0. This results in $(KL/r)_m$ = $(KL/r)_o$.

P-Little Delta Analysis - An incremental P-Delta re-iterative approach is used in RISA 3D. This method does NOT automatically account for P-Little Delta effects. If these effects are expected to be significant please refer to the P-Little Delta section.

Notional Loads - Currently Notional Loads are not automatically included in the analysis and it is expected that the user will create their own set of Notional Loads when required. Please see the Load Generation - Notional Loads topic for more information on using the program to generate and apply notional loads.

Limitations - AISC

ASD 9th Edition Limitations

Wide Flange Shapes - Code checks for shapes that qualify as plate girders, as defined by Chapter G, are not performed. Plate girders that can be checked by the provisions of Chapter F will have code checks calculated.

Channels - The AISC 9th Edition (ASD) specification does not specifically address the allowable stress for weak axis bending of channels. Therefore, the program uses the most similar formula for the weak axis bending of wide flanges (0.75*Fy). For a complete and thorough treatment of channel code checks, refer to the LRFD specification.

WT and LL Shapes - ASD allowable bending stresses calculated for WT and LL shapes use Chapter F for cases when the stem is in compression. This is not technically correct, but the ASD code does not provide direction regarding other means of calculating the allowable bending stress in this situation. This issue will be addressed in a future program release. In the interim, the LRFD code directly addresses this situation, so it is recommended that you use the LRFD code to check WT and LL shapes that have their stems in compression.

Neither the ASD or LRFD codes address the rare case where Lateral Torsional (or Flexural Torsional) Buckling occurs for WT's and double angles bent about their weak axis.

RE Shapes - Rectangular bar members (on-line shapes) are assigned allowable bending stresses for the yielding limit state only. Lateral torsional buckling is not considered because the ASD code doesn't directly address this for rectangular shapes. The strong axis bending stress is assigned as 0.66*Fy and the weak axis bending stress is assigned as 0.75*Fy. If you have a case where lateral torsional buckling may govern, you should use the LRFD code, since the LRFD code does address this limit state.

LRFD 3rd and 2nd Edition Limitations

Wide Flange Shapes - LRFD code checks for shapes that qualify as plate girders as defined by Chapter G are not performed.

Single Angles - Single angles are only checked for Euler buckling. They are not checked for Flexural-Torsional buckling.

WT and LL Shapes - Neither the ASD or LRFD codes address the rare case where Lateral Torsional (or Flexural Torsional) Buckling occurs for WT's and double angles bent about their weak axis.

Tapered Wide Flanges - ASD 9th edition code checks can be performed on tapered members with equal or unequal top/bottom flanges, with the restriction that the compression flange area must be equal to or larger than the tension flange area. LRFD 2nd edition code checks are limited to tapered members with equal area flanges. Code checks are performed using Appendix F, Chapter F, and Chapter D as applicable. Note that the rate of taper is limited by Appendix F, and the program enforces this. The interaction equations in Appendix F are used to compute the final code check value. These equations also include the effects of weak axis bending, if present. Torsional warping effects on Tapered WF members are NOT considered.

Prismatic Wide Flanges with Unequal Flanges - ASD code checks for prismatic WF members with unequal flanges are also limited to shapes that have the compression flange area equal to or larger than the tension flange area. LRFD code checks currently cannot be performed for prismatic WF members with unequal flanges.

Pipes and Bars - For pipes and round bars, the code check is calculated based on an SRSS summation of the y and z-axis stresses calculated for the pipe or bar. This is done because these circular shapes bend in a strictly uniaxial fashion and calculating the code check based on a biaxial procedure (as is done for all the other shapes) is overly conservative.

Single Angles - Code checking (LRFD or ASD) on single angle shapes is based on P/A (axial load/axial strength or axial stress/allowable axial stress) only. This is because the load eccentricity information needed for a meaningful bending calculation is not available. Only Euler buckling is considered for single angles, flexural-torsional buckling is NOT considered. Single angles will have the following message displayed on the Code Check Spreadsheet to remind the user of the axial only code check: "Code check based on z-z Axial ONLY"

Please see Single Angle Stresses for more information on the calculation of single angle stresses.

Limitations - Canadian

It is assumed the transverse load on the member is occurring through the member's shear center. This means secondary torsional moments that may occur if the load is not applied through the shear center are not considered.

Slender Shapes - Shapes with any slender elements are not supported for axial compression. Shapes with slender webs or flanges are not supported for flexure. These shapes use the criteria in the CAN/CSA S136 code, which is not supported at this time.

Compressive Strength - For the equations in section 13.3.1, the parameter “n” is assigned a value of 1.34 for all shapes. This is conservative for WWF shapes and HSS shapes that are stress-relieved.

WT and LL Shapes - The criteria in the AISC LRFD 2nd Edition code is used to perform code checks on WT and LL shapes since the Canadian code does not explicitly specify how to calculate the flexural strength of WT and LL shapes.

The Canadian code does not address the rare case where Lateral Torsional (or Flexural Torsional) Buckling occurs for WT's and double angles bent about their weak axis.

Tapered Wide Flanges - The AISC LRFD 2nd code is used to perform code checks on Tapered WF shapes when the Canadian code is specified. The Canadian code CAN/CSA S16.1-94 does not address web-tapered members.

Pipes and Bars - For pipes and round bars, the code check is calculated based on an SRSS summation of the y and z-axis stresses calculated for the pipe or bar. This is done because these circular shapes bend in a strictly uniaxial fashion and calculating the unity check based on a biaxial procedure (as is done for all the other shapes) is overly conservative.

Single Angles - Code checking on single angle shapes is performed for tension only. Single angles will have the following message displayed on the Steel Code Check Spreadsheet to remind the user of the tension only code check: "Single Angle code check based on Axial Tension ONLY"

Please see Single Angle Stresses for more information on the calculation of single angle stresses.

Limitations - British

It is assumed the transverse load on the member is occurring through the member's shear center. This means secondary torsional moments that may occur if the load is not applied through the shear center are not considered.

Tapered Members - Tapered WF shapes are done per the AISC LRFD 2nd specification at this time. The appropriate sections of the BS5950-1 specification will be used in a later release.

Torsional Warping Effects - Combined bending and warping torsional stresses in WF and Channel shapes are handled per the AISC publication "Torsional Analysis of Steel Members". A later release will use the SCI publication "Design of Members Subject to Combined Bending and Torsion" for combined bending and warping stresses when the British Hot Rolled Steel code is selected.

Single Angles - Single angles are checked for axial and shear forces only. No bending is considered at this time. A later release will consider the requirements in Annex I4 for single angles.

Secondary Moments per Annex I - The program does not yet consider the internal secondary moments described in Annex I.

Limitations - EuroCode

Tapered Members - Tapered WF shapes are done per the AISC LRFD 2nd specification at this time.

Torsional Warping Effects - Combined bending and warping torsional stresses in WF and Channel shapes are handled per the AISC publication "Torsional Analysis of Steel Members".

WT and Double angle Limitations - The EuroCode does not address the rare case where Lateral Torsional (or Flexural Torsional) Buckling occurs for WT's and double angles bent about their weak axis.

Lateral Torsional Buckling - The value M_{cr} used in the lateral-torsional buckling capacity of beams relies on a factor C1. When the C1 field is left blank it is automatically calculated per the AISC 360-05 (13th Edition) code, as there is no suitable generic formula presented in the EuroCode.

Limitations - Indian

Tapered Members - Tapered WF shapes are done per the AISC ASD 9th (for the 1998 code) and LRFD 2nd (for the 2007 code) specifications at this time. The appropriate sections of the IS:800 specification will be used in a later release.

Torsional Warping Effects - Combined bending and warping torsional stresses in WF and Channel shapes are handled per the AISC publication "Torsional Analysis of Steel Members".

WT and Double angle Limitations - The Indian code does not address the case where Lateral Torsional (or Flexural Torsional) Buckling occurs for WT's and double angles bent about their weak axis. For the 2007 code, RISA will use the AISC 360-05 (aka AISC 13th edtion) formulas for calculating member capacity for these failure states.

Shear Stress - For all shapes, the average shear stress is not checked per section 6.4.2 at this time.

Class 4 (Slender) Sections - No code checking is performed for class 4 sections for the IS 800: 2007 code.

Single Angles - Single angles are checked for axial and shear forces only. No bending is considered at this time.

Limitations - New Zealand and Australia

Tapered Members - Tapered WF shapes are done per the AISC LRFD 2nd specification at this time. The appropriate sections of the ENV 1993-1-1 specification will be used in a later release.

Torsional Warping Effects - Combined bending and warping torsional stresses in WF and Channel shapes are handled per the AISC publication "Torsional Analysis of Steel Members".

WT and Double angle Limitations - The NZ / AS codes do not address the rare case where Lateral Torsional (or Flexural Torsional) Buckling occurs for WT's and double angles bent about their weak axis.

Torsional Unbraced Length - At present the program assumes 'Lcomp' as 'Le' in case of lateral torsional buckling. Section 5.6.3 provides a procedure for calculating 'Le' which is not being addressed at this time.

Special Provisions for Cantilevers - At present the program does not address section 5.6.2 for cantilever elements. The value of alpha_m is being calculated using equation 5.6.1.1(2) in all cases. The user has an option to input his own value of alpha_m which would override the value calculated by the program.

Special Messages - AISC

In some instances, code checks are not performed for a particular member. A message explaining why a code check is not possible will be listed instead of the code check value. You may click the cell that contains the message and look to the status bar to view the full message. The following are the messages that may be listed:

"AISC Code Check Not Calculated"

This message is displayed when the member is not defined with a database shape, a steel code is not specified on the **Global Parameters** or no units were specified. For LRFD this message is displayed if the steel yield stress is less than 10ksi.

"Web is slender per Table B5.1, handle as a plate girder"

The ratio h/tw exceeds the limiting criteria listed in Table B5.1. This means Chapter G (plate girders) governs.

"Compressive stress fa exceeds F'e (Euler bucking limit)"

The axial compressive stress for the member is greater than the Euler buckling stress (per ASD criteria), meaning the member is unstable. A code check can not be calculated.

*"Tube depth>6*width (Sec F3-1) where width=bf-3*tw, Sec B5-1"*

A tube is failing to meet the depth/width requirements laid out in Section F3-1 of the ASD code. The depth of the tube is the full nominal depth, which the width is taken as the full width minus 3 times the thickness. Section B5-1 specifies this calculation for the width when the fillet radius is not known.

"Tee or Channel fails on Table A-B5.1 (Appendix B)"

This message appears for ASD code checking when Appendix B calculations are being done for a Tee or Channel shape and the shape fails to meet the requirements of Table A-B5.1, Limiting Proportions for Channels and Tees.

"Pipe diameter/thickness ratio exceeds 13,000/Fy (App. B)"

This message appears when Appendix B calculations are being done for a pipe shape and the diameter/thickness ratio exceeds the limit of 13000/Fy specified in Section B5-b for ASD, Section B5-3b for LRFD.

"KL/r > 200 for compression member (LRFD)"

Section B7 recommends that KL/r for compression members not exceed 200. For the ASD code, a procedure is presented to handle when KL/r exceeds 200, so for ASD, KL/r>200 is permitted. For LRFD, no guidance is provided as to what to do if KL/r>200, so for LRFD this is not permitted. You may override this in the **Preferences** on the **Tools** menu.

"Taper Flange area is not constant per App. A-F7-1 (b)"

The limitations of Appendix F for the design of web tapered members include the restriction that the flange area shall be constant along the length of the member. This member's flange area changes along its length. See Appendix Section F7.1 (b).

"Taper rate exceeds gamma limit per App. A-F7-1 (c)"

The limitations of Appendix F for the design of web tapered members include a limit on how steep the rate of taper can be along the member length. This member's taper rate exceeds the limit given by equation A-F7-1. See Appendix Section F7.1 (c).

"Flanges not equal, currently don't do LRFD App. F1 calcs"

The requirements for Wide Flange members with unequal flanges in the LRFD, Appendix F1, are not addressed.

"Taper Comp. flange < Tension flange, per App A-F7-1 (b)"

The limitations of Appendix F for the design of web-tapered members include the restriction that the flange areas of the top and bottom flange must be equal. The compression flange may be larger than the tension flange. However equation A-F7-4 is unconservative for cases where the compression flange is smaller than the tension flange. See Appendix Sections F7.1 (b) and F7.4.

Special Messages - Canadian

When a code check is not performed for a particular member a message explaining why a code check is not possible will be listed instead of the code check value. You may click the cell that contains the message and look to the status bar to view the full message. Following are the messages that may be listed specifically for the Canadian Code:

"Tension member with L/r ratio > 300 (CAN Sect. 10)"

The maximum L/r ratio for this tension member exceeds the limit shown in Section 10.2.2 of the CAN/CSA S16.1-94 code.

"KL/r > 200 for compression member (CAN Sect. 10)"

The maximum L/r ratio for this compression member exceeds the limit shown in Section 10.2.1 of the CAN/CSA S16.1-94 code. You may override this check in the **Preferences** on the **Tools** menu.

"Can't do unity check for slender compression member"

Compression strengths are not calculated for shapes that contain elements where the width to thickness ratios are classified as “slender”.

"Can't do unity check for flexural member with slender web"

Flexural strengths are not calculated for shapes which have a web depth to thickness ratio classified as “slender”.

"Currently can't do unity check for member with slender flanges"

Flexural strengths are not calculated for shapes which have a flange width to thickness ratio classified as “slender”.

"Can't do unity check for single angles in compression"

Compression strengths are not calculated for single angle members.

Special Messages - Eurocode

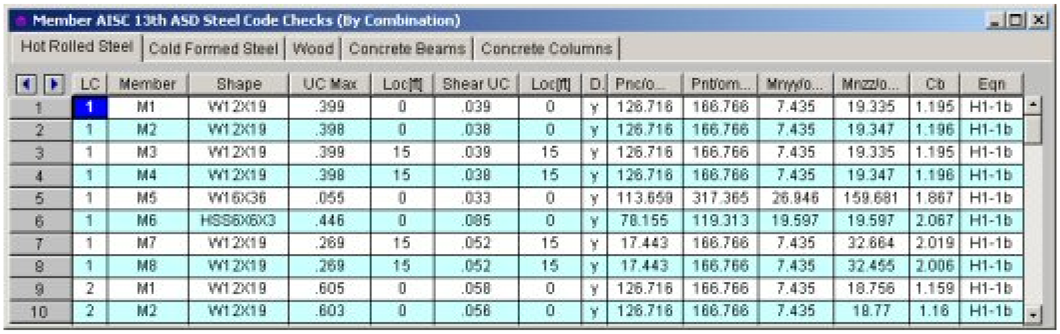
"Rho >= 1.0, No code check calculated"

In beams with high shear the code implements a moment strength reduction factor (ρ), but does not place an upper limit on it. If this value exceeds 1.0 the beam has a negative moment capacity, which is irrational. The program therefore prevents a code check for these circumstances.

Hot Rolled Steel - Design Results

AISC Code Check Results

Access the **Steel Code Check** spreadsheet by selecting the **Results** menu and then selecting **Members** ‣ **Steel Code Checks**.


Member AISC 13th ASD Steel Code Checks (By Combination)

Hot Rolled Steel | Cold Formed Steel | Wood | Concrete Beams | Concrete Columns

	LC	Member	Shape	UC Max	Loc[ft]	Shear UC	Loc[ft]	D.	Pnc/o...	Pnt/om...	Mnyy/o...	Mnzz/o...	Cb	Eqn
1	1	M1	W12X19	.399	0	.039	0	y	126.716	166.766	7.435	19.335	1.195	H1-1b
2	1	M2	W12X19	.398	0	.038	0	y	126.716	166.766	7.435	19.347	1.196	H1-1b
3	1	M3	W12X19	.399	15	.039	15	y	126.716	166.766	7.435	19.335	1.195	H1-1b
4	1	M4	W12X19	.398	15	.038	15	y	126.716	166.766	7.435	19.347	1.196	H1-1b
5	1	M5	W16X36	.055	0	.033	0	y	113.659	317.365	26.946	159.681	1.867	H1-1b
6	1	M6	HSS6X6X3	.446	0	.085	0	y	78.155	119.313	19.597	19.597	2.067	H1-1b
7	1	M7	W12X19	.269	15	.052	15	y	17.443	166.766	7.435	32.664	2.019	H1-1b
8	1	M8	W12X19	.269	15	.052	15	y	17.443	166.766	7.435	32.455	2.006	H1-1b
9	2	M1	W12X19	.605	0	.058	0	y	126.716	166.766	7.435	18.756	1.159	H1-1b
10	2	M2	W12X19	.603	0	.056	0	y	126.716	166.766	7.435	18.77	1.16	H1-1b

The final results of the code checking are two values a **UC Max** and a **Shear UC** code check value.

The UC Max stands for maximum Unity Check. This would be a factored ratio of actual to allowable stress, or demand vs capacity. The Shear UC represents a similar ratio based on the shear provisions of chapter F. The location for the shear check is followed by "y" or "z" to indicate the direction of the shear.

So, if UC Checks are less than 1.0, the member passes. If either of them is greater than 1.0, the member fails. If the value is greater than 9.999 it will be listed as "9.999". The **Loc** field tells at what location the maximum code check occurs measured from the I-joint location of the member. See Plot Options – Members to learn how to view the code check results graphically.

The remaining columns, discussed below, provide some of the values used in the code check with the equation number itself given in the last column. The Member Detail Report gives more values used to perform the code check.

For ASD 13th Edition code checking, Pnc/Om, Pnt/Om, and Mnyy/Om and Mnzz/Om are the Design Capacities divided by the Omega safety factors. Whereas the LRFD code checks will display the capacities multiplied by the Phi factor, Phi*Pnc, Phi*Pnt, Phi*Mnyy, and Phi*Mnzz.

For Pnc is calculated according to the provisions of AISC 13th Edition, Chapter E. Pnt is based on Chapter D. The Mn values are calculated based on Chapter F. Note that for RISA-3D, "zz" corresponds to "xx" in the AISC code, i.e. RISA-3D substitutes Mnzz for Mnx, to maintain consistency with the member local axis system.

Note that LRFD code checking requires a P-Delta analysis to satisfy the requirements of Chapter C, so if P-Delta analysis is NOT turned on (via the P-Delta flag) code checks won't be performed.

The Cb coefficient is calculated based on the description presented in Chapter F, section F1. Refer to the section on Cb input for more information about this calculation.

The final field lists the controlling equation for the code check. This will be one of the equations from Chapter H (for ASD and LRFD) or section 7 for the HSS code.

Note that the requirements of Section H3 of the LRFD code are satisfied since RISA-3D calculates and includes torsional warping effects.

For enveloped results the combination that produced the listed code and shear checks is given in the column "lc". The other values are the corresponding values and are not necessarily the maximums across all the combinations.

Tapered Wide Flanges (ASD) - For Tapered WF shapes, the Cb value shown will be the Cb value that RISA-3D calculated internally, NOT the value that was entered and used for the "B" value. The values shown in the Cm fields will be the C'm values that were used for the Appendix F calculations. The controlling equation from Appendix F will be listed.

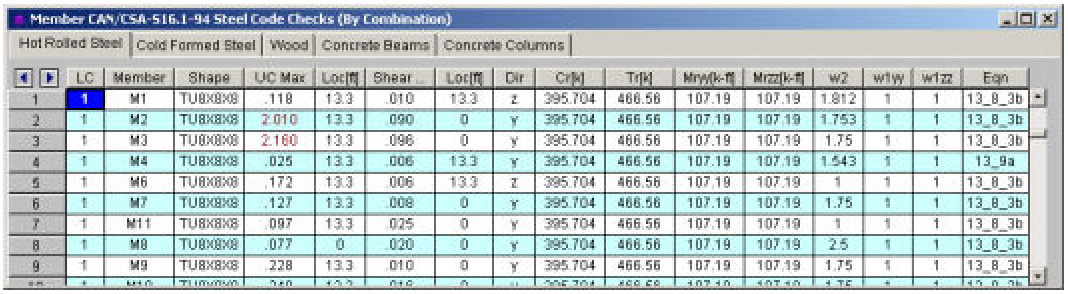
Tapered Wide Flanges (LRFD) - For Tapered WF shapes, the Cb value shown will be the Cb value that RISA-3D calculated, NOT the Cb value that was entered and used for the “B” value. The controlling equation from Chapter H will be listed.

Note

- The Member Detail Report gives more values used to perform the code check.
- See Spreadsheet Operations to learn how to use **Find**, **Sort** and other options.
- See Plot Options – Members to learn how to plot member results.
- See Limitations - ASD Code or Limitations - LRFD code for additional restrictions / limitations.

Canadian Code Check Results

Access the **Steel Code Check** spreadsheet by selecting the **Results** menu and then selecting **Members** ▸ **Steel Code Checks**.


Member CAN/CSA-S16.1-94 Steel Code Checks (By Combination)

Hot Rolled Steel | Cold Formed Steel | Wood | Concrete Beams | Concrete Columns

	LC	Member	Shape	UC Max	Loc[ft]	Shear ...	Loc[ft]	Dir	Cr[k]	Tr[k]	Mryy[k-ft]	Mrzz[k-ft]	w2	w1yy	w1zz	Eqn
1	1	M1	TU8X8X8	.118	13.3	.010	13.3	z	395.704	466.56	107.19	107.19	1.812	1	1	13_8_3b
2	1	M2	TU8X8X8	2.010	13.3	.090	0	y	395.704	466.56	107.19	107.19	1.753	1	1	13_8_3b
3	1	M3	TU8X8X8	2.160	13.3	.096	0	y	395.704	466.56	107.19	107.19	1.75	1	1	13_8_3b
4	1	M4	TU8X8X8	.025	13.3	.006	13.3	y	395.704	466.56	107.19	107.19	1.543	1	1	13_9a
5	1	M6	TU8X8X8	.172	13.3	.006	13.3	z	395.704	466.56	107.19	107.19	1	1	1	13_8_3b
6	1	M7	TU8X8X8	.127	13.3	.008	0	y	395.704	466.56	107.19	107.19	1.75	1	1	13_8_3b
7	1	M11	TU8X8X8	.097	13.3	.025	0	y	395.704	466.56	107.19	107.19	1	1	1	13_8_3b
8	1	M8	TU8X8X8	.077	0	.020	0	y	395.704	466.56	107.19	107.19	2.5	1	1	13_8_3b
9	1	M9	TU8X8X8	.228	13.3	.010	0	y	395.704	466.56	107.19	107.19	1.75	1	1	13_8_3b

The steel design code checks are calculated based on the internal sections used to check the section. The number of internal sections is selected on the Solution tab of the **Global Parameters** dialog.

These are the results of the Canadian code checking for the members. These are referred to as "unity" checks because the final number represents a ratio of factored loads divided by the design resistance. Thus, any value greater than unity (1) means the member fails. If a unity check is greater than 9.999, it will be listed here as "9.999". The "Loc" field tells at what location the maximum unity check occurs measured from the I-joint location of the member (in the current length units).

The Shear Check is the maximum ratio of the factored shear force to the shear resistance. The location for the shear check is followed by "y" or "z" to indicate the direction of the governing shear check.

Cr, Mryy and Mrzz are the factored resistances calculated for the member. Cr is calculated according to the provisions of CAN/CSA S16.1-94, Section 13.3.1. The Mr values are calculated based on Sections 13.5 and 13.6. Note that for RISA-3D, "zz" corresponds to "xx" in the Canadian code, i.e. RISA-3D substitutes Mrzz for Mrx, to maintain consistency with the member local axis system. Note that Canadian code checking requires a P-Delta analysis to satisfy the requirements of Section 8.6, so if P-Delta analysis is NOT turned on (via the P-Delta flag) Canadian code checks won't be done.

The w2 coefficient is calculated based on the description presented in Section 13.6. Whether or not the member is subject to sidesway is determined from the setting for the strong axis sway flag on the **Member Design** spreadsheet. The w1 coefficients, described in Section 13.8.4 are also listed. These are also influenced by the sway flag settings. The final field lists the controlling equation for the unity check. This will be one of the equations from Section 13.8.1, 13.8.2, or 13.9.

Note that the requirements of Section 15.11 are satisfied since RISA-3D calculates and includes torsional warping effects.

For enveloped results the combination that produced the listed unity and shear checks is given in the column "lc". The other values listed are the values calculated for the controlling load combination. They are not necessarily the maximums across all the combinations.

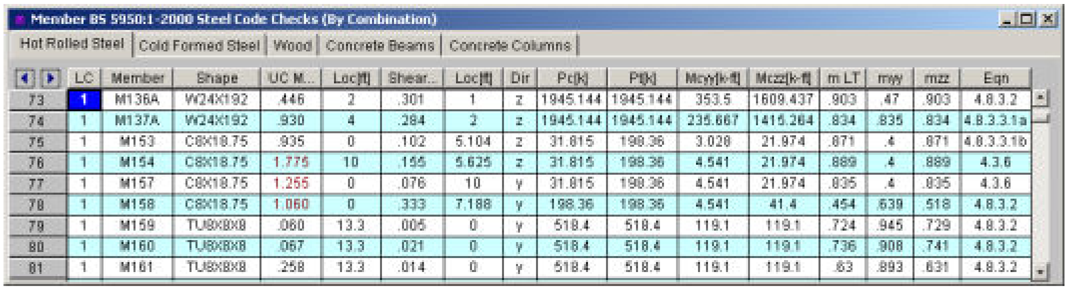
Tapered Wide Flanges - For Tapered WF shapes, the provisions of the AISC LRFD 2nd code are used. The w2 value shown will be the w2\Cb value that RISA-3D calculated, NOT the w2 value that was entered and used for the “B” value. The controlling equation from Appendix F in the LRFD code will be listed. The values shown under the headers for Cr and Mr will instead be the Pn and Mn values calculated for the LRFD code. These are the unfactored member strengths. (Similar to Cr and Mr, but without the “phi” reduction factor.)

Note

- The Member Detail Report gives more values used to perform the code check.
- See Spreadsheet Operations to learn how to use **Find**, **Sort** and other options.
- See Plot Options – Members to learn how to plot member results.
- See Limitations - Canadian Code for additional restrictions / limitations.

British Code Check Results

Access the **Steel Code Check** spreadsheet by selecting the **Results** menu and then selecting **Members** ‣ **Steel Code Checks**.


Member BS 5950:1-2000 Steel Code Checks (By Combination)

Hot Rolled Steel | Cold Formed Steel | Wood | Concrete Beams | Concrete Columns

	LC	Member	Shape	UC M...	Loc[ft]	Shear...	Loc[ft]	Dir	Pc[k]	Pt[k]	Mcyy[k-ft]	Mczz[k-ft]	m LT	myy	mzz	Eqn
73	1	M136A	W24X192	.446	2	.301	1	z	1945.144	1945.144	353.5	1609.437	.903	.47	.903	4.8.3.2
74	1	M137A	W24X192	.930	4	.284	2	z	1945.144	1945.144	235.667	1415.264	.834	.835	.834	4.8.3.3.1a
75	1	M153	C8X18.75	.935	0	.102	5.104	z	31.815	198.36	3.028	21.974	.871	.4	.871	4.8.3.3.1b
76	1	M154	C8X18.75	1.775	10	.155	5.625	z	31.815	198.36	4.541	21.974	.889	.4	.889	4.3.6
77	1	M157	C8X18.75	1.255	0	.076	10	y	31.815	198.36	4.541	21.974	.835	.4	.835	4.3.6
78	1	M158	C8X18.75	1.060	0	.333	7.188	y	198.36	198.36	4.541	41.4	.454	.639	.518	4.8.3.2
79	1	M159	TU8X8X8	.060	13.3	.005	0	y	518.4	518.4	119.1	119.1	.724	.945	.729	4.8.3.2
80	1	M160	TU8X8X8	.067	13.3	.021	0	y	518.4	518.4	119.1	119.1	.736	.908	.741	4.8.3.2
81	1	M161	TU8X8X8	.258	13.3	.014	0	y	518.4	518.4	119.1	119.1	.63	.893	.631	4.8.3.2

The steel design code checks are calculated based on the internal sections used to check the section. The number of internal sections is selected on the Solution tab of the **Global Parameters** dialog.

These are the results of the British code checking for the members based on BS5950-1:2000. WF, Channel, RHS, CHS, WT, LL, L, solid circular bar, and solid rectangular section types are supported. These are referred to as "unity" checks because the final number represents a ratio of factored loads divided by the design resistance. Thus, any value greater than unity (1) means the member fails. If a unity check is greater than 9.999, it will be listed here as "9.999". The "Loc" field tells at what location the maximum unity check occurs measured from the I-joint location of the member (in the current length units).

The shear capacity, Pv, is calculated per section 4.2.3 using the listed equations for the shear area of each section type. Shear buckling per section 4.4.5 is also considered. The Shear Check is the maximum ratio of the factored shear force to the shear resistance. The location for the shear check is followed by "y" or "z" to indicate the direction of the governing shear check.

The reduction in design strength, “py”, per table 9 in section 3.1.1 is automatically considered by the program. The reduction is based on the yield stress of the steel material used for the member and the maximum thickness of the cross section.

The compression capacity, Pc is calculated based on section 4.7. The effective lengths shown in table 22 are shown as “K” factors in the program (K*length is the effective length) and these can be approximated by the program based on the sway flags and the end release conditions of the member (the connection type is NOT considered.) The user can also specify the effective length by entering the appropriate K factor. For the case of a sway member having pinned ends, a value of 2.1 for K is used as table 22 does not address this condition. The program considers all H sections as I sections for purposes of picking the proper strut curve in table 23 in section 4.7.5. For the same table and section, the program does not do interpolation for shapes that have maximum thicknesses between 40 and 50mm, per footnote 1.

The tension capacity, Pt, is calculated based on section 4.6 and currently done using the gross section.

Width to thickness checks are done for all shapes based on Tables 11 and 12 in section 3.5. Complete Effective section properties for the Area, Plastic Modulus, and Section Modulus, are done per section 3.5.6 and 3.6.

The moment capacity, Mc, is calculated per section 4.2.5, including consideration for the amount of shear at the section. The limit states of lateral torsional buckling, per section 4.3.6 and web buckling per section 4.4.4.2 are also considered.

The equivalent uniform moment factors (my, mz, mLT) used in section 4.8.3 and shown in table 26 of section 4 are calculated automatically by the program based on the sway flag and member moment diagram. The user can also enter their own values.

Combined axial and bending stresses are computed based on the equation shown in section 4.8, with the governing equation reported. Combined bending and warping is calculated per the AISC specification and design guide.

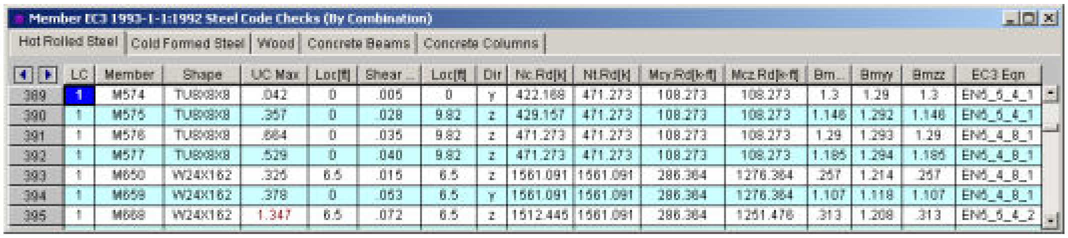
Biaxial bending and biaxial shears on Pipes and solid circular bars are done using the square root sum of the squares of the forces applied in or about the local y and local z directions.

Note

- The Member Detail Report gives more values used to perform the code check.
- See Spreadsheet Operations to learn how to use **Find**, **Sort** and other options.
- See Plot Options – Members to learn how to plot member results.
- See Limitations - British Code for additional restrictions / limitations.

EuroCode Code Check Results

Access the **Steel Code Check** spreadsheet by selecting the **Results** menu and then selecting **Members** ▸ **Steel Code Checks**.



	LC	Member	Shape	UC Max	Loc[ft]	Shear ...	Loc[ft]	Dir	Nc.Rd[k]	Nt.Rd[k]	Mcy.Rd[k-ft]	Mcz.Rd[k-ft]	Bm...	Bmyy	Bmzz	EC3 Eqn
389	1	M574	TU8X8X8	.042	0	.005	0	y	422.168	471.273	108.273	108.273	1.3	1.29	1.3	EN5_5_4_1
390	1	M575	TU8X8X8	.357	0	.028	9.82	z	429.157	471.273	108.273	108.273	1.146	1.292	1.146	EN5_5_4_1
391	1	M576	TU8X8X8	.664	0	.035	9.82	z	471.273	471.273	108.273	108.273	1.29	1.293	1.29	EN5_4_8_1
392	1	M577	TU8X8X8	.529	0	.040	9.82	z	471.273	471.273	108.273	108.273	1.185	1.294	1.185	EN5_4_8_1
393	1	M650	W24X162	.326	6.5	.015	6.5	z	1561.091	1561.091	286.364	1276.364	.257	1.214	.257	EN5_4_8_1
394	1	M659	W24X162	.378	0	.053	6.5	y	1561.091	1561.091	286.364	1276.364	1.107	1.118	1.107	EN5_4_8_1
395	1	M668	W24X162	1.347	6.5	.072	6.5	z	1512.445	1561.091	286.364	1251.476	.313	1.208	.313	EN5_5_4_2

The steel design code checks are calculated based on the internal sections used to check the section. The number of internal sections is selected on the Solution tab of the **Global Parameters** dialog.

These are the results of the Euro code checking for the members based on ENV 1993-1-1:1992. WF, Channel, RHS, CHS, WT, LL, L, solid circular bar, and solid rectangular section types are supported. These are referred to as "unity" checks because the final number represents a ratio of factored loads divided by the design resistance. Thus, any value greater than unity (1) means the member fails. If a unity check is greater than 9.999, it will be listed here as "9.999". The "Loc" field tells at what location the maximum unity check occurs measured from the I-joint location of the member (in the current length units).

Important Note: The bending axes in the member local coordinate system are reversed between RISA and the Eurocode. All input and results are shown in the member local coordinate system used by RISA. For a wide flange shape, the "z-z" axis is parallel to the flanges and the "y-y" axis is parallel to the web.

The reduction in yield strength, "fy", per table 3.1 in section 3.2.2.1 is automatically considered by the program. The reduction is based only on the maximum thickness of the cross section. Yield strengths for sections with a maximum thickness of 40mm or less are not reduced. Yield strengths for sections with a maximum thickness greater than 40mm are reduced by 20 N/mm2

The compression capacity, Nc.Rd is calculated based on section 5.4.4 and 5.5.1. The effective lengths shown in tables E.2.1 and E.2.2 are shown as "K" factors in the program (K*length is the effective length.) These can be approximated by the program based on the sway flags and the end release conditions of the member (the connection type is NOT considered.) The approximate K factor values for reasonable "pinned" and "fixed" conditions are taken from the British BS 5950-1:2000 code. The user can also specify the effective length by entering the appropriate K factor. For the case of a sway member having pinned ends, a value of 2.1 for K is used as a reasonable limit.

The tension capacity, Nt.Rd, is calculated based on section 4.3.4 and currently done using the gross section.

Width to thickness checks are done for all shapes based on Table 5.3.1 in section 5.3. Complete Effective section properties for the Area, Plastic Modulus, and Section Modulus, are done per section 5.3.5.

The shear capacity, Vpl.Rd, is calculated per section 5.4.6 using the listed equations for the shear area of each section type. Shear buckling per section 5.6.3 is also considered.

The moment capacity, Mc.Rd, is calculated per section 5.4.5, including consideration for the high shear per section 5.4.7.

The equivalent uniform moment factors (βmy, βmy, βmLT) used in section 5.5.4 and shown in figure 5.5.3 of section 5.5.4 are calculated automatically by the program based on the sway flag and member moment diagram. The user can also enter their own values.

Combined axial and moment is checked for the limit state of stress and high shear per the equations shown in sections 5.4.8, The limit states of flexural buckling and lateral torsional buckling are checked per section 5.5.2, and web buckling per section 5.6.7.2 is also considered. The governing section is reported. Combined bending and warping is calculated per the AISC specification and increases the moment about the y-y axis.

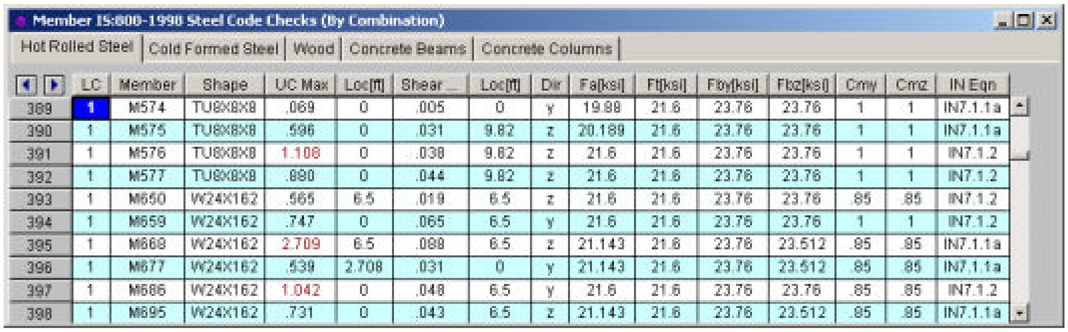
Biaxial bending and biaxial shears on Pipes and solid circular bars are done using the square root sum of the squares of the forces applied in or about the local axes.

Note

- The Member Detail Report gives more values used to perform the code check.
- See Spreadsheet Operations to learn how to use **Find**, **Sort** and other options.
- See Plot Options – Members to learn how to plot member results.
- See Limitations - EuroCode for additional restrictions / limitations.

Indian Code Check Results

Access the **Steel Code Check** spreadsheet by selecting the **Results** menu and then selecting **Members** ‣ **Steel Code Checks**.



	LC	Member	Shape	UC Max	Loc[ft]	Shear ...	Loc[ft]	Dir	Fa[ksi]	Ft[ksi]	Fby[ksi]	Fbz[ksi]	Cmy	Cmz	IN Eqn
389	1	M574	TU8X8X8	.069	0	.005	0	y	19.88	21.6	23.76	23.76	1	1	IN7.1.1a
390	1	M575	TU8X8X8	.596	0	.031	9.82	z	20.189	21.6	23.76	23.76	1	1	IN7.1.1a
391	1	M576	TU8X8X8	1.108	0	.038	9.82	z	21.6	21.6	23.76	23.76	1	1	IN7.1.2
392	1	M577	TU8X8X8	.880	0	.044	9.82	z	21.6	21.6	23.76	23.76	1	1	IN7.1.2
393	1	M650	W24X162	.565	6.5	.019	6.5	z	21.6	21.6	23.76	23.76	.85	.85	IN7.1.2
394	1	M659	W24X162	.747	0	.065	6.5	y	21.6	21.6	23.76	23.76	1	1	IN7.1.2
395	1	M668	W24X162	2.709	6.5	.088	6.5	z	21.143	21.6	23.76	23.512	.85	.85	IN7.1.1a
396	1	M677	W24X162	.539	2.708	.031	0	y	21.143	21.6	23.76	23.512	.85	.85	IN7.1.1a
397	1	M686	W24X162	1.042	0	.048	6.5	y	21.6	21.6	23.76	23.76	.85	.85	IN7.1.2
398	1	M695	W24X162	.731	0	.043	6.5	z	21.143	21.6	23.76	23.512	.85	.85	IN7.1.1a

The steel design code checks are calculated based on the internal sections used to check the section. The number of internal sections is selected on the Solution tab of the **Global Parameters** dialog.

These are the results of the Indian code checking for the members based on IS:800 – 1984. This code was reaffirmed in 1998 and reprinted in 2001. Wide Flange, Channel, Round Hollow Sections, Circular Hollow Sections, WT, LL, L, solid circular bar, and solid rectangular section types are supported. These are referred to as "unity" checks because the final number represents a ratio of factored loads divided by the design resistance. Thus, any value greater than unity (1) means the member fails. If a unity check is greater than 9.999, it will be listed here as "9.999". The "Loc" field tells at what location the maximum unity check occurs measured from the I-joint location of the member (in the current length units).

The allowable compressive stress, Fa, is calculated based on section 5.1. The effective lengths shown in table 5.2 are shown as “K” factors in the program (K*length is the effective length) and these can be approximated by the program based on the sway flags and the end release conditions of the member (the connection type is NOT considered.) The user can also specify the effective length by entering the appropriate K factor. For the case of a sway member having pinned ends, a value of 2.1 for K is used as table 5.2 does not address this condition.

The allowable tensile stress, Ft, is calculated based on section 4.1 and is currently done using the gross section.

The maximum slenderness ratios shown in table 3.1 are reported in the warning log when they are exceeded. The program still calculates capacities for the members and reports a code check.

Width to thickness checks are done for all shapes based on section 3.5.2.1. Complete effective section properties for the Area and Section Modulus are calculated.

The allowable shear stress, Fv, is used per section 6.4.1. This value is checked per the maximum shear stress in the section based on elastic theory.

The strong axis allowable bending stress, Fb, is calculated per section 6.2. The limit state of lateral torsional buckling, per sections 6.2.4 and 6.2.6, is considered. The allowable bending stress is increased per section 6.2.4.1 as appropriate. The weak axis bending stress is assigned per section 6.2.5.

The equivalent uniform moment factor, Cm, used in section 7.1 are calculated automatically by the program based on the sway flag per section 7.1.3. The user can also enter their own values.

Combined axial and bending stresses are computed based on the equations shown in section 7.1,with the governing equation reported. Combined bending and warping is calculated per the AISC ASD specification.

Biaxial bending and biaxial shears on Pipes and solid circular bars are done using the square root sum of the squares of the forces applied in or about the local y and local z directions.

Note

- The Member Detail Report gives more values used to perform the code check.
- See Spreadsheet Operations to learn how to use **Find**, **Sort** and other options.
- See Plot Options – Members to learn how to plot member results.
- See Limitations - Indian Code for additional restrictions / limitations.

Joints

Joints are used to define the ends of members and plate corners. Joints are also used to specify boundary conditions, diaphragms, story drift locations, and joint loads. Each joint is a point in space defined by coordinates in the global X, Y, and Z directions and temperature that may be used in conjunction with thermal loads.

Note

- The terms "node" and "joint" are interchangeable and are both used in this manual and in the program.

Joints may be input manually, or they may be created automatically as you draw new members and plates on the drawing grid. Once defined, the joints may be edited in three ways: graphically, in the **Joint Information Dialog**, or in the **Joint Coordinates Spreadsheet**.

To Define Joints

- Select the **Joint** Coordinates **Spreadsheet** from the **Spreadsheets Menu** and define the joint coordinates and temperature.

Note

- You may use cut and paste, block fill, and block math to enter and edit joints.
- You may choose the prefix that is used to label the joints.
- To modify one joint you may double click that joint to view and edit its properties.

To Relabel Joints

- After sorting the **Joints Spreadsheet** into the desired order select the **Tools Menu** and choose **Relabel Joints**.

Rounding Off Joint Coordinates

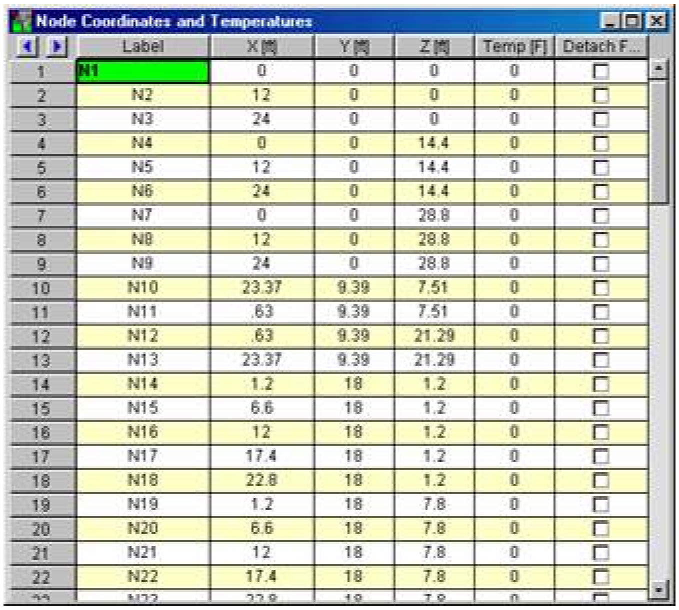
- Select the **Tools Menu** and choose **Round Off Joint Coordinates**.

Note

- This utility will round off all the joint coordinates to 1, 2, or 3 decimal places (user selected). This sometimes is useful for models that have been generated or created via DXF import and have joint coordinates with a large number of decimal places that aren't significant. Rounding off those coordinates makes the data more consistent and can help avoid problems such as non-coplanar plate joints. Also, the default member orientations can change based on whether the member is exactly parallel to a global axis and slight unintended coordinate differences can have an impact here as well.

Joint Coordinates Spreadsheet

The **Joint Coordinates Spreadsheet** records the joint coordinates and ambient temperatures for the joints and may be accessed by selecting **Joint Coordinates** on the **Spreadsheets Menu**.



	Label	X [ft]	Y [ft]	Z [ft]	Temp [F]	Detach F...
1	N1	0	0	0	0	☐
2	N2	12	0	0	0	☐
3	N3	24	0	0	0	☐
4	N4	0	0	14.4	0	☐
5	N5	12	0	14.4	0	☐
6	N6	24	0	14.4	0	☐
7	N7	0	0	28.8	0	☐
8	N8	12	0	28.8	0	☐
9	N9	24	0	28.8	0	☐
10	N10	23.37	9.39	7.51	0	☐
11	N11	.63	9.39	7.51	0	☐
12	N12	.63	9.39	21.29	0	☐
13	N13	23.37	9.39	21.29	0	☐
14	N14	1.2	18	1.2	0	☐
15	N15	6.6	18	1.2	0	☐
16	N16	12	18	1.2	0	☐
17	N17	17.4	18	1.2	0	☐
18	N18	22.8	18	1.2	0	☐
19	N19	1.2	18	7.8	0	☐
20	N20	6.6	18	7.8	0	☐
21	N21	12	18	7.8	0	☐
22	N22	17.4	18	7.8	0	☐

The first column is used to assign a unique **Label** to every joint. You can then refer to the joint by its label. Each label has to be unique, so if you try to enter the same label more than once you will receive an error message. As you create new lines, the program will automatically create a new, unique label for each joint.

The next three columns contain the coordinates of the joint in each of the global directions. These represent the relative offsets of the joints from the origin of the coordinate system (0, 0, 0). The appropriate units are listed at the top of each column.

The next column is used to define the ambient, no-stress, temperature at the joint. Temperature loads are then calculated based on the differential between the ambient temperature interpolated across the member, and the applied thermal load. See "Loads - Thermal Loads" for more information.

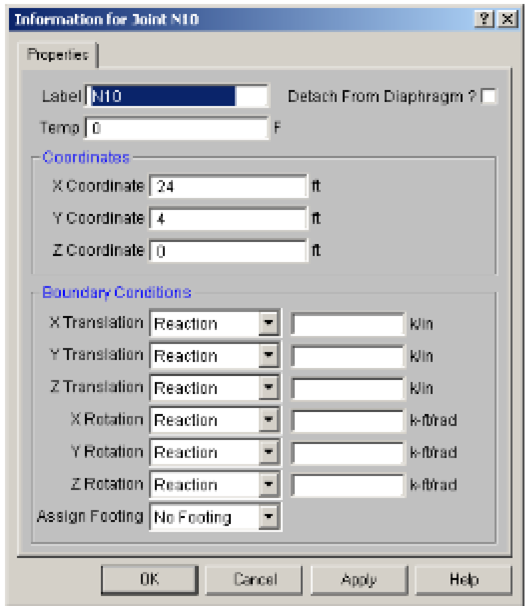
The last column is used to detach joints from a diaphragm. See Partial Diaphragms to learn more about this.

Two options on the **Tools Menu** help you work with joints. Selecting **Relabel Joints** will let you define a prefix to be used for the joint labels before creating a new label for each joint by using this prefix with a sequential number. For example, if you were to enter a prefix of "FLR", the first joint would get label FLR1, the second one would get FLR2, etc.

The second option is **Round Off Joint Coordinates**. Selecting this will round off all the joint coordinates to 1, 2, or 3 decimal places. This is useful for models that have been created using generation functions or DXF files where a high number of decimal places is present for some of the coordinates.

Joint Information Dialog

Just as with the members and plates you may double-click any joint to view it's properties. All of the same information that is stored in the **Joints** spreadsheet is displayed for the joint you choose, and may be edited. This is a quick way to view and change joint properties. For large selections of joints however the spreadsheet and graphic editing tools may be the faster solution.



Label - You can view and edit the joint's label.

Temperature - Allows you to set the ambient temperature at that node for Thermal Loading.

Coordinates - You can view and edit the joint coordinates.

Boundary Conditions - Here you can view and edit the joint boundary conditions. The edit boxes to the right of the boundary condition list boxes are where you would enter additional data for a boundary condition such as a spring stiffness, a Story number, or a master joint label for a Slaved joint.

Assign Footing - For use with the RISAFoot add-on, this allows you to define an individual spread footing at this joint.

Note

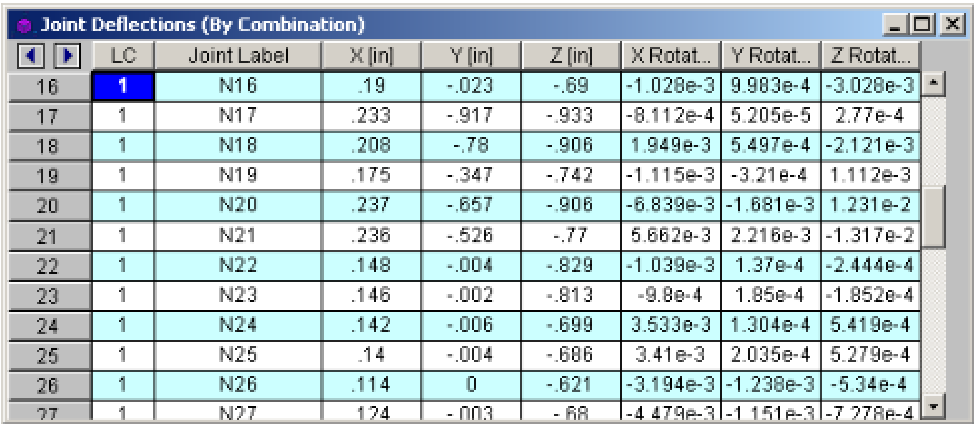
- It's generally more efficient to use the Graphic Editing features if you want to change the properties for many joints at once.

Joints - Results

When the model is solved, there are several groups of results specifically for the joints. Story Drift calculation results are discussed in Drift Results, the others are discussed here.

Joint Deflections Results

Access the **Joint Deflections Spreadsheet** by selecting it from the **Results Menu**.



Joint Deflections (By Combination)

	LC	Joint Label	X [in]	Y [in]	Z [in]	X Rotat...	Y Rotat...	Z Rotat...
16	1	N16	.19	-.023	-.69	-1.028e-3	9.983e-4	-3.028e-3
17	1	N17	.233	-.917	-.933	-8.112e-4	5.205e-5	2.77e-4
18	1	N18	.208	-.78	-.906	1.949e-3	5.497e-4	-2.121e-3
19	1	N19	.175	-.347	-.742	-1.115e-3	-3.21e-4	1.112e-3
20	1	N20	.237	-.657	-.906	-6.839e-3	-1.681e-3	1.231e-2
21	1	N21	.236	-.526	-.77	5.662e-3	2.216e-3	-1.317e-2
22	1	N22	.148	-.004	-.829	-1.039e-3	1.37e-4	-2.444e-4
23	1	N23	.146	-.002	-.813	-9.8e-4	1.85e-4	-1.852e-4
24	1	N24	.142	-.006	-.699	3.533e-3	1.304e-4	5.419e-4
25	1	N25	.14	-.004	-.686	3.41e-3	2.035e-4	5.279e-4
26	1	N26	.114	0	-.621	-3.194e-3	-1.238e-3	-5.34e-4
27	1	N27	.124	-.003	-.68	-4.479e-3	-1.151e-3	-7.278e-4

These are the deflections, or displacements, for every joint in the structure. Each joint has 6 values calculated, 1 for each of the 6 global degrees of freedom. Those being 3 translations and 3 rotations. The units for the displacements are shown at the top of each column. The rotations are shown in units of radians (1 radian = 57.3 degrees, approximately).

For enveloped results the maximum and minimum value for each displacement is listed. The load combination producing the maximum or minimum is also listed, in the "lc" column. To include a particular Load Combination in the envelope analysis, open the **Load Combinations Spreadsheet** and check the box in the **Solve** column.

Note

- See Spreadsheet Operations to learn how to use **Find**, **Sort**, and other options.
- See Plot Options – Joints to learn how to plot joint results. You will NOT be able to plot the deflected shape for an envelope analysis. This is because the various maximum and minimum displacements probably correspond to different load combinations, so a deflected shape based on these values would be meaningless.
- You can see the deflection results for a joint in any model view by moving your mouse near the joint and then reading the deflection results from the right hand section of the status bar. The joint displacement reported will coincide with the joint nearest to the tip of the mouse pointer.

Joint Reaction Results

Access the **Joint Reactions Spreadsheet** by selecting it from the **Results Menu**.

Joint Reactions (By Combination)

Regular | Overstrength

	L...	Joint Label	X [k]	Y [k]	Z [k]	MX [k-ft]	MY [k-ft]	MZ [k-ft]
1	1	N1	0	-.315	NC	NC	NC	0
2	1	N3	0	3.571	NC	NC	NC	0
3	1	N5	0	2.32	NC	NC	NC	0
4	1	Totals:	0	5.576	0			
5	1	COG (ft):	X: 15.557	Y: 7.864	Z: 0			
6	2	N1	0	.932	NC	NC	NC	0
7	2	N3	0	5.534	NC	NC	NC	0
8	2	N5	0	2.109	NC	NC	NC	0
9	2	Totals:	0	8.576	0			
10	2	COG (ft):	X: 11.864	Y: 7.912	Z: 0			
11	3	N1	-1.2	-3.925	NC	NC	NC	0
12	3	N3	-1.075	6.39	NC	NC	NC	0
13	3	N5	.375	2.111	NC	NC	NC	0
14	3	Totals:	-1.9	4.576	0			
15	3	COG (ft):	X: 22.236	Y: 7.834	Z: 0			

These are the reactive forces **applied to the structure** at its points of support. A positive reaction is applied in the direction of the global axis and a negative reaction is applied in the direction opposite the global axis. A positive moment is given according to the right hand rule assuming the thumb is pointing in the positive global axis direction. Assuming a reaction has been calculated at all points of support, the total of the reactive forces in each direction should equal the total applied force in each direction. The units for the reactions are listed at the top of each column, and a total reaction for each direction is summed at the bottom of each translation column.

The last line provides the center of gravity (COG) for the applied loads. This "COG" is based on the load components acting in the VERTICAL direction. If there are no vertical loads in the combination a "COG" will not be calculated.

For enveloped results the maximum and minimum reaction value is listed. The load combination producing the maximum or minimum is also listed, in the "lc" column.

Overstrength Tab

Joint Reactions - Overstrength (By Combination)

Regular | Overstrength

	L...	Joint Label	X [k]	Y [k]	Z [k]	MX [k-ft]	MY [k-ft]	MZ [k-ft]
1	4	N1	0	0	NC	NC	NC	0
2	4	N3	0	0	NC	NC	NC	0
3	4	N5	0	0	NC	NC	NC	0
4	4	Totals:	0	0	0			
5	4	COG (ft):	NC	NC	NC			
6	5	N1	0	0	NC	NC	NC	0
7	5	N3	0	0	NC	NC	NC	0
8	5	N5	0	0	NC	NC	NC	0
9	5	Totals:	0	0	0			
10	5	COG (ft):	NC	NC	NC			

If your solution included a load combination that included overstrength load factors, a second tab will be included in the Joint Reactions spreadsheet. This tab contains all the reactive forces per the overstrength load combinations.

Note

- See Spreadsheet Operations to learn how to use **Find**, **Sort**, and other options.

- See Plot Options – Joints to learn how to plot joint results.
- An 'NC' listing means "No Calculation". This occurs for boundary conditions defined with the 'Fixed' option and joints with an enforced displacement in that degree of freedom.

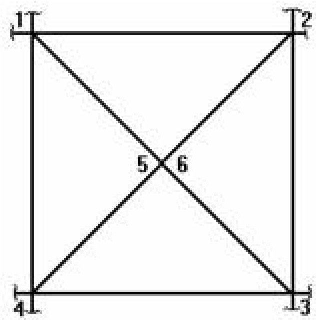
Joints - Slaving Joints

When a joint is "slaved", it is linked in one or more directions to other joints (the "master" joints). The slave and master joints actually share the same degree of freedom for the direction of slaving. The slaving can be for any or all of the six global degrees of freedom. A joint can be slaved to more than one master (in different directions). Any number of joints may be slaved to the same master, but the master joint itself may not be slaved to another master.

Note

- Since the advent of rigid links there are not many circumstances where nodal slaving is the best solution for modeling.

An example of the use of joint slaving is in X bracing, where the bracing members overlay and are pinned to each other. Modeling this connection makes the analysis far more complex than simply modeling the braces as separate pieces, and is ***generally not recommended***. However, if it is desired to specifically account for interaction between the braces then this pinning will force the brace midpoints to deflect together for translations, but still leave them free to rotate independently.



In this diagram the braces are NOT modeled as physical members. Brace 1 to 3 is modeled with non physical members from joints 1 to 5 and 5 to 3. Brace 2 to 4 is modeled with non physical members from joints 2 to 6 and 6 to 4. Joints 5 and 6 are at the same location where the braces overlap and are pinned together.

We would slave joint 6 to joint 5 for the translation directions. To define a slave joint, use the **Boundary Conditions** spreadsheet. Enter the label of the joint to be slaved as the "Joint No.", and for the directions in which it is slaved, enter **SLAVE nnn**, where **nnn** is the label of the master joint.

For this X brace example, we would enter the following on the **Boundary Conditions** spreadsheet:

Joint Boundary Conditions

	Joint Label	X [k/in]	Y [k/in]	Z [k/in]	X Rot.[k-ft/r...	Y Rot.[k-ft/r...	Z Rot.[k-ft/r...
1	6	Slave5	Slave5	Slave5			

Here we have "SLAVE 5" entered for the X, Y and Z translations. The rotation fields areleft blank because joint 6 is free to rotate independently. We don't have to enter anything for joint 5.

Note

- Slave joints should NOT be used to build rigid diaphragms. Slaving the translations will not give correct diaphragm behavior. If you need a rigid diaphragm, you should use the Rigid Diaphragm feature.

Loads

As you input loads they are grouped into separate sets of Basic Load Cases. These are the basic building blocks of the loads applied to the structure, consisting of discrete loads and self-weight of the structure. The basic load cases may also be grouped into load categories such as dead load and live load. In the end, the model will be solved against load combinations that are built with the basic load cases and categories.

You may view the loads on your model. This is an excellent way to verify the loads that are on the model. You may plot them by load case, load category or load combination. See Plot Options – Loads to learn how to do this.

Each of the discrete load types is described in it's own section:

- Area Loads
- Distributed Loads
- Joint Loads/Forced Displacements
- Moving Loads
- Point Loads
- Surface Loads
- Thermal Loads

Self Weight (Gravity Load)

The structure's self-weight may be automatically calculated and included in a solution. The inclusion of self-weight may be specified within a Basic Load Case or as part of a Load Combination.

The self-weight is calculated as a full-length uniform load across each member of the model and as a surface load on the plates. The magnitude of the load is the area times the material weight density.

Note

- If a member has offsets defined, the offset distances are not included in the self-weight calculation.

Adding Self Weight to a Basic Load Case

- Select the **Basic Load Case** spreadsheet from the **Spreadsheets** menu. On the line of the load case you wish to contain self-weight, enter a load factor in the appropriate direction column. For example, if you wish to apply 90% of the self-weight in the negative Y direction (down), you would put -0.9 as the factor in the **Y Gravity** column.

Adding Gravity Load to a Load Combination

- To include the self-weight in a Load Combination, put "X", "Y" or "Z" in the BLC field. The letter used indicates which global direction the self-weight is applied in. Also put a factor in the Factor field. For example, if you wish to apply 90% of the self-weight in the negative Y direction (down), you would put "Y" as the BLC and "-0.9" as the BLC factor.

Note

- You may have multiple self-weight definitions, i.e. you could have self-weight applied in the "Y" and "X" directions in the same or different load cases if you wished.
- Don't inadvertently include the self-weight of the structure twice by specifying it within a combination that includes a Basic Load Case that also contains the self-weight.

Drawing Loads

You may graphically apply loads to the model. You may apply one load at a time by clicking the mouse, or apply loads to entire selections at once. All of the graphical loading tools may be found on the **Drawing Toolbar.**

Modifying Loads

Modifying loads may be done within the spreadsheets. Select **Loads** from the **Spreadsheets** menu and then choose the spreadsheet that you wish to modify. You may then move through different Basic Load Cases by using the drop-down list on the **Window Toolbar.**

See Spreadsheet Operations to learn how you can modify the loads. You may also copy or delete entire Basic Load Cases from the **Basic Load Case** spreadsheet. See Copying Basic Load Cases and Deleting Basic Load Cases for more information.

Deleting Loads

Deleting loads may be done within the spreadsheets. Select **Loads** from the **Spreadsheets** menu and then choose the spreadsheet that you wish to modify. You may then move through different Basic Load Cases by using the drop-down list on the **Window Toolbar.**

You may use spreadsheet operations such to help modify the loads. You may also delete entire Basic Load Cases from the **Basic Load Case** spreadsheet. See Deleting Basic Load Cases.

Loads - Basic Load Cases

When loads are defined they are grouped into basic load cases (BLC's). You are allowed up to 1000 separate Basic Load Cases. These are the basic building blocks of the final load combinations applied to the structure. The basic load cases may be assigned to load categories such as dead load and live load. These basic cases and categories are then combined to define load combinations used in analysis. A BLC can be comprised of any type of load, such as joint loads, distributed loads, member point loads, etc.

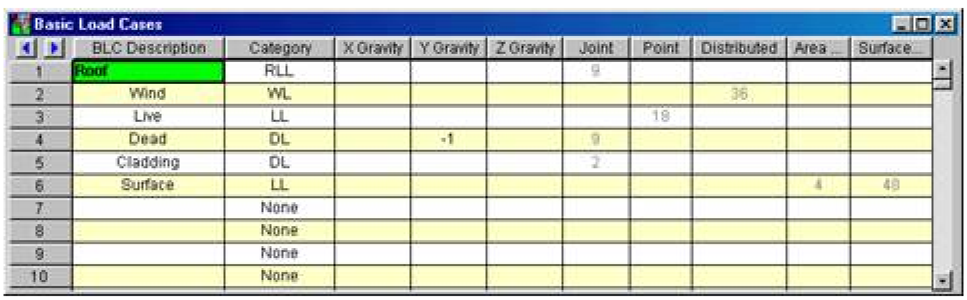
Any time you are applying or viewing loads, graphically or in the spreadsheets, they are assigned to the current basic load case. This current basic load case is displayed in the **Window Toolbar** in a drop down list. To change to another BLC simply choose it from this list. If you wish to name the BLC's you may do this in the **Basic Load Case** spreadsheet.

To Add Loads to a Basic Load Case

- When adding loads graphically click on the toolbar button for the type of load you wish to add and specify the BLC along with the load definition.
- When adding loads with the spreadsheets, open the spreadsheet for the load type you want to add and select the BLC from the list on the Window Toolbar.

Basic Load Case Spreadsheet

The **Basic Load Cases** spreadsheet groups and summarizes the loads applied to the model and may be accessed by selecting **Basic Load Cases** on the **Spreadsheets** menu.


Basic Load Cases

	BLC Description	Category	X Gravity	Y Gravity	Z Gravity	Joint	Point	Distributed	Area ...	Surface...
1	Roof	RLL				9				
2	Wind	WL						36		
3	Live	LL					18			
4	Dead	DL		-1		9				
5	Cladding	DL				2				
6	Surface	LL							4	48
7		None								
8		None								
9		None								
10		None								

Information about Basic Load Cases (BLC's) is recorded on the **Basic Load Case Spreadsheet** and the loads themselves are recorded in the load spreadsheets for each load type.

You may enter descriptions for each BLC in the first column. These descriptions are primarily for your own use. The descriptions will print the descriptions as part of the input printout and can also display the description when plotting the loads for the BLC or choosing them from the list.

The second column is used to assign the BLC to a load category such as Dead Load or Live Load. Simply choose the category from the drop down list. You can then build load combinations for analysis by referring to the categories rather than list each basic load case.

The next three columns may be used to specify that the self-weight of the structure be included in a load case. Simply enter a factor in the column that represents the direction of the self-weight. Typically you will enter a value of "-1" in the **Y Gravity** column assuming that Y is the vertical axis.

The remaining columns display the quantity of each type of load that is contained in the BLC. You may not edit these values **but you may click on the quantities to open the spreadsheet and view the loads that it represents**. For example, the previous figure has 9 joint loads as part of BLC 1. Clicking on the number 9 with the mouse will open the Joint Loads spreadsheet and display these loads.

Copying Basic Load Cases

You may copy the loads from one BLC into another BLC. This can be useful when one load case is similar to another and can be entered quickly by copying a load case and then making changes to the copy with features such as block math.

To do this, open the **Basic Load Case** spreadsheet and then click . Specify what case to copy loads from and which case the loads are to be copied into. You may further specify which types of loads are to be copied. For example, if you check **Joint Loads** and uncheck all the other load types, only the joint loads will be copied.

Any loads copied into a BLC will be added to any loads that may already be in that BLC.

Deleting Basic Load Cases

You may automatically clear all the loads in a BLC. To do this, open the **Basic Load Case** spreadsheet and then click the **Delete BLC** button on the **Window** toolbar. Select the BLC you wish to delete.

All of the loads will be deleted including category information and any self-weight information. The BLC Description will remain in the **Description** field.

Load Categories

The basic load cases may be assigned to load categories, such as dead load and live load, which are commonly used in building codes. You may do this on the **Basic Load Cases** spreadsheet. These categories may then be combined to define load combinations used in analysis.

Categories are easy to use and are very helpful. They allow you to put your loads into groups that are commonly used in load combinations, while keeping them in separate load cases. This is especially helpful in large models where the loads might occupy many different load cases. They also allow you to define load combinations that are easily understood because they resemble the combinations as they appear in the building code.

Load Category	Description
DL	Dead Load
LL	Live Load
EL	Earthquake Load
WL	Wind Load
SL	Snow Load
RLL	Roof Live Load
LLS	Live Load Special (public assembly, garage, storage, etc.)
TL	Long Term Load (creep, shrinkage, settlement, thermal, etc.)
SLN	Snow Load Non-shedding
HL	Hydrostatic Load
FL	Fluid Pressure Load
RL	Rain Load
PL	Ponding Load
EPL	Earth Pressure Load
IL	Impact Load
OL1 - OL10	Other Load 1 - 10 (generic)
ELX, ELY, ELZ	Earthquake Load along global X-axis, Y-axis, Z-axis
WLX, WLY, WLZ	Wind Load along global X-axis, Y-axis, Z-axis
WL+X, WL+Y, WL+Z	Wind Load along positive global X-axis, Y-axis, Z-axis
WL-X, WL-Y, WL-Z	Wind Load along negative global X-axis, Y-axis, Z-axis
WLXP1, WLYP1, WLZP1	Partial Wind Load 1 along global X-axis, Y-axis, Z-axis

Load Category	Description
WLXP2, WLYP2, WLZP2	Partial Wind Load 2 along global X-axis, Y-axis, Z-axis
ELX+Z, ELX+Y	Eccentric Earthquake Load along global X-axis shifted along positive global Z-axis, Y-axis
ELX-Z, ELX-Y	Eccentric Earthquake Load along global X-axis shifted along negative global Z-axis, Y-axis
ELZ+X, ELZ+Y	Eccentric Earthquake Load along global Z-axis shifted along positive global X-axis, Y-axis
ELZ-X, ELZ-Y	Eccentric Earthquake Load along global Z-axis shifted along negative global X-axis, Y-axis
ELY+X, ELY+Z	Eccentric Earthquake Load along global Y-axis shifted along positive global X-axis, Z-axis
ELY-X, ELY-Z	Eccentric Earthquake Load along global Y-axis shifted along negative global X-axis, Z-axis
NL, NLX, NLY, NLZ	General notional load and along global X-axis, Y-axis, Z-axis
WLX+R, WLY+R, WLZ+R, WLX-R, WLY-R, WLZ-R,	Roof wind loads in the positive and negative X-axis, Y-axis, Z-axis

Using categories is optional. Remember though that if you define combinations of categories you must define these categories in the **Basic Load Case** spreadsheet. If you don't the combinations will have no loads.

Loads - Load Combinations

During solution the model is loaded with a combination of factored Load Categories and/or Basic Load Cases, both of which are defined on the **Basic Load Cases Spreadsheet**. These combinations, load factors, and other parameters are defined on the **Load Combinations Spreadsheet**. Most standard load combinations are included in the program. See Solution to learn how to solve load combinations.

To Add Load Combinations Manually

1. From the **Spreadsheets Menu** select **Load Combinations**.
2. Enter load combinations by pairing loads in the **BLC** fields with factors in the **Factor** fields.

To Add Auto-Generated Building Code Combinations

1. From the **Spreadsheets Menu** select **Load Combinations**.
2. Select the button from the **Window Toolbar**.
3. Select the Load Combination **Region** and **Code** from the drop down lists provided in the Gravity tab.
4. Select the desired **Roof Live Load** and **Notional Load** options and click **Generate**.
5. Click in the **Wind** tab, select the desired **Wind Load** options, and click **Generate**.
6. Click in the **Seismic** tab, select the desired **Seismic Load** options, and click **Generate**.
7. Modify the generated combinations and options in the spreadsheet as necessary.

Note

- The generated building code combinations are made up of Load Categories and Factors. Loads that are not assigned to these categories will not be included in the combinations upon solution.
- All generated combinations are added and marked for the envelope solution. You may remove combinations from the envelope after adding them.
- You may specify P-Delta options and SRSS combinations for each combination after you have added them.
- Verify the Wind/Seismic ASIF (allowable stress increase factor), the Footing ABIF (allowable bearing increase factor), and the Timber CD settings on the **Design** tab for combinations after you add them.
- You may save any preferred load combinations as the default by clicking the **Save As Defaults** button on the **Window Toolbar**.

To Solve Load Combinations

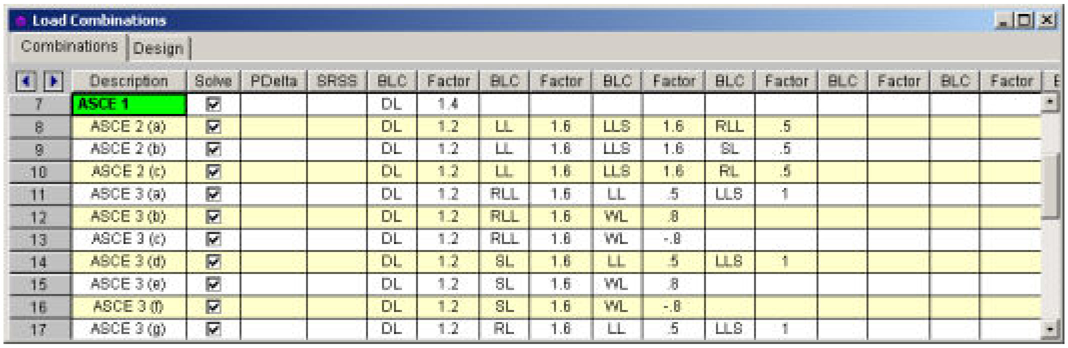
- To solve a single load combination, click the **Solve** button on the **RISA Toolbar**. Select the load combination from the drop down list and click the **Solve** button.
- To solve multiple load combinations, first mark the combinations to be solved by checking the **Solve** box on the **Load Combinations** spreadsheet Click the **Solve** button on the **RISA Toolbar**, and choose the **Envelope** or **Batch** option before clicking **Solve**.

Note

- The **Envelope** solution is where all combinations with a checkmark in the **Solve** field are solved simultaneously. The **Maximum and Minimum** results of these solutions are listed, along with the number of the controlling load combination for each solution result.
- The **Batch** solution is where all combinations with a checkmark in the **Solve** field are solved simultaneously. **All** results of these solutions are listed.

Load Combinations Spreadsheet

The **Load Combinations Spreadsheet** records the combinations of loads for solution and may be accessed by selecting **Load Combinations** on the **Spreadsheets Menu**.



	Description	Solve	PDelta	SRSS	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor
7	ASCE 1	☑			DL	1.4										
8	ASCE 2 (a)	☑			DL	1.2	LL	1.6	LLS	1.6	RLL	.5				
9	ASCE 2 (b)	☑			DL	1.2	LL	1.6	LLS	1.6	SL	.5				
10	ASCE 2 (c)	☑			DL	1.2	LL	1.6	LLS	1.6	RL	.5				
11	ASCE 3 (a)	☑			DL	1.2	RLL	1.6	LL	.5	LLS	1				
12	ASCE 3 (b)	☑			DL	1.2	RLL	1.6	WL	.8						
13	ASCE 3 (c)	☑			DL	1.2	RLL	1.6	WL	-.8						
14	ASCE 3 (d)	☑			DL	1.2	SL	1.6	LL	.5	LLS	1				
15	ASCE 3 (e)	☑			DL	1.2	SL	1.6	WL	.8						
16	ASCE 3 (f)	☑			DL	1.2	SL	1.6	WL	-.8						
17	ASCE 3 (g)	☑			DL	1.2	RL	1.6	LL	.5	LLS	1				

Combinations Tab

The first field, the **Description**, is strictly for the user's reference. Enter any descriptive label you wish and it may be displayed with the results when the combination is solved.

The next three fields are for options that apply to each load combination.

The **Solve** box is used to indicate which combinations should be included in batch or envelope solutions. See Solution for more information.

The **P-Delta** entry is used to enable an analysis of member secondary effects. See P-Delta Analysis for more information.

Note

- Per section 7.3.(1) of Appendix 7 (Direct Analysis Method) of the AISC 13th Edition steel code, a factor of 1.6 will be applied to all load combinations for which a P-Delta analysis is to be performed (and for which the Hot Rolled box is checked on the Design tab) whenever 'AISC 13th: ASD' is selected on the Codes tab of the Global Parameters Dialog. This factor is applied prior to conducting the P-Delta analysis and the results are subsequently divided by a factor of 1.6.

The **SRSS** entry is used to combine response spectra results by the Square Root of Sum of Squares. See SRSS Combination of RSA Results for more information.

The next eight pairs of fields (BLC, **Factor**) are for defining what loads are to be part of the combination, along with factors for each. Select load categories from the drop down lists in the **Category** columns of the spreadsheet. The following are the entries you can use in the **BLC** field:

Entry	**Description**
Number	Entering a number includes that particular Basic Load Case, I.e. enter “3" for BLC 3
Category	Enter a category code such as DL, LL, etc to include all loads in that category
Lnn	Enter “Lnn” to nest the loads from another combination, where “nn” is the number of the other combination, i.e. “L3" means include all loads from load combination 3.
Mnn	Use this entry to include a moving load where “Mnn” is the moving load tag from the moving load spreadsheet

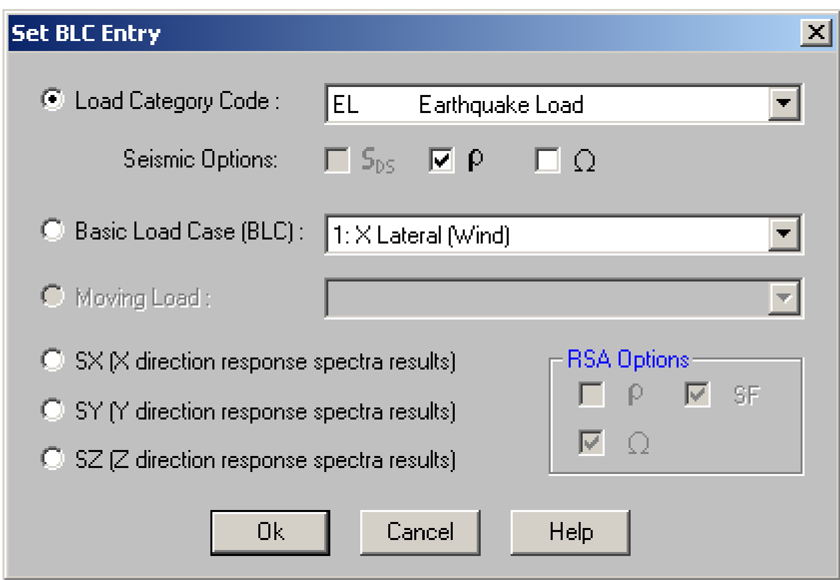
Sn	Includes response spectra results for the global direction “n”, I.e. enter “SX” to include response spectra results calculated in the global X direction, “SY” for global Y , etc.

There are also a number of BLC Prefixes that may be included in your load combinations. You can have these generated by the LC Generator or you may click on the red arrow button in the BLC cell to select them manually. These are defined in the Seismic tab of the Global Parameters. They are explained below:

Prefix	Description
SF	Scaling Factor - applied to RSA results
ρ	Redundancy factor - applied to EL loads for seismic load combinations (or RSA results) Note: If ELX and ELZ used, the ρ from the X and Z direction will be used directionally. If EL is used, the ρ from the X direction will be used and the Z direction will be disregarded. If ELY is used, the ρ from the Z direction will be used and the X direction will be disregarded.
SDS	Spectral Response Acceleration Parameter for Short Periods - applied to DL for seismic load combinations
Ω	Overstrength Factor - applied to EL loads for seismic load combinations (or RSA results) Note: If ELX and ELZ used, the Ω from the X and Z direction will be used directionally. If EL is used, the Ω from the X direction will be used and the Z direction will be disregarded. If ELY is used, the ρ from the Z direction will be used and the X direction will be disregarded.

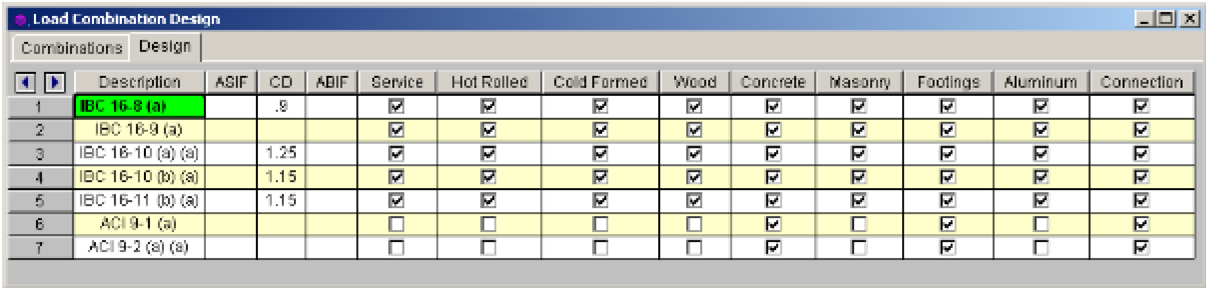
Enter in the **Factor** field a multiplier to be applied to the loads being included.

You may also use the button in the BLC cell to help you specify the loads. Choose from load categories, basic load cases, or spectral results or moving loads from the drop-down lists.



Notice the **Seismic Options** and **RSA Options** that are described in the Prefix table above. For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Seismic Factors**.

Design Tab



	Description	ASIF	CD	ABIF	Service	Hot Rolled	Cold Formed	Wood	Concrete	Masonry	Footings	Aluminum	Connection
1	IBC 16-8 (a)		.9		☑	☑	☑	☑	☑	☑	☑	☑	☑
2	IBC 16-9 (a)				☑	☑	☑	☑	☑	☑	☑	☑	☑
3	IBC 16-10 (a) (a)		1.25		☑	☑	☑	☑	☑	☑	☑	☑	☑
4	IBC 16-10 (b) (a)		1.15		☑	☑	☑	☑	☑	☑	☑	☑	☑
5	IBC 16-11 (b) (a)		1.15		☑	☑	☑	☑	☑	☑	☑	☑	☑
6	ACI 9-1 (a)				☐	☐	☐	☐	☑	☐	☑	☐	☑
7	ACI 9-2 (a) (a)				☐	☐	☐	☐	☑	☐	☑	☐	☑

The first field, the **Description**, is strictly for the user's reference. Enter any descriptive label you wish and it may be displayed with the results when the combination is solved.

The **ASIF** column is used to set the allowable stress increase factor used in the AISC: ASD code checking. This factor is only used for Hot Rolled Steel (see Allowable Stress Increase).

The **CD** factor is the load duration factor and is only necessary for NDS timber design (see Load Duration Factor).

The **ABIF** column is used to set the allowable bearing increase factor used for the soil bearing check in Footing design. This factor is only available when you have installed a current copy of the RISAFoot program.

The **Service** checkbox has two applications:

- **Cracked Concrete Section Properties**: Per the provisions of ACI 318-11 Sections 8.8 and R10.10.4.1 the cracked section properties of beams and columns may be multiplied by a factor of 1.43 for service level loads. All concrete members apply this factor when the Service box is checked.
- **RISAFoot Integration**: For models which use the integrated RISAFoot program, this checkbox is also used to determine which Load Combinations are used for the service level soil bearing and stability checks. Whereas, the Footing load combinations which do NOT have this box checked will be used for the concrete strength design of the footing and pedestal.

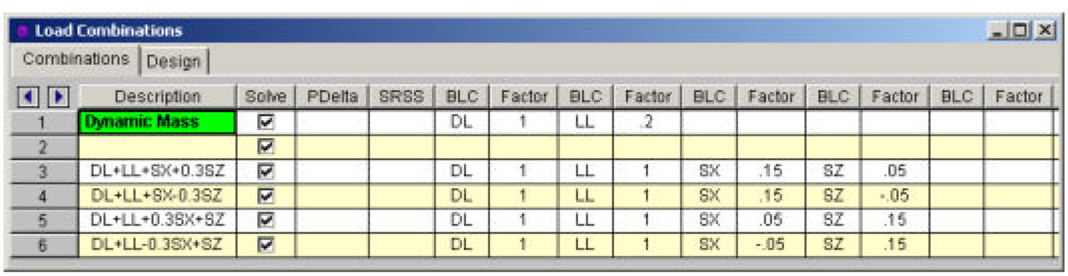
The next seven check boxes designate which load combinations should be used for the code checking of each material type. The example shown uses one set of Load Combinations for the NDS wood design, and another for the ACI concrete design. Member results (Forces, Stresses, Torsion) will only be reported for a members if it's material type has been checked for that load combination. Member results for general material will be shown for all load combinations. For example when using steel and wood in your model, you may design your steel for one set of load combinations and your wood for another.

The last checkbox for **Connection** defines:

- If this LC should be used in designing hot-rolled steel connections with RISAConnection.

Load Combinations with RSA Results

The results from response spectra analyses in the X, Y, and Z direction may also be included in the load combinations. Remember, when you perform a response spectra analysis (RSA), you specify in which global direction the spectrum is applied. RISA-3D will retain the three RSA solutions (one for each direction) simultaneously.



	Description	Solve	PDelta	SRSS	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor
1	Dynamic Mass	☑			DL	1	LL	.2						
2		☑												
3	DL+LL+SX+0.3SZ	☑			DL	1	LL	1	SX	.15	SZ	.05		
4	DL+LL+SX-0.3SZ	☑			DL	1	LL	1	SX	.15	SZ	-.05		
5	DL+LL+0.3SX+SZ	☑			DL	1	LL	1	SX	.05	SZ	.15		
6	DL+LL-0.3SX+SZ	☑			DL	1	LL	1	SX	-.05	SZ	.15		

To include the RSA results for a particular direction in a load combination, enter "**Sn**" in the BLC field, where n is the global direction. Suppose you wanted to include X direction RSA results, you would enter **SX** in the BLC field. You would enter **SY** for Y direction and SZ for Z direction RSA results. Also be sure to put the RSA Scaling Factor for the RSA results in the Factor field. You can have more than one RSA entry in a load combination.

Note

- Remember that RSA results are typically unsigned (all positive) and you should provide some means of accounting for this. The figure above uses two combinations for each RSA result, one with a positive factor and the other with a negative factor to capture the maximum deflections, stresses and forces. See Unsigned (All Positive) Results for more information.
- If you have to combine 2 or 3 different spectra results with many static load combinations, it is convenient to put all the spectra results (SX, SY, and SZ) and factors in one load combination and nest that combination in other combinations. You must set the RSA SRSS flag on each load combination for it to be performed.

SRSS Combination of RSA Results

This is used to cause orthogonal RSA results in the combination to be summed together using an SRSS (Square Root of Sum of Squares) summation. This gives a good approximation of MAXIMUM responses but it also causes all the RSA results to be positive.

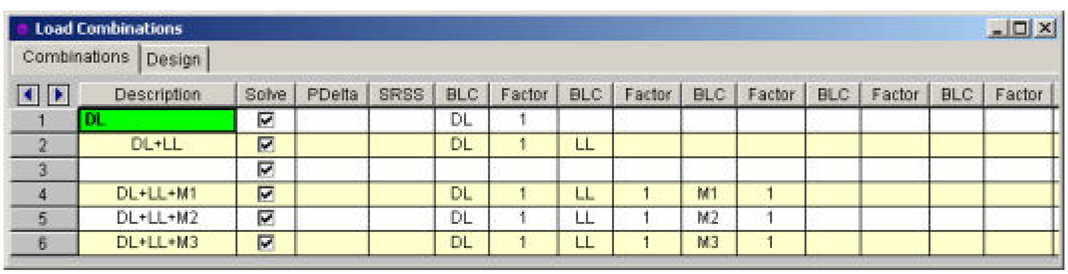
You may choose from the options by clicking on the button. The entry is “+” or “-” to indicate whether the combined RSA results are to be added (+) or subtracted (-) from the other loads in the combination.

Note

- This flag is used to combine different spectra that are acting in different directions. This is different than the modal combination method that combines results for one direction and is specified in the **Response Spectra** settings.

Load Combinations with Moving Loads

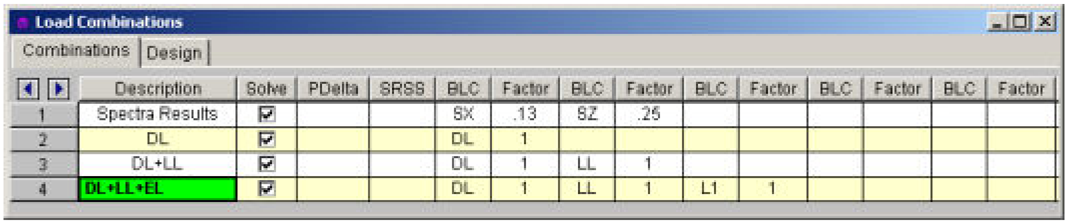
Moving loads are included in your analysis by referencing them on the Load Combinations spreadsheet. For example, to reference moving load number “**n**” you would enter “**Mn**” in one of the BLC fields, and then also enter the corresponding BLC factor. The moving load numbers are shown for each moving load on the **Moving Loads** spreadsheet.


Load Combinations
Combinations | Design

	Description	Solve	PDelta	SRSS	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor
1	DL	☑			DL	1								
2	DL+LL	☑			DL	1	LL							
3		☑												
4	DL+LL+M1	☑			DL	1	LL	1	M1	1				
5	DL+LL+M2	☑			DL	1	LL	1	M2	1				
6	DL+LL+M3	☑			DL	1	LL	1	M3	1				

Nesting Load Combinations

You are allowed only 8 BLC's per load combination, which may not be enough. For this reason you can define "combinations of load combinations". This means if you need more than 8 BLC entries in a single combination, you can define the needed BLC's and self-weights over several load combinations and then pull these combinations together into another load combination.



Load Combinations

Combinations | Design

	Description	Solve	PDelta	SRSS	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor
1	Spectra Results	☑			SX	.13	SZ	.25						
2	DL	☑			DL	1								
3	DL+LL	☑			DL	1	LL	1						
4	DL+LL+EL	☑			DL	1	LL	1	L1	1				

Entering "**Lnn**" in the BLC field means include all the BLC entries (with their factors) from Load Combination "**nn**". For example, say Load Combination 4 has "L1" entered for one of its BLC's. This specifies to include all the BLC's (with their factors) entered in Load Combination 1 as part of Load Combination 4 (this includes self-weight and RSA entries as well). The flags for Load Combination 1 (i.e. Solve, PDelta, and SRSS entries) apply only to Load Combination 1 and will not be used when Load Combination 4 is solved.

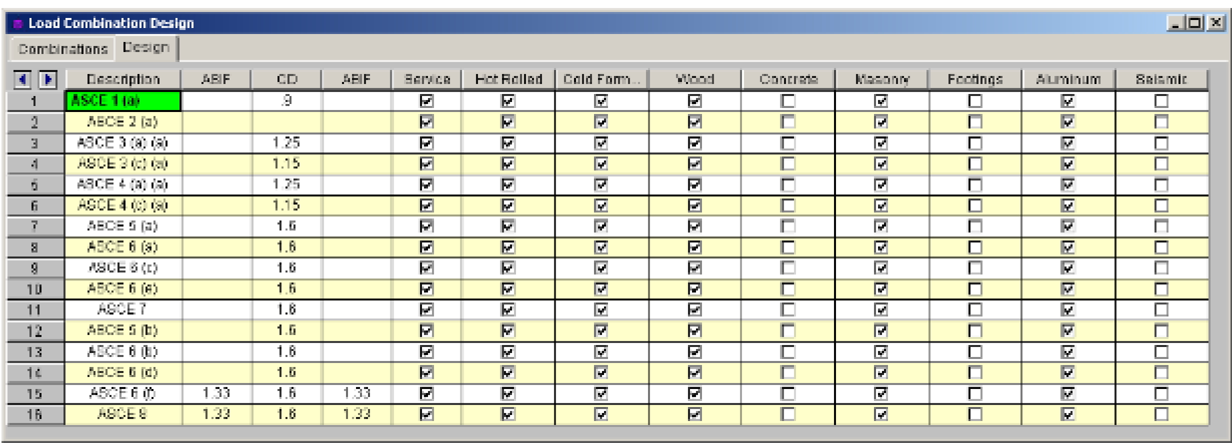
Also, the factor we enter with the "**Lnn**" entry will be applied to the BLC factors entered for LC **nn**. Thus, if we enter "L1" with a factor of "0.9", we're including 90% of the BLC entries of Load Combination 1.

Note

- These "combinations of load combinations" can only be nested to one level; i.e. the load combs referenced with the **Lnn** entries may not themselves have **Lnn** entries.

Transient Load Combinations

Entering a value/factor in the field labeled **ASIF** (Allowable Stress Increase Factor) indicates that the load combination should be treated as a wind/seismic load combination.



Load Combination Design

Combinations | Design

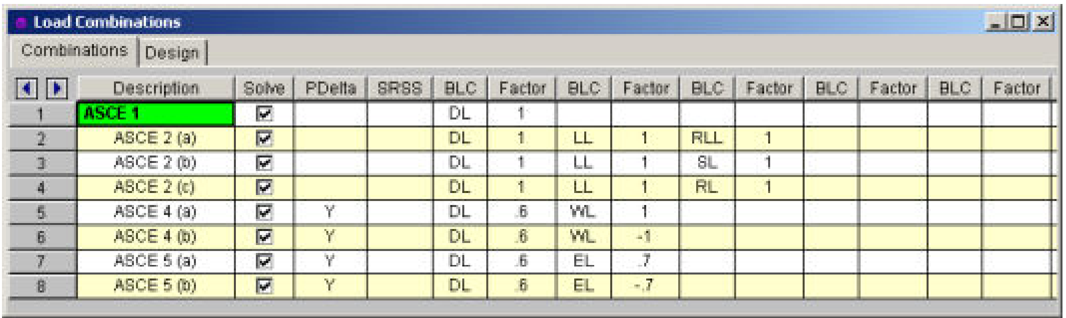
	Description	ASIF	CD	ASIF	Service	Hot Rolled	Cold Form...	Wood	Concrete	Masonry	Footings	Aluminum	Seismic
1	ASCE 1 (a)		.9		☑	☑	☑	☑	☐	☑	☐	☑	☐
2	ASCE 2 (a)				☑	☑	☑	☑	☐	☑	☐	☑	☐
3	ASCE 3 (a) (a)		1.25		☑	☑	☑	☑	☐	☑	☐	☑	☐
4	ASCE 3 (c) (a)		1.15		☑	☑	☑	☑	☐	☑	☐	☑	☐
5	ASCE 4 (a) (a)		1.25		☑	☑	☑	☑	☐	☑	☐	☑	☐
6	ASCE 4 (c) (a)		1.15		☑	☑	☑	☑	☐	☑	☐	☑	☐
7	ASCE 5 (a)		1.6		☑	☑	☑	☑	☐	☑	☐	☑	☐
8	ASCE 6 (a)		1.6		☑	☑	☑	☑	☐	☑	☐	☑	☐
9	ASCE 6 (c)		1.6		☑	☑	☑	☑	☐	☑	☐	☑	☐
10	ASCE 6 (e)		1.6		☑	☑	☑	☑	☐	☑	☐	☑	☐
11	ASCE 7		1.6		☑	☑	☑	☑	☐	☑	☐	☑	☐
12	ASCE 5 (b)		1.6		☑	☑	☑	☑	☐	☑	☐	☑	☐
13	ASCE 6 (b)		1.6		☑	☑	☑	☑	☐	☑	☐	☑	☐
14	ASCE 6 (d)		1.6		☑	☑	☑	☑	☐	☑	☐	☑	☐
15	ASCE 6 (f)	1.33	1.6	1.33	☑	☑	☑	☑	☐	☑	☐	☑	☐
16	ASCE 8	1.33	1.6	1.33	☑	☑	☑	☑	☐	☑	☐	☑	☐

If AISC 9th Edition ASD code checking is selected, the allowable stress increase factor entered in this field is applied to this load combination.

If LRFD code checking is selected, then any value of ASIF greater than 1.0 will indicate that the seismic provisions for the WF compactness checks are to be used (Table 8-1, p. 6-317 of the 2nd ed. LRFD).

P-Delta Load Combinations

The **P Delta** field is used to perform a P-Delta calculation for that load combination. You may choose from the options by clicking on the  button. A blank field indicates no P-Delta analysis, **Y** indicates that a P-Delta analysis is to be performed for the combination.



	Description	Solve	PDelta	SRSS	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor
1	ASCE 1	☑			DL	1								
2	ASCE 2 (a)	☑			DL	1	LL	1	RLL	1				
3	ASCE 2 (b)	☑			DL	1	LL	1	SL	1				
4	ASCE 2 (c)	☑			DL	1	LL	1	RL	1				
5	ASCE 4 (a)	☑	Y		DL	.6	WL	1						
6	ASCE 4 (b)	☑	Y		DL	.6	WL	-1						
7	ASCE 5 (a)	☑	Y		DL	.6	EL	.7						
8	ASCE 5 (b)	☑	Y		DL	.6	EL	-.7						

You may also perform a compression only P-Delta analysis. Invoke this option by putting a **C** in the **P Delta** field. See P-Delta Analysis for more information.

Note

- P-Delta analysis is normally required for LRFD based code checks. If this is not desired, the user may change the setting by selecting Preferences on the Tools menu and clicking the Solution and Results tab.
- P-Delta analysis is not performed on plates.

Timber Design Load Duration Factor

For Timber design, the load duration factor (CD) is entered in the **CD** field on the row that the particular CD factor applies to. Different load combinations would have different CD factors. For example, per the NDS code, a load combination that had only dead load, would have a CD factor of "0.9", while another combination that was comprised of dead load plus wind load would have a CD factor of "1.6".

The CD factor will only be applied to wood code checks on wood members. "Wood" members are those members whose material properties are defined on the **Wood** tab of the **Material Properties Spreadsheet**.

Note

- See Table 2.3.2 in the NDS 2005 code for the CD factors to be applied for typical loads. Note that the CD factor used for a load combination should be for the load with the *shortest* load duration in that load combination.

Footing Design Combinations

RISAFoot recognizes two types of combinations: Service and Design. Service combinations are used to determine if the soil bearing capacity and overturning resistance are adequate. Design combinations are used to design the footings and pedestals for flexure and shear per the chosen code.

The **ABIF** column stands for "Allowable Bearing Increase Factor". For transient loads, such as wind or seismic loads, you may want to specify an allowable bearing increase factor (ABIF) to increase the allowable soil bearing pressure for that combination. The increase factor is typically 1.333 (a 1/3 increase).

Service Combinations

The service combinations are used to calculate actual soil bearing for comparison with allowable soil bearing defined in the Criteria and Materials window.

An exact biaxial analysis is performed to calculate the soil bearing stress distribution on the footings. The service combinations are also used to calculate the overturning moment safety factor. The overburden and self weight loads are included in the dead load (DL) category for the service combinations.

Design Combinations

The design combinations are used to calculate required reinforcing steel and also to check both shear and bearing on the concrete footings.

The overburden and footing self weight loads are typically NOT included in the dead load (DL) basic load category for the design combinations. The pedestal self weight is always included in the DL load category. See Overburden and Self Weight for more on this.

Generating Building Code Combinations

Major portions of the load combinations that are specified by building codes are included and may be applied to the model for solution. These combinations may be inserted by selecting the LC Generator button from the **Window Toolbar** once the **Load Combinations Spreadsheet** is open and active. This will activate the **Load Combination Generator Dialog** shown below:

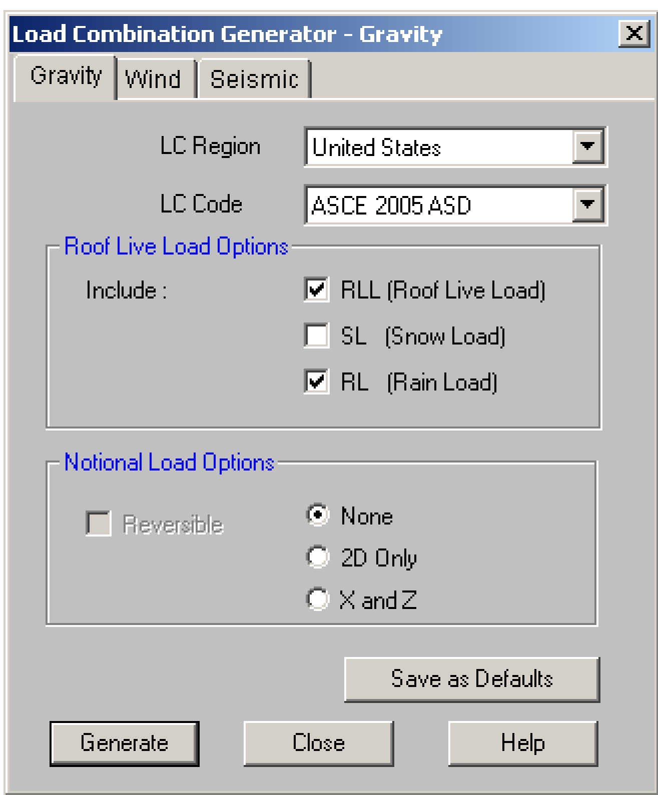
Gravity Options

The first tab of the Load Combination Generator is the **Gravity** tab. This tab contains **Roof Live Load** and **Notional Load** Options.

The **LC Region** refers to the various regions supported by the program (U.S., Canada, India, British, et cetera).

The **LC Code** refers to the actual code used to build the load combinations. For the United States, there are a number of different codes that could be used to build load combinations. If the only option is *Sample*, that means that no load combinations have been input for that region. See Customizing the Load Combination Generator for more information on how to add or edit combinations for that region.

These settings can be changed on any of the three tabs will automatically update on the other tabs. Therefore making it easy to simply move through the tabs, selecting the options you require, but not having to re-select the region and code data on each page.



Roof Live Load Options

The **Roof Live Load Options** allow the user to specify how complex the load combinations including roof live loads should be.

Placing a check in the box next to **RLL (Roof Live Load)**, **SL (Snow Load)**, and/or **RL (Rain Load)** will indicate that the selected load categories are to be included in the generated load combinations.

Notional Load Options

The **Notional Load Options** allow the user to specify how complex the load combinations including notional loads should be.

When **None** is selected, the program will not generate any Load Combinations that include the NL load category.

The **2D Only** option is used to indicate that only the most basic notional load category (NL) will be used.

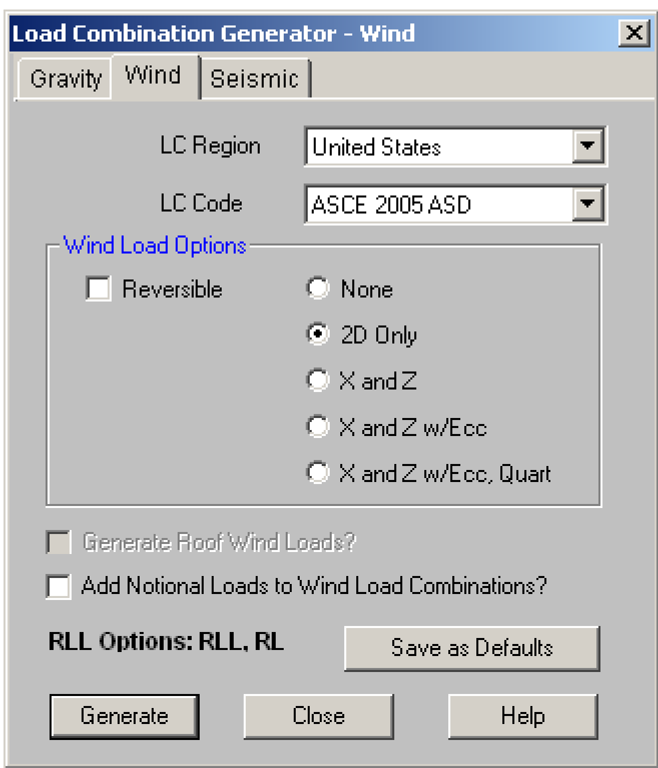
The **X and Z** option is used to indicate that the program should generate separate notional load combinations for each horizontal direction (NLX and NLZ).

When the **Reversible** box is checked, the program will generate every notional load combination twice. Once with a positive load factor, once with a negative load factor.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Notional Loads**.

Wind Options

The second tab of the Load Combination Generator is the **Wind** tab. This tab contains **Wind Load Options**.



Wind Load Options

The **Wind Load Options** specify how detailed the generated wind load combinations should be.

When **None** is selected as the wind load option, the program will not generate any Load Combinations that include wind load categories.

The **2D Only** option generates only the most basic wind load category (WL).

The **X and Z** option generates separate wind load combinations for each horizontal direction (WLX and WLZ when Y is set at the vertical axis).

The **X and Z w/ Ecc** option generates all possible wind load combinations that include partial / eccentric wind loading (WLX, WLXP1, WLXP2, et cetera) per Case 2 from Figure 27.4-8 in the ASCE 7-10.

The **X and Z w/ Ecc, Quart** option generates all possible wind load combinations per Cases 2, 3 and 4 from Figure 27.4-8 in the ASCE 7-10.. For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keyword: **Quartering**.

The **Reversible** option generates two combinations for each wind load, one with positive load factors and one with negative load factors.

The **Generate Roof Wind Loads** option generates a roof wind load BLC for use with perpendicular roof wind loads per Figure 27.4-1 in ASCE 7-10.

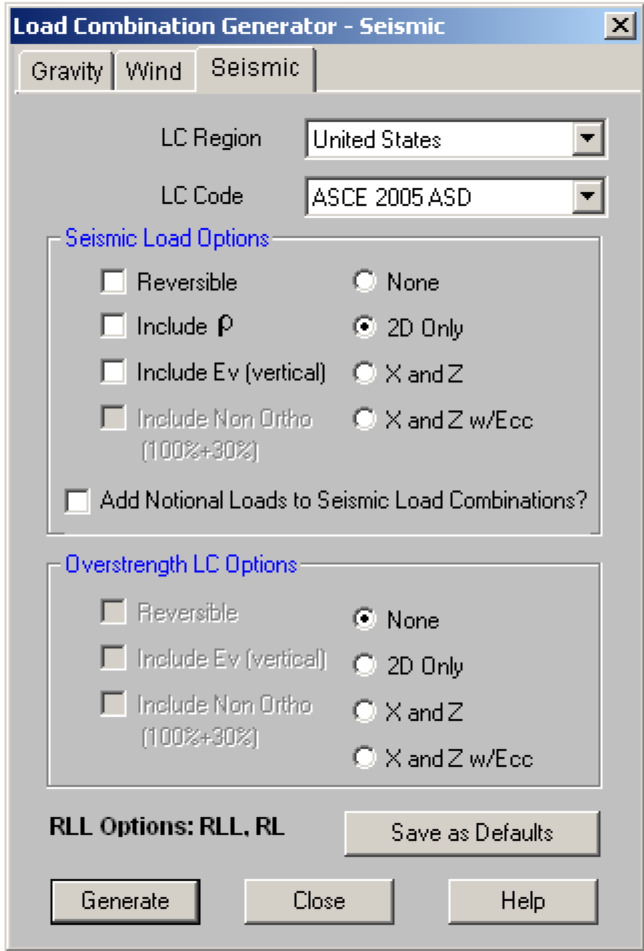
The **Add Notional Loads to Wind Load Combinations** option generates wind loads which include notional loads per the ASEC 7 and AISC codes.

Note:

- Eccentric and Quartering options are not available unless ASCE or NBC is chosen as the wind code.
- Notional loads are not generated with the Ecc/Quart specific Load Combinations.

Seismic Options

The third tab of the Load Combination Generator is the **Seismic** tab. This tab contains **Seismic Load Options**.



Seismic Load Options

The **Seismic Load Options** allow the user to specify how complex his or her seismic load combinations should be.

When **None** is selected as the seismic load option, the program will not generate any Load Combinations that include the EL load category.

The **2D Only** option is used to indicate that only the most basic seismic load category (EL) will be used.

The **X and Z** option is used to indicate that the program should generate separate seismic load combinations for each horizontal direction (ELX and ELZ).

The **X and Z w/ Eccentric** option is used to indicate that the program should generate all possible seismic load combinations that include eccentricities (ELX, ELX+Z, ELX-Z, et cetera).

If you have selected the X and Z or X and Z w/ Eccentric option, you will also have the ability to include non-orthogonal seismic loads. Simply check the **Include Non Ortho (100% + 30%)** checkbox to include these loads per the Orthogonal Combination Procedure of section 12.5.3(a) of the ASCE 7-10.

When the **Reversible** box is checked, the program will generate every seismic load combination twice. Once with a positive load factor, once with a negative load factor.

Check the **Include ρ** checkbox if you want to include the redundancy factor in your seismic load combinations. This factor is set in the Seismic tab of Global Parameters.

Check the **Include Ev (vertical)** checkbox if you want to include the vertical seismic load effect in your seismic load combinations. This value is calculated per equation 12.14-6 of the ASCE 7-10 using the S_{DS} value set on the Seismic tab of Global Parameters.

Check the **Add Notional Loads to Seismic Load Combinations** checkbox if you want to include notional loads applied as lateral seismic loads per the ASCE 7 and AISC codes.

The **Save as Defaults** button may be used to establish the current load combination generator settings as the defaults for future use. Clicking the **Generate** button will generate load combinations in the **Load Combinations Spreadsheet** based on the selected options.

General Notes on the Load Combination Generator

The following loads are not generally included in the standard combinations but may be added by editing the combinations in the spreadsheet or by modifying the source document itself. For more information, see Customizing the Load Combination Generator below:

Category Code	Description
SX, SY, SZ	Response Spectra Results
TL	Long Term Load
HL	Hydrostatic Load
FL	Fluid Pressure Load
PL	Ponding Load Category
EPL	Earth Pressure Load
OL#	Other Load Categories

Note

- The standard combinations are made up of Load Categories and Factors. Loads that are not assigned to these categories will not be included in the combinations upon solution.
- Some load categories do not occur in all of the design codes. Loads placed in categories that are not part of the standard combinations will not be included in the solution of these combinations.
- All combinations added from the drop down list are added to the envelope solution. You may remove combinations from the envelope after adding them.
- Verify the Wind/Seismic ASIF settings for combinations after you add them.
- After you add the combinations, verify the settings for the NDS Load Duration factor, CD.
- You may specify P-Delta options and SRSS combinations for each combination after you have added them.
- See Editing Spreadsheets - Inserting, Deleting and Clearing Cells for quick ways to edit or add combinations.
- When a new installation is performed the program will back-up any existing XML files that contain load combination data rather than overwriting them. The back up copies will be given the extension BAK.

Customizing the Load Combination Generator

The Load Combinations for each region are contained in an XML file which can be opened and edited using a standard spreadsheet program. An example of on of these XML files is shown below:

	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R	S	T
1	Label	Solve	pDelta	SRSS	ASIF	CD	ABIF	Service	Hot Rolled	Cold Formed	Wood	Concrete	Footings	BLC	Factor	BLC	Factor	BLC	Factor	BLC
2	ACI 9-1	1										1	1	DL	1.4	LL	1.7			
3	ACI 9-2	1										1	1	DL	1.05	LL	1.275	WL	1.275	
4	ACI 9-3	1										1	1	DL	0.9	WL	1.3			
5	ACI 9-2E	1										1	1	DL	1.05	LL	1.28	EL	1.4	
6	ACI 9-3E	1										1	1	DL	0.9	EL	1.43			
7																				
8																				
9																				
10																				

ACI Strength 99 / ACI Strength 03 / ASCE 1998 ASD / ASCE 1998 Strength / ASCE 2002 ASD / ASCE 2002 Strength

The name of the file itself becomes the name of the Load Combination Region that appears in the Load Combination Generator. Each XML file has a series of "worksheets" and the name of each worksheet will be the name of the Load Combination Code that appears in the Load Combination Generator. The user may add, edit or modify these documents to completely customize the available load combinations.

Note

- When the program is updated for new versions, the existing XML files will be "backed up" and saved with a '.bak' extension while the new XML files in the update will replace them. If any customizations were made to the XML files, they can be retrieved from the back ups.

The first row of each sheet is reserved for the column headers. The recognized column headers are as follows: Label, Solve, pDelta, SRSS, ASIF, CD, ABIF, Service, Hot Rolled, Cold Formed, Wood, Concrete, and Footings. In addition, there will be multiple pairs of BLC & Factor headers.

Other than Label and the BLC / Factor pairs, the user may omit columns. If a column is omitted, the default values will be used for those entries. The order of the column labels is optional except for pairs of BLC and Factor. For the program to correspond the Factor with correct BLC, the user should always provide BLC label PRIOR to the corresponding Factor. The user may insert blank columns or columns with other labels than described above. In those cases, the program omits those undefined columns.

Note

- As shown in the above example, the wind and seismic loads should be entered as WL and EL in order for them to be "expanded" using the Wind Load Options and Seismic Load Options.
- The program will read these files from the directory specified in Tool - Preferences - File Locations. See Customizing RISA for more information.

Loads - Joint Load / Displacement

You may specify joint loads, and enforced joint displacements and joint mass in any of the global degrees of freedom. Loads and displacements may be applied in any non-global direction by defining components of the load in the global directions. This may be accomplished graphically or in the spreadsheets. See Drawing Joint Loads below to learn how to apply joint loads/displacements/masses graphically.

Drawing Joint Loads

You can apply joint loads to joints. You must enter the load direction, magnitude and type. Make sure that you are careful to enter the correct BLC number that you want the loads assigned to. See Joint Load/Forced Displacement above for more information on joint loads.

For help on an item, click ? and then click the item.

To Apply Joint Loads, Mass and Enforced Displacements

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Click the **Apply Joint Loads** button and define the load. For help on an item, click ? and then click the item.
3. You may apply the load by choosing joints on the fly or apply it to a selection of joints.

   To choose joints on the fly choose **Apply by Clicking/Boxing Joints** and click **Apply**. Click/Box on the joints with the left mouse button.

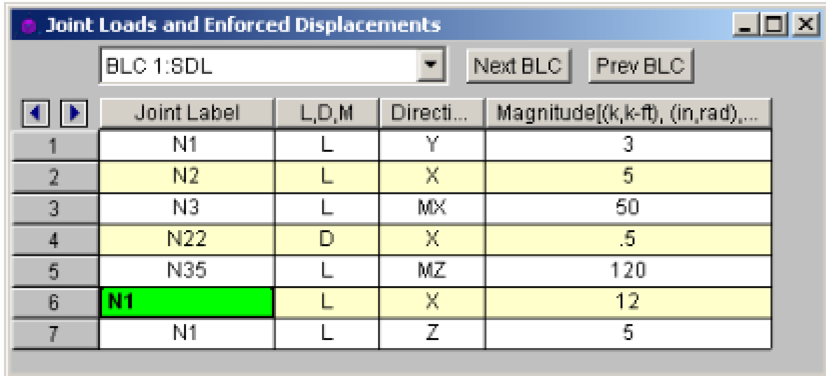
   To apply the load to a selection, choose **Apply Entries to All Selected Items** and click **Apply**.

Note

- To apply more loads with different parameters, press CTRL-D to recall the **Joint Loads** settings.
- You may also specify or edit joint loads/displacements in the **Joint Load Spreadsheet**.
- You may undo any mistakes by clicking the Undo button.

Joint Load Spreadsheet

The **Joint Load Spreadsheet** records the loads for the joints and may be accessed by selecting **Loads ▸ Joint Loads** on the **Spreadsheets** menu.



	Joint Label	L,D,M	Directi...	Magnitude[(k,k-ft), (in,rad),...
1	N1	L	Y	3
2	N2	L	X	5
3	N3	L	MX	50
4	N22	D	X	.5
5	N35	L	MZ	120
6	N1	L	X	12
7	N1	L	Z	5

When you open this spreadsheet you may view only one basic load case at a time. Use the drop down list on the toolbar to specify a different load case. The current load case is also displayed in the title bar at the top of the spreadsheet.

The Joint Label specifies the joint that receives the load or displacement. The same joint may be listed any number of times.

The next column indicates the value is a load or an enforced displacement. Enter "L" if it's a load, "D" if it's a displacement and "M" if it is a mass.

The direction code indicates in which of the global directions the value is applied. Valid entries are X, Y or Z for the translational directions, or MX, MY or MZ for the rotational directions.

The last column holds the value of the load, displacement or mass. The appropriate units for the magnitudes are displayed at the top of the column. Which units apply depends upon whether the value is a load, displacement or mass, and whether the direction is translational or rotational.

Note

- If you have a "Reaction" or a "Spring" boundary condition for the same degree of freedom that you have an enforced displacement assigned, NO reaction will be calculated. See Reactions at Joints with Enforced Displacements to learn how work around this limitation.

Joint Mass

For more sophisticated dynamics modeling, you can enter your mass directly as a mass rather than have the program convert it from a load. Using joint masses offers several advantages such as being able to define directional mass and also the ability to specify mass moment of inertia's to account for rotational inertial effects.

The units used for Joint Mass are derived from the current **Force** and **Length** units as specified on the **Units** settings. For example, if the current force units are Kips and the current length units are Feet, you will need to specify your mass as kips / g and mass moments of inertia as kip-ft^2 / g where g is the acceleration of gravity given in those units (feet per seconds squared).

When specifying a joint mass on the **Joint Loads** spreadsheet, enter an "**M**" for the load type. The directions are defined relative to the global axes. Enter translational mass using the global **X**, **Y,** or **Z** codes and mass moments of inertia by specifying the global **MX**, **MY** or **MZ**.

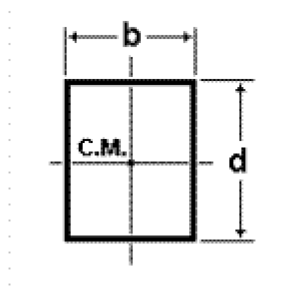
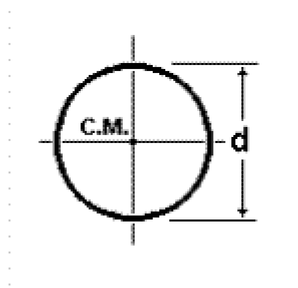
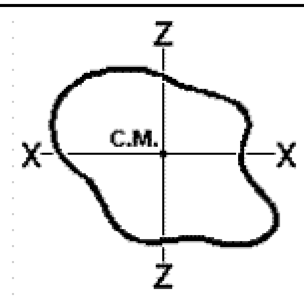
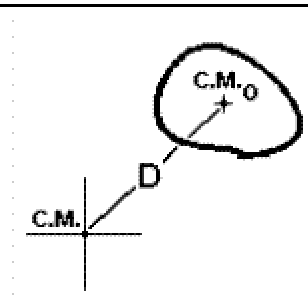
Joint masses only allow dynamic response in the direction that they've been applied. This can be a very effective way to prevent local modes. A good example is a floor diaphragm modeled with plate/shell finite elements. If the mass is only specified for the two lateral directions, you will prevent any unwanted vertical modes. Care must be taken in limiting dynamic response using directional mass for complicated structures. A structure that has "coupled modes" will not give the "real" dynamic response when mass is only specified in one or two directions. A coupled mode is a mode that has mass participate in two or three directions at one time.

Joint masses also allow you to account for rotational inertia effects by specifying a mass moment of inertia. These are particularly important when you're using a rigid diaphragm and you've also lumped all your mass at one point (typically the center of mass). The rotational inertia effects contribute to the torsion on the diaphragm and should not be neglected. The following table shows some typical diaphragm shapes and the formulas to calculate their mass moment of inertias. Note that

you can use the axis transformation equation to calculate the mass moment of inertia for diaphragms that are combinations of these basic shapes. For very irregular diaphragms, a more general equation is given based on the in-plane moment of inertia and the area of the diaphragm.

Mass Moment of Inertia About an Axis Through the Center of Mass

In the table below C.M. is the center of mass point. M is the total Mass of the area (typically including self weight, dead load, and a percentage of the live load) and is assumed to be uniformly distributed throughout. Ixx is the moment of inertia about the X-X axis. Izz is the moment of inertia about the Z-Z axis. A is the area. MMIo is the mass moment of inertia about some other point.

<table>
<tr><th>Area Plan View</th><th>Formula</th></tr>
<tr><td>
</td><td>$M (b^2 + d^2) / 12$</td></tr>
<tr><td>
</td><td>$M d^2 / 8$</td></tr>
<tr><td>
</td><td>M (Ixx + Izz) / A</td></tr>
<tr><td>
</td><td>$MMIo + M D^2$</td></tr>
</table>

Loads - Area Loads

Area loads are loads that are applied to a planar area and automatically attributed to the members in that plane. This gives you the ability to model the loading effects on a membrane, such as live load on a deck system or wind on a curtain wall, without adding unwanted stiffness to the model. (To model load **and** stiffness you may use plate shell elements loaded with surface loads.)

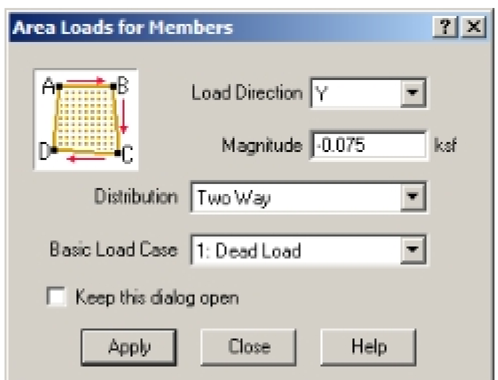
The area loads are defined by polygons of three or four sides. They may be assigned a direction for one-way span situations or considered as two-way membranes that transfer load in all directions. The loads from the area are assigned to the nearest members, not to plates. Area loads may also be assigned with an "Open Structure" load distribution which will apply loads to members based solely on their projected surface area.

Note

- Area loads are not assigned to Tension-only and Compression-only members.
- The joints that define an area load must be coplanar.

Drawing Area Loads

To apply area loads to members enter the load direction and magnitude. Make sure that you are careful to enter the correct BLC number that you want the loads assigned to.



For help on an item, click ? and then click the item.

To Apply Member Area Loads

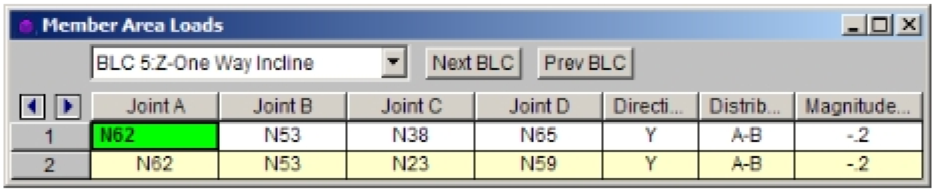
1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Click the **Area Load** button and define the load. For help on an item, click ? and then click the item.
3. Click on four joints to define the area. For three sided areas click on the last joint twice.

Note

- To apply more loads with different parameters, press CTRL-D to recall the **Area Loads** settings.
- You may also specify or edit area loads in the **Area Load** Spreadsheet.
- You may undo any mistakes by clicking the Undo button.

Area Loads Spreadsheet

The Area Load Spreadsheet records the area loads to be attributed to the members and may be accessed by selecting **Loads** ▸ **Area Loads** on the **Spreadsheets** menu.



Member Area Loads

BLC 5:Z-One Way Incline | Next BLC | Prev BLC

	Joint A	Joint B	Joint C	Joint D	Directi...	Distrib...	Magnitude...
1	N62	N53	N38	N65	Y	A-B	-.2
2	N62	N53	N23	N59	Y	A-B	-.2

When you open this spreadsheet you may view only one basic load case at a time. Use the drop down list or the 

button to specify a different load case. The current load case is also displayed in the title bar at the top of the spreadsheet.

The first four columns contain the joints that define the area to be loaded. The fifth and sixth columns indicate the direction and distribution of the load, both are discussed below. The last column holds the load magnitude.

Area Load Direction

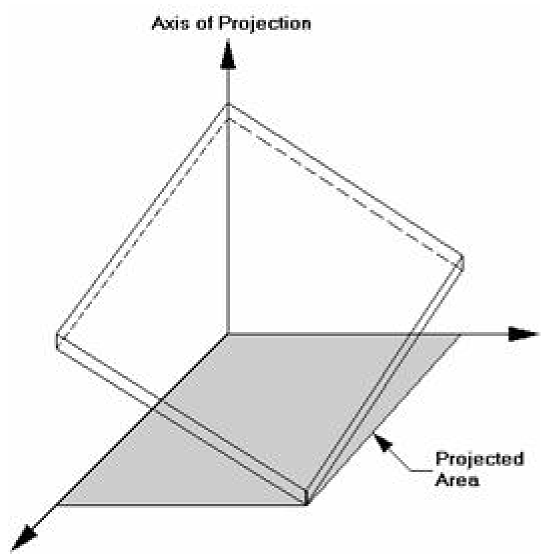
The direction code indicates the direction of application for the area load. The following directions are available:

Direction Code	Application Direction
X, Y, or Z	Global X, Y, or Z direction
PX	Projected in the direction of the global X axis
PY	Projected in the direction of the global Y axis
PZ	Projected in the direction of the global Z axis
Perp	Perpendicular to the plane of the area load polygon

Global loads are applied without being modified for projection. For example, a global Y-direction load of 1 ksf applied to an inclined plane with an area of 10 sq.ft. generates a total force of 10 kips, no matter what the incline is.

Projected loads, on the other hand, are applied in the global directions, but their actual magnitude is influenced by the planar orientation. The load is applied to the projected area of the element that is perpendicular to the load.

For example, a "PY" direction load is a projected load applied in the global Y direction. The actual magnitude of the load is the entered magnitude reduced by the ratio A/Axz. "A" is the actual area of the element and Axz is the element's projected area on the X-Z plane, which is always less than or equal to the actual area. See the following figure:



If the "Axis of Projection" in the figure is the Y-axis, then the shaded area is the total element area "projected" onto the plane perpendicular to the Y-axis (which happens to be the X-Z plane). The total load generated is equal to the input magnitude applied to the projected area. The generated load is then applied to the whole area, so the generated load magnitude is reduced accordingly.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Projected Loads**.

Note

- For the loading perpendicular to the plane of the area polygon, the order in which you click on the joints will determine whether the load direction is into our away from the plane of the area. Clicking on the ABCD joints in a clockwise order will create a loading heading down towards the plane.

Area Load Distribution

The distribution of the area load determines which members "support" the area load you define. You may choose for the load to span in all directions as a two-way load or assign a one-way span direction. The distribution direction is indicated on the area load in the model view. For one-way spans, two corners of the area define the span direction. This allows you to set the span to be parallel to one side or to be diagonal across the area.

As a third option, you may elect for the load to be distributed as an "Open Structure" in which the area load is applied to each member based solely on the projected surface area (in the direction of the loading) of that member. For an "Open Structure" area load, no distribution direction is indicated in the model view because the distribution is member specific. For example, the "Open Structure" distribution option is generally intended for open/lattice type structures for which a wind pressure is acting uniformly on all the exposed structural surfaces of the structure.

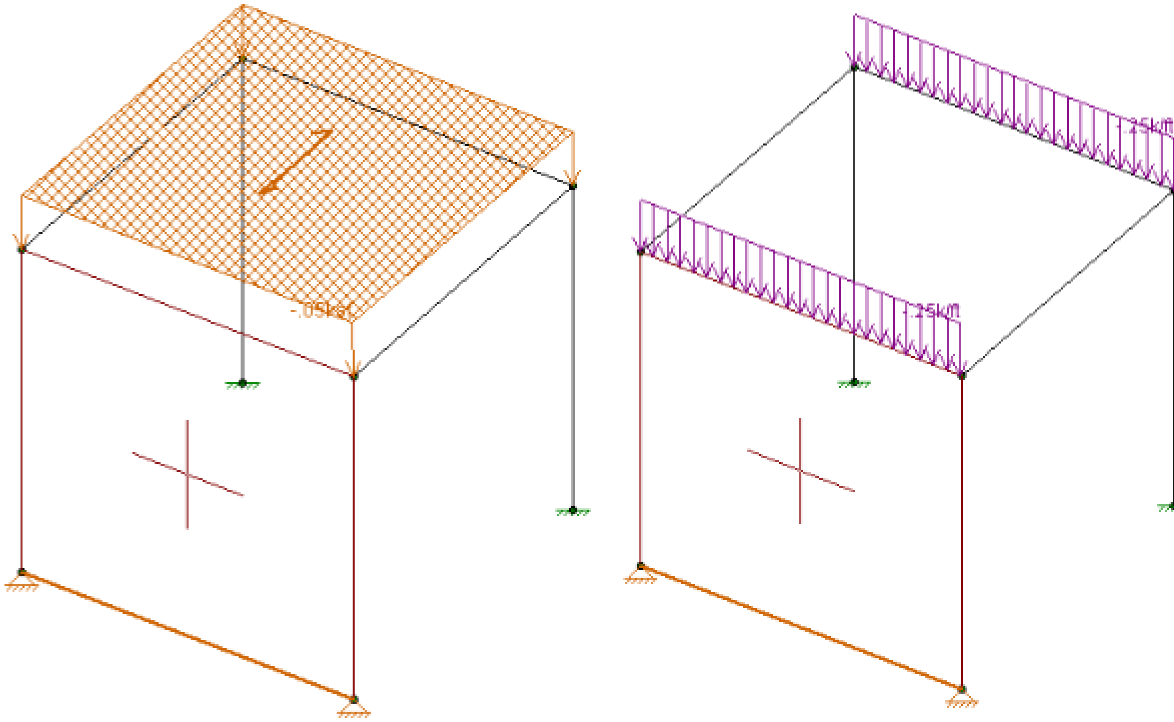
Note

- The "Open Structure" load distribution option applies load to the members in the plane of the applied load as well as the members in the structure in-front of or behind that plane.

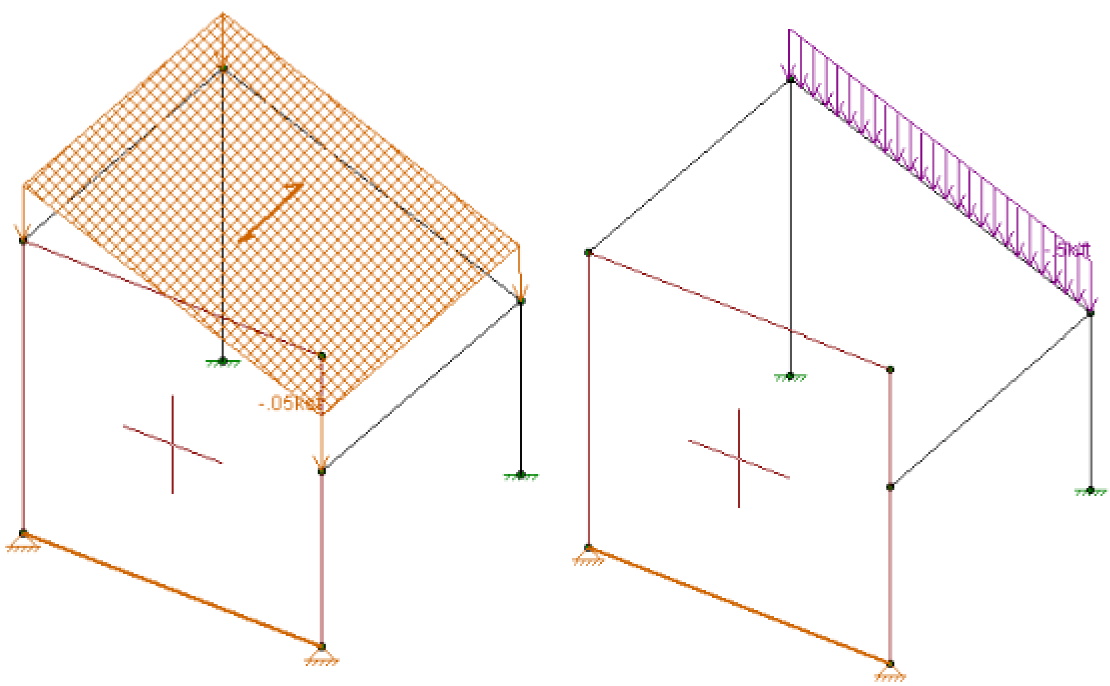
For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Area Surface Loads**.

Area Loads and Wall Panels

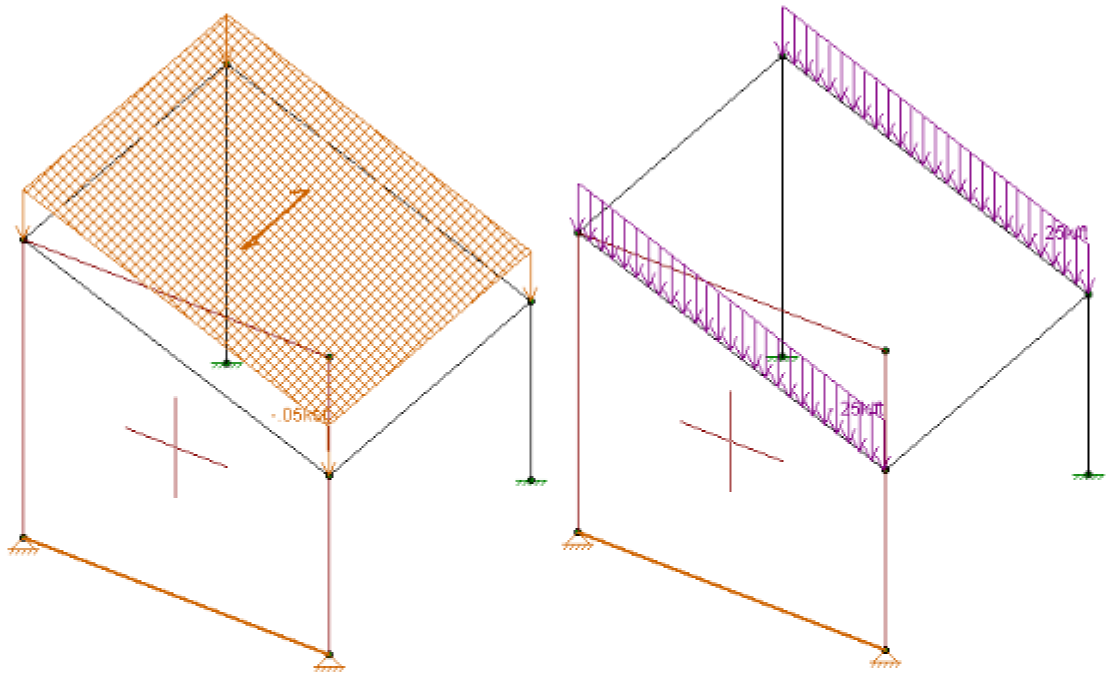
Wall panels can now accept area loads as well. This, however, will only occur at the tops of walls and at diaphragm locations. If there is an area load applied at the top of a wall or where there is a diaphragm applied, then the wall panel will receive area load in the same way as a member.



This, however, will not work if the area load is not applied at the top of the wall, or there is no diaphragm defined where the area load intersects the wall panel. In this case no load will be applied to the wall. All of the load will go in the opposite direction.



The user can force area loads to attribute to walls even if there is not a diaphragm at that elevation. This can be done by drawing in a "dummy" member or ledger beam at the same location as the wall. This member will receive area load like any other member, but should then transfer its load directly into the wall.

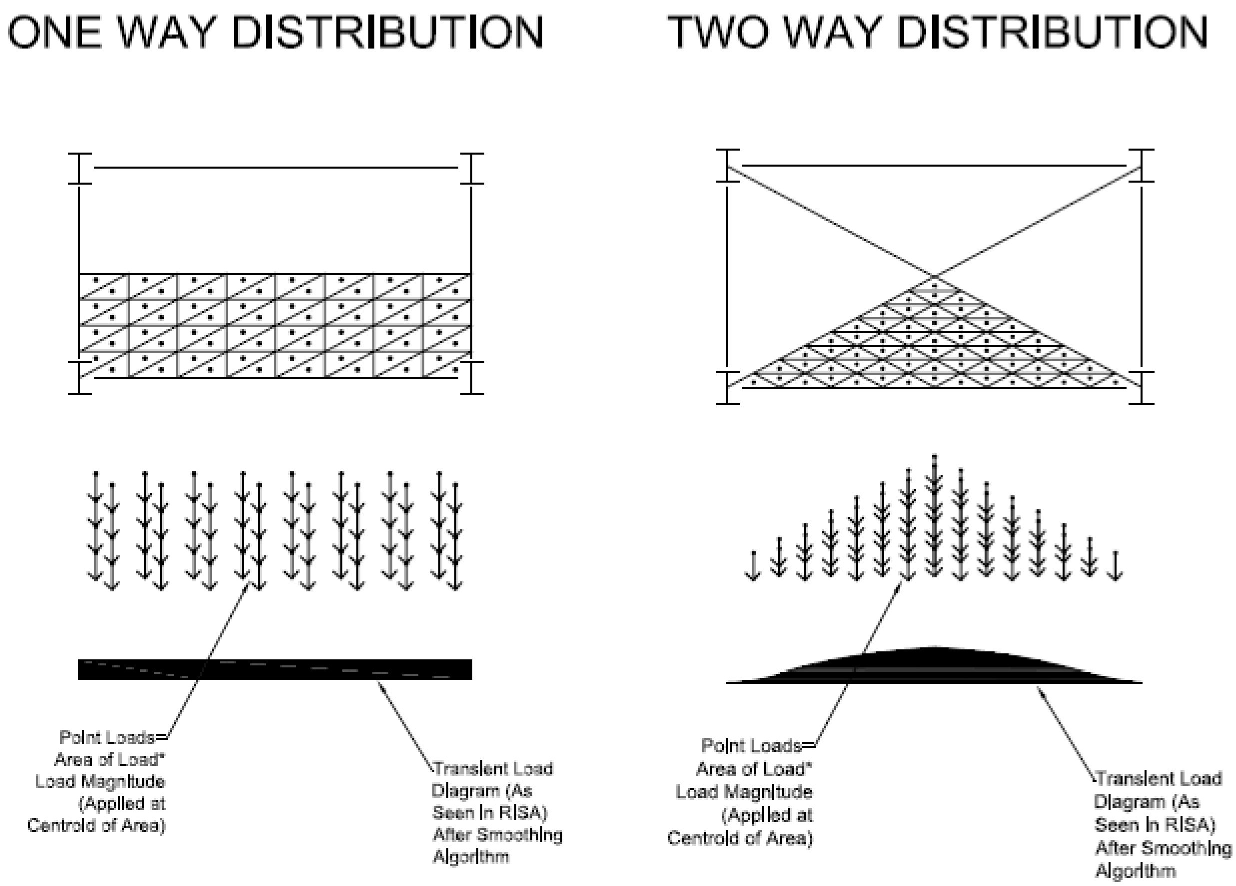


Area Load Attribution

Meshing

The program takes the area loads defined by the user and breaks the loads down into finite "pieces". These pieces are broken down until the side dimension of the "piece" is less than that defined in **Wall Mesh Size** in the [Global Parameters - Solution](#) tab. These pieces of load are then attributed to the members in the plane of the load according to the distribution defined.

For additional advice on this topic, please see the RISA News website: [www.risanews.com](#). Type in Search keywords: **Verify Area Loads**.

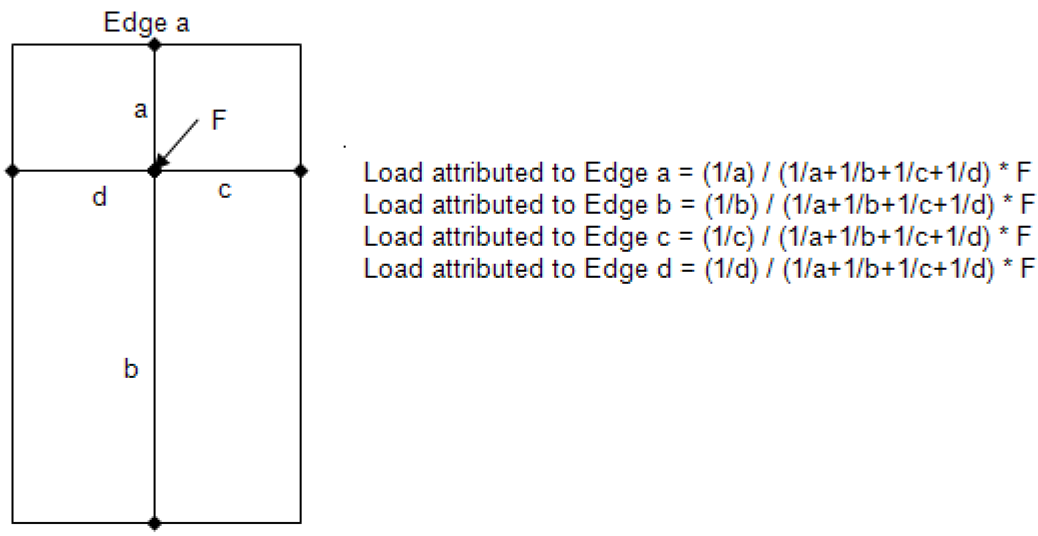


One way Attribution

One-way areas are attributed in the span direction based on the the assumption of a simply supported deck. Each point load "piece" is attributed to supporting mebmers based on the ratio of the distance of that member compared to the total deck span at that location. If there is no member in that direction then the load is attributed as a two-way load.

Two Way Attribution

Two-way areas are attributed to the members in all 4 directions based on an extension of a simply supported deck. Each point load "piece" is attriubed to the members base on the ratio of the distance of that member compared to the distance of that point from the other members as shown in the image below:



This results in a load distribution that agrees very closely with with traditional hand calc methods. But, which will not be as sensitive to the area load mesh size.

Transient Area Loads

After solution, the resulting distribution of the area load is stored in the first available basic load case with "BLC # Transient Area Loads" as the description. They may be viewed graphically or in the spreadsheets as a basic load case. These loads are transient which means they are only a result of the area load and will be deleted when the results are cleared and re-determined each time you solve the model.

Note

- Members that are designated as Tension Only members will not receive any load during the Area Load Attribution.
- Members that are designated with a member type of VBRACE or HBRACE will not receive any load during the Area Load Attribution.

You may disconnect the attributed loads and take control of them by assigning a category to the automatically created basic load case. The original area loads will be left as they were so a subsequent solution will produce another distribution of loads that are a result of the defined area loads.

Solution Speed

Note that the area load algorithm used by RISA-3D is very versatile in that it can work in any of the global directions in a three dimensional model, so that it can be used for vertical dead or live load modeling as well as wind loads. But it can add a fair amount to the analysis time when small mesh sizes are used with large models. Sometimes a small mesh is necessary to get the desired accuracy for the area load attribution.

Loads - Point Loads

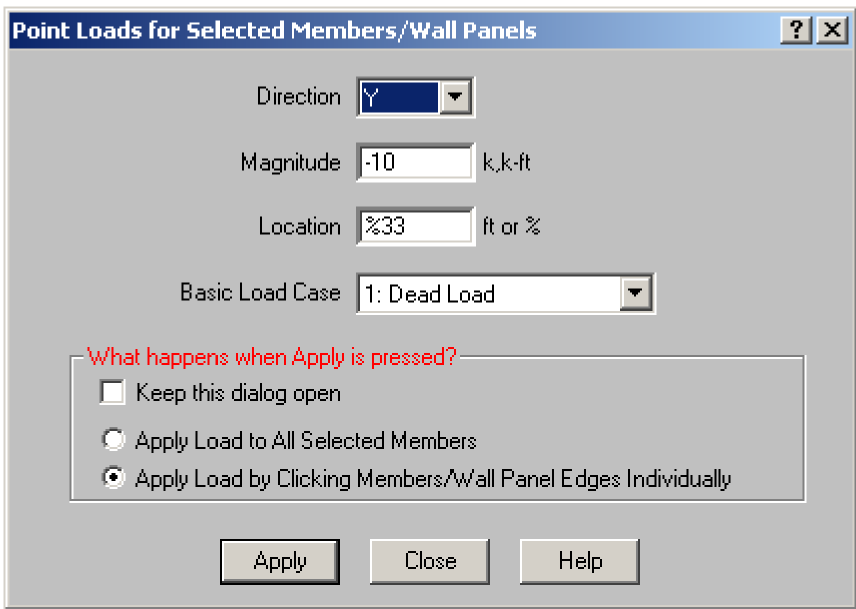
Point loads are concentrated loads applied along the span of a member or the edge of a wall panel. Defining point loads may be accomplished graphically or in the spreadsheets. See Drawing Point Loads below to learn how to draw joint loads graphically.

Drawing Point Loads

To apply point loads, enter the load direction, magnitude, and location. Make sure that you are careful to enter the correct BLC number that you want the loads assigned to.

Note

- In the event that a member end offset exists and the load lies on the offset the load will be applied at the end of the offset.



For help on an item, click ? and then click the item.

To Apply Point Loads

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Click the **Apply Point Loads** button and define the load.
3. You may choose to apply the load to a single member/wall panel at a time or to an entire selection of members/wall panels.

   To apply the load to just a few elements choose **Apply Entry by Clicking Items Individually** and click **Apply**. Click on the members or wall panels with the left mouse button.

   To apply the load to a selection, choose **Apply Entries to All Selected Items** and click **Apply**.

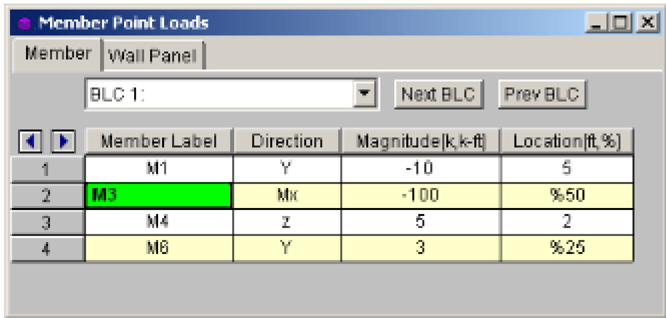
Note

- To apply more loads with different parameters, press CTRL-D to recall the **Point Loads** settings.

- You may also specify or edit point loads in the **Point Load Spreadsheet.**
- You may undo any mistakes by clicking the **Undo** button.

Point Load Spreadsheet

The **Point Load Spreadsheet** records the point loads for the member elements and may be accessed by selecting **Loads ▸ Point Loads** on the **Spreadsheets** menu.



The first column contains the label of the member or wall panel to receive the load.

The direction in the second column represents the direction of the load as one of the options mentioned above.

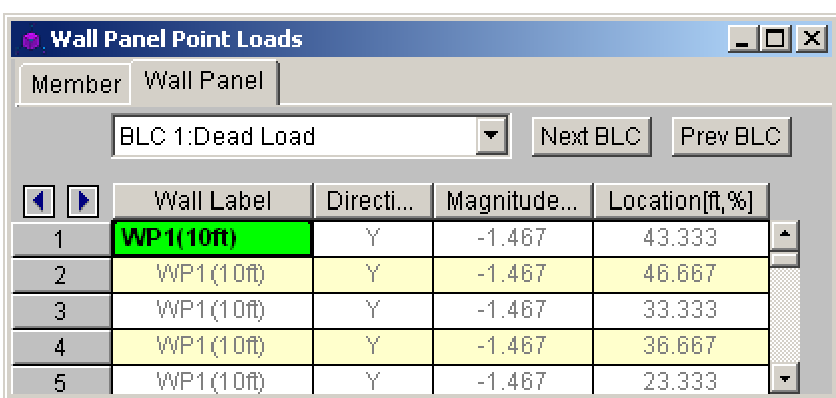
The load magnitude is recorded in the third column. The units for the magnitude are listed at the top of the column, depending upon whether the load is a force or a moment.

The final column contains the location of the load. The location is the distance from the I-joint of the member and is unaffected by any member offsets. The location of the load may be defined as a percentage of member length. To define the distance from the I-joint as a percentage of member length, enter the percentage value (0 to 100), preceded by the symbol "%". For example, a load in the center of the member would be defined with a location of "%50". Using a percentage value is handy if the member's length will be changing due to editing of the model coordinates and you wish to have the load some proportional distance from the I end.

The **Wall Panel Point Loads Spreadsheet** records the point loads on your wall panels. Note that the loads are specific to the BLCs and that you can use the drop-down list to scroll between your various load cases. The columns in the spreadsheet are the same as the member point loads spreadsheet.

Note:

- If you are using RISAFloor to bring your model into RISA-3D, this spreadsheet will be automatically populated with the point loads from RISAFloor. These lines will be grayed out because they cannot be edited from RISA-3D since they are tied to the RISAFloor analysis. Even if you detach the model from RISAFloor in RISA-3D, the loads will remain and you will not be able to edit them.



Point Load Directions

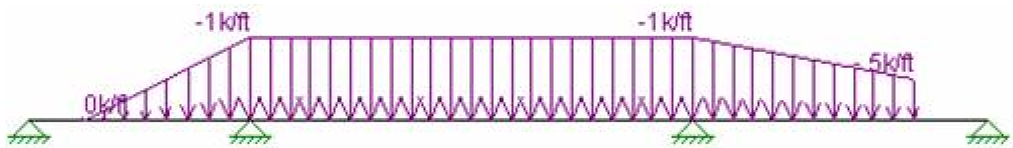
x, y, z - Load applied in local x, y or z direction

X, Y, Z - Load applied in global X, Y or Z direction

My, Mz - Moment about member local y or z axis

Mx - Torsional Moment about member local x axis

Loads - Distributed Loads



Distributed loads are loads that are spread across all or part of a member or wall panel and can be of uniform, stepped, or varying magnitude such as triangular or trapezoidal. You may define distributed loads graphically or by using the spreadsheets. See Drawing Distributed Loads below to learn how to draw distributed loads graphically.

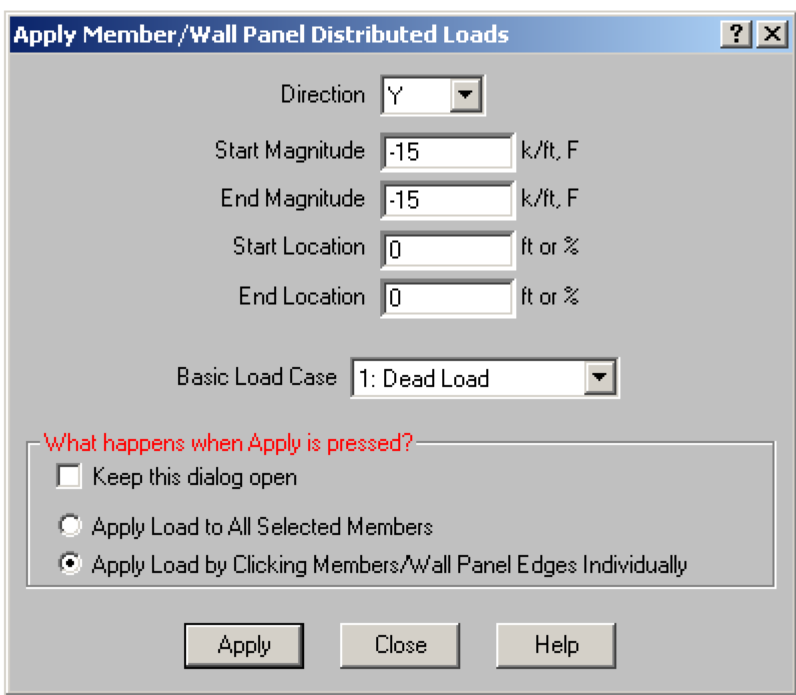
Note

- If a member has offsets defined, the offset distances will NOT be loaded. The locations are still relative to the I joint, but if the start location is less than the I end offset, the part of the load applied along the offset distance will be ignored. The same is true for the end location and the J end offset. So a full-length load is actually applied to a length equal to the full I to J joint distance minus the I-end and J-end offset distances.

Drawing Distributed Loads

You can apply distributed loads to members or wall panels. The direction of the load may either be defined in the global axes or the local axes of the member and may also be projected. For full length loads leave the Start and End Locations as zero.

Make sure that you are careful to enter the correct Basic Load Case number that you want the loads assigned to.



For help on an item, click ? and then click the item.

To Apply Distributed Loads

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.

2. Click the **Apply Distributed Loads** button and define the load.

    - For full member length loads leave the Start and End Locations as zero.

3. You may choose to apply the load to a single element at a time or to an entire selection of members and/or wall panels.

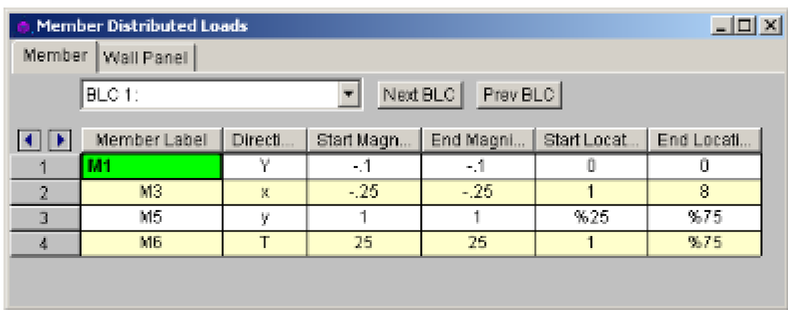
    - To apply the load to a few members/wall panels choose **Apply Entry by Clicking Items Individually** and click **Apply**. Click on the members/wall panels with the left mouse button.
    - To apply the load to a selection, choose **Apply Entries to All Selected Items** and click **Apply**.

Note

- To apply more loads with different parameters, press CTRL-D to recall the **Distributed Loads** settings.
- You may also specify or edit distributed loads in the **Distributed Load Spreadsheet.**
- You may undo any mistakes by clicking the **Undo** button.

Distributed Loads Spreadsheet

The **Member Distributed Load Spreadsheet** records the distributed loads for the member elements and may be accessed by selecting **Loads** ▸ **Distributed Loads** on the **Spreadsheets** menu or by clicking on **Distributed Loads** on the **Data Entry** toolbar.



The first column contains the label of the member/wall panel that receives the load.

The direction specified in the second column indicates which axes are to be used to define the load directions and whether or not the load is to be projected. Directions are discussed in the next section.

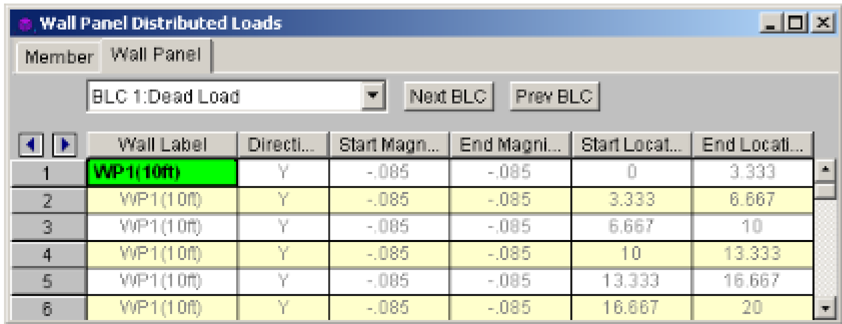
Start and end magnitudes of the load must be specified. Start and end locations for the load need only be specified if the load is not across the full member length. If both locations are left as zero then the load will be applied across the full member length.

The location columns contains the location of the load. The location is the distance from the I-joint of the member and is unaffected by any member offsets. The location of the load may be defined as a percentage of member length. To define the distance from the I-joint as a percentage of member length, enter the percentage value (0 to 100), preceded by the symbol "%". For example, a load starting or ending in the center of the member would be defined with a start or end location of "%50". Using a percentage value is handy if the member's length will be changing due to editing of the model coordinates and you wish to have the load some proportional distance from the I end.

The **Wall Panel Distributed Loads Spreadsheet** records the distributed loads on your wall panels. Note that the loads are specific to the BLCs and that you can use the drop-down list to choose a different one. The columns in the spreadsheet are the same as the member distributed load spreadsheet.

Note:

- If you are using RISAFloor to bring your model into RISA-3D, this spreadsheet will be automatically populated with the distributed loads from RISAFloor. These lines will be grayed out because they cannot be edited from RISA-3D since they are tied to the RISAFloor analysis. Even if you detach the model from RISAFloor in RISA-3D, the loads will remain and you will not be able to edit them.



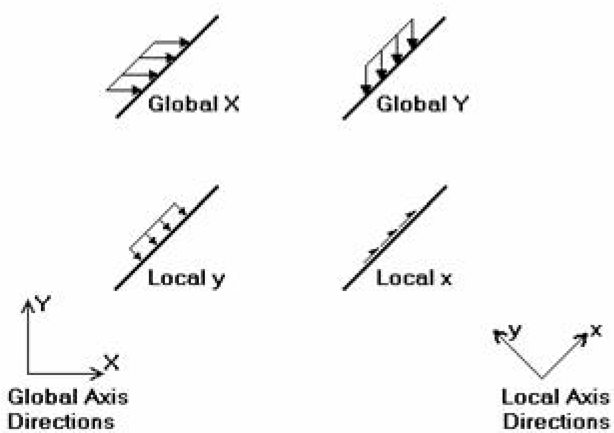
	Wall Label	Directi...	Start Magn...	End Magni...	Start Locat...	End Locati...
1	WP1(10ft)	Y	-.085	-.085	0	3.333
2	WP1(10ft)	Y	-.085	-.085	3.333	6.667
3	WP1(10ft)	Y	-.085	-.085	6.667	10
4	WP1(10ft)	Y	-.085	-.085	10	13.333
5	WP1(10ft)	Y	-.085	-.085	13.333	16.667
6	WP1(10ft)	Y	-.085	-.085	16.667	20

Distributed Load Directions

The direction code indicates how the distributed load is to be applied. Following are the valid entries:

Entry	Load Direction
x, y or z	Applied in the member's local x, y or z-axis
X, Y or Z	Applied in the global X, Y or Z-axis
Mx	Torque applied about the member's local x-axis
T (or t)	Thermal (temperature differential) load
PX	Projected load in the global X-axis direction
PY	Projected load in the global Y-axis direction
PZ	Projected load in the global Z-axis direction

This diagram illustrates the difference between local (x, y, z) and global (X, Y, Z) direction loads:

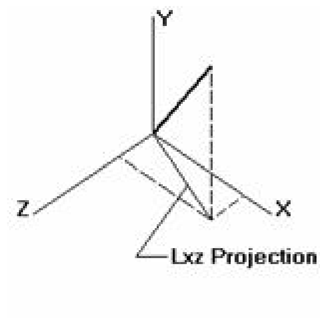


In this diagram, the local y and global Y loads shown are both negative, while the local x and global X loads are positive. As can be seen, local direction loads line up with the element's local axis directions, so their direction relative to the rest of the

model changes if the member/wall panel orientation changes. Global loads have the same direction regardless of the element's orientation.

A distributed load in the Mx direction will be applied to the member/wall panel according to the right hand rule. Remember that the positive local x-axis extends from the I joint of a member towards the J joint.

Keep in mind that global loads are applied without being modified for projection. For example, a full length Y direction load of 1 kip/foot applied to a 10 foot member inclined at 45 degrees generates a total force of 10 kips. Projected loads, on the other hand, are applied in the global directions but their actual magnitude is influenced by the member's orientation. The load is applied to the "projection" of the member perpendicular to the direction of the load. For example, a "PY" direction load is a projected load applied in the global Y direction. The actual magnitude of the load is the entered magnitudes reduced by the ratio L/Lxz, where L is the member's full length and Lxz is the member's projected length on the global X-Z plane. See the following figure:



So the total load generated is equal to the input magnitudes applied along the projected length. This generated force is distributed along the full member length, so the applied magnitudes are reduced accordingly.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Projected Loads**.

Loads - Moving Loads

The standard AASHTO loads are built into the moving loads database, however you can add and save custom moving loads as well. The moving loads can be applied in any direction, so they can be used to model crane loadings (which are typically applied in 2 or 3 directions at the same location). You can have up to 1000 moving loads in each model.

To Apply a Moving Load

1. From the **Spreadsheets** menu, select the **Moving Loads** spreadsheet.
2. Specify a pattern in the **Moving Load Pattern** field by selecting it from the drop down list.
3. In the **Load Increment** field specify the distance for the load pattern to be stepped through the path.
4. Specify the path in the remaining fields and indicate if you wish the pattern to be moved both ways through the path.

Note

- You may skip joints when specifying the path. The moving load feature is "smart" in the sense that it will try to find a way to get from one joint to the next joint in the load path sequence. The load path taken will usually be the most direct route between the joints and may be verified by animating the moving load.

To Animate a Moving Load

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and make any adjustments you wish to appear in the animation.
2. On the **Window Toolbar** click the **Plot Options** button.
3. Select the moving load from the drop-down list at the bottom of the options and then click on the **Animate** button.

Note

- You may repeat step 4 to generate animations of multiple moving loads simultaneously.

To Include a Moving Load in a Load Combination

- To include a moving load in your analysis, specify it in one of the BLC fields and enter a corresponding BLC factor. You may either type in the moving load label directly or you may select it by clicking and choose the moving load label from the drop down list.

Moving Loads Spreadsheet

The **Moving Loads Spreadsheet** records the moving loads for the member elements and may be accessed by selecting **Loads ▸ Moving Loads** on the **Spreadsheets** menu.

Moving Loads

	Tag	Pattern	Incre...	Both ...	1st Jo...	2nd J...	3rd Jo...	4th Jo...	5th Jo...	6th Jo...	7th Jo...
1	M1	HS20-36	1	☐	N1	N11	N15	N20			
2	M2	HS20-44	1	☐	N1	N11	N25	N35			
3	M3	HS20-44	1	☐	N1	N11	N30	N40			

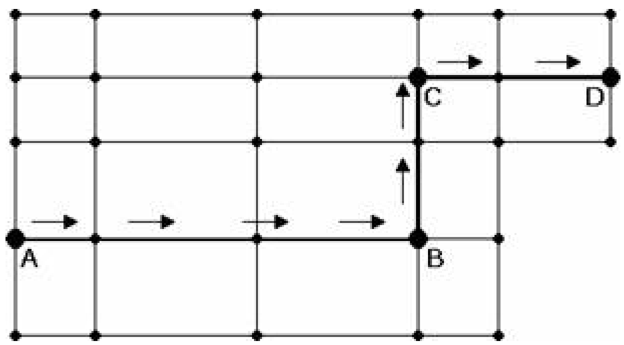
Each moving load definition is automatically assigned a **Tag** on the left side of the spreadsheet. These labels may not be edited. Moving loads are included in your analysis by referencing this label on the **Load Combinations** spreadsheet.

The **Pattern** field is the name of the moving load pattern used for that particular moving load definition. You can access the drop-down list of valid pattern names by clicking the down arrow in this cell. You may access the Moving Load Patterns and add or edit your own patterns by clicking on the **Moving Load Patterns** button on the **RISA Toolbar.**

The **Increment** field is the distance that the moving load will be moved for each step in the moving load analysis.

The **Both Ways** field is a check box that indicates whether the moving load pattern is to be applied in both directions of the load path or just one way along the load path. If the box is checked the load is first run from the start joint all the way to the last joint of the load path. The load is then turned around and the last joint is now treated as the first joint in the load path. The load is then run back to the first joint in the load path.

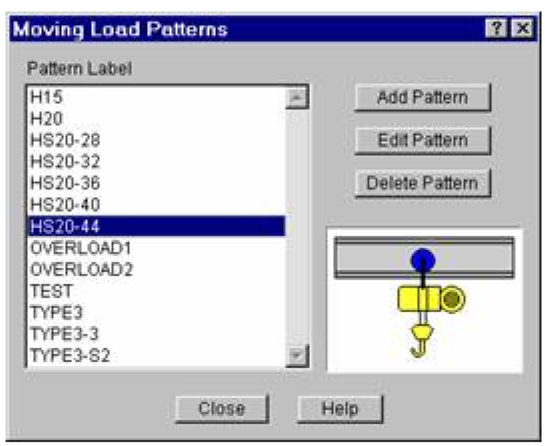
The last 10 fields are the joint numbers that are used to define the load path for the moving load. The moving load feature is smart in the sense that it will try to always find a way to get from one joint to the next joint in the load path sequence. The load path taken will usually be the most direct route between the joints. If you have a long series of co-linear members, or if there is only one valid path between your start and end joints, you usually will only need to specify the first joint and the last joint in the series. If there are several members that branch from a joint that are all part of valid paths to the next joint in the sequence, the member with the lowest member number will be the one chosen. To control exactly which route is taken in this situation, use joints at each intersection point. See the figure below:



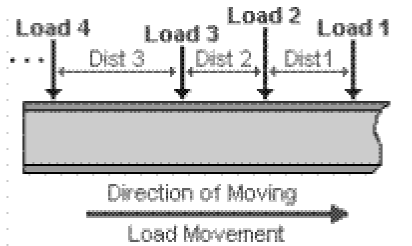
In the example moving load path shown, you would need to specify joints A, B, C, and D as the load path joints. You would not have to specify the joints that were in between the points where the load path changed direction; I.e., the moving load would automatically go in a straight line from joint A to B, etc.

Moving Load Patterns

You may access the Moving Load Patterns and add or edit your own patterns by clicking on the **Moving Load Patterns**  button on the **RISA Toolbar** and then clicking on **Add Pattern** or **Edit Pattern.**



The file that the moving load pattern database is stored in is ML_LIB32.FIL. The path to this file is specified in the **Preferences** on the **Tools** menu. You can add up to 500 different moving load patterns in the pattern database.



When you add a new pattern, the new pattern must have a unique name and can consist of up to 50 different loads. The sign of the load **Magnitude** will control which way the load is pointing in the direction specified in the **Direction** field. The direction field can be any of the 3 global directions or the 3 local directions for the members that the load will travel over. Note that if your load travels over multiple members, a local direction load will be applied based on the local axes of each member it crosses over. There is also a special code, "V", which causes the load to be applied in the direction of the current vertical axis, whatever it is (X, Y, or Z). The **Distance** is the distance between the loads.

Note

- The moving load library is not currently stored with the RISA model file. Instead it is stored in the ml_lib32.FIL database file located in the directory set using Tools - Preference - File Locations. If a file with a custom moving load needs to be transferred to another computer, then this file must also be transferred to the new computer.

Moving Loads Procedure

Moving loads are handled internally by applying the loads at discrete locations that are then moved through the model. A static solution is performed for the model at each load location. Typically, once the first solution is solved, the remaining loads are just solved against the existing stiffness matrix, so the stiffness matrix would not be rebuilt for each load position.

Note

- Models that contain tension/compression only items will have their stiffness matrix rebuilt at least once at each load position. This can make the model solution take much longer than usual.
- Multiple moving loads may be assigned to a single load combination. However, they cannot be assigned different "start times". If a delay between moving loads is required, it must be accomplished by adjusting the moving load pattern.
- The load increment may be adjusted to "slow down" a moving load when multiple moving loads are applied.

Moving Loads Results

Load combinations that contain a moving load, will step the moving load through the load path and perform a solution for each position. The results are enveloped, giving maximum and minimum results of these solutions.

For these result spreadsheets, the maximum and minimum values are shown for each section location, for each active member.

Note

- The governing load combination is always the same since the envelope solution is just for the load case that contains the moving load.
- The results for every load position are not stored; just the maximums and minimums.

Loads - Thermal Loads

You can model the effects of temperature differentials in members and plates. For members, these loads cause the axial expansion or contraction of the member along its length, i.e. axial stress only. The temperature is assumed constant across the member's depth. For plates, these loads cause an in-plane expansion or contraction of the plate. The temperature is assumed constant through the thickness of the plate.

Note

- The internal axial deflections for beam members are the average of the end deflections for thermal loads.

To Apply a Thermal Load

- Select the members you wish to assign a thermal load.
- Click to turn on the **Drawing Toolbar** if it is not already displayed and click the **Apply Distributed Load** button or the **Apply S**urface Loadbutton .
- Define a distributed load with a direction of T and a magnitude, in temperature.

Note

- You may also specify or edit thermal loads in the **Distributed Load Spreadsheet or the Plate Surface Load Spreadsheet.**
- An easy way to do thermal loadings is to define the joint temperatures (on the Joint Coordinates Spreadsheet) as all zero, so any defined thermal loads are the full stress inducing temperatures.

Recording Thermal Loads for Members

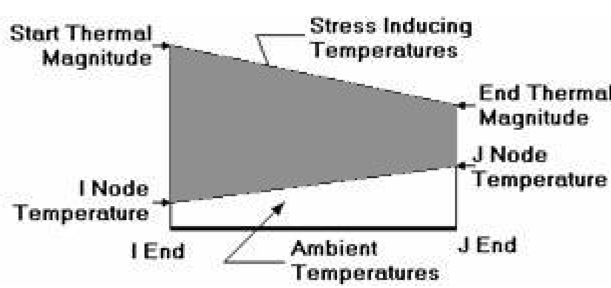
The Coefficient of Thermal Expansion (α) is entered on the **Materials** spreadsheet. Note that this value is entered per 100,000 degrees (it is sometimes listed per 1,000 degrees).

The joint temperatures recorded on the **Joint Coordinates** spreadsheet define the ambient thermal state of the structure. Thermal loads, entered as distributed loads on the **Distributed Loads** spreadsheet, induce axial stress in the member.

Member Direct Distributed Loads (BLC 1)

	Member Label	Direction	Start Mag...	End Mag...	Start Loc...	End Loca...
1	M1	T	150	150	0	0
2	M1	T	150	150	0	0
3	M1	T	150	150	0	0
4	M1	T	150	150	0	0
5	M1	T	150	150	0	0

The difference between the applied thermal load and the ambient temperature is the stress inducing temperature.

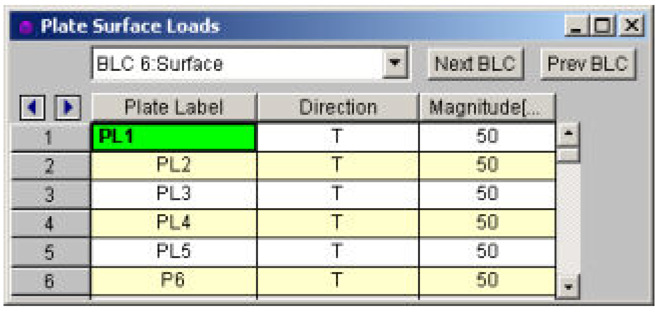


Since you can define start and end locations for the thermal load, you can define up to three separate thermal regions. Interpolating from the I-end temperature to the start thermal load for the first region, from the start thermal load to the end thermal load for the second region and from the end thermal load to the J end temperature for the third region.

Recording Thermal Loads for Plates

The Coefficient of Thermal Expansion (α) is entered on the **Materials** spreadsheet. Note that this value is entered per 100,000 degrees (it is sometimes listed per 1,000 degrees).

The joint temperatures recorded on the **Joint Coordinates** spreadsheet define the ambient thermal state of the structure. Thermal loads on the **Surface Loads** spreadsheet induce in-plane stresses in the plate.



The difference between the applied thermal load and the ambient temperature is the stress inducing temperature. For plates, you cannot define start and locations for the thermal load the way you can for member thermal / distributed loads.

Thermal Force Calculation

The joint temperatures recorded on the **Joint Coordinates** Spreadsheet define the ambient thermal state of the structure. The joint temperature at the I-end of the member is interpolated across to the J end temperature to define the ambient state of the member. Thermal loads, entered as distributed loads on the **Distributed Loads** Spreadsheet, induce axial stress in the member. The difference between the applied thermal load and the ambient temperature is the stress inducing temperature.

Thermal forces are calculated thusly:

$$Ft = A*E*\alpha*\Delta T$$

Where,

F_t = Calculated Thermal Force

A = Member Cross Sectional Area

E = Elastic Modulus

α = Coefficient of Thermal Expansion

ΔT = Stress Inducing Temperature

Prestressing with Thermal Loads

Thermal loads provide a way to introduce pre-stressing in a model. Given a desired prestress force, just back-solve the thermal force equation for the needed ΔT. Remember, as the model expands (or contracts), the prestress force may be altered.

Loads - Surface Loads

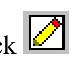
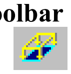
Surface loads are loads that are spread out over the surface of a plate element or wall panel. RISA-3D allows surface loads directed along the global axes, the local axes, or projected in the direction of the global axes. Loads may be input manually or assigned graphically. See Drawing Surface Loads below to learn how to draw surface loads graphically.

Drawing Plate Surface Loads

To apply surface loads to plates enter the load direction and magnitude. Make sure that you are careful to enter the correct BLC number that you want the loads assigned to. See Surface Loads above for more information on surface loads.

For help on an item, click  and then click the item.

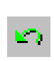
To Apply Plate Surface Loads

1. If there is not a model view already open then click  on the **RISA Toolbar** to open a new view and click  to turn on the **Drawing Toolbar** if it is not already displayed.
2. Click the **Surface Load**  button and define the load. For help on an item, click and then click the item.
3. You may choose to apply the load to a single plate at a time or to an entire selection of plates.

   To apply the load to a few plates choose **Apply Entry by Clicking Items Individually** and click **Apply**. Click on the plates with the left mouse button.

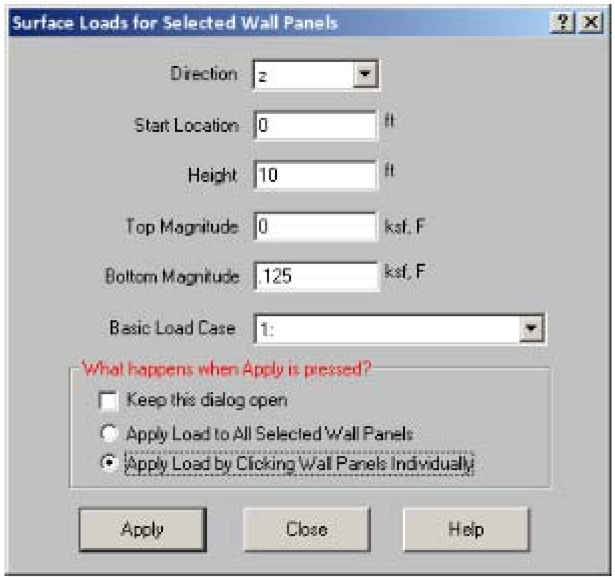
   To apply the load to a selection, choose **Apply Entries to All Selected Items** and click **Apply**.

Note

- To apply more loads with different parameters, press CTRL-D to recall the **Surface Loads** settings.
- You may also specify or edit surface loads in the **Surface Load** Spreadsheet.
- Applied surface loads are converted to equivalent corner joint loads based on the plate area tributary to each corner joint.
- You may undo any mistakes by clicking the Undo  button.

Drawing Wall Panel Surface Loads

To apply surface loads to wall panels enter the load direction, start location (from bottom), height, and magnitudes (top and bottom). Make sure that you are careful to enter the correct BLC number that you want the loads assigned to. See Surface Loads above for more information on surface loads.



For help on an item click and then click the item.

To Apply Wall Panel Surface Loads

1. If there is not a model view already open click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Click the **Wall Panel Surface Load** button and define the load.
3. You may choose to apply the load to a single wall panel at a time or to an entire selection of wall panels.

To apply the load to a few wall panels choose **Apply Load by Clicking Plates Individually** and click **Apply.** Click on the wall panels with the left mouse button.

To apply the load to a selection, choose **Apply Load to All Selected Wall Panels** and click **Apply**.

For loads that apply to the full height of the wall, leave the start location and height as zero.

Note

- To apply more loads with different parameters, press CTRL-D to recall the **Surface Loads** settings.
- You may also specify or edit surface loads in the **Surface Load** Spreadsheet.
- You may undo any mistakes by clicking the Undo button.

Surface Loads Spreadsheet

The Surface Load Spreadsheet records the surface loads for the plate/shell elements and wall panels, and may be accessed by selecting **Loads** ▸ **Surface Loads** on the **Spreadsheets** menu.

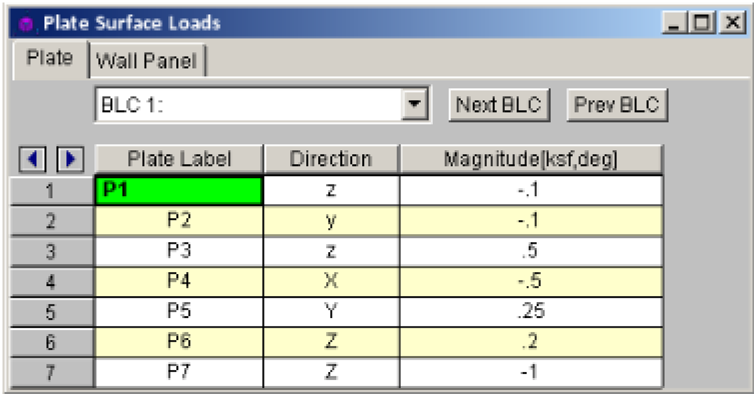
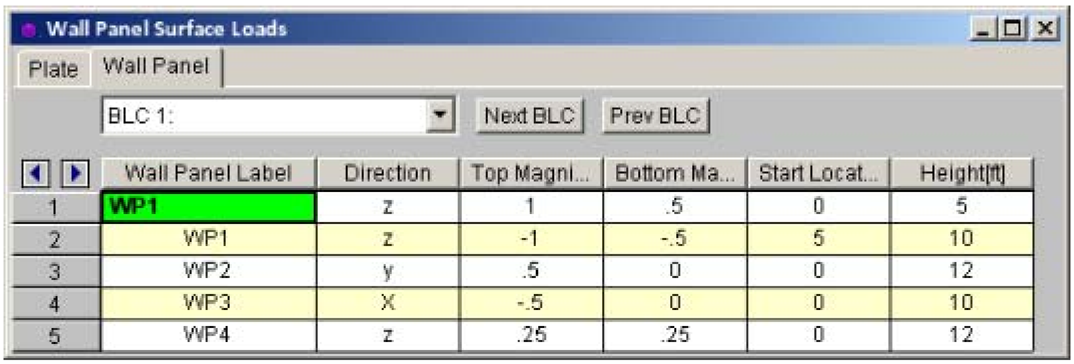


Plate Surface Loads

Plate | Wall Panel

BLC 1: | Next BLC | Prev BLC

	Plate Label	Direction	Magnitude[ksf,deg]
1	P1	z	-.1
2	P2	y	-.1
3	P3	z	.5
4	P4	X	-.5
5	P5	Y	.25
6	P6	Z	.2
7	P7	Z	-1



Wall Panel Surface Loads

Plate | Wall Panel

BLC 1: | Next BLC | Prev BLC

	Wall Panel Label	Direction	Top Magni...	Bottom Ma...	Start Locat...	Height[ft]
1	WP1	z	1	.5	0	5
2	WP1	z	-1	-.5	5	10
3	WP2	y	.5	0	0	12
4	WP3	X	-.5	0	0	10
5	WP4	z	.25	.25	0	12

When you open this spreadsheet you may view only one basic load case at a time. Use the drop down list on the toolbar to specify a different load case. The current load case is also displayed in the title bar at the top of the spreadsheet.

Under the **Plate Tab** the first column contains the label of the plate to be loaded. The second column defines the direction of the load. Direction options are discussed in the next section. The third column holds the magnitude of the load.

Under the **Wall Panel Tab** the first column contains the label of the wall panel to be loaded. The second column defines the direction of the load. Direction options are discussed in the next section. The third and fourth columns hold the magnitude of the load. The fifth and sixth columns define the Start Location (bottom) and Height of the load.

For loads that apply to the full height of the wall, leave the start location and height as zero.

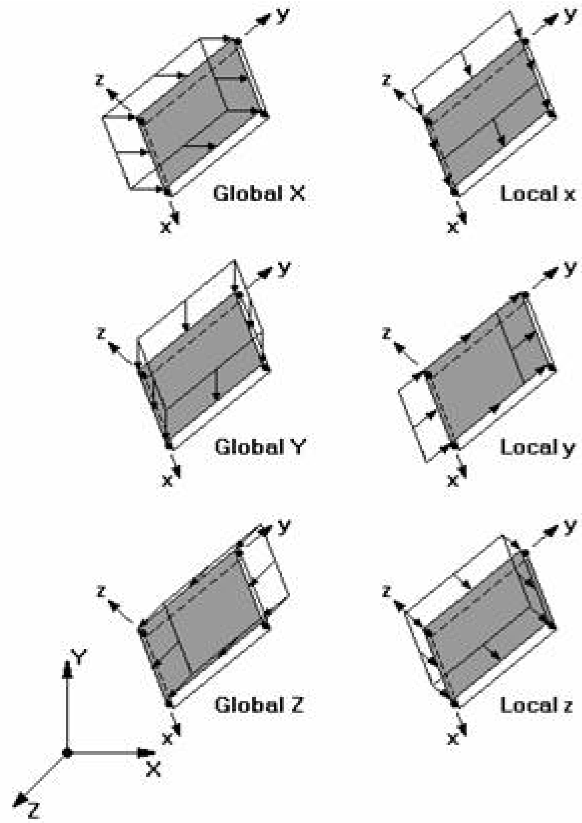
Surface Load Directions

The direction code indicates the direction of application for the surface load. The following directions are available:

Direction Code	Applied Direction
x, y, or z	Element's local x, y, or z direction.
X, Y, or Z	Global X, Y, or Z direction
PY	Projected in the direction of the global Y axis
PX	Projected in the direction of the global X axis
PZ	Projected in the direction of the global Z axis
T	Thermal (temperature differential) load

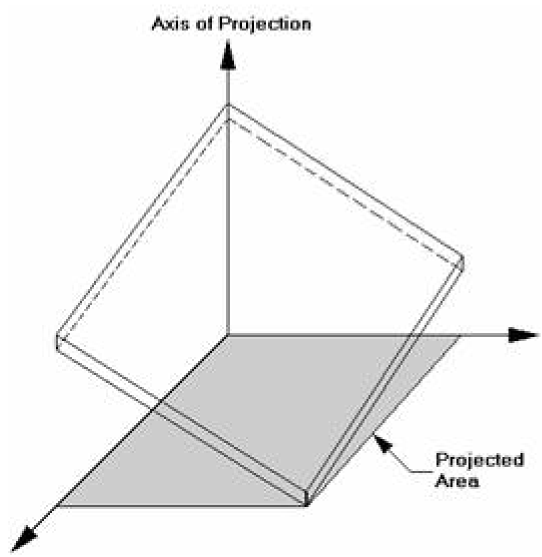
The following diagram illustrates the difference between local (x, y, z) and global (X, Y, Z) direction loads. The local direction loads line up with the element's local axis directions, so their direction relative to the rest of the model changes if

the element orientation changes. Global loads have the same direction regardless of the member's orientation. Keep in mind that global loads are applied without being modified for projection. For example, a global Y direction load of 1 kip/sq.ft. applied to an element with an area of 10 sq.ft., which is inclined at 45 degrees, generates a total force of 10 kips.



Projected loads, on the other hand, are applied in the global directions, but their actual magnitude is influenced by the element orientation. The load is applied to the projected area of the element that is perpendicular to the load.

For example, a "V" direction load is a projected load applied in the global Y direction. The actual magnitude of the load is the entered magnitude reduced by the ratio A/Axz. "A" is the actual area of the element and Axz is the element's projected area on the X-Z plane, which is always less than or equal to the actual area. See the following figure:

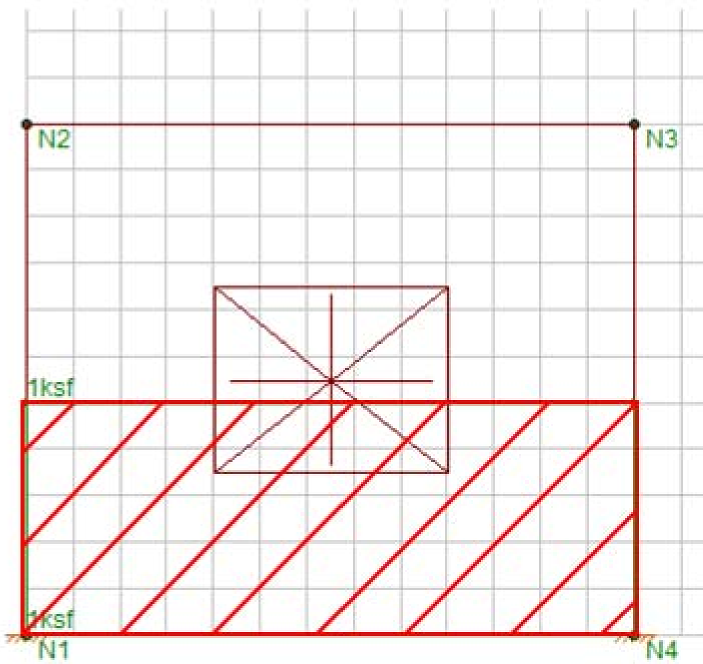


If the "Axis of Projection" in the figure is the Y-axis, then the shaded area is the total element area "projected" onto the plane perpendicular to the Y-axis (which happens to be the X-Z plane). The total load generated is equal to the input magnitude applied to the projected area. The generated load is then applied to the whole area, so the generated load magnitude is reduced accordingly.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Area Surface Loads**.

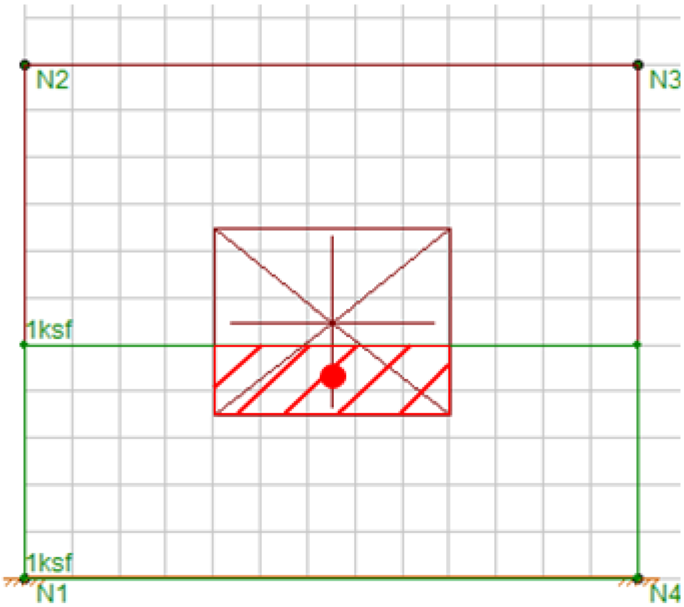
Surface Loads at Openings (Wall Panels)

When a surface load is applied to a wall panel with openings it is converted into equivalent nodal loads around the openings. The example below is a wall panel with an opening, and a surface load applied to the lower portion of the wall.

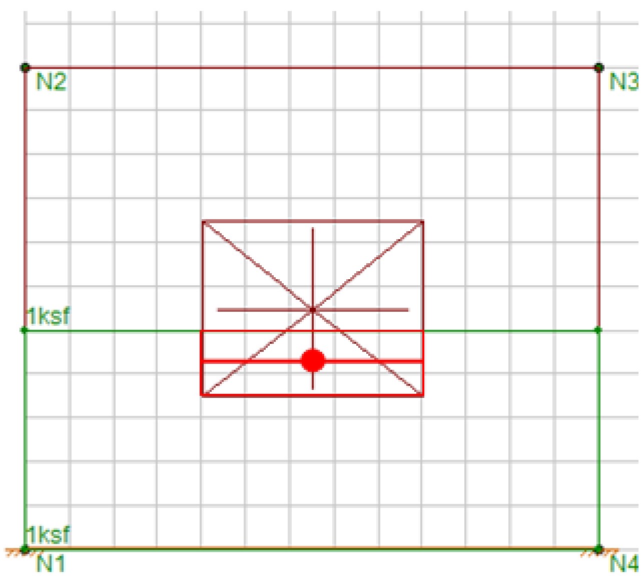


The program calculates transient nodal loads according to the following approximation procedure:

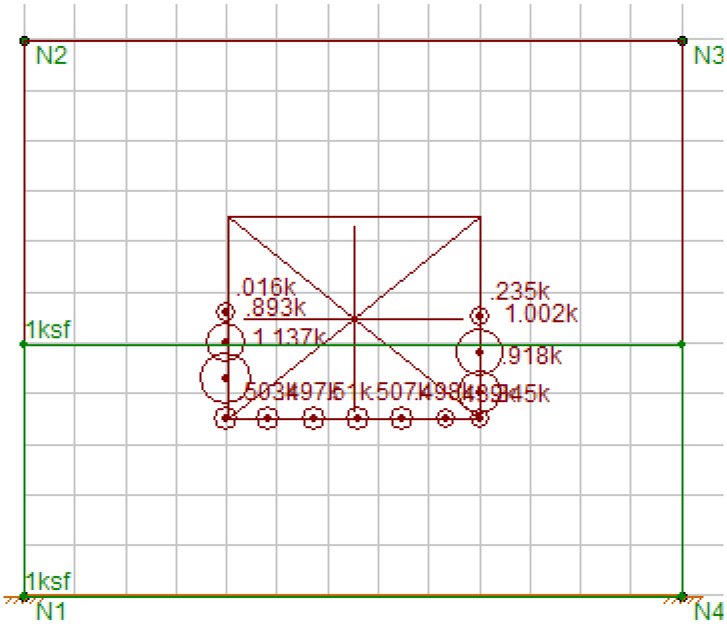
1. The centroid of the union of the surface load and the opening is determined



2. Using the centroid location, the surface load within the opening region is split into two parts



3. Each of the two parts the surface load are summed up and smeared as point loads along the edge nodes of the opening. The forces and moments are conserved with this approximation method. These generated nodal loads are transient, and will be deleted when the solution result is deleted. These loads are not visible by default, however if you would like to view them you can select **Include Transient WP Loads** under [Plot Options](). Below is an example:


N2
N3
1ksf
1ksf
N1
N4
.016k
.893k
1.137k
.235k
1.002k
.918k
.45k

Load Generation - Notional Loads

Notional Loads are used by some building codes for the stability design of a structure. They serve as a minimum lateral load, or as an alternative to modeling the actual out-of-plumbness or out-of-straightness of the structure. Instead of changing the geometry of the structure, an equivalent de-stabilizing load is added to the structure. There are numerical benefits to handling this out-of-plumbness issue with loads rather than geometry. Essentially, it is quicker and easier to adjust the loading on a structure than it is to modify the stiffness matrix of the structure.

The implementation of these notional loads is not based on a single code, but on the concept of using lateral forces equal to a percentage of the applied vertical load at each floor level. Codes that may require the use of notional loads include the following:

- ASCE 7: A minimum lateral load of 1% of the Dead Load of the structure should be applied at each floor as a notional load.
- AISC 360: A notional load to account for out-of-plumbness of the structure of 0.2% to 0.3% of the total gravity load (DL + LL) shoudl be applied at each floor as a notional load.
- AS 4100: Has a default of notional load of 0.2%
- NZS 3404: Has a default notional load of 0.2%
- BS 5950: Has a default notional load equal to 0.5%
- EC 1993-1-1: Has a notional load that can vary, but which will not normally exceed 0.5% of the applied vertical load

These notional loads are normally only assumed to act for load cases which do not include other lateral forces. However, the specific requirements of the individual code may require the use of these loads for other load cases depending on the sensitivity of the structure to stability effects.

Notional loads can only be automatically generated for diaphragm/floor levels. The program will automatically calculate the center of mass and use that point as the location to apply the Notional Loads.

Note

- The Notional Loads generated by the program are calculated for Building Structures ONLY and may not apply to non-building structures, horizontal trusses, towers, or other specialty structures.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Notional Loads**.

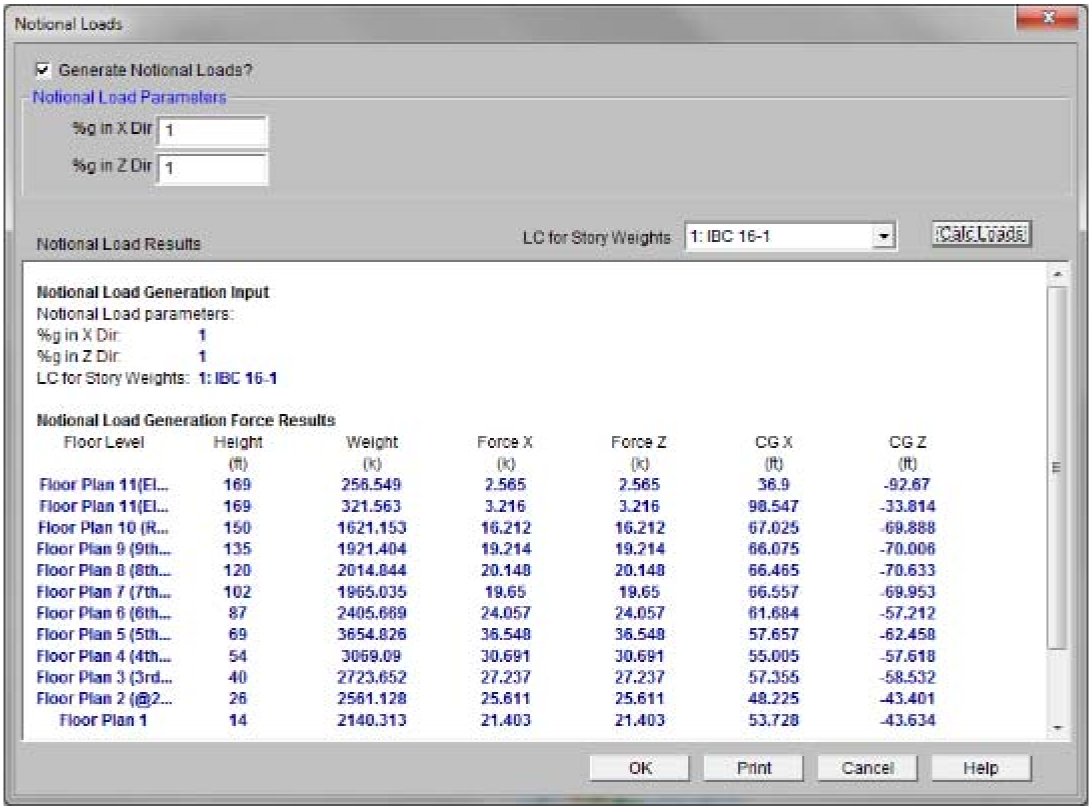
Vertical Load

The **Weight** of each diaphragm is the total tributary weight associated with it depending on the Load Combination chosen for calculating these forces on the Notional Loads dialog box.

While computing the tributary weight at a particular diaphragm, the self weight of the members/columns and plates between any two diaphragms is equally distributed amongst these diaphragms. Any weight, or load included in the specified load combination, supported between diaphragms is distributed to the diaphragm above and below in inverse proportion to its distance from each diaphragm.

Notional Load Generation Dialog

The notional load dialog only asks for a load combination and the % of gravity loads that should be applied in the lateral direction. Notional loads are only generated for a single load combination. Therefore, the user is encouraged to choose a load combination which includes the most severe loading vertical loading.



Notional Load Results

The program will only calculate the notional loads when the Insert-Notional Loads option is selected from the main menu toolbar, and when the **Generate Notional Loads?** box is checked.

Notional Load Generation Input

This section echoes back to the user all the relevant design data entered so that it can be included on a print out with the Notional Load results.

Notional Generation Detail Results

This section reports back important calculated information such as the **Weight** associated for each floor level, the Center of Gravity location for this weight and the notional load applied to each diaphragm or floor level.

Note

- The Joints / Nodes used to apply seismic load may appear as a "floating joint" at that diaphragm level. While these nodes may not be attached to any beams or framing, they are attached to the diaphragm at that floor level.

Load Generation - Seismic Loads

Building Seismic Loads can be automatically generated according to the equivalent static methods of the following codes:

- ASCE 7-2010 / IBC 2012
- ASCE 7-2005 / IBC 2006 / IBC 2009
- ASCE 7-2002 / IBC 2003
- IBC 2000
- 1997 UBC
- 2001 CBC (California Amended UBC)
- Mexican NTC-04
- Indian IS 1893:02
- Canadian NBC-2005

Seismic load can only be applied at diaphragm/floor levels. The program will automatically calculate the center of mass and the 5% accidental eccentricity for the various seismic load cases.

Note

- The Seismic Loads generated by the program are calculated for Building Structures ONLY and may not apply to non-building structures.
- The Seismic Loads generated by the program consider accidental eccentric loading as well. If you plan on using the Load Combination Generator in the Load Combinations spreadsheet, you must use the "X and Z w/Eccentric" Seismic Load option. Otherwise the eccentric BLCs that have been generated will never actually be applied.

Seismic Weight

The **Seismic Weight** of each diaphragm is the total tributary weight associated with it depending on the Load Combination chosen for calculating these forces on the Seismic Loads dialog box.

While computing the seismic weight at a particular diaphragm, the self weight of the members/columns and plates between any two diaphragms is equally distributed amongst these diaphragms. Any weight, or load included in the specified load combination, supported between diaphragms is distributed to the diaphragm above and below in inverse proportion to its distance from each diaphragm.

The total seismic weight of the whole structure is the sum of the seismic weights associated with all diaphragms and the weight associated with the base level. The base shear is always computed using the total seismic weight. The total seismic weight can be viewed using the Scaling Factor Dialog.To get here click the 

button in the Load Combinations spreadsheet.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keyword: **Diaphragms**.

Seismic Design Parameters

The parameters used in the seismic calculations may be viewed or changed by selecting Insert - Seismic Load from the Main Menu Toolbar in RISA-3D

In RISA-3D, the weight used for the calculation of seismic loads is based solely upon the Load Combination specified as the Mass LC entered in the Seismic Loads Dialog shown below:

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Generate Seismic**.

Seismic Load Parameters

Seismic Code currently allows you to choose the code which will be used for seismic load generation. For reference, sections of the 2010 edition of ASCE-7 will be cited to explain the various entries.

T represents the natural period from an eigensolution or Method B analysis. If these values are not entered, then the program will use the approximate Method A period as defined in section 12.8.2.1 of ASCE 7-10. This value is entered for each of the Global horizontal directions.

Ct is the building period coefficient as defined in 12.8.2.1 of ASCE 7-10. It is used in conjuction with the Ct exponent "x" to determine the Method A period. These are defined for each of the Global horizontal directions.

Note:

- You can either input the period manually in the Global horizontal directions, or you can input Ct and the Ct exponent "x" and the program will use Eq. 12.8-7 of the ASCE 7-10 for calculation of the period.

R is the Response Modification Factor as defined in table 12.14-1 of ASCE 7-10. It provides a reduction for the design seismic force based on the ductility of the system. This is defined for each of the Global horizontal directions.

Note:

- The program offers a single R value input in each direction. There are situations where the lower portion of the structure may have a different R value than the upper portion. This can be typical with a concrete pedestal supporting a wood structure. In this case, two sets of load combinations would be created, one set in which only the Wood design checkbox would be checked in the Load Combinations spreadsheet and one set in which only the Concrete design checkbox is checked. In this case, the R value for the concrete pedestal would be input in the Seismic Load generator. Then, in the Load Combinations spreadsheet, the wood load combinations would have their seismic load factors

factored by the ratio of the wood R value over the concrete R value. In this way, the two R values can be taken into account in the same direction.

Base Elevation determines the height at which the structure is assumed to be connected to the ground. This is important for hillside structures or structures with sub-grade floor levels. A certain amount of structure self weight may be associated with base level (or sub-grade levels) of the structure. The **Add Base Weight** checkbox determines if that self weight will be added into the base shear to be distributed as lateral force through the height of the structure per section 12.8.3 of ASCE 7-10. If no elevation is chosen for base elevation, then the lowest joint in the structure will be assumed to be the base elevation.

Risk Categoryis used to determine the importance factor assigned to the structure per table 1.5-2 of ASCE 7-10.

S_D1 represents the 5% damped spectral response Design acceleration for a 1.0 second period.

S_DS represents the 5% damped spectral response Design acceleration for short period response.

S_1 represents the 5% damped spectral response Mapped acceleration for a 1.0 second period.

TL represents the point at which the structural response is assumed to transition from a velocity controlled response to a displacement controlled response. These values are shown on Figures 22-12 through 22-16 in the ASCE 7-10.

Mass LC is used to dictate which load combination should be used to define the weight of the structure when the seismic event is assumed to occur. In ASCE 7-10 this would be based on the criteria in section 12.7.2.

At the bottom of the dialog box, there is a drop down list for **Add Loads to BLC starting...**, this is to define where the program will add the earthquake basic load combinations to the Basic Load Cases spreadsheet. Be careful not to overwrite basic load cases that have already been created.

Seismic Load Results

The program will calculate the appropriate seismic loads and present the calculations in a printable report. You may open the seismic load generator at any time to view, print, or recalculate the seismic loads.

Seismic Generation Input

This section echoes back to the user all the relevant design data entered so that it can be included on print out with the Seismic Load results.

Seismic Generation Detail Results

This section reports the values used to obtain the Base Shear in each of the two global directions.

T Used is the period which was actually used to determine an upper limit for Cs in equations 12.8-3 and 12.8-4 of ASCE 7-10 (if applicable). A user defined period may not be used if it exceeds the upper limit period.

T Method A is the approximate period calculated per equation 12.8-7 of ASCE 7-10

T Upper Limit is the maximum allowable period to be used in equations 12.8-3 and 12.8-4, calculated per Table 12.8-1 of ASCE 7-10

Importance Factor is determined from Table 1.5-2 of ASCE 7-10, based on the specified Risk Category

Design Category is determined in Section 11.6 of the ASCE 7-10 and reported here.

V (Base Shear) is calculated using the Governing Equation listed next to it.

Governing Equation is the equation which was used to calculate the base shear. This is typically from 12.8 of ASCE 7-10.

Cs is the seismic response coefficient used to calculate the seismic base shear, V.

Seismic Generation Force Results

This section reports back information used in distributing the seismic force to each diaphragm or story level. This includes the calculated **Height** and **Weight** of each diaphragm, the calculated **Force** in each horizontal direction and the calculated location of the Center of Gravity of the diaphragm (**CG**).

Note:

- In ASCE 7-10 there is no required seismic loading required for structures which fall under Seismic Design Category A. Instead, notional loads should be applied.

Seismic Generation Diaphragm Results

This section reports back information used in calculating the accidental torsion values. This includes the **Width** and **Length** of each diaphragm and the distance used for the accidental eccentricity.

Note

- The Joints / Nodes used to apply seismic load may appear as a "diamond" shape pattern of unattached nodes at that diaphragm level. While these nodes may not be attached to any beams or framing, they are attached to the diaphragm at that floor level. To get rid of these nodes, it is necessary to re-run the Seismic force generation with the design code set to NONE.
- Any equations based on NON-building structures are not currently taken into account for the seismic load calculations.

Load Generation - Wind Loads

"Building" Wind Loads can be automatically generated per the following codes:

- ASCE 7-10
- ASCE 7-05
- ASCE 7-02
- ASCE 7-98
- ASCE 7-95
- NBC 2005 (Canadian)
- NTC 2004 (Mexican)
- IS 875: 1987 (Indian)

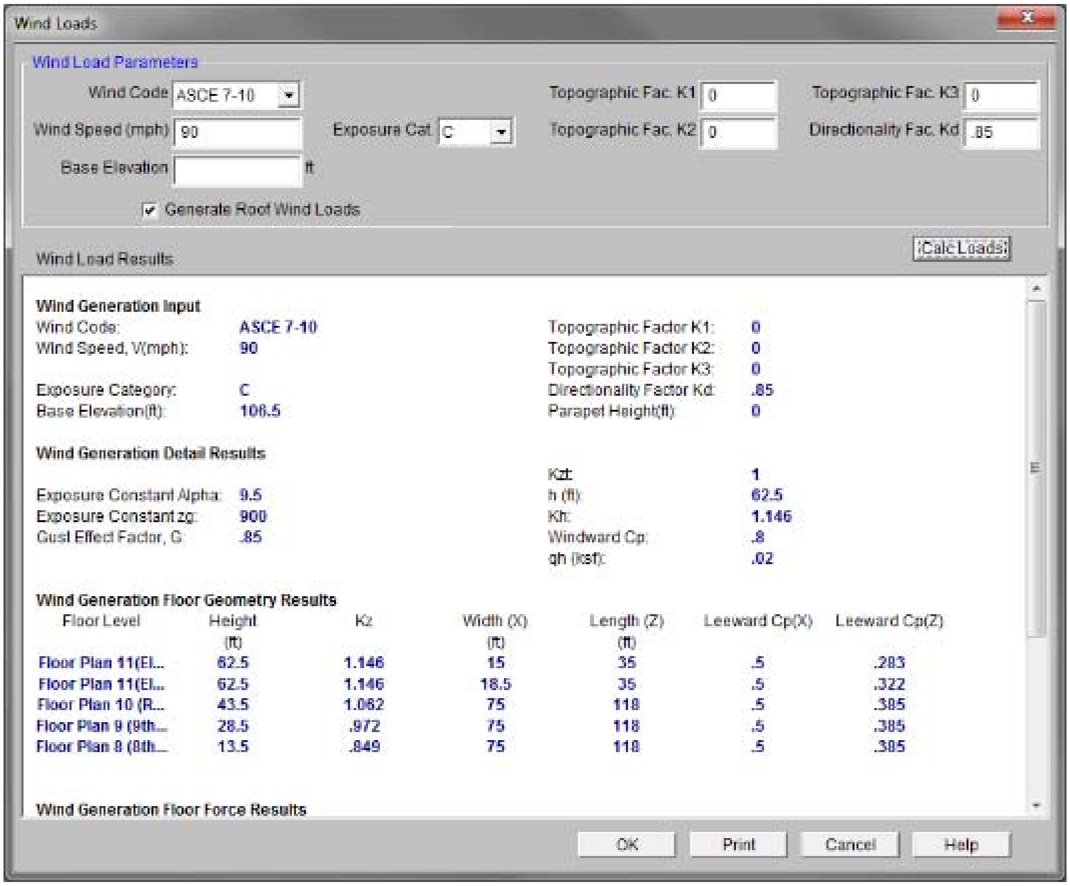
Note:

- The wind load generator creates Basic Load Cases. You must generate wind load combinations to have the wind loads actually applied to the structure.

Wind Load Parameters

The parameters used for automated wind load generation may be viewed or changed by selecting Insert - Wind Load from the Main Menu Toolbar. These settings may also be changed when the wind load dialog pops up when transferring between RISAFloor and RISA-3D.

The parameters and results shown below are specific to ASCE 7-10, however the concepts apply to all wind codes. For questions specific to other wind codes contact RISA Technical Support.



Wind Code specifies which code will be used to generate the loads.

Wind Speed (V) is used to calculate wind pressures. See ASCE 7-10, Section 26.5.1

Base Elevation defines the elevation that the program considers as the "ground elevation". This is typically used for structures which have basements, or base coordinates other than 0.

Exposure Category is used for multiple wind load calculations, and is defined in ASCE 7-10, Section 26.7.3

Topographic Factors (K1, K2, K3) are used to calculate the Topographic Factor (K_{zt}) per ASCE 7-10, Section 26.8.

Directionality Factor (K_d) is used to calculate wind pressures. See ASCE 7-10, Section 26.6.

Generate Roof Wind Loads is only present in RISA-3D models which are integrated with RISAFloor. This toggles the automatic generation of sloped roof wind loads.

Wind Load Results

The program calculates the appropriate wind loads and presents the calculations in a printable report. You may open the wind load generator at any time to view, print or recalculate the wind loads.

Changes made to RISAFloor models which are integrated with RISA-3D models will be reflected in automatically updated wind loads when transferring between the programs.

Wind Generation Input

This section reports values which were directly or indirectly input from the Wind Load Parameters.

Wind Generation Detail Results

This section reports important values that were calculated using the Wind Load Parameters, as well as the height and base elevation of the structure.

Importance Factor is calculated using the input Occupancy Category.

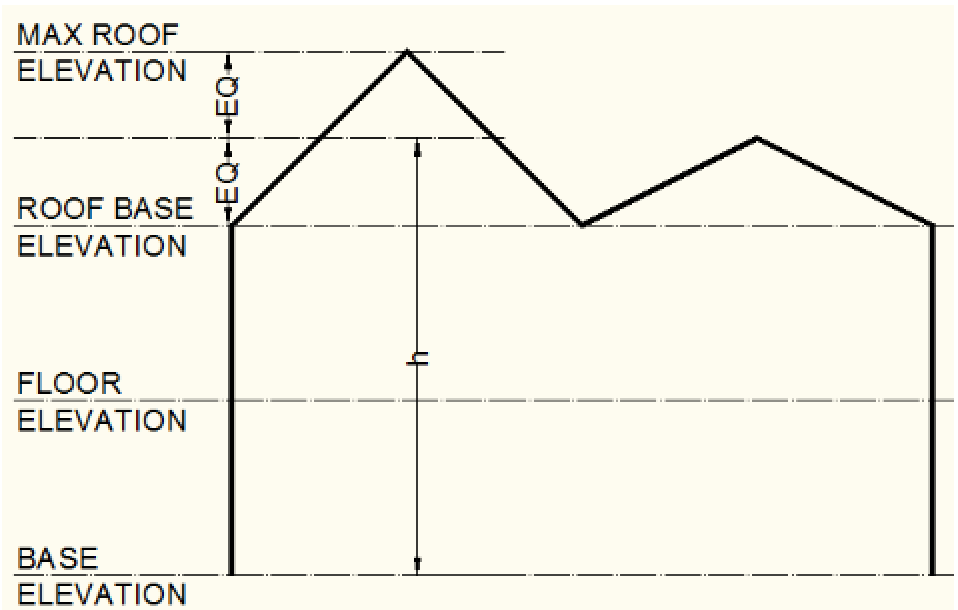
Exposure Constant Alpha (α) is determined from ASCE 7-10, Table 26.9-1 based on the input Exposure Category.

Exposure Constant z_g is determined from ASCE 7-10, Table 26.9-1 based on the input Exposure Category.

Gust Effect Factor (G) is explained in ASCE 7-10, Section 26.9. The wind load generator always sets this value at 0.85.

Topographic Factor (K_{zt}) is calculated per ASCE 7-10, Section 26.8.2 using the input Topographic Factors (K1, K2, K3)

Mean Roof Height (h) is determined as illustrated below:



Exposure Coefficient (Kh) is calculated per ASCE 7-10, Table 27.3-1 at the mean roof height as illustrated above. The Exposure Constants (z_g, α) are used in conjunction with the formula in Table 27.3-1, Note 2 to calculate the Exposure Coefficient.

Windward Pressure Coefficient (Cp) is calculated per ASCE 7-10, Figure 27.4-1. The wind load generator always sets this value at 0.8 per ASCE 7-10, Figure 27.4-1.

Mean Roof Velocity Pressure (qh) is calculated per ASCE 7-10, Section 27.3.2. The input Directionality Factor and Wind Speeds are used.

Wind Generation Floor Geometry Results

The wall geometry used to calculate the wind loads is reported here.

Floor Level is the name of the Floor from RISAFloor. Multiple diaphragms on the same floor will each be reported separately. For sloped wall calculations the level will always be reported as "Sloped Roof".

Height is the height of each diaphragm above the input Base Elevation. For "Sloped Roof" floors from RISAFloor the height reported is the height of the highest point in each diaphragm.

Exposure Coefficient (Kz) is the calculated exposure coefficient at the diaphragm elevation. For "Sloped Roof" floors from RISAFloor the coefficient is calculated for the highest point in each diaphragm.

Width is calculated as the difference between the highest and lowest magnitude X-Coordinates on the diaphragm.

Length is calculated as the difference between the highest and lowest magnitude Z-Coordinates on the diaphragm.

Pressure Coefficient (Cp) is calculated per ASCE 7-10, Figure 27.4-1. The Height/Width and Height/Length ratios (see above) are used to calculate leeward pressure coefficients.

Area is used to account for the wind load on walls which project above the base Roof elevation. This only occurs in models which are integrated with RISAFloor. The program determines area by going from support-to-support finding vertical polygons along the slab edge, and calculating their projected area in both the X and Z directions.

The area reported is equal to one half the total area calculated, as wind is not expected to act on both sides of the same polygon simultaneously. In addition to the standard wall wind force calculated, there is a wall wind force calculated as the reported area multiplied by the sum of the windward and leeward wind pressures.

Wind Generation Floor Force Results

The wind forces applied to each diaphragm are reported in this section.

Floor Level is the name of the Floor from RISAFloor. Multiple diaphragms on the same floor will each be reported separately. For sloped wall calculations the level will always be reported as "Sloped Roof".

Velocity Pressure (q_z) is calculated per ASCE 7-10, Section 27.3.2. The input Directionality Factor and Wind Speeds are used.

Windward Pressure is calculated per ASCE 7-10, Section 27.4.1. The calculated Velocity Pressure and Pressure Coefficient are used, as well as the standard Gust Effect Factor (0.85).

Leeward Pressure is calculated per ASCE 7-10, Section 27.4.1. The calculated Velocity Pressure and Pressure Coefficient are used, as well as the standard Gust Effect Factor (0.85).

Force is calculated as the Length/Width (see above) of the diaphragm, multiplied by the tributary height of the diaphragm, multiplied by the sum of the windward and leeward pressures. The additional "Sloped Roof" wall wind force in RISAFloor integrated models is also added to this value for the Roof.

The tributary height of the diaphragm is defined as half the distance the diaphragm immediately above, and half the distance to the diaphragm immediately below. At the roof the tributary height is just half the distance to the diaphragm immediately below. At the lowest level of the structure the tributary height is half the distance to the Base Elevation, as it is assumed that all wind load below that is tributary to the ground.

The Forces calculated for each diaphragm are applied as horizontal joint loads at the center of exposure, and four "eccentric" points (per ASCE 7-10, Section 27.4.6). These joints form a diamond pattern, which can be viewed in the model.

Sloped Roof Wind Loads

When RISA-3D is integrated with RISAFloor the program is capable of automatically calculating the roof wind loads per ASCE 7-10, Figure 27.1-1. These loads are calculated using the mean roof height for each diaphragm, the standard Gust Effect Factor (0.85), and a Roof Pressure Coefficient (C_p) for each roof plane.

The angle (θ) for each roof plane is taken as the angle between the roof plane and the horizontal plane, as projected along the Global X and Z axes. To determine a Roof Pressure Coefficient the wind load is always assumed to act "Normal to Ridge" with θ not less than 10°. Roof planes that do not meet these conditions will still be treated as though they do.

Only the most positive values of C_p are used from the table in ASCE 7-10, Figure 27.4-1. The more negative values typically result in less base shear within the Main Wind Force Resisting System, so they are ignored. Interpolation is used between values given in the table.

The sloped roof wind loads are applied as two-way Member Area Loads to each roof plane, and are created within roof wind load Basic Load Cases.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Sloped Roofs**.

Wind Load Limitations

General

- Each diaphragm receives a full wind load, with no shielding effects considered from other diaphragms.
- Provisions for Parapet wind loading are ignored.

ASCE 7-10

- The Directional MWFRS Procedure (Chapter 27) is used for all wind load calculations
- The "low rise" provisions of Part 2 of Chapter 27 are ignored
- Components and Cladding loads are not calculated or applied.
- The building is assumed to be "Regular Shaped" per the definition in Chapter 26
- The building is assumed to not have response characteristics making it subject to across wind loading, vortex shedding, or instability due to galloping or flutter.
- The building is assumed to be located in a site for which channeling effects and buffeting in the wake of upwind obstructions do not warrant special consideration.
- The building is assumed to be fully enclosed with non-air-permeable cladding.
- The same exposure category is used for all wind directions.
- Roof overhangs are not addressed.
- Internal wind pressure is ignored. Because it must apply outwardly or inwardly to every surface simultaneously, it would cancel itself out for wall wind loading.
- Vertical uplift loads on flat roofs and flat portions of Mansard roofs are not included.

ASCE 7-05

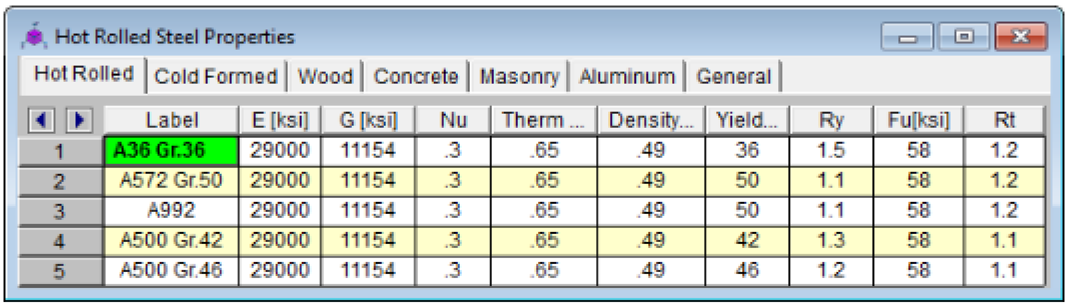
- Method 1 (Section 6.4) is not considered.
- Method 3 (Section 6.6) is not considered.
- The building is assumed to be "Regular Shaped" per the definition in Chapter 6
- The building is assumed to not have response characteristics making it subject to across wind loading, vortex shedding, or instability due to galloping or flutter.
- The building is assumed to be located in a site for which channeling effects and buffeting in the wake of upwind obstructions do not warrant special consideration.
- The building is assumed to be fully enclosed with non-air-permeable cladding.
- The same exposure category is used for all wind directions.
- Special provisions for "Low-Rise Buildings" are not considered.
- Components and Cladding loads are not calculated or applied.
- Roof overhangs are not addressed.
- Internal wind pressure is ignored. Because it must apply outwardly or inwardly to every surface simultaneously, it would cancel itself out for wall wind loading.
- Vertical uplift loads on flat roofs and flat portions of Mansard roofs are not included.

Material Properties

Material properties are defined on the **Material Properties Spreadsheet** and then are referred to as you build section sets, members, and plates. You may perform analysis using any type of material; simply define the properties for the material here. You may use up to 500 materials in a single model although most models will only have one or two. For example, your model might be made up of members of various grades of steel along with different concrete materials, timber or aluminum.

Material Properties Spreadsheet

The **Material Properties Spreadsheet** records the material properties to be used in the model and may be accessed by selecting **Materials** on the **Spreadsheets Menu**. The entries are explained below.



	Label	E [ksi]	G [ksi]	Nu	Therm ...	Density...	Yield...	Ry	Fu[ksi]	Rt
1	A36 Gr.36	29000	11154	.3	.65	.49	36	1.5	58	1.2
2	A572 Gr.50	29000	11154	.3	.65	.49	50	1.1	58	1.2
3	A992	29000	11154	.3	.65	.49	50	1.1	58	1.2
4	A500 Gr.42	29000	11154	.3	.65	.49	42	1.3	58	1.1
5	A500 Gr.46	29000	11154	.3	.65	.49	46	1.2	58	1.1

The values for A36 steel are the default material set. You of course don't have to use the A36 properties; you can change these and also add as many other materials as you need. You may then save your preferred materials as the default materials by clicking the button on the **Window Toolbar**.

Label is the material label you wish to use to describe the entered material properties. This label is how you will reference the set of properties later when defining section sets, members, and plates.

E is Young's modulus that describes the material stiffness.

G is the shear modulus and may be left blank if you would like it calculated automatically. The equation for "G" is:

$$G = \frac{E}{2.0 * (1.0 + Poisson's\ Ratio)}$$

Note

- If you enter a value for Shear Modulus that does not match the value calculated using the above equation you will be given a warning.

Nu is Poisson's ratio. Besides being used for the "G" calculation, this value is also used for shear deformation calculations. The value of Poisson's ratio may not exceed '0.5'.

Therm is the coefficient of thermal expansion and is entered per 10^5 (100,000) degrees. This coefficient is used in the calculation of thermal loads.

Density is the material density and is used in the calculation of the member and plate self weight. This density times the area gives the self weight per unit length for the members; density times plate volume gives the total weight of a given plate.

Yield is the yield stress and is used only for Hot Rolled and Cold Formed steel design.

Hot Rolled Specific Material Data

The **Hot Rolled** tab records a number of hot rolled steel specific material properties that do not exist for the other materials. These entries are described below:

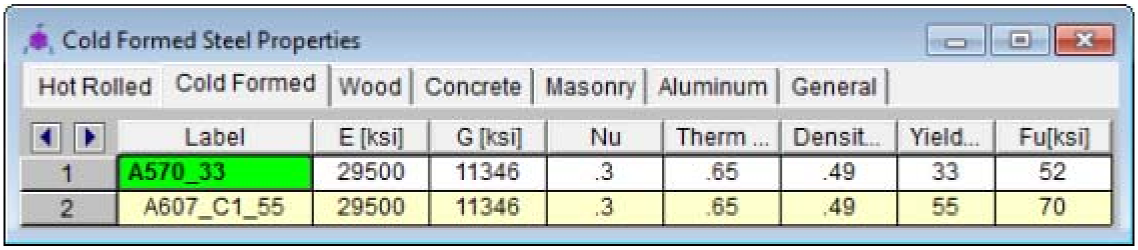
Ry is the ratio of the expected yield stress to the specified minimum yield stress. This value is used in determining the required strength of an element for the seismic detailing checks.

Fu is the specified minimum tensile strength. This value is used per *AISC 358* to calculate C_{pr} which is in turn used to calculate the probable maximum moment at a plastic hinge in the seismic detailing calculations.

Rt is currently not used by the program. In a future version of the program this value will be used to calculate the expected tensile strength for the seismic detailing checks.

Cold Formed Specific Material Data

The **Cold Formed** tab records a cold formed material property that may not exist for the other materials. This entry is described below:

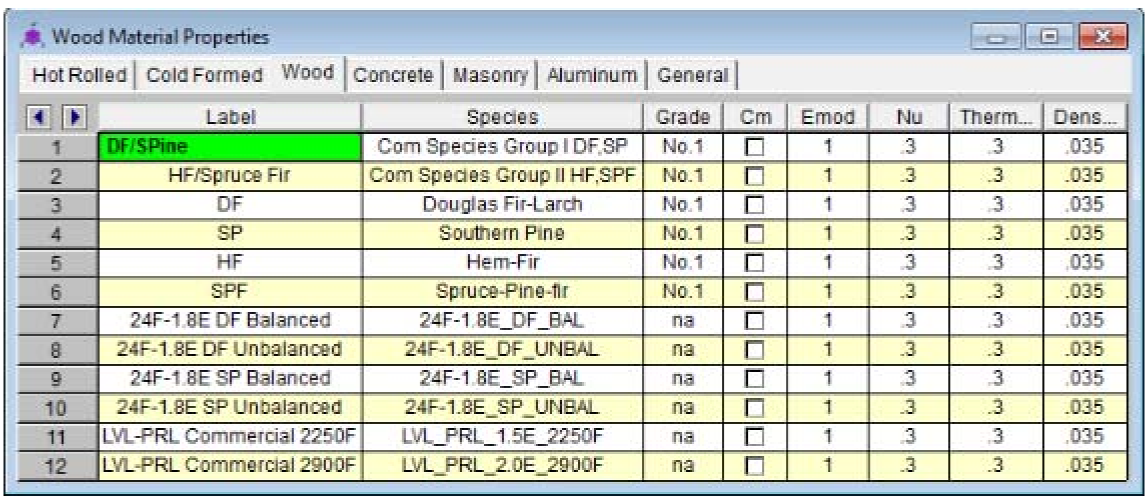


	Label	E [ksi]	G [ksi]	Nu	Therm ...	Densit...	Yield...	Fu[ksi]
1	A570_33	29500	11346	.3	.65	.49	33	52
2	A607_C1_55	29500	11346	.3	.65	.49	55	70

Fu (Ultimate) is the ultimate tensile stress.

Wood Specific Material Data

The **Wood** tab records a number of wood specific material properties that do not exist for the other materials. These entries are described below:



	Label	Species	Grade	Cm	Emod	Nu	Therm...	Dens...
1	DF/SPine	Com Species Group I DF,SP	No.1	☐	1	.3	.3	.035
2	HF/Spruce Fir	Com Species Group II HF,SPF	No.1	☐	1	.3	.3	.035
3	DF	Douglas Fir-Larch	No.1	☐	1	.3	.3	.035
4	SP	Southern Pine	No.1	☐	1	.3	.3	.035
5	HF	Hem-Fir	No.1	☐	1	.3	.3	.035
6	SPF	Spruce-Pine-fir	No.1	☐	1	.3	.3	.035
7	24F-1.8E DF Balanced	24F-1.8E_DF_BAL	na	☐	1	.3	.3	.035
8	24F-1.8E DF Unbalanced	24F-1.8E_DF_UNBAL	na	☐	1	.3	.3	.035
9	24F-1.8E SP Balanced	24F-1.8E_SP_BAL	na	☐	1	.3	.3	.035
10	24F-1.8E SP Unbalanced	24F-1.8E_SP_UNBAL	na	☐	1	.3	.3	.035
11	LVL-PRL Commercial 2250F	LVL_PRL_1.5E_2250F	na	☐	1	.3	.3	.035
12	LVL-PRL Commercial 2900F	LVL_PRL_2.0E_2900F	na	☐	1	.3	.3	.035

Species is the NDS wood species designation. This column has a drop down list where you can pick from NDS wood species as well as glulam types. If you add your own material species with the **Modify - Custom Wood Species**spreadsheet, you will find your new species name located between the NDS wood species and the glulam types.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Custom Wood Species**.

Grade is the NDS wood grade designation. This has a drop down list where you can select the appropriate grade.

The **Cm** checkbox determines if the wet service / moisture content factor should be applied. If you put a check in the Cm field, the appropriate factors will be applied to the allowable stresses and Young's Modulus (E), per the tables in the NDS supplement.

Emod is a factor that is applied to the Young's modulus modifier to reflect the Appendix F criteria.

Note:

- Refer to Custom Wood to specify a custom wood material.

Commercial Species Groups:

There are two new species listed that are not specifically shown in the NDS. These are the Commercial Species Group I - DF/SPine and Commercial Species Group II - HF/Spruce Fir. These are meant to be simplified groupings of the most commonly used wood species. It is meant to simplify the selection procession for wood member design in the United States.

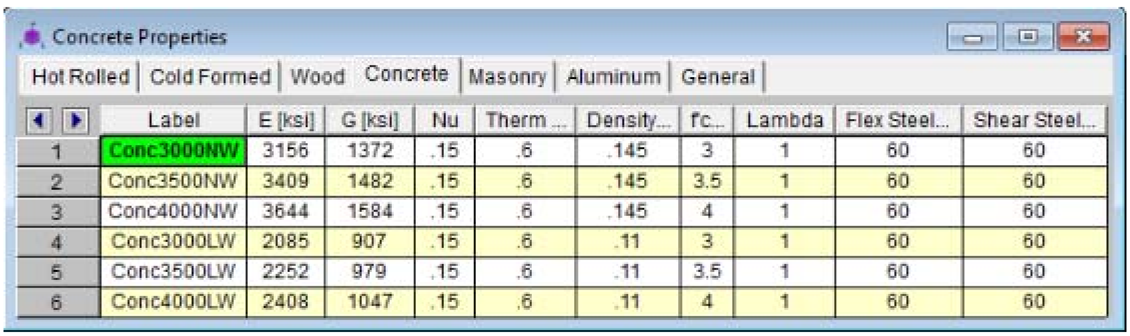
The design values for Group I take the NDS allowable stress values for two of the most widely used species (Doug Fir-Larch and Southern Pine) and uses the lower bound allowable stress value for each size and grade. To verify exactly what base allowable stress values were used, the user may review the properties in the Member Double-Click dialog.

Similarly, the design values for Group II take the allowable stress values for Hem Fir and Spruce-Pine-Fir and use the lower bound allowable stress values for each size and grade. To verify exactly what base allowable stress values were used, the user may review the properties in the Member Double-Click dialog.

These commercial grade species are currently only provided for NDS 05/08 wood design.

Concrete Specific Material Data

The **Concrete** tab records a few concrete specific material properties that do not exist for the other materials. These entries are described below:



	Label	E [ksi]	G [ksi]	Nu	Therm ...	Density...	f'c...	Lambda	Flex Steel...	Shear Steel...
1	Conc3000NW	3156	1372	.15	.6	.145	3	1	60	60
2	Conc3500NW	3409	1482	.15	.6	.145	3.5	1	60	60
3	Conc4000NW	3644	1584	.15	.6	.145	4	1	60	60
4	Conc3000LW	2085	907	.15	.6	.11	3	1	60	60
5	Conc3500LW	2252	979	.15	.6	.11	3.5	1	60	60
6	Conc4000LW	2408	1047	.15	.6	.11	4	1	60	60

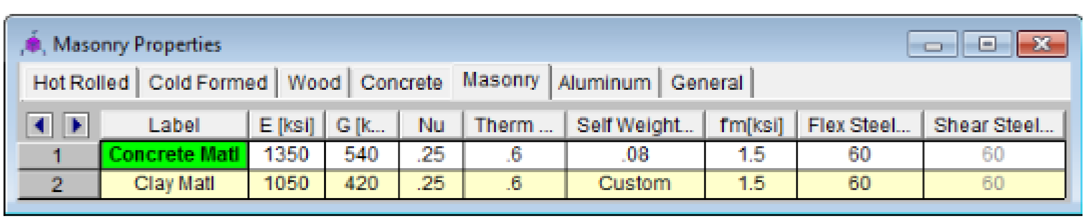
f'c is the concrete compressive strength used for concrete design.

Lambda, **λ** is the lightweight concrete modification factor. This factor only applies to the *ACI 318-08*, *ACI 318-11*, and *CSA A23.3-04* codes. For all other codes the Density value of the material determines any strength reduction. The program will automatically calculate the correct value if it is left blank. Only values between 0.75 and 1.0 will be considered.

Flex Steel is the reinforcement yield strength for flexural bars in members and vertical bars in walls. **Shear Steel** is the reinforcement yield strength for shear bars in members and horizontal bars in walls.

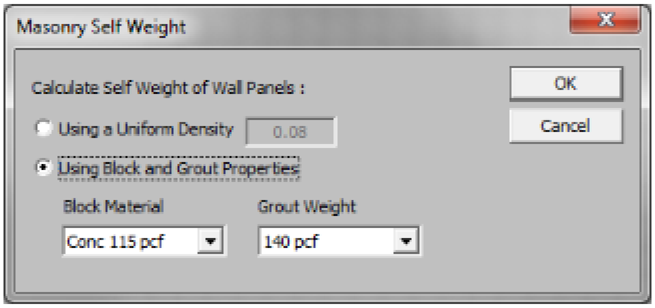
Masonry Specific Material Data

The **Masonry** tab records a masonry specific material property that does not exist for the other materials. This entry is described below:



	Label	E [ksi]	G [k...	Nu	Therm ...	Self Weight...	fm[ksi]	Flex Steel...	Shear Steel...
1	Concrete Matl	1350	540	.25	.6	.08	1.5	60	60
2	Clay Matl	1050	420	.25	.6	Custom	1.5	60	60

Masonry **Self Weight** can be accounted for by two different methods. Entering a number for the density will result in masonry walls which have a self weight equal to that density multiplied by the wall cross-sectional area. Otherwise, click within the cell to launch the masonry self weight dialog:



Set the Using **Block and Grout Properties** option to have the program automatically calculate the self weight of the wall using the weights from tableB3 of the Reinforced Masonry Engineering Handbook (RMEH). The self weight will be listed as **Custom**.

f'm is the masonry compressive strength used for masonry design.

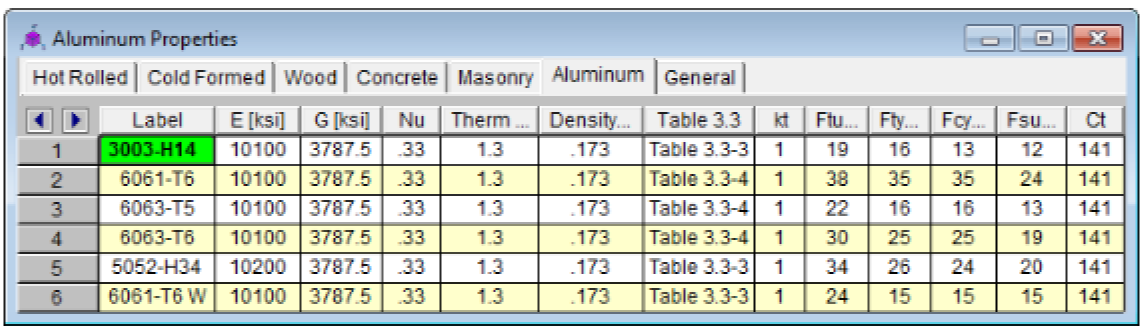
Flex and **Shear Steel** are the rebar yield strengths for flexural and shear bars used to reinforce the masonry.

Note:

- Masonry Shear Steel is automatically set to the Flex steel strength.

Aluminum Specific Material Data

The **Aluminum** tab records a number of aluminum specific material properties that do not exist for the other materials. These entries are described below:



	Label	E [ksi]	G [ksi]	Nu	Therm ...	Density...	Table 3.3	kt	Ftu...	Fty...	Fcy...	Fsu...	Ct
1	3003-H14	10100	3787.5	.33	1.3	.173	Table 3.3-3	1	19	16	13	12	141
2	6061-T6	10100	3787.5	.33	1.3	.173	Table 3.3-4	1	38	35	35	24	141
3	6063-T5	10100	3787.5	.33	1.3	.173	Table 3.3-4	1	22	16	16	13	141
4	6063-T6	10100	3787.5	.33	1.3	.173	Table 3.3-4	1	30	25	25	19	141
5	5052-H34	10200	3787.5	.33	1.3	.173	Table 3.3-3	1	34	26	24	20	141
6	6061-T6 W	10100	3787.5	.33	1.3	.173	Table 3.3-3	1	24	15	15	15	141

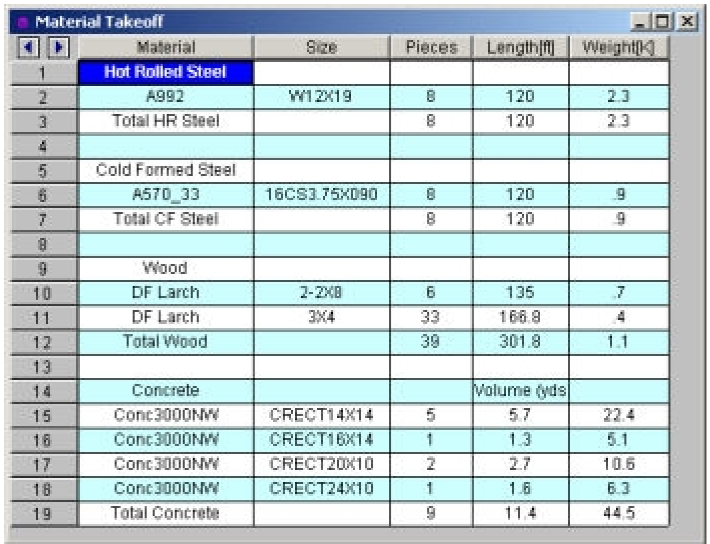
Table 3.3 is the table that refers to the Buckling Constants for a series of Temper designations. This column has a drop down list where you can pick from Table 3.3-3 equations or Table 3.3-4 equations.

$\mathbf{k_t}$ refers to coefficient for tension members listed in the ADM 05 Table 3.4-2.

Ct refers to Buckling Constant Intersection for Axial Compression in Curved Elements per Table 3.3-3 and Table 3.3-4 ADM 05.

Material Take-Off

Access the **Material Take-Off** spreadsheet by selecting it from the **Results** menu.



Material Takeoff

	Material	Size	Pieces	Length[ft]	Weight[K]
1	Hot Rolled Steel				
2	A992	W12X19	8	120	2.3
3	Total HR Steel		8	120	2.3
4					
5	Cold Formed Steel				
6	A570_33	16CS3.75X090	8	120	.9
7	Total CF Steel		8	120	.9
8					
9	Wood				
10	DF Larch	2-2X8	6	135	.7
11	DF Larch	3X4	33	166.8	.4
12	Total Wood		39	301.8	1.1
13					
14	Concrete			Volume (yds	
15	Conc3000NW	CRECT14X14	5	5.7	22.4
16	Conc3000NW	CRECT16X14	1	1.3	5.1
17	Conc3000NW	CRECT20X10	2	2.7	10.6
18	Conc3000NW	CRECT24X10	1	1.6	6.3
19	Total Concrete		9	11.4	44.5

This spreadsheet shows material takeoff information for each material and shape in the model. For each material the shapes are listed with the total length and weight. The total length and weight for each material is summed beneath the listing of shapes.

The length listed is the sum of the lengths of all the members assigned to this shape. Member end offsets are deducted from the length. The weight is the sum of the self-weight for all the members assigned to section set. This weight is calculated as Area * Density * Length for each member, again with offset distances deducted from the length.

For Concrete members, the volume of concrete is shown rather than the length of the members.

Notes:

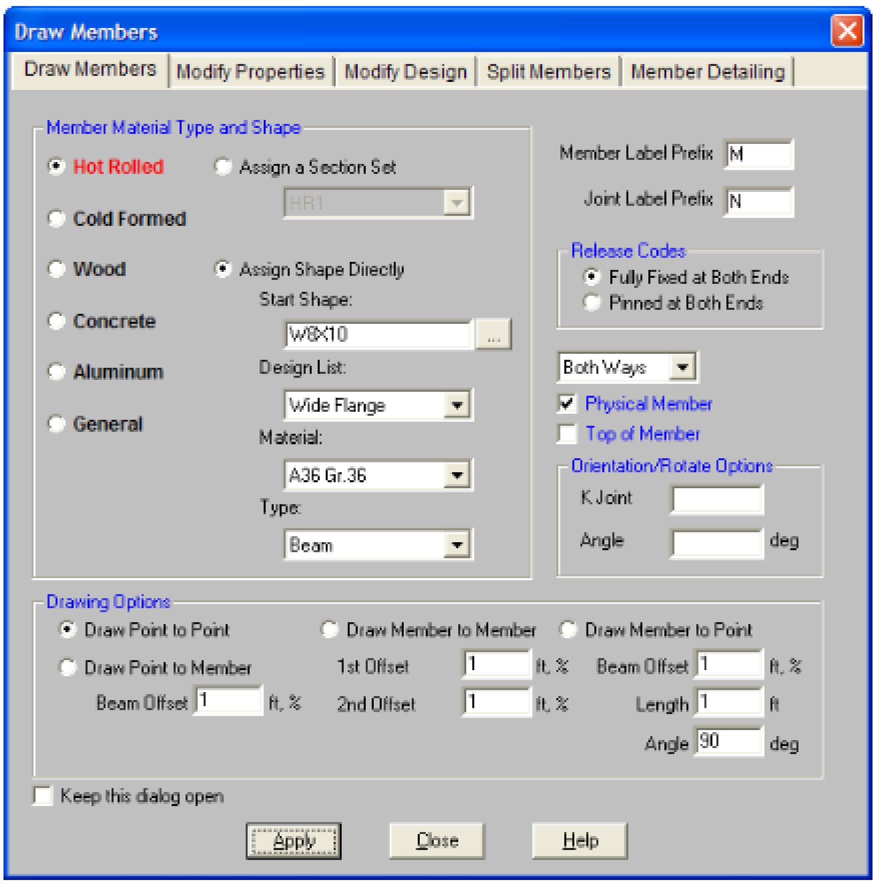
- This material takeoff report is independent of the loads applied to the model, i.e. the applied loads do not influence this report.
- Plates are not included in the Material Take-Off.

Members

RISA-3D uses a Physical Member that is automatically sub-meshed into general-purpose beam elements. With the use of the member End Releases you may define truss members or any other end condition for that matter. Member data may be viewed and edited in three ways: graphically in a model view, in the **Member Information Dialog**, or in the **Members Spreadsheet**. See Members Spreadsheet for descriptions of the member data. Design parameters for steel, wood or concrete design are recorded on the material tabs of the Members spreadsheet and are discussed in the Hot Rolled Steel - Design, Cold Formed Steel - Design, Concrete - Design, and Wood - Design sections.

Drawing Members

To create new members you can draw them using a Drawing Grid, Project Grid, or drawing "dot to dot" clicking existing joints. You can set all of the member properties up front or you can modify these properties after you draw the member. Graphically modifying properties is discussed in the next section.



The Member Drawing Options are as follows:

Draw Point to Point

Use the first mouse click to select the start point of the new member. Use the second mouse click to select the end point of the new member.

Draw Point to Member

Use the first mouse click to select the start point of the new member. Use the second mouse click to select an existing member which would contain the end point of the new member. 'Beam Offset' is the offset of the end point of the new member from the start point of the existing member.

Draw Member to Member

Use the first mouse click to select an existing member which would contain the start node of the new member. Use the second mouse click to select an existing member which would contain the end point of the new member. '1st Offset' and '2nd Offset' are the location offsets of the start and end points of the new member from the start points of the existing members.

Draw Member to Point

Use the mouse click to select the existing member which would contain the start node of the new member. The new member would have a length equal to 'Length'. The new member would be created at an angle 'Angle' to the projection of the existing member on the horizontal plane (plane perpendicular to the vertical axis).

To Draw Members

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. If you are not drawing between existing joints, you will need to create a drawing grid or define joints on the Joint **Coordinates Spreadsheet**.
3. Click the button and set the member properties. For help on an item, click and then click the item.
4. Click **Apply** to start drawing members by clicking on joints, members or grids with the left mouse button. The coordinates of the closest joint or grid intersection to your cursor are displayed in the lower right hand corner of the status bar.

   The first click will define the **I-end** of the first member. The second click, and each click thereafter, will define the **J-end** of the first member and also the **I-end** of the next member so that you may continue to draw as if your pencil is down. To "pick up" the pencil, click the right mouse button. You may then start drawing somewhere else with the left button.

   The new members will be shown on screen and will be recorded in the **Members Spreadsheet**. For help on an item, click and then click the item.

5. To stop drawing altogether click the right mouse button or press the Esc key.

Note

- Press CTRL-D to quickly recall the last dialog accessed and make changes.
- You may also specify or edit members in the **Members Spreadsheet**.
- You may undo any mistakes by clicking the **Undo** button.

Modifying Member Properties

There are a number of ways to modify members. You may view and edit the member data in the Members Spreadsheet. You may double-click a single member to view and edit its properties. You can use the **Modify Members** tool to graphically modify a possibly large selection of members.

The graphical **Modify Members** tool discussed here lets you modify the properties of members that already exist in your model. To use this, you will typically specify parameters you want changed, then select the members that you want to modify. You can modify members one at a time by selecting the **Apply by Clicking Individually** option and then click on

the members you wish to modify. You may also modify entire selections of members by selecting the members in the model view first and then use the **Apply to All Selected** option. See the Graphic Selection topic for more on selecting.

To Modify Members

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Click the **Draw/Modify Members** button and select the **Modify Properties** tab.
3. Set the parameters for the new members. Check the **Use?** box for the items that are to be applied. For help on an item, click and then click the item.
4. You may choose to modify a single member at a time or an entire selection of members.

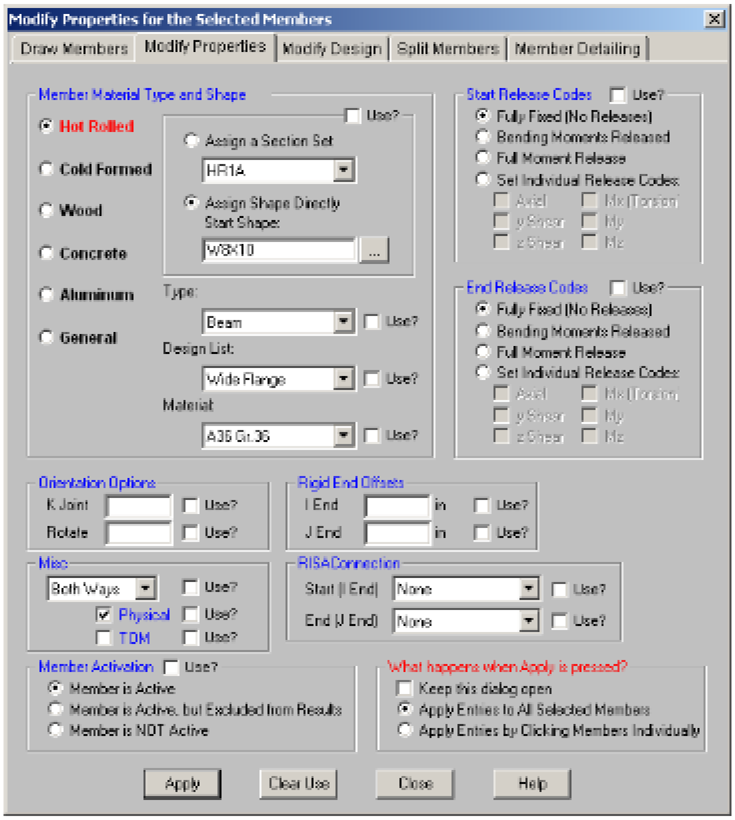
   To modify a few members, choose **Apply Entries by Clicking Items Individually**, click **Apply**, and click on the members with the left mouse button.

   To modify a selection of members, choose **Apply Entries to All Selected Items** and click **Apply**.

Note

- To modify more members with different parameters, press CTRL-D to recall the **Modify Members Dialog**.
- You may also modify members in the **Members Spreadsheet**.
- To modify one member you may double click that member to view and edit its properties.
- To relabel members first sort the **Members Spreadsheet** into the desired order, select the **Tools Menu** and choose **Relabel Members**.
- You may undo any mistakes by clicking the **Undo** button.

The parameters shown are nearly the same as those used to define new members. For help on an item, click and then click the item.



The **Use?** check boxes next to the data fields indicate whether the particular parameter will be used or not when the modification is applied. If the box next to a field is checked, that parameter will be applied to any selected members, if the box is NOT checked, the parameter will NOT be applied, even if a value is entered in the field. This lets you easily change one or two properties on members without affecting all the rest of the properties. Note that if no value is entered in a field (I.e. the field is blank) and the corresponding check box is checked, clicking **Apply** will have the effect of clearing the data for these fields. For example, to remove all member end-releases for the currently selected members, just erase any text in the I and J release fields, click their **Use?** check boxes so that they're checked, and then click on **Apply**.

Material and Cross Section Properties

Member material and cross section properties may be assigned to a member in one of two ways – either directly by specifying the shape and material explicitly, or by assigning a Section Set to the member. Section Sets allow you to control properties for a group of members that share the same properties. Combining section sets with the Member Redesign feature gives you great control over how new member sizes are picked and what members get updated. Section sets must be used when the desired shape is not in the database or when steel redesign (optimization) is desired. See Section Sets for more information.

All member properties may be assigned graphically either as you draw or later as a modification to the members.

Modifying Member Design Parameters

You can graphically specify the Hot Rolled, Cold Formed, Wood, Aluminum, or Concrete design parameters such as unbraced lengths and K factors. See Steel Design, Cold Formed Steel Design, Concrete Design, Aluminum Design, and Wood Design for information on the design parameters themselves.

To use this, you will specify the appropriate member parameters, then select the members that you want to modify, and then click on the "Apply" button. Alternately you can set the parameters and then choose the "Click to Apply" option, which allows you to then click on the members you want modified.

To Modify Member Design Parameters

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Click the **Draw / Modify Members** button and select the **Modify Design** tab.
3. Set the design parameters for the members. Check the **Use?** Box for the items to apply.
4. You may choose to modify a single member at a time or an entire selection of members.

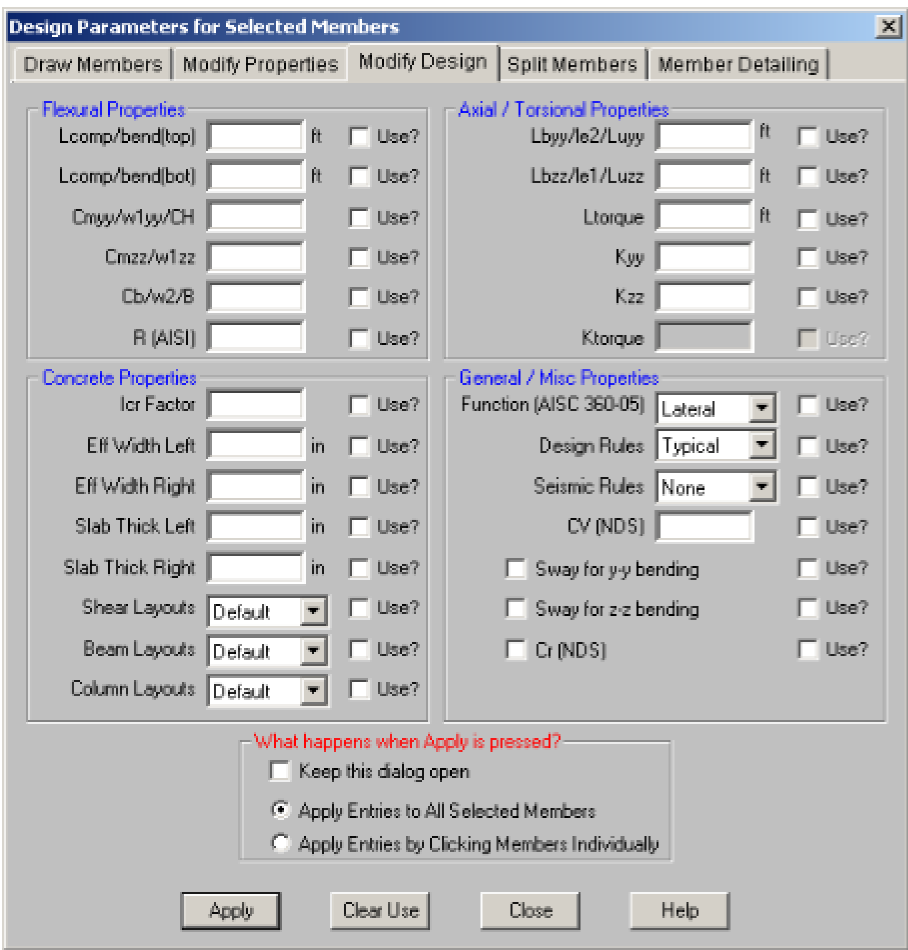
   To modify a few members choose **Apply Entry by Clicking Items Individually** and click **Apply**. Click on the members with the left mouse button.

   To modify a selection, choose **Apply Entries to All Selected Items** and click **Apply**.

Note

- To modify more members with different parameters, press CTRL-D to recall the **Modify Members** settings.
- You may also specify or edit design parameters in the **Material** tab of the **Members Spreadsheet.**
- You may undo any mistakes by clicking the **Undo** button.

The parameters shown are the same as those on the Hot Rolled, Cold Formed, Wood, Aluminum, or Concrete tab of the Members spreadsheet.



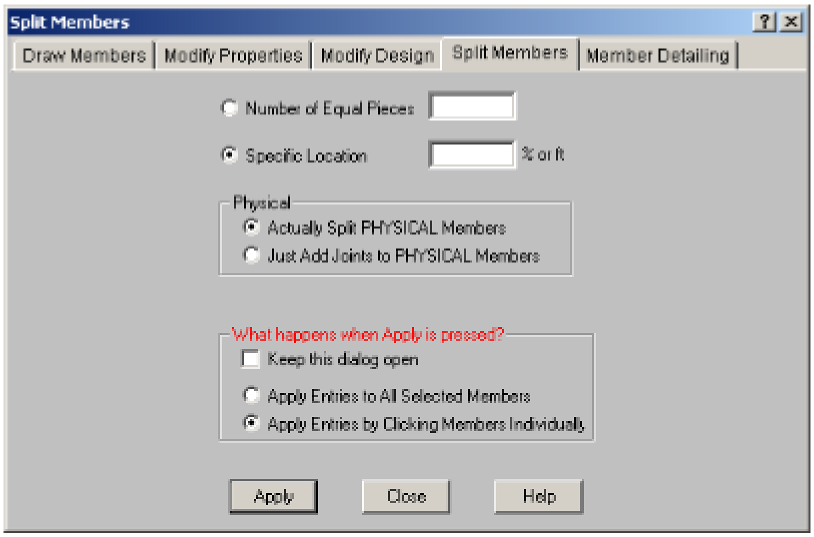
The **Use?** check boxes next to the data fields indicate whether the particular parameter will be used or not when the modification is applied. If the box next to a field is checked, that parameter will be applied to any selected members, if the box is NOT checked, the parameter will NOT be applied, even if a value is entered in the field. This lets you easily change one or two properties on members without affecting all the rest of the properties. Note that if no value is entered in a field

(I.e. the field is blank) and the corresponding check box is checked, clicking "Apply" will have the effect of clearing the data for these fields. For example, to remove all member end-releases for the currently selected members, just erase any text in the I and J release fields, click their **Use?** check boxes so that they're checked, and then click on "Apply".

Splitting Members

You can split members into a number of equal length pieces or you may split into two members at a specified location from the I joint. You may specify the split location as a length or a percentage of length. For example specifying %50 will split the members at the midpoints.

You may choose between **Add Joints to Physical Members** and **Actually Split Physical Members**. For Physical members it is likely that you will not want to actually split the member but prefer instead that joints be placed along the member so that you may specify other members, plates, loads or boundary conditions.



If you use both parameters at the same time, each parameter will be applied independently to the original member. For example if you specify '3' equal pieces and also specify a split at '%50', you will get a member that is broken up in 3 places. There will be 2 breaks at its third points, and also a break that would be at the middle (50% of the length) of the original member.

To Split Members

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Click the **Draw / Modify Members** button and select the **Split Members** tab.
3. Set the parameters for splitting the members. For help on an item, click and then click the item.
4. You may choose to modify a single member at a time or an entire selection of members.

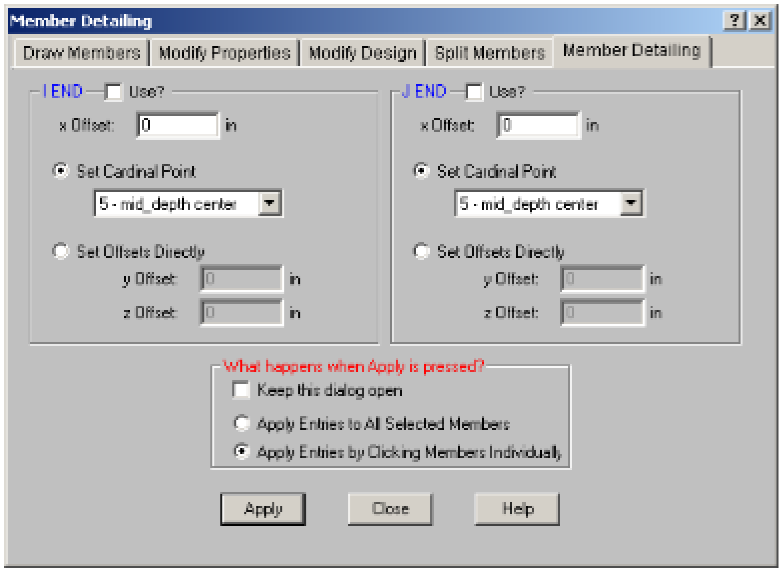
   To modify a few members choose **Apply Entry by Clicking Items Individually** and click **Apply**. Click on the members with the left mouse button.

   To modify a selection, choose **Apply Entries to All Selected Items** and click **Apply**.

Note

- To modify more members with different parameters, press CTRL-D to recall the **Split Members** settings.
- You may undo any mistakes by clicking the **Undo** button.

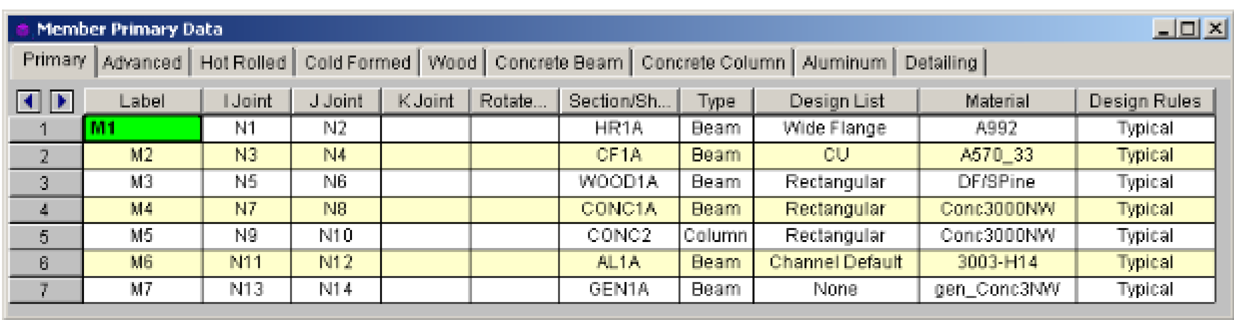
Member Detailing



The Member Detailing tab in RISA-3D is only used to export data via the CIS/2 translator for use in detailing software. For more information on this, see the help file for the Cardinal Points.

Members Spreadsheet - Primary Data

The **Members Spreadsheet** records the properties for the member elements and may be accessed by selecting **Members** on the **Spreadsheets Menu**.


Member Primary Data

Primary | Advanced | Hot Rolled | Cold Formed | Wood | Concrete Beam | Concrete Column | Aluminum | Detailing

	Label	I Joint	J Joint	K Joint	Rotate...	Section/Sh...	Type	Design List	Material	Design Rules
1	M1	N1	N2			HR1A	Beam	Wide Flange	A992	Typical
2	M2	N3	N4			CF1A	Beam	CU	A570_33	Typical
3	M3	N5	N6			WOOD1A	Beam	Rectangular	DF/SPine	Typical
4	M4	N7	N8			CONC1A	Beam	Rectangular	Conc3000NW	Typical
5	M5	N9	N10			CONC2	Column	Rectangular	Conc3000NW	Typical
6	M6	N11	N12			AL1A	Beam	Channel Default	3003-H14	Typical
7	M7	N13	N14			GEN1A	Beam	None	gen_Conc3NW	Typical

The following data columns hold the **Primary** data for the members:

Member Labels

You may assign a unique **Label** to any or all of the members. You can then refer to the member by its label. Each label has to be unique, so if you try to enter the same label more than once you will get an error message. You may relabel members at any time with the **Relabel Members** option on the **Tools Menu**.

Member End Joints

The **I-Joint** and **J-Joint** entries define the start (I-joint) and end (J-joint) locations of the member. The member local axes are defined based on these joints. See Member Local Axes for more information.

K-Joint and X-axis rotation

These two parameters may be used separately or together to define the rotation of a member. See Member Local Axes and Defining Member Orientation for more information.

Section / Shape

If you are explicitly assigning shapes to each member, then enter the database shape you wish to use for the member. You can select this by clicking on the arrow in the cell. Alternatively, you may choose a section set to represent the section properties, material properties, and re-design parameters.

Member Type

If you are explicitly assigning shapes to each member, then you may enter the **member type** that you wish to use. The choices are *Column, Beam, Vertical Brace, and Horizontal Brace*. This member type will affect the detailing rules that apply to your concrete columns and beams. It will also allow you to use design lists that were developed specifically for that member type.

Note

- If you are using **Section Sets** to define your member, then the information in this field will be generated automatically based on the referenced section set.
- In concrete design member types affect the code required minimum flexural steel and shear tie requirements.
- In load attribution, HBrace and VBrace members do not receive load from applied member area loads. See the Member Area Load section for more detail.

Design List

If you are explicitly assigning shapes to each member, then you may enter the **design list** type that you wish to use. This entry will affect the members that are available to program when it is suggesting alternate or optimized shapes. Refer to Design Optimization for more information on the member optimization procedure. Also refer to Appendix A – Redesign Lists for information on creating or editing these lists.

Note

- If you are using **Section Sets** to define your member, then the information in this field will be generated automatically based on the referenced section set.

Material

If you are explicitly assigning shapes to each member, then you may enter the **material** that you wish to use. You can select this by clicking on the arrow in the cell.

Note

- If you are using **Section Sets** to define your member, then the information in this field will be generated automatically based on the referenced section set.

Design Rules

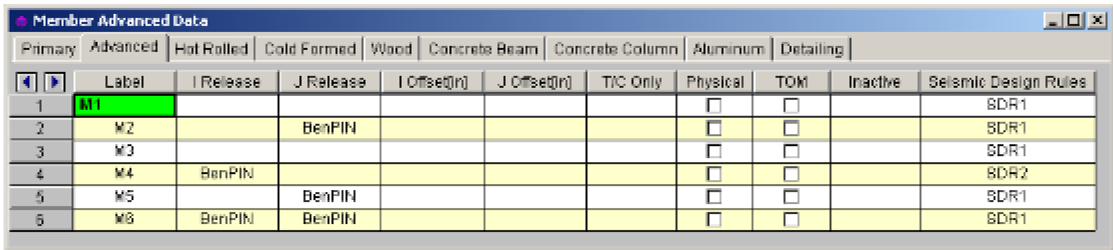
If you are explicitly assigning shapes to each member, then you may enter the **design rules** type that you wish to use. When the program is checking alternate or optimized shapes, it will restrict its selections to members that obey the chosen design rules. Refer to Design Rules– Size / U.C. for more information.

Note

- If you are using **Section Sets** to define your member, then the information in this field will be generated automatically based on the referenced section set.

Members Spreadsheet - Advanced Data

The **Advanced** tab records the properties for the member elements and may be accessed by selecting **Members** on the **Spreadsheets Menu**.



	Label	I Release	J Release	I Offset[in]	J Offset[in]	T/C Only	Physical	TOM	Inactive	Seismic Design Rules
1	M1						☐	☐		SDR1
2	M2		BenPIN				☐	☐		SDR1
3	M3						☐	☐		SDR1
4	M4	BenPIN					☐	☐		SDR2
5	M5		BenPIN				☐	☐		SDR1
6	M6	BenPIN	BenPIN				☐	☐		SDR1

The following data columns hold the **Advanced** data for the members:

I and J Releases

Releases control the forces that may be resisted by a member. You may use these to define pinned connections, truss members, and any other end condition. See Member End Releases for more information.

I and J End Offsets

End offsets may be used to model a rigid end zone for a member. See Member End Offsets for more information.

T/C Members

The **T/C** field is used to indicate that a member is to be Tension or Compression only. When a member is flagged as **C**, any members it will only be able to take compressive loads. The member will have no stiffness to resist tensile loads. When a member is flagged as **T**, the member will only be able to take tension loads. When a section is flagged as **E**, the member will primarily take only tension loads, however it will also take some compression load, up to its Euler buckling load.

Physical

This box is checked if the member is a Physical Member. See Physical Members for more information.

TOM – Top of Member

Members may be automatically offset to align their tops for situations that make each member centerline unique to its depth. You may define the members with joints located at their top and specify the TOM flag to have them rigidly offset downward a distance half of their depth. See Member Top Offset to learn more.

Inactive Members

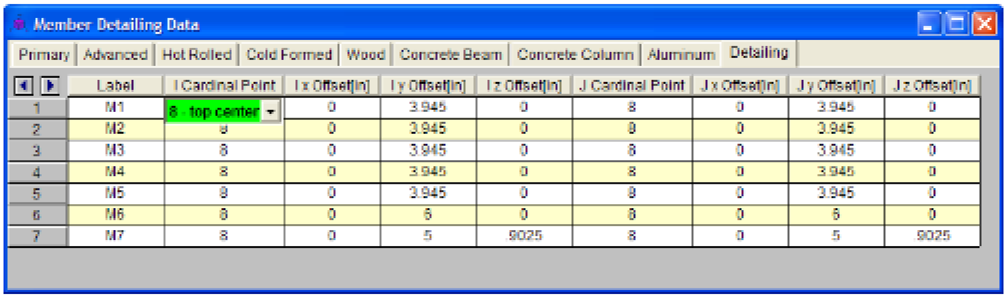
Members may be removed from the solution without deleting them from your model by making them inactive. See Inactive and Excluded Items for more information.

Seismic Design Rules

This entry may be used to assign a Seismic Design Rule to each member individually. This can be left as **None** if you are not including seismic detailing in your design. This entry will only apply to Hot Rolled Steel members.

Members Spreadsheet - Detailing Data

The **Detailing** tab records the detailing data for the members that are necessary for full 2-way data transfer between RISA and steel detailing packages. For more information on this subject refer to the Help file for the RISA CIS/2 Translator which can be downloaded from our website.



Tension/Compression-Only Members

Members defined as *Compression Only* elements will have no stiffness to resist tensile loads. Members defined as *Tension Only* elements will have no stiffness to resist compressive loads. Members defined as *Euler Buckling* will behave as two way members for any loads that are less than 52% of their Euler buckling load. If the compression load exceeds this load, then the *Euler Buckling* member will be removed from the model.

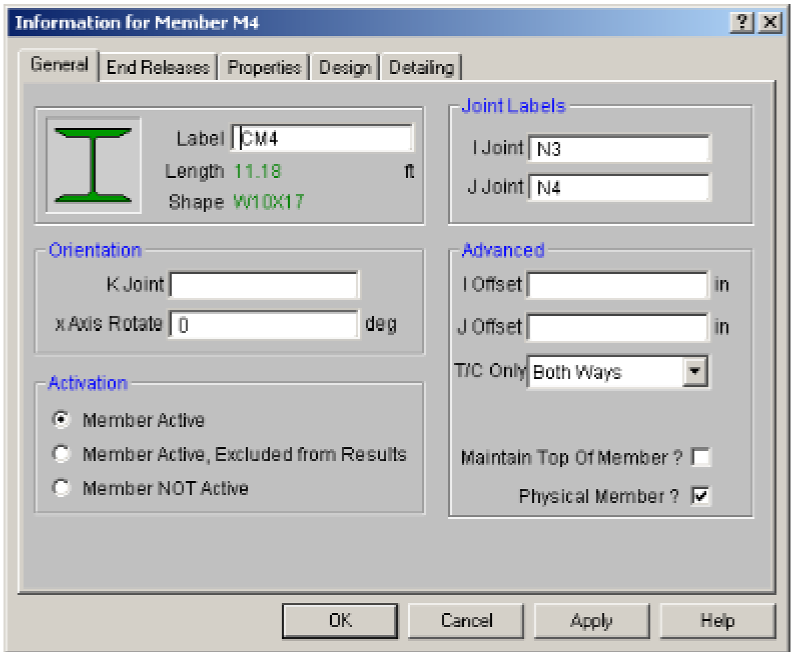
Letting the tension only members take a little compression helps model convergence immensely. You can control the amount of compression that a Euler member takes by increasing or decreasing the Kl/r ratio that is used to calculate the Euler buckling load. The best way to alter the Kl/r ratio is to modify the Lb parameters for that member on theHot Rolled, Cold Formed or NDS Wood tabs of the **Members** spreadsheet. The larger the Kl/r ratio, the less compression that a tension only member will take.

Note

- For tension-only and compression-only members, normal force/stress code checks are restricted to axial forces/stresses only in tension or compression respectively. Flexural effects are not included. Shear effects are considered in separate code checks for shear.
- When a model contains T/C only members, the program must iterate the solution until it converges. Convergence is achieved when no more load reversals are detected in the T/C only members. During the iteration process, each T/C only member is checked. If any members are turned off (or turned back on), the stiffness matrix is rebuilt and model is resolved. For models with lots of T/C only members, this can take a bit longer than a regular static solution.

Member Information Dialog

Just as with the joints and plates you may double-click any member to view it's properties. All of the same information that is stored in the **Members** and **Member Design** spreadsheets is displayed for the member you choose, and may be edited. This is a quick way to view and change member properties. For large selections of members however the spreadsheet and graphic editing tools may be the faster solution.



General - This tab let's you view edit the member's label and end joints. The rigid end offsets, orientation, and activation state can also be viewed and modified here. The shape pictured will change depending on the shape type of the member. It will also change to reflect wood members. If the member is made inactive here, you will need to activate the member from the Members spreadsheet, or by using the Criteria Select feature to find and select inactive members.

End Releases - This tab allows you to view and modify the end releases for the member. It also allows you to define Connection Rules for your hot-rolled steel connection design to be used in conjunction with RISAConnection. See the RISAConnection Integration topic for more information.

Properties - This tab lets you view and edit the member's section set. The type of material information displayed will change depending on whether the section set material properties are defined on the General Materials spreadsheet.

Design - This tab allows you to view and edit the design properties for the member. The types of design information will change depending on the material type of the member and the code setting on the Global Parameters window.

Detailing - This tab is specifically for you with the CIS\2 translator export application. THIS TAB DOES NOT AFFECT ANALYSIS

Note

- It's generally more efficient to use the Graphic Editing features if you want to change the properties for many members at once.

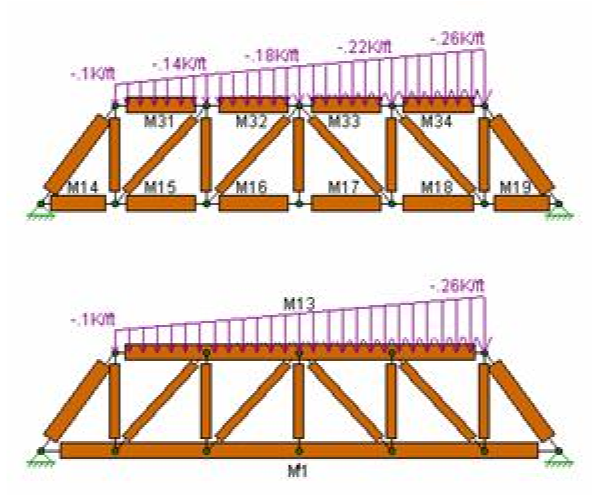
Physical Members

Physical Members provide fixity to all joints that occur along the length of the member, without breaking that member into multiple smaller members. You may use the physical member feature to avoid defining one 'field' member with multiple members in your model. This saves time in building and editing your model and in understanding your results.

To define a Physical Member, check the Physical Member box in the **Set Member Properties** settings or in the **Modify Member Parameters** settings. You may also specify Physical Members in the **Members Spreadsheet** and in the **Member Information** settings.

To understand the benefits of Physical Members see the trusses in the following figure. The first truss shows the chords modeled without physical members and thus with **multiple** members modeling each chord. The second truss models each chord with **one** Physical Member. Both models will yield the same results. However the Physical Member model is more intuitive and easier to work with because you don't have to work with multiple members when you create the chords, load

them, edit them or evaluate their results. Notice that the distributed load may be defined from start to end for the Physical Member. In the other model the distributed load end magnitudes have to be specified for each of five members in order for the entire load to be defined.



Continuous beams, multi-story columns and truss chords are examples of continuous 'field' members that can be modeled with one Physical Member. You may define them from start to finish without having to explicitly define their connection to other elements or boundaries through intermediate joints. Subsequently, when it comes time to make changes to a member you can edit the properties of one Physical Member rather than the multiple members that might otherwise represent it.

Physical Members are also effective in managing results because the results for one Physical Member are reported together in the results spreadsheets and the member detail report. With the truss example above the Physical Member model allows you to look at results for one member making it easier to track maximum forces and other values.

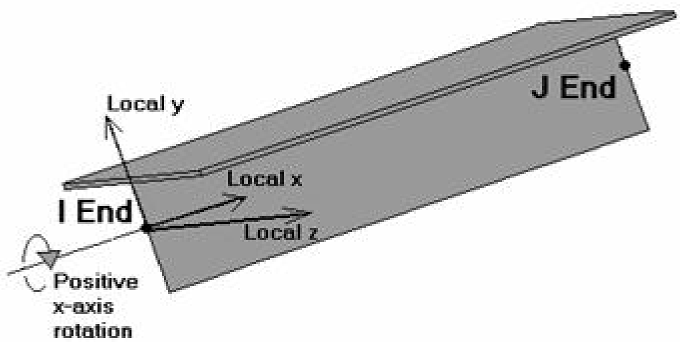
Note

- You may convert a Physical Member into multiple members by removing the Physical Member property and then performing a Model Merge to split the member for connectivity. See Model Merge for more information.
- You cannot convert multiple members into one Physical Member. To achieve this you must delete the multiple members and define a new Physical Member.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **What are Physical Members**.

Member Local Axes

The following diagram illustrates the directions of the member's local axes that are used to define member forces, stresses, and deflections as well as loads defined in local axes directions:



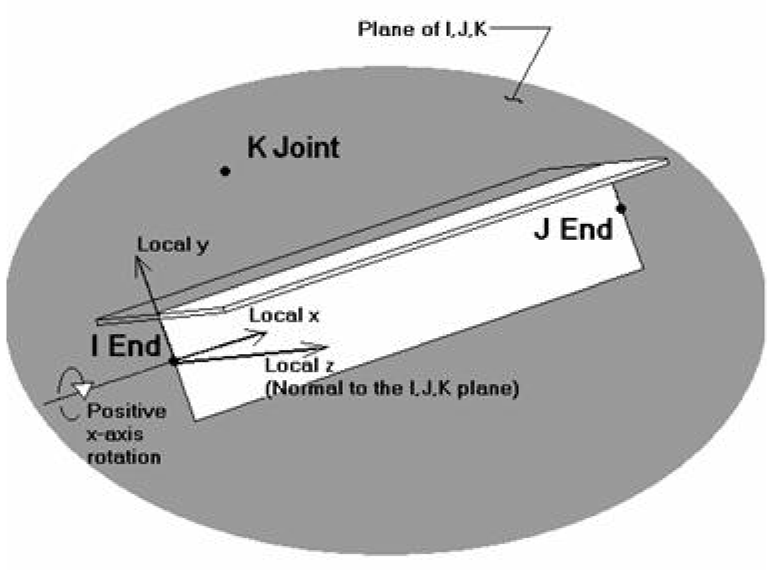
As can be seen from the diagram, the local x-axis corresponds to the member centerline. The positive direction of this local x-axis is from the I-joint towards the J joint. The complicated part is defining the orientation for the local y and z-axes. Of course, we only have to define the direction for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

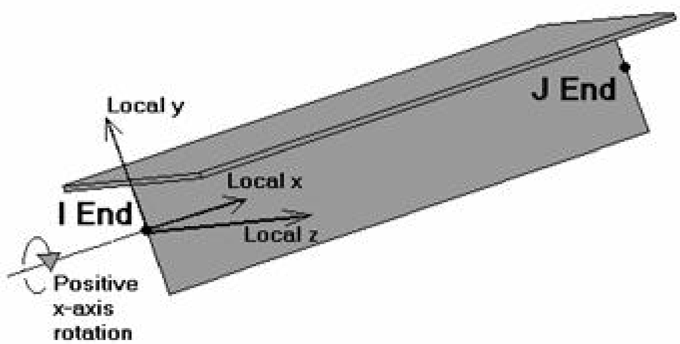
If you do not explicitly define the orientation for a member, the default is for the member's local z-axis to lie in the global X-Z plane or as near as possible. If the member is defined in the global Y-direction, the member's local y and z axes both lie in the global X-Z plane, so the local z-axis is made parallel to the global Z-axis. This works well for models with the Y-axis as the vertical axis because any beam members are typically oriented such that vertical loads are resisted by the strong axis bending of the member. You can change the default orientation of the members with the **Global Parameters**. For example if your vertical axis is the Z-axis then you can specify the default so that vertical loads are resisted by the strong axis bending of the member.

Defining Member Orientation

RISA-3D provides two ways to explicitly set the orientation of the y-axis. The first is by rotating the member about the local x-axis. This member rotation is entered in the x-Axis Rotate column on the Member spreadsheet or may be specified in the graphic editing tools. For this rotation, positive is counter-clockwise about the x-axis, with the x-axis pointing towards you.

The second way to explicitly define the orientation is by specifying a K joint for the member. If a K joint is defined the three joints (I,J,K) entered for the member are used to define a plane. This plane is the plane of the member's x and y-axes. The z-axis is defined based on the right hand rule using the x and y-axes. See below:





Note

- To be sure of a member's orientation you can always view the rendered shape of a member by clicking the button on the **Window** toolbar.

Member End Releases

The **I Release** and **J Release** fields are used to designate whether the forces and moments at the ends of the member are considered fixed to or released from the member's points of attachment (the I and J joints). Each member has 6 force components at each end (axial, y-y & z-z shear, torque, and y-y & z-z bending). Any or all of these force components can be released from the member's point of attachment. If a force component is released, that force is not transferred between the joint and the member.

To specify member releases go to the **I Release** or **J Release** field for the member in the Members Spreadsheet on the Advanced Tab, click the button, and specify the condition.

Alternatively, you may specify the end condition by directly typing in the field. To indicate that a force component is released, put a **X** for that component in the release field. You can move within the release field using the space bar which will result in a **O** for no release.

RISA-3D has two special "keyword" release configurations built-in. These are:

AllPIN => Mx, My, Mz (all moments) released (OOOXXX)

BenPIN => My, Mz (bending only) released (OOOOXX)

These keyword entries are included because 99% of the release configurations you'll ever want to define will be one of these two (98% will be BenPIN). You can call out the keyword entry by just entering the first letter of the keyword. So if you go to a release field and enter "bp", the keyword "BenPIN" will be filled in automatically.

Note

- RISA-3D will not allow you to release the member torsion at both ends. This is because it will be unstable as it would be free to spin about its centerline. For this reason, pinned end conditions should be modeled using the "BenPIN" entry instead of "AllPIN".

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **End Release**.

Top of Member Offset

The Member Top Offset is an offset applied perpendicular to the member which acts to make the top of the member line up with the end joint elevation (i.e. the end joint elevation becomes the “top of the member”). When the TOM flag is checked on the Advanced tab of the Members spreadsheet, RISA-3D first calculates the distance from the member centroid to its extreme top fiber (in the current configuration) then offsets the member that distance in the negative Global vertical direction.

This feature reflects the fact that members may be aligned by their faces rather than by their centerline elevations such as when you have a W10 floor joist framing into a W24 girder. Members with this flag set will be offset downward (in the negative vertical direction) from their joint endpoints. The distance of the offset will be the distance from the centroid to the top of the beam. For example, a beam in strong axis bending will be offset half its depth. A beam turned 90 degrees so that it is in weak-axis bending will be offset by half its width. For singly-symmetric sections such as single and double angles and WTs, RISA-3D will correctly identify the member orientation and vertical offset distance necessary to properly align the member’s “top” edge. This offset affects the member stiffness and results. All member results for offset members will be reported at the ends of the member, not at the joint locations.

Note

- Top of member offsets only apply to members that are running perpendicular to the vertical direction and have their local y or local z axes parallel to the vertical direction.
- Combining top of member offsets with rigid elements such as boundary conditions, diaphragms and rigid links often gives undesirable behavior such as large axial loads in the offset members.
- Using this feature on braced frames (or on any beams that have a diagonal member framing into them) will introduce eccentric moments and axial loads into the beam and is generally not recommended.
- Top of member offsets can not be used on tapered members. You may wish to provide rigid links similar to the automated way it is being done, but be aware of the limitations.

Member End Offsets

Member offsets reflect the fact that the member ends may not be attached at the centerline of the member being attached to. For example, a beam connected to the flange of a column is offset from the centerline of the column. The distance of the offset would be ½ the depth of the column.

You may enter explicit offset distances or have them calculated automatically. To enter offsets explicitly simply enter the value of the offset. To have the offset calculated, enter the non-numeric label of the member whose depth defines the offset distance.

For example, say your member is framing into the flange of a 12" deep column. The offset distance would be 6", so you would enter '6' for the offset. Now, if that column gets changed to a 14" shape, you would have to go back and change the offset distance to 7". This can be time consuming if you have many offsets.

If instead the column has a label of M100, specifying M100 as the member offset causes the offset to be calculated as half of the depth of the member M100. For the W12 column the offset would be 6" and when the column is changed to a W14 the offset becomes 7".

Note

- When you use a member label to define the end offset, this value is ALWAYS taken as simply the depth of the member. Thus, if you are framing at an angle into a member or into the weak axis of a member you should not use this feature.
- When the model is solved the member length is adjusted in the stiffness matrix by the offset distance resulting in a shorter, stiffer member. Also the results listed for members with offsets do take into account the offset distances. The I-end and J-end results are the results at the offset locations, and the report locations are determined by dividing the member length minus the offset distances by the **Number Of Sections** on the **Global Parameters**.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keyword: **Offsets**.

Inactive and Excluded Items

Making an item such as a member or plate inactive allows you to analyze the structure without the item, without having to delete the information that defines it. This leaves data intact so the item may be easily reactivated. This is handy if you want to try a frame with and then without certain items, without having to actually delete the data.

Putting a “y” in the **Inactive** field makes the item inactive, i.e. the item is not included when the model is solved or plotted.

Another option is to put an “E”. The “E” code means include the item in the solution, but exclude it from the results list. So, an item with an “E” in the “Inactive?” field will be treated like any other member in the solution and plotting of the model, but the member will not be listed in the solution results (forces, stresses, deflections, etc.). This is useful if there are certain items whose results you're not interested in. You don't have to clutter up the results with these items and can concentrate on the items you're most interested in. See Printing for more limiting printed results.

Note

- When making members inactive you may need to update the unbraced lengths of the adjacent members.

Member Shear Deformations

Including shear deformation models the effects of shearing forces on the lateral displacements of the members. Shear deformation is included in the analysis by checking the appropriate box on the **Global Parameters**.

Shear deformation effects are based on the material shear modulus (G) and the shear area.

Including shear deformation causes the member stiffness matrix to be modified by the term M, where:

$$M := \frac{12 \cdot E \cdot I}{G \cdot A_s \cdot L^2}$$

Where A_s = Area * Shear Deformation Factor

These shear area coefficients will be automatically calculated for all standard members, but must be entered for arbitrary sections. While these shear deformation factors are always less than 1.0, they can vary from 0.85 for solid rectangular shapes, to 0.88 for solid circular shapes, to 0.53 for thin pipes. These coefficients are used strictly for shear deformation effects and should not to be confused with shear stress factors described in the next section. A good reference for the calculation of both of these coefficients is *Stress, Strain, and Structural Matrices* by Walter D. Pilkey.

For members whose length is much greater than the depth, shear deformation has a relatively minor impact. When the length of the member is less than 10 times its depth, shear deformation begins to have a significant impact on the solution. Keep this in mind if you are creating models where members are being broken up into several pieces because the length used to calculate the term M is the joint-to-joint member length.

Note

- Shear deformation effects are included for the joint deflections only and not for the internal member deflections.
- Shear deformation can play a significant role in the stiffness of the member and thus the results.

Member Shear Stresses

Since shear stress is not equally distributed over a cross section, it is unconservative to take the maximum shear stress as simply the beam shear (V) divided by the beam's cross sectional area. To achieve more accurate results you must reduce the shear area by a shear stress factor, $\alpha_{stress} := \frac{\tau_{avg}}{\tau_{max}}$

where:

$$\tau_{max} := \frac{V \cdot Q}{I \cdot t}$$

and

$$\tau_{avg} := \frac{V}{Area}$$

These shear stress factors will be automatically calculated for all standard members, but must be entered for arbitrary sections. While these shear area coefficients are always less than 1.0, they can vary significantly. Common values include 0.67 for solid rectangular shapes, and 0.75 for solid circular shapes.

Note

- For wide flange and channel shapes, RISA will replace this calculation with the shear stress calculations required by the Steel Code specifications.

Torsion

A twisting of the member induces torsional forces and stresses. The primary reference used in the development of RISA-3D's torsional calculations was **Torsional Analysis of Steel Members**, available from the AISC. The equations used for torsional stresses won't all be repeated here, but they can be found in the reference. RISA-3D models warping members using CASE 2, as shown in the Torsion reference.

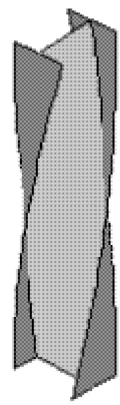
Note:

- Torsion considerations for any other cases can be grossly over or underestimated. RISA's calculation of torsion capacity is based on torsion due to racking of the structure, not point or line torques on an individual member. That type of check is not supported.

It is more accurate to consider warping effects when calculating member stiffnesses and stresses, but there is a way you can turn off these effects. On the **Global Parameters**, you'll see the checkbox to **Include Warping**. If this box is not checked warping effects will not be considered, i.e. stress and stiffness calculations for wide flanges and channels will be done just like all the other shapes (k = GJ/L). You may wish to do this to compare the RISA-3D results with and then without warping, or to compare RISA-3D results with a program that does not include warping.

Warping

A primary consideration in the calculation of torsional properties and stresses is whether the cross section is subject to warping. Solid cross sections are NOT subject to warping. For RISA-3D, all closed shapes such as pipes and tubes are considered to be NOT subject to warping. Cross sections composed of rectangular elements whose centerlines all intersect at a common point are NOT subject to warping. Examples are Tee shapes and angle shapes. For simplification, double angle cross sections are also assumed to be not subject to warping. So, the only shapes RISA-3D considers subject to warping effects are wide flanges and channels (I's and C's). The importance of this extends beyond the stress calculations, however. Warping considerations also impact the calculation of torsional stiffness for these shapes.



For a nonwarping member or a warping member with warping unrestrained, the member's torsional stiffness is given by:

$$k := \frac{G \cdot J}{L}$$

G = Material Shear Modulus

J = Cross Section Torsional Stiffness

L = Member Length

For a member subject to warping, if the warping of the member is restrained its torsional stiffness is:

$$k := \frac{G \cdot J}{A \cdot \left[\tanh\left(\frac{L}{2A}\right) \cdot \cosh \cdot \left(\frac{L}{A}\right) - \tanh\left(\frac{L}{2A}\right) + \frac{L}{A} - \sinh\left(\frac{L}{A}\right) \right]}$$

$$A := \sqrt{\frac{E \cdot C_w}{G \cdot J}}$$

C_w = Cross Section Warping Constant

E = Material Modulus of Elasticity

L = Torque Length

Thus restraining the warping effects for a cross section subject to warping (I's or C's) makes the shape much stiffer in torsion. Think of it this way; if you twist a wide flange, the flanges want to warp. If you restrain the flanges from warping its much harder to twist the wide flange as it is stiffer in torsion.

Member Releases

If a member is released for any rotational degree of freedom at either end, warping is not considered for that member. For example, if you model a wide flange member with a "BenPIN" release code (at either or both ends), warping would not be considered for that member. This is because any connection that doesn't resist bending moments is certainly not going to restrain warping. RISA-3D does not consider the effect of warping "pins" at this time.

Warp Length

The "warp length" is the length between points of torsional restraint (or release). This may be equal to or greater than the member's actual joint to joint length. This warp length is calculated automatically by RISA-3D and is used for the member's torsional stiffness and stress calculations. Each member's warp length is shown on the member detail report (see Member Detail Report).

Physical members that are subject to torsion will always have their warp length be at least the length of the physical member. Members framing in along the length of the physical member are assumed to not reduce the warp length.

Note that the calculation of the warp length by RISA-3D can be "fooled" by beams that are modeled by several segments. The warp length used for each member will be the length of each straight-line segment rather than the whole length. Using Physical Members rather than member segments will avoid this. This is not possible however when modeling curved members with straight line segments.

Warping Pins

A member that is subject to warping effects, like a WF or channel shape, will still experience warping stresses, even if warping restraint is not provided at the ends of the member. RISA-3D currently does NOT consider any warping effects for members that have warping "pinned" end conditions. The addition of warping effects for members with warping pins will be addressed in a future program version.

Torsional Stresses

RISA-3D calculates and lists the torsional stresses for the members of the structure, including the warping stresses.

Pure torsional shear is calculated for all non-warping "open section" shapes based on the equation:

$$\tau := \frac{M_x \cdot t}{J}$$

M_x = Torsion Moment

t = Maximum Thickness of Any Part of the Cross Section

This is the only torsional stress calculated for non-warping shapes. Shape types that are not "open cross sections" will have their shear stresses calculated with equations that are appropriate for each type.

For warping shapes (I's and C's), three separate stresses are calculated: pure torsion shear, warping shear and warping normal (bending) stresses. These results are all listed for review. The equations used to calculate these values won't be listed here but they are contained in the reference.

Code Check

These torsional stresses are included when the AISC code check (ASD or LRFD) is calculated for the member. The shear stresses (pure torsion and warping) are included in the shear check, and the warping normal stresses are added to the weak

axis bending stresses for calculation of the combined code check. By “weak axis bending stresses”, we mean the bending stresses produced by moments about the local y-axis.

When including shear stresses from torsion in the shear code check, the program uses the worst case of the torsional shear stresses (top and bottom flanges or web) and combines that with the actual shear stresses due to pure flexure. This is intentionally conservative in cases where the worst case torsional shear occurs in the flange, but the worst case flexural shear occurs in the web.

Applied Torsional Loads

You may apply member point torques along the length of the member. However, these loads are intended to work only for non-warping members. The reason is that we (RISA Technologies, LLC) have not yet found the time to work out the derivatives necessary to properly handle these member point torques when applied to warping members. The calculations for non-warping members is quite simple, but for warping members they're complex. This may be added in a future version.

Cardinal Points

Overview

A detailing layer has been built for both 3D and Floor that lets the user set the "true" elevations and locations start/end, top of steel, etc...) for all members. For each member (both column and beam in the case of Floor), a new data structure has been added to describe the connection point at member ends.

The areas in RISAFloor and RISA-3D where member detailing parameters can be added/viewed are:

- RISAFloor
    - Beams Data Entry Spreadsheet>Detailing tab
    - Columns Data Entry Spreadsheet>Detailing tab
    - Plot Options>Beams/Columns/Walls tab
    -
- RISA-3D
    - Members Data Entry Spreadsheet>Detailing tab
    - Double-clicking Individual Member>Detailing tab
    - Graphic Member Drawing>Member Detailing tab
    - Plot Options>Members tab

For each end of the member, both cardinal point positions and decimal local offsets are used to described the connection location. The location of the cardinal point is plotted as follows:

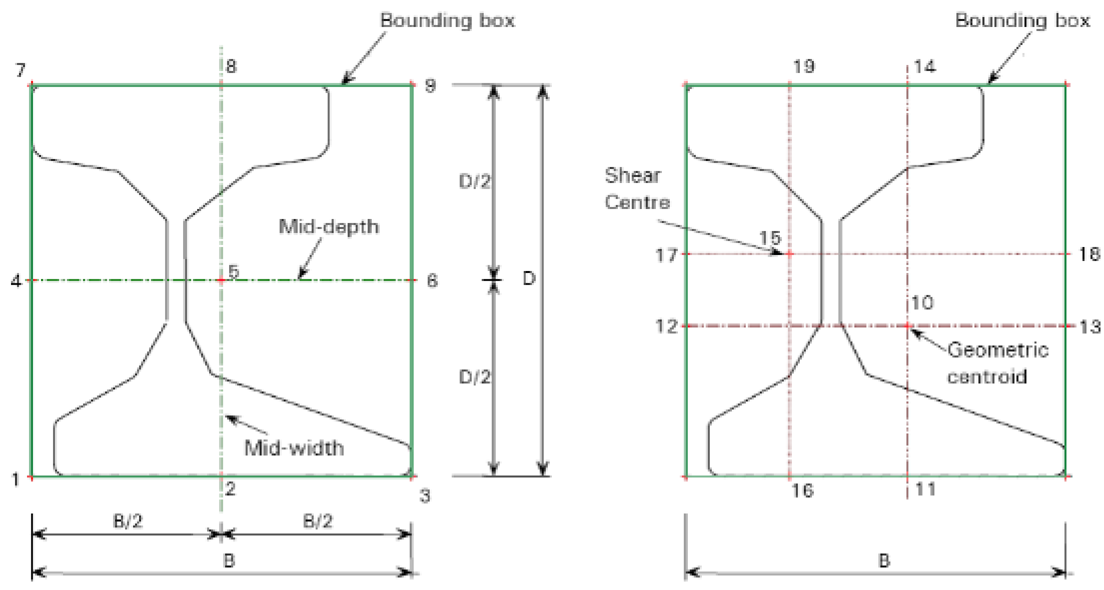


Figure 1 Cardinal point Location and Local Axis

Cardinal point 1:lower left corner of the member section bounding box
Cardinal point 2:lower center point in the member section bounding box

Cardinal point 3:lower right corner of the member section bounding box
Cardinal point 4:mid-depth left point in the member section bounding box
Cardinal point 5:mid-depth center point in the member section bounding box
Cardinal point 6:mid-depth right point in the member section bounding box
Cardinal point 7:upper left corner of the member section bounding box
Cardinal point 8:upper center point in the member section bounding box
Cardinal point 9:upper right corner of the member section bounding box
Cardinal point 10:Geometric centroid of the member section

Note:

- Only cardinal point 1-10 are supported by RISA. You may instead use an offset.

The x, y, and z offsets are based on the local axis of the member. The x local axis is defined along the member from I to J. It coincides with the geometric centroid of the member section. The standard cardinal point positions (1-10), as well as the decimal local offsets are both supported in current detailing definition. If the cardinal point is set, the y and z offset values will be automatically calculated and filled in the data structure. If there is a situation that doesn't match a cardinal point (angle brace as an example) you can also set the y and z offsets directly.

Note:

- The TOM and Rigid End Offset are completely different concepts from the detailing offset. The purpose of the detailing offsets is more realistic visualization and plotting of the model. The detailing offsets are not considered during the analysis. In the analysis, all members are still connected at their geometric centroid. TOM and Rigid End Offsets, on the other hand, affect the load distribution and design of the member. They are analysis parameters. In our program, the TOM / Rigid End Offset are completely separate data structure from the detailing offsets. When TOM is set for a model, for example, the detailing information will NOT be automatically updated to cardinal point 8 accordingly.

Detailing Input and Modification:

There are three ways for setting and modifying the detailing layer information. User can use the graphical member modify, the double-click dialog and the member detailing spreadsheet to set the detailing information.

Default

When a member is drawn, by default, x offsets are set to be 0 on both ends. If the member type is beam , by default the y and z detailing offsets will be on cardinal point 8 (top center) for both ends. If the member type is column, by default the detailing offset will be on cardinal point 10 for both ends.

Graphical member modify

A **Member Detailing** tab has been added to the **Graphical member draw/modify** window. The user can choose to “set y, z offset using cardinal point” or “set y, z offsets directly”. When a cardinal point is picked, the value in the “y Offset” and “z Offset” text box will automatically be updated.

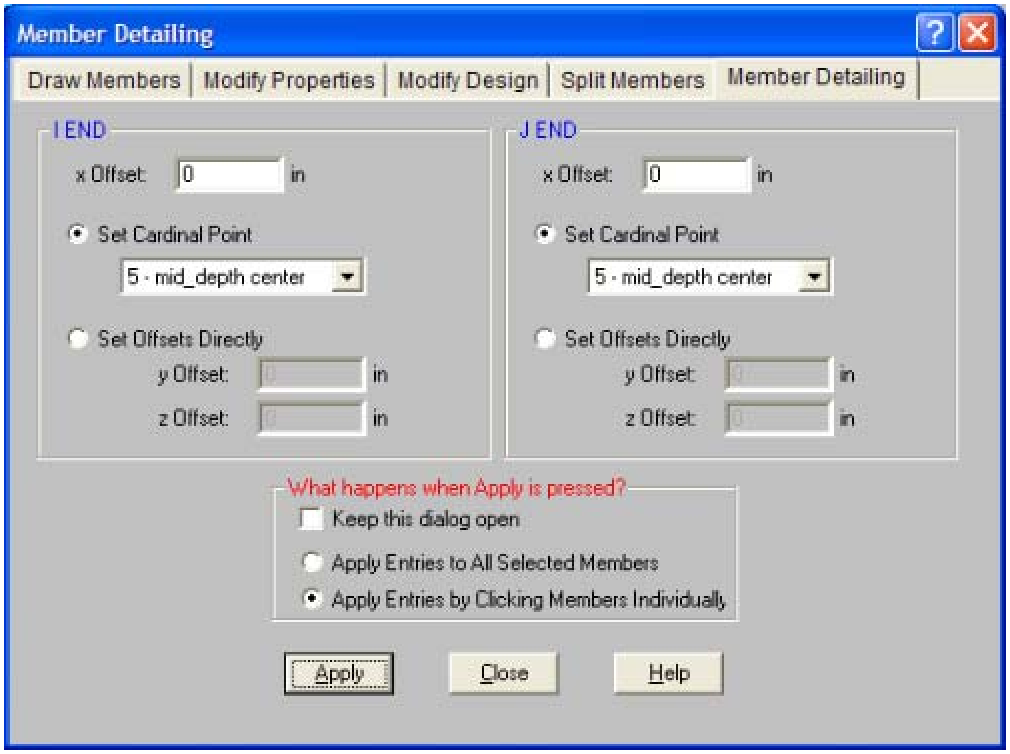


Figure 2.A Member Detailing Tab in Graphical Member Modify (3D)

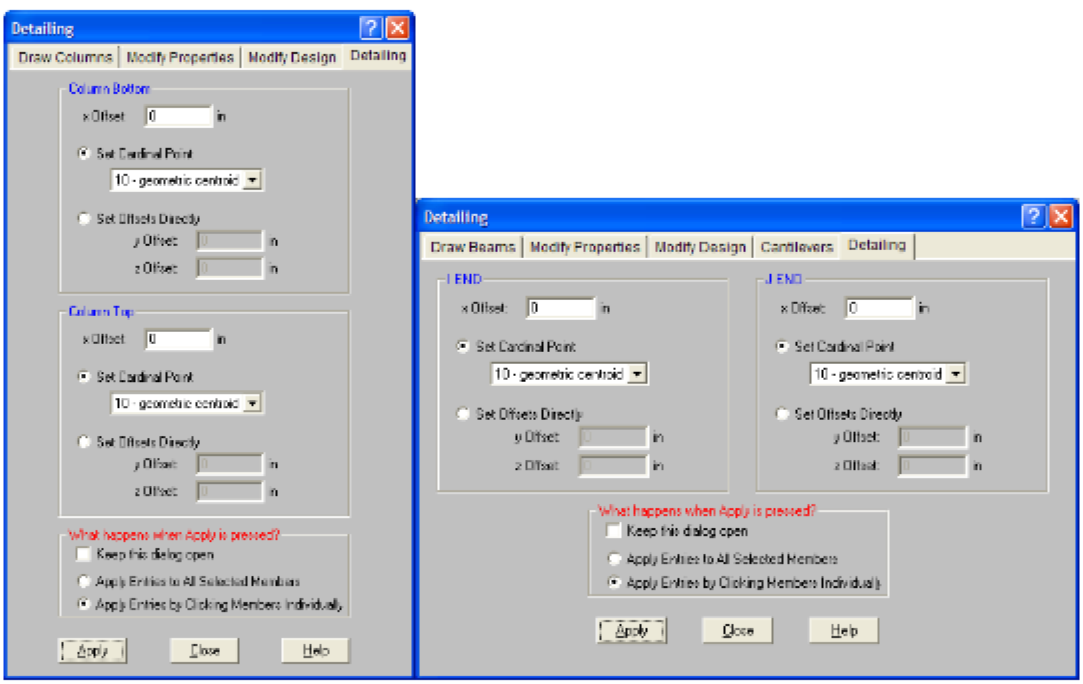


Figure 2.B Column/Beam Detailing Tab in Graphical Member Modify (FLOOR)

Double click dialog

A **Detailing** tab has been added to the double click dialog for the displaying and modification of the detailing info. The user can choose to "set y, z offset using cardinal point" or "set y, z offsets directly". When a cardinal point is picked, the value in the "y Offset" and "z Offset" text box will automatically be updated.

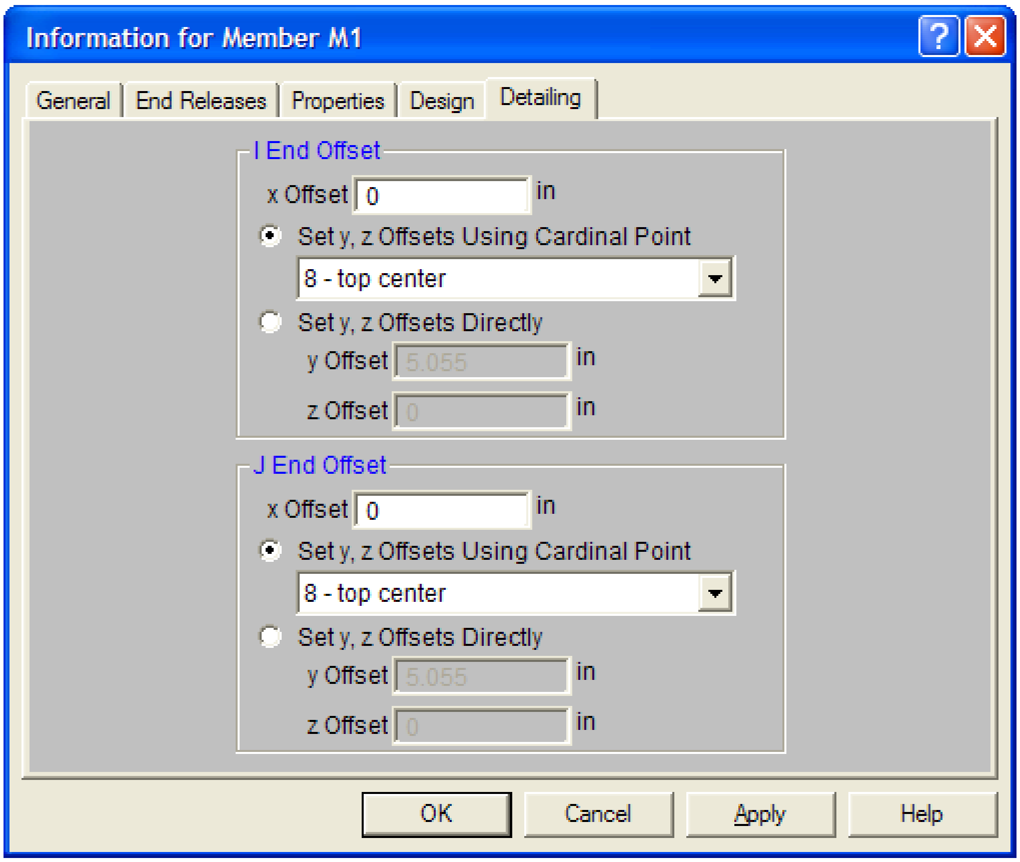


Figure 3 Detailing Tab in double click dialog

Spreadsheets

A detailing tab has been added for the member spread sheet (Column and Beam grid in the case of FLOOR). The spreadsheet shows a label and 8 values for each member. The "I cardinal point" and "J Cardinal Point" are pull down list with numbers and a description. When user picks a cardinal point, the y and z offset values will update automatically based on the current shape. When a new y and z offset are input by the user and it doesn't fall on any cardinal point, the "Cardinal Point" will automatically change to "None". When the "Member Detailing" grid is active, a toolbar button is added to recalculate values based on new shapes (Figure 5).

Note:

- When the design shape of the member is changed, the detailing offsets of the member is not automatically updated. The recalculate tool bar button needs to click to get new y, z offset with respect to the cardinal point picked)

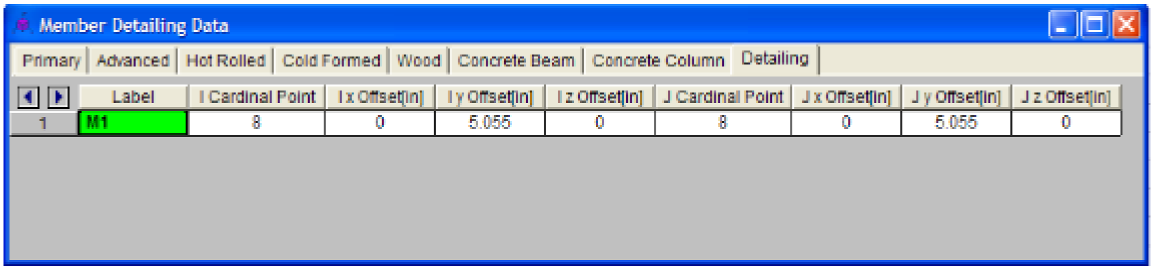


Figure 4.A Detailing Tab in Member Spreadsheet(3D)

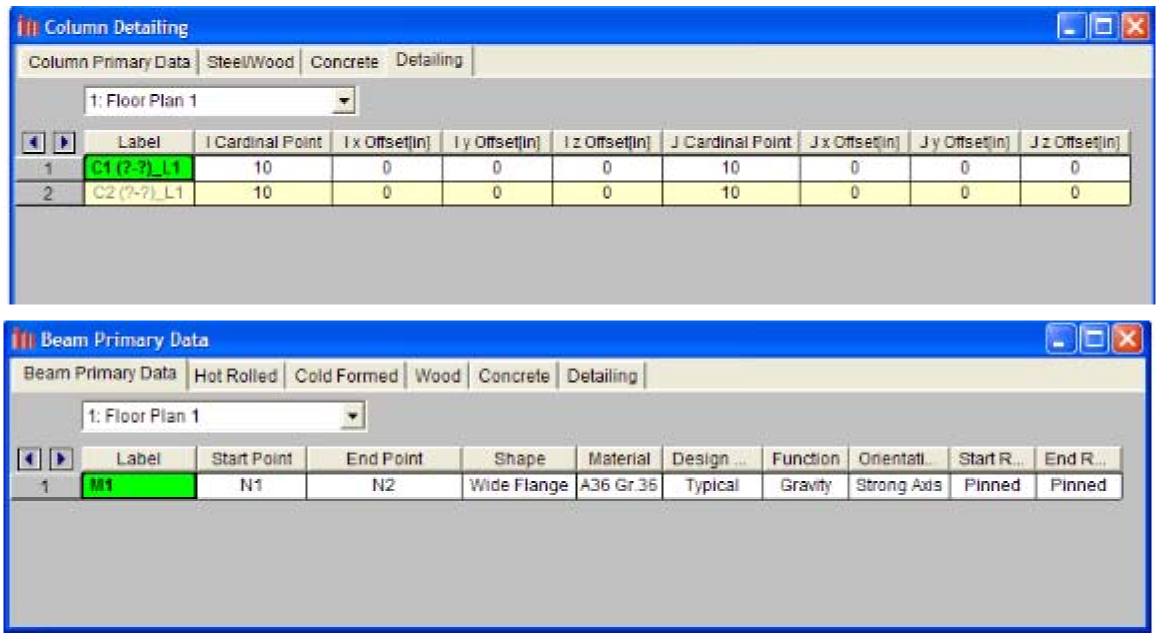


Figure 4.Beam/Column Detailing Tab in Member Spreadsheet(FLOOR)

Figure 5 Recalculate Detailing Offset button when Member Detailing Spreadsheet in Active

Visualization of the Detailing Layer

To visualize the model with the detailing layer, go to the plot option dialog box and member tab. Check the “Detailing Info” check box (Figure 6).

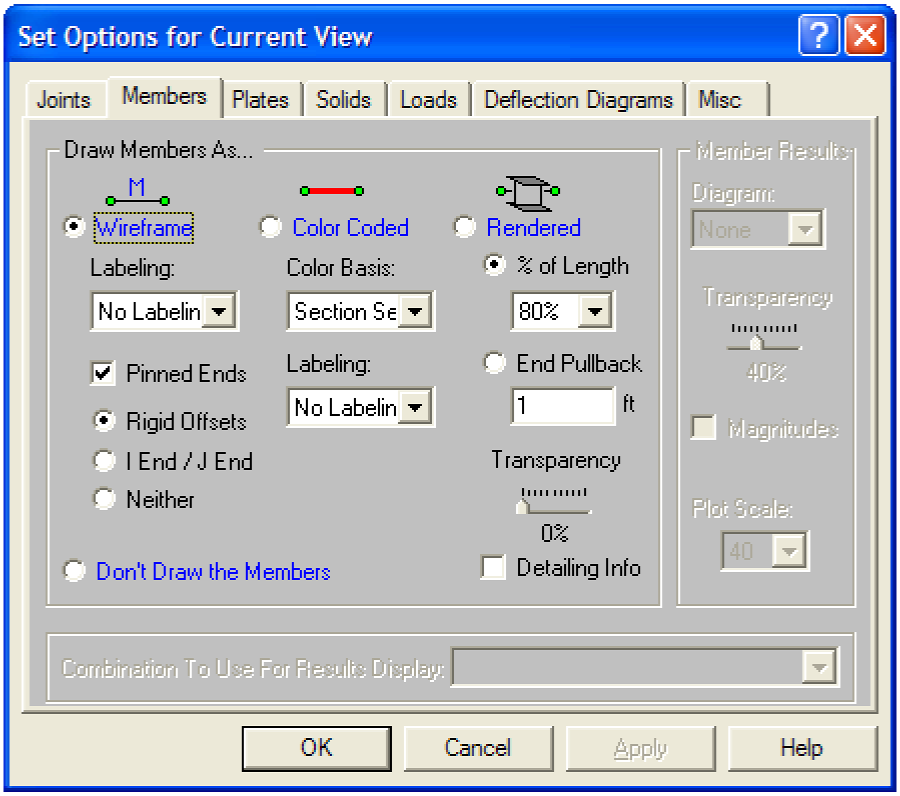


Figure 6.A Detailing Info checkbox in plot option window (3D)

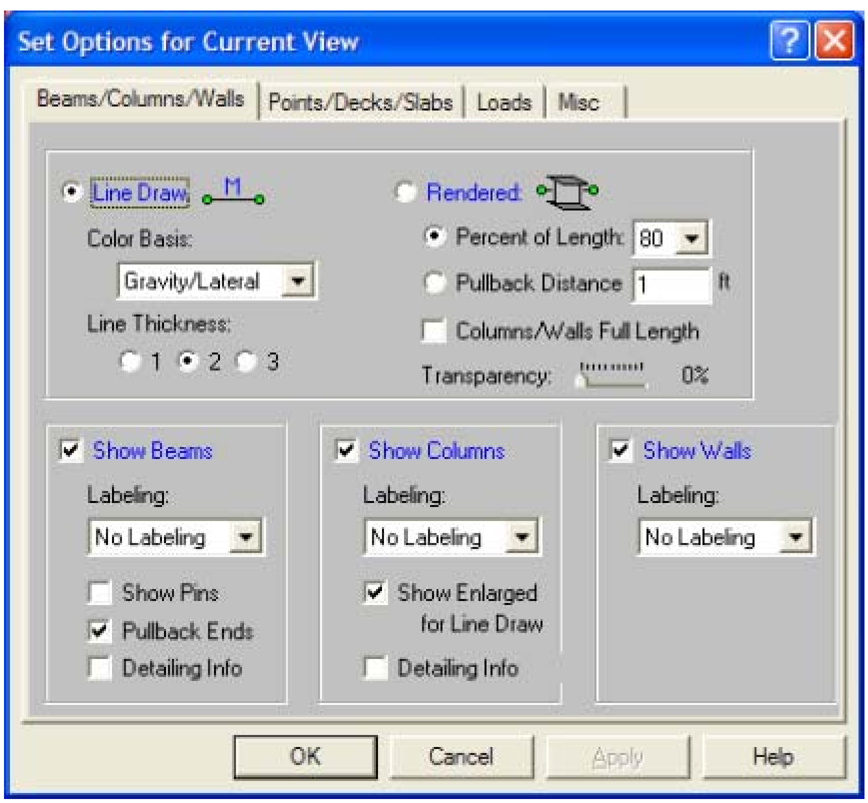


Figure 6.B Detailing Info checkbox in plot option window (FLOOR)

Note:

- When the Detailing info box is checked. The members will be plotted solely using the x, y and z offsets. The “TOM” and “Rigid End Offset” will not be considered in the plotted here.

File I/O

[.MEMBERS_DETAILING_DATA]data section is added to the .R3D file to hold the member true location data. [..RF_COLUMNS_DETAILING_DATA] and [..RF_BEAMS_DETAILING_DATA] data section are added to the .RFL file to hold member true location data. The CIS2 translator is updated to read/write this new data section.

Members - Results

When the model is solved, the results are separated into material specific design results and generic results. The generic member results are discussed in this section. The material specific design results are discussed in the following sections of the manual: Hot Rolled Steel - Design, Concrete - Design, Cold Formed Steel - Design, and Wood - Design. For information on **Member Detail Reports** see the Results section.

Number of Reported Sections

Note that the member results (forces, stresses, code checks) are only reported at the section locations. For example, if you set the **Number Of Sections** in the **Global Parameters Dialog** to be '2', you will not get any results for the middle of your member, you will only get results for the end points. If you have a point load applied to your member at a location that is not a section location, you will probably not report the maximum moment in the section if it does not occur at an endpoint.

To Adjust the Number of Sections

- In the **Global Parameters Dialog**, select the **Solution** tab.
- Adjust the **Number of Sections** as needed.

Note

- Adjusting the number of sections affects the amount of output.

Number of Internal Sections

Internally, the program subdivides the member into equally spaced sections to calculate forces, stresses, code checks, etc. The number of **Internal Sections** can be adjusted in the **Global Parameters Dialog**. If this value were set to 100, this would mean that for a member that is 100ft long RISA-3D will calculate these values at approximately every foot. These values are then used in the member steel, wood, and concrete code checks, the diagrams in the model view, and in the detail reports. The locations of the maximum code checks are reported at a distance from the I-joint.

To Adjust the Number of Internal Sections

- In the **Global Parameters Dialog**, select the **Solution** tab.
- Adjust the **INTERNAL Sections** as needed.

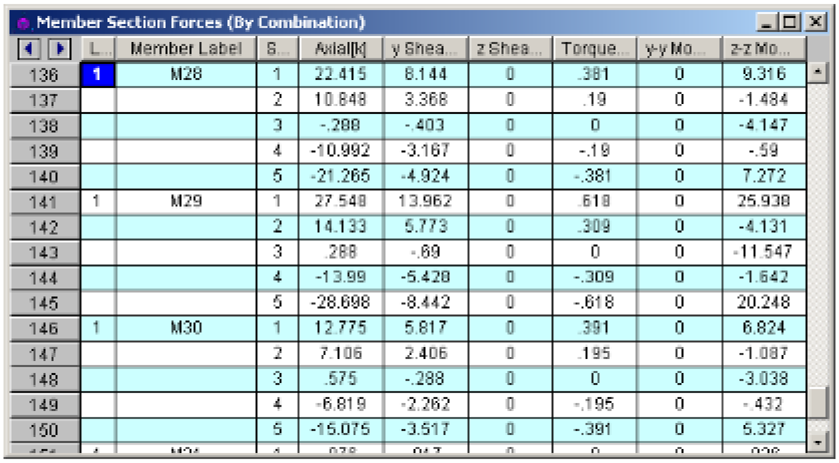
All other results are reported at the **Number of Sections** that you specify in the **Global Parameters Dialog**. This controls how many places you receive **reported** or **printed** member force, stress, torsion, and deflection results. These locations are also equally spaced so setting the value to 5 will give you 5 equally spaced sections; at the ends, the middle and the quarter points.

Note

- Adjusting the number of **Internal Sections** will not affect the amount of output in the results spreadsheets.
- You may want to stick with odd numbers for the **Number of Sections**. Setting the **Number of Sections** to an even number will not report forces/stresses at the midpoint of the member, which is often the location of maximum moment.
- Setting the **Number of Sections** to '2' will only report end forces which might be desirable for connection calculation but not when looking for maximum forces along a members length. There is a printing option for end forces that is the equivalent to setting the **Number of Sections** to '2' but allows you to see more results on screen while printing only the end forces.

Member Force Results

Access the **Member Section Forces Spreadsheet** by selecting the **Results Menu** and then selecting **Members** ‣ **Forces**.



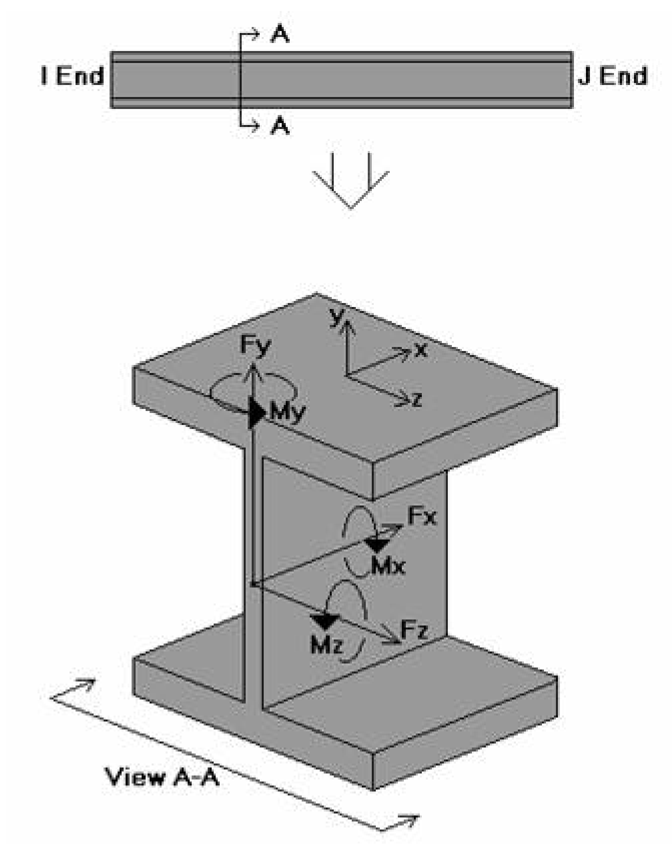
Member Section Forces (By Combination)

	L...	Member Label	S...	Axial[k]	y Shea...	z Shea...	Torque...	y-y Mo...	z-z Mo...
136	1	M28	1	22.415	8.144	0	.381	0	9.316
137			2	10.848	3.368	0	.19	0	-1.484
138			3	-.288	-.403	0	0	0	-4.147
139			4	-10.992	-3.167	0	-.19	0	-.59
140			5	-21.265	-4.924	0	-.381	0	7.272
141	1	M29	1	27.548	13.962	0	.618	0	25.938
142			2	14.133	5.773	0	.309	0	-4.131
143			3	.288	-.69	0	0	0	-11.547
144			4	-13.99	-5.428	0	-.309	0	-1.642
145			5	-28.698	-8.442	0	-.618	0	20.248
146	1	M30	1	12.775	5.817	0	.391	0	6.824
147			2	7.106	2.406	0	.195	0	-1.087
148			3	.575	-.288	0	0	0	-3.038
149			4	-6.819	-2.262	0	-.195	0	-.432
150			5	-15.075	-3.517	0	-.391	0	5.327

These are the member forces calculated along each active member taking into account any member offsets. The number of sections for which forces are reported is controlled by the **Number Of Sections** specified in the **Global Parameters Dialog**.

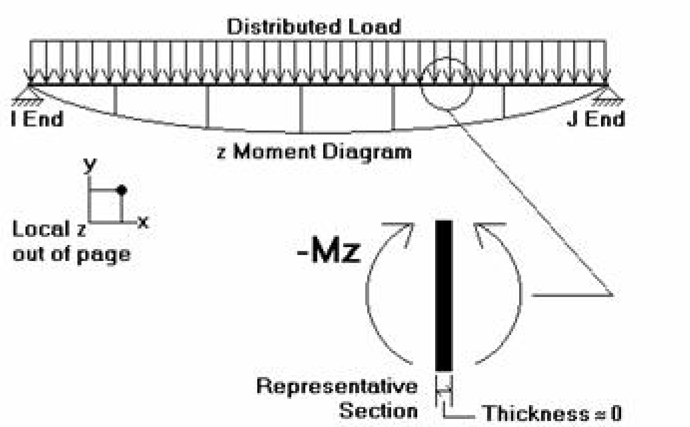
The units for the forces are shown at the top of each column. As for the sign convention, the signs of these results correspond to the member's local axes, using the right hand rule. The left side forces at each section location are displayed. There are six force values for each section location.

These are axial, shear parallel to the local y axis (Shear y-y), shear parallel to the local z axis (Shear z-z), torque moment, moment about the member's local y axis (Moment y-y) and moment about the member's local z axis (Moment z-z). Please see the diagram below:



This diagram shows a member section location with all positive section forces. As can be seen, the section forces listed at any given section are the left side forces. For axial forces, compression is positive. For moments, counter-clockwise around the member axis is positive.

These section forces may also be displayed graphically. Remember that the section forces used for the plot are the left side forces. For an example of what you would see for the graphic plot of the moment diagram for a member, please see below:



RISA-3D uses the right hand rule **joint convention** and is always consistent with this convention. Since the left side moment is being used, a member under negative Mz moment would have the "holds water" deflected shape, which is contrary to some **beam conventions**. The opposite is true for My moments which will tend to "hold water" under a positive moment and "shed water" under a negative moment.

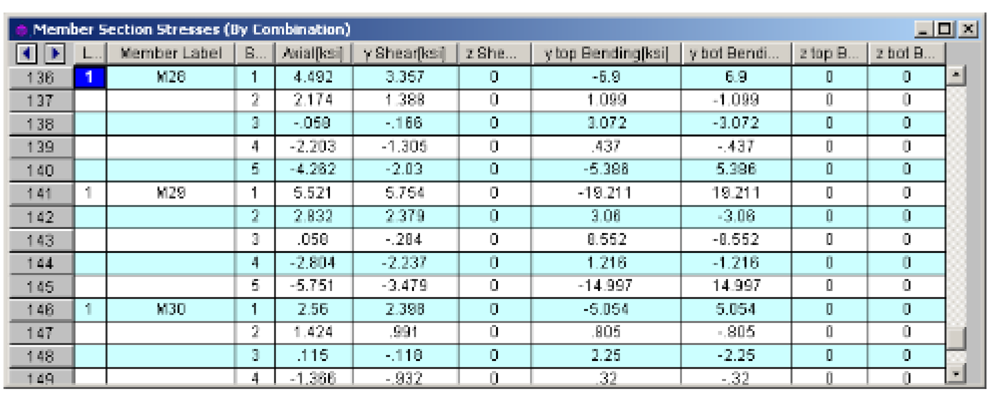
For enveloped results the maximum and minimum value at each location is listed. The load combination producing the maximum or minimum is also listed, in the "lc" column.

Note

- See [Spreadsheet Operations] to learn how to use **Find**, **Sort**, and other options.
- See [Plot Options – Members] to learn how to plot member results.

Member Stress Results

Access the **Member Section Stresses Spreadsheet** by selecting the **Results Menu** and then selecting **Members** ▸ **Stresses**.



Member Section Stresses (By Combination)

	L...	Member Label	S...	Axial[ksi]	y Shear[ksi]	z She...	y top Bending[ksi]	y bot Bendi...	z top B...	z bot B...
136	1	M28	1	4.492	3.357	0	-6.9	6.9	0	0
137			2	2.174	1.388	0	1.099	-1.099	0	0
138			3	-.059	-.166	0	3.072	-3.072	0	0
139			4	-2.203	-1.305	0	.437	-.437	0	0
140			5	-4.282	-2.03	0	-5.388	5.386	0	0
141	1	M29	1	5.521	5.754	0	-19.211	19.211	0	0
142			2	2.832	2.379	0	3.08	-3.08	0	0
143			3	.050	-.204	0	8.552	-8.552	0	0
144			4	-2.804	-2.237	0	1.216	-1.216	0	0
145			5	-5.751	-3.479	0	-14.997	14.997	0	0
146	1	M30	1	2.56	2.398	0	-5.054	5.054	0	0
147			2	1.424	.991	0	.805	-.805	0	0
148			3	.115	-.118	0	2.25	-2.25	0	0
149			4	-1.366	-.932	0	.32	-.32	0	0

These are the member stresses calculated along each active member. The number of sections for which stresses are reported is controlled by the **Number Of Sections** specified in the **Global Parameters Dialog**.

There will be seven stress values listed for each section location along the member taking into account any member offsets. The units for the stresses are shown at the top of each column. As for the sign convention, the signs of these results

correspond to the signs of the forces. These line up as positive or negative according to the member local axis directions. Compression is positive and tension is negative.

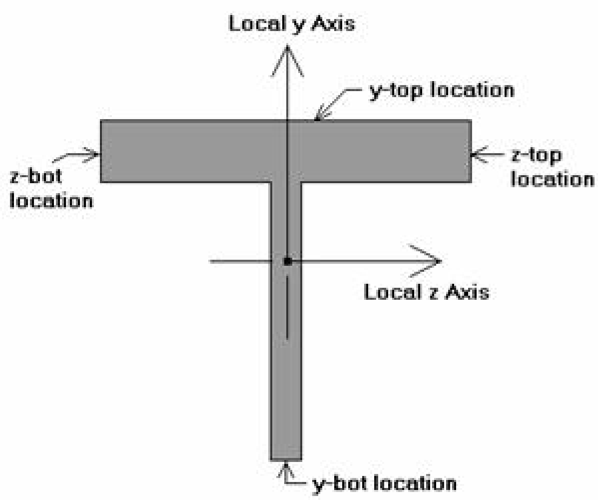
The axial stress is the ratio P/A, where P is the section axial force. A positive stress is compressive, since the sign of the stress follows the sign of the force.

The shear stresses are calculated as V/A_s, where A_s is the effective shear area. The program obtains A_s by multiplying the total area by the shear stress factor. This factor is calculated automatically for most cross sections, but must be entered for Arbitrary members. Refer to Member Shear Stresses.

The bending stresses are calculated using the familiar equation M * c / I, where "M" is the bending moment, "c" is the distance from the neutral axis to the extreme fiber, and "I" is the moment of inertia. RISA-3D calculates and lists the stress for the section's extreme edge with respect to the positive and negative directions of the local y and/or z axis. A positive stress is compressive and a negative stress is tensile.

Note that two stress values are listed for each bending axis. This is because the stress values for a bending axis will not be the same if the shape isn't symmetric for bending about the axis, as with Tee and Channel shapes. The y-top and y-bot values are the extreme fiber stress for the + or – y-axis locations. The same is true for the z-top and z-bot stresses.

The locations for the calculated stresses are illustrated in this diagram:



So, the y-top location is the extreme fiber of the shape in the positive local y direction, y-bot is the extreme fiber in the negative local y direction, etc. The y-top,bot stresses are calculated using Mz and the z-top,bot stresses are calculated using My.

For enveloped results the maximum and minimum value at each location is listed. The load combination producing the maximum or minimum is also listed, in the "lc" column.

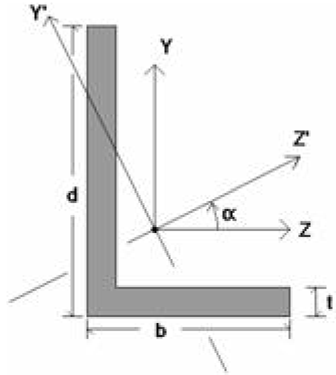
Note

- A special case is bending stress calculations for single angles. The bending stresses for single angles are reported for bending about the principal axes.
- Torsional stress results are listed separately on the Torsion spreadsheet.
- See Spreadsheet Operations to learn how to use **Find**, **Sort**, and other options.
- See Plot Options – Members to learn how to plot member results.

Single Angle Results

Depending on whether a single angle has been fully restrained against rotation or not it will either behave about its geometric axes or its principal axes. This behavior can be controlled by correctly specifying the unbraced lengths for the angle. In the

diagram below the z and y axes are the geometric axes. The z' and y' are the principal axes. The y' axis is considered to be the weak axis for principal behavior, and the z' is considered to be the strong axis.



The orientation of the shape is defined using the local y and z axes shown in the above diagram, but for principal axis behavior the bending calculations are done with respect to the y' and z' axes shown (the principal axes). The y' axis is the axis of minimum 'I' and the z' axis is the axis of maximum 'I'. RISA calculates the angle "α" and transposes the moments as shown below:

Mz' = Mz * cos(α) + My * sin(α)

My' = -Mz * sin(α) + My * cos(α)

The My' and Mz' moments are the moments shown as My and Mz respectively in the member forces results. Likewise, the y-top and y-bot bending stresses are relative to the extreme fibers along the y' axis (for the Mz' bending moment). The z-top, z-bot stresses are for My' bending at the extreme fiber locations along the z' axis.

Note

- If both LcompTop and LcompBot have been set to zero then the angle will behave about its geometric axes and the member forces and stresses will be displayed relative to the geometric axes. Alternatively, setting the L-torque value to zero will also constrain the single angle to behave about its geometric axes.

Member Torsion Results

Access the **Member Torsion Stresses Spreadsheet** by selecting the **Results Menu** and then selecting **Members** ▸ **Torsion.**

Member Torsion Stresses (By Combination)

	L...	Member Label	S...	Torque[k-ft]	Shear[ksi]	y Warp...	z Warp...	z-Bot ...	z-Top ...
136	1	M28	1	.381	0	0	.527	16.481	16.481
137			2	.19	1.677	0	.17	3.449	3.449
138			3	0	0	0	0	0	0
139			4	-.19	-1.677	0	-.17	3.449	3.449
140			5	-.381	0	0	-.527	16.481	16.481
141	1	M29	1	.618	0	0	.856	30.629	30.629
142			2	.309	4.507	0	.176	4.97	4.97
143			3	0	0	0	0	0	0
144			4	-.309	-4.507	0	-.176	4.97	4.97
145			5	-.618	0	0	-.856	30.629	30.629
146	1	M30	1	.391	0	0	.541	17.08	17.08
147			2	.195	1.775	0	.171	3.543	3.543
148			3	0	0	0	0	0	0
149			4	-.195	-1.775	0	-.171	3.543	3.543
150			5	-.391	0	0	-.541	17.08	17.08
151	1	M31	1	0	0	0	0	0	0
152			2	0	0	0	0	0	0

These are the torsional stresses calculated along each member. The number of sections for which torsional stresses are reported is controlled by the **Number of Sections** option on the **Global Parameters Dialog**.

The units for the torsion stresses are shown at the top of each column. RISA-3D calculates pure torsion shear for any shape type; this value is based on the maximum thickness of any part of the cross section. Closed shapes such as tubes and pipes do not warp, nor do solid rectangular or circular shapes. For these shapes there are no warping stresses to report. Warping only occurs in open cross sections where the rectangular pieces that make up the cross section do not all intersect at a single point. For example, a Tee shape could be thought of as two rectangular pieces, the flange and the stem. These two pieces intersect at the midpoint of the flange, so there is no warping. A channel, on the other hand, is comprised of three pieces, the two flanges and the web. These three pieces do NOT share a common point, so a Channel will warp. The same is true for a Wide Flange, so warping stresses are calculated only for I shapes (WF,S,H) and Channel shapes with warping restrained.

The shear and bending stresses caused by torsion are integrated into the code check and shear check calculations for the member, so your final code check (and final shear check) values DO include torsional effects. Warping shear is a shear stress acting parallel to the member's local y-and z-axis. Warping bending stress is a triangular stress normal to the cross section acting on the flanges, with the maximum stress at the outer edges of the cross section, the z-top and z-bot locations. As for the sign convention, the signs of these results correspond to the signs of the forces. These line up as positive or negative according to the member local axis directions. Compression is positive and tension is negative. See Torsion for more information on these calculations.

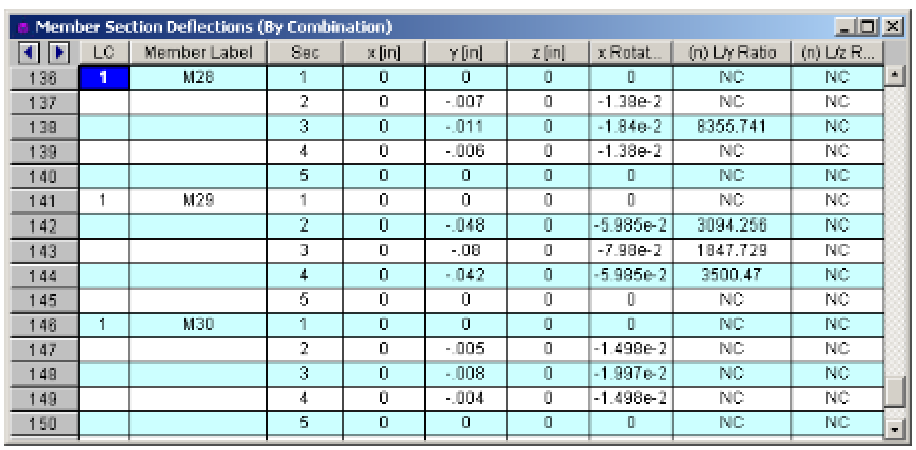
For enveloped results the maximum and minimum value at each location is listed. The load combination producing the maximum or minimum is also listed, in the "lc" column.

Note

- See Spreadsheet Operations to learn how to use **Find**, **Sort**, and other options.
- See Plot Options – Members to learn how to plot member results.

Member Deflection Results

Access the **Member Section Deflections Spreadsheet** by selecting the **Results Menu** and then selecting **Members** ▸ **Deflections**.



Member Section Deflections (By Combination)

	LC	Member Label	Sec	x [in]	y [in]	z [in]	x Rotat...	(n) L/y Ratio	(n) L/z R...
136	1	M28	1	0	0	0	0	NC	NC
137			2	0	-.007	0	-1.38e-2	NC	NC
138			3	0	-.011	0	-1.84e-2	8355.741	NC
139			4	0	-.006	0	-1.38e-2	NC	NC
140			5	0	0	0	0	NC	NC
141	1	M29	1	0	0	0	0	NC	NC
142			2	0	-.048	0	-5.985e-2	3094.256	NC
143			3	0	-.08	0	-7.98e-2	1847.729	NC
144			4	0	-.042	0	-5.985e-2	3500.47	NC
145			5	0	0	0	0	NC	NC
146	1	M30	1	0	0	0	0	NC	NC
147			2	0	-.005	0	-1.498e-2	NC	NC
148			3	0	-.008	0	-1.997e-2	NC	NC
149			4	0	-.004	0	-1.498e-2	NC	NC
150			5	0	0	0	0	NC	NC

These are the member deflections calculated along each active member. The number of sections for which deflections are reported is controlled by the **Number Of Sections** specified on the **Global Parameters Dialog**.

The member section deflections are comprised of 3 translations in the member local axis directions, the rotation (x Rotate) about the local x-axis (the twist), and the deflection to length ratios for the y and z deflections. The units for the deflections are shown at the top of each column. As for the sign convention, the signs of these results correspond to the member's local axes, using the right hand rule.

The 'L/y' and 'L/z' ratios are the member length (minus member offsets) divided by the deflection. The deflection in this calculation is not the deflection shown in the columns to the left, which are the absolute deflections. The deflection used is relative to the straight line between the deflected positions of the end joints. For cantilevers the deflection is relative to the original position of the member.

Expressed as an equation, n = L/deflection, where **n** is what is tabulated in the spreadsheet. The smaller the deflection, the larger the value. If 'NC' is listed, that means the 'n' value is greater than 10000 which is a very small deflection. The minimum value that will be shown is '1'. For example, if the deflection criteria is L/360, check here to make sure no tabulated values are **less than** 360. Greater than 360 is OK.

For enveloped results the maximum and minimum value at each location is listed. The load combination producing the maximum or minimum is also listed, in the "lc" column.

Note

- See Spreadsheet Operations to learn how to use **Find**, **Sort**, and other options.
- If you wish, you can go to the model view to plot and animate the deflected structure. The amount of deflection shown on the plot is controlled by the magnification factor. The joints are plotted based on the joint displacements, and these member deflections are used to plot the member's curvature between the joints. See Plot Options – Members to learn how to plot member results.

Model Merge

Model Merge is a feature located on the **Tools Menu** that scans through your model and automatically merges elements in the model. Model Merge detects unconnected joints along member spans, unconnected crossing members and duplicate joints, members and plates. You can use Model Merge to build models faster as well as to detect and fix modeling errors.

Knowing what Model Merge does allows you to skip modeling steps as you build your model and let the software perform these steps for you. You can take advantage of Model Merge in modeling your structure in many ways, a few of which are:

Note

- Model Merge will "connect" Physical Members by inserting a joint at their intersection but will not break them up into smaller members. Model Merge also eliminates duplicate Physical Members so you should still use model merge to detect errors when working with these types of members. See Physical Members to learn more.

What do you want to know?

Model Merge Options

There are three main options for the model merge.

The **Merge Tolerance** is set in the Global Parameters screen and defines 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.

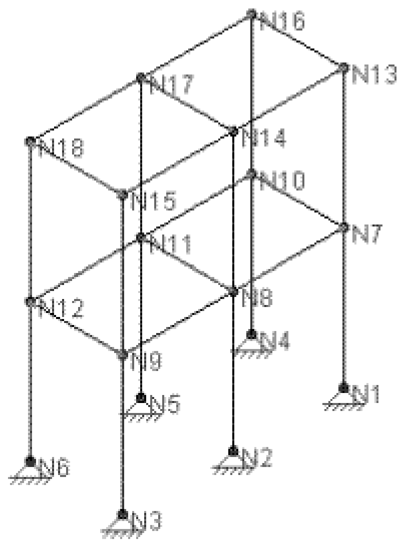
If the **Merge Crossing Members** box is checked then as part of the merge process, all members will be scanned along their lengths for crossing members. Crossing members will be merged together at their intersection points. If you have cross bracing you may or may not want them to be merged. A finite element crossing a physical member will cause only the finite element to be broken up at the point of intersection. If two physical members are crossing, a joint will be placed at the intersection point.

Merge Inactive Members if left unchecked allows you to limit the merge process to the parts of your model that are selected. This allows you to prevent the program from merging portions of your model where you may have intentionally put joints at the same location or have two members next to each other.

Trim/Extend Crossing Beams can help connect member ends that are within the merge tolerance from an adjacent beam. Checking this box will move the end node of that member so that it lies on the adjacent beam.

Trim/Extend Crossing Wall Panels can help when you have two wall panels that are overlapping or intersecting by a distance smaller than the merge tolerance. This allows you to easily correct minor modeling flaws. This will also correct the condition where a defined wall panel is non-coplanar.

Model Merge Examples



Looking at this frame, consider the column line on the right side, members 1-7 and 7-13. If you did not use Physical Members you would define this just that way, as two separate members. With the model merge capability you could instead enter a single member definition, 1-13, and let the model merge function break it up for you.

Other convenient uses of the model merge function are laying out floor plans and being able to draw all the joists right over the main girders or defining truss chords as full length rather than specifying each panel point. The model merge will take of breaking up the members at all the intersecting points. Of course the Physical Member feature goes a step farther for these situations because these members **never** need to be split to model a connection – allowing you to make edits and understand results more readily. See Physical Members to learn more.

Model Merge Limitations

Certain types of shape types and certain load types can cause members to not get broken up by the model merge function. In particular, members that are Tapered WF shapes will not get broken up by the model merge. Even if such members have intermediate unattached joints, or crossing members within their spans, they will not be broken up.

Model Merge Process

1. Duplicate joints are merged together.
2. All the members are scanned for other members crossing along their span. If a crossing member is found, a joint is created at the intersection point.
3. All the members are scanned for joints along their span. If found, the member is broken up into pieces to incorporate the joint.
4. Duplicate members are merged together. Physical members take precedence over finite element members. Longer members take precedence over shorter members that are fully coincident with the longer member. When duplicate members with different section sets are merged, the set listed first in the Sections Sets spreadsheet are used.
5. Duplicate plates are merged together.

To better understand how the model merge function works, please refer to this figure:

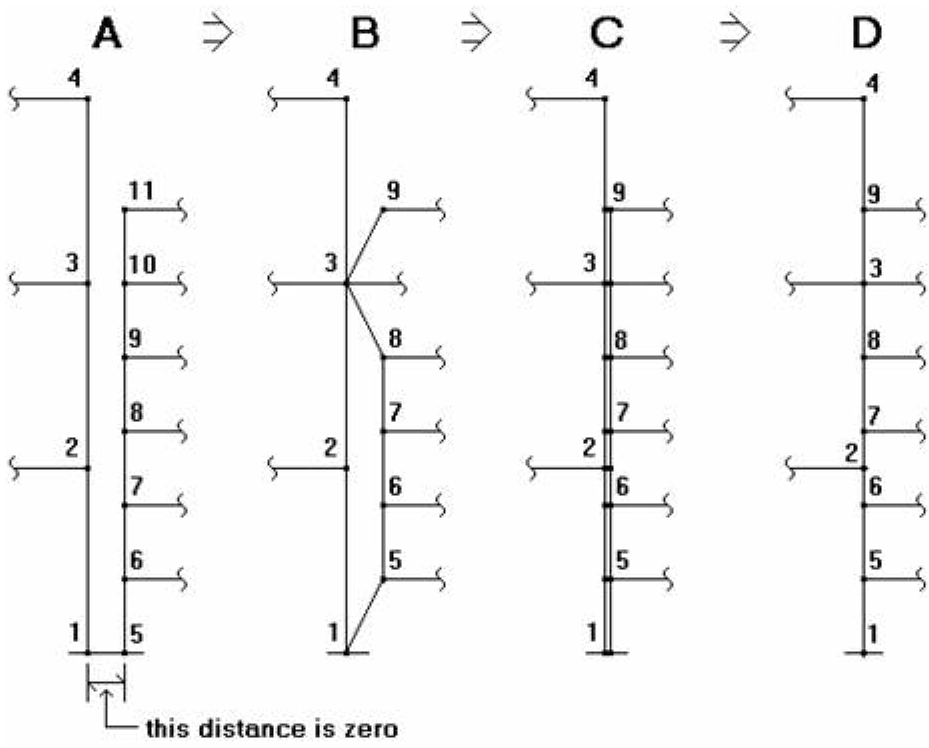


Diagram A shows the model before a merge. The two column lines are separated in diagrams A and B strictly for ease of viewing, they should be considered to be right on top of each other. Joints 1 and 5 (in diagram A) have exactly the same coordinates, as do joints 3 and 10. None of these members are Physical Members.

Step 1 of the merge eliminates duplicate joints changing the model from diagram A to diagram B. On diagram A, joints 1 and 5 are duplicates (same coordinates), as are joints 3 and 10. Joints 5 and 10 are merged into joints 1 and 3 respectively. This means any loads applied to joints 5 and 10 are now applied to joints 1 and 3. Any members connected to 5 and 10 are now connected to joints 1 and 3 (these members are shown with the inclined lines in diagram B).

Step 2 looks for crossing members, however, there aren't any for this particular example. Members that are parallel to each other aren't treated as "crossing" since the end joints of overlapping members will be merged in Step 3.

Step 3 is where the members are scanned for intermediate span joints. This takes us from diagram B to diagram C. Referring to diagram B, member 1-2 has two intermediate joints (5 and 6), member 6-7 has one intermediate joint (joint 2), and so on. The members with intermediate joints are broken up, shown in diagram C.

Step 4 eliminates duplicate members, in this case those that were created in step 2. This takes us from diagram C to diagram D. Looking at diagram C, the duplicate members are shown as the double lines. The first member listed on the **Member** spreadsheet is maintained and the other member is deleted. Any loads applied to the deleted member are transferred to the remaining member.

The final merged model is shown in diagram D. The column line is now comprised of 8 members, 1-5, 5-6, 6-2, etc. up to member 9-4.

Note

- The direction code of loads merged for duplicate members and plates is kept the same. Elements with loads in local directions and different orientations may result in a load direction that is not be the same as the original direction.

To Perform a Model Merge

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Select the items you wish to merge. Typically you will want everything to be selected.
3. Click the **Merge** button and set the parameters for the new merge. For help on an item, click and then click the item.

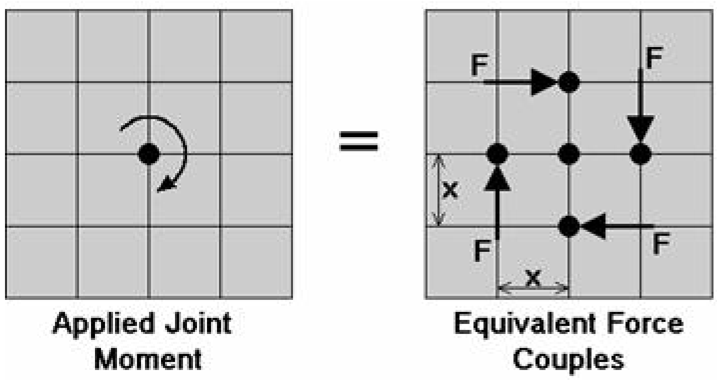
Note

- You may undo any mistakes by clicking the **Undo** button.

Modeling Tips

Applying In-Plane Moment to Plates

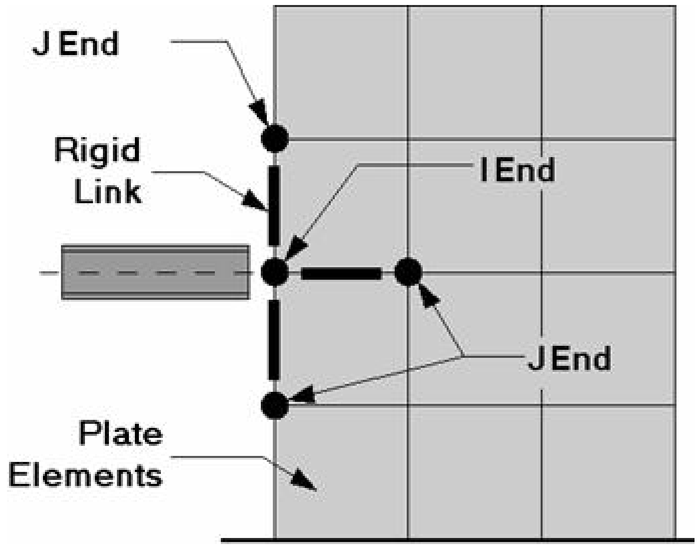
Occasionally you may need to model an applied in-plane moment at a joint connected to plate elements. The plate/shell element cannot directly model in-plane rotations. One way around this is to model the in-plane moment as a force couple of in-plane forces. You would replace the applied in-plane joint moment at 1 joint with 2 or 4 in-plane forces at 2 or 4 joints, which would produce the same magnitude in-plane moment. See below:



This might require re-meshing the area receiving the moment into smaller plates so that the load area can be more accurately modeled. If a beam member is attached to the joint and will be used to transfer the moment, than you will want to look at the topic **Modeling a Beam Fixed to a Shear Wall** below.

Modeling a Beam Fixed to a Shear Wall

Occasionally you may need to model the situation where you have a beam element that is fixed into a shear wall. A situation where this may occur would be a concrete beam that was cast integrally with the shear wall or a steel beam that was cast into the shear wall. The beam cannot just be attached to the joint at the wall because the plate/shell element does not model in-plane rotational stiffness. A fairly simple work around is to use rigid links to transfer the bending moment from the joint at the wall as shear force to the surrounding joints in the wall. See Rigid Links in the Modeling Tips section to learn how to create rigid links. This modeling method provides a more accurate analysis than trying to use a plate/shell element with a "drilling degree of freedom" which attempts to directly model the in-plane rotation. See the figure below:



The only trick to this method is getting the proper member end releases for the rigid links. We want to transfer shear forces from the wall joint to the interior wall joints without having the rigid links affect the stiffness of the shear wall. Notice from the figure that the I-joint for all the links is the joint connected to the beam element, while the J-joints are the ends that extend into the shear wall. The J-ends of all the rigid links should have their **x, Mx, My,** and **Mz** degrees of freedom released. Only the **y** and **z** degrees of freedom (local axes shears) should be connected from the J-ends to the interior wall joints. This release configuration will allow the shears to be transferred into the wall, but the wall stiffness will not be adversely affected by the presence of the rigid links.

Modeling a Cable

While there is not a true "cable element", there is a tension only element. A true cable element will include the effects of axial pre-stress as well as large deflection theory, such that the flexural stiffness of the cable will be a function of the axial force in the cable. In other words, for a true cable element the axial force will be applied to the deflected shape of the cable instead of being applied to the initial (undeflected) shape. If you try to model a cable element by just using members with very weak Iyy and Izz properties and then applying a transverse load, you will not get cable action. What will happen is that the beam elements will deflect enormously with NO increase in axial force. This is because the change in geometry due to the transverse loading will occur after all the loads are applied, so none of the load will be converted into an axial force.

Guyed Structure ("Straight" Cables)

You can easily model cables that are straight and effectively experience only axial loading. If the cable is not straight or experiences force other than axial force then see the next section.

When modeling guyed structures you can model the cables with a weightless material so that the transverse cable member deflections are not reported. If you do this you should place all of the cable self-weight elsewhere on the structure as a point load. If you do not do this then the cable deflections (other than the axial deflection) will be reported as very large since it is cable action that keeps a guyed cable straight. If you are interested in the deflection of the cable the calculation is a function of the length and the force and you would have to calculate this by hand.

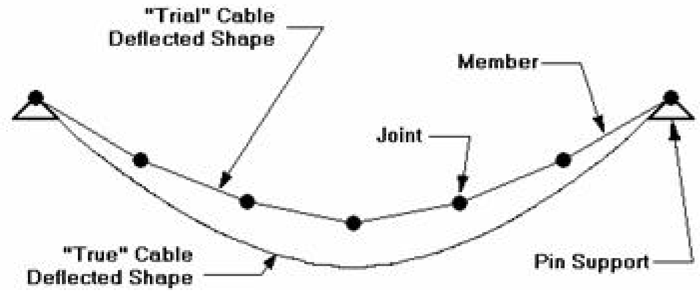
The section set for the cable should be modeled as a tension only member so that the cable is not allowed to take compression. See T/C Members for more on this.

To prestress the cable you can apply a thermal load to create the pre-tension of the cable. See Prestressing with Thermal Loads to learn how to do this.

Sagging Cables (Large Deflections) & Transverse Loads

One way, (although not an easy one) to model a sagging cable is as follows: First you would define members with the correct area and material properties of the cable. You should use a value of 1.0 for the Iyy, Izz, and J shape properties. Next you

will want to set the coordinates for your joints at a trial deflected shape for the cable. Usually you can use just one member in between concentrated joint loads. If you are trying to model the effects of cable self-weight, you will need to use at least 7 joints to obtain reasonable results. See the figure below for an example of a cable with 5 concentrated loads:



You will want to set the vertical location of each joint at the approximate location of the “final” deflected shape position. Next you will connect your members to your joints and then assign your boundary conditions. Do NOT use member end releases on your members. Make sure you do NOT use point loads, all concentrated loads should be applied as joint loads. You can model pre-stress in the cable by applying an equivalent thermal load to cause shortening of the cable.

Now you will solve the model with a P-Delta Analysis, and take note of the new vertical deflected locations of the joints. If the new location is more than a few percent from the original guess, you should move the joint to the midpoint of the trial and new location. You will need to do this for all your joints. You will repeat this procedure until the joints end up very close to the original position. If you are getting a lot of “stretch” in the cable (more than a few percent), you may not be able to accurately model the cable.

Once you are close to converging, a quick way to change all the middle joint coordinates is to use the Block Math operation. That way you shift many joints up or down by a small amount in one step.

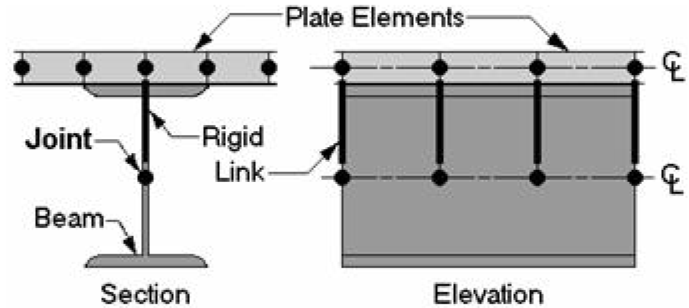
Modeling Composite Behavior

Occasionally you may want to model a structure with composite behavior. A practical situation where this arises is with composite concrete floor slabs, which have concrete slabs over steel or concrete beams. Another common case where composite behavior may be considered is where you have a steel tank with stiffeners. The stiffeners might be single angles or WT shapes.

The most accurate way to model this behavior for composite floor systems is to use a program (such a RISAFloor) that was created explicitly for this purpose. If that is not an option you may want to use an arbitrary member with an effective moment of inertia, I_{eff}, calculated according to the AISC or Canadian code provisions. This method has the advantage of being able to account for the effects of partial composite behavior, while the methods described below assume perfect connectivity between the steel beam and the concrete slab.

If you have already modeled the concrete deck with plate elements, then the situation can be quickly modeled for horizontal beams by using one beam modeled as a physical member and also setting the TOM flag. The beam would simply be drawn along the plates/joints for the composite section. This method has the advantage of being very fast to model, but it only works for horizontal beams (since the TOM feature only offsets beams that are horizontal) and it also neglects the depth of the concrete in computing the total vertical offset between the centerline of the beam and the centerline of the slab. Note that you have to have an appropriate number of joints along the span of the beam to model the shear transfer between the slab and the beam. A more refined (and complicated) method using rigid links is described next.

An example of a plate/beam model with composite action included using rigid links is shown here:



Note that beams and plates are each modeled at their respective centerlines. It is this offset of the beam and plate centerlines that causes the composite behavior. The distance between the centerlines is typically half the depth of the beam plus half the thickness of the plate elements. If the beam is an unsymmetrical shape, like a WT about the z-z axis or a single angle, then you would use the distance from the flange face to the neutral axis.

As shown above, a rigid link is used to connect each set of joints between the beam and the plates. This rigid link is fixed to each joint and therefore has no member end releases. See Rigid Links in the Modeling Tips section to learn how to create rigid links.

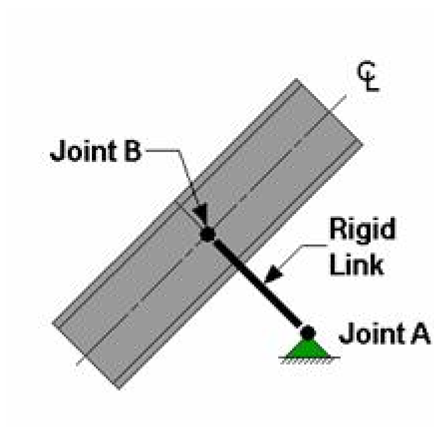
Graphical editing offers the fastest way to model composite action. It is usually best to model the plates with an appropriately fine mesh first. Then you would copy the plate joints that are directly over the beam down to the centerline elevation of the beams. Next you would draw your beam to the two outer joints at the beam centerline elevation.

There are a number of ways to build your rigid link member between all the corresponding plate and beam joints. A good way to connect all the links is to generate grid members (Grid Member Generation). You can also draw the first one and then copy it along the length of the member.

Now perform a model merge. If you have used finite members rather than Physical Members the merge Model Merge will break up finite element beams at all the intermediate locations. See Model Merge for more information. If you're using physical members, performing the model merge will clean up duplicate joints and members but not break up the beam member since the links are automatically attached to Physical Members as long as their nodes lie on the member.

Modeling Inclined Supports

You may model inclined supports by using a short rigid link to span between a joint which is restrained in the global directions and the item to receive the inclined support. See the figure below:

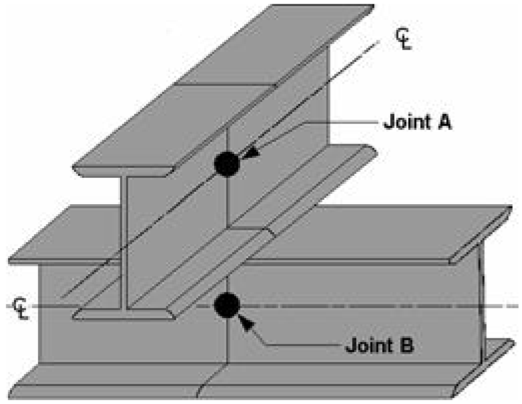


The rigid link should be "short", say no more than 0.1 ft. The member end releases for the rigid link at joint "B" are used to control which degrees of freedom are pinned or fixed in the inclined directions. This works because the member end releases are in the local member axes. See Rigid Links in the Modeling Tips section to learn how to create rigid links.

The section forces in the rigid link **are** the inclined reactions. Note that you need to make sure the rigid link is connected to the members/plates at the correct inclined angle. You can control the incline of the angle using the coordinates of joints "A" and "B". You can also rotate the rigid link to the proper angle.

Modeling One Member Over Another

Occasionally you may need to model the situation where one member crosses over another member. A common situation where this occurs is in the design of framing for crane rails, where the crane rail sits on top of, or is hanging beneath, the supporting beam. See the figure below:



The two beams are each modeled at their correct centerline elevations. Both the top and bottom members need to have a joint at the point of intersection. The distance between the joints would be half the depth of the top beam plus half the depth of the bottom beam. The method to model this is to connect joint A to joint B is with a rigid link. The member end releases at the A or B end can be used to control which degrees of freedom get transferred between the beams. For example, if Beam A is free to pivot over Beam B then you would apply a Pin end release to the top of the rigid link, and a Fixed end release to the bottom of the rigid link. Don't pin both ends of the rigid link though, because then there can be no shear transfer between the beams. See Rigid Links in the Modeling Tips section to learn how to create rigid links.

Reactions at Joints w/ Enforced Displ.

The reaction at an enforced displacement can be obtained by inserting a very short (.02' or so) rigid link between the joint with the enforced displacement and any attached members. The member forces in this rigid link will be the reactions at the joint with the enforced displacement. It is helpful to align the link to be parallel with one of the global axes, that way the local member forces will be parallel to the global directions unless of course you are modeling inclined supports. See Rigid Links below to learn how to create rigid links.

Rigid Links

Rigid links are used to rigidly transfer the forces from one point to another and to also account for any secondary moments that may occur due to moving the force. This is in contrast to using the slave feature for joints where the forces are shared by 2 or more joint degrees of freedom (DOF), but any secondary moments are lost when slaving the joints. Slaved joints actually share common DOF and so do not account for the distances between them. Rigid links do not have any practical internal deformation, I.e. there is no differential movement between the I-joint and the J-joint. Rigid links may be used to model situations such as composite behavior or beams fixed to walls modeled with plate elements. They are also useful for getting information such as reactions at inclined supports or reactions at joints with enforced displacements.

To Make a Rigid Link

1. On the **General** tab of the **Materials** spreadsheet create a material **Label** called “LINK”. Enter “1e6” for the value of **E**. Blank out the value for **G** by going to that field and pressing the space bar. Set the **Density** to zero. Leave all the other values as their defaults.
2. On the **General** tab of the **Sections** spreadsheet create a section set **Label** called “RIGID” with the “LINK” **Material**.
3. Move the cursor to the **Shape** field and hit your space bar to erase the information in this field.
4. Set the **A**,and **I**y, **I**z and **J**values to “1e6”.
5. To make a rigid link, on the **Member** spreadsheet create a member that references the RIGID section set created above. You can control which DOF are transferred through the link by using the member end releases.

Note

- The density is set to zero in case the self-weight is used in a loading condition. If the density is left as the default then any gravity loading would cause the rigid link to add a very large load into your model due to it's large area.
- To keep the model merge from deleting your rigid links, be careful not to create links whose lengths are less than your merge tolerance.

The weight density should be set to zero in case self-weight is used as a loading condition. If the material used is not weightless, then any gravity loading would cause the rigid link to add a very large load into your model. (Gravity load is applied as a distributed load with a magnitude equal to the member area times the weight density).

For models with very stiff elements, like concrete shear walls, the rigid link may not be rigid in comparison. If you see that the rigid link is deforming, then you may have to increase the stiffness of the link. The easiest way to do this is to increase the A, Iy, Iz, and J values for the RIGID section set. Make sure that the combination of **E*I** or **E*A** does not exceed 1e17 because 1e20 and 8.33e18 are the internal stiffnesses of the translational and rotational Reaction boundary conditions. If you make a member too stiff, you may get ghost reactions, which tend to pull load out of the model. (The total reactions will no longer add up to the applied loads.)

Solving Large Models

Large models are those where the stiffness matrix size greatly exceeds the amount of available free RAM on your computer. Solving large models can take a long time, so it is useful to have an understanding of what steps can help speed up the solution. The time it takes to solve a model is dependent on several things; these include the bandwidth of the stiffness matrix, the number of terms that need to be stored for the stiffness matrix, and the amount of RAM in your computer.

A bandwidth minimizer is used at the beginning of the solution to try to reorder your degrees of freedom to get a reduced bandwidth stiffness matrix size. Sometimes, however, the bandwidth minimizer can be fooled and will give a poor matrix column height and a huge number of matrix terms.

If you are getting a stiffness matrix that is larger than you would expect and you don't think that you have any modeling errors, you can try a few things to reduce the bandwidth. The first thing you can try is to sort your joints. Typically you will want to sort your joints on the Coordinates spreadsheet from “Low to High” in the 2 lateral directions and then lastly in the vertical direction. After you sort your joints, try to solve again and check the matrix size. The order of sorting depends on the model, so you might want to try a couple of different combinations and check the model each time. Sometimes, sorting the joints will result in cutting the height and number of terms by a factor of 2.

You will also want to make sure you don't have separate structures in the same model where one is big and the other small. You will get a very large matrix height if the bandwidth minimization starts on the small model and then jumps to the big model. Instead, you will probably want to split these into 2 separate files.

The amount of “address space” available to solve your model is based on several things: the amount of RAM in your computer, the amount of free hard disk space, the operating system, the Virtual memory settings in the Windows Control Panel, and the internal limitations of your operating system.

If you get an error that states “You have run out of memory, try increasing your virtual memory…” you will want to note the amount of memory that was requested at the time versus the amount that was available. This amount should displayed along with the error message. This amount will give a starting point from which you can increase the available address space. You may need to increase the amount of Virtual Memory so that you have enough address space to run the model and your other applications. (You typically do this by double clicking on “My Computer”, then “Control Panel”, then “System”. Within the System options, you would click on the “Performance” tab and then you click on the “Virtual Memory” button.) Make sure that you are specifying more Virtual Memory than is needed to solve your model.

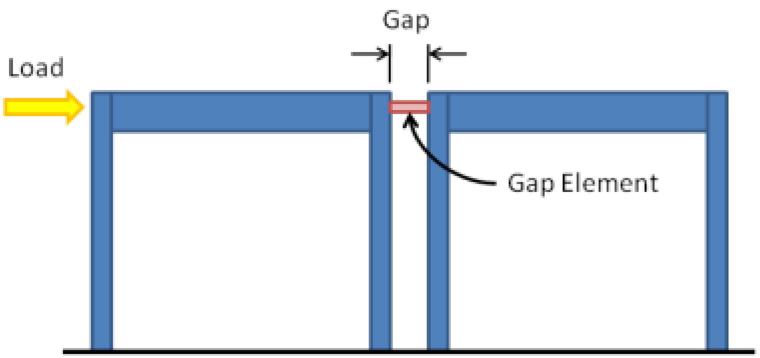
There is an internal limitation to the amount memory that Windows will allocate to the RISAProgram. Within a 32 bit addressing space, Windows has a basic limit of 4 Gigabytes. Of that, they reserve 2 Gigs for the operating system. Therefore, RISA can only use a maximum of 2 Gigabytes with large models (where the stiffness matrix alone is well in excess of 1.0 Gigs), your only option may be to re-model your structure using fewer degrees of freedom.

Note

- All of the model views and spreadsheets are updated as you edit the model. For large models this update can take a while. You may turn off the **Full Synchronization** option by clicking on the **Tools** menu and selecting **Preferences**. You may then use the **Refresh All**  button on the **RISA Toolbar** to update the views and spreadsheets manually.

Modeling a "Gap" (Expansion Joint) Between Structures

A gap element is a member that mimics the behavior of a gap or expansion joint between adjacent structures.



While RISA does not offer the capability to directly create a gap element, one may be indirectly created using the properties of the member and an applied thermal load. The concept is to place a ‘shrunk’ member between adjacent structures. The shrinkage of the member is achieved with a negative thermal load, in the form of a member distributed load.

The amount of shrinkage should be equal to the width of the gap, such that the structures act independently until they move close enough to each other to ‘touch’ and thereby transmit loads to each other. To calculate the thermal load required for a gap use the following formula:

$$\Delta T = \frac{-1 * (Gap)}{L * (\alpha)}$$

Where:

ΔT = Applied thermal load

Gap = Distance between two structures

L = Length of gap element

α = Coefficient of thermal expansion

In order to prevent the gap element from ‘pulling’ its connected structures towards it due to shrinkage it must be defined as a ‘compression only’ member under the advanced options tab. It is also advisable to define the gap element as a rigid material such that the amount of load it transfers once the gap is closed is not affected by elastic shortening.

Lastly, in cases where the applied temperature would need to be of an extraordinary magnitude, it might be useful to increase the coefficient of thermal expansion of the material such that a smaller temperature load would achieve the same shrinkage.

P-Delta - Analysis

When a model is loaded, it deflects. The deflections in the members of the model may induce secondary moments due to the fact that the ends of the member may no longer be co-linear in the deflected position. These secondary effects, for members (not plates), can be accurately approximated through the use of P-Delta analysis. This type of analysis is called "P-Delta" because the magnitude of the secondary moment is equal to "P", the axial force in the member, times "Delta", the distance one end of the member is offset from the other end.

Since RISAFloor is designing entirely for gravity loads it does not need to account for the P-Delta effect. However, elements of the lateral force resisting system do need to consider this effect when being designed for lateral forces.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keyword: **P-Delta**.

To Perform a P-Delta Analysis

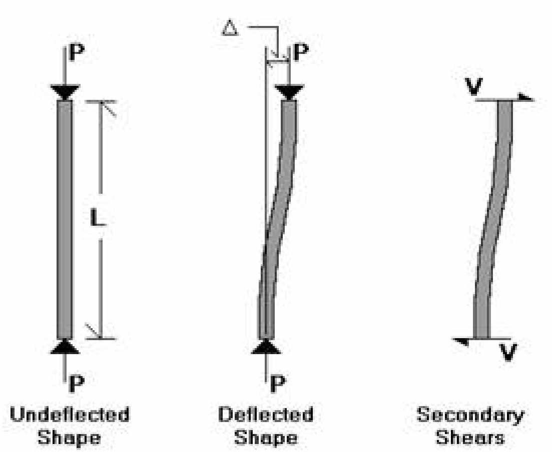
- On the **Load Combination** spreadsheet, indicate the P-Delta combinations by placing a "Y" in the **P-Delta** column.

Note

- You may indicate a compression only P-Delta analysis by placing a "C" in the **P-Delta** column.

P-Delta Procedure

The actual modeling of these secondary moments is done through the calculation of secondary shears (shown as V in the diagram):



$$P * \Delta = V * L$$

$$\text{so,} \quad V = P * \Delta / L$$

These shear forces are applied at the member ends. For a 3D model, this P-Delta calculation is done for the member's local y and local z directions.

1. The solution sequence is as follows:
2. Solve the model with original applied loads
3. Calculate the shears (V's) for every member in the model
4. Add these the shears (V's) to the original loads and re-solve
5. Compare the displacements for this new solution to those obtained from the previous solution. If they fall within the convergence tolerance the solution has converged. If not, return to step 2 and repeat.

If the P-Delta process is diverging dramatically, it will be stopped before numerical problems develop and an error will be displayed. If this error is displayed, the P-Delta displacements have reached a level where they are more than 1000 times greater than the maximum original displacements. If this happens with your model, the model may be unstable under the given loads, or there may be local instabilities present. See P-Delta Troubleshooting and Testing Instabilities to learn how to solve these problems.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keyword: **P-Delta**.

P-Delta Limitations

The P-Delta algorithm is based on end-joint displacements and will not automatically account for the effect of interior span forces on members, or member end rotations. As such the RISA implementation of P-Delta can be referred to as a P-Big Delta analysis.

In addition to the P-Big Delta effect, there can also be a P-Little Delta effect caused by end node rotation or transverse loading between node points. Please refer to the P-Little Delta topic for more information.

Note

- P-Delta effects are not calculated for plates.
- P-Delta effects are not currently include in Dynamic / Modal Analysis.

Compression Only P-Delta

The P-Delta effect can be thought of as decreasing the flexural stiffness of members in compression and increasing the flexural stiffness of members in tension. It is possible that if you have members with extremely large tensile forces, and intermediate joints that are not connected to supports or other "stiff" members, the P-Delta algorithm could cause a joint displacement to reverse direction, instead of converging to zero. This is an incorrect result. (A practical example where this could happen would be a truss chord with a large amount of tension that also has extra joints in-between the panel points.) If you have members with very large tensile forces, and intermediate joints, you may want to do a "compression only" P-Delta analysis. You invoke this by putting a "C" in the **P-Delta** field, instead of a "Y". A "compression only" P-Delta analysis will only affect members that are in axial compression. The P-Delta analysis will not modify members that are in tension.

P-Delta Convergence

The default convergence tolerance is 0.5%. This means that the displacements from one solution to the next must vary by no more than ½ of 1 percent for the solution to be considered converged. You may adjust this tolerance on the **Global Parameters**. If you have a model that does converge but takes a lot of iterations, you may want to increase this tolerance so convergence is faster. Be careful! If you set this value too high, unstable models may falsely converge. It is not advisable to set this value above 2 or 3 percent.

P-Delta Troubleshooting

The first step in troubleshooting a P-Delta model that won't converge is to run the load combination without P-Delta analysis and make sure there are no instabilities. If it turns out that degrees of freedom are being locked, this indicates instabilities that you will want to fix.

In some ways, a P-Delta divergence may indicate an elastic buckling failure. Therefore, any P-Delta instabilities should be taken seriously and investigated thoroughly.

Local Instabilities

By far, the most common cause of P-Delta convergence problems is *local* instabilities. A local instability is when one part of the model is unstable causing the P-Delta analysis to diverge. To locate local instabilities, run the solution with P-Delta analysis turned OFF. Now plot the exaggerated deflected shape and animate it. Any local instabilities should be apparent. See Testing Instabilities for more information.

If you are trying to model P-Delta effects on a 2D frame, you will want to make sure that you restrain the out-of-plane degrees of freedom. See Boundary Condition at All Joints to learn how to do this.

Flexible Structures

In some cases, a model may be so flexible that it is not possible to run a P-Delta analysis. A situation where this might occur would be a wood frame where all the connections were modeled as pins, but the boundary conditions did not provide positive lateral support. In the real world, the connections will take some moment and the structure would be fine, but in the idealized model, there is zero moment resistance at each connection. The total lateral stiffness would be very small and this would make convergence of a P-Delta analysis unlikely.

Stiffness Reduction for the AISC 13th Edition / Direct Analysis Method

The Direct Analysis method requires a reduction in the axial and flexural stiffness of some members. This is done to account for the non-linear material effects caused by residual stresses. This stiffness reduction can be problematic during the early stages of design when your members may be smaller than they will be after basic drift and stress criteria are met. Therefore, RISA gives you the option (on the Codes tab of the Global Parameters) to turn off this stiffness reduction or set it to a constant value (tau = 1.0).

P-Delta Amplification for AISC 13th Edition ASD

In order to ensure that using LRFD load factors does not result in a penalizing effect for P-Delta analysis, the AISC specification requires that the forces be multiplied by 1.6 during ASD analysis. Therefore, if AISC 13th ASD is specified as the hot rolled code, RISA automatically multiplies all load combinations by 1.6 during solution, then divides the force results by 1.6 prior to displaying the results.

P-Delta for ACI Concrete

See the P-Little-Delta Topic.

Wall Panels

The P-Delta effect is handled the same way for Wall Panels. The main difference is that the shears for a wall panel are only generated at the story locations or the diaphragm locations. The average deflection at the story height (or diaphragm location) is used to calculate the P-Delta shears for the top and bottom of the story. These shears are then distributed over the width of the wall at those locations.

Leaning Column Effect

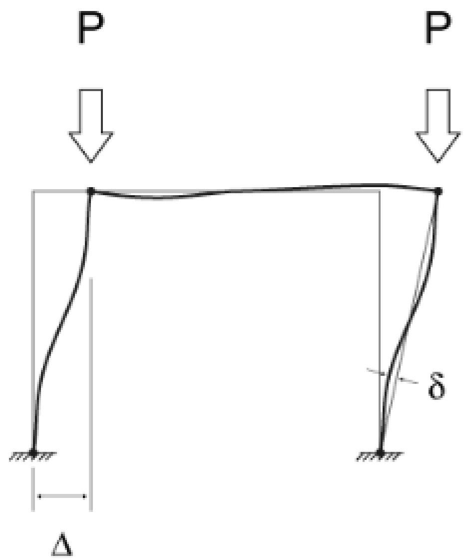
Gravity columns and walls that are modeled in RISAFloor, but which are NOT included in the Lateral analysis will still have an effect on the P-Delta analysis in RISA-3D. This is because RISA-3D automatically includes this leaning column / leaning wall effect. This is done only for columns or walls that are contained within the a slab edge at that floor level. The program then uses the RISAFloor column / wall axial loads (without considering LL Reduction) along with the rigid diaphragm displacements (projected to each column / wall) to come up with equivalent leaning column shears for the diaphragm.

Note

- The leaning column effect does not get accounted for columns or walls which do not connect to a diaprhagm.

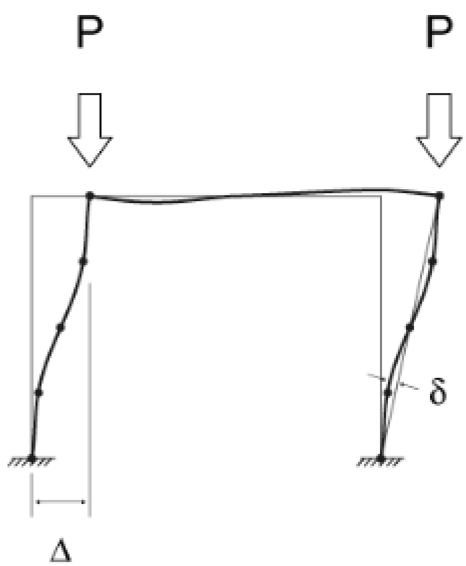
P - Little Delta Analysis

The P-Little Delta effect is essentially the destabilizing effect of individual member curvature on the axially loaded members. In practical applications, the effect of P-Little Delta will normally be significantly smaller than the P-Big Delta effect. The following figure demonstrates the displacements that cause the P-Delta and P-Little Delta effect in a typical moment frame.



P-Little Delta Procedure

The current implementation of P-Delta analysis does not directly account for the Little Delta effect. While this is true, the model can be easily adjusted to account for the effect as shown in the figure below.



Because RISA's P-Delta method is based entirely on nodal deflections, the introduction of nodes at the locations along the column where member displacement effects are at their maximum will adequately account for this effect.

There are a number of "benchmark" tests given in various publications to determine if a program is capable of properly considering this effect. Some of these benchmark problems (from the AISC Commentary) have been posted to our website with comparisons to theoretically correct solutions. These comparisons can be used as a basis for when the P-Little Delta

effect is significant enough to consider in the analysis. They can also be used to determine how many intermediate nodes are required to adequately account for the effect at a given load level.

Note

- P-Little Delta effects will have more of an impact as a member approaches its elastic euler buckling load. Special attention should be paid to cantilevered compression members, or members with significant weak axis bending moments.

AISC Direct Analysis Method

The 13th edition of the AISC manual specifically requires the consideration of the P-Little Delta effect. However, the Direct Analysis Method (Appendix 7) inherently acknowledges that P-Little Delta may not be important by stating that it may be neglected when, “the axial loads in all members whose flexural stiffnesses are considered to contribute to the lateral stability of the structure” are less than 15% of the Elastic / Euler buckling load of the member.

This may seem like it would be a low axial force, but it can actually be a very large force greater than the total axial capacity of the column. Situations where this code provision may justify ignoring the P-Little Delta effect would include strong axis bending of a wide flange column with equal unbraced lengths in the strong and weak axes.

ACI Concrete Design

As of the 2008 edition, ACI 318 offers three options to account for slenderness and curvature in columns and walls (Section 10.10.2):

- Nonlinear Second Order Analysis (10.10.3)
- Elastic Second Order Analysis (10.10.4)
- Moment Magnification (10.10.5)

Both forms of second order analysis are more accurate than the moment magnification procedure, which is a hand calculation method intended to be used when computer analysis is not available. RISA does not have a true nonlinear solver though, so the program performs an Elastic Second Order Analysis using cracked section properties.

Columns in RISA-3D

In order to meet the requirements of Section 10.10.4 (thereby meeting the requirements of Section 10.10.2) you must turn on P-Delta in the Load Combinations spreadsheet, and possibly add intermediate joints along the length of the columns. The Split Member tool can be used to add joints along the physical columns.

When ACI 318-08 and newer is used the slenderness effects are *not* neglected per the provisions of Section 10.10.1.

Walls in RISA-3D

In order to meet the requirements of Section 10.10.4 (thereby meeting the requirements of Section 10.10.2) you must turn on P-Delta in the Load Combinations spreadsheet and include P-Delta for walls in the Global Parameters. The effects of P-Little-Delta are accounted for in walls using the Non-Sway Moment Magnification Procedure of ACI 318-11 Section 10.10.6.

Limitations

- The provisions of Section 10.10.2.1 (P-Delta moments should not exceed 1.4 times non-P-Delta moments) are not considered.

Plates/Shells

The plate/shell finite element allows you to easily model shear walls, diaphragms, shells, tanks and many other surface structures. We refer to the elements as plate elements, but they are actually plate/shell elements. Plate data may be viewed and edited in three ways: graphically, in the **Information** dialog or in the **Plates** spreadsheet.

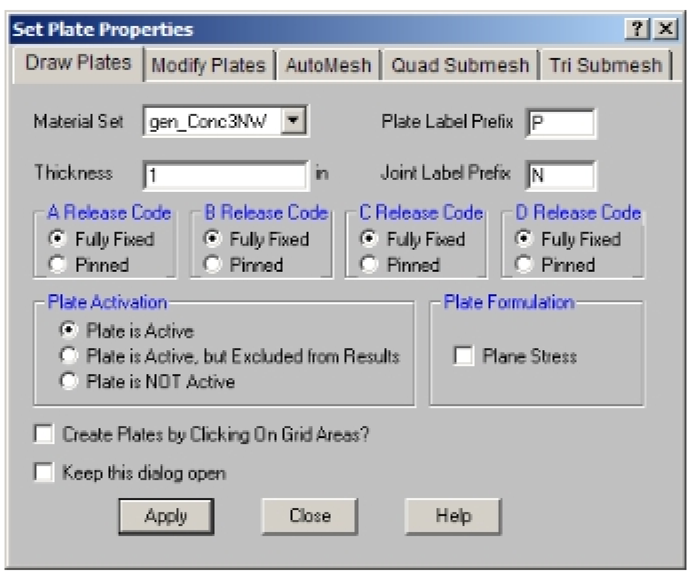
Drawing Plates

There are several graphic-editing features that make the creation and modification of models quite easy. Use the Insert and Modify menus or the Drawing Toolbar to use these features in the model view. To create new members or plates, you can draw them using a drawing grid or draw "dot to dot" from existing joints. Once you have created these items you may use other graphic features to load the model and set boundary conditions.

Creating plate models requires more forethought than beam models. See Plate Modeling Tips and Plate Modeling Examples for tips on building plate models. To create new plates you can draw them using a drawing grid, a project grid, or draw "dot to dot" from existing joints. You can set all of the plate properties up front or you can modify these properties after you draw them. Modifying properties is discussed in the next sections. See Plate Spreadsheet for information on plates and their properties.

The **Draw Plates** tool lets you graphically draw plates in your model. Enter the appropriate plate parameters, click **OK** and draw plates between existing joints or on the drawing grid. You will also notice that the coordinates of the joint or grid point that is closest to your cursor are displayed in the lower right hand corner of the model view. The new plates will be shown on screen and will be recorded in the **Plates Spreadsheet**.

To actually draw a plate, you have two options. The fastest way is to use the **Create Plates by Clicking on Grid Areas** option, and then create plates by clicking on the grid areas formed by the intersecting grid lines. As you click on an area, a plate will automatically be created in that area. The second option is to create plates by drawing them one joint at a time. You click on the grid point or joint that you want to be the "A" joint for the plate, then you click on the "B" joint, "C" joint, and then the "D" joint in either clockwise or counter clockwise order. The plate will "stretch" like a rubber band as you draw from joint to joint.



The parameters shown are the same parameters that you would enter on the **Plates Spreadsheet**. For help on an item, click ? and then click the item.

To Draw Plates

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. If you are not drawing between existing joints, you will need to create a drawing grid or define joints on the **Joint Coordinates** spreadsheet.
3. Click the **Draw / Modify Plates** button and select the **Draw Plates** tab. Then set the plate properties. For help on an item, click and then click the item.
    a. You must click four points in a clockwise or counter-clockwise order. To create a triangular plate click on the third joint twice.
    b. If in step 3 you chose to click in grid areas then you create plates by clicking between the drawing grids.
4. Click **OK** to start drawing plates by clicking on the joints or grid points with the left mouse button.
5. To stop drawing altogether right click or press the **Esc** key.

Note

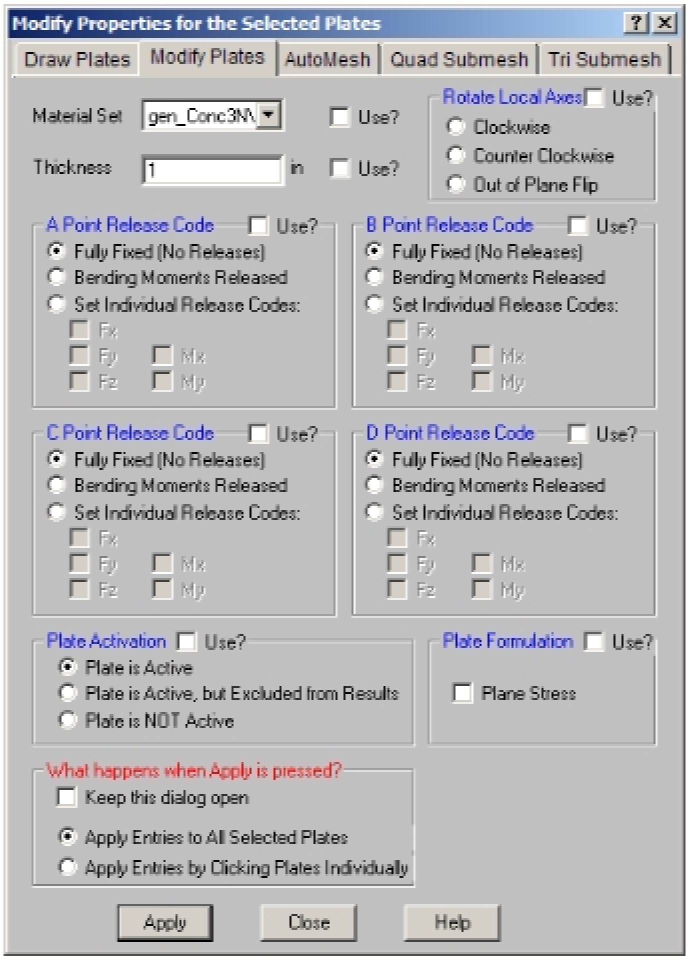
- To draw more plates with different properties, press CTRL-D to recall the **Plate Properties** settings.
- You may also specify or edit plates in the **Plates Spreadsheet.**
- You may also view and edit plate properties by double-clicking on a plate.
- You may undo any mistakes by clicking the Undo button.

Modifying Plates

There are a number of ways to modify plates. You may view and edit the member data in the Plates Spreadsheet. You may double-click a plate to view and edit its properties. You can use the Modify Members tool to graphically modify a possibly large selection of members.

The graphical Plate Modify tool discussed here lets you modify the properties of plates that already exist in your model. To use this, you will typically specify the properties you want to change, then select the plates that you want to modify. You can modify plates one at a time by selecting the **Click to Apply** option and then click on the plates you wish to modify. You may also modify entire selections of plates by selecting the plates first and then use the **Apply to Selected** option. See the Graphic Selection topic for more on selecting.

The parameters shown are the same as those used to define new plates. For help on an item, click and then click the item.



The **Use?** check boxes next to the data fields indicate whether the particular parameter will be used or not when the modification is applied. If the box next to a field is checked, that parameter will be applied to any selected plates. If the box is NOT checked, the parameter will NOT be applied, even if a value is entered in the field. This lets you easily change one or two properties on members without affecting all the rest of the properties. Note that if a no value is entered in a field (I.e. the field is blank) and the corresponding check box is checked, clicking “Apply” will have the effect of clearing the data for these fields.

Rotating Plates

These options allow the user to perform a clockwise, or counter clockwise rotation so that they can better align the plate local axes. It also allows the user to flip the local z-axis so that it is headed in the other direction.

To Modify Plates

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Click the **Draw / Modify Plates** button and select the **Modify Plates** tab. Then set the parameters for the new plates. Check the **Use?** Box for the items to apply. For help on an item, click and then click the item.
3. You may choose to modify a single plate at a time or to an entire selection of plates.
   a. To modify a few plates choose **Apply Entry by Clicking Items Individually** and click **Apply**. Click on the plates with the left mouse button.
   b. To modify a selection, choose **Apply Entries to All Selected Items** and click **Apply.**

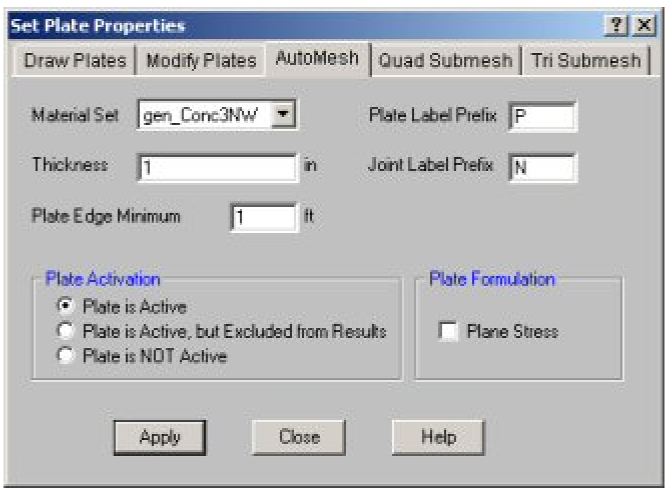
Note

- To modify more plates with different parameters, press CTRL-D to recall the **Modify Plates** settings.
- You may also modify plates in the **Plates** Spreadsheet.
- You may undo any mistakes by clicking the **Undo** button.
- It may not be possible to perfectly align the plate local axes.

Submeshing Plates

AutoMesh Plates

The AutoMesh tool allows you to draw a polygon that RISA-3D will automatically submesh into smaller quadrilateral plate elements. Just as with Drawing Plates, the material set and plate thickness of the plates within the mesh may be indicated prior to drawing the polygon. In addition to these parameters, a plate edge minimum can also be provided. Polygons of virtually any size and shape may be drawn provided the drawing lines do not cross. Polygons may be drawn in either a clockwise or counter-clockwise direction. To complete a polygon, simply double click on the last joint/grid intersection, or click on the starting joint/grid intersection. Once a polygon is drawn, RISA-3Dwill create a submesh of quadrilateral plate elements, limited by the edges of the polygon, and of a size corresponding to the plate edge minimum indicated by the user. Note that only quadrilateral plate elements are created with the AutoMesh tool.



When drawing a polygon with the AutoMesh tool, any existing joints within the boundary and in the plane of the polygon will be considered control points. These **Control Points** will be considered "fixed" points within the mesh and will dictate the layout of individual plates surrounding them. It is important to note that only currently selected joints at the time the polygon is drawn will be used as control points.

The AutoMesh tool will attempt to use the Plate Edge Minimum that the user enters. However, if it cannot successfully create a valid mesh, it will automatically re-set the Plate Edge Minimum entry to an edge minimum equal to the minimum distance between two control points.

To AutoMesh a Polygon

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. If you are not drawing between existing joints, you will need to create a drawing grid or define joints on the **Joint Coordinates** spreadsheet.
3. Make sure that all joints that are to be considered **Control Points** are selected.
4. Click the **Draw / Modify Plates** button and select the **AutoMesh** tab. Then set the plate properties including the plate edge minimum. For help on an item, click and then click the item.
5. Click **OK** to start drawing a polygon by clicking on the joints or grid points with the left mouse button.

6. To complete the drawing of a polygon, double click the last point or click on the original starting point.
7. To stop drawing altogether, click the button, right click or press the **Esc** key.

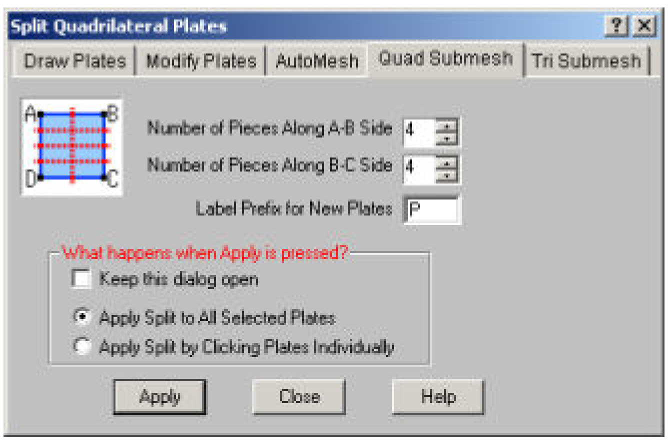
Note

- The AutoMesh feature is only available for plates that lie entirely within one of the global planes (XY, XZ, or YZ). A future update may extend this feature into co-planar area which do NOT lie in the main global planes.
- To draw more plates with different properties, press CTRL-D to recall the **Plate Properties** settings.
- You may also specify or edit plates in the **Plates Spreadsheet.**
- You may also view and edit plate properties by double-clicking on a plate.
- You may undo any mistakes by clicking the Undo button.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Plate Mesh**.

Quadrilateral Plates

You can submesh quadrilateral (4 sided) plate elements into a mesh of smaller elements. This new mesh can be any size up to the program limits for joints and/or plates. This is very useful for refining a coarse mesh of elements, just make sure that all adjacent plate elements (elements sharing an edge) maintain connectivity.



You can define different submesh increments in each direction. The A,B,C and D joints for each plate are displayed in the plates spreadsheet. You can determine which side is which by displaying the plate local axes and realizing that the local "x" axis is parallel to the D-C edge of the plate. The A joint is the first joint clicked on when you created the plate. The B joint is the second and so on.

You can submesh the plates one at a time by selecting the **Click to Apply** option and then clicking on the plates you wish to submesh. You may also modify entire selections of plates by selecting the plates and then using the **Apply to Selected** option.

To Submesh Quadrilateral Plates

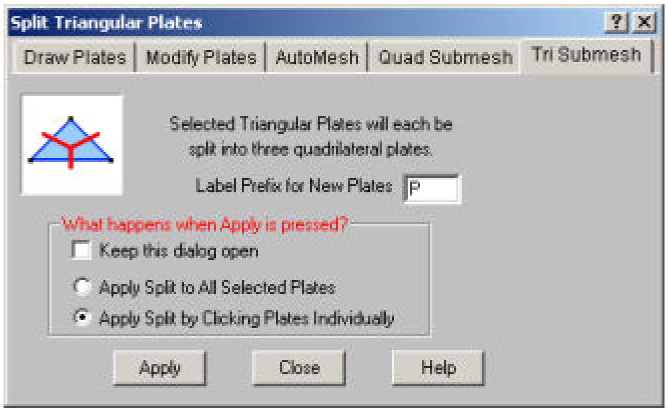
1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Select the plates you want to sub mesh.
3. Click the **Draw / Modify Plates** button and select the **Submesh Quads** tab. Then specify the number of plates. For help on an item, click and then click the item.

Note

- To submesh more plates with different parameters, press CTRL-D to recall the **Submesh Plates** settings.
- You may undo any mistakes by clicking the **Undo** button.

Triangular Plates

This is used to sub-mesh the selected triangular (3 sided) elements into a mesh of 3 quadrilaterals. This is done by first creating a new joint at the center of each selected triangular element and also at the center point along each edge of the triangular element. These new joints are then used to create three quadrilateral elements that replace the triangular element.



<u>To Submesh Triangular Plates</u>

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Select the plates you want to sub mesh.
3. Click the **Draw / Modify Plates** button and select the **Submesh Tris** tab.. For help on an item, click and then click the item.

Note

- To submesh more plates with different parameters, press CTRL-D to recall the **Submesh Plates** settings.
- You may undo any mistakes by clicking the **Undo** button.

Plates Spreadsheet - Primary Data

The Plates Spreadsheet records the properties for the plate/shell elements of the model and may be accessed by selecting Plates on the Spreadsheets menu.

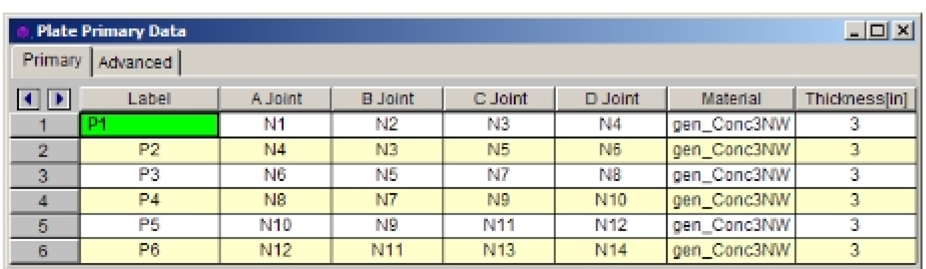


Plate Primary Data

Primary | Advanced

	Label	A Joint	B Joint	C Joint	D Joint	Material	Thickness[in]
1	P1	N1	N2	N3	N4	gen_Conc3NW	3
2	P2	N4	N3	N5	N6	gen_Conc3NW	3
3	P3	N6	N5	N7	N8	gen_Conc3NW	3
4	P4	N8	N7	N9	N10	gen_Conc3NW	3
5	P5	N10	N9	N11	N12	gen_Conc3NW	3
6	P6	N12	N11	N13	N14	gen_Conc3NW	3

The following data columns hold the **Primary** data for the plates:

Plate Labels

You may assign a unique label to any or all of the plates. You can then refer to the plate by its label. Each label has to be unique, so if you try to enter the same label more than once you will get an error message. You may relabel plates at any time with the Relabel Plates option on the Tools menu.

Plate Joints

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

The direction and sequence in which you define the joints determines how the elements local coordinate system is set up. This is discussed in the section on Plate Local Axes.

Plate Material

The material set label links the plate with the desired material defined on the Material Spreadsheet.

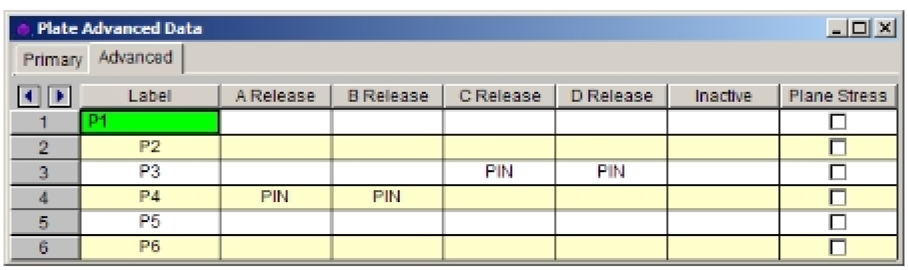
Note

- Plates are always defined with general materials. This is because the other material sets (Hot Rolled, Cold Formed, Wood, and Concrete) are used to designate member code checking specifications. Since plates are only used for analysis, no code checking is provided and the material must be designated as a general material.

Plate Thickness

The thickness field on the Plates spreadsheet is the thickness of the element. This thickness is constant over the entire element.

Plates Spreadsheet - Advanced Data



	Label	A Release	B Release	C Release	D Release	Inactive	Plane Stress
1	P1						☐
2	P2						☐
3	P3			PIN	PIN		☐
4	P4	PIN	PIN				☐
5	P5						☐
6	P6						☐

The following data columns hold the **Advanced** data for the plates:

A, B, C, & D Release

The **Plate Corner Releases** for joints A, B, C, and D of each plate may be set in these four data columns. See Plate Corner Releases for more information.

Inactive

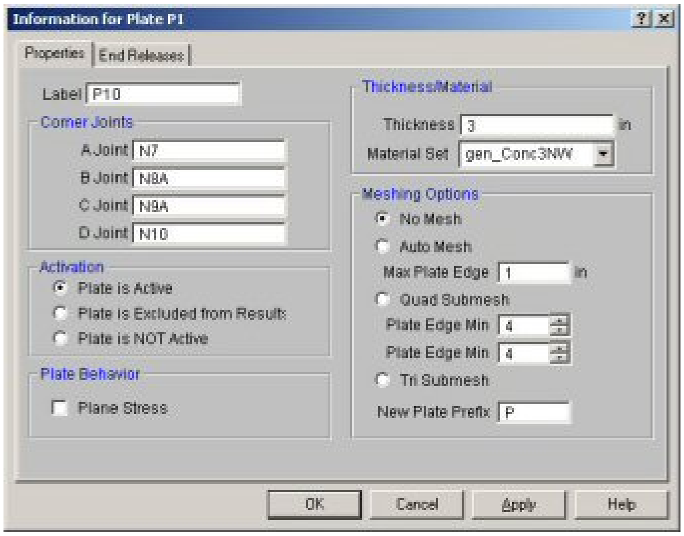
The **Inactive** data column allows for plates to be set to 'Active', 'Inactive', or 'Excluded'. These choices may be made by selecting them from the drop down list or by entering an 'I' for 'Inactive' or an 'E' for 'Excluded'. If the cell is left blank the plate will be 'Active' by default. See Inactive and Excluded Plates for more information.

Plane Stress

The plate formulation may be changed to indicate that the plate is a **Plane Stress** plate by checking the box in this data column. This is useful for creating diaphragms that have only in-plane stiffness and will not attract out of plane shear or moment. The default is to have this box unchecked which allows for both in-plane and out of plane behavior.

Plate Information Dialog

Just as with the joints and members you may double-click any plate to view it's properties. All of the same information that is stored in the **Plates Spreadsheet** is displayed for the plate you choose, and may be edited. This is a quick way to view and change plate properties. For large selections of plates however the spreadsheet and graphic editing tools may be the faster solution.



Label - You can view and edit the plate label.

Corner Joints - The corner joint labels are displayed for you to view or edit.

Thickness Material - The plate thickness (in the current dimension units) and the material may be viewed or edited.

Activation - The activation state of the plate may be changed. If the plate is made inactive, you will need to activate the plate from the Plates spreadsheet, or by using the Criteria Select feature to find and select inactive plates.

Plate Behavior- The plate formulation may be changed to indicate that the plate is a Plane Stress plate. This is useful for creating diaphragms that have only in-plane stiffness and will not attract out of plane moments. The default is to leave this box unchecked which allows for both in-plane and out of plane behavior.

Meshing Options - The section allows you to mesh the current plate. You may choose between the auto-mesh function or the quad and tri submesh. Refer to the section on Modifying plates for information on meshing.

Note

- It's generally more efficient to use the Graphic Editing features if you want to change the properties for many plates at once.

Plate Corner Releases

The **A, B, C, & D Release** fields are used to designate whether the forces and moments at the corners of the plate are considered fixed to or released from the plates's points of attachment (the A, B, C, and D joints). Each plate has 5 force components at each corner (Fx, Fy, Fz, Mx, and My). Any or all of these force components can be released from the plate's point of attachment. If a force component is released, that force is not transferred between the joint and the plate.

To specify plate corner releases go to the **A, B, C, or D Release** fields for the plate on the **Advanced Tab** of the **Plates Spreadsheet**, click the button, and specify the condition.

Alternatively, you may specify the corner condition by directly typing in the field. To indicate that a force component is released, put a **'X'** for that component in the release field. You can move within the release field using the space bar which will result in a **'O'** for no release.

RISA-3D has a special "keyword" release configuration built-in. That is:

PIN => Mx and My (all moments) released (OOOXX)

This keyword entry is included because 99% of the release configurations you'll ever want to define will be "PIN". You can call out the keyword entry by just entering the first letter of the keyword, "p". So if you go to a release field and enter "p", the keyword "PIN" will be filled in automatically.

Note

- It's generally more efficient to use the **Graphic Editing** features if you want to change the properties for many plates at once.

Inactive and Excluded Plates

Making an item such as a member or plate inactive allows you to analyze the structure without the item, without having to delete the information that defines it. This leaves data intact so the item may be easily reactivated. This is handy if you want to try a model with and then without certain items, without having to actually delete the data.

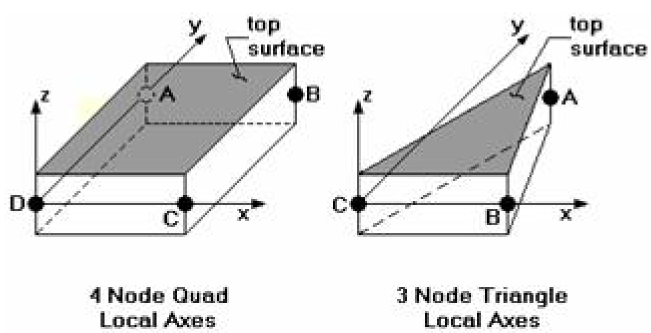
Putting a "y" in the Inactive field makes the item inactive, i.e. the item is not included when the model is solved or plotted.

Another option is to put an "E". The "E" code means include the item in the solution, but exclude it from the results list. So, an item with an "E" in the "Inactive?" field will be treated like any other plate in the solution and plotting of the model, but the plate will not be listed in the solution results (forces, stresses etc.). This is useful if there are certain items whose results you're not interested in. You don't have to clutter up the results with these items and can concentrate on the items you're most interested in. See Printing for more limiting printed results.

Plate Local Axes

The A, B, C, and D joints are used to define the corners of a quadrilateral element. (To define a 3-joint element, just leave the D joint blank, or make it the same as the C joint.) The joints must all lie on the same plane and be entered in either a clockwise or counter-clockwise direction.

The direction and sequence in which you define the joints determines how the elements local coordinate system is set up. The following diagrams illustrate how the elements local coordinate system is related to the joint numbering sequence and direction:



The local x-axis is defined as positive from the D joint towards the C joint for 4 joint elements and from C towards B for 3 joint elements. The local y-axis is then placed as close to pointing towards the A-joint as possible. Note that for triangular elements, the y-axis will probably not pass through the A-joint. For 3 joint elements, the y-axis is "towards" the A-joint and perpendicular to the x-axis. Once the x and y axes are defined, the positive local z-axis is found using the right hand rule.

Plate/Shell Element Formulation

The element used is a mixed interpolation 4 joint quadrilateral element. By mixed interpolation, we mean that the in-plane and transverse shear strain components are derived independently. This allows the element to be easily simplified into a plane stress element in cases where transverse shear and bending are not desired. A reference for this element is Finite Element Procedures, by K.J. Bathe, Prentice-Hall, 1996. The book also provides many references for papers on the elements convergence and other characteristics. In brief, the element can model isotropic behavior for plane stress, plate bending and out-of-plane transverse shear.

This is accomplished by starting with the Mindlin-Reissner plate assumptions and adding interpolating functions for the out-of-plane transverse shear. This approach is analogous to incorporating shear deformation with flexural effects in beam theory. This results in an element that can be used for thin and thick plate applications. Traditional plate elements do not model out-of-plane transverse shear well (if at all) and cannot be used for thick plate applications. The element is also very insensitive to distortion.

RISA-3D also provides a 3-joint triangle element that can be used to build transitional meshes. The stress characteristics of the triangle are not as accurate as the 4-joint quad and use of the triangle should be limited. It is not recommended that the stresses from the 3-joint triangle be used at all. In fact, RISA-3D's AutoMesh tool will only create quadrilateral plate elements for this very reason. RISA-3D provides a way to convert your triangular plates to quadrilaterals, see Submeshing Triangular Plates.

Orthotropic Behavior

The RISA plate element allows a limited degree of Orthotropic material behavior. Specifically, the In-Plane shearing of the plate will be almost entirely controlled by the G value for the material whereas the direct In-Plane compressive stiffness will be controlled by the Evalue of the material.

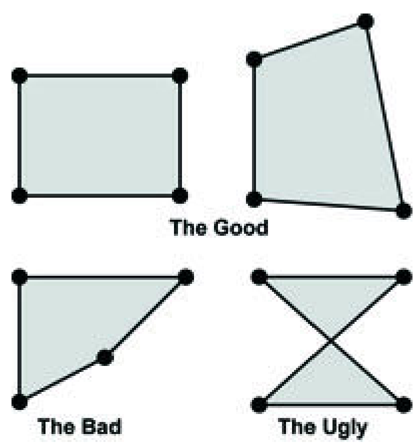
Plate Modeling Tips

A word of caution is in order if you are new to plate modeling. Unlike modeling with beam elements, plate elements require some understanding of finite element behavior to successfully obtain meaningful results. It is easy to build a finite element model using the powerful generators and graphic editing tools. However, without understanding the limitations of the analysis method used, you can end up with an impressive looking but very inaccurate model. Even if you’ve been engaged in structural engineering for years, modeling with plates is not something most engineers do frequently. It is therefore not realistic to have the expectation that you should be able to perform complicated analysis with plate elements in a short amount of time. Good plate modeling takes time, knowledge of plate and finite element behavior, and experience.

The first tip is to read all the Plate documentation before embarking on an ambitious modeling project. This will save you much aggravation down the road.

Plate Distortion

The finite elements in a model should be as undistorted as possible. See the following figure:



Non-Planar Plates

The plate element formulation used in RISA is not very sensitive to distortion within the plane of the plate itself (see the plate Distortion topic above). However, the plate element formation is particularly sensitive to non-planar plates. This means that it is critical that all 4 joints that define a quadrilateral plate remain in the same plane. There is an internal tolerance of 0.01 inches built into the plate element formulation for maximum allowable planar distortion. This is appropriate for concrete slabs and shear walls and such. But, extra care should be given to avoiding out-of-plane plates for extremely small and thin plates.

Plate Generation

A fast way to build a new mesh of finite elements is with the generation features. RISA-3D currently provides several generation features to quickly build common structures providing an easy way to create cylinders, cones, grids, radial grids and disks of plates. The best way to see what these features do is to experiment with them. See Generation to learn more about generation.

Another time-saving method is to draw large elements to represent continuums such as slabs and shear walls and then use the submesh features mentioned below to refine the mesh.

Note

- Before sub-meshing, make sure that any adjacent “large” elements connect at their corner joints. That way, any subsequent sub-mesh operations will produce element meshes that automatically connect at the intermediate joints.

Automatic Plate Sub-Meshing

What if you’ve already built a model and you now decide that your finite element mesh is too coarse? To submesh elements see Submeshing Plates. Performing a Model Merge afterwards will insure that all the new elements get connected to existing beam elements and that duplicate joints get merged. See Model Merge for more information.

Finite Element Basics

While this will not be a comprehensive treatment of plate and finite element fundamentals, a review of certain key basic concepts and terminology will be valuable to the engineer who has not worked with finite elements, or has not had the opportunity to use them recently.

A place to start is with the types of forces or stresses that can occur in a plate. One term that is commonly used is “plane stress”. This term is used to describe a state of stress in a plate where all the stresses occur in the plane of the plate. A real world example would be a shear wall with forces applied only in the plane of the wall. The resulting plate forces would be just the normal stresses (Fx, Fy) and the in-plane shear stresses (Fxy). There would be no plate moments or out-of-plane shears generated.

Stress vs. Force

It should be pointed out that the results for a plate are always a stress. These stresses are multiplied by the plate thickness and the width or length to obtain a force. Note that this force obtained is just an average value for the plate, since the stress was for a point on the plate and it undoubtedly will vary throughout the plate area. The fact that the stresses vary within a plate is why a good finite element mesh is so critical to obtain accurate results. Stresses tend to vary more around point loads and supports, and less in regions that are far from supports and have a uniform load.

A different example of plate forces would be a horizontal diaphragm that is loaded only in the out-of-plane direction. The plate results would be plate moments, out-of-plane shears, but no membrane (plane stress) stresses. The reason for no membrane stresses is that there was no in-plane loading.

Sign Convention

One other comment on plate results is to point out the convention used for moments in plates. With beams, the My moment describes the moment about the local y-axis. However, with a plate element, the My moment is the moment that produces stresses in the local y-direction. The My moment in a plate is actually about the local x-axis.

Why Meshing Is Required

In a nutshell, finite elements tend to work by trying to approximate the correct deflected shape of the real world item being modeled. For example, if we are trying to model a horizontal diaphragm, simply supported on all edges, and loaded out-of-plane, our finite element model must able to approximately recreate the deflected shape of the diaphragm.

In order to do this with some accuracy, we must use a mesh of elements to represent the physical diaphragm. If we try to model the diaphragm with only one element (which is what everyone tries to do at least once), we will get very inaccurate results because one finite element cannot accurately model the deflected shape of the physical diaphragm. The multiple reasons for this are beyond the scope of this file, and if you want to understand the "why" please study a reference on finite element analysis such as Bathe's book.

The most important concept to understand is that finite elements require a certain number of free or unrestrained joints in order to produce accurate results. Using enough elements in your mesh will produce accurate results for the deflection and stresses in the structural item being modeled. The gage of "enough" for common structural elements is addressed in the Plate Model Examples section of the Reference Manual.

Distortion

Finite elements are also affected by geometric distortion. The best shape for the 4-joint quadrilateral is a square. In practice, elements are frequently distorted, which is fine as long as they aren't squashed too far out of shape. The largest internal angle should never be equal to or greater than 180 degrees, and preferably shouldn't even approach 180 degrees.

Drilling Degree of Freedom

One last item is that the element used by RISA-3D, like other plate/shell elements, cannot accurately model in-plane rotations. I.e., a plate/shell element will not provide resistance to a moment applied about the plate's local z-axis. For example, let's say you have a 4x4 grid of elements, simply supported about the edges, and you apply a joint moment to one of the internal joints so that the moment is about the local z-axis of the elements. RISA-3D will solve such a model, however you will get all zeros for the joint reactions and the element stresses. See Applying In-Plane Moments to Plates to learn how to work around this limitation.

Plates/Shells - Results

When the model is solved, there are several groups of results spreadsheets specifically for the plates.

Plate Stress Results

Access the **Plate Stresses Spreadsheet** by selecting the **Results Menu** and then selecting **Plates** ▸ **Stresses.**

Plate Principal Stresses (By Combination)

	LC	Plate Label	Loc	Sigma1[ksi]	Sigma2[ksi]	Tau Max[ksi]	Angle[rad]	Von Mises[ksi]
1	1	P1	T	126.532	1.988	62.272	1.75	125.55
2			B	-1.988	-126.528	62.27	.18	125.546
3	1	P2	T	120.552	13.154	53.699	1.641	114.543
4			B	-13.153	-120.548	53.698	.07	114.539
5	1	P3	T	119.355	16.015	51.67	1.589	112.208
6			B	-16.014	-119.351	51.668	.018	112.204
7	1	P4	T	119.303	17.6	50.851	1.572	111.549
8			B	-17.599	-119.299	50.85	.001	111.545
9	1	P5	T	119.358	17.826	50.766	1.568	111.518
10			B	-17.826	-119.354	50.764	-.003	111.515
11	1	P6	T	119.445	18.093	50.676	1.57	111.505

The plate stresses are listed for the top and bottom of each active plate. The principal stresses sigma1 (σ_1) and sigma2 (σ_2) are the maximum and minimum normal stresses on the element at the geometric center of the plate. The Tau Max (τ_{max}) stress is the maximum shear stress. The Angle entry is the angle between the element's local x-axis, and the direction of the σ_1 stress (in radians). The Von Mises value is calculated using σ_1 and σ_2, but not σ_3 which isn't available for a surface (plate/shell) element, so this Von Mises stress should be considered to be a "plane stress" value.

The equations are:

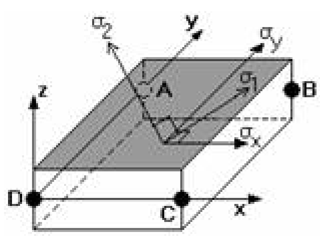


Plate Principal Stresses

$$\sigma_1 = \frac{(\sigma_x + \sigma_y)}{2} + \sqrt{\frac{(\sigma_x - \sigma_y)^2}{4} + \tau_{xy}^2}$$

$$\sigma_2 = \frac{(\sigma_x + \sigma_y)}{2} - \sqrt{\frac{(\sigma_x - \sigma_y)^2}{4} + \tau_{xy}^2}$$

$$\tau_{max} = \frac{(\sigma_1 - \sigma_2)}{2} \quad , \quad \phi = \frac{1}{2}\arctan\left[\frac{2\tau_{xy}}{(\sigma_x - \sigma_y)}\right]$$

$$Von\ Mises = \sqrt{\sigma_1^2 - \sigma_1\sigma_2 + \sigma_2^2}$$

The angle, ϕ, is the angle in radians between the maximum normal stress and the local x-axis. The direction of the maximum shear stress, τ_{max}, is $\pm$ $\pi/4$ radians from the principal stress directions.

The Von Mises stress is a combination of the principal stresses and represents the maximum energy of distortion within the element. This stress can be compared to the tensile yield stress of ductile materials for design purposes. For example, if a steel plate has a tensile yield stress of 36ksi, then a Von Mises stress of 36ksi or higher would indicate yielding of the material at some point in the plate.

The σ_x , σ_y , and σ_{xy} values used to calculate the stresses are a combination of the plate bending and membrane stresses, thus the results are listed for the top and bottom surfaces of the element. The “Top” is the extreme fiber of the element in the positive local z direction, and the “Bottom” is the extreme fiber of the element in the negative local z direction. The membrane stresses are constant through the thickness of the element, while the bending stresses vary through the thickness of the element, very similar to the bending stress distribution in a beam.

For enveloped results the maximum and minimum value at each location is listed. The load combination producing the maximum or minimum is also listed, in the "lc" column.

Note

- See Spreadsheet Operations to learn how to use **Find**, **Sort** and other options.
- See Plot Options – Plates to learn how to plot joint results.

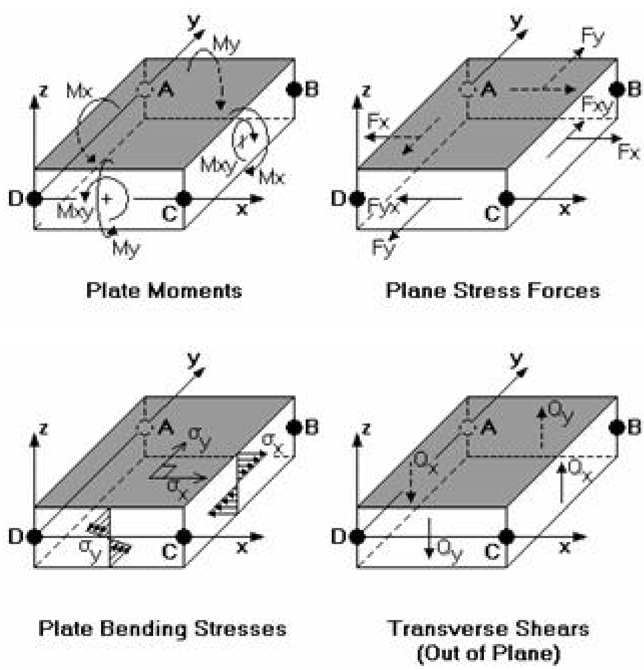
Plate Force Results

Access the **Plate Forces Spreadsheet** by selecting the **Results Menu** and then selecting **Plates** ‣ **Forces**.

Plate Forces (per ft) (By Combination)

	L...	Plate Label	Qx[k]	Qy[k]	Mx[k-ft]	My[k-ft]	Mxy[k-ft]	Fx[k]	Fy[k]	Fxy[k]
1	1	P1	7.644	-44.587	.994	20.426	-3.649	.002	.024	0
2	1	P2	.351	-36.489	2.279	20.005	-1.244	.003	.024	0
3	1	P3	2.145	-39.79	2.675	19.886	-.318	.003	.024	0
4	1	P4	-1.262	-39.466	2.933	19.883	-.022	.004	.024	0
5	1	P5	1.291	-39.821	2.971	19.892	.047	.004	.024	0
6	1	P6	-1.279	-39.846	3.015	19.907	.019	.004	.024	0
7	1	P7	1.279	-39.846	3.015	19.907	-.019	.004	.024	0
8	1	P8	-1.291	-39.821	2.971	19.892	-.047	.004	.024	0
9	1	P9	1.262	-39.466	2.933	19.883	.022	.004	.024	0
10	1	P10	-2.145	-39.79	2.675	19.886	.318	.003	.024	0
11	1	P11	-.351	-36.489	2.279	20.005	1.244	.003	.024	0

The **Plate Forces** are listed for each active plate. Interpretation of output results is perhaps the most challenging aspect in using the plate/shell element. The results for the plates are shown for the geometric center of the plate.



The forces (Q_x and Q_y) are the out-of-plane (also called "transverse") shears that occur through the thickness of the element. The Q_x shear occurs on the element faces that are perpendicular to the local x-axis, and the Q_y shear occurs on the element faces that are perpendicular to the local y-axis. Q_x is positive in the z-direction on the element face whose normal vector is in the positive x-direction. This is also the σ_x face. Q_y is positive in the z-direction on the element face whose normal vector is in the positive y-direction. This is also the σ_y face. The total transverse shear on an element face is found by multiplying the given force by the width of the element face.

The plate bending moments (M_x, M_y and M_{xy}) are the plate forces that induce linearly varying bending stresses through the thickness of the element. M_x is the moment that causes stresses in the positive x-direction on the top of the element. Likewise, M_y is the moment that causes stresses in the positive y-direction on the top of the element. M_x can then be thought of as occurring on element faces that are perpendicular to the local x-axis, and the M_y moment occurs on faces that are perpendicular to the local y axis. To calculate the total M_x or M_y on the face of an element, multiply the given value by the length of the element that is parallel to the axis of the moment. For example, looking at the 'Plate Moments' figure above, the

total M_x moment could be obtained by multiplying the given M_x force by the length of side BC (the distance from joint B to joint C). The total M_y force can be calculated in the same way by instead using the length of side DC.

The M_{xy} moment is the out-of-plane twist or warp in the element. This moment can be added to the M_x or M_y moment to obtain the 'total' M_x or M_y moment in the element for design purposes. This direct addition is valid since on either the top or bottom surface, the bending stresses from M_{xy} will be going in the same direction as the M_x and M_y moments.

Note

- For the placement of concrete reinforcement, it is helpful to realize that laying reinforcement parallel to the local x-axis will resist the M_x moment.
- A positive M_x or M_y moment will put the top fiber of the plate in tension.

The plane stress forces (F_x , F_y and F_{xy}) are those forces that occur in the plane of the plate. These forces, which are also called "membrane" forces, are constant through the thickness of the element. F_x and F_y are the normal forces that occur respectively in the direction of the local plate x and y-axes, positive values indicating tension. These forces are reported as a force/unit length. To get the total force on an element, you would need to multiply the given value by the length of the element that is perpendicular to the normal force. For example, looking at the 'Plane Stress Forces' figure, the total Fx force could be obtained by multiplying the given F_x force by the length of side BC (the distance from joint B to joint C).

The F_{xy} force is the in-plane shear force that occurs along the side of the element. The subscript 'xy' indicates that the shear occurs on the face of the element that is perpendicular to the x-axis and is pointing in the y-direction. F_{yx} is the complementary shear force, where the subscript 'yx' indicates that the shear occurs on the face of the element that is perpendicular to the y-axis and is pointing in the x-direction. RISA-3D only gives values for F_{xy} because F_{xy} and F_{yx} are numerically equal. The total in-plane shear can be obtained by multiplying the given force value by the length of the element that is parallel to the shear force. For example, when looking at the 'Plane Stress Forces' figure, the total F_{xy} force which is parallel to the local y-axis could be obtained by multiplying the given F_{xy} force by the length of side BC.

Note that the plate bending (Q_x, Q_y, M_x, M_y, M_{xy}) and membrane (F_x, F_y, F_{xy}) results are forces per unit length. For example, a rectangular element with a B to C length of 10 feet showing a F_x force of 20K would have a total normal force on the B-C face of the element of 20K (per foot) times 10 feet, or 200K.

For enveloped results the maximum and minimum value is listed. The load combination producing the maximum or minimum is also listed, in the "lc" column.

Note

- See Spreadsheet Operations to learn how to use **Find**, **Sort** and other options.
- See Plot Options – Plates to learn how to plot joint results.

Plate Corner Force Results

Access the **Plate Corner Forces Spreadsheet** by selecting the **Results Menu** and then selecting **Plates** ▸ **Corner Forces**.

Plate Corner Forces (By Combination)

	LC	Plate Label	Joint	X[k]	Y[k]	Z[k]	MX[k-ft]	MY[k-ft]	MZ[k-ft]
1	1	P1	N58	-.002	.016	-34.879	0	0	0
2			N60	0	.014	-19.708	1.868	1.924	0
3			N72	.002	-.01	17.352	22.061	2.892	0
4			N71	0	-.008	17.235	20.658	2.828	0
5	1	P2	N60	-.002	.015	-22.66	-1.868	-1.924	0
6			N61	.001	.015	-23.829	.108	.758	0
7			N73	.002	-.009	14.181	20.045	1.696	0
8			N72	-.001	-.009	12.309	18.205	-.18	0
9	1	P3	N61	-.002	.015	-26.592	-.108	-.758	0
10			N62	.002	.015	-23.198	.116	2.025	0
11			N74	.002	-.009	15.342	19.983	1.723	0
12			N73	-.002	-.009	14.448	19.798	-.844	0
13	1	P4	N62	-.002	.015	-23.632	-.116	-2.025	0
14			N63	.002	.015	-25.834	-.034	.961	0
15			N75	.002	-.009	14.572	19.848	1.341	0
16			N74	-.002	-.009	14.894	19.768	-1.54	0

The plate corner forces are the global forces at the corner of each plate and are listed for each active plate.

These are the forces and moments calculated at the corners of the plates, in the GLOBAL directions. These values are obtained by multiplying the plate's corner displacements with the global stiffness matrix. Unlike the local stresses and forces, which are very accurate approximations, these corner forces represent EXACT results based on linear elastic theory. Also, the local forces are listed on a 'per unit length' basis, whereas these global direction corner forces represent the total force on the plate at the corner in the given direction, very similar to beam end forces. At any given joint, the corner forces for all plates connected to that joint should sum to zero (a requirement of equilibrium), assuming no members or boundary conditions are also present at the joint.

As an example of how to use these corner forces, you can obtain the total shear at a given level in a shear wall by adding the proper corner forces for the plates at that level. See Plate Modeling Examples to learn how to use the plate corner forces to get shear wall story shears and moments, as well as slab moments and shears.

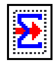
For enveloped results the maximum and minimum value is listed. The load combination producing the maximum or minimum is also listed, in the "lc" column.

Note

- See Spreadsheet Operations to learn how to use **Find**, **Sort** and other options.
- See Plot Options – Plates to learn how to plot joint results.

Plates/Shells - Design Tools

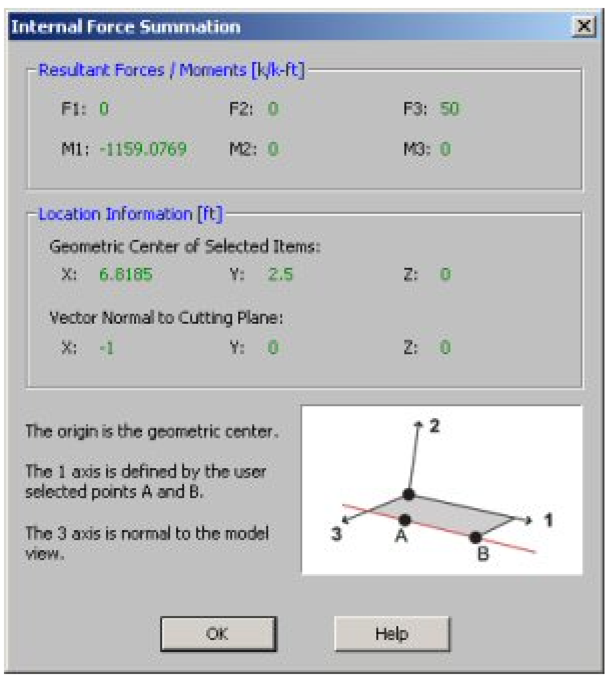
Internal Force Summation Tool

This tool may be activated by clicking the  icon below the Selection Toolbar or by using the Right-Click menu from within a model view. The tool is only available from within a model view that has active results for a single combination or a batch solution. This tool is not available for an envelope solution.

The Internal Force Summation Tool can be used to quickly come up with information on story shears in a building, or internal moments in a elevated or on grade slab.

When initiated, the tool requires the user to select two points (A and B). These points will be used to define the plane in which the internal force summation will be performed. That defines a "cutting plane" perpendicular to the screen. A red line will appear to indicate the edge of this plane. As you slide the cutting plane back and forth, the status bar at the bottom of the screen will give a summary of the internal forces for the currently selected items. These forces are reported at the cutting plane location with respect to the local 1,2,3 axes that defined the cutting plane.

Clicking on a third point will lock the cutting plane to that exact location and provide a detailed summary of the internal forces at that location. These summary results are summarized below:



The Internal Force Summation report is separated into three regions. The first region gives the resultant forces and moments. The second region gives information needed to locate the reference plane and origin of local axes. The third region gives a basic reference for the orientation of the local axes and forces.

Since the **Forces** and **Moments** are summed only for the selected members, this tool can be easily used to determine the overall story forces in a shear wall or moment frame (if the whole model is selected), as well as the forces in an individual bent or pier (if only a portion of the model is selected).

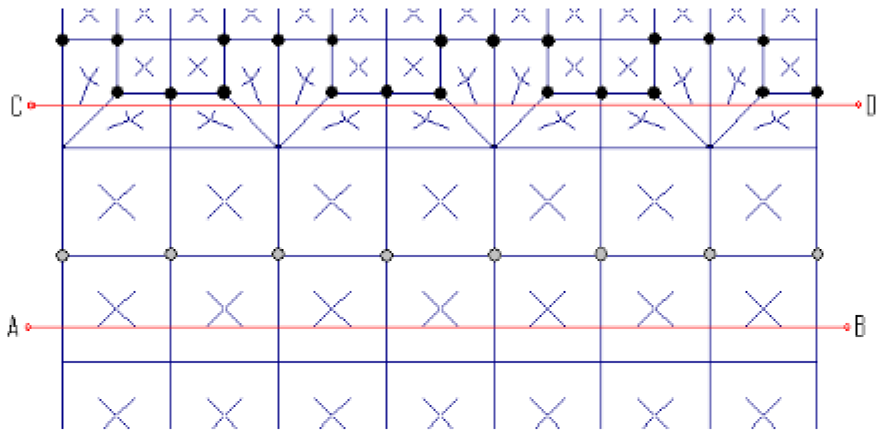
The **Geometric Center of the Selected Items** defines the origin at which the forces (F1, F2, and F3) are reported. This also corresponds to the point used to define the moments (M1, M2 and M3). These forces and moment are all given with respect to the local 1, 2, 3 axes. The local 3 axis is always perpendicular to the current model view. The local 1 axis is always defined parallel to the points A and B which were are selected by the user. The 2 axis is then defined by the right hand rule.

The **Vector Normal to the Cutting Plane** is used to define the plane where the internal force summation was performed. This was determined by the user selected points A and B.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Story Shears**or **IFST**.

Calculation and Theory

When cutting through a plate, the internal force summation tool uses the plate corner forces to derive the force in the cutting plate. When cutting through the interior of a plate, any corner node that is "above" the cutting plane is included in the summation. Moments are then interpreted based on the location of the forces compared to the centroid of the cutting plane. See the figure below:

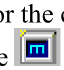


Consider the cutting plane A-B defined above. This was created by clicking from left to right on the page (from A to B). Therefore, there are 8 nodes above cutting plane A-B. These are highlighted in a gray color and are the nodes whose corner forces will be used to create the base value of the cutting plane force. Because the cutting plane is below the row of nodes, and because there may be applied surface or self weight loading at the plates, some interpolation must used between the values above the plane and the values below the plane. This consists of essentially a linear interpolation.

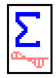
For cutting plan C-D there are 22 nodes above the cutting plane that will be used to determine the base value of the cutting plane force.

If the cutting plane were defined from right to left (i.e. from B to A), then the program will actually reverse the local 1 axis as if the user defined the plane by clicking from A to B.

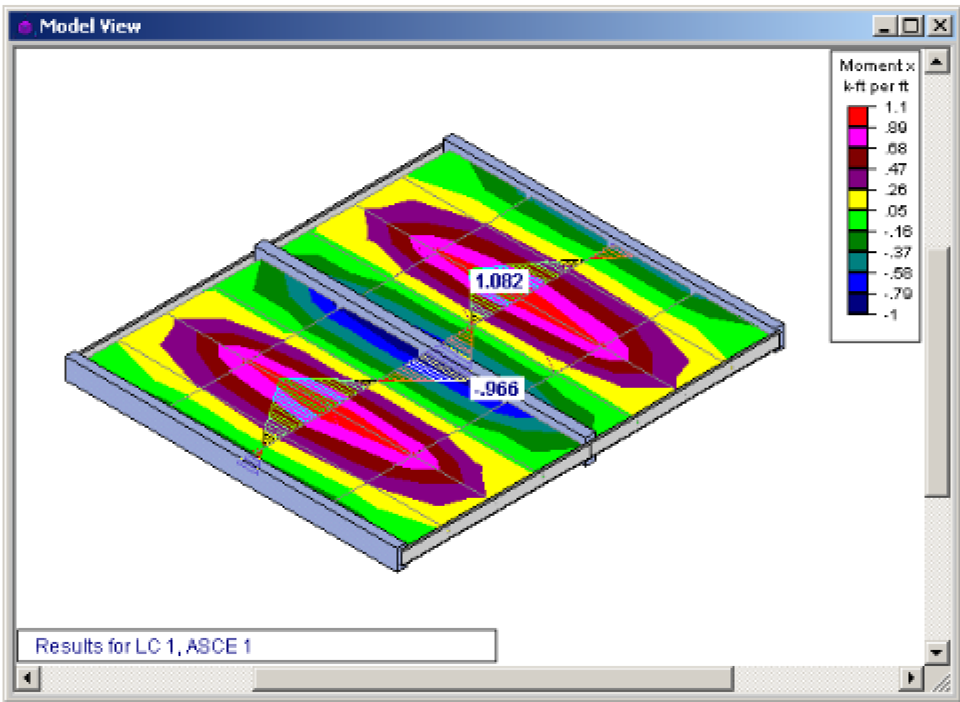
Note

- This tool only computes forces for elements that are selected in your model. In the same sense, it will sum up forces for **everything that is selected** that is in the cutting plane. You may use the selection tools to unselect portions of the model for which you do not want force results for.
- Because these forces are reported with respect to the local 1,2,3 axes of the cutting plane it is NOT recommended that this tool be used from an isometric view of the structure. It will function best in a pure plan or a pure elevation view.
- The results for the internal force summation tool are always given for the currently displayed results. The currently displayed results show up in the Plot Options selection (the  icon). If the model has not been run and there are no results, then the tool will be unavailable. Similarly, the tool is not available with Envelope Results.

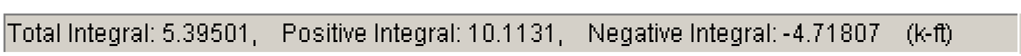
Contour Display Details

This tool may be activated by clicking the  icon below the Selection Toolbar or by using the Right-Click menu from within a model view and selecting **Make a Cut Over Displayed Contour**. The tool is only available from within a model view that is currently plotting a plate contour.

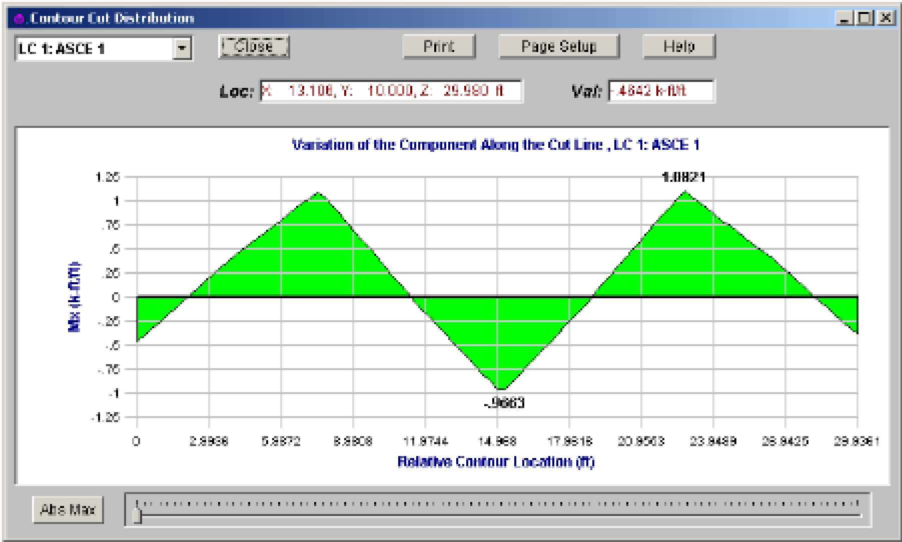
The Contour Display Detail is used to turn a visual plate contour into a more traditional shear or moment diagram. The tool merely transforms the visual contour display into actual numerical that can be viewed by the user. In the image below, the contour display shows the numerical values associated with the contour at the cutting line.



This tool can also be used for curved plate models such as tanks and vessels. The integral of the force diagram is displayed in the Status Bar at the bottom of the screen. This may be useful for summing up shear forces in a wall or lintel.



When a Contour Display Detail is shown, the right click menu gives access to a Detailed Diagram of the contour cut as shown below:



This detailed diagram will show you all the results for every section along the length of the diagram. this information may even be copied to the clipboard for use in a spreadsheet program. This can be done by selecting **Copy Data** from the right click menu.

Note

- Because this display is just a currently displayed plate contours, the units for the detailed diagrams are always the same as for the displayed contours.

//

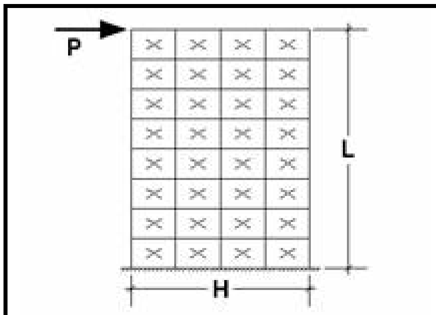
Plates/Shells - Modeling Examples

The finite element method is an approaching method. This means that the results will never be exact and can be made better and better, within reason, by sub-dividing the element mesh. Accurate results are dependent on the modeling of the problem.

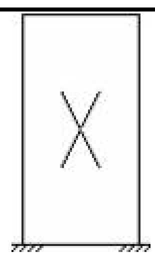
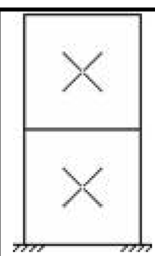
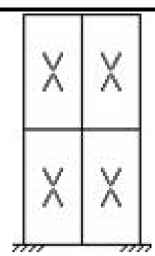
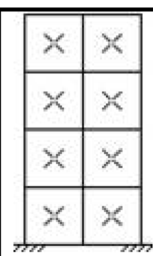
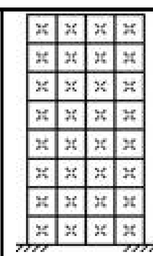
The following pages are studies of finite element mesh fineness and its relationship to accurate stress and deflection results. These studies are meant to be an aide to help you select appropriate mesh fineness for a structure you are trying to model. These studies will also answer the "why" many people ask when told they must use a "mesh" of elements to model a structural item (such as a shear wall) instead of using one giant element. Obviously these studies only give an overview of some basic elements and the engineer must be the final judge as to whether a specific finite element model is a good reflection of the "real" structure.

Shear Wall Modeling

Theoretical Deflection of Shear Wall with Point Load

	Shear Wall Properties L = 240 in Area = 1440 in2 B = 12 in H = 120 in. E = 4000 ksi $\nu = 0.30$ G = 1538.5 ksi P = 15,000 kip	$I = BH^3/12$ $= 12(120)^3 / 12$ $= 1{,}728{,}000\ in^4$ $\Delta = PL^3/3EI +$ $1.2\ PL / AG$ $= 11.95$ in $K = P / \Delta$ = 15,000k /11.95in = 1255 kips/in

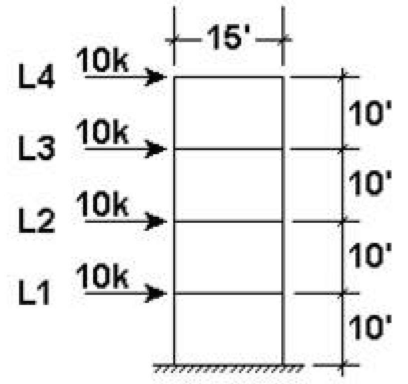
Stiffness as a Function of Mesh Fineness

					
Element Mesh	1x1	1x2	2x2	2x4	4x8
Deflection (in.)	4.54	8.07	8.26	10.43	11.29
Error	62%	33%	31%	13%	6%
Stiffness K (kips/in)	3304.0	1858.7	1816.0	1438.2	1328.6

Note on Methodology:

Since the theoretical solution is based on an assumption that plane sections shall remain plane after deformation, the last model (4x8 mesh) had very stiff axial members included at the 2nd, 4th, 6th and top level across the width of the wall. This prevented horizontal differential joint movement and allows for a more meaningful comparison with the theoretical solution.

Shear Wall Design Forces



Floor	Shear	Moment	Elements
4	9.99k	100.05k-ft	P49-P52
3	20k	300.08k-ft	P33-P36
2	30k	600k-ft	P17-P20
1	40k	999.98k-ft	P1-P4

Shown above are the analysis results of a 4-story shear wall. This example is for a straight shear wall, however the method and results are valid for box, channel, or any other shear wall shapes.

The RISA-3D files that were used to obtain these results are included as “4X1WALL.R3D” and “4X4WALL.R3D”. The 10 kip story loads were applied uniformly across each story. This was done to more accurately model loads being applied to the wall from a rigid or semi-rigid floor.

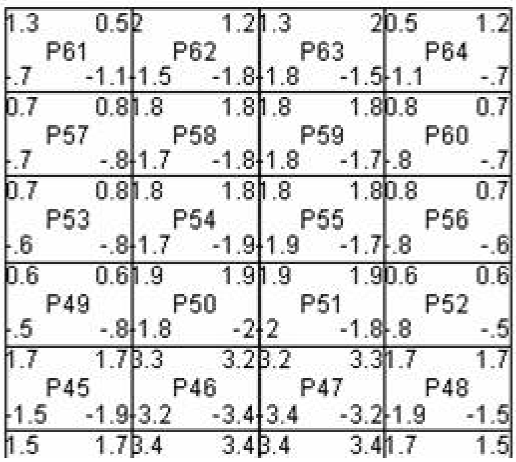
The story shears at each level were calculated as the sum of the FX corner forces. The story moments at each level are calculated from the FY corner forces as shown below:

Mi = (Fyouter joint * 15ft) + (Fyinner joint * 7.5ft)

Story Shears - Hand Calculation

The story shears were calculated as shown below from the corner forces. See the screen shot close up of the FX corner forces on the next page.

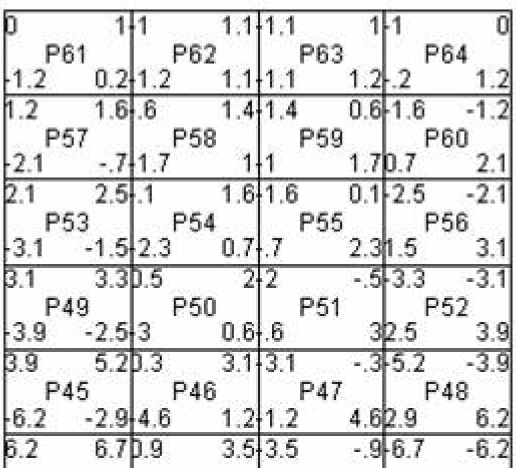
Story	Sum FX Corner Forces at Story Level (k)	Shear	Elements
4	[0.462 + 0.786 + 1.77 + 1.98] * 2 = 9.996	9.9k	P49-P52
3	[1.2 + 1.65 + 3.45 + 3.7] * 2 = 20.000	20k	P33-P36
2	[1.9 + 2.5 + 5.14 + 5.46] * 2 = 30.000	30k	P17-P20
1	[11.76 – 0.32 + 6.09 + 2.47] * 2 = 40.000	40k	P1-P4



Level 4 Global FX Corner Forces

For the graphical display of the corner forces, there are 4 corner forces shown for each plate. This is similar to a beam element which has 2 member end forces.

To get the story shear at any line, just sum up all the FX corner forces along the line.



Level 4 Global FY Corner Forces

To get the story moment at any line, just sum the moments obtained by multiplying the Fy corner forces along a line, times the moment arm such as the distance of each Fy force to the center of the wall.

Story Shears - Summation Tool

A quicker way to calculate story forces is to use the Internal Force Summation tool to have the program automatically calculate the global forces that pass through a given elevation of the shear wall. To do this, just click on the icon that appears below the selection toolbar whenever you have a set of valid, non-envelope solution results. This tool will sum the forces for the displayed items at the desired elevation and will report them back to the user in terms of the Global X, Y and Z directions.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Story Shears**.

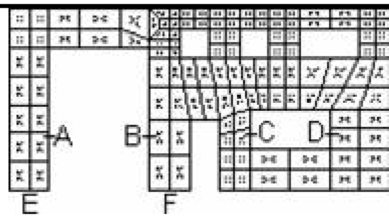
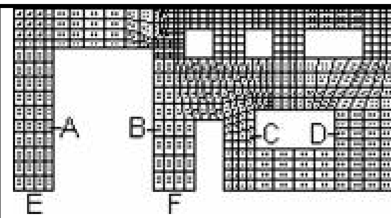
Story Moments

The story moments are calculated as the FY Corner forces times their moment arms. The forces are symmetrical so each force on one side is multiplied by twice the arm length.

Story	Sum FY Corner Forces * Moment Arm (k)	Story Moment	Elements

4	[3.92 k * 15' + (2.48 k + 3.02 k) * 7.5'] = 100.05 k'	100.1 kip-ft	P49-P52
3	[12.33 k * 15' + (7.9 k + 7.45 k) * 7.5'] = 300.075 k'	300.1 kip-ft	P33-P36
2	[24.93 k * 15' + (16.5 k + 13.64 k) * 7.5'] = 600.00 k'	600 kip-ft	P17-P20
1	[45.81 k * 15' + (25.16 k + 16.55 k) * 7.5'] = 999.975 k'	999.9 kip-ft	P1-P4

Shear Wall Penetrations

		
Horizontal Deflection at Top	0.028 in.	0.033 in.
Shear @ A-A	10.82 kips	10.53 kips
Shear @ B_B	24.2 kips	23.08 kips
Shear @ C_C	33.74 kips	33.05 kips
Shear @ D-D	45.26 kips	47.35 kips
Reactions at E	10.8 kips	10.53 kips
Reactions at F	24.2 kips	23.06 kips

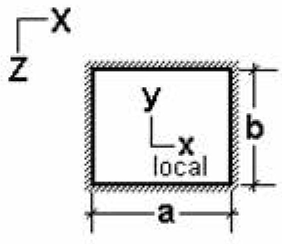
This is an example of a typical concrete shear wall with penetrations for windows and doors of various sizes. The files for the models are WALLPEN1.R3D (coarse mesh) and WALLPEN2.R3D (fine mesh). No theoretical solution results are given to compare with, however the two finite element densities are compared to observe the rate of convergence to the “true” answer. The shears at the various lines are computed by adding up the X corner forces for the element corners closest to the lines. The horizontal deflection is for the top of the wall. A very rigid link is added to the top of the wall to simulate the effect of a concrete horizontal diaphragm. This has the effect of stiffening the walls and spreading the load uniformly across the top of the wall. The load is applied as a uniform load of 3.0 kips/ft. across the top of the wall. The total width of the wall is 38 ft, so the total applied load is 114 kips. The total height of the wall is 18 ft.

The “coarse” mesh on the left is an example of the minimum finite element mesh that should be used to model this type of wall. Notice that the course mesh gives good results for the wall shears and reactions. The overall deflection of the coarse mesh is off by about 15% from the “fine” mesh. The coarse mesh tends to give too much stiffness to the slender walls around the loading door opening on the left, this can be seen in the larger reactions at points E and F as well in the horizontal deflections. The fine mesh on the right shows that the slender wall sections are more flexible than shown by the coarse mesh and thus the reactions and wall shears are reduced for the slender wall sections.

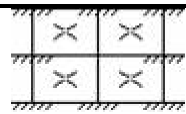
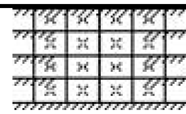
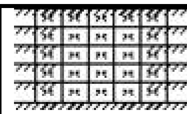
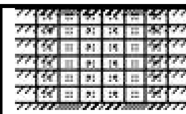
Diaphragm Modeling

Theoretical Solution for Plate with Fixed Edges

(Results from Roark’s Formulas for Stress and Strain, 5th Ed., pg. 392)

	Plate Properties a = 21 ft. b = 15 ft. thickness = 8 in. E = 3122ksi q = 60 psi α = 0.0226 β1= 0.4356 β2= 0.2094	Deflection at center (y): $= \alpha q b^4 / E t^3 = 0.891$ in. $\sigma_{Center} = \beta_2 q b^2 / t^2 = 6361$ psi $M_{Center} = \sigma_{Center}(t^2/6)(1kip/1000\#)$ = 67.8 k-ft/ft $\sigma_{Max} = \beta_1 q b^2 / t^2 = 13{,}231$ psi M Max $= \sigma_{Max}(t^2/6)(1kip/1000\#)$ = 141.1 k-ft/ft

Stiffness and Stress as a Function of Mesh Fineness

				
Element Mesh	2x2	4x4	5x5	6x6
Deflection @ Center Error (%)	0.032 96%	0.895 1%	0.774 13%	0.911 2.2%
My @ Center (K-ft / ft) Error (%)	80.45 19%	75.8 12%	70.3 4%	73.4 8%
Global MX Reaction @ Center of Long Side to Obtain Max. Local My	Joint 6 Reaction 844.8 K-ft	Joint 15 Reaction 717.2K-ft	Joint 18 Reaction 545.3K-ft	Joint 4 Reaction 489.4K-ft
Local My @ Center of Long Side (Mx Reaction divided by tributary length) Error (%)	80.5 k-ft / ft 43%	136.6 k-ft / ft 3%	129.8 k-ft / ft 8%	139.8 k-ft / ft 1%

The condition being modeled is a flat plate with fixed edges and a uniform load over the entire surface. The RISA-3D2DFloor files that were used to obtain these results are included as "2X2FIXED.R3D", "4X4FIXED.R3D", "5X5FIXED.R3D", and "6X6FIXED.R3D".

The plate moments at the center of the long side were calculated by dividing the global Mx reaction at the center of the long side by the tributary length. See the summary results below. (Note that the 5x5 mesh produces good results even though the Mx reaction is not at the exact center of the long side.) Remember that plates with perfectly fixed end conditions have their maximum moments at the center edge of their longest side.

Mesh	Mx Global Reaction	Tributary Length	Equation	My Local Moment
2x2	844.8 k-ft	21ft / 2 = 10.5ft	844.8 / 10.5	80.5 k-ft / ft

4x4	717.2 k-ft	21ft / 4 = 5.25ft	717.2 / 5.25	136.6 k-ft / ft
5x5	545.3 k-ft	21ft / 5 = 4.25ft	545.3 / 4.25	129.8 k-ft / ft
6x6	489.4 k-ft	21ft / 6 = 3.5 ft	489.4 / 3.5	139.8 k-ft / ft

The edge moments only need to be considered as the maximum moments when a plate is fixed at it's edges, since the maximum moments will often occur in the center of the plate for most other support conditions. (The edge moments will still need to be considered for moment reversal if the plate is continuous across the supports).

For the situation of continuous slabs supported by beams between columns, the maximum moment will often occur at mid span and not at the edges. Thus a 3x3 or 5x5 mesh should be used to obtain correct moments. Even numbered meshes (e.g. 6x6, 4x4, or 2x2) should be used to obtain the best deflection information and odd numbered meshes (e.g. 3x3 or 5x5) should be used to obtain the best bending moment results. The 6x6 mesh could be used to obtain good moments and deflection results.

The internal My bending moments are obtained using the Global Corner Forces and the Internal Force Summation tool for the 2x2, 4x4, and 6x6 meshes. The total global MX moment on the side of an element was computed and then divided by the length of the element. (Global MX moments are parallel to local My moments in this model) The internal My @ Center are found using the Plate Forces for the odd plate example (5x5) because there is no node at the center.

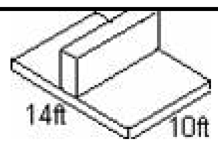
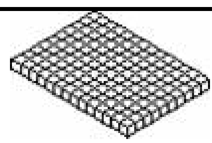
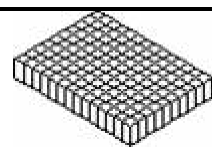
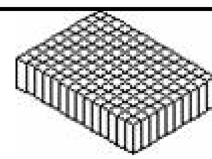
It should be noted that the deflection obtained from the 4x4 and 6x6 meshes is larger than that predicted by the Roark equations because the RISA-3D2DFloor finite element accounts for transverse shear deformation while the Roark equations ignore shear deformation.

These results are for a uniform load. If the loading is more localized, or approaches a point load, a much finer mesh in the vicinity of the load will be needed to model the loading itself and to get accurate results. Also note that RISA-3D2DFloor's finite element (like most commercial finite elements) is based on small strain theory. This means that the in-plane diaphragm stresses are not affected when the transverse deflections become large. According to Roark, (pgs. 405-409), this additional stress becomes significant when the transverse deflection is larger than half the plate thickness.

Spread Footing Modeling

Stress Accuracy as a Function of Distortion

Shown below are the analysis results for an axial wall load on a spread footing, which is then on soil springs. The files used for this parametric study are FLXFTNG.R3D for the "flexible" footing results and RGDFTNG.R3D for the rigid footing results. Note that the theoretical values shown are based on the assumption of an infinitely rigid footing.

			
Footing Thickness	12 in.	24 in.	36 in.
Element Aspect Ratio Thickness:Length	1:1	2:1	3:1
One way shear at "d" from the wall face, Flexible Footing	109.2 kips	92.4 kips	70 kips
Elements Used for Shear	9-135 (by 14) 10-136 (by 14)	10-136 (by 14) 11-137 (by 14)	11-137 (by 14) 12-138 (by 14)
Moment at Wall Face, Flexible Footing	368 k-ft	415.2 k-ft	421.2 k-ft

Elements Used for Moment	9-135 (by 14)	9-135 (by 14)	9-135 (by 14)
Theoretical 1-Way Shear at “d” from the Wall Face	117.9 kips	94.3 kips	70.7 kips
Theoretical Moment at Wall Face	424.3 k-ft	424.3 k-ft	424.3 k-ft

As can be seen in the table, the results are converging to the theoretical solution for a infinitely rigid footing as the footing thickness increases and begins to become “very rigid” when compared to the soil spring stiffness.

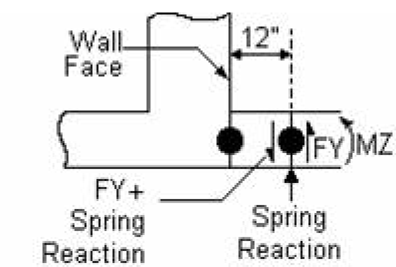
To obtain the One Way Shear values at a distance “d” from the wall face, simply sum the FY global corner force values for the elements on both sides of the appropriate row, and then take the average of these two values. You need to average the two values in this case, because the corner force results are on either side of a soil spring. For example, to obtain the one way shear for the 24” thick footing, sum all the FY corner forces for elements 9 to 135 by 14 (9, 23, 37, …) and then 10 to 136 by 14 (10, 24, 38, …). Then take the average of those two sums. If you don't have soil springs at the corner force locations, you don't have to average the two values. (The sums on each side in this case will be equal). The easiest way to add up the corner forces is to simply sum them from the graphics display. This way you don't have to track the element numbers.

To obtain the Moment values at the face of the wall, just add up the MZ global corner forces for the elements along the wall face. For this example these would be elements 9 to 135 by 14. Again, the easiest way to add up the corner forces is to sum them from the graphics display so you don't have to worry about element numbers.

The finite element corner forces work best when the footing is aligned with the global axes. That way the global corner forces line up with the desired footing shears and moments.

Since the theoretical values for the shear and moment are based on the assumption of an infinitely rigid footing, it is instructive to look at a finite element model where we use an artificially high value of “E” (Elastic Modulus) to approximate an infinitely rigid foundation.

Rigid Footing Results



Footing Thickness	12 in.
1-Way Shear at “d” from the Wall Face, Rigid Footing	117.9 kips
Elements Used for Shear	9-135 (by 14) 10-136 (by 14)
Moment at Wall Face, Rigid Footing	424.3 k-ft
Elements Used for Moment	9-135 (by 14)

These results agree exactly with the theoretical values.

Computing Soil Spring Stiffness

Obtain the subgrade modulus for a 1' by 1' or .3m. x .3m sample plate. A typical value for medium dense dry sand would be say, k1 = 500 kcf. This value must first be modified to account for our actual footing size (10 ft by 14 ft). For this example we will use equations from Principles of Foundation Engineering, 3rd edition, by Braja Das, pgs. 263 - 264. We will assume a 1ft x 1ft sample plate.

$$K_{10x10} = k_1\left(\frac{B+1}{2B}\right)^2 = 500\left(\frac{10+1}{2*10}\right)^2 = 151kcf$$

$$K_{10x14} = K_{10x10} * \left(\frac{1+0.5*B/L}{1.5}\right) = 151*\left(\frac{1+0.5*10/14}{1.5}\right) = 136.6\,kcf = 0.07906\ kip/in^3$$

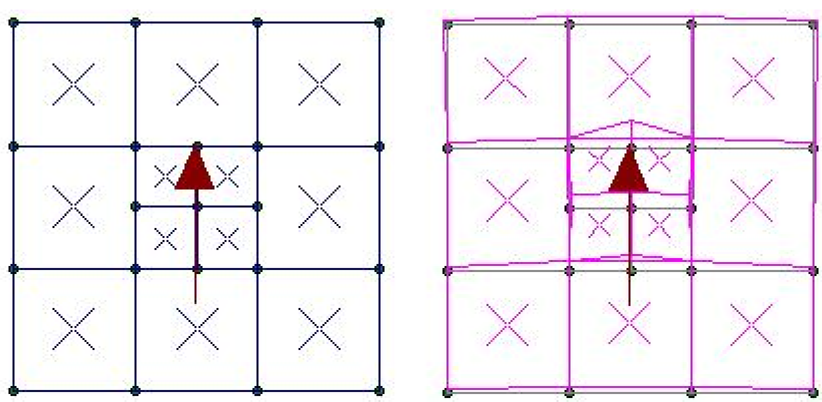
We can now calculate the spring stiffnesses for all the joints in the model based on tributary area. The program is capable of automatically generating these soil springs based on the tributary are of each node in the plate element mesh. Refer to Generating Soil Springs of the Boundary Conditions s section for more information.

Tributary Area	K(10x14) k/in^3	K(spring) k/in.	Example Joint
1 sq.ft = 144 sq.in.	0.07906	11.38	13
0.5 sq.ft = 72 sq.in.	0.07906	5.69	34
0.25 sq.ft = 36 sq.in.	0.07906	2.85	1

Although a little out of date, the ACI publication 336.2R-88, "Suggested Analysis and Design Procedures for Combined Footing and Mats", is another good reference for the modeling of mat foundations.

Plate Connectivity Problems

Shown below is a common modeling problem with plate elements. Since plates only have connectivity at their corner nodes, the applied load at middle joint connects to the plates below the joint, but not to the one above it. Because of this lack of connectivity, you see the joint "pushing through" the plate edge above in the plotted deflected shape.



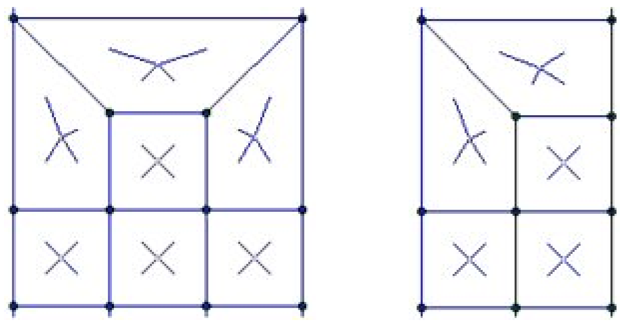
The proper way to hand this type of mesh is with one of the mesh transitions described in the following section.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Plate Connectivity**.

Mesh Transition Examples

Coarse Mesh to Fine Mesh

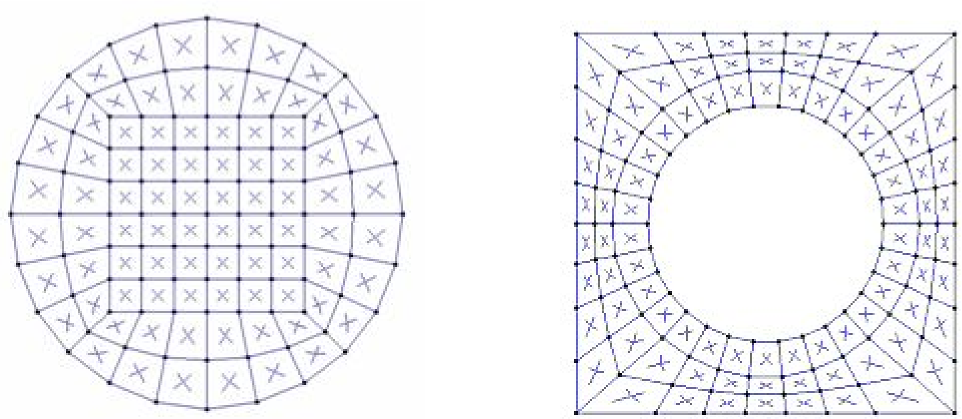
Shown below are two methods for transitioning from an area with a fine mesh to an area with a larger / coarser mesh.



Three to One Two to One

Rectangular Mesh to Radial Mesh

Shown below are two methods for transitioning from an area with a rectangular mesh into an area with a radial mesh.



Square to Round Round to Square

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Plate Mesh**.

Printing

You may print graphics as they appear on the screen or print all or part of the tabulated results. If a spreadsheet is currently active and you click the print button the following options are presented. If a model view is currently active, and you select the print button, Graphic Print Options are presented.

If you have lines selected in the spreadsheet then you may print only those lines by selecting the first option. If you wish to print the entire spreadsheet you may choose the second option. The last option allows you to print multiple spreadsheets in a custom report that is discussed in the next section. You may combine the last two options with the Exclude feature to hide some data and print only the data you want.

To Print a Report

- While in a spreadsheet click on the **Print** button and choose to print the current spreadsheet, a selected portion of the spreadsheet, or multiple sections by printing a report. If you are in a model view the Graphic Print Options are opened so click 'Print a Report Instead'.

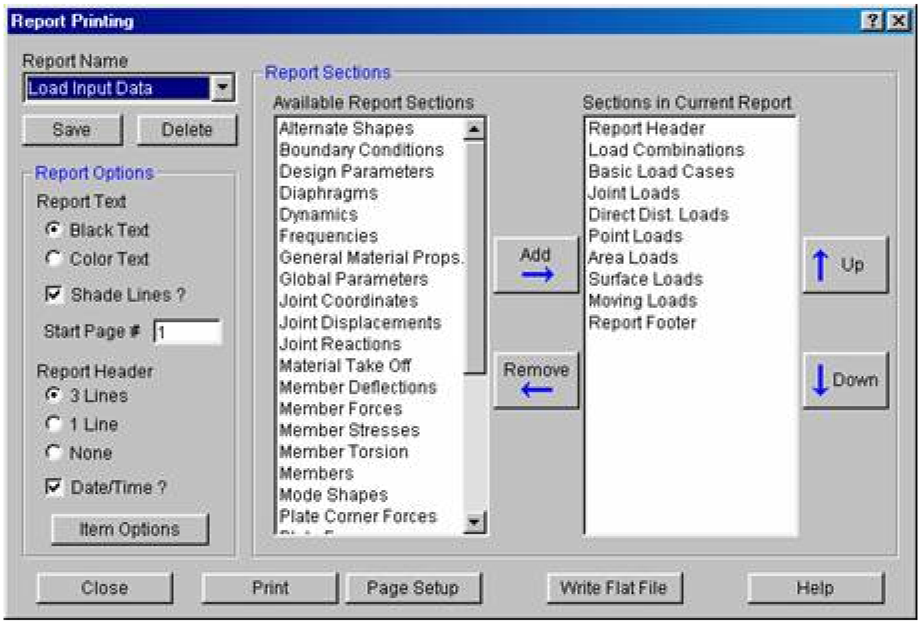
To Print Graphics

- While in a model view click on the **Print** button and choose from the options.

Note

- If you are not in a model view the Data Printing settings are opened instead. Close this and click on the model view that you want to print before clicking the **Print** button.

Printing Reports



The **Report Printing Dialog** options help you build your reports. There are standard reports for you to choose from and you may also name and save any report format you custom build. To choose a standard report simply pick it from the **Report Name** drop down list.

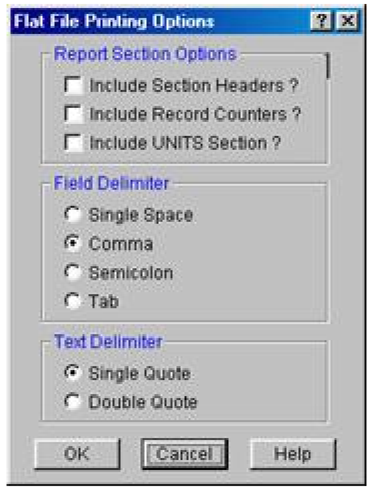
To build your own custom report you may double-click on report sections in order to move them from the list of available sections on the left, to the current report defined in the list on the right. You may use the mouse and the SHIFT or CTRL keys to pick multiple sections and then move them with the **Add** button.

Formatting options allow you to specify the text color as black or blue, shade every other line to enhance readability and select the starting page number. The number shown will be the next page number in the current sequence, but you can override this for occasions where you need to insert your calculation pages into an existing report and you need the page numbering to match. All reports have a footer with version information, the file name and path, and the page number. The single line header option will include the Model Title specified in the **Global Parameters** along with the date. The triple line header adds company, designer and job number to the header as well as a place to initial any checking. There is also an option to turn off the Date/Time stamp so that this will not appear on your printed reports.

The **Item Options** allow you to select member and plate related options. You may specify that you want the member results to be listed for each member section (specified in the **Global Parameters**) or just for the member ends, which can be useful for connection design. For plates you may indicate which surface forces you desire.

Printing to a File

A flat file is a file without column headings, print formatting, or graphical elements and is useful for importing and parsing into spreadsheets, database tables, or as post processor input data.



There are several options available to make the flat file easier to parse. Note that printing and then looking at a sample output file with all the options selected will make it easier to understand what the options do.

You have an option to include the 'Section Headers' which will print a text description of each block of data. For example, the Joint Coordinates data would be preceded by a [JOINT COORDINATES] header on it's own line before the data. You may also include a 'Record Count' which is useful if you're writing looping code to read in the number of records within each data block. The number of records prints on it's own line. You can also have a special 'UNITS Section' be printed out which will give you the units used for all the values in the program.

The field delimiter options let you choose what character will be used to separate each field of data within record. For example, each coordinate value in the Joint Coordinates data would be separated by the character selected. Programs that you might be importing the file into like MS Excel or MS Access often have options to select what the field delimiter will be for records of data.

The text delimiter works like the field delimiter, except that it's used to set apart text labels. All text in the flat file will be enclosed at the beginning and the end by the selected text delimiter character. This is very useful when trying to read in label strings that contain embedded spaces. As an example, the Joint labels in the Joint Coordinates data would each be enclosed by a single or double quote.

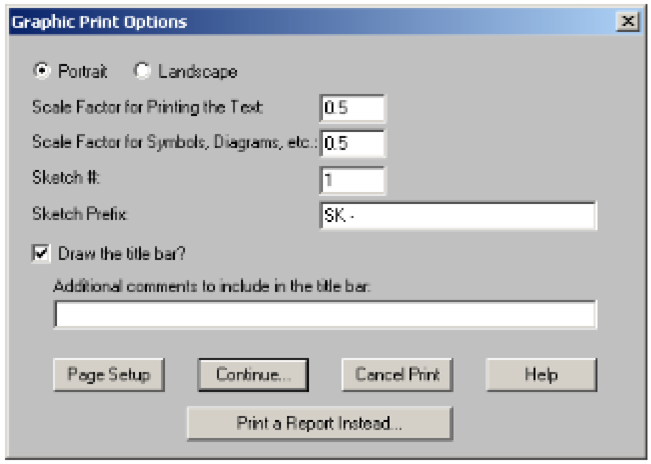
The current flat file options are saved each time the program is closed.

To Print to a Flat File

- While in a spreadsheet click on the **Print**  button, Choose **Use Report Printing Options** and click **OK**, then choose **Write Flat File.**

Graphics Printing

These options control the graphics printing. You may choose to print in **Portrait** or **Landscape** mode.



Next, two **Scale Factors** are defined. These scale factors are used to make the text and symbols displayed as part of the graphic larger or smaller. A higher scale factor makes the text or symbol bigger. Since the resolution of the printer is probably much greater than the resolution of your screen, you can probably make the text and symbols smaller (by using a scale factor of less than 1.) than they appear on screen and they will still be easily readable. This makes for a cleaner looking graphic print. As far as what scale factors you should use, the only way to be sure is to experiment a little.

You can specify a **Sketch #** and a **Sketch Prefix** to label your graphic prints as well.

You may include the **Title Bar** to display the model title, designer, company name, date and time. You can also enter a comment that will be printed in the title bar. **Page Setup** allows you to set the margins.

Once you have everything the way you want it choose **Continue**. This will bring you to the print settings that are specific to your printer. The choices are different for each printer but generally allow you to choose the printer, orientation, quality and quantity.

The bottom button **Print a Report Instead** is provided so you can get to the report printing dialog directly from a graphic view.

Results

You may work with the results of a solution by viewing and sorting data in the spreadsheets, graphically plotting them with the model or by viewing detailed member reports. You may also print the results in any of these forms. To learn about printing results, see Printing.

Upon the completion of a static solution, RISA opens the **Results Toolbar** and the **Reactions Spreadsheet**. You may specify that other results be displayed automatically as well. You may then proceed to view any results and make any changes for further analysis.

If you make any changes to the model that would void the results, such as moving joints or adding members, the results will be purged and another solution will be required before the results can be viewed again.

Each of the result types is described in it's own section:

Results Spreadsheets

- Joint Deflection Results
- Joint Reaction Results
- Joint Drift Results
- Member Force Results
- Member Stress Results
- Member Torsion Results
- Member Deflection Results
- Hot-Rolled Design Results
- Cold-Formed Design Results
- Wood Design Results
- Concrete Design Results
- Aluminum Design Results
- Concrete Wall Results
- Masonry Wall Results
- Wood Wall Results
- Plate Stress Results
- Plate Force Results
- Plate Corner Force Results
- Modal Frequency Results
- Mode Shape Results

Detail Reports

- Member Detail Reports (Hot Rolled Steel, Cold Formed Steel, & Wood)
- Concrete Beam Detail Reports
- Concrete Column Detail Reports

Saving Results

When you save a file that has been solved you may also save the results. The next time that the file is opened the saved results will be opened as well. You may use the **Preferences** on the **Tools Menu** to change the way that you are prompted to save results.

If changes are made to the model, any saved results are deleted. Saved results for models that no longer exist in the same directory are also deleted.

Results Spreadsheets

You may access the result spreadsheets by selecting them from the **Results Menu**. You may use the **Find**, **Sort**, and **Exclude** features to find the results you are interested in. For example you might sort the member stresses from high to low, bringing all of the highly stressed members to the top. You might then exclude members that do not have significant axial stresses so that they do not distract you or so that they are not printed.

Finding Results

- To go to a certain item while in a spreadsheet click the **Find** button and type in the desired member, plate or joint label.

Sorting Results

- To sort the results click on the column of results you wish to sort and then click the **Sort** button to specify preferences. You can sort based on maximum, minimum, absolute maximum or input order.

Excluding Results

There are three ways to exclude results so that you can work with the results that are important to you.

Excluding Results Before the Solution

Excluding items before the solution allows you to remove the items from the results while leaving them as part of the model. This exclusion is permanent for that solution and any exclusion changes will then require another solution. This exclusion may be applied graphically with the **Draw** or **Modify** features or within the spreadsheets by recording an "E" in the **Inactive** field of the **Members** and **Plates** spreadsheets.

A member with an "E" in the "Inactive?" field will be treated like any other member in the solution and plotting of the model, but the member will not be listed in the solution results (forces, stresses, deflections, etc.). This is useful if there are certain members whose results aren't of interest. You don't have to clutter up the results with these members and can concentrate on the members you are most interested in.

Excluding Results After the Solution

You may run the solution and then graphically select the joints, members, and plates that are of interest. By clicking the Exclude button you may then update the spreadsheets and printed reports so that they will only have results for the selected items. All of the spreadsheets and reports will be controlled by this selection and you may adjust this selection at any time.

You may also select the results of interest on each spreadsheet. While viewing spreadsheet results you may select the last line of interest and exclude the rest. Exclusions are applied independently for each results spreadsheet. Any exclusion applied to the Member Forces spreadsheet will not affect the Member Stresses spreadsheet, etc. Excluding items graphically will reset **all** of the spreadsheets to match the graphic selection. If you wish to combine these two features to fine tune the results perform the graphic selection first.

Excluding Results While in Spreadsheets

To exclude results click on the last line of results you wish to keep and then click the **Exclude After** button. You can bring the results back by clicking on the **UN-Exclude** button.

Note

- It might be best to first sort the results before excluding. As an example, let's say that you only wanted to view members with Code checks greater than or equal to 0.7. To do this you would open the Steel Code Checks spreadsheet and sort the members by Code Check magnitude. Then you would visually identify the last member with a Code Check of 0.7 and use the Exclude feature to exclude all members after that one.

Graphic Results

Most of the analysis results may be viewed graphically as well as in the spreadsheets. For the joints you may plot the reactions. For the members you may plot force diagrams as well as color-code the plotted members by code check or stress levels. Plate stress contours and corner forces may also be viewed graphically. Deflected shapes and mode shapes may be viewed and animated. See Plot Options for more information.

To Plot Results Graphically

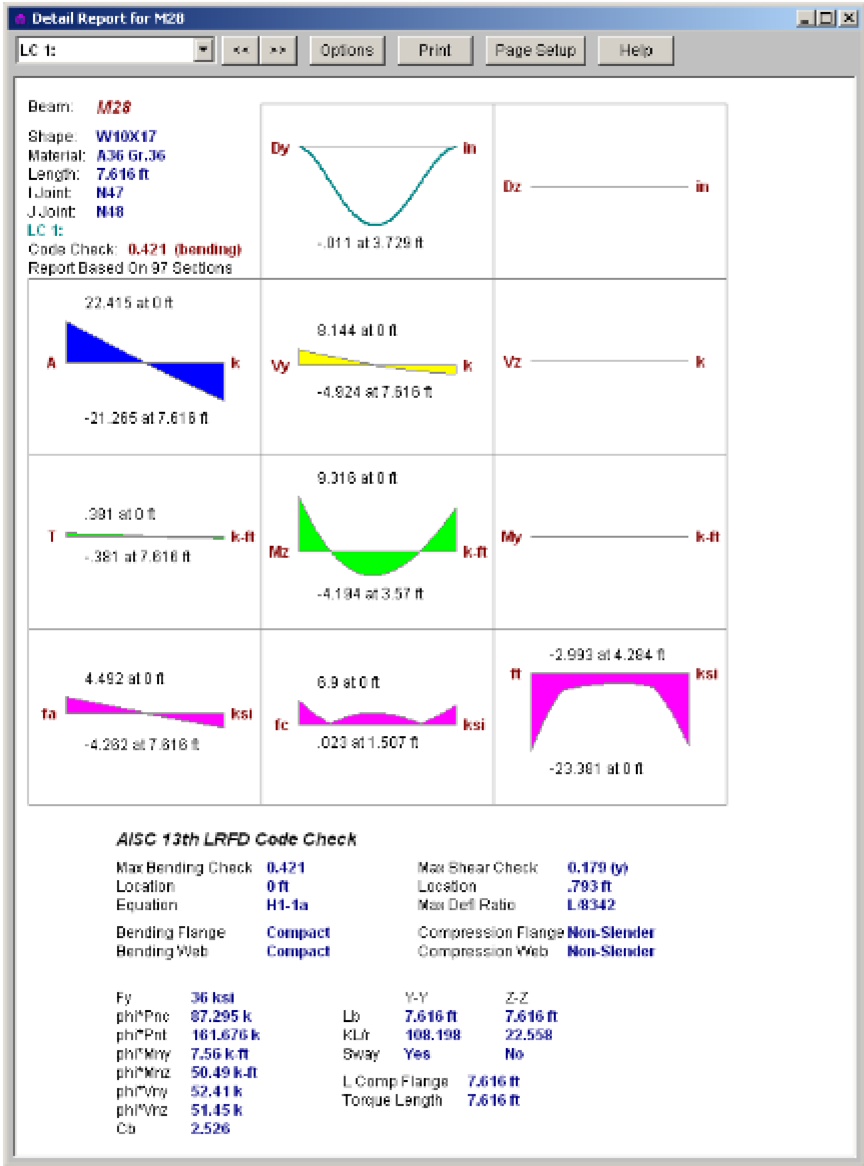
1. If you do not already have the model view open click the **New View** button on the **RISA Toolbar.**
2. Click the **Plot Options** button on the **Window Toolbar** and select the **Loads** tab.
3. Select the options you would like to view and click **Apply**.

Clearing Results

You will be provided with a warning if changes are made to the model that might invalidate the current results. Should you decide to proceed, the results will automatically be cleared and you will have to re-solve the model to get results once you are finished making changes. This warning may be disabled in the **Preferences** on the **Tools Menu**. To manually clear results, click the **Clear Results** button.

Member Detail Report

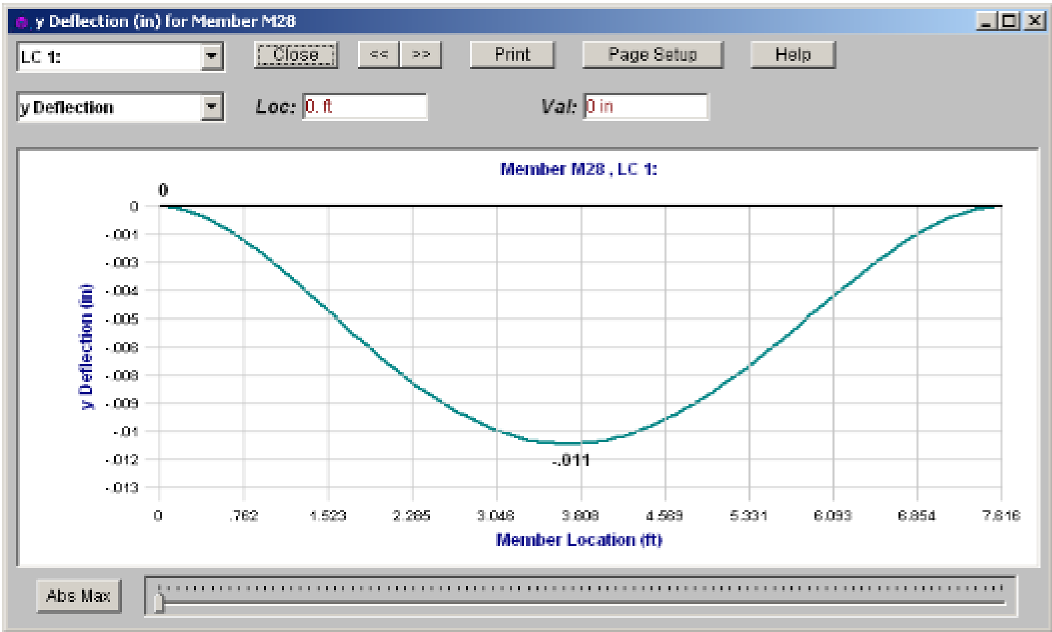
The detail report allows you to see the overall force/stress/deflection state for any particular member. This report shows diagrams for all these quantities and also lists detailed information on the code checks for Hot Rolled or Cold Formed Steel Design or Timber Design. If there is a (y) or (z) value next to the code check, it defines whether the controlling case is in the local y or z axes respectively. If there is an (s) this tells you that the maximum check occurred at some other axes where a round shape is used and the cross-sectional properties for the member are the same in all directions. Once a report is open, any of the diagrams may be expanded by clicking on them.



To View a Detail Report

- To view a member detail report after running a solution, click on the Detail button on the **Selection** Toolbar and then click on that member. If you do not already have the model view open click the **New View** button on the **RISA Toolbar.**
- You may also access the detail report while viewing a member results spreadsheet such as **Member Stresses** by clicking the Detail Report for Current Member button on the **Window** toolbar.
- You may also use the **Results** ‣ **Members** ‣ **Detail Report** menu option to open the report for any member.

This detail report is available for any member following the solution of any individual or batch load combination. This report is **NOT** available if the solution performed was an envelope solution.



The detail report diagrams the force, stress, and deflected state of the member. Click on the diagrams to open an enlarged diagram view with calculated values at specific member distances.

The diagrams provided are:

Plot Designation	Plotted Value
A	Axial Force
T	Torsional Moment
Vy	Shear Force Parallel to y-y Axis
Vz	Shear Force Parallel to z-z Axis
My	Bending Moment About y-y Axis
Mz	Bending Moment About z-z Axis
Dy	Deflection in Local y Direction
Dz	Deflection in Local z Direction
fa	Axial Stress
fc	Bending Compressive Stress
ft	Bending Tension Stress

The "fa" stresses are the stresses resulting from the axial force A. The "fc" and "ft" stresses incorporate both bending moment and torsional normal warping stresses.

The diagrams are scaled in groups to give a good representation of relative values. For example, the force diagrams (A, Vy and Vz) are scaled such that the force of maximum magnitude fills the diagram space. The other diagrams are then plotted using that same scale. The moments, deflections and stresses are similarly scaled together.

When an output file is saved, the program will discard much of the unneeded force, moment and deflection data used to create the detail report plots. When this saved file is later retrieved, these plots will appear more coarse and inexact. However, the maximums, minimums and controlling code checks are always maintained regardless of how coarse the plots appear.

Concrete Member Detail Reports

Concrete member detail reports are similar in function to detail reports for other materials, but they are different in the type and amount of information they convey. One of the largest differences is that the force diagrams are always **envelope** force diagrams, because the majority of the concrete design results are based on the envelope forces.

The detail reports for concrete Beams and Columns are discussed in greater detail in the Concrete - Design Results section.

RISAFoot Integration - Design

Design Procedure for Integrating RISAFoot and RISA-3D

The Criteria and Materials information is entered on the Footing tab of the Global Parameters.

1. The Footing spreadsheet has tabs for entering the basic geometry and soil parameters for each footing type.
2. The Boundary Conditions allows you to specify which footings will apply to which column supports.
3. The Design Tab of the Load Combinations spreadsheet let's you select which Load Combinations are going to be used for your footing design.

Notes:

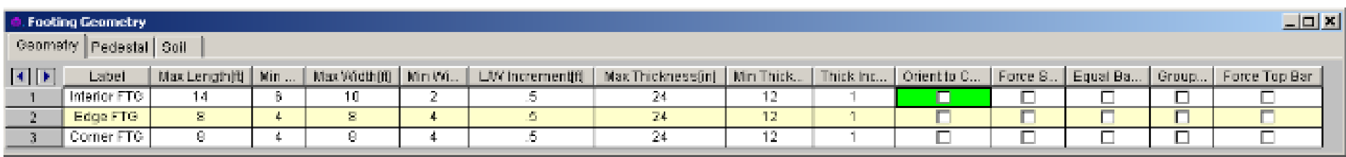
- Make sure that your batch solution has at least one **Service** load combination that also has the **Footing** box checked. These service combinations are the ones that will be use for the soil bearing and stability checks.
- Make sure that your batch solution has a least one **non-Service** load combination that has the **Footing** box checked. These load combinations will be used for the concrete code checks and rebar design of the footing and pedestal.
- You must make sure that your load combinations are built using the load CATEGORIES (DL, LL, WL… et cetera) rather than the BLC Number. This is because Foot requires information about DL factors so that it can apply factors to the soil overburden and footing self weight.

5. Solve a **Batch** solution to get all your reaction results.
6. Solve again, this time selecting the option for **Design Footings** and your footings will be designed / analyzed. Results will be presented in the Footing Results spreadsheet.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **RISAFoot**.

Footing Geometry

The Geometry tab contains the basic information that will control the size and thickness of the footing.


Footing Geometry

Geometry | Pedestal | Soil

	Label	Max Length[ft]	Min ...	Max Width[ft]	Min Wi...	L/W Increment[ft]	Max Thickness[in]	Min Thick...	Thick Inc...	Orient to C...	Force S...	Equal Ba...	Group...	Force Top Bar
1	Interior FTG	14	6	10	2	.5	24	12	1	☐	☐	☐	☐	☐
2	Edge FTG	8	4	8	4	.5	24	12	1	☐	☐	☐	☐	☐
3	Corner FTG	8	4	8	4	.5	24	12	1	☐	☐	☐	☐	☐

These parameters define the minimum and maximum length, width and thickness for the footing. When RISAFoot sizes the footing, it optimizes for the smallest footing that fits within the parameters and does not exceed the allowable soil bearing pressure. RISAFoot favors footings with the lowest length/width ratio (as close to a square as possible).

If you are investigating an existing footing size that you don't want RISAFoot to change, you may either specify the same maximum and minimum length, width and/or thickness OR specify the maximum values and set the design increments to zero.

Orient to Column

This will orient the footing to the local axes of the column that is defined by the similar joint. This will only work if the Member Type for the column is set to a Column.

Force Footing to be Square

Checking the box in the **Force Square** column will restrict the design to square footings only.

Force Bars to be Equally Spaced

Checking the **Equal Bar Spacing** option allows you to indicate that you do not want to specify banded reinforcement. If you check this box, RISAFoot will determine the tightest spacing required per ACI 318-11 section 15.4.4 and then use that spacing throughout the footing. The results browser will still state (banded) so that you know why this tight spacing occurred.

Enveloping Multiple Footing Designs

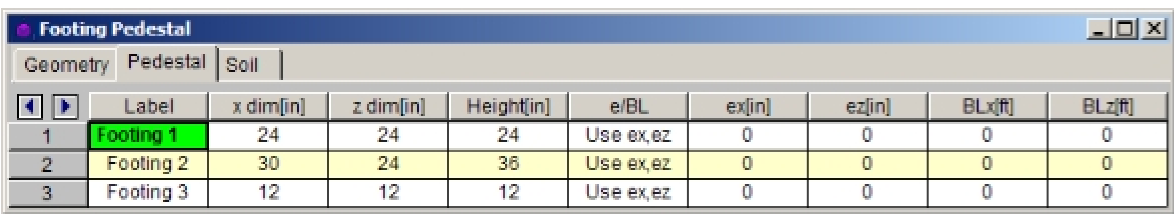
Checking the box in the **Group Design** column will cause all footings assigned to the current footing definition to use the same final size. The largest required size within the group of footings will be used for all footings within the group. The maximum required reinforcement for the footings and pedestals will also be enveloped.

Force Top Bars

The **Force Top Bars** option allows the user to to indicate that top bars are required for this footing even for cases where the footing does not experience significant uplift or negative moment. One reason for checking this box would be to spread out the Temperature / Shrinkage reinforcement betwee the top and bottom layers of steel.

Footing Pedestal

The Pedestal tab contains the basic information that will control the width and height of the pedestal. It will also control the location of the pedestal within of the footing if you have an eccentric footing.



	Label	x dim[in]	z dim[in]	Height[in]	e/BL	ex[in]	ez[in]	BLx[ft]	BLz[ft]
1	Footing 1	24	24	24	Use ex,ez	0	0	0	0
2	Footing 2	30	24	36	Use ex,ez	0	0	0	0
3	Footing 3	12	12	12	Use ex,ez	0	0	0	0

The pedestal dimensions are entered in these fields. The x dim (px) and z dim (px) values are the pedestal dimensions parallel to the local x and z axes, respectively. The height is the distance from the upper surface of the footing to the top of the pedestal. The pedestal location may be controlled in the Boundaries and Eccentricities section discussed below.

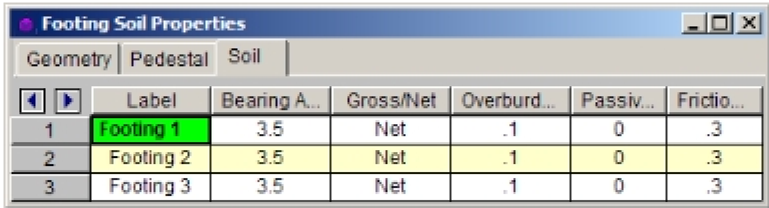
Boundaries and Eccentricities

When specifying an eccentric pedestal the **eX** and **eZ** values are the offsets from the center of the footing in the local X and Z directions, respectively. These values may be either positive or negative.

You may use the **BLx** or **BLz** entries to set boundary lines for the footing in one or two directions or you may specify the pedestal eccentricity from the centroid of the footing. If you set boundary lines, RISAFoot will keep the sides of the footing from violating the boundary. These values may be either positive or negative.

Soil Properties

The Soil Properties tab contains the basic information related to design for soil bearing.



	Label	Bearing A...	Gross/Net	Overburd...	Passiv...	Frictio...
1	Footing 1	3.5	Net	.1	0	.3
2	Footing 2	3.5	Net	.1	0	.3
3	Footing 3	3.5	Net	.1	0	.3

Soil Bearing

Allowable Soil Bearing values are typically supplied by the Geotechnical Engineer as either *Gross* or *Net* allowable values. The soil bearing check uses the same load to check soil bearing (Footing Self Weight + Overburden + Applied Load) regardless of whether Gross or Net is specified.

When a Gross soil pressure is specified the load is compared directly against the Bearing Allowable (user-entered).

When a Net soil pressure is specified the load is compared against a modified soil capacity as shown below:

Soil Capacity = Bearing Allowable (user-entered) + Footing Self Weight + Overburden (user-entered)

If the sum of Footing Self Weight and Overburden do *not* equal the soil weight which was present above the footing/soil interface strata prior to excavation then the modified soil capacity provided by the program is incorrect. In this circumstance the Net Allowable Soil Bearing value should be specified as a Gross value in the program, and the magnitude of Bearing Allowable should be increased by that pre-excavation soil weight.

For transient loads (such as wind or seismic) the allowable pressure may be increased, typically by 1/3. RISAFoot allows the input of an Allowable Bearing Increase Factor along with the loads. The factor is applied to the allowable soil bearing for those combinations that include transient loads. See Combinations for more information.

Overburden

Overburden is the weight of soil or other dead loads (e.g. pavement) which exist above the footing in its final (constructed) condition. RISAFoot applies the overburden pressure uniformly on the footing, except for at the pedestal. The overburden is automatically included in the DL category for service load combinations. Typically it is not included in the design combinations.

Passive Resistance of Soil

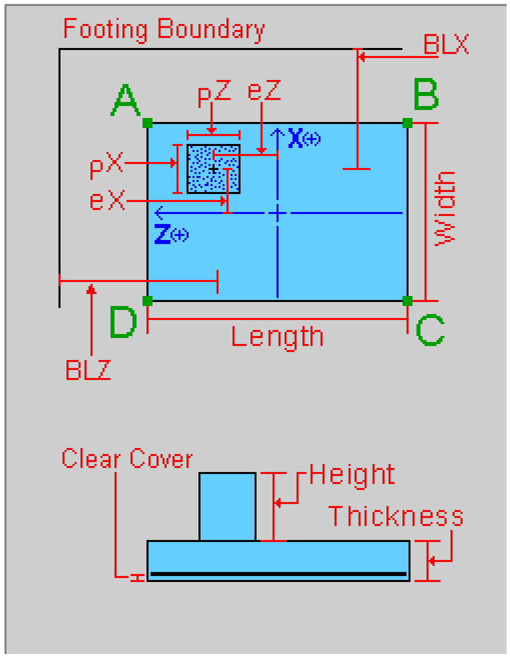
For additional sliding resistance you may enter the passive resistance of the soil as a lump sum value. RISAFoot applies this value in both directions.

Coefficient of Friction

RISAFoot multiplies this coefficient of friction by the total vertical forces to determine the soil sliding resistance.

Local Axes

The “A,B,C,D” notation provides a local reference system for the footing. The local z axis is defined as positive from B towards A, and the local x axis is defined as positive from D towards A. The origin for the local axes is the geometric center of the footing.



It is important to realize that the local footing axes do NOT necessarily correspond to the Global axis of the RISA model or the local axis of the column member.

Limitations

Non-Linear Effects

The Joint Reaction results can differ from the design results used for the OTM safety factor calculation. The reason for this is that the OTM safety factor needs to determine whether each component (P, V, or M) of each load creates a stabilizing or de-stabilizing moment.

RISAFoot Integration - Footing Results

Upon completion of the analysis and footing design the results are presented in results spreadsheets and a detail report for each footing. The detail report may be may customize and printed. You may choose which sections you wish to view or print by clicking the Report Options button. The results sections are shown below.

You can control what information is included in the detail report by clicking the Report Options button at the top of the detail report. Available report sections are listed for you to check those that you wish to include.

Solution Methodology

It is important to note that if the location of the load resultant is not within the foot print of the footing then the footing is unstable. In this instance the user is warned of the instability which must be corrected by increasing the footing size or overturning resisting force.

The method RISAFoot utilizes to obtain the soil pressure distribution provides an exact solution. RISAFoot assumes the footing behaves as a rigid body and satisfies equilibrium equations. The contribution of the soil to the above equations is determined through direct integration.

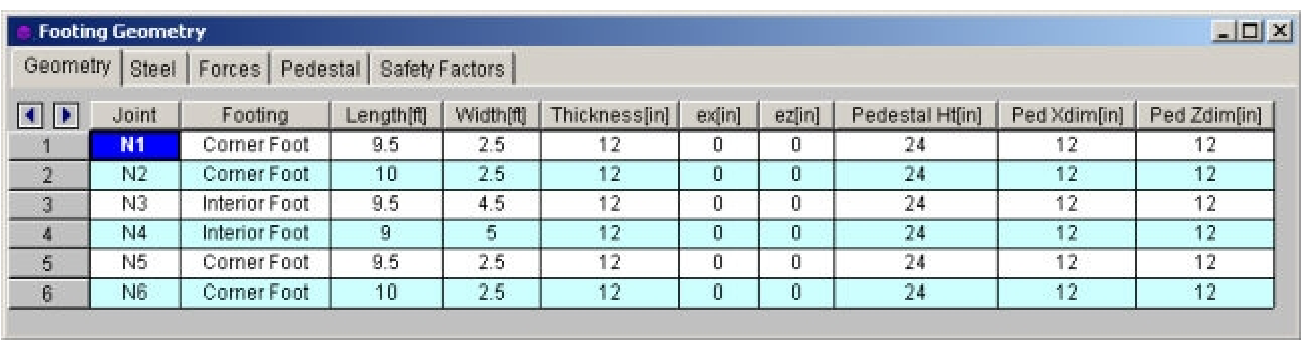
The calculation of the soil bearing profile for a two way (biaxial) load state is the most complex calculation RISAFoot makes. The soil is considered to resist compression only. If uplift is present, RISAFoot allows that part of the soil to “let go”, so no soil tension is ever part of the soil bearing solution.

Soil bearing controls the plan dimensions of the footing; i.e. the footing is sized such that the actual maximum soil bearing pressure does not exceed the user defined allowable soil bearing. The calculations use the service load combinations only. When and allowable bearing increase factor is specified, the allowable soil bearing is scaled up by the factor (ABIF), typically a 1/3 increase.

Once the soil stress distribution has been solved, the load on the footing is known at every point. Direct integration on the contributing portion of the soil stress provides the shears and moments at the lines of interest. See the Results section for more information on the calculations that RISAFoot performs.

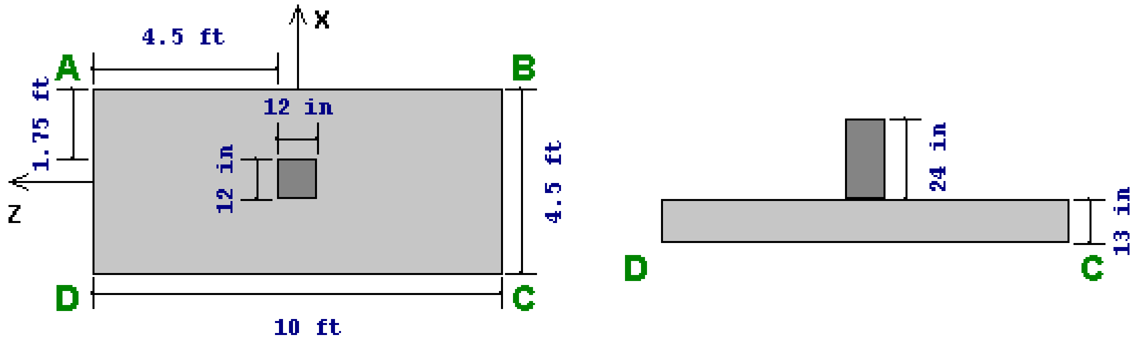
Sketch and Details

RISAFoot will optimize the dimensions of the footing for optimum concrete volume. The final footing geometry is presented in tabular form within Footing Results spreadsheet and graphically in the Footing detail report. The orientation of the footing is based on the footing's local x and z axes, respectively.



	Joint	Footing	Length[ft]	Width[ft]	Thickness[in]	ex[in]	ez[in]	Pedestal Ht[in]	Ped Xdim[in]	Ped Zdim[in]
1	N1	Corner Foot	9.5	2.5	12	0	0	24	12	12
2	N2	Corner Foot	10	2.5	12	0	0	24	12	12
3	N3	Interior Foot	9.5	4.5	12	0	0	24	12	12
4	N4	Interior Foot	9	5	12	0	0	24	12	12
5	N5	Corner Foot	9.5	2.5	12	0	0	24	12	12
6	N6	Corner Foot	10	2.5	12	0	0	24	12	12

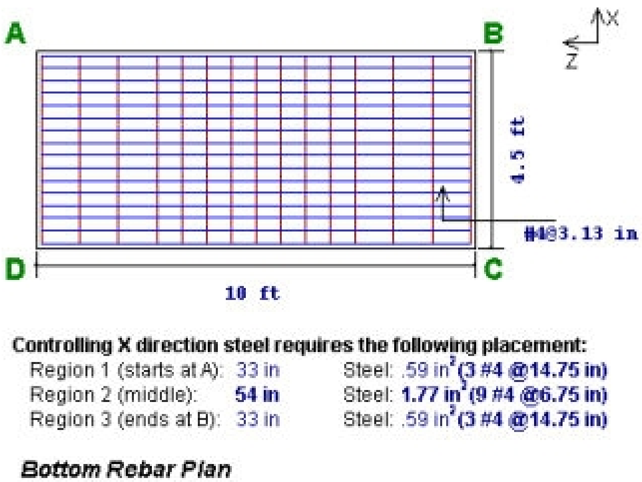
Footing Sketch



The detail report presents a sketch of the footing with dimensions (using the A,B,C,D reference system discussed in the geometry section).

Footing Details

Footing details, shown below, may be included in the report. The details may also be exported to a DXF CAD file using the Export option on the File menu.

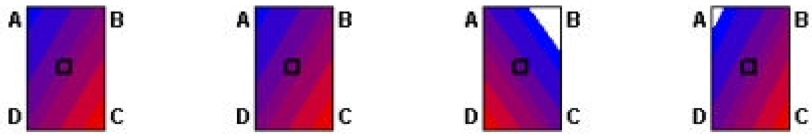


You may export the footing details to a DXF file for use in your CAD drawings.

Soil Bearing Results

RISAFoot displays the Soil Bearing check in a spreadsheet format on the Code Checks tab of the Footing results spreadsheet. The RISAFoot detail report presents the soil bearing results for all service combinations. A soil bearing contour plot for each combination can also be displayed.

Description	Categories and Factors	Gross Allow.(ksf)	Max Bearing (ksf)	Max/Allowable Ratio
ASCE 2 (b)	1DL+1LL+1SL	2.2483	2.2197 (C)	.9873
ASCE 2 (c)	1DL+1LL+1RL	2.2483	2.2197 (C)	.9873
ASCE 4 (a)	.6DL+1.5WL	2.2483	1.2717 (D)	.5656
ASCE 4 (b)	.6DL-1WL	2.2483	1.6084 (C)	.7154



1DL+1LL+1SL	1DL+1LL+1RL	.6DL+1.5WL	.6DL-1WL
QA: .5698 ksf	QA: .5698 ksf	QA: .4224 ksf	QA: 0 ksf
QB: 1.4164 ksf	QB: 1.4164 ksf	QB: 0 ksf	QB: .7986 ksf
QC: 2.2197 ksf	QC: 2.2197 ksf	QC: .5171 ksf	QC: 1.6084 ksf
QD: 1.373 ksf	QD: 1.373 ksf	QD: 1.2717 ksf	QD: .6296 ksf
NAZ: 188.7615 in	NAZ: 188.7615 in	NAZ: 121.3328 in	NAZ: 118.3107 in
NAX: 315.0307 in	NAX: 315.0307 in	NAX: 170.6975 in	NAX: 226.4333 in

RISAFoot displays color contoured images of the soil bearing profile along with the bearing pressure magnitudes at each footing corner (QA through QD). Also provided is the location of the neutral axis (the line between compressive stress and no stress due to uplift). The location is given by NAX and NAZ values. These are the distances from the maximum stress corner to the neutral axis in the X and Z directions respectively.

Footing Flexure Design

The flexural design results include the maximum moment and required steel for each direction for each design load combination. The controlling required steel values are highlighted. This information is reported in the Forces and Steel tabs of the Footing Results spreadsheet and also from within the Footing Detail report. The direction of the xx and zz moments is based on the orientation of the footing and the footing's local x and z axes, respectively.

Minimum flexural steel is listed here (along with temperature and shrinkage minimum) even though temperature and shrinkage or 4/3 As_required will normally control the design of the footing steel.

Footing Flexure Design (Bottom Bars)

As-min x-dir (Top Flexure): **22.945 in^2**
As-min z-dir (Top Flexure): **22.945 in^2**
As-min x-dir (Bot Flexure): **22.945 in^2**
As-min z-dir (Bot Flexure): **22.945 in^2**

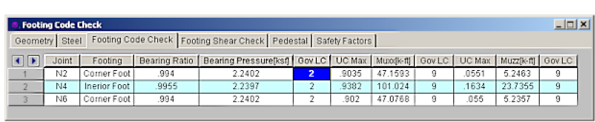
As-min x-dir (T & S): **13.4784 in^2**
As-min z-dir (T & S): **13.4784 in^2**

Description	Categories and Factors	Mu-xx UC Max	Mu-xx (k-ft)	z-Dir As Required (in^2)	z-Dir As Provided (in^2)	Mu-zz UC Max	Mu-zz (k-ft)	x-Dir As Required (in^2)	x-Dir As Provided (in^2)
ACI 9-1	1.4DL+1.7LL	.01346	35.27	.178	13.548	.01787	46.82	.237	13.548
ACI 9-2	1.05DL+1.275LL+1...	0	0	0	13.548	.01121	29.38	.148	13.548
ACI 9-3	.9DL+1.3WL	0	0	0	13.548	.00666	17.46	.088	13.548
ACI 9-2E	1.05DL+1.28LL+1.4..	.01441	37.77	.191	13.548	.02241	58.72	.297	13.548
ACI 9-3E	.9DL+1.43EL	.00709	18.57	.094	13.548	.0126	33.01	.167	13.548

Footing Flexure Design (Top Bars)

Description	Categories and Factors	Mu-xx (k-ft)	z Dir As (in^2)	Mu-zz (k-ft)	x Dir As (in^2)
SW+OB	1SW+1OB-(ACI 9-3,ACI 9-3)	69.4814	0	139.4879	0

Moment Capacity of Plain Concrete Section Along xx and zz= **816.1157k-ft,816.1157k-ft** Per Chapter 22 of ACI 318.


Footing Code Check

Geometry | Steel | Footing Code Check | Footing Shear Check | Pedestal | Safety Factors

	Joint	Footing	Bearing Ratio	Bearing Pressure[ksf]	Gov LC	UC Max	Muxx[k-ft]	Gov LC	UC Max	Muzz[k-ft]	Gov LC
1	N2	Corner Foot	.994	2.2402	2	.9035	47.1593	9	.0551	5.2463	9
2	N4	Inerior Foot	.9955	2.2397	2	.9382	101.024	9	.1634	23.7355	9
3	N6	Corner Foot	.994	2.2402	2	.902	47.0768	9	.055	5.2357	9

Footing Steel

Geometry | Steel | Forces | Pedestal | Safety Factors

	Joint	Footing	Bot X Steel[in^2]	Bot Z Steel[in^2]	Top X Steel[in^2]	Top Z Steel[in^2]	Ped Long	Ped Shear
1	N1	Corner Foot	2.749(banded)	1.571	0	0	8#3	#3@4 in
2	N2	Corner Foot	2.749(banded)	1.767	0	0	8#3	#3@4 in
3	N3	Interior Foot	2.945(banded)	3.142	0	0	8#3	#3@4 in
4	N4	Interior Foot	2.945	2.945	0	0	8#3	#3@4 in
5	N5	Corner Foot	2.749(banded)	1.571	0	0	8#3	#3@4 in
6	N6	Corner Foot	2.749(banded)	1.767	0	0	8#3	#3@4 in

To select the required flexural reinforcing steel for the footing, RISAFoot considers moments at the face of the pedestal on all four sides. The soil bearing profile from the edge of the footing to the face of the pedestal is integrated to obtain its volume and centroid, which are in turn used to calculate the moments and shears. This moment is then used to calculate the required steel using standard flexural methods.

RISAFoot reports the reinforcing steel as a total required area for each direction. ACI 318 also requires that short direction reinforcing in a rectangular footing be concentrated under the pedestal (this is referred to as banding), per the requirements of Section 15.4.4. RISAFoot will report the proper steel distribution for this circumstance.

The minimum steel required for the footing is maintained based on the ratio entered on the Footings tab of the Global Parameters dialog. ACI 318 stipulates that the minimum steel ratio for Grade 40 or 50 steel is .0020. For Grade 60 the ratio is .0018. This ratio cannot be less than .0014 for any grade of steel and RISAFoot will check this.

Top reinforcing may be required when the footing goes into partial uplift resulting in a negative moment due to soil overburden and footing self weight. When this happens, RISAFoot will first check to see if the plain (unreinforced) concrete section is sufficient to resist the negative moment per chapter 22 of ACI 318. If not, then top reinforcing will be provided. The Load Combination shown under "Categories and Factors" shows (in parentheses) the controlling Z direction, then the controlling X direction for Top bar design.

Footing Shear Check

An important part of footing design is insuring the footing can adequately resist the shearing stresses. Shear checks for each design combination are shown below. RISAFoot checks punching shear as well as one way (beam) shear in each direction. The capacity and shear check results are presented, both as ultimate shear and as demand vs. capacity ratios where the actual shear is divided by the available shear capacity, thus any ratio above 1.0 would represent failure.

This information is displayed on the Forces tab of the Footing results spreadsheet also from within the Footing Detail report.

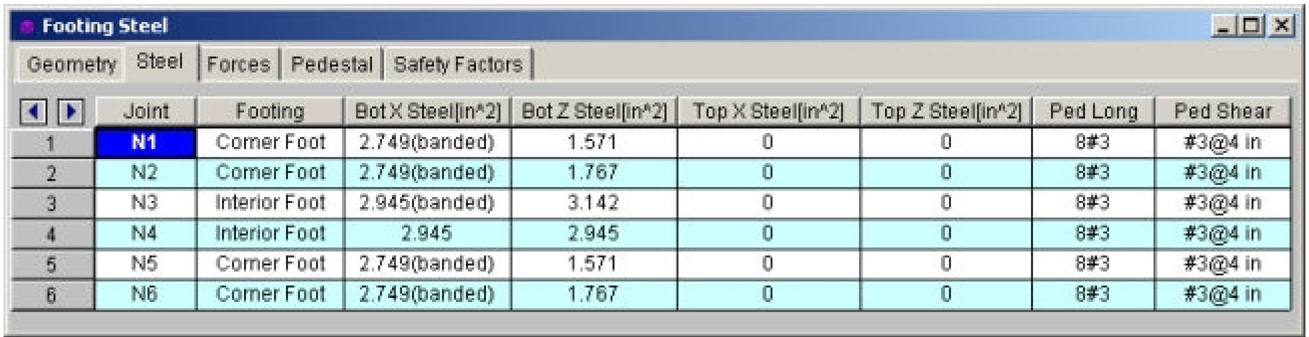
Two Way (Punching) Vc: 146.4062 k One Way (X Dir. Cut) Vc 103.0266 k One Way (Z Dir. Cut) Vc: 65.0694 k

		Punching		X Dir. Cut		Z Dir. Cut	
Description	Categories and Factors	Vu(k)	Vu/⌀Vc	Vu(k)	Vu/⌀Vc	Vu(k)	Vu/⌀Vc
ACI 9-1	1.4DL+1.7LL	93.4135	.7506	37.4541	.4277	36.8722	.6667
ACI 9-2	1.05DL+1.275LL+1.275WL	63.5717	.5108	20.8855	.2385	25.0932	.4537
ACI 9-3	.9DL+1.3WL	32.0711	.2577	12.7402	.1455	12.6593	.2289

Footing Shear Check

Geometry | Steel | Footing Code Check | Footing Shear Check | Pedestal | Safety Factors

	Joint	Footing	UC Shear	Vux[k]	Gov LC	UC Shear	Vuz[k]	Gov LC
1	N2	Corner Foot	.9766	19.2561	9	.0223	1.58	9
2	N4	Inerior Foot	.9141	36.0476	9	.1427	14.0655	9
3	N6	Corner Foot	.9748	19.2216	9	.0222	1.5768	9



Footing Steel

Geometry | Steel | Forces | Pedestal | Safety Factors

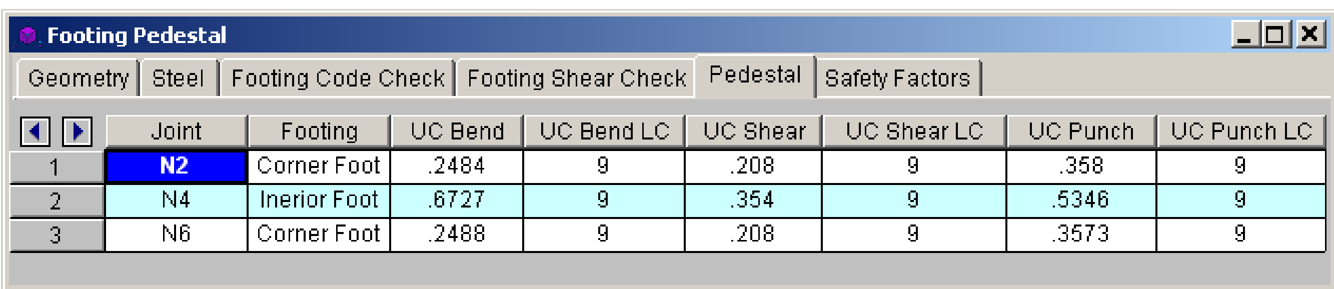
	Joint	Footing	Bot X Steel[in^2]	Bot Z Steel[in^2]	Top X Steel[in^2]	Top Z Steel[in^2]	Ped Long	Ped Shear
1	N1	Corner Foot	2.749(banded)	1.571	0	0	8#3	#3@4 in
2	N2	Corner Foot	2.749(banded)	1.767	0	0	8#3	#3@4 in
3	N3	Interior Foot	2.945(banded)	3.142	0	0	8#3	#3@4 in
4	N4	Interior Foot	2.945	2.945	0	0	8#3	#3@4 in
5	N5	Corner Foot	2.749(banded)	1.571	0	0	8#3	#3@4 in
6	N6	Corner Foot	2.749(banded)	1.767	0	0	8#3	#3@4 in

One-way and two-way shear are checked per ACI 318-11 Section 15.5. RISAFoot checks shear assuming only the concrete resists the applied shear; the contribution of the reinforcing steel to shear resistance is ignored. Two-way punching shear checks for the pedestal are reported on the Pedestal tab of the footing results spreadsheet.

One way shear is calculated for a “cut” through the footing a distance “d” from the face of the footing, where “d” is the distance from the top of the footing to the centerline of the reinforcing steel. For RISAFoot, the “d” distance value is equal to the footing thickness minus the rebar cover and one bar diameter (we do not know how the r/f is placed so we use the interface between the bars in each direction. The soil bearing profile from the edge of the footing to the cut location is integrated to obtain the total shearing force that must be resisted by the concrete. This is done for both directions (width cut and length cut) on both sides of the pedestal for a total of four one way shear calculations.

Pedestal Design

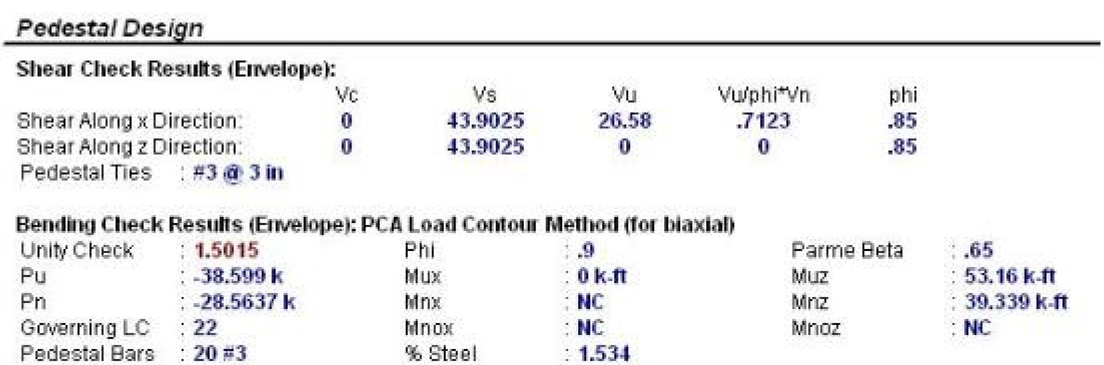
RISAFoot will design the longitudinal and shear reinforcement for rectangular pedestals. This information is presented in tabular form within Footing Results spreadsheet and graphically in the Footing detail report.



Footing Pedestal

Geometry | Steel | Footing Code Check | Footing Shear Check | Pedestal | Safety Factors

	Joint	Footing	UC Bend	UC Bend LC	UC Shear	UC Shear LC	UC Punch	UC Punch LC
1	N2	Corner Foot	.2484	9	.208	9	.358	9
2	N4	Inerior Foot	.6727	9	.354	9	.5346	9
3	N6	Corner Foot	.2488	9	.208	9	.3573	9



Pedestal Design

Shear Check Results (Envelope):

	Vc	Vs	Vu	Vu/phi*Vn	phi
Shear Along x Direction:	0	43.9025	26.58	.7123	.85
Shear Along z Direction:	0	43.9025	0	0	.85

Pedestal Ties : #3 @ 3 in

Bending Check Results (Envelope): PCA Load Contour Method (for biaxial)

Unity Check	: 1.5015	Phi	: .9	Parme Beta	: .65
Pu	: -38.599 k	Mux	: 0 k-ft	Muz	: 53.16 k-ft
Pn	: -28.5637 k	Mnx	: NC	Mnz	: 39.339 k-ft
Governing LC	: 22	Mnox	: NC	Mnoz	: NC
Pedestal Bars	: 20 #3	% Steel	: 1.534		

Pedestal Flexure

For flexure RISAFoot uses a rectangular stress block. For biaxial bending situations the load contour method is used. RISAFoot does not account for slenderness effects.

For flexure design RISAFoot presents the required longitudinal reinforcement assuming an equal distribution of bars around the perimeter of the pedestal. Supporting information includes the axial and bending contributions to resistance Pn, Mnx and

Mnz. The total factored axial and bending loads Pu, Mux and Muz are also listed. The contour method beta factor and Mnox and Mnoz are given, the moments representing the nominal uniaxial bending capacity at the axial load Pn. Finally, the Phi factor and the resulting unity check (demand vs. capacity) ratio is given.

For vertical reinforcing design, note that RISAFoot will meet the provisions of the ACI code in section 15.8.2.1 that states that in pedestals you must have a minimum reinforcement of 0.5% of the pedestal cross section.

Pedestal Shear

For shear design in either direction RISAFoot presents the required tie spacing. Also given are the concrete and steel contributions to resisting shear Vc and Vs. The total factored load Vu is listed. Finally, the Phi factor and the resulting demand vs. capacity ratios are given.

Pedestal Punching Shear

Punching shear is calculated for a perimeter a distance d/2 from the pedestal all the way around the pedestal. The soil bearing profile is integrated, minus the volume bounded by this perimeter, to obtain the total punching force to be resisted. For cases where the pedestal is near a corner of the footing, RISAFoot considers the punching perimeter to extend out to the footing edge if the pedestal is within a distance d from the edge of the footing.

RISAFoot presents the Vu and Vc values for punching and both one way shears as well as the shear demand vs. capacity ratio. So as long as this ratio is less than 1.0 the footing is adequate for shear. The **Forces** tab of the **Footings** spreadsheet gives the two way and one way shear capacities in tabular form.

Stability Results

RISAFoot will check the stability results for overturning and sliding. This information is presented in tabular form within Footing Results spreadsheet and graphically in the Footing detail report.

Footing Safety Factors

Geometry | Steel | Forces | Pedestal | Safety Factors

	Joint	Footing	OSF-xx	LC	OSF-zz	LC	SR-xx	LC	SR-zz	LC
1	N1	Corner Foot	2.826	8	3.988	8	32.561	3	1.064	7
2	N2	Corner Foot	2.685	7	4.032	7	32.838	3	.936	8
3	N3	Interior Foot	4.372	8	6.382	8		1	1.515	7
4	N4	Interior Foot	3.8	7	6.433	7		1	1.247	8
5	N5	Corner Foot	2.826	8	3.989	8	32.561	3	1.064	7
6	N6	Corner Foot	2.683	7	4.033	7	32.838	3	.935	8

Overturning Check (Service)

Description	Categories and Factors	Mo-XX (k-ft)	Ms-XX (k-ft)	Mo-ZZ (k-ft)	Ms-ZZ (k-ft)	OSF-XX	OSF-ZZ
ASCE 2 (b)	1DL+1LL+1SL	13.7738	122.2019	0	188.8576	8.8721	NA
ASCE 2 (c)	1DL+1LL+1RL	13.7738	122.2019	0	188.8576	8.8721	NA
ASCE 4 (a)	.6DL+1.5WL	33.392	62.8353	21.6066	89.3476	1.8817	4.1352
ASCE 4 (b)	.6DL-1WL	17.963	67.1336	0	103.752	3.7373	NA

Mo-XX: Governing Overturning Moment about AD or BC
Ms-XX: Governing Stablizing Moment about AD or BC
OSF-XX: Ratio of Ms-XX to Mo-XX

Sliding Check (Service)

Description	Categories and Factors	Va-XX (k)	Vr-XX (k)	Va-ZZ (k)	Vr-ZZ (k)	SR-XX	SR-ZZ
ASCE 2 (b)	1DL+1LL+1SL	13.3859	17.5139	4.9188	17.5139	1.3084	3.5606
ASCE 2 (c)	1DL+1LL+1RL	13.3859	17.5139	4.9188	17.5139	1.3084	3.5606
ASCE 4 (a)	.6DL+1.5WL	13.1902	8.9532	4.4009	8.9532	.6788	2.0344
ASCE 4 (b)	.6DL-1WL	13.1858	11.4855	6.3512	11.4855	.8711	1.8084

Va-XX: Applied Lateral Force to Cause Sliding Along XX Axis
Vr-XX: Resisting Lateral Force Against Sliding Along XX Axis
SR-XX: Ratio of Vr-XX to Va-XX

Overturning Check

Overturning calculations are presented as shown in the figure above. For all service combinations both the overturning moments and the stabilizing moments are given along with the calculated safety factor.

Overturning moments are those applied moments, shears, and uplift forces that seek to cause the footing to become unstable and turn over. Resisting moments are those moments that resist overturning and seek to stabilize the footing. The overturning safety factor (OSF) is the sum of resisting moments divided by the sum of overturning moments. Most codes require that this factor be greater than 1.5. Overturning safety factor calculations are based on the service load combinations only and are calculated in both the X and Z directions.

Overturning results from applied moments and applied shears and the summation of all these forces becomes the overall "overturning moment".

Note

- The OSF is calculated separately in both the X and Z directions as: resisting moment / overturning moment. Overturning is evaluated about all edges of the footing and the worst case in each direction is reported.
- The width and length of the footing are not increased or optimized to prevent the stability checks from violating the required overturning or sliding safety factors entered by the user.

Sliding Check

RISAFoot provides sliding checks for all of the service combinations.

The applied forces and the resisting forces, due to friction and passive soil resistance, are calculated and displayed. The final result is a sliding ratio of resistance vs. applied force that indicates a footing susceptible to sliding when the ratio is less than 1.0.

Concrete Bearing Check

The concrete bearing and demand /capacity ratios for each design combination are presented.

IMPORTANT: These concrete bearing checks are only based on the vertical applied loads, i.e. bending effects that increase or decrease bearing area and pressure are not considered.

The concrete bearing check in RISAFoot checks whether the footing concrete is adequate to resist in bearing the vertical loads imposed on the pedestal. These calculations are per Section 10.14 of the ACI 318-11 code. The results are presented as the demand vs. capacity ratio.

Footing Stability & Overturning Calcs

Calculation of OTM Stability Ratio

One of the important results from any footing analysis is a ratio of the stabilizing moments to the de-stabilizing moments. This is referred to as the Stability Ratio or the Safety Factor for overturning. RISAFoot calculates this value differently than the way many engineers would traditionally approach a hand calculation. The RISA method is a bit more complicated, but will provide a more realistic representation of the true safety factor versus overturning whenever uplift loads are present. A comparison of the two methods is provided below.

The overturning safety factor (OSF) is the sum of resisting moments divided by the sum of overturning moments. Most codes require that this factor be greater than 1.5. Overturning safety factor calculations are based on the service load combinations only and are calculated in both the X and Z directions.

Traditional / Simplistic OTM Calculation

Most hand calculations are done on the assumption that all vertical loads are stabilizing loads that are only capable of producing a stabilizing moment whereas all lateral shear and moments will act to de-stabilize the structure. As an example consider the case where you have the following Loads applied to a 4x4 footing with a pedestd266al that extends 2 ft above the bottom of the footing.

Force Effect	Dead Load	Wind Load
Axial	25 kips	+/- 10 kips
Shear	0	5 kips

In the traditional method, the axial forces are combined together to form a total vertical load. Since the TOTAL EFFECT of the axial loads is considered a stabilizing effect the resisting moment is considered to be P*L/2. Since the only de-stabilizing effect is assumed to be the shear force the de-stabilizing moment would be calculated as V*H. These calculations are summarized in the table below.

Wind Effect	Stabilizing Moment	DeStabilizing Moment	OTM Safety Factor
Downward	70 k-ft	10 k-ft	7.0
Uplift	30 k-ft	10 k-ft	3.0

True Safety Factor Method (used by RISAFoot)

The True Safety Factor method used by RISAFoot involves taking a look at EVERY component of every load and deciding whether it has a stabilizing or a de-stabilizing effect. Overturning moments are those applied moments, shears, and uplift forces that seek to cause the footing to become unstable and turn over. Resisting moments are those moments that resist overturning and seek to stabilize the footing. These overturning checks are performed for overturning about each edge of the footing. This is important for footings with eccentric pedestals where the vertical loads will have different stabilizing effects for each face.

By summing up all the resisting moments and comparing them to the sum of all the overturning moments we get a better sense of the true tendency of the structure to overturn. Taking the example listed above, it is clear that when the wind forces act in compression, the two method will produce identical results. That means that if the Wind shear is increased by a factor of 7.0, then the footing would be in net overturning.

However, the stabilizing moment in the uplift case would be viewed as the FULL dead load * L/2 rather than the reduced axial load used in the traditional method. Similarly, the de-stabilizing load would be viewed as the sum of the moments about the edge of the footing. This includes the de-stabilizing effect of the wind uplift and is equal to P_wind* L/2 + V*H.

The OTM Safety Factor calculations for the True Safety Factor Method are summarized in the table below:

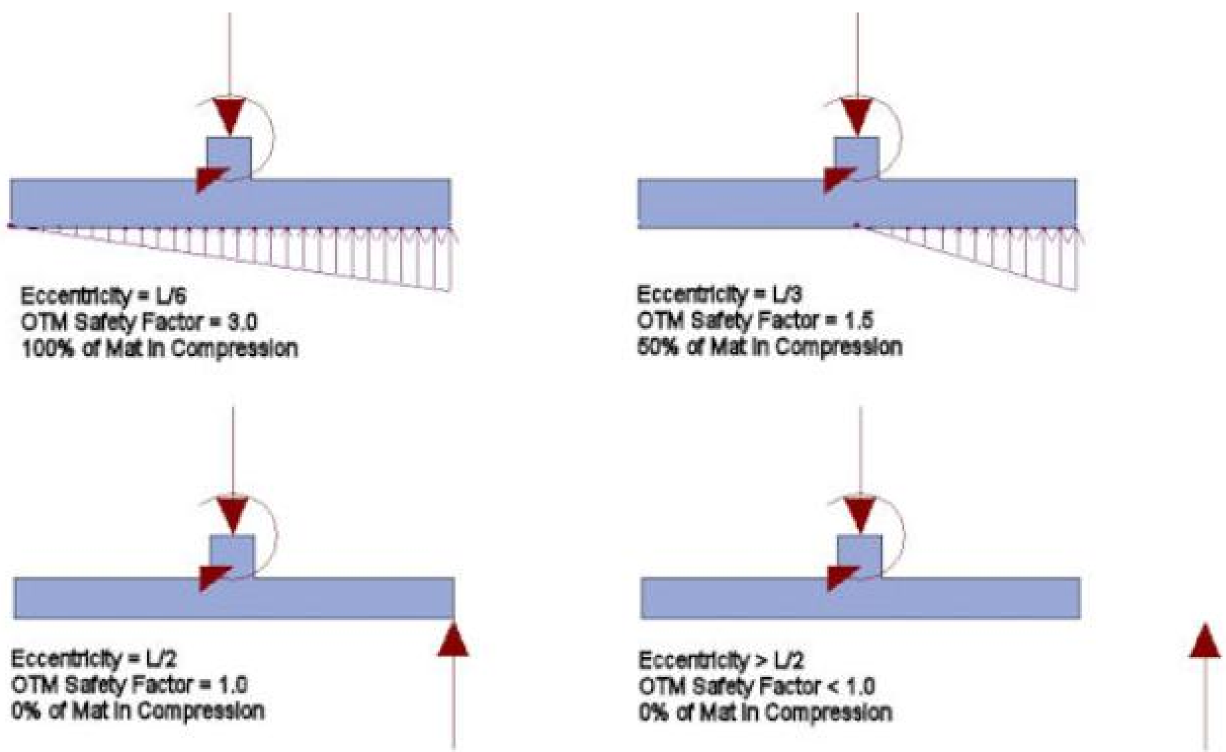
Wind Effect	Stabilizing Moment	DeStabilizing Moment	OTM Safety Factor
Downward	70 k-ft	10 k-ft	7.0
Uplift	50 k-ft	30 k-ft	1.67

Comparing the Two Methods

Which is a more accurate representation of the safety factor of stability ratio for the uplift case? The best way to judge this is to multiple the wind load by a factor equal to the Safety factor and to see if the footing still works. If the wind effects (shear and uplift) are multiplied by a factor of 3.0 as given by the traditional method, then the footing will be unstable due to net uplift and net overturning. The Safety Factor would be calculated a negative value which has little physical meaning. If the wind effects (shear and uplift) are multiplied by a factor of 1.67 as given by the True Safety Factor method, then the stabilizing moment will equal the de-stabilizing moment and the safety factor becomes 1.0 as predicted.

Calculation of Moment and Shear Demand for Unstable Footings

Older versions of the program could not handle situations where the Ultimate Level load combinations resulted in net overturning of the Footing. This is because the previous versions always relied on the soil bearing pressure calculations to derive the moment and shear demand in the footing. For net overturning cases, the current version of the program assumes that the design shear and moment can be based on the total vertical load and the net eccentricity of that load.



In these cases, where the resultant is off the footing, the design shear force in the footing will be constant equal to the net resultant minus any effects of self weight or over burden. Also, the basis for the design moment at the face of the pedestal

will be the load resultant times the distance from the resultant to the face of the pedestal. Any negative moment effect due to the self weight of the footing slab and overburden would be subtracted out when determining the final design moment.

RISAFoundation Interaction

RISAFoundation has the ability to transfer load information from RISA-3D and RISAFloor and use this information to design foundations. The following entry will describe how these loads come across the design modules so you can accurately use RISAFoundation for your foundation design.

RISAFoundation Interaction with RISA-3D

RISAFoundation transfers joint reactions from RISA-3D into RISAFoundation by way of load categories. This is not the same as load combinations. Load categories are defined in the Basic Load Cases spreadsheet of RISA-3D.

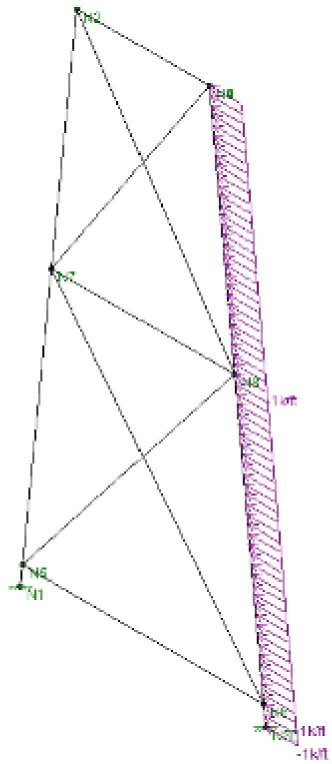
To get your reactions to transfer for use into RISAFoundation, you must have your loads split into specific load categories under the Basic Load Cases spreadsheet in RISA-3D.

Note:

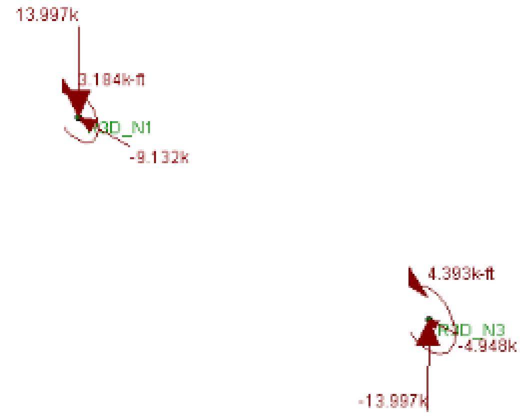
- If you do not define a load category for a basic load case, then the loads from that case will not be transferred into RISAFoundation.
- You must run at least a single load combination.
- When you are working between RISA-3D and RISAFoundation global parameters will only be brought over to RISAFoundation the first time you transfer from the RISA-3D module. After this first transfer, any time you change something in RISA-3D from global parameters that would affect RISAFoundation, then you would also have to go into RISAFoundation and make the change.
- Boundary conditions defined as **Reaction** or **Spring** will be transferred but **Fixed** boundary conditions will not.

Example of Interaction

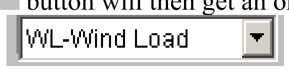
A simple example would be the braced frame structure seen below on the left. This has been created in RISA-3D. The load applied to this structure came from Load Category WL (wind load). Note that this structure had other loads applied as well. But the wind loads are used as an example.



RISA-3D Wind Loads Graphical Representation



RISAFoundation Wind Loads Graphical Representation

First a load combination is run (note that this does not have to have wind in the load combination), then RISAFoundation is chosen from the Director in RISA-3D. The Director is the means by which navigation to different modules in the program is possible. When RISAFoundation is first entered dead loads applied to points in a plan view of your structure are displayed. Clicking the  button will then get an orientation similar to the one shown above. Next, choose WL-Wind Load from the drop down list. 

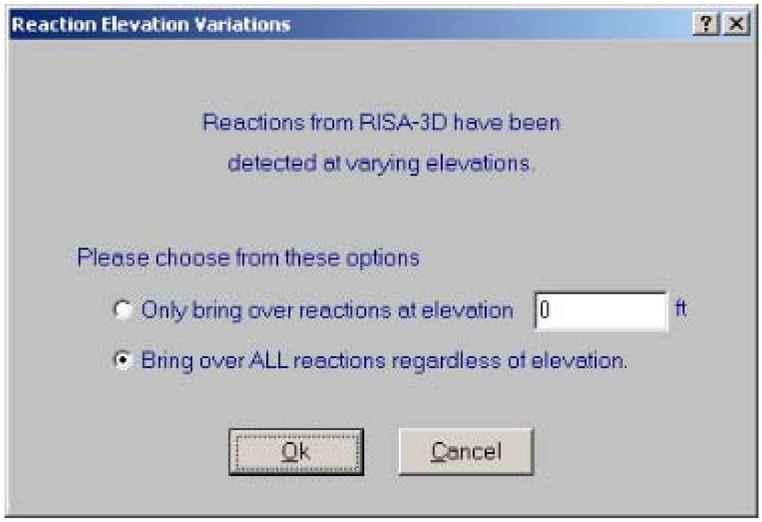
The loads shown above on the right will be displayed. Note that these loads appear consistent for the type of loading present.

Note:

- The points are relabeled in RISAFoundation, though labeled in a way that can be easily compared to the original RISA-3D model joints. You can scroll through the different types of loads and compare them to RISA-3D for accuracy verification.
- Load combinations do not transfer over from RISA-3D, thus they will need to be re-created in RISAFoundation. An easy way to do this would be to just copy the spreadsheet over from RISA-3D.

Foundations at Different Elevations

If, when moving into the RISAFoundation from RISA-3D, there are foundations that are at different elevations, a dialog box will pop-up that looks like this:



You may choose to bring over reactions at a single elevation of your choice. This may be helpful if there are some reactions that are due to an adjacent structure or support that is not a foundation.

You may choose to bring over all reactions regardless of elevation. This is a good option to have if there is a partial basement level or any condition where there truly are foundations at different levels.

Note:

- All of the differing elevations will be brought in at a single elevation.

RISAFoundation Interaction with RISAFloor

This interaction is a little more involved because it involves three programs (RISA-3D, RISAFloor and RISAFoundation). For RISAFoundation, the bases of columns and the bottom of walls will be considered boundary conditions. Thus, loads are applied at the bases of these elements for foundation design in RISAFoundation.

Gravity Elements in RISAFloor

Elements (columns, walls) defined as gravity only are the only elements which transfer load directly from RISAFloor into RISAFoundation. Wall reactions come in as line loads and column reactions come in as point loads.

Lateral Elements in RISAFloor

When applying loads to lateral members within RISAFloor, those loads are simply attributed to the members brought over into RISA-3D. Thus, RISAFoundation does not bring any information in from RISAFloor concerning lateral members. All of this information is taken directly from RISA-3D. From this point we are just working as if we only had RISA-3D and RISAFoundation working together. See above.

Limitations

RISAFoundation can not recognize moving loads

Moving Loads, not described as a load category, and their reactions can't be read as loading within RISAFoundation, thus loads will need to be applied manually to the foundations to account for these.

RISAFoundation can not recognize reactions due to dynamic analyses

Because RISAFoundation considers loads based on load categories, it is not possible to consider dynamic loads in calculation of footings, thus these loads need to be applied manually.

RISAFoundation will not consider non-linear load effects

In a true Non-Linear analysis (with Tension Only or Compression Only members), the individual reactions would vary depending on which LC was used to create it because different members may be active for different LC's. In that manner the Load Category reactions that make their way into RISAfoundation are imperfect because a DL or LL reaction would technically be different for the WLX and WLZ cases.

RISAFoundation will not consider second order effects

In a true second order analysis, the P-Delta effect would vary depending on which LC was used to create it because of the non-linearity of the effect when multiple loads are combined together. In that manner the Load Category reactions that make their way into RISAfoundation are imperfect because the P-Delta effect on a DL or LL reaction would technically be different depending on whether the final Load Combination included WLX or WLZ cases.

RISAFoundation requires your Y-axis to be vertical

Models coming from RISA-3D need to have the Y-axis to be vertical to be brought into RISAFoundation. If the Y-axis is not the vertical axis, then a dialog box will pop up when you try to link to RISAFoundation with this message.

Section Sets

Cross-section properties may be assigned to members in one of two ways; either by choosing a shape directly from the steel database, or by using a section set. Section sets provide a way to group members so that they have the same properties. Adjusting the set properties, rather than selecting and adjusting each member can achieve changes to the properties for all of the members in the set. You must use section sets if you want to perform timber code checks or steel shape optimization.

The cross section data for the members is recorded on the **Section Sets** spreadsheet. Cross sectional properties can be entered manually or may be retrieved from one of the shape databases. Currently the databases include Hot Rolled Steel, Cold Formed Steel, Wood, Concrete, and Aluminum. Once the section is defined, it is then referenced on the **Members** spreadsheet when assigning properties to a member.

To Define a New Section Set

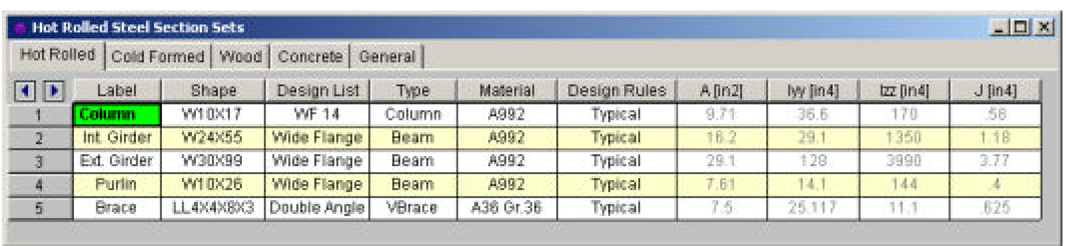
1. On the **Spreadsheets** menu click on **Section** to open the spreadsheet.
2. Select the tab for the Material type that you will be using
3. Enter the section set label and other information. You may choose the material by clicking on the arrow in the cells. For help on an item, click and then click the item.
4. You may open the database by clicking on the arrow in the **Shape** column.

Note

- If you leave the Shape name blank for **General Sections**, then you may enter the rest of the properties (A, I, J) yourself without having to define an arbitrary shape.
- For rectangular, circular, and pipe shapes you can use the On-Line shapes to automatically define the shape properties. See On-Line Shapes.

Section Sets Spreadsheet

The **Section Sets** spreadsheet records the section properties for the member elements and may be accessed by selecting **Sections** on the **Spreadsheets** menu.


Hot Rolled Steel Section Sets

Hot Rolled | Cold Formed | Wood | Concrete | General

	Label	Shape	Design List	Type	Material	Design Rules	A [in2]	Iyy [in4]	Izz [in4]	J [in4]
1	Column	W10X17	WF 14	Column	A992	Typical	9.71	36.6	170	.56
2	Int. Girder	W24X55	Wide Flange	Beam	A992	Typical	16.2	29.1	1350	1.18
3	Ext. Girder	W30X99	Wide Flange	Beam	A992	Typical	29.1	128	3990	3.77
4	Purlin	W10X26	Wide Flange	Beam	A992	Typical	7.61	14.1	144	.4
5	Brace	LL4X4X8X3	Double Angle	VBrace	A36 Gr.36	Typical	7.5	25.117	11.1	.625

The following are input columns on the spreadsheet that may be used to specify cross section data for the members in the model.

Section Label

The **Section Label** is the label you'll reference on the **Members** spreadsheet to assign properties to a member. This label can be anything you wish, so long as it's not the same as any other section set label.

Section Database Shape

The **Shape** field is used to obtain properties from the shape databases. To use the database, simply enter the name of the database shape and the shape properties (A, I, J) will be filled in automatically. Click to pick from the database.

If you don't want to use a database shape and wish to enter the shape properties directly, just leave this field blank.

Note

- For rectangular, circular, and pipe shapes you can use the On-Line shapes to automatically define the shape properties. See On-Line Shapes.

Section Material

The **Material** field is used to enter the label of the material. The material must be defined on the **Materials** spreadsheet or on the **Wood Properties** spreadsheet.

Member Type

Enter the member type for the section set. The choices are *Column, Beam, Vertical Brace, and Horizontal Brace.*

Here are the main effects that the member type will have on your structure:

1) If you are using concrete, this will define the rebar layout (column vs beam) .

2) If you are using design lists, they specifically reference the member type.

3) If you are using member area loads, loads will not be attributed to members defined as Hbraces or Vbraces.

4) If you are using the RISA-Revit link the link will not work properly unless you use member types.

5) If you are integrating with RISAConnection, the connection validation requires proper use of member types.

6) If you are using the seismic detailing provisions then you must use proper member types.

Design List

Enter the **design list** type that you wish to use for this section set. This entry will affect the members that are available to program when it is suggesting alternate or optimized shapes. Refer to Design Optimization for more information on the member optimization procedure. Also refer to Appendix A – Redesign Lists for information on creating or editing these lists.

Design Rules

Enter the **design rules** type that you wish to use for this section set. When the program is checking alternate or optimized shapes, it will restrict its selections to members that obey the chosen design rules. Refer to Design Rules– Size / U.C. for more information.

Cross Section Properties

The cross section properties will be filled in automatically if you use a database shape. For **General Materials**, you may leave the shape field blank and enter these directly. **I**yy and **I**zz are for bending about the respective member local axes. Note that it's not a good idea to edit these fields if you already have a database shape assigned. Any changes will not be saved and will be replaced with the original values when the file is reopened or the shape is reentered in some other manner. You should create a new shape in the database to avoid this situation.

Note

- For Tapered WF shapes, only the shape properties at the I-end will be shown on the **Sections** spreadsheet. To view properties for both ends use the **Edit** button in the **Shape Selection** settings.

Shape Databases

For each material, there are several databases of common structural shapes such as Hot Rolled Steel Wide Flanges, Cold Formed Shapes, Wood, Concrete Tees, etc. You may also choose from shapes created in RISASection. You may type in the names directly, select shapes from these databases or add your own shapes.

- What Aluminum Shapes are available?

Database Shape Types

There are different types of shapes for each material type including General shapes. Names for each shape type follow a syntax so that they may be typed directly into the **Shape** field on the **Sections** spreadsheet or on the **Primary** tab of the **Members** spreadsheet. Alternately you may click the button to look up a shape and select it.

Hot Rolled Shapes

AISC, Canadian, Trade Arbed and custom Hot Rolled shapes are accessed by clicking the **Edit Shape Database** button from the **RISA Toolbar**, and then clicking the **Hot Rolled** tab. The hot rolled shapes and databases are more fully described in the Hot Rolled Steel Design section. See Hot Rolled Steel Databases for more information.

Cold Formed Shapes

Manufacturer and custom cold formed shapes are accessed by clicking the **Edit Shape Database** button from the **RISA Toolbar**, and then clicking the **Cold Formed** tab. The cold formed shapes and database are more fully described in the Cold Formed Design section. See Cold Formed Steel Databases for more information.

Concrete Shapes

Concrete shapes do not have a predefined database like hot rolled and cold formed steel. Instead, they are defined using a parametric shape code that may be assigned any depth or width. There are two types of shapes currently supported: Rectangular and Round. See Concrete Database for more information.

Wood Shapes

The available wood shapes are based on the dimension lumber and post and timber shapes given in the NDS. You may also double up or triple up these shapes. Note that these are all *nominal* sizes.

Allowable stress values for each shape are based on the NDS species and grade information given in the supplement for the NDS specification per the code chosen in the Codes tab of Global Parameters. See Wood Database for more information.

Aluminum Shapes

US, Canadian and custom Aluminum shapes are accessed by clicking the **Edit Shape Database** button from the **RISA Toolbar**, and then clicking the **Aluminum** tab. The aluminum shapes and databases are more fully described in the Aluminum - Databases section.

General Shapes

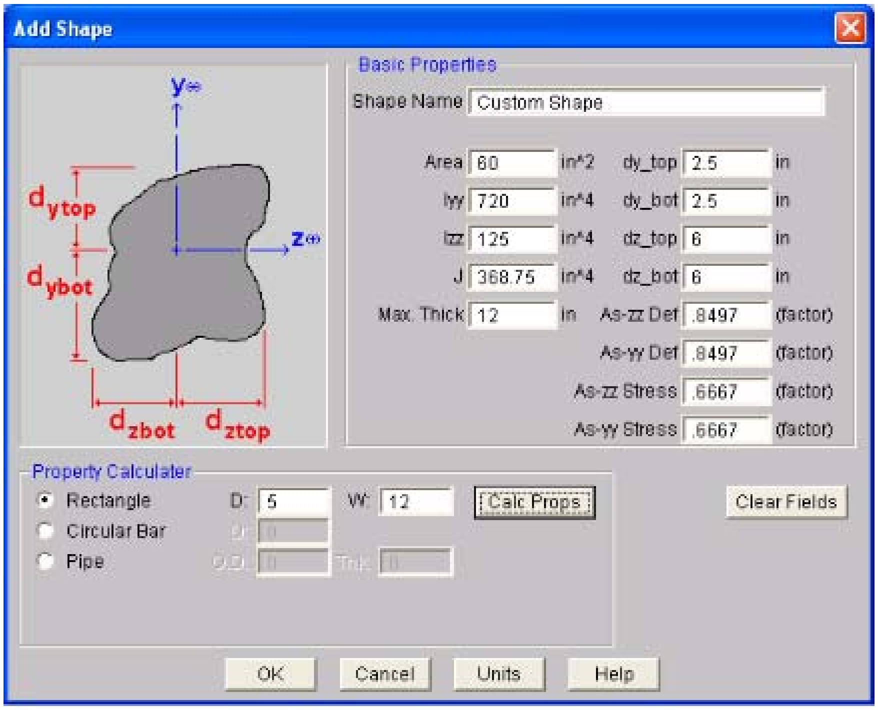
Arbitrary Shapes

Arbitrary Shapes are a special, catch-all shape. This arbitrary shape type is provided so that any shape can be added to the shape database.

AISC code checks are not calculated for arbitrary shapes since their place in the specification is unknown. Everything else will be calculated for them (forces, deflections, stresses). The max thickness (Max thick) value for the cross section is used to determine the pure torsional shear stress for the shape. "J" is the torsional constant. The "d" values (the distances to the extreme fibers) allow the program to calculate stresses at the extreme fibers.

Note:

- These shapes will generally be rendered using a greenish cruciform shape. The center of the cruciform will reflect the centroid of the section with the tips of the cruciform representing the distances from the neutral axis to the extreme fiber.
- Refer to Member Shear Deformations and Member Shear Stresses for more information on the shear area factors (As-zz Def, As-yy Def, .As-zz Stress, & As-yy Stress) shown in the figure below.



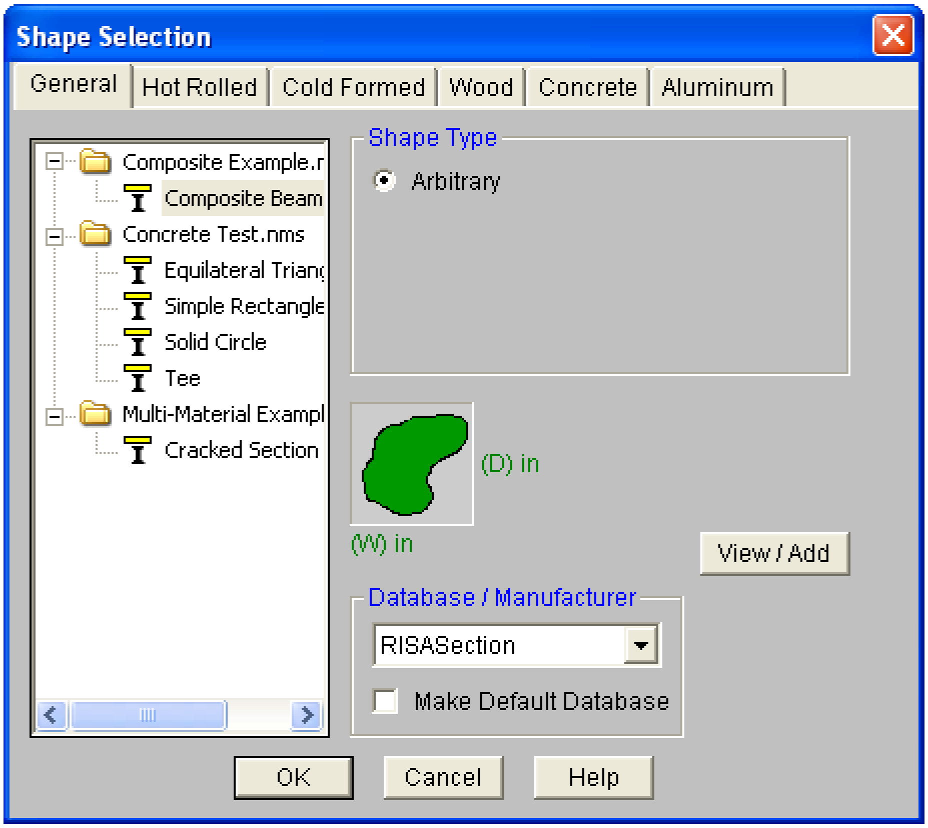
To Add an Arbitrary Shape to the Database

- To enter an arbitrary shape in the database click  , select the **General** tab. Set the shape type to Arbitrary and click **Add.** Enter the shape name and properties.

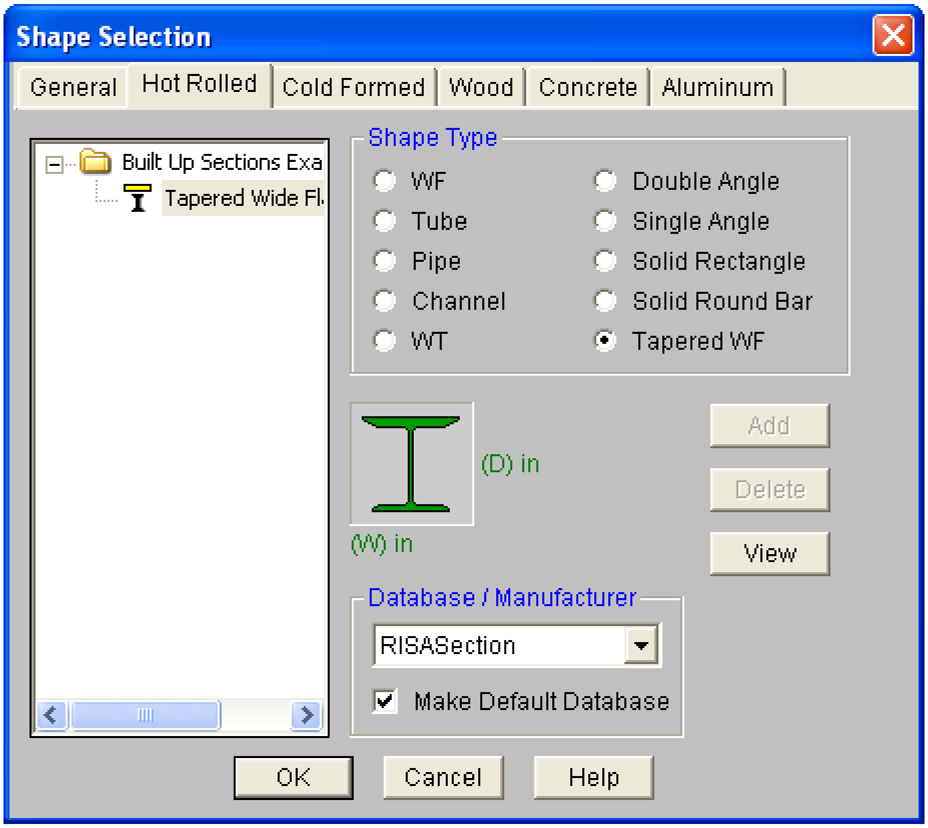
RISASection Files

You may create simple or complex sections in RISASection and then import those sections for use in your model. Sections that exist in RISASection (files located in the file specified in **Tools - Preferences - File Locations**) will be available for use in the model. Each section must have a unique name for it to be available.

The shapes that are designated as General Material, Arbitrary Shape Type in RISASection will show up in the **General** tab under the "**RISASection**" **Database/Manufacturer**.



The shapes that are designated as Hot Rolled Steel Material in RISASection will show up in the **Hot Rolled** tab under the appropriate Shape Type (Channel, Wide Flange, etc.) when "**RISASection**" is selected as the **Database/Manufacturer**.



Note: Currently, RISASection can only import General and Hot Rolled Steel shapes. More material choices will be available in a future version.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **RISASection Integration**.

Troubleshooting RISASection / RISA-3D Interaction

There are a few common issues that arise when attempting to access RISASection files from within RISA-3D.

- When RISA-3D launches, it will only import RISASection files that exist in the directory specified in **Tools-Preferences - File Locations**.
- RISA-3D only reads in database files when it is first launched. Therefore, if a RISASection file is created or modified, then RISA-3D must be closed and re-started before it will recognize the new or modified section.

There are some common mistakes that are made from within the RISASection program that may cause an issue when trying to read that shape from within RISA-3D.

- The name of the RISASection "File" is actually different from the name of the section itself. Both names will appear in the RISA-3D database, but it is important to give your sections unique names in order to properly tell them apart. Note: each RISASection file can contain many different sections. Hence the need to name each individual section.
- Unless specifically identified as a Hot Rolled Material type and shape (from within RISASection), the shapes will be imported as Arbitrary shapes with a General Material type
- RISASection can save files with a new *.nmsx file type (for RISASection 2.0 or higher), or an old *.nms (for RISASection version 1.1 or older). If saved as the old file type, the sections will only come in as Arbitrary members with a General material type.
- If the file is saved with a *.nmsx extension it will not be read into older versions of the RISA programs (prior to RISA-3D version 9.1.1.

Structural Desktop (SDT) Shapes

The SDT database is provided by the Structural Desktop software in a file called SDT.FIL. Structural Desktop automates drawing production of RISA models. The SDT database is provided for shapes that are not directly supported in RISA models (such as bar joists) but are available in Structural Desktop. For more information on Structural Desktop see www.structuraldesktop.com.

On-Line Shapes

On-Line shapes are shapes whose dimensions are defined directly in the syntax of the shape name. On-line shapes are not stored in the shape database because there is enough information from the label syntax to calculate all the shape properties. A pipe, for example, can be fully defined by specifying the thickness and diameter.

These shapes are treated just like database shapes for stress calculations. Currently, Pipes, Solid Rectangular and Solid Circular shapes are defined on-line as discussed below in Pipe Database Shapes, Solid Rectangular Shapes, and Solid Circular Shapes.

Pipe Database Shapes

Pipe shapes, which are hollow circular shapes, are entered as on-line shapes. The syntax for these shapes is "PIdiaXthick", where "dia" is the pipe outside diameter and "thick" is the pipe thickness (in inches or centimeters). For example (assuming US Standard units), PI10X.5 would be a 10" diameter pipe with a wall thickness of 1/2".

Solid Rectangular Shapes

These shapes can be defined as on-line shapes. The syntax is "REhtXbase", where "ht" is the rectangle height and "base" is the rectangle base (in inches or cm). For example, RE10X4 would be a 10" deep, 4" width rectangular shape (assuming US Standard units). These shapes can also be defined in the Shape Editor. When defined in the Shape Editor the depth of the solid rectangular section must always be greater than or equal to the width.

Solid Circular Shapes

These shapes are defined as on-line shapes. The syntax is "BARdia", where "dia" is the circle diameter. For example (assuming metric units), BAR2 would be a circular bar with a diameter of 2 cm.

Database Files

The shape databases are stored in the database files (*.FIL). These files may only be edited through the program. The path to these files is set in the **Preferences** on the **Tools** menu.

Note

- Alterations to the shape databases are not permanent unless you agree to save them. Changes that are not saved only remain valid for the current session and will not be present the next time you start RISA.
- New shapes are always added to the bottom of the database.
- To delete a shape specify the database and shape type you wish to delete and then click the **Delete** button.
- To edit a shape click the **Edit** button and edit the shape properties. Values can only be manually edited here, nothing will be recalculated. If you wish to have all the values for a shape recalculated, you will need to delete the shape and then add it again with the new properties. For Tapered WF shapes only, you can click **Calc Props** to recalculate the shape properties based on the basic dimensions. Tapered WF shapes are stored parametrically, so there are no database values to edit since they are calculated on the fly whenever an analysis is performed.

Seismic Detailing - Input / Design Rules

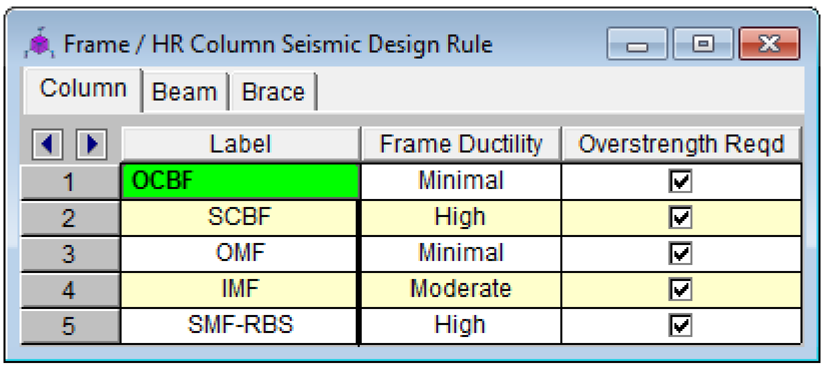
Seismic Design Rules can be applied to Column, Beam and VBrace members in the model. These will invoke various design or code check requirements according to AISC design provisions (AISC 360-2005, AISC 341-2005, AISC 358-2009).

These design provisions primarily apply only to Hot Rolled steel members. However, the **Overstrength Req'd** design option applies to all members to which that seismic design rule has been applied. The reason for this is so that members (collectors or such) which require design to the overstrength provisions per the requirements of ASCE-7 can be automatically designed to the higher force requirements of the Overstrength load combinations.

The default entry for a member's seismic design rule is None which means that no special seismic detailing provisions will apply to the code checking provisions for that member.

Seismic Design Rules - Hot Rolled Columns / General Frame

The first tab of the Seismic Design Rules spreadsheet applies to the frame in general and applies to the Hot Rolled columns in particular.



Label

The label is a user defined text string which is used as a unique identifier for each of the seismic design rules defined for the structure. The program comes pre-loaded with number of generic seismic design rules based on the AISC 341-2005 seismic detailing specification.

Ductility

This defines the basic ductility requirements for the frame. These ductility requirements apply equally to Beams, Columns and Braces.

High Ductility refers to a frame which requires **Seismically Compact** sections for members such as a Special Concentrically Braced Frame (SCBF) or a Special Moment Frame (SMF).

Moderate Ductility refers to a frame which require **Compact** sections for frame members, but which does not require the special Seismic Compactness defined in AISC 341. One example would be an Intermediate Moment Frame (IMF).

Minimal Ductility refers to a frame which does not have specific compactness requirements beyond the normal AISC specification. This can even include members with slender elements. One example would be an Ordinary Moment Frame (OMF).

Note

- Ordinary Concentrically Braced Frames (OCBF) require High ductility out of the brace members, but allow for Minimal ductility out of the other members. The program will automatically account for this brace ductility requirement. Therefore, the Frame Ductility may still be entered as Minimal.

- When checking the seismic compactness of members, the axial load used to calculate Ca is based on the worst case axial compression from the normal load combinations. It will not consider any axial force that occurs from an "overstrength" load combination.
- The Frame Ductility setting is used by the program when it is checking some of the miscellaneous beam-column moment connection requirements per AISC 358. For example: if the beam to column connection is specified by the user as a Reduced Beam Section (RBS) then the span to depth ratio of the beam must be greater that 7.0 to be considered a highly ductile frame, and greater than 5.0 to be considered a moderately ductile frame.

Overstrength Required

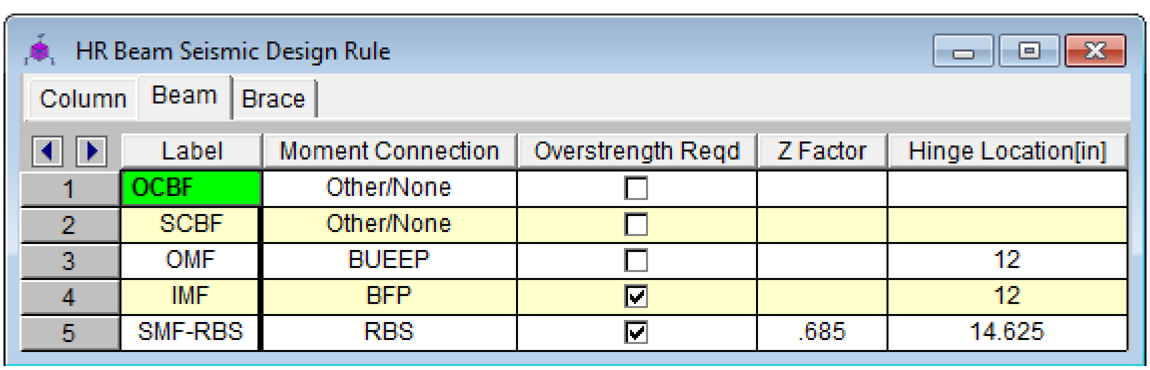
In seismic design some members may be required to be designed to an elevated / overstrength load. In RISA, this is done by creating load combinations with the Omega overstrength factors applied to the earthquake loads. If the column members are required to be designed to these load combinations, then this check box will be checked.

Note

- In many cases, it's only the axial effect of the overstrength loads that needs to be considered in the column overstrength code checks. In these cases, it could be overly conservative to consider the moment with these elevated axial forces. A future revision to the program will add an **Axial Only** option to this input field which would allow the program to ignore the effects of moment when the amplified axial force is taken into account.
- Generally speaking the 2005 version of AISC 341 requires that moment frame columns be designed for the overstrength loads whenever the axial force for the regular load combinations exceed 40% of the column's axial capacity. However, the 2010 version of the same specification appears to **always** require that these columns be designed to these overstrength loads.
- The program does not consider any of the limitations related to the "sum of shears which can be transmitted to the column" or the "sum of the expected strength of the braces".
- If a Seismic Design Rule is applied to a column member which is not hot rolled steel (such as a wood drag strut or collector), that member will ignore the design rules except for the Overstrength required flag. If this is selected, then the non-steel member's capacity will be checked against the forces derived from the overstrength seismic load combinations.
- Refer to the LC Generator and the Set BLC Entry sub-topics in the Load Combinations section for more information regarding the creation of overstrength load combinations.

Seismic Design Rules - Hot Rolled Beams

This tab contains information pertinent mostly for the design of hot rolled beams that are part of moment frames. However, the Overstrength Req'd option applies to any beam member which is assigned the seismic design rule. Hot Rolled beams which are part of braced frames will have their element slenderness checked against the ductility limitations set on the Frames / Columns tab.



Moment Connection

This field designates which of the pre-qualified moment connections defined in AISC 358-2005 or its supplement are being used for the beam to column moment connection. The options include the following:

Bolted Flange Plate (BFP) as described in Chapter 7 of the 2009 supplement to AISC 358.

Reduced Beam Section (RBS) as described in Chapter 5 of AISC 358.

Bolted Unstiffened Extended End Plate (BUEEP) and **Bolted Stiffened Extended End Plate (BSEEP)** as described in Chapter 6 of AISC 358.

Welded Unreinforced Flange - Welded Web (WUF-W) as described in Chapter 8 of the 2009 supplement to AISC 358.

Other/None: There are times where AISC 341 and AISC 358 have conflicting provisions. This includes the definition of the probable maximum moment at the hinge and the definition of the Strong Column / Weak Beam Ratio. When one of the pre-qualified connections is selected then the program will enforce the AISC 358 version of the provisions. When the **Other** connection is selected then, the AISC 341 version of these provisions will be enforced instead. This "connection option" should normally be used when the beam will not have moment connections (i.e. it is part of a braced frame).

Note

- The moment connection setting is used (in combination with the Frame Ductility setting on the column tab) to check some of the miscellaneous beam-column moment connection requirements per AISC 358. For example: if the beam to column connection is specified by the user as an Reduced Beam Section (RBS) then the column depth must be limited to a maximum of W36, and the beam weight cannot exceed 300 lbs / ft.

Overstrength Required

In seismic design some members may be required to be designed to an elevated / overstrength load. In RISA, this is done by creating load combinations with the Omega overstrength factors applied to the earthquake loads. If the beam members are required to be designed to these load combinations, then this box would be checked.

Note

- Beams in a braced frame may act as some form of a collector or drag strut. As such, the ASCE-7 seismic provisions would require that they be designed for the overstrength load combinations.
- If a Seismic Design Rule is applied to a beam member which is not hot rolled steel (such as a wood drag strut), that member will ignore the design rules except for the Overstrength required flag. If this is selected, then the non-steel member's capacity will be checked against the forces derived from the overstrength seismic load combinations.
- Refer to the LC Generator and the Set BLC Entry sub-topics in the Load Combinations section for more information regarding the creation of overstrength load combinations.

Z Factor

This factor is used to define the reduction in plastic hinge moment expected for Reduced Beam Sections. Enter in the ratio between the plastic section modulus for the reduced beam section and the unreduced beam. For RBS connections this value will vary greatly, but will always be less than 1.0. The program will not allow a value of less than 0.1 to be entered in by the user. If this value is left blank, then no reduction in moment is considered.

This factor will be used to determine the probable design strength and the strong column / weak beam moment ratio for the connection. It does NOT currently reduce the stiffness of the beam used in the analysis.

Note

- AISC 358 has some restrictions on the length and depth of cut that is allowed for the RBS section. RISA does not make any attempt to enforce these restrictions.

Hinge Location

This entry defines the location of the assumed plastic hinge (in inches) from the face of the column. This is used to determine the design moment at the face of the column as well as the strong column / weak beam ratio.

For RBS connections, this entry should be the distance from the face of column to the center of the reduced beam section.

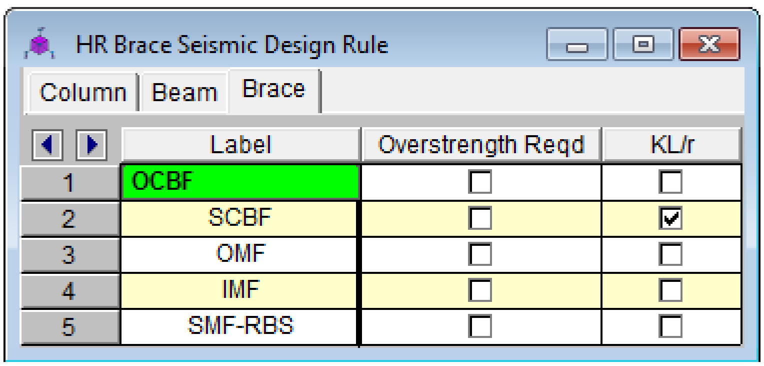
For stiffened connections (such as BSEEP, or BFP), this will usually be the distance from the face of column to the end of the stiffener, haunch, or flange plate.

For unstiffened end plate connections, this entry will usually be the lesser of 50% of the beam depth or 3 times the beam flange width.

For WUF-W connections, this entry will normally be set to 0.0.

Seismic Design Rules - Hot Rolled Braces

This tab contains information pertinent to the design of hot rolled braces. The only exception to this is the Overstrength Reqd option which applies to any VBrace member which is assigned the seismic design rule.



Overstrength Required

In seismic design some members may be required to be designed to an elevated / overstrength load. In RISA this is done by creating load combinations with the (Ω_0) overstrength factors applied to the earthquake loads. If the brace members are required to be designed to these load combinations, then this box would be checked.

Note

- If a Seismic Design Rule is applied to a brace member which is not hot rolled steel, that member will ignore the design rules except for the Overstrength required flag. If this is selected, then the non-steel member's capacity will be checked against the forces derived from the overstrength seismic load combinations.
- Refer to the LC Generator and the Set BLC Entry sub-topics in the Load Combinations section for more information regarding the creation of overstrength load combinations.

Max KL/r

In seismic design some members may have the following restriction on the maximum slenderness (KL/r) value that they are allowed to have.

$$\frac{KL}{r} \le 4\sqrt{\frac{E}{F_y}}$$

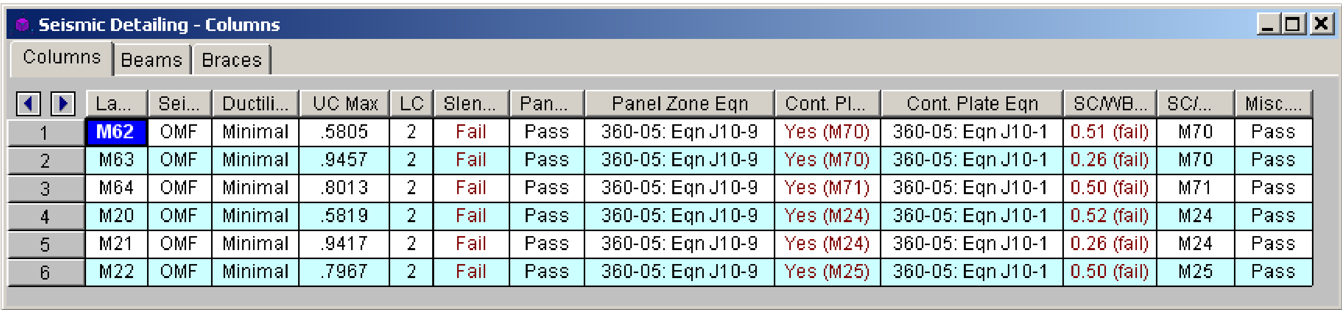
Examples would be braces that are part of a Special Concentrically Braced Frame (SCBF) as well as K, V, or inverted V braces in an Ordinary Concentrically Braced Frame (OCBF).

Seismic Detailing - Results

The seismic detailing results are presented in two ways: the seismic detailing results spreadsheet and the seismic detailing sections of the member detail reports. The seismic detailing results spreadsheet is intended to be a summary report. More detailed information can be found for each member on that member's detail report.

Seismic Results Spreadsheet - Columns

The first tab of the Seismic Design Rules spreadsheet applies to the Hot Rolled members which have been assigned to the Member Type of Column and which have a seismic design rule applied.



Seismic Detailing - Columns

Columns | Beams | Braces

	La...	Sei...	Ductili...	UC Max	LC	Slen...	Pan...	Panel Zone Eqn	Cont. Pl...	Cont. Plate Eqn	SC/WB...	SC/...	Misc....
1	M62	OMF	Minimal	.5805	2	Fail	Pass	360-05: Eqn J10-9	Yes (M70)	360-05: Eqn J10-1	0.51 (fail)	M70	Pass
2	M63	OMF	Minimal	.9457	2	Fail	Pass	360-05: Eqn J10-9	Yes (M70)	360-05: Eqn J10-1	0.26 (fail)	M70	Pass
3	M64	OMF	Minimal	.8013	2	Fail	Pass	360-05: Eqn J10-9	Yes (M71)	360-05: Eqn J10-1	0.50 (fail)	M71	Pass
4	M20	OMF	Minimal	.5819	2	Fail	Pass	360-05: Eqn J10-9	Yes (M24)	360-05: Eqn J10-1	0.52 (fail)	M24	Pass
5	M21	OMF	Minimal	.9417	2	Fail	Pass	360-05: Eqn J10-9	Yes (M24)	360-05: Eqn J10-1	0.26 (fail)	M24	Pass
6	M22	OMF	Minimal	.7967	2	Fail	Pass	360-05: Eqn J10-9	Yes (M25)	360-05: Eqn J10-1	0.50 (fail)	M25	Pass

Label, Seismic Design Rules and Req'd Ductility

These fields display the member's basic input data for reference purposes.

Max Code Check and Load Combination

This field summarizes the member code checks from all the load combinations that included seismic force. It does NOT include code checks related to connection design like panel zone shear checks or continuity plate checks. If the load combination that controlled the design used the overstrength seismic forces (meaning it had an Ω factor in the load combination) then the governing load combination will be followed by an asterisk (*).

Element Slenderness Checks

This field summarizes whether the member passed the required element slenderness checks based on the **Req'd Ductility** setting. Highly ductile frames are considered to require a "seismically compact" member per the provisions of AISC 341. Whereas moderately ductile frames are considered to require a "compact" member per the requirements of the regular AISC 360 design standard.

Panel Zone Checks and Eqn

Whenever the program detects a valid seismic moment connection between a beam and a column it will check the panel zone of the column per the regular AISC 360 chapter J checks.

For frames that require high ductility (SMF), the required panel zone shear force will be based on the maximum probable moment projected to the face of column per section 9.3a of AISC 341-2005.

For frames which do not require high ductility (IMF and OMF), the panel zone shear demand will be based on the end moments from the solved load combinations. If the Overstrength Req'd flag has been set for the column, then the Ω_o load combinations will be used as well. This is because sections 10.3 and 11.3 of AISC 341-2005 give no additional panel zone requirements beyond those given in the AISC 360 specification.

In addition, the program will check the panel zone for column beam connections per the extra seismic detailing checks of AISC 341-2005 section 9.3b. That is the requirement that the thickness of the column web is greater than the sum of the depth and width of the panel zone divided by 90.

Only the worst case / controlling code equation for the panel zone checks will be reported here. However, detailed information on the other equations are reported in the column detail report.

Note

- The regular panel zone checks are done using the formulas which assume that panel zone deformation is not included in the analysis.
- Currently, the effects of column story shear (which tend to decrease the panel zone shear demand) are not taken into account.
- Per the seismic detailing specification, the extra panel zone thickness checks are only checked for "highly ductile" frames (i.e. Special Moment Frames).
- These conditions are checked for each beam connected to the column with a valid seismic moment connection. However, the total panel zone shear demand is based on the sum of the panel zone shears from the individual beams. This may be overconservative for cases with an IMF or OMF frame where the maximum moments in the beams occur for different load cases.

Continuity Plate

Whenever the program detects a valid seismic moment connection between a beam and a column, it will automatically check the column's capacity to resist the concentrated beam flange forces from the moment connection per section J10.6 of the regular AISC 360 specification. If any of these checks fail, then the program will report that a continuity plate is required at that connection location.

When the user has selected one of the pre-qualified moment connections from AISC 358, then the program will enforce the additional continuity requirements of AISC 358-2005 section 2.4.4 or Chapter 6 (for bolted end plates). Because the AISC 358 pre-qualification is not required for frames with minimal ductility (OMF's) this check will not be enforced in those cases.

Only the worst case / controlling code equation for the continuity plate checks will be reported here. However, detailed information on the other equations are reported in the column detail report.

Note

- The demand moment used to calculate the beam flange forces is based on the Req'd Moment reported on the Beams tab of the Seismic Detailing results.

Strong Column / Weak Beam Ratio and Controlling Beam

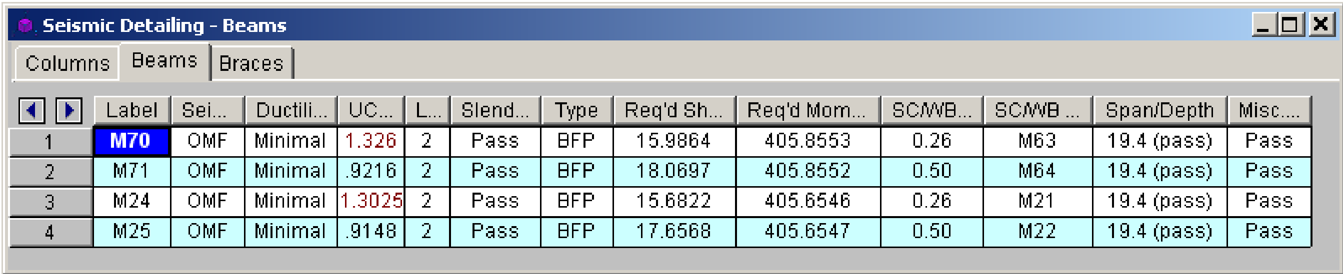
The Strong Column / Weak Beam ratio will be checked for all moment connections between a beam(s) and the column. The worst case (lowest) ratio will be reported here. If the frame ductility requirements dictate a minimum value for this ratio, then the program will also list a pass / fail check in this column. The controlling beam is reported to identify the location on the column which resulted in the lowest SC/WB ratio. If the worst case connection had beams framing in from either side then only one of the beams is reported as only one is required to identify that location.

Miscellaneous Checks

This section relates to a number of miscellaneous checks required by AISC 358. For columns this is normally related to the maximum depth or weight of the column, or whether the member type (tube, wide flange, pipe) is valid with the moment connection assigned to it. Other connection limitations (non-orthogonal / skewed connections) are reported with the beam's Misc Checks.

Seismic Results Spreadsheet - Beams

The second tab of the Seismic Design Rules spreadsheet applies only to Hot Rolled steel members which have been assigned a Member Type of Beam and which have a seismic design rule applied.



Seismic Detailing - Beams

Columns | Beams | Braces

	Label	Sei...	Ductili...	UC...	L...	Slend...	Type	Req'd Sh...	Req'd Mom...	SC/WB...	SC/WB ...	Span/Depth	Misc....
1	M70	OMF	Minimal	1.326	2	Pass	BFP	15.9864	405.8553	0.26	M63	19.4 (pass)	Pass
2	M71	OMF	Minimal	.9216	2	Pass	BFP	18.0697	405.8552	0.50	M64	19.4 (pass)	Pass
3	M24	OMF	Minimal	1.3025	2	Pass	BFP	15.6822	405.6546	0.26	M21	19.4 (pass)	Pass
4	M25	OMF	Minimal	.9148	2	Pass	BFP	17.6568	405.6547	0.50	M22	19.4 (pass)	Pass

Label, Seismic Design Rules and Req'd Ductility

These fields display the member's basic input data for reference purposes.

Connection Type

This field merely lists the type of moment connection that was specified for the member in the Seismic Design Rules applied to the member.

Max Code Check and Load Combination

This field summarizes the member code checks from all the load combinations that included seismic force. It does NOT include code checks related to connection design like panel zone shear checks or continuity plate checks. If the load combination that controlled the design used the overstrength seismic forces (meaning it had an Ω factor in the load combination) then the governing load combination will be followed by an asterisk (*).

Element Slenderness Checks

This field summarizes whether the member passed the required element slenderness checks based on the **Req'd Ductility** setting. Highly ductile frames are considered to require a "seismically compact" member per the provisions of AISC 341, whereas moderately ductile frames are considered to require a "compact" member per the requirements of the regular AISC 360 design standard.

Required Shear

The required shear strength for connection design is listed here. For Highly ductile connections this will be the shear force required to develop a plastic hinge on either side of the beam plus the contribution of shears from the gravity loads. In the member's detail report we refer to these forces as Vpr and Vg.

Connections which require Minimal ductility will be designed for the lesser value of the required hinge shear force (Vpr +Vg) or the Ω shear developed when considering the overstrength load combinations.

For Moderately ductile connections the required shear force for connection design will be based on the connection type. Connections listed with a connection type of "other" will use the same criteria as the minimally ductile connections (based on section 10.2a of AISC 341). The pre-qualified connections (BUEEP, RBS, et cetera) will typically require the same design shear values (Vpr +Vg) as required for the Highly ductile connections.

Note

- The Other connections will assume a value of 1.1 to account for strain hardening (rather than the Cpr factor used in AISC 358). See AISC 341 equation 9-1 for details.
- The Vg values are obtained during a batch or envelope solution by looking at the beam shears at the hinge location for every solved load combination which does not include any wind or seismic loading.
- If there were no gravity-only load combinations solved then Vg is approximated by taking half the difference in the shear force between the two ends of the beam member.

Required Moment

The required Moment strength for connection design is listed here. Currently all moment frame beams connections will require that the connection be designed for the maximum probable moment projected to the face of the column. this is based on an interpretation of section 11.2a of AISC 341-2005. If this level of moment is required for OMF connections, then it should also be required for IMF and SMF.

Note

- This moment is not used to code check the actual beam and column members. It is used, however, to determine if the column will require continuity plate stiffeners to resist the connection forces.
- The Beam Detail Report also lists the Overstrength Moments (Ω_0*Mu) obtained from solving the overstrength load combinations. These are reported for reference only.

Strong Column / Weak Beam Ratio and Controlling Column

The Strong Column / Weak Beam ratio will be checked for all moment connections between a beam(s) and the column. The worst case (lowest) ratio will be reported here. If the frame ductility requirements dictate a minimum value for this ratio, then the program will also list a pass / fail check in this column. The controlling column member is reported to identify which end of the beam resulted in the lowest SC/WB ratio.

Span To Depth Ratio

This field summarizes reports the span to depth ratio for the beam. This value is important because it is used in testing for the AISC 358 pre-qualified connections. Each of the pre-qualified connections will have limits on the minimum span to depth ratio for which that moment connection can be used. When the frame ductility requirements and connection type have limits on this value then the program will also list a pass / fail check in this column.

Note

- The span used in this calculation is the clear span from column flange to column flange as specified in AISC 358.

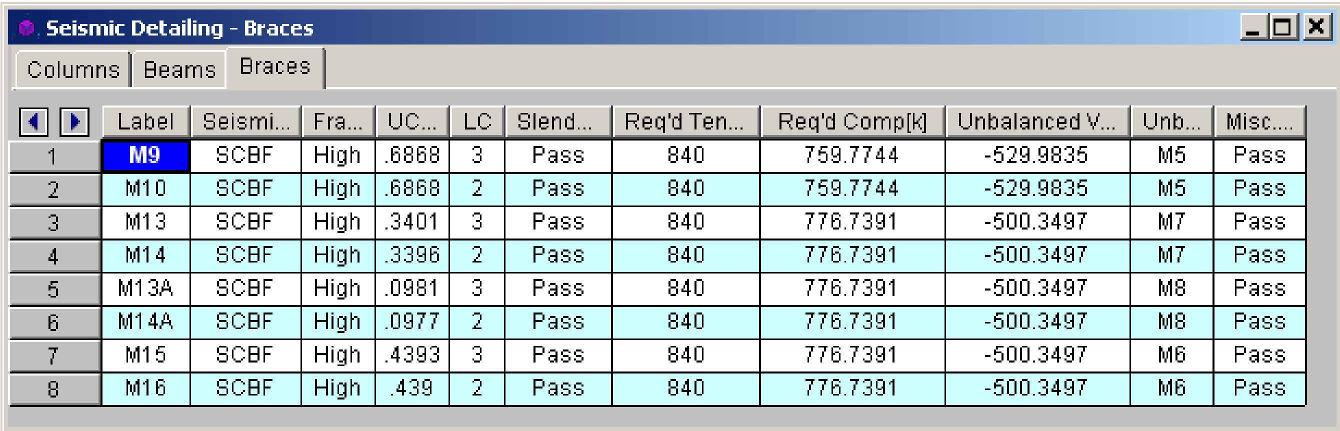
Miscellaneous Checks

This section relates to a number of miscellaneous checks required by AISC 358.

- Geometry Checks on the Beam (i.e. max. Weight, Depth, Flange Thickness, Flange
- Sloped connections: If beam and column webs are in the same plane, the connection need not be orthogonal. But, the program will not perform seismic moment connection calculations when the slope exceeds 15 degrees.
- Strong Axis Connections: The program currently only supports strong axis moment connections for seismic code checking. The tolerance for determining if a connected is skewed enough to be disqualified is 5 degrees.

Seismic Results Spreadsheet - Braces

The third tab of the Seismic Design Rules spreadsheet applies only to Hot Rolled steel members which have been assigned a Member Type VBrace and which have a seismic design rule applied.



	Label	Seismi...	Fra...	UC...	LC	Slend...	Req'd Ten...	Req'd Comp[k]	Unbalanced V...	Unb...	Misc....
1	M9	SCBF	High	.6868	3	Pass	840	759.7744	-529.9835	M5	Pass
2	M10	SCBF	High	.6868	2	Pass	840	759.7744	-529.9835	M5	Pass
3	M13	SCBF	High	.3401	3	Pass	840	776.7391	-500.3497	M7	Pass
4	M14	SCBF	High	.3396	2	Pass	840	776.7391	-500.3497	M7	Pass
5	M13A	SCBF	High	.0981	3	Pass	840	776.7391	-500.3497	M8	Pass
6	M14A	SCBF	High	.0977	2	Pass	840	776.7391	-500.3497	M8	Pass
7	M15	SCBF	High	.4393	3	Pass	840	776.7391	-500.3497	M6	Pass
8	M16	SCBF	High	.439	2	Pass	840	776.7391	-500.3497	M6	Pass

Label, Seismic Design Rules and Req'd Ductility

These fields display the member's basic input data for reference purposes.

Note:

- For OCBF frames, the required frame ductility will be listed as minimal even though the element slenderness requirements will enforce the highly ductile (i.e. seismically compact) limits for the brace.

UC Max and LC

This is the combined bending and axial code check for the brace. This code check is based on non-overstrength load combinations only, unless the Seismic Design Rule for the brace has been designated as Overstrength Req'd. The Load Combination number which resulted in the highest UC value is reported. If the governing load combination contained overstrength seismic forces (Ω_0) it will be followed by an asterisk (*).

Element Slenderness Checks

This field summarizes whether the brace passed the element slenderness checks based on the **Req'd Ductility** setting. Special Concentrically Braced Frames require a "seismically compact" member per the provisions of AISC 341.

Note:

- For OCBF frames (which have minimal required frame ductility) still require that the bracing member's element slenderness meet the highly ductile (i.e. seismically compact) limits of AISC 341.

Required Tension and Compression

This field summarizes the axial forces required for the design of the brace connections. The connection design forces are separated into tension and compression design forces. These forces are determined based on the capacity of the brace itself.

Note

- For frames with Minimal ductility, where Overstrength is required for the brace, the maximum brace axial load from Overstrength (Ω_0) load combinations is reported if it exceeds the capacity-based connection design forces.

Unbalanced Beam Forces

For brace configurations which the program identifies as V or Inverted-V (Chevron) the program automatically calculates the unbalanced beam forces per the provisions of AISC 341-05 Sections 13.4a and 14.3. The resultant unbalanced vertical force on the beam is reported.

Note

- For braces that are non-symmetric the most unbalanced combination of tension/compression is reported.
- The horizontal and vertical components are shown relative to the beam's local axes. (Up) and (Down) refer to the direction of the resultant force on the beam relative to its positive local y axis.
- These unbalanced forces are not actually used in the design of the beam member. They are reported for reference only.
- The beam must have a seismic design rule assigned to it in order to view these forces.
- The requirement that the beam be analyzed as though the brace carried no dead or live load cannot be met by RISA-3D, since the program cannot force certain members to only carry certain components of a load. This can be accomplished by integrating the model with RISAFloor though, which will analyze the beam for dead and live loads as though the brace were not present.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keyword: **Unbalanced**.

Miscellaneous Checks

There are no miscellaneous brace checks in the program at this time. This field has been reserved for future additions, and will always report "Pass".

Seismic Detailing - Detail Reports

The seismic detailing results are presented in two ways: the seismic detailing results spreadsheet and the seismic detailing sections of the member detail reports. The seismic detailing results spreadsheet is intended to be a summary report. More detailed information can be found for each member on that members' detail report as described below. The information described here is based on enveloped information from a batch solution.

Design Forces for Moment Connections

The seismic portion of hot-rolled steel detail reports for beams in moment frames contain sections called **Required Connection Shear Strength** and **Required Connection Moment Strength**.

Required Connection Shear Strength

Column	Vg (k)	Vpr (k)	Ω_o*Vu(k)	Demand
M20	0	116.6421	-9.1642	116.6421
M21	0	116.6421	-9.1642	116.6421

Required Connection Moment Strength

Column	Hinge Loc(in)	Ze(in3)	Mpr(k-ft)	Ω_o*Mu(k-ft)	Demand(k-ft)
M20	-	200	990	-113.4372	990
M21	-	200	990	65.265	0

Required Connection Shear Strength

The Vg listed is the shear at the plastic hinge location in the beam due to gravity loads. RISA determines this value by taking the worst case shear from the load combinations which do NOT include any wind or seismic loads. If there are no solved LC's which meet this criteria, then the value is taken as half the difference in the end shears from the seismic LC's.

RISA uses the term Vpr to refer to the shear required to produce the maximum probable hinge moment (Mpr) on both sides of the beam.. This value will be equal to 2*Mpr / Distance between plastic hinge locations. If the connection is Other / None then the Mpr will be based on a Cpr of 1.1. If the connection is a WUF-W connection, this will be based on a Cpr of 1.4. Otherwise, the calculation will be based on the Cpr of the material as defined in AISC 358-2005 equation 2.4.3-2.

The Ω_o*Vu value listed in the detail report is the shear demand based on the worst case of the Overstrength Load Combinations. This is presented because AISC 341 section 10.2a) says that moderately and minimally ductile frames need not have their shear connection designed to a force greater than this value.

Per AISC 341-2005 Section 9.2, the required shear strength Demand for a connection requiring high ductility will be based on the Vpr value plus the shear due to gravity loading (Vg). Per sections 10.2 and 11.2 of the same document, the required shear strength Demand will be based on the lesser of this shear (Vpr + Vg) or the shear demand from the overstrength load combinations.

Note

- The Vg values are obtained during a batch or envelope solution by looking at the beam shears at the hinge location for every solved load combination which does not include any wind or seismic loading.
- If there were no gravity-only load combinations solved then Vg is approximated by taking half the difference in the shear force between the two ends of the beam member.

Required Connection Moment Strength

The required Moment strength for connection design is listed here along with the values used to compute it.

The **Hinge Location** reported here is the Hinge Location from the assigned Seismic Design Rule. This represents the distance from the face of column to the center of the plastic hinge.

The **Ze** value is the effective plastic section modulus of the hinge. It is equal to the plastic section modulus of the beam multiplied by the Z Factor from the assigned Seismic Design rule.

The **Mpr** value is the probable plastic moment projected to the face of the column. This would be equal to Cpr*Ry*Fy*Ze.

The **Demand Moment** is the Mpr moment projected to the face of the column. The projection of this moment must account for the extra moment due to the shear force at the hinge location times the distance from the hinge location to the face of column. This would be equal to Mpr + (Vpr+Vg)*Hinge Location distance. This value will be used as the demand moment for calculating the beam flange force demand to determine if the column requires continuity plate stiffeners.

The **Ω_o*Mu** is reported mostly for reference. However, it may be used to determine the shear demand on the column's panel zone.

Column Panel Zone Capacity Calculations for Moment Frames

The seismic portion of hot-rolled steel detail reports for columns in moment frames contains a section called **Panel Zone Summary**.

Panel Zone Summary	**(Pass)**			
Beam M24	Demand	LC	Capacity	UC
360-05: Eqn J10-9	Vu = 32.74 k	2	Phi*Vn = 112.5 k	.291
Beam M25	Demand	LC	Capacity	UC
360-05: Eqn J10-9	Vu = 32.74 k	2	Phi*Vn = 112.5 k	.291

For frames that require high ductility (SMF), the Demand Vu is based on the maximum probable moment projected to the face of column per section 9.3a of AISC 341-2005. In addition, the program will check the panel zone for column beam connections per the extra seismic detailing checks of AISC 341-2005 section 9.3b. That is the requirement that the thickness of the column web is greater than the sum of the depth and width of the panel zone divided by 90.

For frames which do not require high ductility (IMF and OMF), the panel zone shear demand will be based on the end moments from the solved load combinations. If the Overstrength Req'd flag has been set for the column, then the Ω_oMu values listed in the section above (Design Forces for Moment Connections) will be used instead.

Note

- Currently, the effects of column story shear (which tend to decrease the panel zone shear demand) are not taken into account.
- These conditions are checked for each beam connected to the column with a valid seismic moment connection. However, the total panel zone shear demand is based on the sum of the panel zone shears from the individual beams. This may be over-conservative for cases with an IMF or OMF frame where the maximum moments in the beams occur for different load cases.

Continuity Plate Checks for Columns in Moment Frames

The seismic portion of hot-rolled steel detail reports for columns in moment frames contains a section called **Continuity Plate Summary**.

Continuity Plate Summary	**Fail (Flange Stiffener Required)**				
Beam M3	Demand	LC	Capacity	UC	Eqn
Flange Bending	Puf = 40.8807 k	1	Phi*Pn = 48.503 k	.843	360-05: Eqn J10-1
Web Yielding	Puf = 40.8807 k	1	Phi*Pn = 73.7317 k	.554	360-05: Eqn J10-3
Web Crippling	Puf = 40.8807 k	1	Phi*Pn = 65.1303 k	.628	360-05: Eqn J10-5a
Flange Thick	tcf > 1.2773 in	N/A	tcf = .72in	1.774	358-05: Eqn 2.4.4-1
Flange Thick	tcf > 1.3783 in	N/A	tcf = .72 in	1.914	358-05: Eqn 2.4.4-2

These checks come from either the AISC 358-05, the AISC 341-05 or the AISC 360-05 depending on the required ductility of the system and the type of moment connection used.

Required Flange Force Calculations

The flange force, P_{uf}, is the demand force that the column has to resist. This force is based on the probable maximum moment at the face of column (see the Demand Moment from the Design Forces for Moment Connections section listed above). This will be used for the AISC 358-05 Continuity Plate Checks.

Note

- When the WUF-W connection is used the program will calculate the probable maximum moment at the face of the column based on a C_{pr} of 1.4. This connection assumes a large degree of strain hardening which artificially increases the probably maximum moment over the other pre-qualified connections with a similar beam and material.
- When an Other / None connection is used, the $C_{pr}$is always assumed to be equal to 1.1.
- When ASD design is used the required Flange Force for connection design is determined by dividing the probably maximum moment at the face of the column by a factor of 1.5.

AISC 341-05 Continuity Plate Checks

Column Flange Thickness Requirement (AISC 341-05: Section 11.5)

For wide flange columns in moment frames which require minimal ductility (OMF's), continuity plates are required if the equations from section 11.5 are not met. For connections which require higher ductility (SMF or IMF) see the AISC 358-05 Continuity Plate Checks listed below.

AISC 358-05 Continuity Plate Checks

Column Flange Thickness Requirement (AISC 358-05: Section 2.4.4)

For wide flange columns with a pre-qualified moment connection (except for Extended End Plates) continuity plates are required if equations 2.4.4-1 and 2.4.4-2 are not met. This check is enforced for only high and intermediate ductility requirements (SMFs and IMF's).

Note

- If the connection type is listed as Other / None then this check is not enforced even for frames with high ductility requirements.

End Plate Moment Connections (BUEEP and BSEEP from AISC 358-05: Section 6.10)

Moment connections requiring high or intermediate ductility (SMFs and IMFs) which use an end plate moment connections require the continuity plate checks described in Section 6.10 of AISC 358-05.

For End Plate Moment Connections (BUEEP and BSEEP) the program will check following concentrated force failure modes to determine if continuity plates are required:

- Column Web Yielding (per AISC 358-05: Equation 6.9-24)
- Column Web Crippling (per AISC 358-05: Equation 6.9-29 through 6.9-31): With the assumption that $N = 2*t_{bf}$
- Column Web Compression Buckling (per AISC 358-05: Equation 6.9-26 and 6.9-27): With the caveat that this check is only performed if the moment connection has beams framing in on both sides of the column.

Note

- These checks are really just variations of the normal AISC 360 continuity plate checks. However, they have been customized to account for the presence of the end plates. For this reason, the BUEEP and BSEEP moment connections do NOT check the AISC 360 Continuity Plate requirements as they would be considered redundant.

- The Column Flange Bending checks of equation 6.9-21 are not performed. These checks require the calculation of the Y_c term which depends on the number of bolts and the geometry of the connection. Thus, it can not be calculated and an N/A is presented instead.
- The bolted end plate moment connection equations from AISC 358 are only formulated for LRFD design. Therefore, in cases where ϕ equals 0.75, an Ω value of 2.0 will be assumed. Similarly, in cases where ϕ equals 0.9, an Ω value of 1.67 will be used.

AISC 360-05 Continuity Plate Checks

For SMFs, IMFs and OMFs that are not using BUEEP and BSEEP connections AISC 341-05 requires that the connections still meet the requirements of AISC 360-05 Section J10. The program will check the following concentrated force failure modes to determine if continuity plates are required:

- Column Flange Bending (per AISC 360-05: Equation J10-1.)
- Column Web Yielding (per AISC 360-05: Equation J10-2 and J10-3.) With the assumption that $N = t_{bf}$
- Column Web Crippling (per AISC 360-05: Equation J10-4, J10-5a and J10-5b.) With the assumption that $N = t_{bf}$
- Column Web Compression Buckling (per AISC 360-05: Equation J10-8.) With the caveat that this check is only performed if the moment connection has beams framing in on both sides of the column.

Multiple Moment Connections Framing to a Column

When a column with seismic design rules has a beam framing in from each side, each beam will show up separately in the detail report with its own continuity plate checks. Since these checks are local to the beam flange to column flange force transfer, the program does not assume any interaction between the continuity plate checks of the two beams.

Strong Column / Weak Beam (SC/WB) Moment Ratios

The seismic portion of hot-rolled steel detail reports for columns in moment frames contains a section displaying the **Column Beam Moment Ratio** which is frequently referred to as the Strong Column / Weak Beam Moment ratio.

Column Beam Moment Ratio (For Reference Only)

Beam	Sum M*pc (k-ft)	Sum M*pb (k-ft)	Ratio	Eqn
M24	213.3333	1669.5413	.1278	358-05 5.4(2a)
M25	213.3333	1669.5413	.1278	358-05 5.4(2a)

This value is reported for reference only for frames that require Minimal or Moderate Ductility (OMF's and IMF's).

For frames that require High Ductility (SMF's), any value less than 1.0 is considered to fail the SC/WB code check per AISC 341-2005 Section 9.6. These frames report a warning that connection bracing is required anytime the SC/WB ratio is calculated as less that 2.0 per AISC 341-2005 Section 9.7.

Note

- The Sum M^*_{pb}value is based on the maximum probable moment at the plastic hinge location projected to the centerline of the column.
- For two sided moment connections, this Sum M^*_{pb}value assumes that the shear force from gravity loads (Vg) adds to the plastic hinging shear force (Vpr) on one side of the connection and takes away from it on the other side.

Bracing Requirements for Beams in a Moment Frame

The seismic portion of hot-rolled steel detail reports for beams in moment frames contains a section called **Beam Flange Bracing**. This section reports the required beam flange bracing forces for moment connection. These values depend on ductility requirements of the connection.

Beam Flange Bracing		
Reqd Strength	9.4664 k	360-05: Eqn A-6-7
Reqd Strength	28.3993 k	341-05: 9.8 (adj. to hinge)
Reqd Stiffness	24.7228 k/in	360-05: Eqn A-6-8
Max Spacing	95.7259 in	341-05: 9.8

For connections which require high or moderate ductility (SMF, or IMF), the L_b values used in Equation A-6-8 are based on Maximum Spacing values shown in the detail report. These maximum spacing for beam bracing is based on the requirements of AISC 341 sections 9.8 and 10.8.

For connections which require minimal ductility (OMF), L_b values used in Equation A-6-8 are based on the maximum of Lcomp-top, Lcomp-bottom or L_p as defined in AISC 360-05 equation F2-5.

Note

- The C_d values in equations A-6-7 and A-6-8 are always assumed to be 1.0 based on the commentary to AISC 341 section 9.8.
- For RBS connections, the M_r values used in equations A-6-7 and A-6-8 are assumed to be equal to the plastic section modulus at the reduced beam section.

Requirements for Braced Frames

In addition to the input echo and the Element Slenderness checks, braced frames display the following sections in the member detail report. These sections are displayed for the Brace Member and (in the case of V or inverted V braced systems) the beam member attached to the V bracing.

Brace Connection Strength

Required Connection Axial Strength					
Comp.(1.1*Ry*Pn/1.5)	474.835 k	Tens.(Ry*Fy*Ag/1.5)	594 k	Ω_o*Pu	336.739 k

The three brace connection design forces are all presented for reference, although the actual design of the connection is outside of the scope of RISA-3D. The first two are determined based on the capacity of the brace itself. The third value is based on the maximum force in the brace from the solved load combinations. If the brace has been specified to be designed for Overstrength (Ω_0) then this value will be based on the Overstrength load combinations.

Unbalanced Forces

For brace configurations which the program identifies as V or Inverted-V (Chevron) the program automatically calculates the unbalanced beam forces per the provisions of AISC 341-05 Sections 13.4a and 14.3. The resultant unbalanced forces on the beam are reported.

Unbalanced Force on Beam (unfactored)	--not considered in beam code check--		
Beam	M5		
Brace 1	M10	129.507 k	341-05: 13.4a(1)(b)
Brace 2	M9	840 k	341-05: 13.4a(1)(a)
Horizontal Component		473.1995 k	
Vertical Component		529.9835 k	(Down)

For more information see the Seismic Detailing Results topic; For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keyword: **Unbalanced**.

Brace KL/r Restriction

KL/r = **61.8338 <= 200 Pass per 360-05 E2**

The program will report the KL/r check for certain brace members based on the user input setting for the Seismic Design Rules. Examples where a user would likely select this input would be braces that are part of a Special Concentrically Braced Frame (SCBF) as well as K, V, or inverted V braces in an Ordinary Concentrically Braced Frame (OCBF).

Miscellaneous Seismic Checks

Element Compactness Checks

This section of the detail report displays the results of the element slenderness checks per the provisions of AISC 341 and AISC 360.

Bending Flange	**Compact**	Compression Flange	**Slender**
Bending Web	**Seismically Compact**	Compression Web	**Seismically Compact**

- AISC 341 requires (based on the frame ductility and member type) that the flanges and webs of the cross section satisfy certain compactness or element slenderness criteria. Members whose flanges or webs do not meet the compactness criteria will report their compactness classification (Seismically Compact, Compact, NonCompact or Slender) using red text to indicate that the member has failed the AISC 341 criteria.

Miscellaneous AISC 358 Pre-Qualification Checks

This section of the detail report displays the results of some of the miscellaneous provisions of AISC 358. Specifically those sections related to pre-qualification of the beam-column moment connection. Some of these requirements are related to the beam or column dimensions (size, weight, et cetera). Some of them are related to the clear span to beam depth ratio.

Col-BM Moment Ratio	**.3107 (For Reference Only)**
Span to Depth Ratio	**15.0376 (Pass)**

Note:

- The program is not checking any of the AISC 358 weld restrictions.

Other Miscellaneous Checks

The program produces warning messages (in the detail report only) relating to the vaious AISC 341 restrictions. Among them are the provisions which require columns with a L/r ratio greater than 60 to be braced out of plane as shown below.

L/r = **132.2713> 60 per 341-05 9.7b(2) (For Reference Only)**

Solid Elements

The Solid or 8 node brick element allows you to model structures that are too thick to be modeled by plate elements. Common applications could include dam models, extremely thick pile caps or vibrating equipment with extremely thick support slabs. Essentially, these elements should be used whenever the Mindlin – Reissner assumption of linear strain through the thickness of the element would not be appropriate.

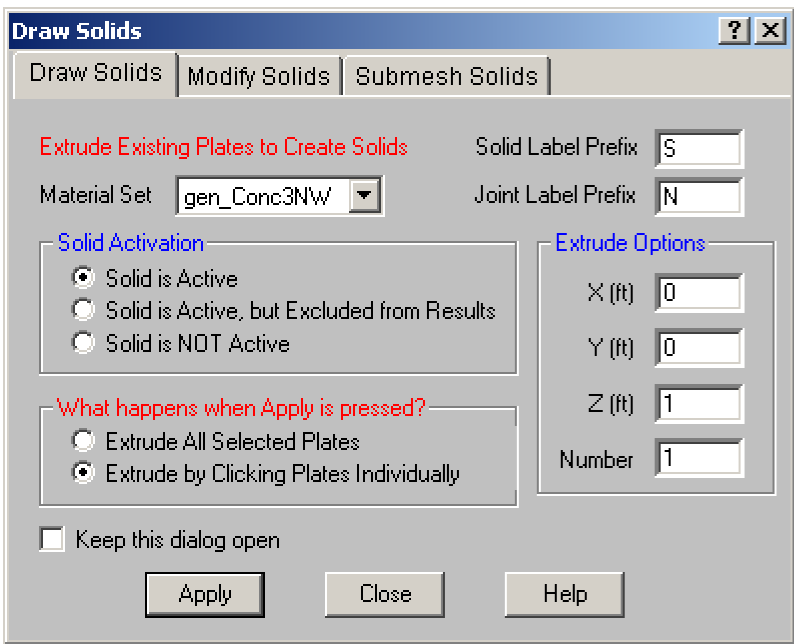
Solid elements may be viewed / edited in a couple of ways, but can only be *created* by extruding existing plate elements.

Creating Solids

Solid Elements are not drawn in RISA-3D. Instead, they must be extruded from a mesh of plate elements. Use the Insert menus or the Drawing Toolbar to create new solids. Once you have created these items, you may use other graphic features to load the model and set boundary conditions.

Creating solid models requires more forethought than either beam or plate elements. To create solids you must first create a plate element mesh and then extrude that mesh into a series of solid elements. See Plate Modeling Tips and Plate Modeling Examples for tip on building plate element meshes. You can set all the element properties up front or you can modify these properties after you draw them. Modifying Properties is discussed in the next section.

The Extrude Plates into Solids dialog is shown below. This lets you take any existing plate and extrude it out into a three dimensional object.



To Extrude Solids:

1. If there is not a model view already open, then click on the RISA Toolbar to open a new view and click to turn on the Drawing Toolbar if it is not already displayed.
2. If you do not already have a plate element mesh, then you must create one. See Drawing Plates for more information on this.
3. Click the Draw/ Extrude Solids button ad select the Draw Solids tab. The set the solid properties. For help on an item, click and then click the item.
4. Enter the extruding options. The X, Y, and Z components of the extruding vector, the thickness of the extrusion, and the number of elements along the extrusion vector.

5. You may choose to extrude a single solid at a time or you may choose to extrude an entire selection of plates. To extrude only a few plates choose Extrude by Clicking Plates Individually and click Apply. Click on plates with the left mouse button. To extrude a selection of plates, choose Extrude All Selected Plates and click Apply.

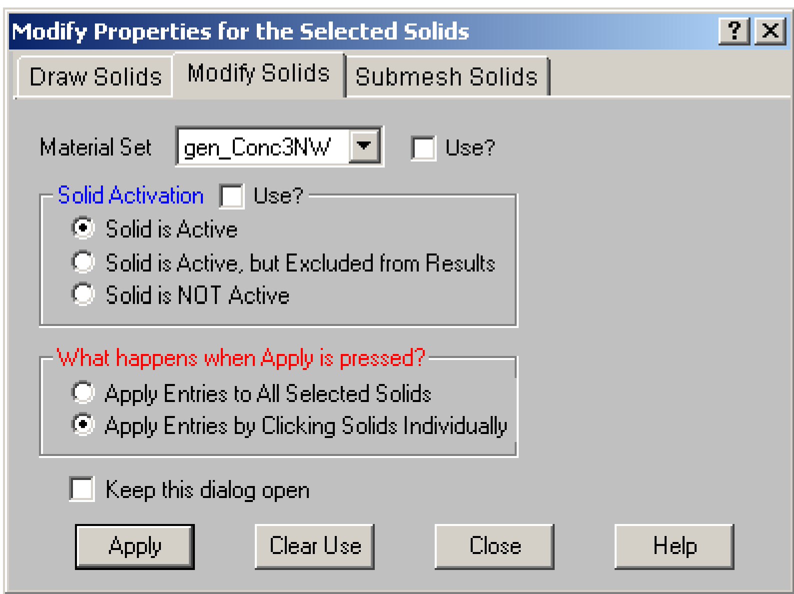
Note

- Only Quadrilateral plates can be extruded.
- To extrude more solids with different properties, press CTRL-D to recall the Draw / Extrude Solids dialog.
- You may also view and edit solid properties by double-clicking on a solid.
- You may undo any mistakes by clicking the Undo button.

Modifying Solids

There are a number of ways to modify solids. You may view and edit the element data in the Solids spreadsheet. You may double-click a solid to view and edit its properties. You can use the Modify Solids tool to graphically modify a possibly large selection of elements.

The graphical Solids Modify tool discussed here lets you modify the properties of solids that already exist in your model. You can modify solids one at a time by selecting the Click to Apply option and then click on the plates you wish to modify. You may also modify entire selections of solids by selecting the solids first and then use the Apply to Selected option. See the Selection topic for more on selecting.

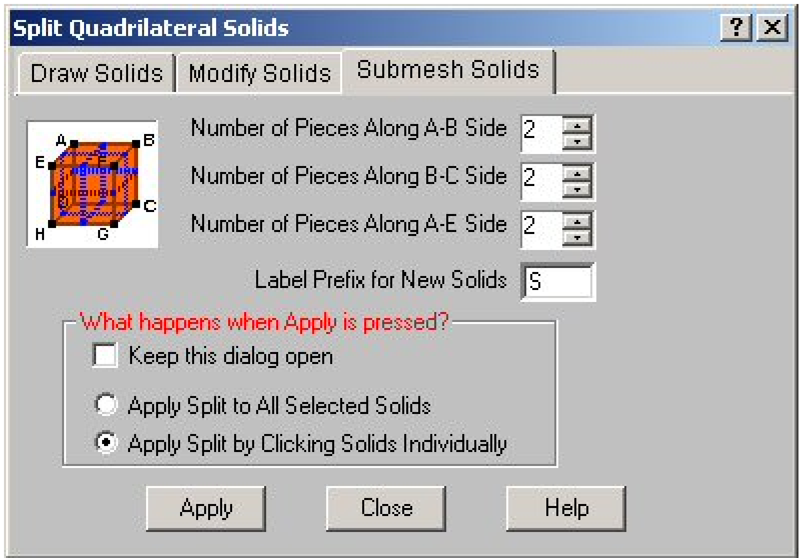


The parameters shown are the same as those used to extrude new solids. For help on an item, click and then click the item.

The **Use?** check boxes next to the data fields indicate whether the particular parameter will be used or not when the modification is applied. If the box next to a field is checked, that parameter will be applied to any selected plates. If the box is NOT checked, the parameter will NOT be applied, even if a value is entered in the field. This lets you easily change one or two properties on members without affecting all the rest of the properties.

Sub-Meshing Solids

You can submesh solid plate elements into a mesh of smaller elements. This new mesh can be any size up to the program limits for joints and/or solids. This is very useful for refining a coarse mesh of elements, just make sure that all adjacent solid elements (elements sharing an edge) maintain connectivity.



You can define different submesh increments in each direction. The A,B,C,D, and E joints for each plate are displayed in the solid info dialog (the double-click dialog).

You can submesh the solids one at a time by selecting the **Click to Apply** option and then clicking on the solids you wish to submesh. You may also modify entire selections of solids by selecting the plates and then using the **Apply to Selected** option.

To Submesh Solid Elements

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Select the Solids you want to sub mesh.
3. Click the **Create / Modify Solid** button and select the **Submesh Quads** tab. Then specify the number of plates.

Note

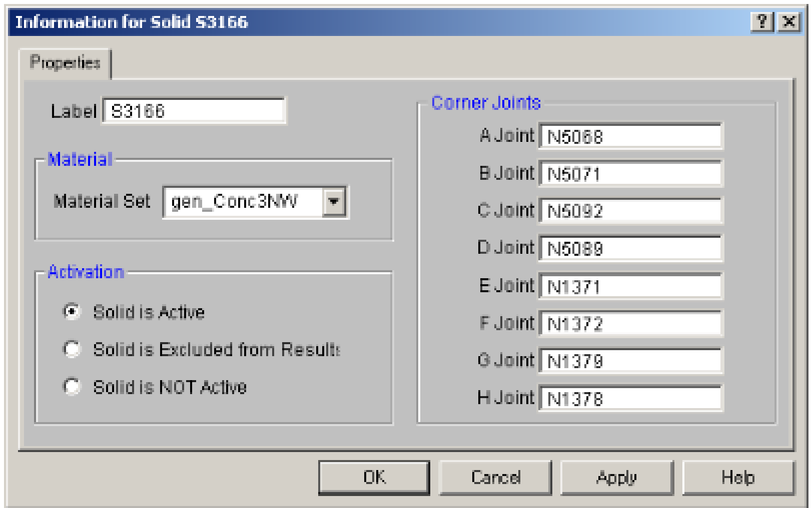
- To submesh more solids with different parameters, press CTRL-D to recall the **Submesh Solids** settings.
- You may undo any mistakes by clicking the **Undo** button.

Solids Spreadsheet

Currently, there is not a Solid Elements spreadsheet. The solid element is very sensitive to the order in which the nodes are defined. Editing these nodes via a spreadsheet is not generally a good idea. The only other information that would have been contained in the spreadsheet are the Element Activation and the Element Material.

Solid Information Dialog

Just as with the joints, members, and plates you may double-click any solid to view it's properties. All of the same information that is stored in the Solids spreadsheet is displayed for the solid you choose, and may be edited. This is a quick way to view and change solid properties. For large selections of solids however the spreadsheet and graphic editing tools may be the faster solution.



Solid Labels

You must assign a unique label to all of the solids. You can then refer to the solid by its label. Each label has to be unique, so if you try to enter the same label more than once you will get an error message. You may relabel solids at any time with the Relabel Solids option on the Tools menu.

Corner Joints

The A, B, C, D, E, F, G, and H joint entries are used to define the 8 corner joints of a hexahedral element.

Solid Material

The material set label links the solid with the desired material defined on the Material spreadsheet.

Activation

The activation state of the element may be changed. If the solid is made inactive, you will need to activate the solid from the solids spreadsheet, or by using the Criteria Select feature to find and select inactive solids.

Note

- If you have a plate that has a side of a length that is less than the merge tolerance that you specify in the Global parameters under the Solutions tab, then we will not extrude a solid element from that plate.
- If you come across this as being a problem, then your plates are probably not well meshed and you should think about cleaning up your mesh.

Inactive and Excluded Solids

Making an item such as a member or solid inactive allows you to analyze the structure without the item, without having to delete the information that defines it. This leaves data intact so the item may be easily reactivated. This is handy if you want to try a model with and then without certain items, without having to actually delete the data.

Putting a “y” in the Inactive field makes the item inactive, i.e. the item is not included when the model is solved or plotted.

Another option is to put an “E”. The “E” code means include the item in the solution, but exclude it from the results list. So, an item with an “E” in the “Inactive?” field will be treated like any other solid in the solution and plotting of the model, but the solid will not be listed in the solution results (forces, stresses etc.). This is useful if there are certain items whose results you're not interested in. You don't have to clutter up the results with these items and can concentrate on the items you're most interested in. See Printing for more limiting printed results.

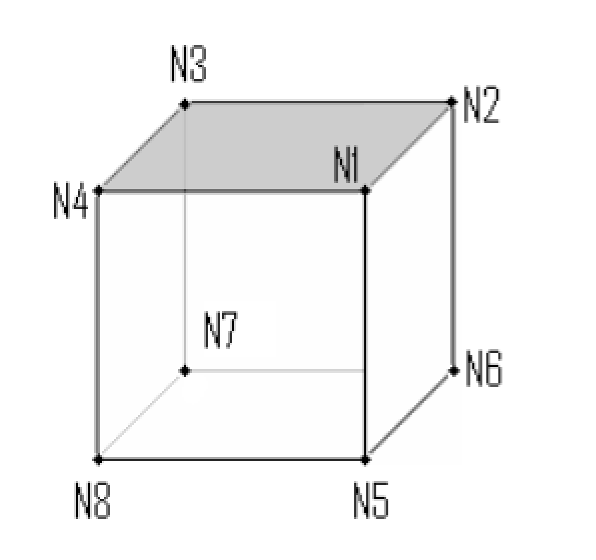
Solids Formulation

A reference for this element is Finite Element Procedures, by K.J. Bathe, Prentice-Hall, 1996. Although the book does not complete the element derivations, it does provide many references for papers on the family of elements. In brief, the element formulation is standard 8-node isoparametric formulation.

Joint Connectivity

The joint connectivity in our current solid formulation follows the “left hand rule”. It means if the first 4 nodes are ordered *counter*clockwise in plane (as shown in the picture below), then node 5 has to be below node 1. However, if the first 4 nodes are ordered *clockwise* in plane, then N5 would have to be above node 1.

The joint connectivity for solid element is listed in the following picture.



If the solid element is generated using the extrusion tool, the joints will automatically be generated in the proper order. However, when user is modifying the location the joints, it is important to keep the order the joints to be consistent with the rules stated above. If not, then the local element matrix will be singular and an error message will be produced at solution time.

Degrees of Freedom

The Solid element activates the THREE translational degrees of freedom at each of its connected joints. Rotational degrees of freedom are NOT activated. This element contributes stiffness to all of these translational degrees of freedom. If a rotational load or constraint is applied to a joint that is only connected to the solid elements, it will be ignored. Modeling these types of rotations would be similar to modeling the "Drilling Degree of Freedom" for plates. Refer to the Modeling Tips section of the general reference manual for more information.

Coordinate System

For the time being, no local coordinate system is defined for the solid element. All the input and output, such as material properties, stresses, displacements are all defined in the global coordinate systems.

Solid Modeling Tips

Number of Elements

The standard iso-parametric formulation needs at least 4 elements through the thickness in order to accurately simulate a bending dominant part (such as a thin beam or thin plate).

Aspect Ratio of Elements

Solid Elements are more sensitive to element distortion than plate elements. For this reason, it is a good idea to keep a solid elements relatively un-distorted. The best formulation for a solid is a cube with equal length sides.

Note

- Solids are always defined with general materials. This is because the other material sets (Hot Rolled, Cold Formed, Wood, and Concrete) are used to designate member code checking specifications. Since solids are only used for analysis, no code checking is provided and the material must be designated as a general material.
- It's generally more efficient to use the Graphic Editing features if you want to change the properties for many solids at once.

Loading

For the time being, only joint loads and self weight can be applied to the solid elements.

Verification Examples

Solid Verification Problem: Solid_Cantilever.R3D

In this example, a straight cantilever beam (modeled with solid elements) is subjected to a unit force at the tip in the three orthogonal direction and the unit moments at the tip about the three orthogonal directions, each in a different load case. The tip displacements are compared with hand calculations as shown below:

Case	Equation	Theory	RISA
Axial Extension	Delta = PL/AE	0.003"	0.003"
SA Bending	Delta = $PL^3/3EI$	10.8"	10.6"
WA Bending*	Delta = $PL^3/3EI$	43.2"	40.7"

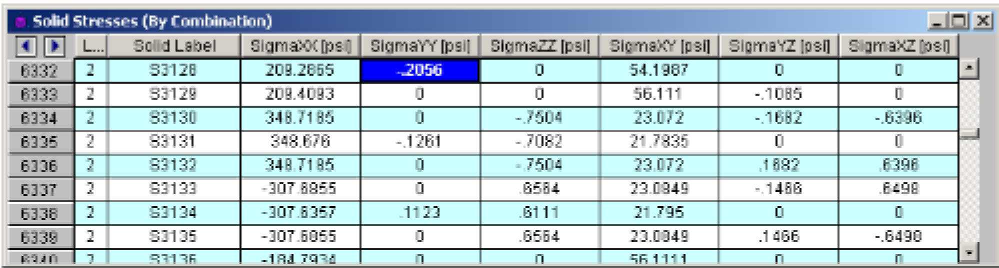
*The weak axis bending results for this example are affected by the fact that only three elements were used through the weak direction in the model. Per the modeling tips, this violates the minimum # of elements capable of modeling bending behavior.

Solid Elements - Results

When the model is solved, there are two types of results specifically for solid elements: Solid Stresses, and Solid Principal Stresses.

Solid (Global) Stress Results

Access the **Solid Stresses Results** by selecting the **Results** menu and then selecting **Solids** ▸ **Solid Stresses**.


Solid Stresses (By Combination)

	L...	Solid Label	SigmaXX [psi]	SigmaYY [psi]	SigmaZZ [psi]	SigmaXY [psi]	SigmaYZ [psi]	SigmaXZ [psi]
6332	2	S3128	209.2865	-.2056	0	54.1987	0	0
6333	2	S3129	209.4093	0	0	56.111	-.1085	0
6334	2	S3130	348.7185	0	-.7504	23.072	-.1682	-.6396
6335	2	S3131	348.676	-.1261	-.7082	21.7835	0	0
6336	2	S3132	348.7185	0	-.7504	23.072	.1682	.6396
6337	2	S3133	-307.8855	0	.6564	23.0849	-.1466	.6498
6338	2	S3134	-307.8357	.1123	.6111	21.795	0	0
6339	2	S3135	-307.8855	0	.6564	23.0849	.1466	-.6498
6340	2	S3136	-184.7934	0	0	56.1111	0	0

The solid element stresses in the global directions are evaluated at the standard 2 x 2 x 2 Gauss integration points of the element and extrapolated/interpolated to the joints and the center of the element for graphical display of the stress contours. The Solid Stresses spreadsheet, however, only displays the center stress value for each element. These stresses are always reported with respect to the global X, Y and Z axes of the model.

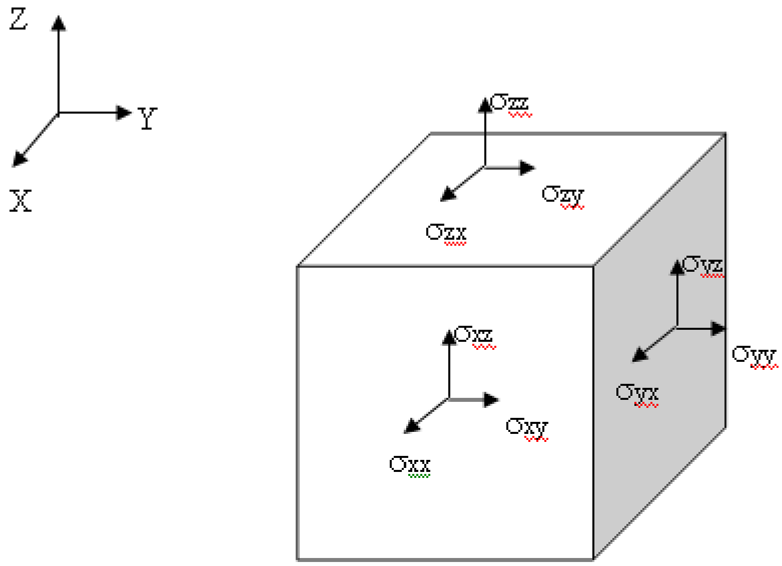
For enveloped results the maximum and minimum value at each location is listed. The load combination producing the maximum or minimum is also listed, in the "lc" column.

Sign Convention for Stresses

For solid element stresses, RISA uses a sign convention where tension is represented by a positive value of Sigma XX, Sigma YY, or Sigma ZZ.

Sign convention for the other stresses is as shown in the image below.

- The first subscript represents the plane on which the stress is acting. In the case of Sigma YZ, the plane is perpendicular to the Global Y plane.
- The second subscript represents the direction of the stress. In the case of SigmaYZ, a positive stress has an orientation in the positive Z direction.
- Since RISA's implementation uses only isotropic material, Sigma YZ = Sigma ZY, Sigma XY = Sigma YX, and Sigma XZ = Sigma ZX.



Note

- See [Spreadsheet Operations] to learn how to use **Find**, **Sort** and other options.
- See [Plot Options – Solids] to learn how to plot solids results.

Solid Principal Stress Results

Principal values and their associated principal directions are also computed. Access the **Solid Principal Stresses Results** by selecting the **Results** menu and then selecting **Solids** ▸ **Solid Principal**.

Solid Principal Stresses (By Combination)

	L...	Soli...	Sigma1 ...	Sigma2 [p...	Sigma3 [...	Von Mise...	Angle1X	Angle1Y	Angle1Z	Angle2X	Angle2Y	Angle2Z	Angle3X	Angle3Y	Angle3Z
5855	2	S2651	4.2414	.2817	-852.8712	855.1296	1.5074	3.0782	1.5708	1.5708	1.5708	0	3.0782	1.5074	1.5708
5856	2	S2652	3.3439	-.2584	-853.6199	855.1684	1.5056	.1396	1.6941	1.5627	1.4478	.1233	.0657	1.6365	1.5708
5857	2	S2653	17.4895	-.0032	-300.7682	309.8828	1.3342	.237	1.5573	1.5875	1.5875	3.1389	.2366	1.8074	1.5708
5858	2	S2654	16.7766	-.2321	-299.8981	308.5222	1.3396	2.9104	1.5708	1.5708	1.5708	0	2.9104	1.3396	1.5708
5859	2	S2655	17.4895	-.0032	-300.7682	309.8828	1.3342	.237	1.5843	1.5875	1.5875	.0047	.2366	1.8074	1.571
5860	2	S2656	300.7681	.0032	-17.4915	309.8828	.2366	1.3342	1.5708	1.5875	1.5875	3.1389	1.3342	2.9046	1.5573
5861	2	S2657	299.8981	.2321	-16.7766	308.5222	.2312	1.802	1.5708	1.5708	1.5708	0	1.802	.2312	1.5708

Principal Stresses

Sigma1, Sigma2, and Sigma3 represent the principal stresses for the element. The principal stresses are the three eigenvalues of the 3 by 3 stress matrix.

$$\begin{bmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{bmatrix}$$

Orientation of Principal Stresses

The direction of the Sigma1 principal stress is defined by Angle1X, Angle1Y, and Angle1Z. Where each of the angles defines the angle between that global axis and the principal stress direction. The same definitions hold true for the Sigma2 and Sigma3 stresses.

Von Mises Stress

The Von Mises stress is a combination of the principal stresses and represents the maximum energy of distortion within the element. This stress can be compared to the tensile yield stress of ductile materials for design purposes. For example, if a steel plate has a tensile yield stress of 36 ksi, then a Von Mises stress of 36 ksi or higher would indicate yielding of the material at some point in the plate.

$$\sigma_v = \sqrt{\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_1 - \sigma_3)^2}{2}}$$

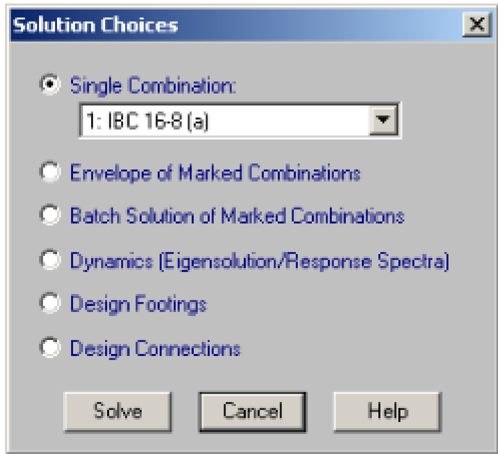
For enveloped results the maximum and minimum value at each location is listed. The load combination producing the maximum or minimum is also listed, in the "lc" column.

Note

- See Spreadsheet Operations to learn how to use **Find**, **Sort** and other options.
- See Plot Options – Solids to learn how to plot solids results.

Solution

Four solution options are presented when you click **Solve** on the menu. Some of the options require other solution types to be performed. See Results for information on solution results. For additional information on Dynamic Analysis and Response Spectra Analysis results refer directly to those sections. For footing design within RISA-3D or RISA-2D see the Footing Design topic. For connection design within RISA-3D see the RISAConnection Integration topic.



What do you want to do?

- Perform a dynamic analysis.
- Perform a response spectra analysis.
- Perform a Static Solution

What do you want to know?

- What is an Envelope Solution?
- What is an Batch Solution?

Static Solutions

Static solutions are based on load combinations and may be performed on any defined load combination. When a static solution has been performed and the results are available the "S" icon on the status bar in the lower left corner will change from to . If any changes are made to the model that invalidate the results then the results will be cleared and another solution will be necessary.

The solution is based on the widely accepted linear elastic stiffness method for solution of the model. The stiffness of each element of the structure is calculated independently. These element stiffnesses are then combined to produce the model's overall (global) stiffness matrix. This global matrix is then solved versus the applied loads to calculate joint deflections. These joint deflections are then used to calculate the individual element stresses. The primary reference for the procedures used is **Finite Element Procedures**, by K. J. Bathe (Prentice-Hall, 1996).

Skyline Solver

This solution method is also sometimes referred to as an Active Column solution method. In finite element analysis, the nonzero terms of the stiffness matrix are always clustered around the main diagonal of the stiffness matrix. Therefore, the Skyline or Active Column solutions take advantage of this by condensing the stiffness matrix to exclude any zero stiffness terms that exist beyond the last non-zero term in that column of the matrix.

Since the majority of terms in a stiffness matrix are zero stiffness terms, this method greatly reduces the storage requirements needed to store the full stiffness matrix. However, for large models (+10,000 joints) , the memory requirements even for a skyline solution can be problematic.

This solution method has been used successfully in RISA for more than 20 years, and has proven its accuracy continuously during that time.

Sparse Solver

The skyline solver described above is moderately efficient because it only stores and performs operations on the terms within the "skyline" of the stiffness matrix. However, that solution still contains a great number of zero stiffness terms within the skyline of the matrix. A Sparse Solver will reduce the matrix size to an absolute minimum by eliminating the storage of ALL zero stiffness terms.

A sparse solver is the most efficient solution methodology possible because it stores and performs operation only on the non-zero terms of the stiffness matrix. For this reason the sparse solution is preferred from both a solution speed and memory requirement standpoint.

Note

- The skyline solver is retained mostly for comparison and verification purposes.

Single Combination Solutions

Choose this option to solve one load combination by itself.

Envelope Solutions

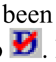
Static solutions may also be performed on multiple combinations and the results enveloped to show only the minimum and maximum results. Each of the results spreadsheets will contain minimum and maximum values for each result and also the corresponding load combination. The member detail report and deflected shape plots are not available for envelope solutions. See Load Combinations to learn how to mark combinations for an envelope solution.

Batch Solutions

Static solutions may be performed on multiple combinations and the results retained for **each** solution. You may group the results by item or by load combination by choosing from the **Results Presentation** options on the **Results Menu**. For example you can have all the combination results for member M1 together or you can have all the member results for Load Combination "D+L" together. See Load Combinations to learn how to mark combinations for a batch solution.

Dynamic Solutions

Dynamic analysis also requires a load combination but this combination is merely used to determine the mass of the model. See Dynamic (Modal) Analysis for much more information.

When a dynamic solution has been performed and the results are available the "D" icon on the status bar in the lower left corner will change from  to . If any changes are made to the mode that invalidates the results then the results will be cleared and another solution is necessary.

Response Spectra Solutions

When a response spectra solution has been performed and the results are available the "D" icon on the status bar in the lower left corner will change from  to . If any changes are made to the mode that invalidates the results then the results will be cleared and another solution is necessary.

Spreadsheet Operations

Powerful spreadsheets may be used to view, sort and edit the input data and results. The spreadsheets and model views are always in tune, unless you turn this feature off in the **Preferences** on the **Tools Menu**. As you edit a model graphically the spreadsheets are automatically updated and as you make changes in the spreadsheets the model views reflect these changes immediately.

The input data may be accessed from the **Spreadsheets Menu**. You may edit the data or you can add new data. You can also paste data from another application directly into the spreadsheet via the Windows clipboard. Any changes made to the input spreadsheets may also be viewed graphically.

After solving the model, results are recorded in spreadsheets for browsing. These spreadsheets may be accessed from the **Results Menu**. You may sort the results in order to find maximums and exclude data that is not important. You may also copy this data to the Windows clipboard and use it in another application.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Advanced Spreadsheet**.

Moving and Scrolling

To move between cells in a spreadsheet, click any cell or use the keyboard arrow keys. When you move to a cell it becomes the active cell. To see a different area of the spreadsheet use the vertical scroll bars on the right side of the spreadsheet.

To Scroll	Do This
One row up or down	Click the arrows on the vertical scroll bar.
One column left or right	Click the arrows on the horizontal scroll bar.
One page up or down	Click above or below the scroll box in the vertical scroll bar.
One page left or right	Click to the left or right of the scroll box in the horizontal scroll bar.
A large distance	Drag the scroll box to the approximate relative position.

Tip

- The size of a scroll box indicates the proportional amount of the spreadsheet that is visible. The position of a scroll box indicates the current location relative to the spreadsheet.
- The mouse wheel is also an excellent tool for scrolling up and down in spreadsheets or graphics. Roll the wheel or click and drag to move around.

Spreadsheet Keyboard Commands

The following keyboard commands are available:

Key	Function
Arrow Keys	Move the active cell one location
TAB	Move right one cell
ENTER	Move to the first column of the next line.
	Adds new line if necessary.
PAGE UP	Move the active cell one full page up.

PAGE DOWN	Move the active cell one full page down.
HOME	Move to the first line of the spreadsheet.
END	Move to the last line of the spreadsheet.
F3	Insert new line below current line.
F4	Delete current line.
F8	Insert new line below current line and repeat the current values in the new line.

Selecting Spreadsheet Cells

Before you can carry out commands or tasks in a spreadsheet, you must select the cells that you want to work with.

To Select	Do This
A single cell	Click the cell, or press the arrow keys to move to the cell.
A range of cells	Click the first cell of the range, and then drag to the last cell.
An entire row	Click the row heading.
An entire column	Click the column heading.
Adjacent rows or columns	Drag across the row or column headings

Note

- To cancel a selection of cells, click any single cell in the spreadsheet.

Undoing Operations

RISA-3D provides you with virtually unlimited **Undo** capability so that you may easily correct mistakes or just back up to try different possibilities. Simply click the  button as many times as you wish to go back a step. The model view and the spreadsheets will visually display the "undoing". Remember that graphic edits are undone as well.

Redoing Operations

RISA-3D provides you with virtually unlimited **Redo** capability so that you may reapply actions that were previously undone. Simply click the  button as many times as the Undo button was used just prior. The model view and the spreadsheets will visually display the "redoing". Remember that graphic edits are redone as well.

Editing Spreadsheets

The spreadsheets have been specifically developed for the input and editing of structural models. There are many ways to edit the spreadsheets allowing you to quickly build your model. You may copy and move data from other locations or other files. You may also fill large blocks of cells automatically and perform math on these cells.

To Change Basic Load Cases in the Spreadsheets

- Use the drop down load list 

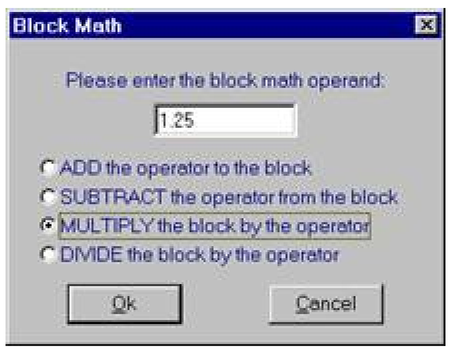
on the window toolbar to control the basic load case in the spreadsheet.

To Fill Selected Cells

1. Select the cells you wish to fill.
2. Click the **Block Fill** button.
3. Type the value you wish to apply.
4. Click **OK**.

To Perform Math on Selected Cells

1. Select the desired cells.
2. Click the **Block Math** button.
3. Specify the value and the operation you wish to apply.
4. Click **OK**.



To Cancel or Undo Editing

- To cancel an entry before you press ENTER press ESC.
- To undo a completed entry, click the **Undo** button.

To Edit Cell Contents

1. Click the cell that contains the data you want to edit.
2. Make any changes to the cell contents.
3. To enter your changes, press ENTER.
4. To cancel your changes, press ESC.

Note

- To replace the contents of a cell, click on the cell and type the new entry.
- To edit the contents of a cell, double-click on the cell and use the arrow keys to locate the cursor and perform the edit.

Inserting, Deleting and Clearing Cells

To insert cells you must insert an entire row. When you delete cells you must delete an entire row. The row is removed from the worksheet and shifts the lower cells to fill the space. When you clear cells, you remove the cell contents but leave the blank cells on the spreadsheet.

To Insert Rows

1. Click a cell in the row immediately above where you want the new row.
2. To insert a blank line, click the **Insert New Line** button.
3. To have the values of the current line copied in the new line, click the **Repeat Current Line** button.

To Clear Cell Contents

1. Select the cells, rows, or columns you want to clear and press the DELETE key.

To Delete Rows of Cells

1. Select the rows you want to delete.
2. Click the **Delete Marked Lines** button. Lower cells shift up to fill the space.

Note

- Some spreadsheets do not allow you to delete lines. For example the Member Design spreadsheet has one line for each member defined. You may leave these lines blank but may not delete them.

Moving and Copying Cell Contents

Standard Windows cut, copy and paste functions are fully supported. By using the clipboard you may move or copy any spreadsheet data to another location in the same spreadsheet, or to another spreadsheet (provided it fits). You may also copy data to or from another model or any other application that supports copy and paste such as a word processor or spreadsheet.

To Move or Copy Cells

1. Select the cells you want to move or copy.
2. To copy the cells select **Copy** on the **Edit Menu**.

   To move cells select **Cut** on the **Edit Menu**.

3. Click on the cell you wish to place the data in and click the **Paste** button.

To Move or Copy Cells between Existing Cells

1. Select the cells you want to move or copy.
2. To copy the cells select **Copy** on the **Edit Menu**.

   To move cells select **Cut** on the **Edit Menu**.

3. Select a cell on the row above where you want to place the data.
4. Click the **Insert New Line** button for each new line that is needed.
5. Click on the upper left cell you wish to place the data in and click the **Paste** button.

To Move or Copy Cells to Another File

1. Select the cells you want to move or copy.
2. To copy the cells select **Copy** on the **Edit Menu**.

   To move cells select **Cut** on the **Edit Menu**.

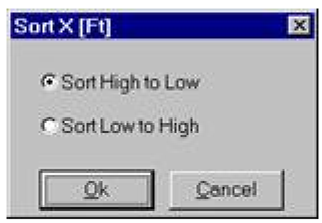
3. Open the file you wish to copy the data to.
4. Click on the cell you wish to place the data in and click the **Paste** button.

Note

- Your data stays in the clipboard until you cut or copy new data. You may repeat step 4 to move or copy data to multiple locations.

Sorting and Finding in Spreadsheets

You may sort spreadsheets by the values in most any column. Simply click in the column you wish to sort, click on the **Sort** button and choose the sorting method. You may relabel the joints, members, or plates after sorting them by using the options in the **Tools Menu**.



To locate or find a specific joint, member, or plate while in a spreadsheet click the **Find** button on the **Window Toolbar**.

Default Spreadsheet Data

Many of the spreadsheets provide the option to save the current data as the default and every new file subsequent to the save will already have that data. This way the office standards that you might use in most of your models are already entered and available in new models. This feature is available in the following spreadsheets: **Materials**, **Design Rules**, **Footings**, and **Load Combinations**.

To save default data simply click the **Save as Defaults** button when you are ready and the current data in the spreadsheet will be used in each new file that you create. The data is saved for the current, active, spreadsheet only and affects no other open or closed spreadsheets.

Special Spreadsheet Functions

There are special functions that help you with a particular spreadsheet. For example you may generate K factors for members on the **Member Design Parameters Spreadsheet**. These features may be accessed on the **Window Toolbar** or you may right click your mouse on the spreadsheet and choose the feature on the **Shortcut Menu**.

Diaphragms Spreadsheet

Click to adjust the diaphragm stiffness. See the Diaphragms chapter before you make any changes to this value.

Member Design Spreadsheet

Click to approximate the K factor for the current member or all of the members. See K Factors (Effective Length Factors) learn more about this feature.

Basic Load Case Spreadsheet

Click to copy loads from one basic load case to another. You may choose certain load types such as distributed loads and point loads. Once you have created a copy you can use the spreadsheet tools to quickly modify the loads.

Click to clear loads from a basic load case.

Load Combinations Spreadsheet

Click Solve Current , Solve Batch or Solve Envelope to solve the current load combination, batch, or envelope solutions.

You may click LC Generator to have the program generate load combinations based on a variety of Building Codes. See Generating Building Code Combinations for more information.

Stability

Instabilities occur whenever a joint can deflect or rotate without limit. Put another way; **a joint is unstable if there is nothing to restrain it**.

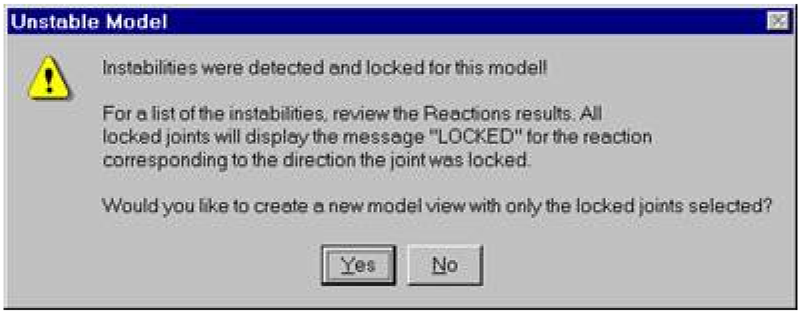
With that one statement you can understand and resolve any instability problem. Instabilities are easy to understand and easy to fix. The next section explains what RISA-3D does with instabilities. The following sections provide some simple examples of instabilities and their resolution.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keyword: **Instabilities**.

Instability Procedure

Because many instabilities are inconsequential to the results yet they prevent a solution RISA-3D locks them as they are discovered and proceeds with the solution. This locking is a boundary condition that removes the degree of freedom from the solution. A reaction (if any) is not calculated and that is one of the dangers of ignoring instabilities. See Testing Instabilities to learn how to test if an instability is affecting the results.

At the end of the solution you will be notified that joints have been locked and that you may view the affected joints in a model view. These locks will also be reported in the **Reactions** spreadsheet.



Note

- Isolated rotational instabilities do not produce a notification that joints have been locked. You may adjust this in the **Preferences** on the **Tools** menu if you wish to be warned about all instabilities.

Instability Causes

Common causes of instabilities are briefly mentioned here and then highlighted in the examples below.

Member End Releases – Boundary Conditions

Overuse of member end releases and/or boundary conditions is by far the most common cause of instability as shown in the examples below. The solution is to either remove a member end release or change a boundary condition so that the joint is restrained. **At least one member or boundary needs to be fixed to each joint to prevent instability.** If you think of a joint as the end of a member and specify no release for that member end this member still will not experience moment at the end if all other elements are left unfixed.

Unconnected Elements

Joints that are not connected into the model cause instabilities. This is much more common in models without Physical Members as can be seen below. The solution may be to merge the model with the model merge feature.

Flexible Elements

Members or plates with relatively small properties such as a long member with a moment of inertia of 1.0 in^4 can cause instabilities. This is usually not a problem unless the members are not used properly.

Instability Examples

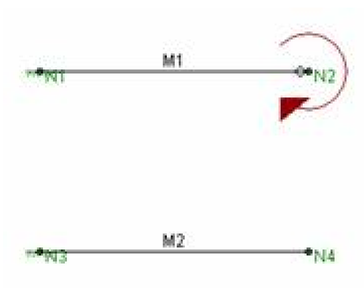
These simple examples are provided to directly address the common scenarios that occur in structural modeling. You will notice a recurring theme so once you understand one or two of them you will have a handle on the causes and resolutions for most instabilities, including those in more complex models.

Remember the golden rule as you look at each example: **A joint is unstable if there is nothing to restrain it.**

Cantilever Beam/Column

If a member end release is specified at the free end of a cantilever the joint becomes unstable because it is free to rotate without any resistance.

Cause: Specifying a member release at the free end of a cantilever member.



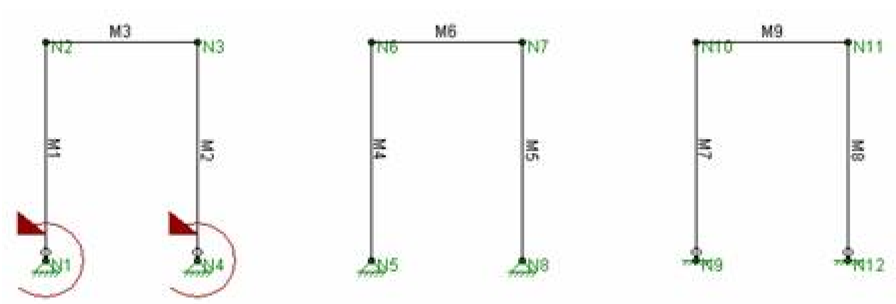
Resolution: Remove the member end release.

If the member is not released from the joint then the member provides resistance to the joint so that it cannot rotate without limit. The member end moves and rotates and the joint goes along for the ride. There will be no moment at the free end of the member since there is nothing there to pass moment.

Column at a Support

If a pinned column base is modeled with a pinned boundary condition **AND** a member end release at the base of the column the joint becomes unstable because it is free to rotate without any resistance.

Cause: Specifying a pinned boundary condition **AND** a member release.



Resolution: Either remove the member end release or specify a fixed boundary condition. Do not do both unless you want a fixed column base that resists moment.

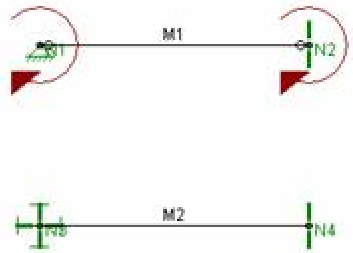
If the member is not released from the joint then the member provides resistance to the joint so that it cannot rotate without limit. The member end rotates and the joint goes along for the ride. There will be no moment at the base of the member since the pinned boundary cannot resist moment.

If instead the boundary is specified as fixed then the boundary provides resistance to the joint so that it cannot rotate. The member end release allows it to rotate while the joint does not. There will be no moment at the base of the member since the fixed boundary cannot pass moment through the member end release.

Simply Supported Beam

If a pinned beam end is modeled with a pinned boundary condition **AND** a member end release the joint becomes unstable because it is free to rotate without any resistance.

Cause: Specifying a pinned boundary condition **AND** a member release.



Resolution: Either remove the member end release or specify a fixed boundary condition. Do not do both unless you want a fixed member end that resists moment.

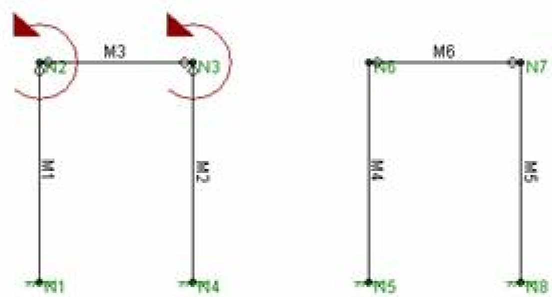
If the member is not released from the joint then the member provides resistance to the joint so that it cannot rotate without limit. The member end rotates and the joint goes along for the ride. There will be no moment at the end of the member since the pinned boundary cannot resist moment.

If instead the boundary is specified as fixed then the boundary provides resistance to the joint so that it cannot rotate. The member end release allows it to rotate while the joint does not. There will be no moment at the end of the member since the fixed boundary cannot pass moment through the member end release.

Beam-Column Connection

If a pinned beam/column connection is modeled with a released column end **AND** a released beam end the joint becomes unstable because it is free to rotate without any resistance.

Cause: Specifying a released column end **AND** a released beam end.



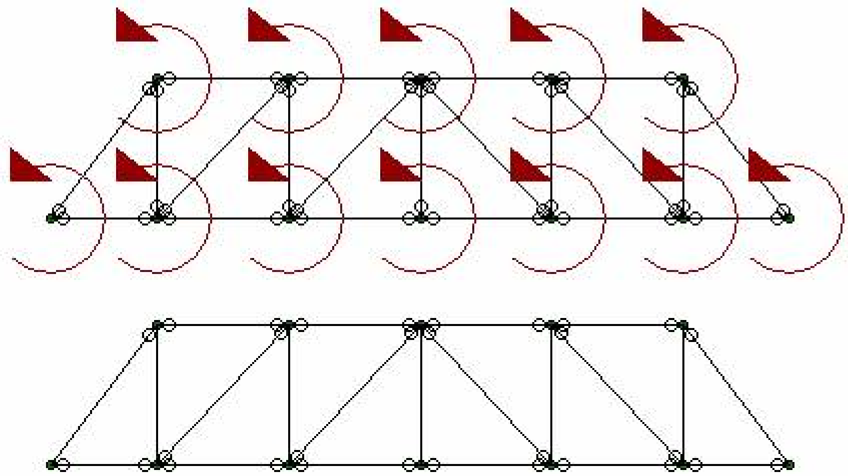
Resolution: Either remove the column end release or the beam end release. Do not do both unless you want a fixed connection that resists moment.

If the column is not released from the joint then the column provides resistance to the joint so that it cannot rotate without limit. There will be no moment at the connection since the beam end release cannot pass moment.

Simple Truss

If a truss panel point is modeled with releases at the ends of EVERY member connecting to that point the joint becomes unstable because it is free to rotate without any resistance.

Cause: Specifying all members with released end**s**.



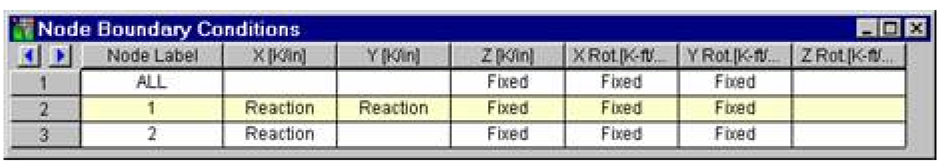
Resolution: Either remove one end release or add a rotational boundary condition at each joint. Do not do both.

If one member end is not released from each joint then the member provides resistance to the joint so that it cannot rotate without limit. There will be no moment at the member end since the remaining members have end releases and cannot pass moment.

You may also solve the problem with a rotational boundary condition at each joint. Using the ALL code you can restrain each joint for rotation and proceed to use end releases at all members.

2D Models

If you are solving a 2D model defined in the XY plane and you're only interested in the planar action, you could enter "ALL" and put an "F" (for Fixed) for Z translation, X Rotation and Y Rotation. See the following figure:



Node Boundary Conditions

	Node Label	X [K/in]	Y [K/in]	Z [K/in]	X Rot.[K-ft/...	Y Rot.[K-ft/...	Z Rot.[K-ft/...
1	ALL			Fixed	Fixed	Fixed	
2	1	Reaction	Reaction	Fixed	Fixed	Fixed	
3	2	Reaction		Fixed	Fixed	Fixed	

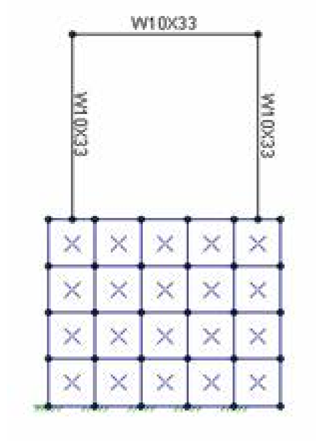
Note

- If a joint is explicitly listed with boundary conditions, those boundary conditions override the "ALL" conditions for all 6 directions. The "ALL" specified boundary codes apply only to those joints NOT otherwise listed on the **Boundary** spreadsheet. This is why joints 1 and 2 in the figure above also have the Fixed code in the Z translation, 2x Rotation and 2y Rotation fields.

Unconnected Elements

If a joint, member or plate is not connected to the model then there will be instability. With the use of Physical Members this is rare in a model that only consists of beam elements however those that have plates or finite members can be defined in a way that they are not connected.

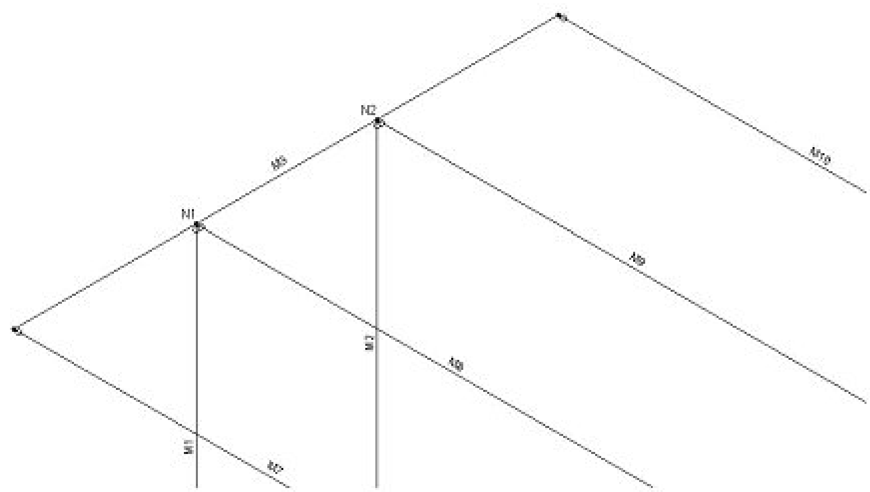
Cause: Unconnected elements. The portal frame is not connected to the plate elements because the bottoms of the columns do not fall on plate corners. The plates are stable however the portal frame is not.



Resolution: For beam models, run the Model Merge feature. For plate models redefine the mesh so that plates are connected at their corners. Model merge will not solve problems caused by lack of plate continuity.

3D Models

For three-dimensional models, torsional instabilities are not uncommon. A "torsional" instability is where a member, or a series of members, is free to spin about its centerline (local x) axis. This diagram illustrates such a situation:

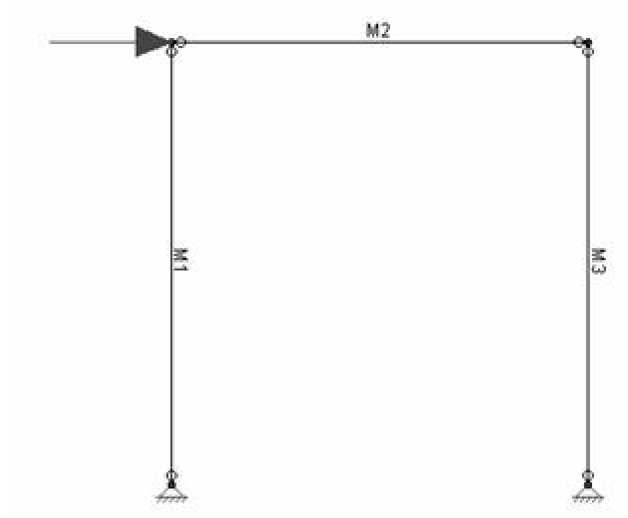


The member M3 as a whole is unstable because there is nothing to restrain it from spinning in torsion. At joints N1 and N2 the columns and beams framing into the member (members M1, M2, M8, M9) are pinned. The same can be said of Members M7 and M10 framing into the member ends. Therefore, there is nothing in the model that will restrict the torsional rotation of the M1 beam.

Another example of a potential local instability is X-bracing with a center joint and loaded with self-weight. X-bracing has almost no out-of-plane stiffness, so even a little bit of out-of-plane load applied at the center joint could cause an instability. (The out-of-plane load could come from a P-Delta analysis, lateral load, etc.) A diaphragm with very weak out-of-plane properties modeled with plate elements can also be a source of potential local instability.

Testing Instabilities

Although some can be ignored, keep in mind that not all instabilities are necessarily inconsequential. Look at the following model:



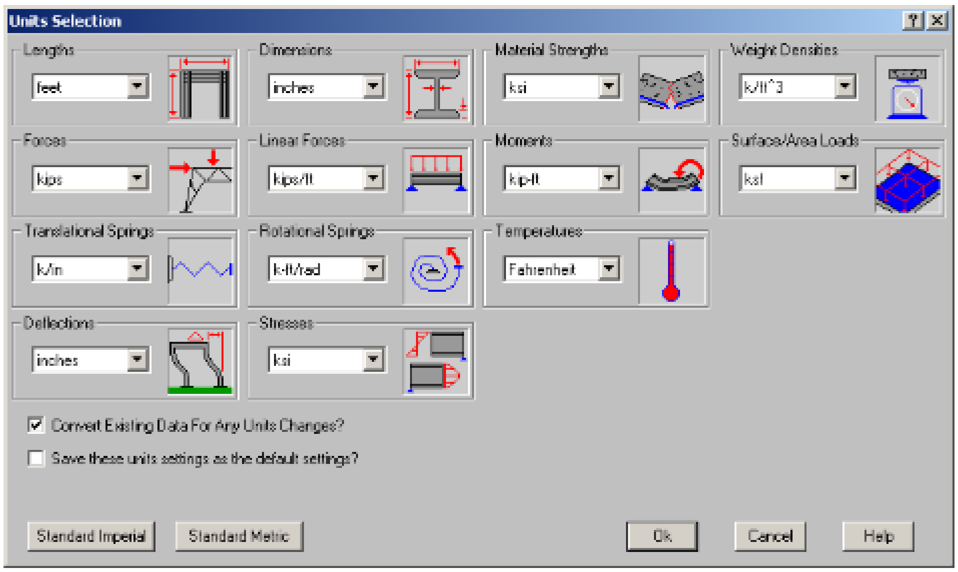
This is an example of a single bent frame that is laterally unstable. To obtain a solution the lateral direction would be locked and a solution obtained, though not a correct one. The warning message may be annoying if you know the instabilities being locked are of no consequence, but there won't be any surprises.

The best way to test whether an instability is inconsequential or not is to apply a Reaction to the joint in the unstable degree of freedom. Then re-run the model and examine the reactions. If the Reaction that is restraining the instability is showing a non-zero force or moment, then you have a problem with the model that must be corrected for you to get valid results. If the Reaction that is restraining the instability is showing a *ZERO* force or moment, then the instability is inconsequential to the results.

Units

You can work with imperial (Kips, inches, etc.) or metric (KN, meters, etc.) units, or any combination of the two. The current units appropriate for each data item are shown at the tops of the data columns in the spreadsheets and with the plot of values in the model view.

You may save any of the units as the default setting so that when you start a new model that information is already there. To do this, simply enter the information that you want to save and check the **Save as Defaults** box and click **OK**.



To Change Units

1. Click the **Units**  button on the **RISA Toolbar**.
2. Specify the units you want for each item in the drop down lists.

   Standard units systems are preset and may be specified by clicking the **Standard Imperial** and **Standard Metric** buttons.

3. If you do not wish to convert values already entered then clear the check box for **Converting Existing Data**.

Standard Imperial Units

This is the units system currently prevalent in the United States. 'Feet' are used for location entries such as joint coordinates and load locations, and 'Inches' are used for section property entries such as area and moment of inertia. Force and weight units are in 'Kips', where 1 Kip = 1000 pounds. Stress units are in 'Ksi' (Kips per square inch).

Standard Metric Units

This units system uses 'Meters' for location entries and 'Centimeters' for section property entries. Force units are in 'kN' (kiloNewtons), where 1 KN = 1000 Newtons. Stress units are in 'MegaPascals' (MPa), where a MegaPascal is 1,000,000 Newtons per square meter. Weight units are in 'Kilograms' and thermal units are in 'Degrees centigrade.

Units Specifications

The following are the unit specifications and their applications:

Measurement	Usage
Lengths	Coordinates, Unbraced Lengths, Load Locations
Dimensions	Shape Properties, Plate Thickness, Member Offsets
Material Strengths	E, G
Weight Densities	Material Density
Forces	Loads, Forces
Linear Forces	Distributed Loads
Moments	Loads, Forces
Surface / Area Loads	Plate/Shell Surface Loads, Area Loads
Translational Springs	X Y Z Boundary Conditions
Rotational Springs	X Rot, Y Rot, Z Rot Boundary Conditions
Temperatures	Thermal Coefficient, Temperatures
Deflections	Deflections, Displacements
Stresses	Fy, Allowable and Actual Stresses

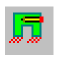
Wall Panels

The wall panel element allows you to easily model walls for in plane and out of plane loads. Wall panel data may be viewed and edited in two ways: graphically in the Wall Panel Editor or in the **Wall Panels Spreadsheet**.

Drawing Wall Panels

There are several graphic-editing features that make the creation and modification of models quite easy. Use the Insert and Modify menus or the Drawing Toolbar to use these features in the model view. To create new wall panels, you can draw them using a drawing grid or draw "dot to dot" existing joints. Once you have created these items you may use other graphic features to apply loads and set boundary conditions.

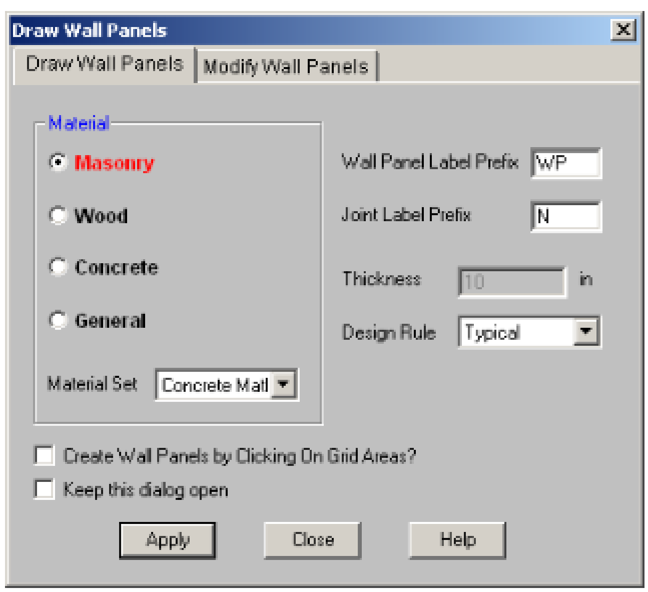
You can set many of the wall panel properties up front or you can modify these properties after you draw them. Modifying properties is discussed in the next sections. See **Wall Panels Spreadsheet** for information on wall panels and their properties.

The **Draw Wall Panels**  button lets you graphically draw wall panels in your model. Enter the appropriate wall panel parameters, click **OK** and draw wall panels between existing joints or on the drawing grid. You will also notice that the coordinates of the joint or grid point that is closest to your cursor are displayed in the lower right hand corner of the model view. The new wall panels will be shown on screen and will be recorded in the **Wall Panels Spreadsheet**.

To actually draw a wall panel, you have two options. One way is to modify your Drawing Grid according to how you wish to lay out your wall panels and use the **Create Wall Panels by Clicking on Grid Areas** option. Wall panels can then be created by clicking in the grid areas formed by the intersecting grid lines. As you click on an area, a wall panel will automatically be created in that area. The second option is to create wall panels by drawing them one joint at a time. First click on the grid point or joint that you want to be the "A" joint for the plate, then the "B" joint, "C" joint, and "D" joint in either clockwise or counter-clockwise order. The wall panel will "stretch" like a rubber band as you draw from joint to joint.

Note:

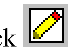
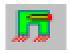
- You must draw wall panels rectangular.
- Wall panels must be oriented vertically in your model.



The parameters shown are the same parameters that you would enter on the **Wall Panels Spreadsheet.**

-

To Draw Wall Panels

1. If there is not a model view already open then click  on the **RISA Toolbar** to open a new view and click  to turn on the **Drawing Toolbar** if it is not already displayed.
2. If you are not drawing between existing joints, you will need to create a drawing grid or define joints on the **Joint Coordinates** spreadsheet.
3. Click the **Draw / Modify Wall Panels**  button and select the **Draw Wall Panels** tab. Then set the wall panel properties.
4. Click **Apply** to start drawing wall panels by clicking on the joints or grid points with the left mouse button.
    1. You must click four points in a clockwise or counter-clockwise order.
    2. Or, click in grid areas.
5. To stop drawing altogether right click or press the **Esc** key.

Note

- To draw more wall panels with different properties, press CTRL-D to recall the **Wall Panel Properties** settings.
- You may also specify or edit wall panels in the **Wall Panels Spreadsheet.**
- You may also view and edit wall panel properties by double-clicking on a wall panel.
- You may undo any mistakes by clicking the Undo  button.

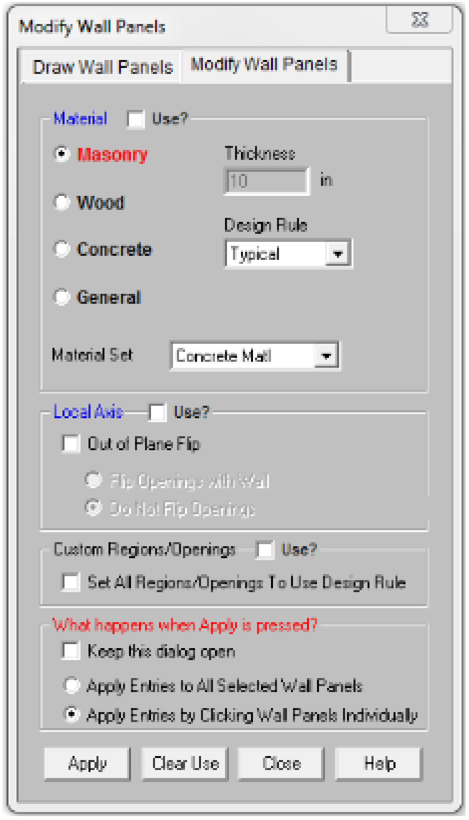
Modifying Wall Panels

There are a number of ways to modify wall panels. You may view and edit the member data in the **Wall Panel Spreadsheet**, you may double-click a wall panel to view and edit its properties, or you can use the **Modify Wall Panels** tool to graphically modify panels.

The graphical **Wall Panel Modify** tool modifies the properties of wall panels that already exist in the model. To use this tool, specify the properties you want to change and then select the wall panels that you want to apply these changes to. Wall panels can be modified one-at-a-time by selecting the **Apply Entries by Clicking** option. This will change the mouse cursor to a modify tool which applies to any wall panels which you click on. A group of selected wall panels can be modified all at once by using the **Apply Entries to All Selected** option. See the Graphic Selection topic for more on selecting.

The **Out of Plane Flip** option will reverse the direction of the wall panel's local z-axis. You can choose to flip the wall and the openings or flip just the wall's local axis without affecting the openings or regions. This feature is useful for aligning the positive z-direction for a group of wall panels which will all receive the same surface load.

The parameters shown are the same as those used to define new wall panels.



The **Use?** check boxes next to the data fields indicate whether the particular parameter will be used or not when the modification is applied. If the box next to a field is checked, that parameter will be applied to any selected wall panels. If the box is NOT checked, the parameter will NOT be applied, even if a value is entered in the field. This allows you to easily change one or two properties on wall panels without affecting all the rest of the properties. Note that if a no value is entered in a field (i.e. the field is blank) and the corresponding check box is checked, clicking "Apply" will have the effect of clearing the data for that field.

To Modify Wall Panels

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Click the **Draw / Modify Wall Panels** button and select the **Modify Wall Panels** tab. Then set the parameters for the new wall panels. Check the **Use?** Box for the items to apply.
3. You may choose to modify a single wall panel at a time or to an entire selection of wall panels.
    1. To modify a few wall panels choose **Apply Entry by Clicking Items Individually** and click **Apply**. Click on the wall panels with the left mouse button.
    2. To modify a selection of wall panels, choose **Apply Entries to All Selected Items** and click **Apply.**

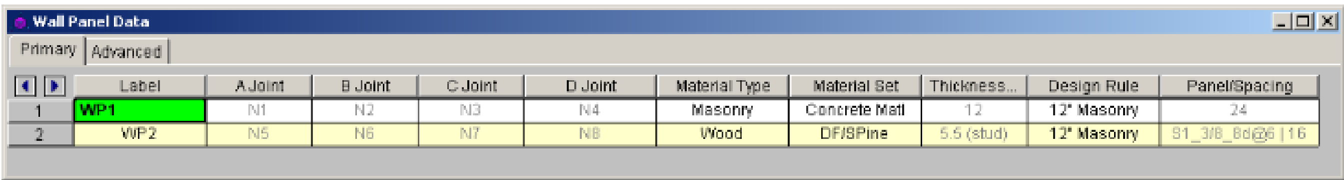
Note

- To modify more wall panels with different parameters, press CTRL-D to recall the **Modify Wall Panels** settings.
- You may also modify wall panels in the **Wall Panels Spreadsheet**.
- You may undo any mistakes by clicking the **Undo** button.
- The thickness option is only available if you are choosing a General or Concrete material. Wood and Masonry require you to change their thickness in the Design Rules spreadsheet.

- For existing models of version 9.1.1or earlier, all wall panels will be brought in using **Custom** regions/openings. Here we provide an option that will reset all wall panels to base their design on the Wall Design Rule.

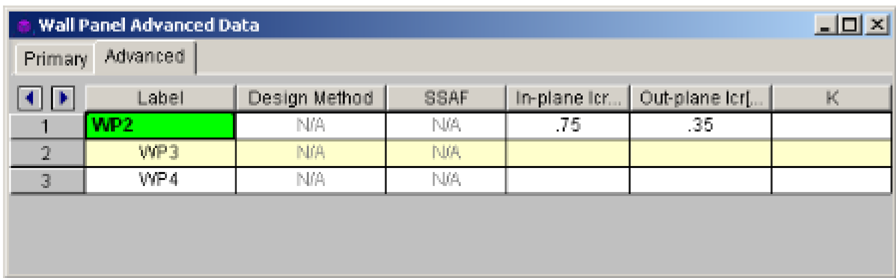
Wall Panel Spreadsheets

Another way of adding or editing wall panels is through the **Wall Panel Spreadsheet**. This spreadsheet is accessible through the **Data Entry Toolbar** and includes data on two tabs: Primary and Advanced.


Wall Panel Data

Primary | Advanced

	Label	A Joint	B Joint	C Joint	D Joint	Material Type	Material Set	Thickness...	Design Rule	Panel/Spacing
1	WP1	N1	N2	N3	N4	Masonry	Concrete Matl	12	12" Masonry	24
2	WP2	N5	N6	N7	N8	Wood	DF/SPine	5.5 (stud)	12" Masonry	81_3/8_8d@6 | 16


Wall Panel Advanced Data

Primary | Advanced

	Label	Design Method	SSAF	In-plane Icr...	Out-plane Icr[...	K
1	WP2	N/A	N/A	.75	.35	
2	WP3	N/A	N/A			
3	WP4	N/A	N/A			

The following data columns hold the primary data for the wall panels:

Wall Panel Labels

You may assign a unique label to any or all of the wall panels. You can then refer to the wall panel by its label. Each label has to be unique, so if you try to enter the same label more than once you will get an error message. You may relabel wall panels at any time with the Relabel Wall Panels option on the Tools menu.

Wall Panel Joints

The A, B, C, and D joint entries are used to define the 4 corner joints of a wall panel. The joints must all lie on the same plane, be oriented parallel to the vertical axis and be entered in either a clockwise or counter-clockwise sequence. They can not be adjusted in the spreadsheet.

Wall Panel Material Type and Material Set

The material set label links the wall panel with the desired material defined on the Material Spreadsheet.

Note

- Currently wall panels can only be made up of concrete, masonry, wood, or general materials.

Wall Panel Thickness

The thickness field on the Wall Panels Spreadsheet is the thickness of the element. This thickness is constant over the entire element. Note that the thickness for Concrete, Masonry and Wood wall panels are set in the Design Rules spreadsheet.

Design Rule

This allows you to choose a specific design rule from the Design Rules spreadsheet. The design rule is where you can specify very detailed information for the wall.

Panel/Spacing

This shows the current panel for masonry and wood walls. Wood and masonry walls can require an iterative solution. This means that the panel properties used for a wall panel may change from solution to solution, so here is the place where that panel is displayed. Note that the panel properties also show up in the output.

When you select a design rule for a wall panel, there may be a range of sheathing call-outs, stud spacings and bar/grout spacings. At the initial solution the program simply uses the first item in the list that meets the criteria of the design rule and then the optimization starts from there. This initial criteria is what will show up in the **Panel/Spacing** column. After you optimize your wall this panel criteria may update/change. If for some reason you want to reset all of these values to the original values, simply press the  button when in the **Wall Panels** spreadsheet.

Note:

- For masonry walls we show the current bar/grout spacing.
- For wood walls we show the current sheathing call-out and the current stud spacing.
- For concrete walls this column is not applicable, as the stiffness of the wall is not affected by the program optimiziation.

Design Method

This is a column specific to wood wall panels and allows you to choose which design method you choose to work with: Segmented, Perforated or Force Transfer. See the Wood Wall Panels topic for more information. These design methods are not applicable for masonry or general wall panels.

SSAF (Shear Stiffness Adjustment Factor)

This is a factor specific to wood wall panels that allows the user to manually adjust the shear stiffness of a particular wall panel. Because the program uses a finite element solution the program does not automatically consider some contributions to the FEM deflections, such as nail slip. With this adjustment factor the user can match up the deflections from their hand calculations with the FEM joint deflections at the top nodes in the wall.

Icr (In Plane and Out of Plane)

These values are only considered in concrete wall design and allow you to modify the stiffness of the wall for cracking considerations. This value will be multiplied by the Igross of the wall. By default (if left blank) the program will use a value of 0.70 for In-plane Icr Factor and 0.35 for Out-plane Icr Factor. If you want to use a different value then you can input it here.

Note:

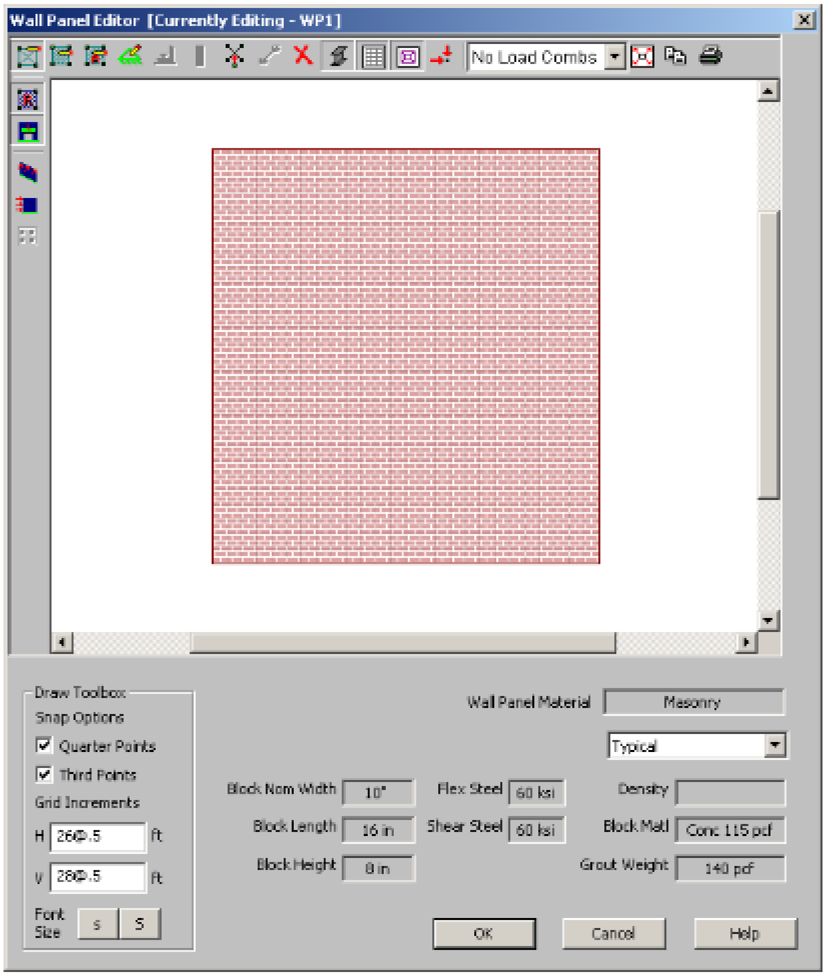
- This factor will only be used if the **Use Cracked Sections** checkbox is checked in the Concrete tab of Global Parameters.

K Factor

This is the effective length factor and is only available for concrete walls. If left blank this will be taken as 1.0.

Wall Panel Editor

The **Wall Panel Editor** allows the user to edit the detailed properties of a wall panel including openings, regions, boundary conditions, and end releases. This also gives design options and details for the specific panel. This application is accessible by double clicking on an existing wall panel.



Note:

- There are many icons, dropdown lists and information shown depending on the type of wall panel you are working with. See the Masonry Wall - Design, Wood Wall - Design and Concrete Wall - Design topics for more information.

Creating Openings

Within the **Wall Panel Editor**, you have the option of adding rectangular openings to the wall panel. To draw an opening, select the **Create New Openings**  button and then select two nodes or grid intersections which make up the two diagonal corners of your opening. Notice that you can view your cursor coordinates in the lower right portion of your screen. To exit this tool right-click your mouse.

Note:

- Drawing an opening in a concrete wall will create a lintel above the opening. We will not design the lintel but we will give analysis results. See the **Concrete Wall - Design** topic on lintels.
- Drawing an opening in a masonry wall will create a lintel above the opening. For more information on defining lintel geometry and design properties, see the **Masonry Wall - Design** topic on lintels.
- Drawing an opening in wood wall will create a header above the opening. For more information on defining the header properties, see the **Wood Wall - Design** topic on headers.
- Drawing an opening in a general wall panel, there is no header/lintel automatically created. The general wall panel is given as an option for analysis only.

- Openings can not overlap a region. Regions must be deleted before you draw an opening in an area. After the opening is created you can go back and redraw the regions.

Creating Regions

Within the Wall Panel Editor, you also have the option of creating different rectangular regions within your wall panel. Regions are used to further define areas of your wall panel for use in analysis/design. If you do not specify a region in a wall panel without openings, then the entire wall panel will be considered a region.

To automatically draw regions you must first have your openings input. Once you have that you can click the **Generate Wall Regions Automatically** button and the program will define regions as we would expect a user to want them.

Note:

- If the regions defined are not located correctly by the generator, you can delete the generated regions with the Delete button and redraw them manually. See below for more information on this.

To manually draw a region, select the **Create New Regions** button and use your cursor to select two nodes or grid intersections which make up the diagonal corners of the region. To exit this tool right-click your mouse.

Note:

- For masonry wall panels, there is a region editor that allows you to define design properties for the region. Double-click inside the boundary of the drawn region to open this editor. See the **Masonry Wall - Design** topic for region information. Note that design and analysis results are displayed by region.
- For wood wall panels using the Segmented method of design, the design and analysis results are displayed by region. The other options, perforated and force transfer around openings, use regions but don't use them for display of results.
- For concrete wall design, the program will automatically create regions at solution around openings. If there are floors/diaphragms that cut through your wall, then separate regions/designs will be created above and below the floors/diaphragms.
- For general wall panels we will not do any design for you. However, you can lay out your regions so that your analysis results will allow you to design your general wall panels much easier.

Boundary Conditions

Within the **Wall Panel Editor**, all boundary conditions are applied as continuous along a wall panel edge. To set boundary conditions within the wall panel editor, select the **Create New Boundary Conditions** button, select your boundary condition criteria, and select **Apply**. To exit out of this tool right-click your mouse. You can also apply boundary conditions to your wall panel outside of the Wall Panel Editor as well, but this is the only place where you can define a continuous boundary condition.

For wood wall panels hold downs and straps are used in the program as well. For more information on adding hold-downs and straps, see the Wood Wall - Design topic.

Draw Toolbox

The draw toolbox, which appears in the lower left corner of the Wall Panel Editor screen gives the user options for drawing within the Wall Panel Editor window. The options include:

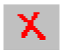
Snap Options allows you to provide snap points at the edges of the wall panel at quarter and third points.

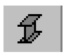
Grid Increments allow you to set a drawing grid within the Wall Panel Editor separate from that in the main model view that you can snap to when drawing openings and regions.

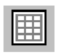
Font Size allows you to increase or decrease the font size associated with the region and opening titles and information shown in the Wall Panel Editor window.

View Controls

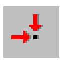
In addition to the wall panel editing tools, the **Wall Panel Editor** window includes the following view controls:

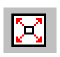
 **Delete** allows you to delete openings or regions from the wall panel.

 **Render** will turn rendering of the current model view on or off, depending on the current setting.

 **Drawing Grid** will turn the display of the Drawing Grid on or off, depending on the current setting.

 **Diaphragm Display** will turn the display of the Diaphragms on or off, depending on the current setting.

 **Loads** will turn the display of the wall panel loads on or off, depending on the current setting.

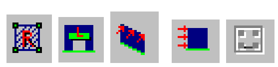
 **Redraw Full Wall View** redraws the wall panel to fit within the **Wall Panel Editor** window.

 **Copy Current View** makes a copy of the current view and saves it to the clipboard.

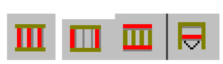
 **Print Current View** prints your current wall panel view.

Note:

- There are also view controls specific to concrete and masonry. For more information see the Concrete Wall - Design and Masonry Wall - Design topic.

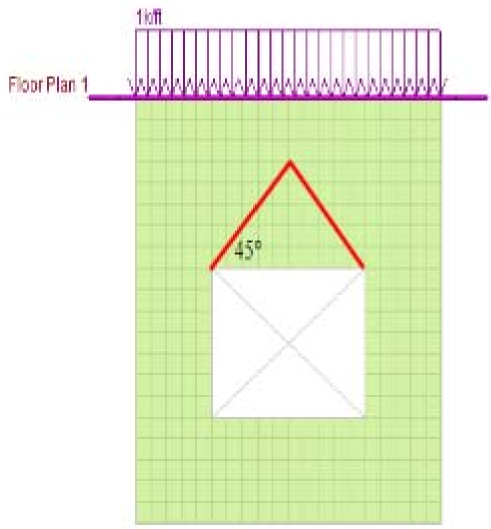


- There are also view controls specific to wood. For more information see the Wood Wall - Design topic.

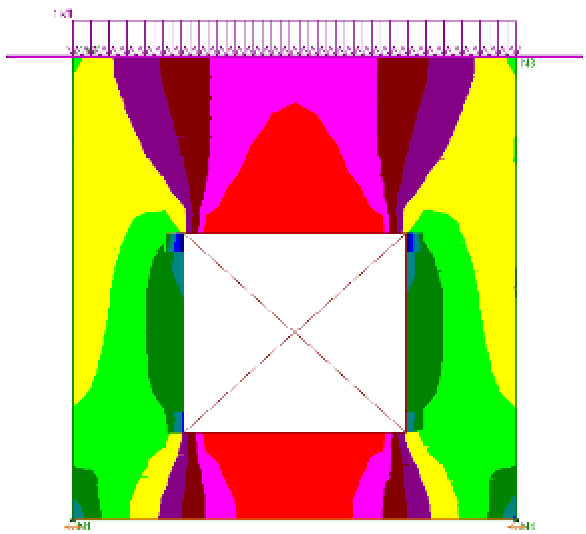


Load Attribution

In RISA-3D, the use of finite elements dictates how loads pass through wall panels. Load is attributed to the structure according to relative stiffnesses of elements. In concrete, wood and masonry design, many empirical equations are formed based on approximations or idealizations. Because of this, you may not get your loading in your wall panel elements (regions and lintels) to match hand calculations exactly. A prime example of this occurs with the 45 degree rule for lintels for masonry. According to theory, arching action occurs in lintels to the point that, if the top of your wall is a sufficient distance away, only the load in the triangular portion above your lintel would actually be taken into the lintel itself. Also, no load applied at the floor level would be felt by the lintel either. See the image below.



Within RISA-3D, this idealization will not hold true. The wall panel is a finite element mesh that attributes load according to the plate mesh FEM behavior. The load that is getting into the lintel is a true representation of how the wall is actually working. There is still arch action taking place as you can tell if you look at the vertical force contours in the wall panel.



In the image above, the red color is an area of very low axial force in the wall. Thus, you can see that, due to plate distribution of force, there is still arching action taking place. This arching action, however, will not be immune to additional loads added to the wall or the opening being located lower in the wall (as is assumed with the idealized arching action in many texts). Thus, though your loads for lintel design may not be identical to what idealized methods might consider, this is a rational loading for the geometry and loading input on the wall panel.

Meshing the Wall Panels

At solution time, the wall panels will be automatically meshed into quadrilateral plate elements. Unlike the plate elements created directly by the user, the automatically generated plate elements are transient in the program and will not be saved in the input file.

The wall panel meshing is treated similar to analysis results. When the results of an analysis are deleted, the wall panel mesh is cleared to be re-built during the next solution. When a solution results file is saved, the meshed elements will be included in that file.

Mesh Size

The global mesh size for the wall panels can be input on the Solution tab of the Global Parameters. The smaller the mesh size, the more accurate the analysis will be. However, smaller mesh size will also lead to longer solution time and more memory usage. The default mesh size is 12 inches RISA-3D.

Localized small mesh sizes are used in the lintel locations for masonry walls, in order to achieve more accuracy for the lintel forces.

Graphical Display of the Wall Panel Mesh

By default, the plate elements associated with the wall panels are not visible to the user. The mesh can be turned on using the setting on the Panels tab of the Plot Options. The **Show Mesh** check box will turn the display of the wall panel mesh on or off.

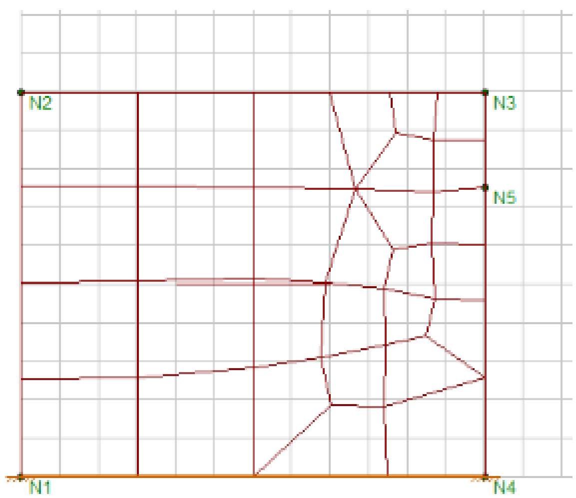
Note

- The display of the mesh is only available when there are active analysis results.

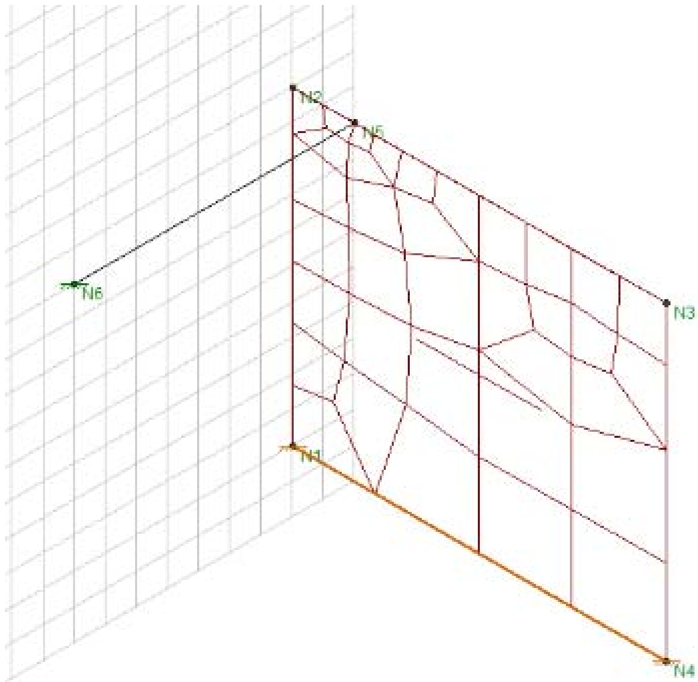
Point Constraints for the Panel Mesh

Point constraints are the locations within the wall panel that require connectivity to the meshed plate elements. The program will automatically generate point constraints at the following locations:

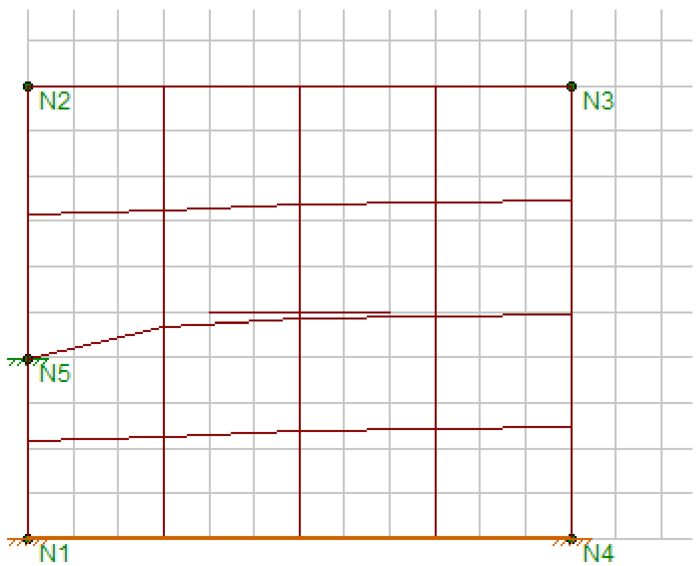
1. Location of an existing joint on the wall panel edges, region boundaries and opening boundaries.



2. Where beams intersects the wall panels (out-of-plane) on the wall panel edges, region boundaries and opening edges.



3. Location of an external boundary condition.



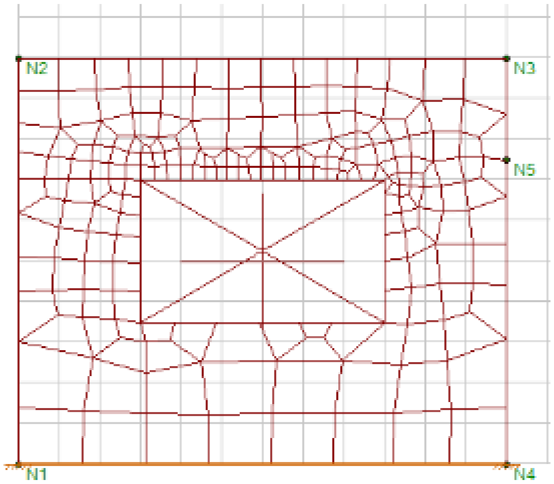
Note:

- Unattached joints that are located on the wall panels can be considered as point constraints and prolong the meshing time. It is highly recommended that the user delete any unattached joints before solving.

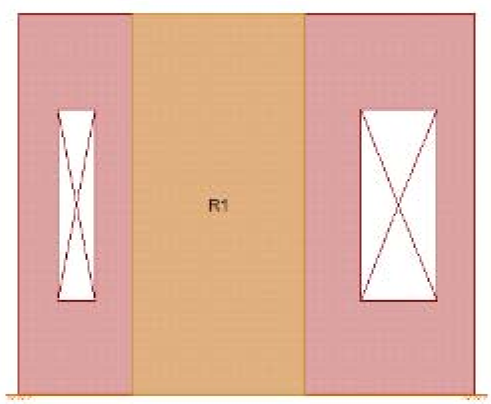
Line Constraints for the Panel Mesh

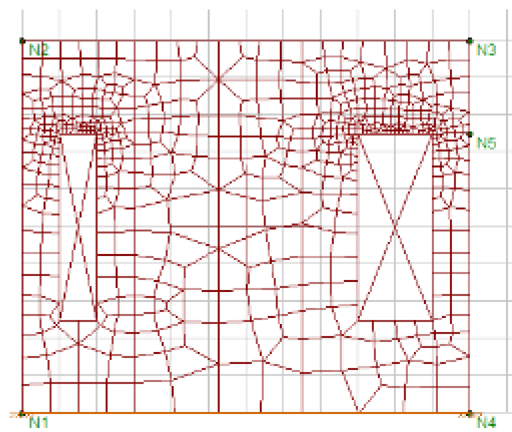
Line constraints are the locations within the wall panel that require continuous connectivity to plate edges rather than a single point. The program will automatically generate line constraints at the following locations:

1. Opening edges.

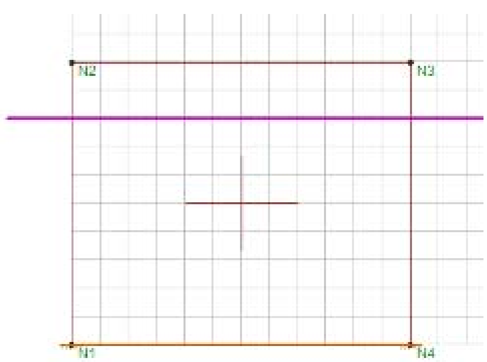


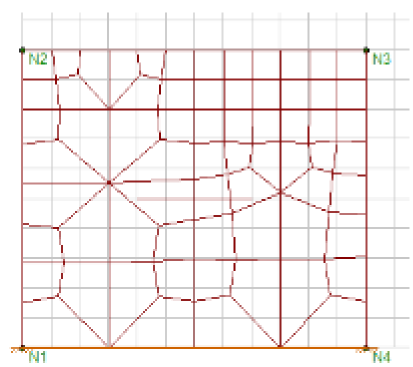
2. The edge and vertical centerline of a defined region.



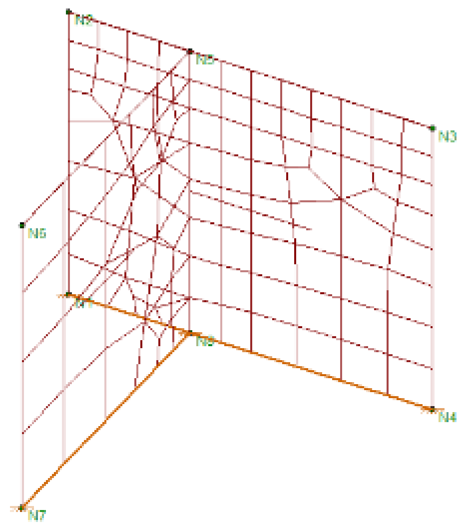


3. Where a diaphragm intersects the wall panel.

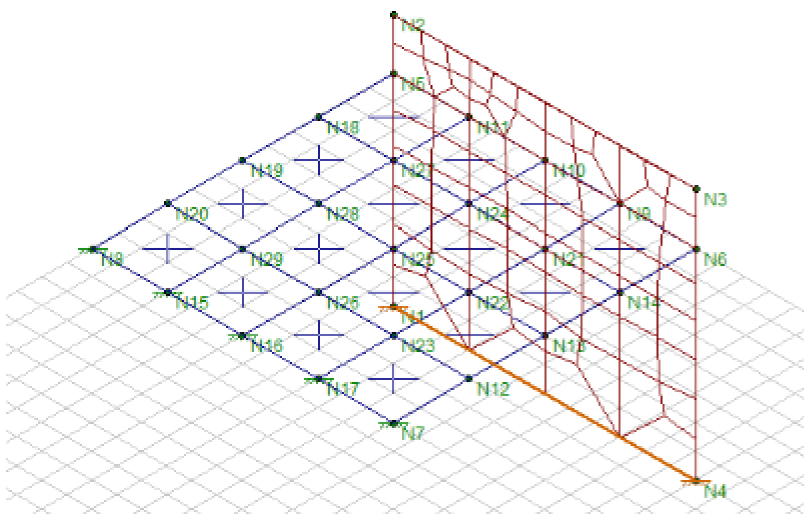




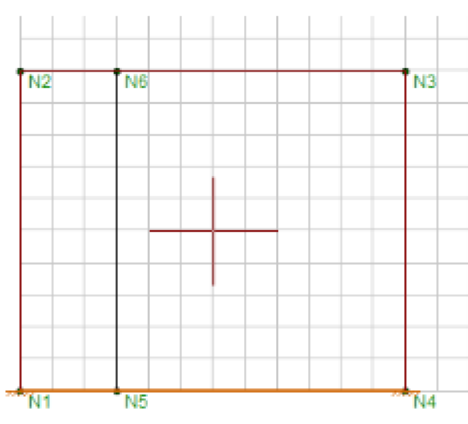
4. At the intersection of multiple wall panels.

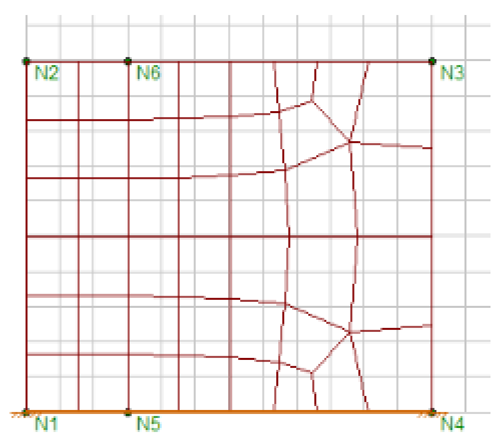


5. Where a plate element mesh intersects the wall panel.



6. Where a beam or column element intersects within the plane of the wall panel.





Tips for Ensuring a Healthy Mesh

In order to generate an efficient mesh that gives accurate results, it is critical to place the line constraints and point constraints correctly. If line constraints or point constraints are very close to each other, the auto mesher will be forced to generate small sized elements in order to satisfy the constraints. Therefore, a large number of plate elements will be generated and the solution will be slowed down significantly.

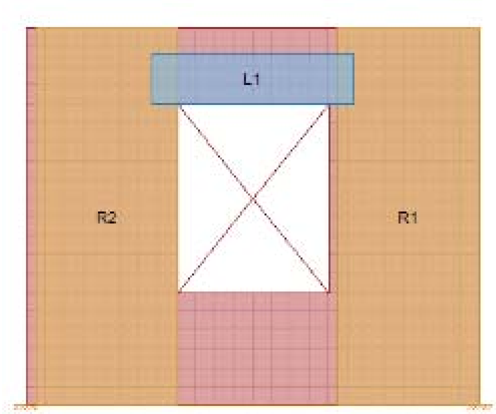
The following guidelines should be followed to ensure a quality mesh:

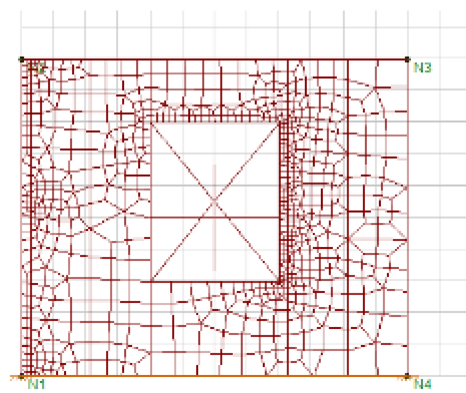
- Avoid generating very narrow regions and openings.
- Avoid small offsets between the external boundary conditions with the location of the region boundaries and wall boundaries.
- Avoid small offsets between opening edges with the region boundaries .
- When a wall panel is intersected by another wall panel, diaphragms, beams or plate elements, keep in mind that the intersection is a line/point constraint. Avoid the small offsets between intersections with the region boundaries or opening edges inside the wall panel.

Example #1: Region Boundaries

1. The left boundary region R1 is placed very close to the opening but not on the opening.
2. The left boundary of region R2 is placed very closed to the left edge of the wall but not exactly on the wall boundary.

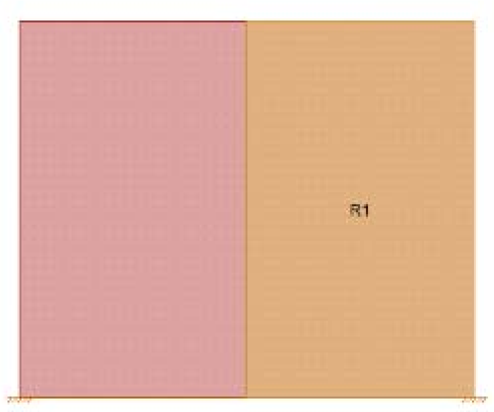
In order to satisfy the line constraints required by the opening edge, region boundaries, and wall boundaries, the program is forced to generate very small meshes in the portion of the wall adjacent to these constraints.

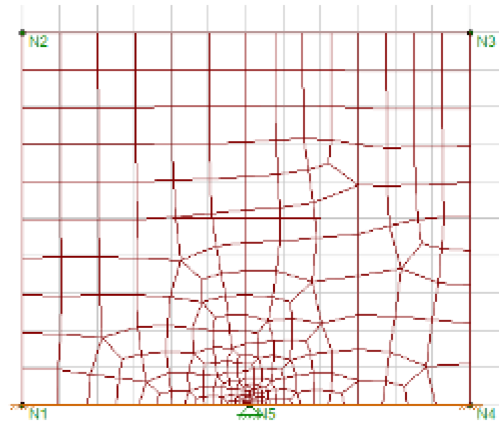




Example #2: Poorly Located Boundary Conditions:

The left boundary of region R1 is at the vertical center line of a wall panel. At the same time, the user placed an external boundary condition at the bottom of the wall panel, which is slightly offset from the center line. In order to accommodate the line constraint of the region boundary and the point constraint of the external boundary condition, the automesher is forced to generate a very small mesh adjacent to these constraints.





Merge Tolerance for Auto-Correction of Mesh

If the distance between the line constraints and point constraints are smaller than the merge tolerance specified on the Global parameters (which defaults to 0.12 inches) then the auto-mesher will automatically snap the constraints together during the meshing. This can eliminate some of the meshing issues that occur in the examples above.

Wall Panels - Results

When the model is solved there is a results spreadsheet specifically for Wall Panel Design. This spreadsheet is divided into different tabs: In plane, Out of Plane, Lintel, Wood Wall Axial and Wood Wall In-Plane. Each tab gives code checks based on the relevant code depending on the material type. These spreadsheets can be used as a summary of all of the panels in your model. To get detailed information about each panel, you can see the Wall Panel Detail Reports.

Concrete Wall Spreadsheet Results

- In Plane Results Spreadsheet
- Out of Plane Results Spreadsheet

Concrete Wall Detail Reports

- Wall Summary Detail Report
- In Plane/Out of Plane Detail Report

Masonry Wall Spreadsheet Results

- In Plane Results Spreadsheet
- Out of Plane Results Spreadsheet
- Lintel Results Spreadsheet

Masonry Detail Reports

- In-Plane / Shear Wall Detail Report
- Out-of-Plane Detail Report
- Slender Wall Detail Report
- Lintel Detail Report

Wood Wall Spreadsheet Results

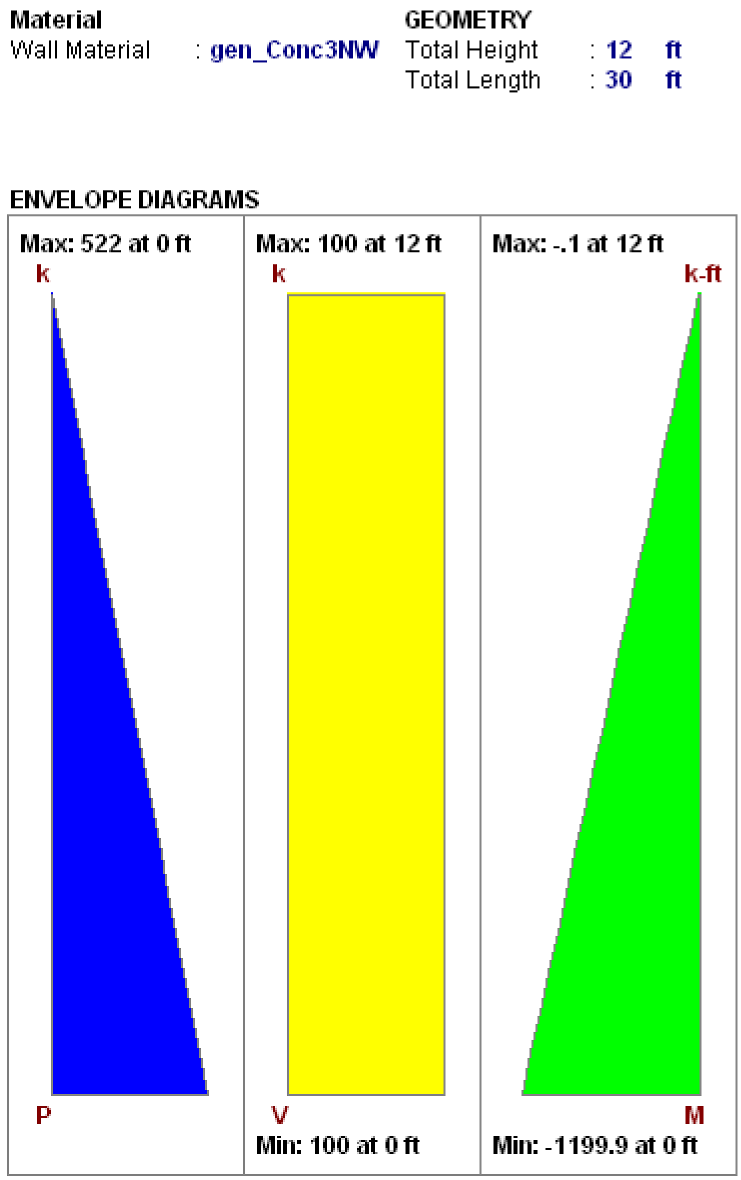
- Axial Results
- In Plane / Shear Results

Wood Wall Detail Reports

- In-Plane or Shear Wall Detail Report

General Wall Detail Report

The general wall detail report shows the material type, height, length, and envelope forces for general walls.



Note

- No detail report is generated for general walls with openings when there is no region defined.

The **Wall Panel Results Spreadsheet** displays the calculated results for wall elements and may be accessed by selecting **Wall Panel Design** on the **Results** menu. The spreadsheet has seven tabs: **Concrete In, Concrete Out, Masonry InMasonry Out, Masonry Lintel, Wood Wall Axial, and Wood Wall In-Plane.**

The first two tabs give results of **Concrete Wall** analysis. For more information on these tabs see Concrete Wall Results.

The next three tabs are results of **Masonry Wall** analysis. For more information on these tabs see Masonry Wall Results.

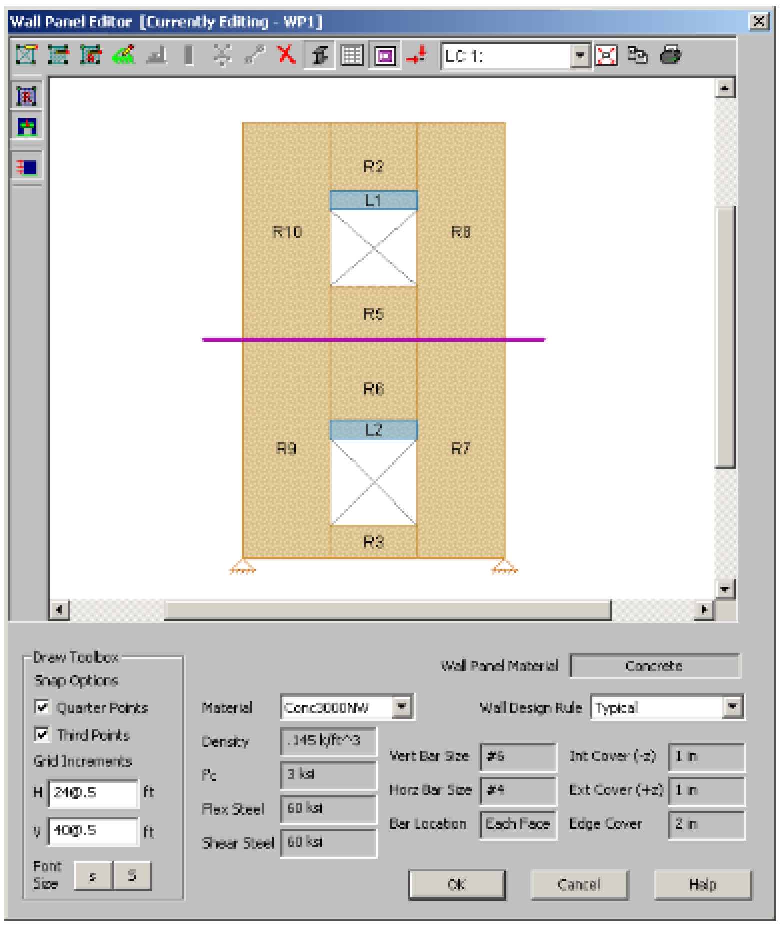
The last two tabs are results of **Wood Wall** analysis. For more information on these tabs see Wood Wall Results.

Concrete Wall Panel - Design

The concrete wall panel element allows you to easily model, analyze, and design concrete walls for in plane and out of plane loads for the ACI 318-05 and newer specifications. Here we will explain the concrete-specific inputs and design considerations. For general wall panel information, see the Wall Panels topic. For concrete wall design rule information, see the Concrete Wall - Design Rules topic. For concrete wall results interpretation, see the Concrete Wall Results topic.

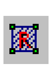
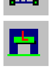
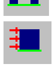
Concrete Wall Input

Double-click on the wall from the model view to open the **Wall Panel Editor.** This dialog displays input information such as Material and Design Rule, as well as gives viewing options for Region and reinforcement display (after solution). Within this dialog you also have control over the application of continuous boundary conditions.



Concrete Wall View Controls

Concrete wall panels have the following view controls:

-  **Toggle Region** display allows you to turn the display of regions on or off.
-  **Toggle Lintel** display allows you to turn the display of lintels on or off.
-  **Toggle Reinforcement** display allows you to turn the display of reinforcement on or off after you have solved your model.
- The **Draw Toolbox** allows you to modify your snap options, drawing grid and font size in the wall panel editor.

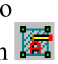
- The **Material** and **Design Rule** drop down lists allow you to modify these parameters in the wall panel editor.

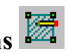
Concrete Wall Regions

Concrete walls depend on Regions for results presentation. The program automatically creates these regions at solution time. If you have a wall panel with no diaphragms then a single region will be created over the entire wall and you will get a single reinforcement design for the entire wall. If there are diaphragms that pass through your wall panel and/or there are openings in the wall, then the wall will be broken up into multiple regions above and below the diaphragms and around the openings, giving a different reinforcement design for each region.

Within each region, the program will optimize the spacing of bars for strength, spacing and minimum reinforcement considerations of the wall.

From within the **Wall Panel Editor**, you have the option of creating rectangular regions within the concrete wall panel. Regions are used to define reinforcement in different parts of the wall. Each region will be assigned a uniform reinforcement, which may be different than the reinforcement in other parts of the same wall (unless you are using Group Story option from Wall Design Rules).

If no regions have been drawn on a wall then they will be automatically generated when a solution is performed. To automatically generate regions prior to running a solution, click the **Generate Wall Regions Automatically** button 

To manually draw regions, select the **Create New Regions**  button and use your cursor to select two nodes or grid intersections which make up the diagonal corners of the region. To exit this tool right-click your mouse.

Concrete Design Considerations

All code references below refer to the ACI 318-11 specification unless noted otherwise.

Reinforcement Design

The program will design the reinforcement spacing for you. For this design reinforcement spacing, rho, and strength requirements are considered for design. If specific reinforcement is defined in the Wall Design Rules spreadsheet then it may be possible for the reinforcement design to not meet code requirements.

Sections 7.6, 14.3 and 11.9 all have provisions regarding min/max spacing, required reinforcement ratios, and proper proportioning of wall reinforcement.

Section 7.6 (General Reinforcement Requirements)

The minimum spacing requirements from 7.6.1 and the maximum spacing requirements from section 7.6.5 are considered for design.

Section 14.3 (Wall Reinforcement Requirements)

The minimum spacing requirements of Sections 14.3.2 and 14.3.3 and maximum spacing requirements of 14.3.5 are also considered for design. Additionally, the thickness requirement from Section 14.3.4 as well as the proportioning and cover checks of 14.3.4 are also considered.

Section 11.9.8 and 11.9.9 (Shear Reinforcement Requirements for Walls)

The program will consider the reinforcement requirements of Section 11.9.9 if the Vu exceeds 0.5*□*Vc (per Section 11.9.8).

Reinforcement Placement

The reinforcement is designed to meet spacing, rho, and strength requirements. This design may cause the reinforcement spacing design to not fit in the wall region at the exact spacing designed for. Therefore the program will add bars to the extreme ends of the wall region to take these remainders into account.

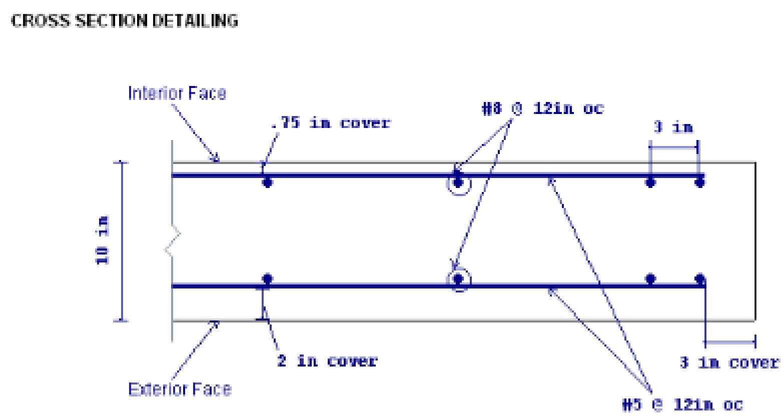
The reinforcement layout algorithm works as follows (picture looking down on a cross section of wall):

First the required spacing is calculated and the wall region length is divided by this spacing.

a. If two bars cannot fit at this spacing, one bar (each face if specified) will be placed at each end of the wall (meeting cover req'ts).
b. If only three bars fit at this spacing then one bar (each face if specified) will be placed at each end of the wall (meeting cover req'ts) and one bar in the center of the wall.
c. If more than three bars are required, then reinforcement is filled in uniformly in the center of the wall and an equal remainder is left between the end bar and the second bar. The image below illustrates this (assume symmetry on both ends of the wall).

Note:

- This process is only required for very skinny walls. Almost all walls will fall into **Step c** above.



Axial Tension

The axial tensile capacity for a wall assumes all reinforcement is fully developed. The capacity (with no bending interaction) equals:

$$\phi T_n := \phi \cdot n \cdot A_s \cdot f_y$$

where n = number of vertical bars in the wall.

Axial Compression

The axial compressive capacity (with no bending interaction) is taken from equation 10-2.

Note:

- Slenderness is taken into account per section 10.10.6. See the Second Order Effects section below.

Bending

Both in plane and out of plane capacity consider beam theory in design. For out of plane design (if no axial force) the capacity is simply defined as:

$$a := \frac{A_s \cdot f_y}{0.85 \cdot f_c \cdot b}$$

$$\phi Mn := \phi \cdot A_s \cdot f_y \cdot \left(d - \frac{a}{2}\right)$$

Note:

- For out of plane reinforcement design, the reinforcement on both faces is taken into account. Thus, the capacity equation above may have two parts to it in order to consider the extreme tension bar and also the bar nearest the compression face that may also be in tension

Minimum Required Moment

Section 10.10.6.5 requires a minimum required moment to be taken into account for each axis in the wall. Thus, if the value from Eq 10-17 is greater than the calculated moment demand this value will be used. This is meant to account for a minimum eccentricity of the axial force in the wall.

The design moment may be factored up due to P-Little Delta effects (see below).

Shear

For in plane shear design, Vc is taken into account using Equations 11-27 and 11-28, where:

- d = 0.8*l_{wall}
- N_u, M_u and V_u are taken at the location of maximum shear demand.

V_s is taken from equation 11-29. Although V_s is only required if V_u >= 0.5*□*V_c, any minimum reinforcement requirement will be used as V_s and added to the shear capacity.

For out of plane shear design the simplified equation from Section 11.9.5 is used for V_c. There is no input for reinforcement to increase the out of plane capacity.

The maximum V_n per Section 11.9.3 is also checked.

Note:

- Specifically in Section 11.9.9, the l_w term here is taken as the FULL length of the wall, not the length of the region. The equations 11-27 and 11-28 take the l_w to be the length of the region.
- Specifically in Section 11.9.9 the h_w term here is taken as the height of the region.

Lambda

Lambda is considered differently for the 2005 and newer codes. For ACI 318-05 the program will always consider lambda = 0.75 if the Density of concrete is <= 115 pcf and lambda = 1.0 otherwise (ACI 318-05 Section 11.2.1). For the ACI 318-08 and newer codes, the program will use the Lambda value directly from the **Materials** spreadsheet.

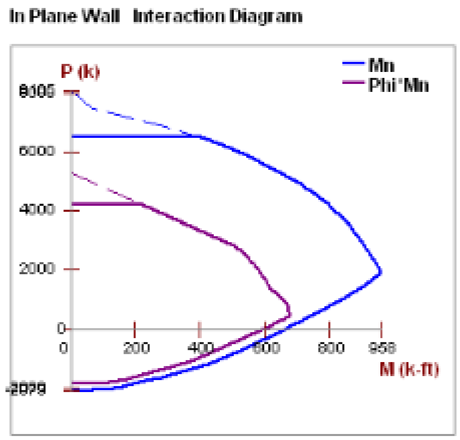
Deflections

The deflection listed in the detail report is based on the finite element analysis of plate elements. This deflection agrees well with beam theory, thus can be calculated based on beam equations.

Note:

- If you are checking a hand calculation of the in-plane deflections, be sure to include the deflection due to shear and consider the cracked moment of inertia.

Interaction Diagrams



The program uses a concrete solver to create the interaction diagram and uses this diagram to calculate the capacity of a wall/wall region based on the demand axial force and moment. The program computes the code check based on making a straight line through the origin, the moment/axial force demand location, and where that line crosses the interaction diagram curve. For the out of plane report there is the possibility of different cover spacing at each face of the wall. Thus, the capacities for each face of the wall are reported.

Moment and Axial Force Thresholds

The program will ignore axial forces and moments that are below a certain threshold. If the moment or axial force is deemed to be inconsequential to the code check then the program will simply not include the interaction of that force. There are two thresholds that are considered:

- Axial Force Threshold: If **Pu < 0.01*f'c*Ag** for that LC, then the axial force in the wall region will be ignored for code checks for that LC.
- Moment Threshold: If **Mu < 0.01*d*Pu** for that LC, then the moment in the wall region will be ignored for code checks for that LC.

These two thresholds allow the concrete solver to work much more efficiently while having little to no effect on code check values.

Second Order Effects

Per Section 10.10.4 an elastic second order analysis will satisfy code requirements. In RISA this means running a P-Delta analysis to consider secondary moments induced due to the displacement of member ends and a P-Little Delta analysis to account for member curvature effects.

The ACI 318-05 specification requires that you follow provisions of Section 10.10.7 (10.13 in the 05 code) for sway frames. In the ACI 318-08 and newer specifications you are required to use either Section 10.10.3, 10.10.4 or 10.10.5.

Because a P-Delta analysis (big and little) is a more robust analysis then the hand calculation methods of 10.10.7, the program uses this same analysis for the ACI 318-05 specification.

P-Delta

The secondary effects due to the displacements of member ends is taken into account with the inclusion of a P-Delta analysis.

To perform a P-Delta Analysis place a "Y" in the **P-Delta** column of the **Load Combinations** spreadsheet. For more information on this, see P-Delta .

Element Curvature Effects (P-Little Delta)

The design moment (max of demand moment and minimum required moment) must be factored up per Section 10.10.6 if the wall is considered slender.

Slenderness requirements are given in Section 10.10.1. The program conservatively only considers equation 10-6. Thus, if the kl/r (in either direction) exceeds 22, then the provisions of Section 10.10.6 are also considered.

Note:

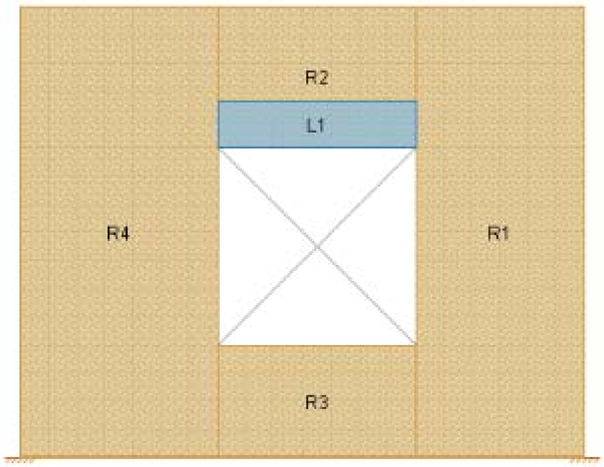
- If Pu > 0.75*Pc the program will fail the wall and not give a code check.
- The moment of inertia (I) in the P_c equation is taken conservatively as $0.25*I_g$ per Commentary section R10.10.6.2.
- L_u for the wall is either defined as the full height of wall or story (a story is broken up by diaphragms that cross the wall.
- P-Little Delta is only considered for full story height regions. Non full-height regions will give a note and not do P-Little Delta.
- For more information on this, see P-Little Delta .

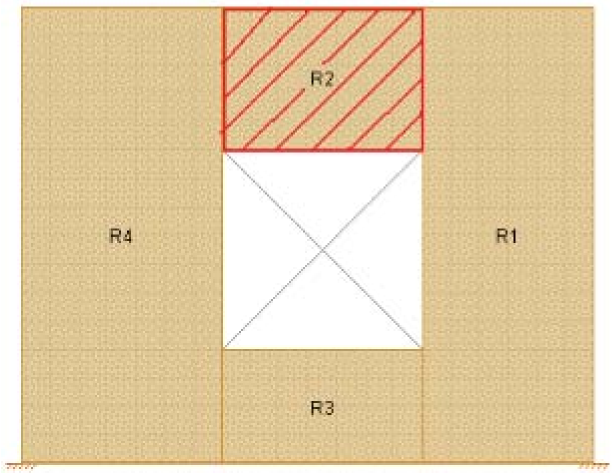
Concrete Lintel Considerations

The addition of openings into a wall in the wall panel editor will automatically create a lintel above the opening. It will be symbolized by a blue bar that has the name of the lintel inside of it. The program will produce axial, shear and moment diagrams for the lintel that can be viewed from the Concrete Wall Detail Report from the Lintel drop-down option. Here we will explain some of the different considerations.

How Lintels are Defined

In RISA-3D the lintel is considered as the entire region directly above an opening (L1 below), even though the blue bar does not cover this entire area.

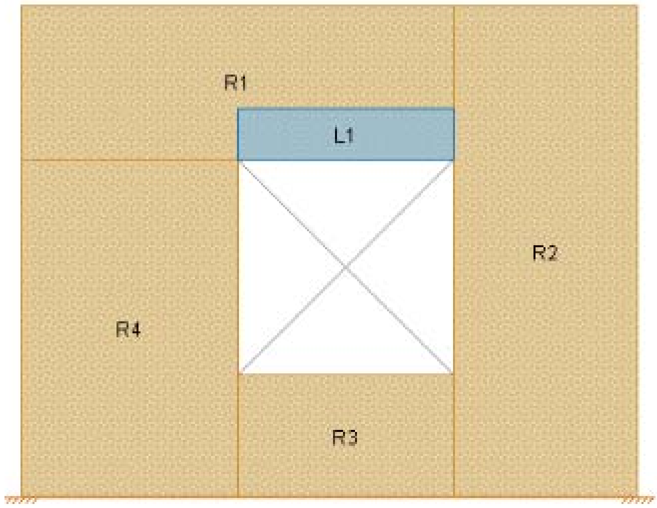




At solution the program will then perform a summation of forces over this entire region and presents the analysis results in the detail report. These results are presented as a "beam" analysis. The program will cut through the entire region vertically multiple times along the length of the region. At each cut the program will calculate the axial, shear and moments. The results from each of these cuts are then combined to form the force diagrams.

Note:

- Keep in mind that the program only reports results in the detail report if there is a region above the opening AND the width of the region matches exactly the width of the opening. Therefore, a region drawn off-center of the opening will not give lintel results.



Load Attribution

See the Wall Panels topic for more information.

Concrete Wall Modeling Considerations

Optimization Procedure

The program will start with the maximum spacing and check that configuration for strength, spacing, and minimum reinforcing requirements. If the max spacing works, then the design is done. If not then the program will reduce the spacing by the spacing increment and then do the same checks. This will occur until a bar spacing is reached to satisfy the code requirements.

RISAFloor/RISA-3D Design Enveloping

If you are using RISAFloor and RISA-3D in tandem to do gravity and lateral design, then the program envelopes the results of both. Thus, RISAFloor will never increase the spacing of bars larger than what was required in RISA-3D and vice versa. Also, when moving between programs, the reinforcement spacing that controls will be carried on into the other program.

For example, let's assume RISAFloor required vertical bars that are #6 @ 12" oc. We then use the **Director** to take the model to RISA-3D. In RISA-3D, the required vertical bars are #6 @ 8" oc. Now, if we take the model back to RISAFloor and solve again, the #6 @ 8" oc spacing is brought over to RISAFloor and code checks are now based on this spacing.

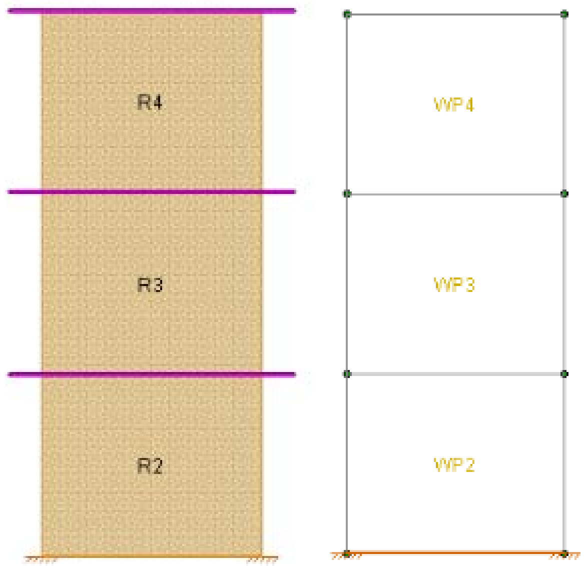
Note:

- When going back and forth between RISAFloor and RISA-3D, you must solve the model in each of the programs to capture this enveloping. If you do not re-solve, then you are then seeing the results from the previous time you solved in that program without updating for the enveloping.

Modeling Multi-Story Shear Walls

For multi-story walls that have diaphragms intersecting, separate regions will be drawn above and below the diaphragms (see Figure 1). The design of each of these regions can be different. This, however, is only true of the spacing of reinforcement. The bar size must be equal for the full-height of the wall, because the Design Rule is for the entire wall and all regions.

If you want to change bar size over the height of the wall, simply create separate wall panels and stack them on top of one another. In this way you can define different Design Rules and thus different reinforcement bar sizes up and down the wall (see Figure 2).



For the stacked regions model where the intention is to drop off bars as you work your way up the wall, lets give an example. Let's say that at the base of the wall you have bars at a 4" o.c. spacing. At some point you want to drop off bars to create an 8" o.c. spacing.

All you need to do here is set your Design Rules such that the Min Vert Bar Spacing is set to 4", the Max Vert Bar Spacing is set to 8" and the Spacing Increment is set to 4".

Wall Cover Dimensions and Local Axes

The program is able to consider different reinforcement cover dimensions for each face of the wall. Because of this the orientation of the wall is important. In the program the **Exterior Face of the wall is oriented in the +z local axis direction**. The **Interior Face of the walls is oriented in the -z local axis direction**.

Note:

- In RISAFloor the wall local axis is irrelevant as we are only doing axial checks.
- If the cover on both faces is identical then the local axis orientation is irrelevant.

A wall drawn in a clock-wise fashion will have its local axis pointed in the positive direction. A wall drawn in a counter-clockwise fashion will have its local axis point in the negative direction.

If the wall local axis is facing the wrong direction then use the Local Axis Flip from the Modify Walls dialog to correct it.

When bringing a model from RISAFloor to RISA-3D then you want to think about how you are drawing in RISAFloor so that the local axes come in properly in RISA-3D. You will want to draw the walls in in RISAFloor in a counter-clockwise manner to get the local z axes to point outward.

Wall Panels Drawn in RISAFloor in Counter-Clockwise Fashion

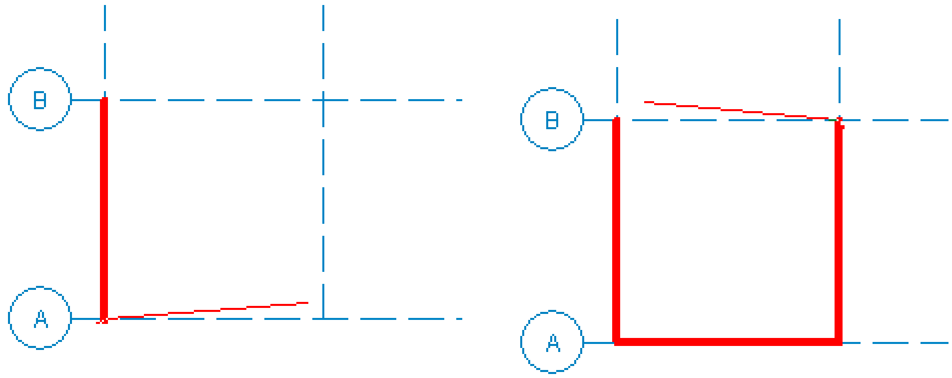
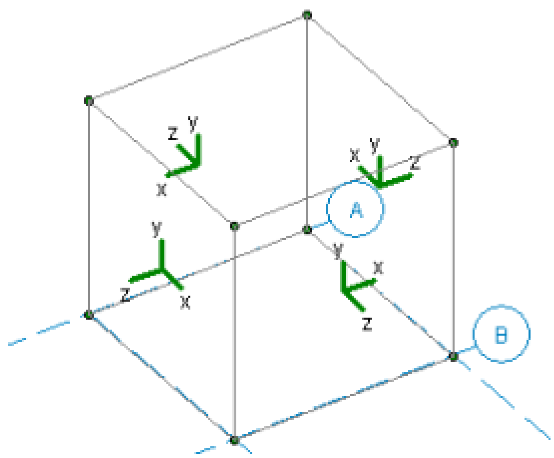


Image of Local Axes in RISA-3D



You can go to Plot Options - Panels tab to view the local axes for the wall.

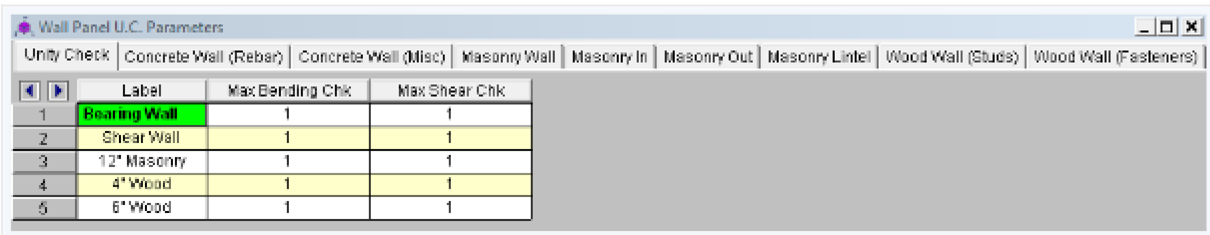
Limitations

- Horizontal reinforcement is designed for in plane shear forces, spacing, and minimum reinforcement checks. They are not used for bending design at this time.
- Walls with bars each face in the wall will require the same size bar and spacing for both faces.
- For sloped walls due to sloping floors, regions cannot be defined in the upper triangular area of the wall panel. Thus, the stiffness of the wall in this area is accurate, but you will get no design results for this portion of the wall.
- Concrete walls are considered completely separately in both the in plane and out of plane directions. Any interaction of the wall, reinforcement, etc., between in plane and out of plane behavior is not considered.
- Concrete wall results are only given for the ACI 318 2005, 2008 and 2011 codes.
- Reinforcement development is not considered. All reinforcement is assumed to be fully developed.

Concrete Wall - Design Rules

The concrete wall panel element allows you to easily model, analyze, and design concrete walls for in plane and out of plane loads. Here we will explain how concrete design rules work. For general wall panel information, see the Wall Panels topic. For information on concrete wall design considerations, see the Concrete Wall - Design topic. For concrete wall results interpretation, see the Concrete Wall Results topic.

Unity Check



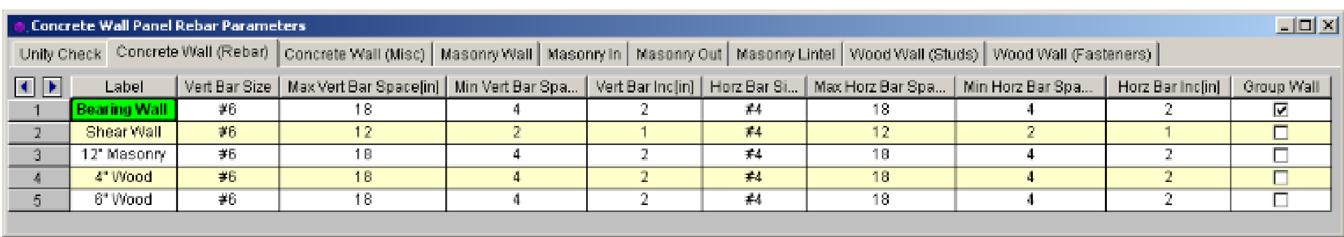
	Label	Max Bending Chk	Max Shear Chk
1	Bearing Wall	1	1
2	Shear Wall	1	1
3	12" Masonry	1	1
4	4" Wood	1	1
5	6" Wood	1	1

Setting a maximum Bending Check (Axial & Bending) or a maximum Shear Check controls the rebar which the program chooses for the wall design. A value of 0.9 denotes that the program may choose a rebar layout that is at 90% of capacity.

Note:

- The same unity check parameters are valid for masonry walls as well. However, these parameters are not considered in wood wall design. For wood walls these values are always assumed to equal 1.0.

Concrete Wall (Rebar) Rules



	Label	Vert Bar Size	Max Vert Bar Space[in]	Min Vert Bar Spa...	Vert Bar Inc[in]	Horz Bar Si...	Max Horz Bar Spa...	Min Horz Bar Spa...	Horz Bar Inc[in]	Group Wall
1	Bearing Wall	#6	18	4	2	#4	18	4	2	☑
2	Shear Wall	#6	12	2	1	#4	12	2	1	☐
3	12" Masonry	#6	18	4	2	#4	18	4	2	☐
4	4" Wood	#6	18	4	2	#4	18	4	2	☐
5	6" Wood	#6	18	4	2	#4	18	4	2	☐

Vert and Horz Bar Size

These are the vertical and horizontal bar sizes used for reinforcement of the wall.

Note:

- The bar size and spacing is assumed to be the same for each face of the wall. Currently reinforcement must be the same for both faces.

Max/Min Vert and Horz Bar Space

The program will design the reinforcement spacing based on these guidelines. If you want the reinforcement to be at an exact spacing, simply enter that spacing as both the min and max in order to force this spacing.

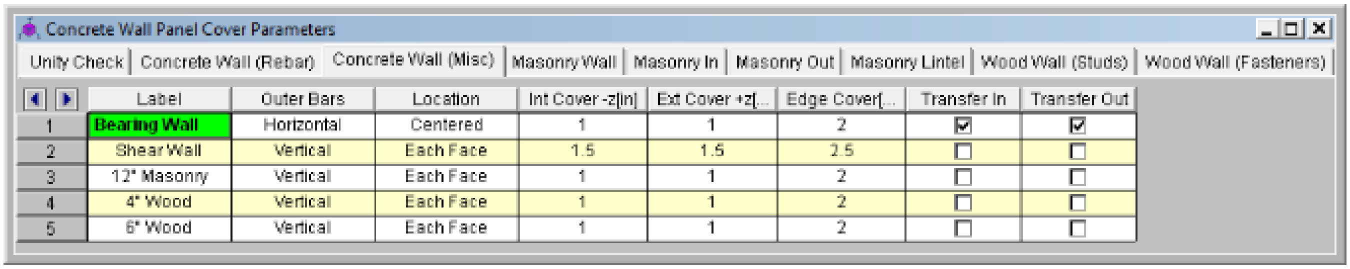
Horz and Vert Bar Increment

This is the spacing change increment that the program will use for design. If the maximum spacing does not work, the spacing will drop by this increment and be checked again. The program will work its way down until it reaches a spacing that meets all reinforcement requirements.

Group Wall

For walls that have multiple regions, this checkbox allows you to group the reinforcement for the regions in a wall. Thus, the worst case vertical and horizontal reinforcement spacing will be used for all regions in the wall.

Concrete Wall (Cover) Rules

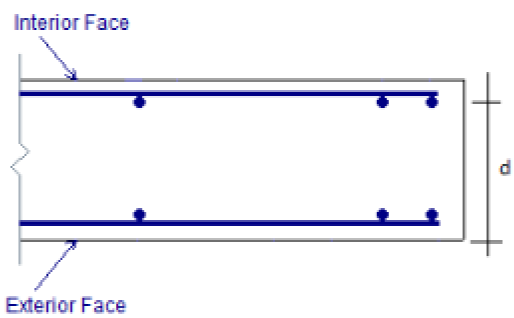


	Label	Outer Bars	Location	Int Cover -z[in]	Ext Cover +z[...	Edge Cover[...	Transfer In	Transfer Out
1	Bearing Wall	Horizontal	Centered	1	1	2	☑	☑
2	Shear Wall	Vertical	Each Face	1.5	1.5	2.5	☐	☐
3	12" Masonry	Vertical	Each Face	1	1	2	☐	☐
4	4" Wood	Vertical	Each Face	1	1	2	☐	☐
5	6" Wood	Vertical	Each Face	1	1	2	☐	☐

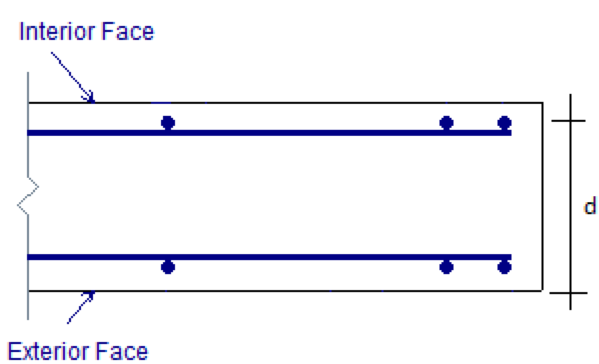
Outer Bars

This defines whether the reinforcement mesh has the Horizontal or Vertical bars closest to the face of concrete. This will affect the "d" calculation for the wall. If the location is Centered then this defines which bar is nearest the outside face of concrete.

Horizontal



Vertical



Location

This allows you to locate reinforcement at each face of wall or centered. If the reinforcement is defined as centered then the program places the vertical bar directly at the center of the wall. The horizontal bar is then placed to one side or the other based on the "Outer Bars" designation.

Note:

- The ACI code requires two curtains of reinforcement if the wall is 10" thick or greater, thus the program will give a warning in the results if you configure your wall like this.

Int Cover (-z)

This is the distance from the interior face of wall to the edge of reinforcement. The interior face of the wall is defined by the negative z local axis direction of the wall.

Ext Cover (+z)

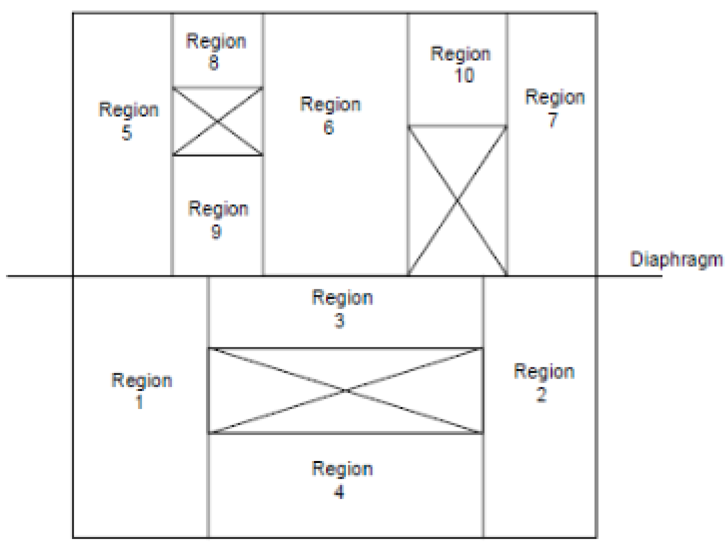
This is the distance from the exterior face of wall to the edge of reinforcement. The exterior face of the wall is defined by the positive z local axis direction of the wall.

Edge Cover

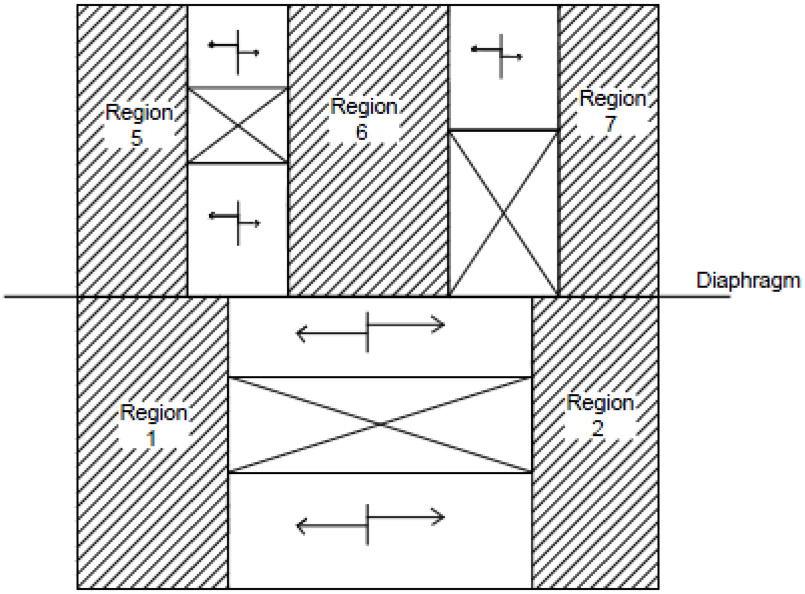
This is the "in plane" cover dimension for the outer edges of walls.

Transfer In and Transfer Out

These options allow you to transfer loads from regions above and below openings to adjacent full-height regions. Transfer in is for in plane loads and transfer out is for out of plane loads. Here is an image of a wall:



If either of the **Transfer** options are turned on for this wall, then any loading in that plane (in plane or out of plane) for regions above and below openings will have their load transferred into the adjacent regions.



A couple of things to keep in mind with the **Transfer** options:

- This is a design-level tool. That is, there is no stiffness change for the model. The program uses the stiffness of the full wall (without openings) for its stiffness. However, after solution, the forces that have accumulated in the regions above/below openings are moved into the adjacent regions. The adjacent region design will then include these forces.
- Results output will not give any information for these "transferred" regions. Only the regions adjacent to the openings will have results.

Concrete Wall Results

Concrete wall results are presented in the **Wall Panel Design** spreadsheet on the **Concrete** tabs, the **Concrete Reinforcing** spreadsheet on the **Concrete Wall** tab and the detail reports. Results are reported on a region by region basis.

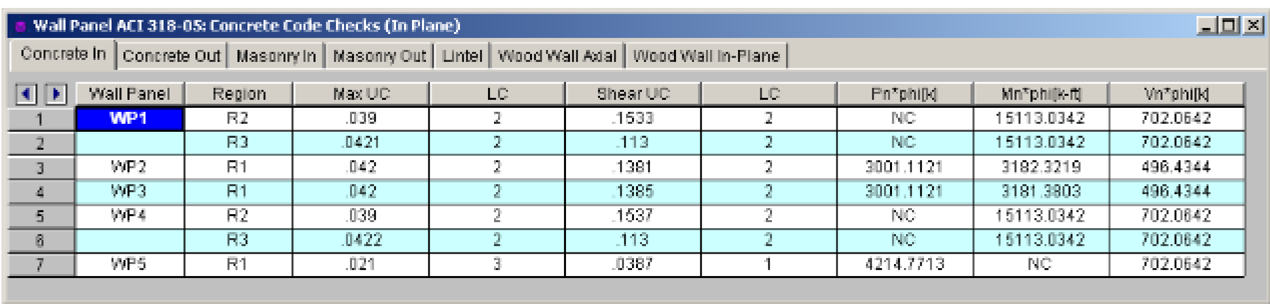
Concrete Wall Spreadsheet Results

The **Wall Panel Design** spreadsheet contains two tabs that involve concrete wall design: **Concrete In**, and **Concrete Out**. Each tab gives code checks based on the chosen concrete code and can be used as a summary of all of the walls and wall regions in your model. To get detailed information about each region, you can see the Wall Panel Detail Report.

Note:

- Concrete lintel results are not given in the output spreadsheets because the program analyzes them without designing. You must go to the detail report to see lintel analysis results.
- If a wall panel is set to Transfer forces, then regions above and below opening will not have results. See the Concrete Wall - Design Rules topic for more information.

In Plane



	Wall Panel	Region	Max UC	LC	Shear UC	LC	Pn*phi[k]	Mn*phi[k-ft]	Vn*phi[k]
1	WP1	R2	.039	2	.1533	2	NC	15113.0342	702.0642
2		R3	.0421	2	.113	2	NC	15113.0342	702.0642
3	WP2	R1	.042	2	.1381	2	3001.1121	3182.3219	496.4344
4	WP3	R1	.042	2	.1385	2	3001.1121	3181.3803	496.4344
5	WP4	R2	.039	2	.1537	2	NC	15113.0342	702.0642
6		R3	.0422	2	.113	2	NC	15113.0342	702.0642
7	WP5	R1	.021	3	.0387	1	4214.7713	NC	702.0642

The **Concrete In** tab provides in plane code checks and capacities relevant to the in plane behavior of the wall.

The **Max UC** entry gives the maximum code check due to axial force plus in-plane bending. The **Shear UC** will show the in plane shear code check. A value greater 1.0 for any of these values would indicate failure.

The **LC** columns report the load combination that produces each of the highest code check values.

The **Pn*Phi** reports the axial capacity of the wall.

Note:

- An NC means that the axial force in the wall is less than the threshold value, so the axial force is not considered.

The **Mn*Phi** reports the calculated in plane moment capacity for the region.

Note:

- An NC means that the bending moment in the wall is less than the threshold value, so the axial force is not considered.

The **Vn*Phi** reports the calculated in plane shear capacity for the region.

Out of Plane

Wall Panel ACI 318-05: Concrete Code Checks (Out Plane)

Concrete In | Concrete Out | Masonry In | Masonry Out | Lintel | Wood Wall Axial | Wood Wall In-Plane

	Wall Panel	Region	Max UC	LC	Shear UC	LC	Pn*phi[k/ft]	Mn*phi[k-ft/ft]	Vn*phi[k/ft]
1	WP1	R2	.0287 (Int)	3	.0067	1	210.7386	NC	6.8724
2		R3	.0045 (Ext)	1	.0043	1	NC	23.406	6.8459
3	WP2	R1	.042 (Int)	2	.0027	3	212.2107	NC	6.8279
4	WP3	R1	.042 (Int)	2	.0026	3	212.2107	NC	6.8288
5	WP4	R2	.0287 (Int)	3	.004	1	210.7386	NC	8.0951
6		R3	0 (Int)	1	.0032	3	NC	NC	6.8731
7	WP5	R1	.021 (Int)	3	.0017	3	210.7386	NC	6.8304

The **Concrete Out** tab provides out of plane code checks and capacities relevant to the out of plane behavior of the wall.

The **Max UC** entry gives the code check due to axial force plus out of plane bending. The **Shear UC** will show the out of plane shear code check. A value greater 1.0 for any of these values would indicate failure.

The **LC** columns report the load combination that produces each of the highest code check values.

The **Pn*Phi** reports the axial capacity of the wall.

Note:

- An NC means that the axial force in the wall is less than the threshold value, so the axial force is not considered.

The **Mn*Phi** reports the calculated in plane moment capacity for the region.

Note:

- An NC means that the bending moment in the wall is less than the threshold value, so the axial force is not considered.

The **Vn*Phi** reports the calculated in plane shear capacity for the region.

Concrete Reinforcing Spreadsheet Results

The **Concrete Wall** tab contains reinforcement results for each region in a concrete wall panel.

Note:

- If a wall panel is set to Transfer forces, then regions above and below opening will not have results. See the Concrete Wall - Design Rules topic for more information.

Concrete Wall

Concrete Wall Reinforcement (By Combination)

Beam Bending | Beam Shear | Column Bending | Column Shear | Concrete Wall | Masonry Wall | Masonry Lintel

	Wall	Region	Thickness[in]	Hor. Bar Size	Vert. Bar Size
1	WP1	R2	8	#4@18in oc (ef)	#6@18in oc (ef)
2		R3	8	#4@18in oc (ef)	#6@18in oc (ef)
3		R4	8	#4@18in oc (ef)	#6@18in oc (ef)
4		R5	8	#4@18in oc (ef)	#6@18in oc (ef)
5		R6	8	#4@18in oc (ef)	#6@18in oc (ef)
6	WP2	R2	8	#4@18in oc (ef)	#6@18in oc (ef)
7		R3	8	#4@18in oc (ef)	#6@18in oc (ef)

The **Concrete Wall** tab displays the thickness, horizontal and vertical reinforcement sizes, and spacing for each region in the concrete wall.

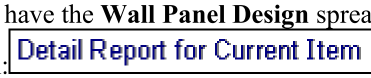
Concrete Wall Detail Reports

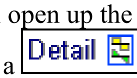
The detail reports show the overall geometry, analysis, and design for the individual regions/stories of the wall panel. The report also shows envelope diagrams for the forces and moments in the region.

Three basic types of detail reports are provided: Wall Summary, Region and Opening. For the region and opening reports there is a drop-down option to view either in-plane or out of plane results.

Accessing the Detail Reports and the Specific Windows

Once you have a solved model, the detail reports become available. They are accessible in two ways:

1. If you have the **Wall Panel Design** spreadsheet open, there will be a button at the top of the screen: 

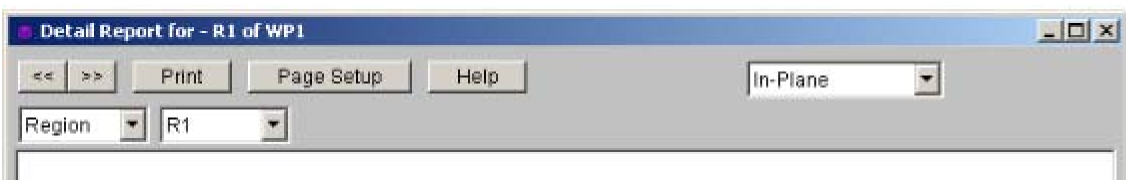
. This will open up the detail report window.
2. If you are in a graphic view of your model, there is a 

button on the Selection toolbar. Clicking this button and clicking on a wall panel will open up the detail report window for that wall panel.

Note:

- Detail Report information is not available for an envelope solution.
- If a wall panel is set to Transfer forces, then regions above and below opening will not have results. See the Concrete Wall - Design Rules topic for more information.

Once the detail report window is open, you will see a dialog area at the top.



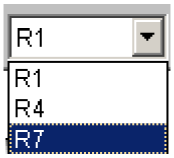
These options control the display of the Detail Report:

- 

- The arrow buttons allow you to scroll quickly between the different wall panels in your model.

- 

- The first drop down list allows you to select between individual Region and Opening (Lintel) results and a summary of the entire Wall. Below we will explain the importance of each of these sections.

- 

- The second drop down list allows you to select between different Regions or Openings within the individual wall panel.

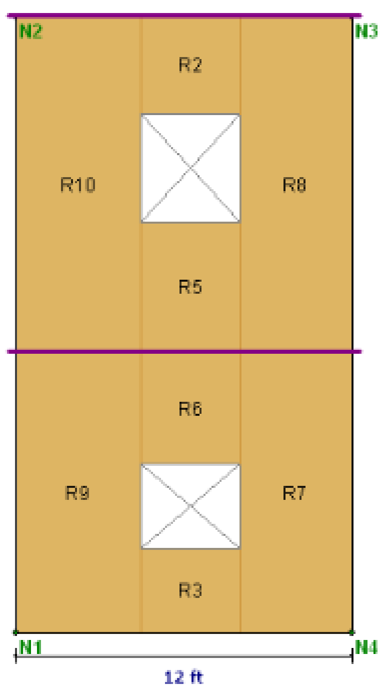
- 

- If you have selected a Region, then you have the option of whether to view the in plane or out of plane report.

Wall Report

This report gives an overview of the wall, a summary of the controlling code checks and deflection information. This report also displays information about the wall, similar to the Region Input Echo and also gives an image of the wall. The image shows region locations, wall length and story dimensions, and the nodes that define the corners of the wall panel.

CRITERIA		MATERIALS			GEOMETRY		
Code	: ACI 318-08	Material Set	: Conc3000NW		Total Height	: 22	ft
Design Rule	: Typical	Concrete f'c	: 3	ksi	Total Length	: 12	ft
Loc of r/f	: Each Face	Concrete E	: 3156	ksi	Thickness	: 12	in
Outer Bars	: Vertical	Concrete G	: 1372	ksi			
		Conc Density	: .145	k/ft^3	Int Cover (-z)	: 1	in
Vert Bar Size	: #6	Lambda	: 1		Ext Cover (+z)	: 1	in
Horz Bar Size	: #4	Conc Str Blk	: Rectangular		Cover Open/Edge	2	in
					K	: 1	
Transfer In?	: No	Vert Bar Fy	: 60	ksi	Use Cracked?	: Yes	
Transfer Out?	: No	Horz Bar Fy	: 60	ksi	In Icr Factor	: .7	
Group Story?	: No	Steel E	: 29000	ksi	Out Icr Factor	: .35	



REGION RESULTS

Region	UC Max In Plane	LC	UC Shear In Plane	LC	Delta Max In Plane(in)	LC	UC Max Out Plane	LC	UC Shear Out Plane	LC	Delta Max Out Plane(in)	LC
R2	.359	3	.432	3	.059	3	.833	2	.314	2	2.332	2
R3	.094	3	.189	3	.061	3	.276	2	.13	2	3.384	2
R7	.476	3	.627	3	.059	3	.833	2	.316	2	2.332	2
R8	.062	3	.189	3	.061	3	.276	2	.13	2	3.384	2

REINFORCEMENT RESULTS

Region	Vertical Reinforcement	Horizontal Reinforcement
R2	#6@6in oc e.f.	#4@18in oc e.f.
R3	#6@18in oc e.f.	#4@18in oc e.f.
R7	#6@6in oc e.f.	#4@18in oc e.f.
R8	#6@18in oc e.f.	#4@18in oc e.f.

The **Region Results** section gives the tabulated results of all regions in the wall for in plane and out of plane design axial/bending, shear and deflection for quick reference. You can view the individual region reports to get a more detailed explanation of these values.

The **Reinforcement Results** section gives the reinforcement results for each region in the wall.

Region Report - In Plane and Out of Plane

This window gives information for your wall on a region by region basis. The Region detail report is split into five portions: input echo, diagrams and design, wall section properties, interaction diagrams and cross section detailing.

Note:

- In RISAFloor, the detail reports are less detailed because RISAFloor does not consider lateral forces.

Input Echo

Below is the input echo portion of the detail report.

CRITERIA			MATERIALS			GEOMETRY		
Code	: ACI 318-08		Material Set	: Conc3000NW		Total Height	: 11	ft
Design Rule	: Typical		Concrete f'c	: 3	ksi	Total Length	: 2.5	ft
Loc of r/f	: Each Face		Concrete E	: 3156	ksi	Thickness	: 12	in
Outer Bars	: Vertical		Concrete G	: 1372	ksi			
			Conc Density	: .145	k/ft^3	Int Cover (-z)	: 1	in
Vert Bar Size	: #6		Lambda	: 1		Ext Cover (+z)	: 1	in
Horz Bar Size	: #4		Conc Str Blk	: Rectangular		Cover Open/Edge	2	in
						K	: 1	
Vert Bar Spac	: 18	in	Vert Bar Fy	: 60	ksi	Use Cracked?	: Yes	
Horz Bar Spac	: 12	in	Horz Bar Fy	: 60	ksi	Icr Factor	: .7	
Group Wall?	: No		Steel E	: 29000	ksi			

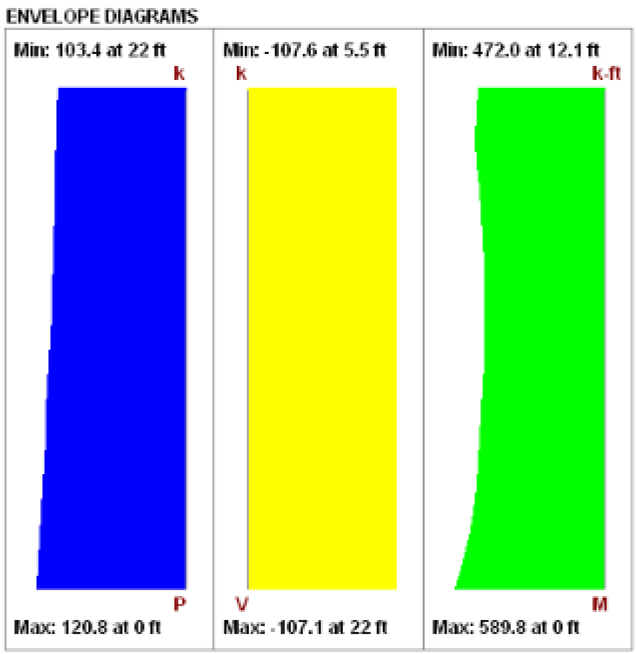
Criteria	Description
Code	Gives the code used to design the wall.
Design Rule	Gives the design rule used to design reinforcement and cover for the wall.
Loc of r/f	States whether reinforcement is defined at each face of the wall or centered.
Outer Bars	States whether the outer bar in the wall is vertical or horizontal.
Bar Size	States the bar size for both horizontal and vertical reinforcement
Bar Spacing	States the bar spacing for both horizontal and vertical reinforcement
Group Wall?	States whether the regions in the wall are grouped or not. See the Concrete Wall - Design Rules topic for more information.

Materials	Description
Material Set	States the Material used for the design of the wall.
Concrete f'c	States the compressive strength of the concrete.
Concrete E	States the modulus of elasticity of the concrete.
Concrete G	States the shear modulus of the concrete.
Conc Density	States the unit density of concrete used for self-weight calculations.
Lambda	States the lightweight concrete factor for shear design.
Conc Str Blk	States whether a rectangular (Whitney's) or parabolic stress block was used.
Bar Fy	States the reinforcement strength for both vertical and horizontal bars
Steel E	States the modulus of elasticity for reinforcement.

Geometry	Description
Wall Dimensions	States the height, length, and thickness of the wall panel region.
Cover Dimensions	States the interior, exterior, and edge reinf. cover dimensions.
K	States the effective length factor which is used in determining slenderness of the wall.
Use Cracked?	States whether a wall is considered to be cracked or not. Defines whether or not to use cracking in the determination of the moment of inertia.
Icr Factor	States the factor that Igross is multiplied by to get the cracked moment of inertia. This defaults to 0.7 for in plane and 0.35 for out of plane.

Diagrams and Design

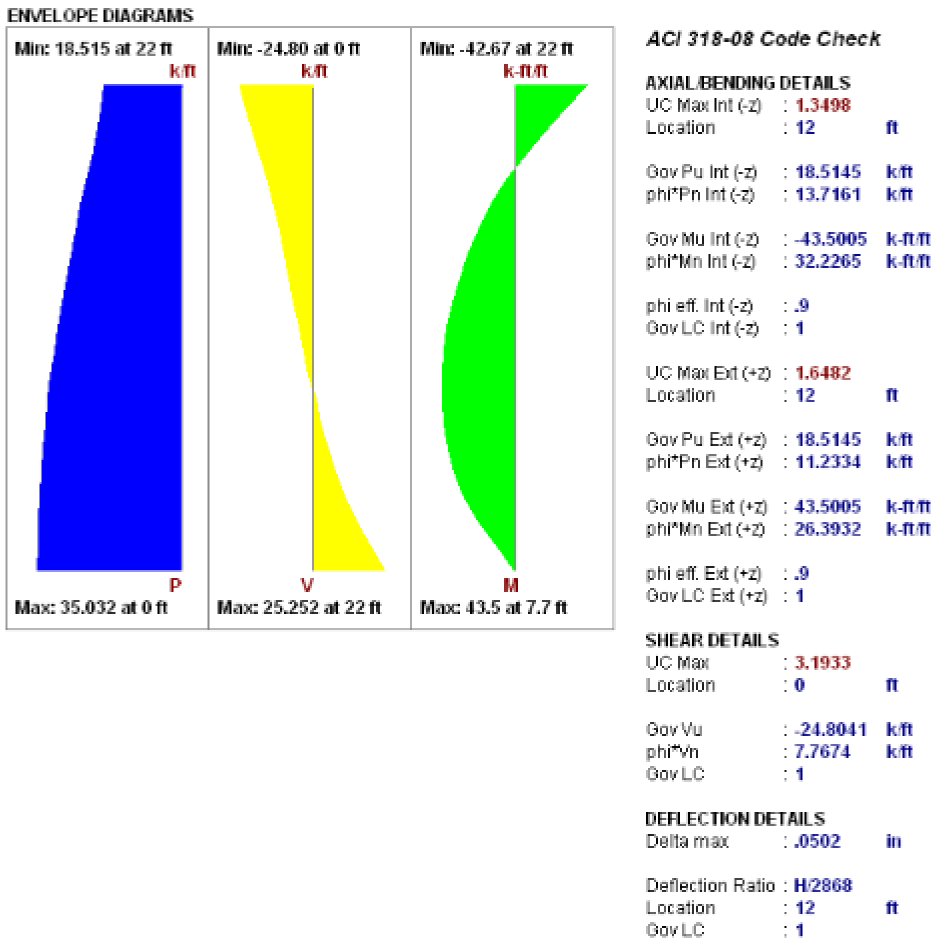
In Plane



ACI 318-08 Code Check

AXIAL/BENDING DETAILS		
UC Max	: .039	
Location	: 0	ft
Gov Pu	: 0	k
phi*Pn	:	k
Gov Mu	: 589.8284	k-ft
phi*Mn	: 15113.0342	k-ft
phi eff.	: .9	
Gov LC	: 2	
SHEAR DETAILS		
UC Max	: .1533	
Location	: 3	ft
Gov Vu	: -107.6151	k
phi*Vn	: 702.0642	k
Vnmax	: 1051.6273	k
Vc	: 347.037	k
Vs	: 589.0486	k
Gov LC	: 2	ft
DEFLECTION DETAILS		
Delta max	: .0092	in
Deflection Ratio	: H/10000	
Location	: 12	ft
Gov LC	: 2	

Out of Plane



Envelope Diagrams

These diagrams show the axial forces, in-plane shear, and in-plane moments of the wall region, as well as the maximum and minimum forces and their locations. The results give an envelope solution of all load combinations .

Because the enveloped results displayed are always the maximum values and because axial and bending forces are checked per their combined effects, the forces in the envelope diagrams won't necessarily be the forces that the wall region is designed to. For example, if there is a high bending moment at the top of a wall region and a high axial force at the base of the wall region, the program will do a check *at each location up the wall region, considering the shear and moment at that location for THAT load combination*. Thus, the maximum axial force given at the location of maximum bending may NOT be the axial force for the LC that produced the maximum bending.

Code Check Summary

This portion of the report gives the capacity and strength values at the section in the wall region where the combined check is maximum, as well as the governing load combination. Much of this information is also reported in the **Concrete Wall Panel Design** spreadsheets.

Axial/Bending Details

The axial and bending capacity are based upon an interaction diagram for the wall region. See below for interaction diagram information. The program computes the code check based on making a straight line through the origin, the moment/axial force demand location and where that line crosses the interaction diagram curve.

For the out of plane report there is the possibility of different cover spacing at each face of the wall region. Therefore the program reports capacities for each face of the wall region. The capacity at the exterior face means that reinforcement at the exterior face of the wall is in tension and vice-versa for the interior face.

Note:

- The program considers walls to act completely separately in the out of plane direction from the in plane direction. Any out of plane/in plane interaction will need to be taken into account by hand.

Shear Details

The shear details section gives the shear demand required in the wall region. The in plane shear strength of the concrete and steel are listed separately, along with the code-prescribed maximum allowed shear. For out of plane shear design, the simplified equation will be used. For more information on shear capacity of concrete walls, see the Concrete Wall - Design topic.

Deflections

The deflection listed in the detail report is based on the finite element analysis of plate elements. This deflection agrees well with beam theory, thus can be calculated based on beam equations.

Note:

- If the deflection ratio is larger than L/10000, then L/10000 will be reported.
- If you are checking a hand calculation for in plane deflections, be sure to include the deflection due to shear and consider the cracked moment of inertia.

Wall Section Properties

WALL SECTION PROPERTIES

Total Length	: 20	ft	r	: 69.282	in	As Provided (H)	: 14.7262	in^2
A	: 2400	in^2	KL/r	: 2.078		rho Provided (H)	: .0061	
Igross	: 1.152e+7	in^4				As min (H)	: 5.28	in^2
Icracked	: 8.064e+6	in^4				rho min (H)	: .0022	
Cracked Mom, Mcr:	3286.3353	k-ft						
						As Provided (V)	: 34.5575	in^2
						rho Provided (V)	: .0144	
						As min (V)	: 3.6	in^2
						rho min (V)	: .0015	

This section reports the properties used to calculate the wall capacities. The reinforcement details (minimums and provided area) are reported.

Slender Wall Considerations (P-Little Delta)

SLENDER BENDING SPAN RESULTS

KL/r out	Cm out	Lu out (ft)	Pc (k/ft)	deltaNS	M2 (k-ft/ft)	M2 min (k-ft/ft)	Mc out (k-ft/ft)
69.282	.8969	20	233.6135	2.5348	9.0571	9.0571	-22.9578

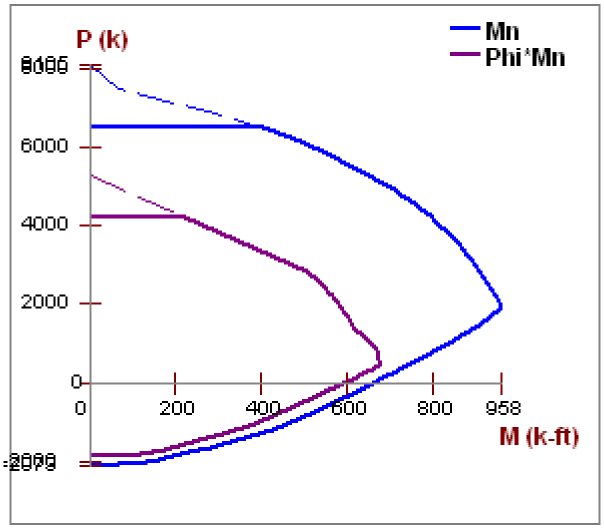
Section 10.10.6 of the ACI 318-11 specification considers moment magnification of non-sway frames. This is essentially the P-little delta effect in the form of an amplified moment due to the effects of element curvature, M_c. This moment replaces the actual demand moment in design checks. For more information on this see the Concrete Wall - Design topic.

Note:

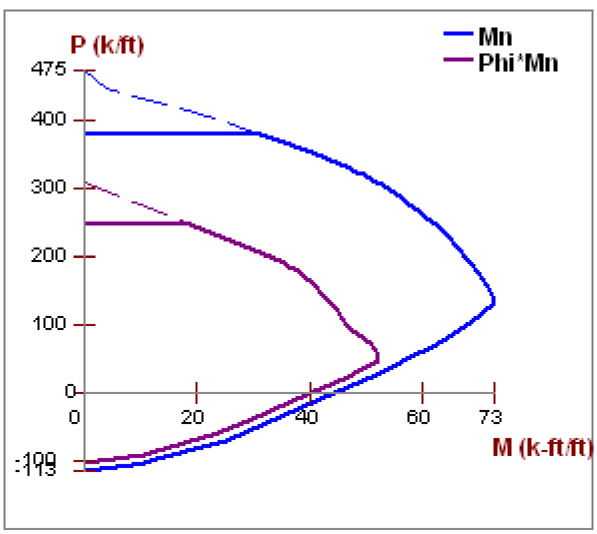
- M_{act} is the actual demand moment in the wall from analysis. The program compares this to the M_{2min} moment and uses the maximum as the M_2 moment.
- Since the moment can be different positive or negative for out of plane bending, the program will provide an Interior and Exterior calculation in this case.

Interaction Diagrams

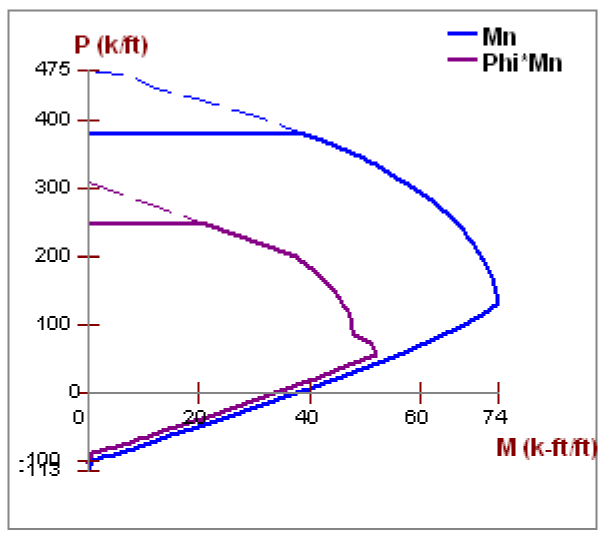
In Plane Wall Interaction Diagram



Interior (-z) Face Wall Interaction Diagram



Exterior (+z) Face Wall Interaction Diagram

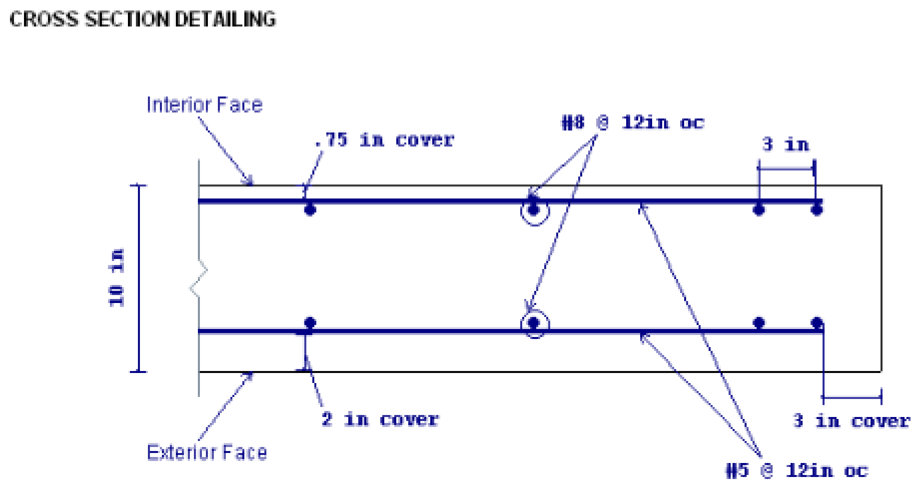


The program uses a concrete solver to create the interaction diagram and uses this diagram to calculate the capacity of each wall region based on the demand axial force and moment. The program then computes the code check based on making a straight line through the origin, the moment/axial force demand location and where that line crosses the interaction diagram curve. For the out of plane report, there is the possibility of different cover spacing at each face of the wall. Thus, the program reports capacities for each face of the wall.

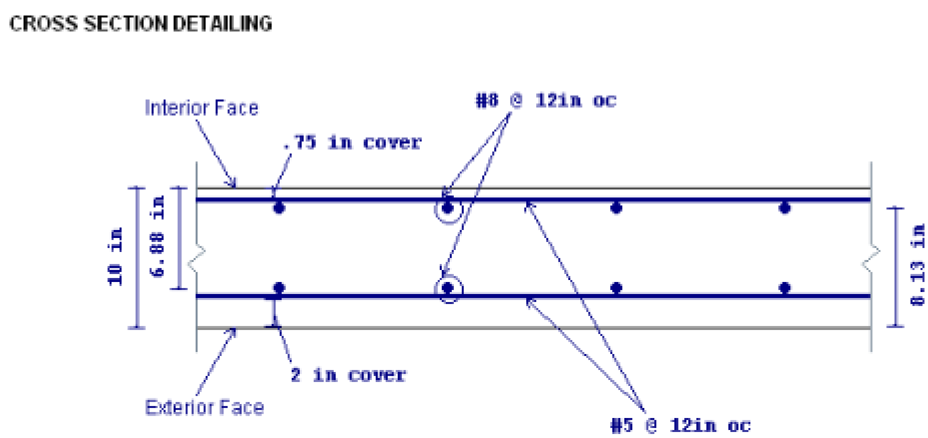
The in plane interaction diagram is based on the entire wall region, while the out of plane interaction diagram is drawn on a per foot basis of wall region.

Cross Section Detailing

In Plane



Out of Plane



The last section of the detail report consists of a graphic cross-sectional view of the wall. This view gives cover dimensions, reinforcement size and spacing, and the wall thickness for placement verification.

Concrete Lintel Report

Lintels - Criteria / Materials / Geometry

The first section of the detail report echoes back the basic input parameters (Criteria, Materials, Geometry) entered by the user. An example is shown below:

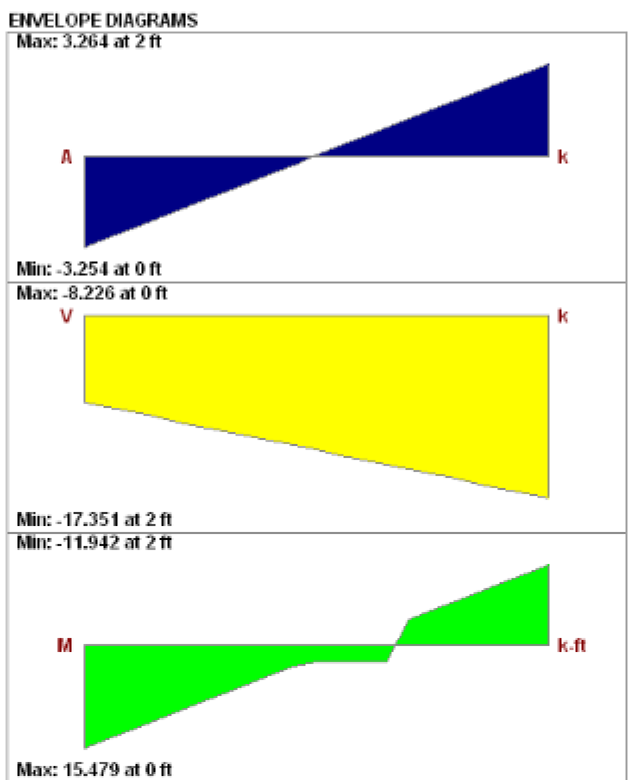
CRITERIA		MATERIALS			GEOMETRY		
Code	: ACI 318-08	Material Set	: Conc3000NW		Total Height	: 3.5	ft
Design Rule	: Typical	Concrete f'c	: 3	ksi	Total Length	: 6	ft
Loc of r/f	: Each Face	Concrete E	: 3156	ksi	Thickness	: 12	in
Outer Bars	: Vertical	Concrete G	: 1372	ksi			
		Conc Density	: .145	k/ft^3	Int Cover (-z)	: 1	in
Vert Bar Size	: #6	Lambda	: 1		Ext Cover (+z)	: 1	in
Horz Bar Size	: #4	Conc Str Blk	: Rectangular		Cover Open/Edge	2	in
Valid Region	: R2	Vert Bar Fy	: 60	ksi			
		Horz Bar Fy	: 60	ksi			
		Steel E	: 29000	ksi			

Since the lintel is not being designed, much of this information is not used.

Information	Description
Valid Region	This is the region that the forces are being considered over. The program will sum forces of the region directly above the opening and the forces given in the detail report come from this portion of the wall.
Total Height	This is the height of the region above the opening that forces are being reported for.
Total Length	This is the width of the opening that the lintel is spanning.
Thickness	This is the thickness of the wall.
Other Information	All of the other information is general information for the overall wall.

Lintel Detail Reports - Diagrams

The next section of the detail report provides the enveloped axial, shear and moment diagrams over the length of the lintel. These can be viewed for both in and out of plane forces and are the "beam" forces for the lintel.

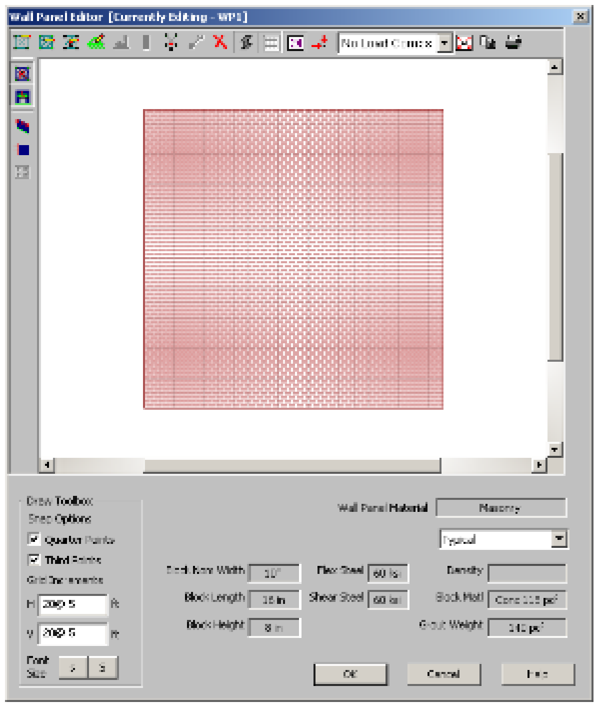


Masonry Wall Panel - Design

The masonry wall panel element allows you to easily model, analyze and design masonry walls for in plane and out of plane loads. Here we will explain the masonry specific inputs and design considerations. For general wall panel information, see the Wall Panels topic. For information on masonry design rules, see the Masonry Wall - Design Rules (this is where you can define block thickness and self-weight). For masonry wall results interpretation, see the Masonry Wall Results topic.

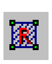
Masonry Wall Input

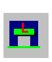
The **Wall Panel Editor** gives some specific information and options for modeling/analysis of masonry walls.

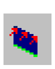


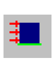
Masonry View Controls

Masonry Wall Panels will have the following view controls:

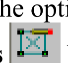
 **Toggle Region Display** allows you to turn the display of regions on or off.

 **Toggle Lintel Display** allows you to turn the display of lintels on or off.

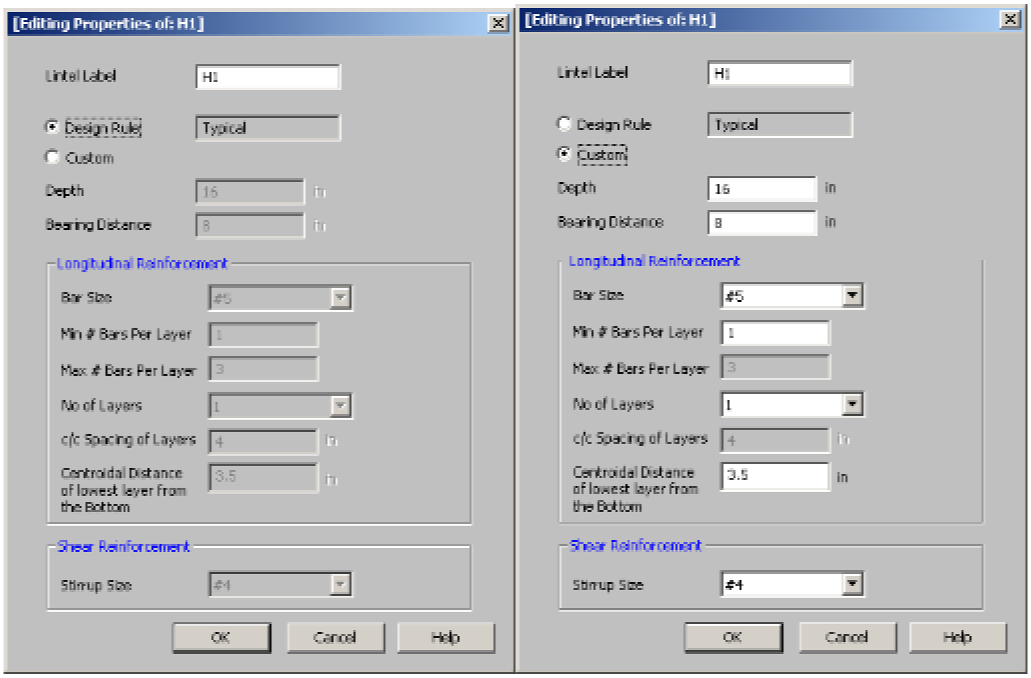
 **Toggle Out of Plane Reinforcement** allows you to turn the display of out of plane reinforcement on or off after you have solved your model.

 **Toggle In Plane Reinforcement** allows you to turn the display of in plane reinforcement on or off after you have solved your model.

Creating Openings in Masonry with Lintels

Within the **Wall Panel Editor**, you have the option of adding rectangular openings to masonry wall panels. To draw an opening, select the **Create New Openings**  button and then select two nodes or grid intersections which make up the two diagonal corners of your opening. When an opening is drawn a lintel is automatically created above the opening. To view or

edit the properties of a masonry lintel, double-click inside the boundary of the drawn opening. This will bring up the Editing Properties window for that particular lintel.



This window will show the design options set in the Wall Design Rules - Lintel to design/analyze your lintel. If you have multiple lintels in a wall and want a specific design that differs from the other lintels, then you can choose the **Custom** option. When using the custom option the program will now use all of the information set in the Lintel Editor and will disregard any information given by the design rule.

Here we will walk through the different input options available for designing/analyzing lintels:

Same as Lintel - This checkbox will allow you to use the same properties as a lintel that has already been created in the same wall panel.

Density - This allows you to make the density of your lintel a different value than the density of the wall panel.

Depth - This is the depth of your lintel.

Bearing Distance - This is the bearing length at either end of the lintel. This is used to calculate the effective length of the lintel.

Bar Size - This is the reinforcement size you wish to use for your main reinforcing in the lintel.

No of Bars Per Layer - This is the number of bars you wish to have in a given layer of reinforcement. There is also an option to have the this value optimized based on geometry of the section and also the number of layers that you have defined.

Number of Layers - This is an option if you need multiple layers of reinforcement in the lintel.

c/c Spacing of Layers - This is the distance between layers (if there is more than one).

Centroidal Distance of lowest layer from the Bottom - This value is used to calculate the "d" value for the lintel.

Stirrup Size - This is the size of stirrup that will be added to the lintel if required.

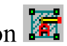
Note:

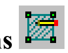
- When inputting bar sizes for your lintels, the program will not allow you to place reinforcement that will not actually fit into the lintel because of width constraints. We use the actual dimensions of the block, the face-shell thickness for the given block chosen and use a 1/2" clear cover between reinforcement and the block.

If you draw an opening in a General wall panel, there is not a lintel automatically created. This is because the program does not currently support concrete lintels and concrete wall panel design. The General wall panel is given as an option for analysis only. In a future release wall panel reinforcing and lintel design will be implemented.

Masonry Regions

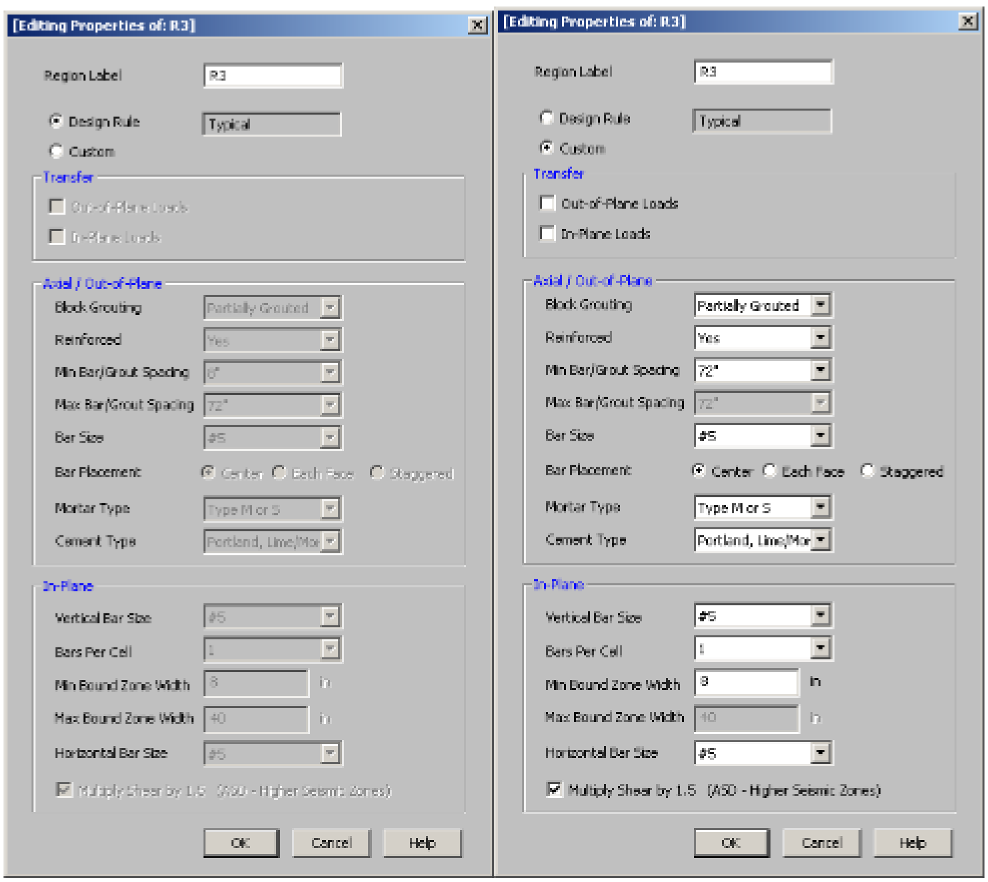
Within the Wall Panel Editor, you have the option of creating rectangular regions within the masonry wall panel. Regions are used to define reinforcement in different parts of the wall. Each region will be assigned a uniform reinforcement, which may be different than the reinforcement in other parts of the same wall.

If no regions have been drawn on a wall then they will be automatically generated when a solution is performed. To automatically generate regions prior to running a solution, click the **Generate Wall Regions Automatically** button 

To manually draw regions, select the **Create New Regions**  button and use your cursor to select two nodes or grid intersections which make up the diagonal corners of the region. To exit this tool right-click your mouse.

To view or edit the properties of a masonry region, double-click inside the boundary of the drawn region. This will bring up the Editing Properties window for that particular region.

This information is populated from the Wall Design Rules spreadsheet for masonry. For more information see the Masonry Wall - Design Rules spreadsheet on setting this up.



In most cases this Region Editor would only be used as a viewer. If, however, you want to change the reinforcement for a region within the wall to make it different than the Wall Design Rules, you can use the **Custom** option. When using the custom option the program will now use all of the information set in the Region Editor and will disregard any information given by the design rule.

Note:

- For models created with version 9.1.1 or earlier ALL regions will come in set to Custom, bringing over the information exactly as it comes from the existing model. For all newly created models in version 10 or later the regions will default to the wall design rule.

Within this dialog you can specify the properties which will be used for the design of the region.. The program can optimize the bar spacing and the boundary zone width based on code checks. The block size, reinforcing strength and the method of self-weight calculation are defined in the Design Rules under the Masonry Wall tab.

Note:

The program designs regions separately for out-of-plane and in-plane forces, thus the Region Editor is divided into different parts.

Here we will walk through the different input options available for designing/analyzing regions:

Transfer - This is an option as to whether or not you want this region to transfer Out of Plane and In Plane Loads. If you check these transfer options we will remove the stiffness from that region and dump those loads into the adjacent regions.

Note:

- These transfer options are only available when you have defined a region above or below an opening.
- Even if you do not choose to check these transfer options, if the region you are considering is supported by adjacent regions, then the load will still transfer into these regions.

Same as Region - This checkbox allows you to define this region exactly as you have in a previous region within the same wall panel.

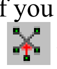
Axial/Out-of-Plane - Allows you to define properties of a region based on out-of-plane/axial forces.

- **Block Grouting** - Allows you to define how you want your wall grouted. If you choose "Partially Grouted" the program will optimize the grout spacing with the Bar/Grout Spacing
- **Reinforced** - Gives the option of designing the wall as reinforced or unreinforced.
- **Bar/Grout Spacing** - This allows you to define the bar/grout spacing. If you have the "Optimize" box checked, the program will optimize the reinforcement spacing based on code checks.
- **Bar Size** - This is the main vertical bar size that will be used in design.
- **Bar Placement** - This defines how you want to lay out the reinforcement in your region. Each Face will put reinforcement on both faces of a given cell. Staggered alternates the bars to either face of wall region.
- **Note:**
    - When using the staggered option you are selecting to space the bars at each face at double the bar/grout spacing defined above. For example, if you have a staggered spacing at 24" oc you have a bar on the outside face at 48" oc and a bar on the inside face at 48" oc. These bars are staggered, thus you have grout filled cores at 24" oc.
- **Mortar Type** - Allows you to specify the type of mortar to design for.
- **Cement Type** - Allows you to specify the type of cement to design for.

In-Plane - Allows you to define properties of a region based on in-plane forces.

- **Vertical Bar Size** - Allows you to define vertical bar size for the boundary zones.
- **Bars Per Cell** - Allows you to define one or two bars per cell.
- **Boundary Zone Width** - The user must define a boundary zone width but RISA will optimize the width if the "Optimize" box is checked.
    - **Note:** If you have the optimize checkbox selected, the program will optimize the boundary zone width based on code checks.
- **Horizontal Bar Size** - Allows you to define horizontal bar size to be used if horizontal reinforcing is required.
- **Multiply Shear by 1.5** - This is an option that may be required in high seismic zones. Section 2106.5.1 of the IBC 2006 gives more information on this.

Merge Lintels

When an opening is drawn in a masonry wall panel, you will notice that a lintel beam region is automatically created above the opening. If you have multiple openings, you may want to merge the individual lintels into one. To do this, select the Merge Lintels  button. If you have two lintels you want to merge, then click within each of the openings to merge them into one. If you have multiple lintels that you want to merge, click inside of the two openings that define the ends of the merged lintel that you want. You will see that this merges your multiple openings into one. To exit out of this tool right-click your mouse.

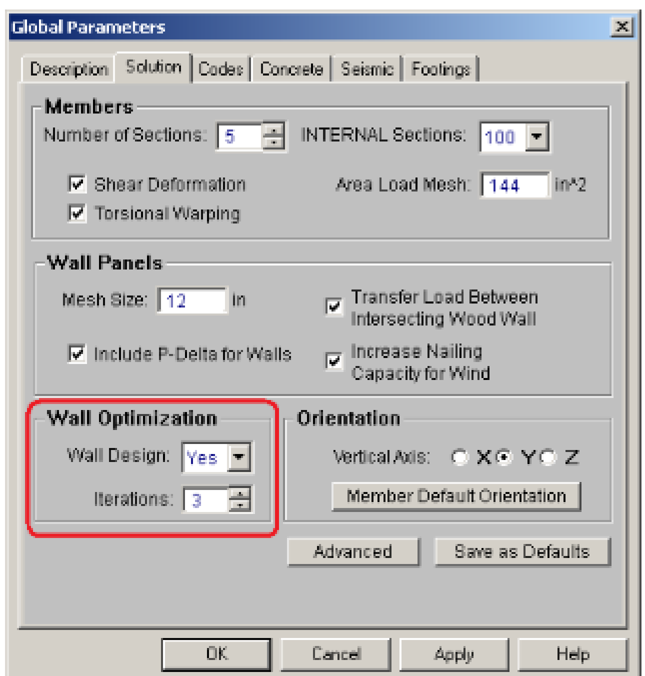
When merging lintels, the top edges of the lintels have to be identical. If, once you have merges lintels, you delete one of the openings the entire lintel will be deleted. At that point you have to delete any opening left in the wall that doesn't have a lintel over it.

Masonry Wall Optimization

The program will optimize masonry walls and lintels based on the required demand forces. The program can optimize:

- Vertical bar/grout spacing for out of plane design.
- Boundary zone widths for in plane flexural design.
- Horizontal bar spacing for in plane shear design.
- Reinforcement bars for lintel flexural design.

Of these optimizations the only one that substantially modifies the stiffness of the wall is the vertical bar/grout spacing. To properly adjust the stiffness requires an iterative solution that updates the stiffness of the model. This includes updating the strength properties of the wall as well as the stiffness. This optimization/iteration can be done automatically (by choosing **Yes**) or can be done manually (by choosing **No**) in the Global Parameters - Solution tab.



To update the stiffness portion of the wall, the program must re-solve your model with these updated stiffnesses as this will change the distribution of forces through the model. By choosing **Yes** you are telling the program to re-solve automatically. Thus, the program will start with it's initial stiffness parameters and solve the model. It will then optimize the wall to meet strength criteria. Another solution will then be run with the new stiffnesses and the program will again optimize the wall to meet strength criteria. This procedure will continue to occur until you reach the number of **Iterations** set or until all wall panel results match those of the previous solution.

By choosing **No** the program will only run the solution once and the results will be based on the original configuration. You can then manually optimize your walls using the Suggested Design spreadsheet.

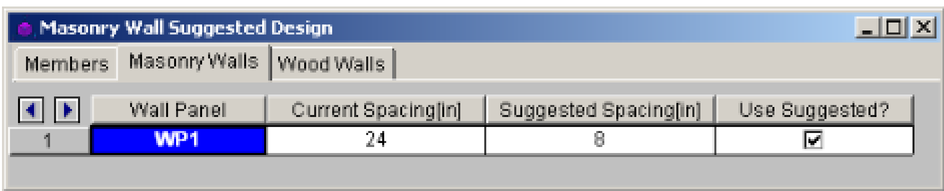
After the solution is run (with or without optimization) the design results are based on the stiffness used in the last iteration (by stating **No** a single iteration is run). The program will then compare the design of the last iteration with the stiffness used in that last iteration. If the two are the same the results shown are the final results. If the two are not the same the program will then provide these two different results in the **Suggested Design** spreadsheet.

The program will always present results in the output that coincide with the stiffness used in the final solution.

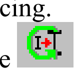
Note:

- The updating of the stiffness for the model is only required for the vertical bar/grout. Thus, boundary zone widths, horizontal shear reinforcement and lintel reinforcement are optimized automatically.

Suggested Design



In the **Suggested Design** spreadsheet you will get a list of wall panels in your model that are not yet fully optimized, showing the bar/grout spacing of the last iteration (Current Spacing) and the program optimized spacing (Suggested Spacing). From here you have the ability to **Use Suggested?** which means that you want to re-run the solution with the Suggested Spacing.

You can choose this for each wall panel individually. Once you have these checkboxes checked appropriately press the  button. After this the stiffness matrix is re-formulated and may cause some redistribution of loads through the model. Because of this the Suggested Spacing may also update and you may need to **Use Suggested?** multiple times to converge on a solution.

Note:

- If the wall does not show up in the Suggested Design spreadsheet then the current wall panel settings used are the optimal ones.
- For more information on wood wall optimization see the Wood Wall - Design topic.
- For more information on member optimization see the Design Optimization topic.
- Concrete walls do not show up here because the reinforcement optimization does not affect the stiffness of the wall.

Lintels

For masonry lintels you must input the dimensions, bar size and number of layers of bars for the lintel, but are given the option of optimizing how many bars are in a given layer. If you provide a max/min number of bars in the Wall Design Rules - Masonry Lintel tab then the program will optimize the number of bars in a layer. Because this is just a change in reinforcement this is an automatic optimization that does not require an iterative solution.

Regions

For masonry regions there are two options for optimizing. For out of plane design the program will optimize the spacing of bars for strength (not deflection) considerations of the wall. For in plane design the boundary zone width will be optimized for strength as well.The spacing of reinforcement/grouting affects the overall stiffness of the wall thus you must iterate your solution to update this spacing. The boundary zone design affects the overall stiffness very little, thus this is an automatic optimization that does not require an iterative solution. If you provide a min/max boundary zone width in the Wall Design Rules - Masonry In tab then the program will optimize the boundary zone width.

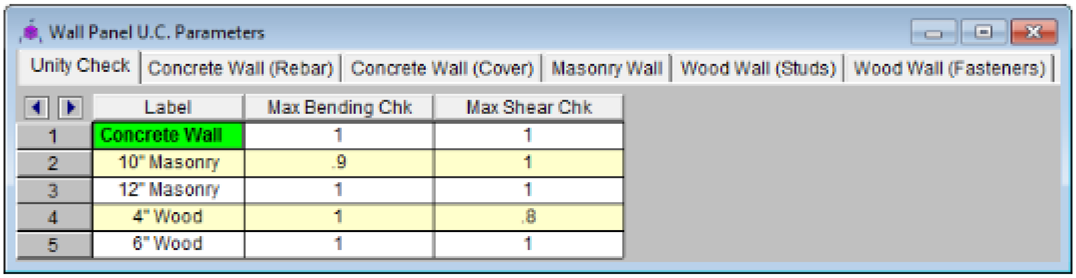
Limitations

- Openings and regions must be input only in RISAFloor if you are using RISA-3D and RISAFloor in tandem.
- For sloped walls due to sloping floors, openings and regions can not be defined in the upper triangular area of the wall panel. These openings are not supported at this time. This will be addressed in a future version.
- For areas of masonry wall panels that are not defined as a region, the stiffness of the wall is assumed to be that of an ungrouted masonry wall and the weight of the wall is assumed to be that of a fully grouted wall.

Masonry Wall - Design Rules

The masonry wall panel element allows you to easily model, analyze and design masonry walls for in plane and out of plane loads. Here we will explain the masonry specific inputs and design considerations. For general wall panel information, see the Wall Panels topic. For information on masonry wall design considerations, see the Masonry Wall - Design topic. For masonry wall results interpretation, see the Masonry Wall Results topic.

Unity Check



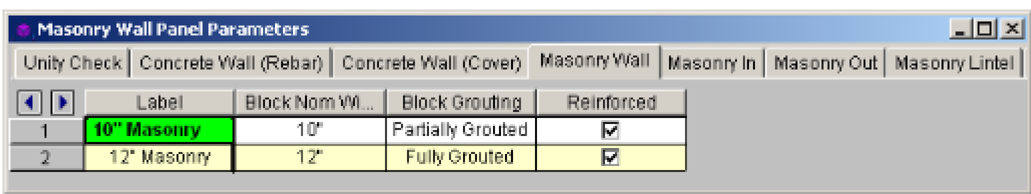
	Label	Max Bending Chk	Max Shear Chk
1	Concrete Wall	1	1
2	10" Masonry	.9	1
3	12" Masonry	1	1
4	4" Wood	1	.8
5	6" Wood	1	1

Setting a maximum Bending Check (Axial & Bending) or a maximum Shear Check controls the rebar which the program chooses for the wall design. A value of 1.0 denotes that the program may choose a rebar layout that is at 100% of capacity.

Note:

- The same unity check parameters are valid for masonry walls as well. However, these parameters are not considered in wood wall design. For wood walls these values are always assumed to equal 1.0.

Masonry Wall General



	Label	Block Nom Wi...	Block Grouting	Reinforced
1	10" Masonry	10"	Partially Grouted	☑
2	12" Masonry	12"	Fully Grouted	☑

Block Nominal Width

This is used to calculate thickness of masonry walls. We use this value along with the value of grout / bar spacing to determine the effective thickness of the wall. The effective thickness is based on table B3 of the Reinforced Masonry Engineering Handbook, by Amrhein, Copyright 1998.

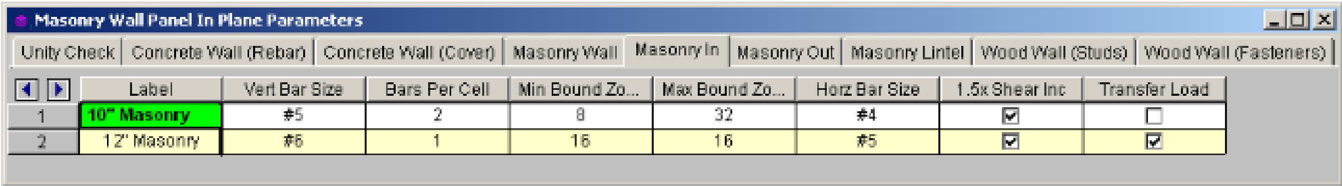
Block Grouting

This option defines how the wall is grouted. If "Partially Grouted" is chosen, then the spacing of grout will be based on the bar spacing defined on the **Masonry Out** tab.

Reinforced

This defines whether the wall is reinforced or not.

Masonry Wall In Plane Design



	Label	Vert Bar Size	Bars Per Cell	Min Bound Zo...	Max Bound Zo...	Horz Bar Size	1.5x Shear Inc	Transfer Load
1	10" Masonry	#5	2	8	32	#4	☑	☐
2	12" Masonry	#6	1	16	16	#5	☑	☑

Vert Bar Size

Allows you to define vertical bar size for the boundary zones.

Bars Per Cell

Allows you to define one or two bars per cell in the boundary zones.

Min/Max Boundary Zone

Allows you to give a maximum and minimum boundary zone width. The program will then design the width based on 8" increments (1/2 of a block length).

Horz Bar Size

Allows you to define horizontal bar size to be used if horizontal reinforcing is required.

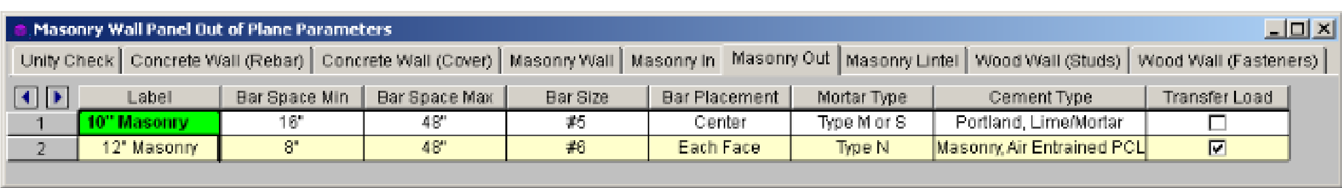
1.5x Shear Inc

This is an option that may be required in high seismic zones. Section 2106.5.1 of the IBC 2006 gives more information on this.

Transfer Load

This option will transfer in plane loads from regions above and below openings into the regions adjacent to the openings.

Masonry Wall Out of Plane Design



	Label	Bar Space Min	Bar Space Max	Bar Size	Bar Placement	Mortar Type	Cement Type	Transfer Load
1	10" Masonry	16"	48"	#5	Center	Type M or S	Portland, Lime/Mortar	☐
2	12" Masonry	8"	48"	#8	Each Face	Type N	Masonry, Air Entrained PCL	☑

Bar Space Min/Max

This allows you to give a maximum and minimum bar/grout spacing. If you give a range between the max and min, then the program will optimize the reinforcement spacing according to strength requirements (code checks).

Bar Size

This is the main vertical bar size that will be used for axial/out of plane design.

Bar Placement

This defines how you want to lay out the reinforcement in your region. **Each Face** will put reinforcement on both faces of a given cell. **Staggered** alternates the bars on either face along the length of the region.

Note:

- When using the staggered option you are selecting to space the bars at each face at double the bar/grout spacing defined above. For example, if you have a staggered spacing at 24" oc you have a bar on the outside face at 48" oc and a bar on the inside face at 48" oc. These bars are staggered, thus you have grout filled cores at 24" oc.

Mortar Type

Allows you to specify the type of mortar in the wall.

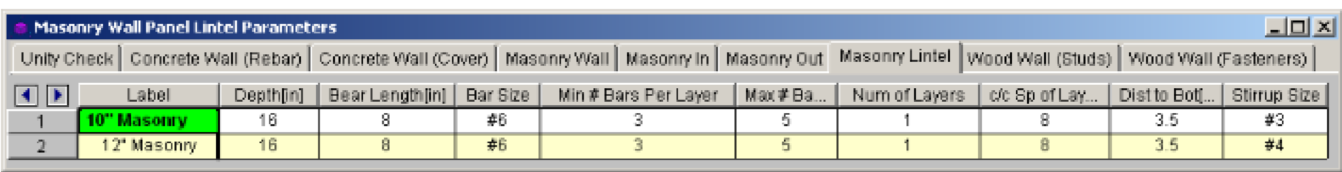
Cement Type

Allows you to specify the type of cement in the wall.

Transfer Load

This option will transfer out of plane loads from regions above and below openings into the regions adjacent to the openings.

Masonry Wall Lintel Design


Masonry Wall Panel Lintel Parameters

Unity Check | Concrete Wall (Rebar) | Concrete Wall (Cover) | Masonry Wall | Masonry In | Masonry Out | Masonry Lintel | Wood Wall (Studs) | Wood Wall (Fasteners)

	Label	Depth[in]	Bear Length[in]	Bar Size	Min # Bars Per Layer	Max # Ba...	Num of Layers	c/c Sp of Lay...	Dist to Bot[...	Stirrup Size
1	10" Masonry	16	8	#6	3	5	1	8	3.5	#3
2	12" Masonry	16	8	#6	3	5	1	8	3.5	#4

Depth

This is the total depth of your lintel.

Bear Length

This is the bearing length at either end of the lintel. This is used to calculate the effective length of the lintel.

Bar Size

This is the reinforcement size for your main reinforcing in the lintel.

Min/Max # Bars Per Layer

This is the number of bars you wish to have in a given layer of reinforcement. If you give a range between the max and min, then the program will optimize the reinforcement spacing based on geometry of the section and also the number of layers that you have defined.

Num of Layers

This is an option if you require multiple layers of reinforcement in the lintel.

c/c Sp of Layers

This is the distance between layers (if there is more than one).

Dist to Bot

This defines the distance from the centerline of the lowest-most bar to the bottom fiber of the lintel.

Stirrup Size

This is the size of stirrup that will be added to the lintel if required.

Masonry Wall Results

Masonry wall results are presented in the wall results spreadsheets and the detail reports. Results are reported on a region by region basis. In addition, the code checks for Lintels spanning openings are reported separately.

Masonry Wall Spreadsheet Results

The information on this spreadsheet is present on three tabs that involve Masonry design: In plane, Out of Plane and Lintel. Each tab gives code checks based on the chosen masonry code and can be used as a summary of all of the panels and panel regions in your model. To get detailed information about each region, you can see the Wall Panel Detail Report.

In Plane

Wall Panel ACI 530-05/08: ASD Masonry Code Checks for Wall Regions (In Plane)

Concrete In | Concrete Out | Masonry In | Masonry Out | Lintel | Wood Wall Axial | Wood Wall In-Plane

	Wall Panel	Region	Combined UC	LC	Shear UC	LC	Fa[ksi]	Fb[ksi]	Fv[ksi]
1	WP6	R1	.136	1	.4673	1	.3503	.5	.0477
2		R2	.0715	1	.4911	1	.372	.5	.0491
3		R3	.1786	1	.5209	1	.3735	.5	.0452
4		R4	.0994	1	.4853	2	.3503	.5	.0496
5	WP7	R1	.0444	2	.3978	1	.3503	.5	.0495

The **In-Plane** results are intended to provide the code checks relevant to shear wall behavior for the wall.

The **Combined UC** entry gives the code check due to axial force plus in-plane bending. Whereas the **Axial UC, Bending UC**, and **Shear UC** will show the relative code checks for the axial check, bending, or shear effects. A value greater 1.0 for any of these values would indicate failure.

The **Fa** or **Pn*Phi** reports the allowable axial stress or axial capacity.

The **Fb** or **Mn*Phi** reports the calculated allowable bending or moment capacity for the region.

The **Fv** or **Vn*Phi** reports the calculated allowable shear stress or Shear Capacity for the region.

Out of Plane

Wall Panel ACI 530-05/08: ASD Masonry Code Checks for Wall Regions (Out of Plane)

Concrete In | Concrete Out | Masonry In | Masonry Out | Lintel | Wood Wall Axial | Wood Wall In-Plane

	Wall Panel	Region	Combined UC	LC	Shear UC	LC	Fa[ksi]	Fb[ksi]	Fv[ksi]
1	WP6	R1	.0318	3	.0003	1	.3503	.5	.0387
2		R2	.0122	3	.0003	1	.372	.5	.0387
3		R3	.0355	1	.0007	1	.3735	.5	.0387
4		R4	.0609	1	.0083	1	.3503	.5	.0387
5	WP7	R1	.0489	3	.0019	1	.3503	.5	.0387

The Out-of-Plane results are intended to provide the code checks relevant to out of plane bending for the wall.

The **Combined UC** entry gives the code check due to axial force plus out of plane bending. Whereas the **Axial UC, Bending UC**, and **Shear UC** will show the relative code checks for the axial check, bending, or shear effects. A value greater 1.0 for any of these values would indicate failure.

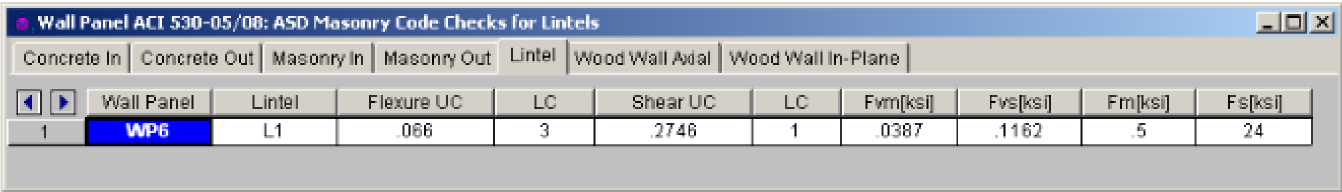
The **Fa** or **Pn*Phi** reports the allowable axial stress or axial capacity.

The **Fb** or **Mn*Phi** reports the calculated allowable bending or moment capacity for the region.

The **Fv** reports the calculated allowable shear stress for the region.

For slender wall design additional checks and analyses are required. These are reported in the region's detail report.

Lintels



The Lintel results give the results for the masonry lintels that span over user defined openings in the wall. They can also be viewed by looking at the detail report associated with each opening.

The **Bending UC** entry gives the code check due pure flexure of the Lintel. Axial force is not considered in this code check at all. The **Shear UC** gives the code check for shear. A value greater 1.0 for either of these values would indicate failure.

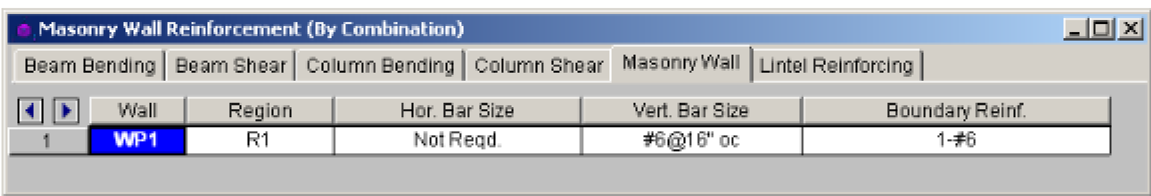
The **Fvm** and **Fvs** or **Vn*Phi** reports the allowable shear stress or shear capacity.

The **Fm** and **Fs** or **Mn*Phi** reports the calculated allowable bending stress or moment capacity for the region.

Concrete Reinforcing Spreadsheet Results

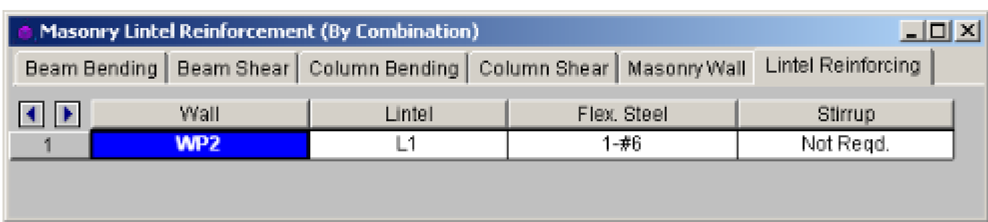
The last two tabs of this spreadsheet contain results for Masonry Wall reinforcement.

Masonry Wall



The Masonry Wall tab displays analysis results for the reinforcement of each region defined in your masonry wall.

Lintel Reinforcing



The Lintel Reinforcing tab displays analysis results for the reinforcement of each lintel defined in your masonry wall.

Masonry Wall Detail Reports

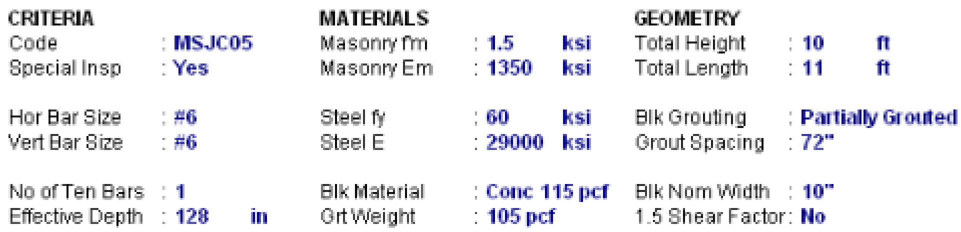
The detail reports show the overall geometry, analysis and design for the individual regions of the wall panel. The report also shows envelope diagrams for the forces and moments in the region.

The report gives detailed information about the user specified regions or lintels. Three basic types of detail reports are provided: in plane, out of plane and lintels. The detail reports are similar, but each contain information specific to one type of code check.

In Plane / Shear Wall Detail Reports

In Plane-Masonry - Input Echo

The top section of the detail report echoes back the input information used in the design of the wall region or lintel. This information is summarized below:



CRITERIA			MATERIALS			GEOMETRY		
Code	: MSJC05		Masonry f'm	: 1.5	ksi	Total Height	: 10	ft
Special Insp	: Yes		Masonry Em	: 1350	ksi	Total Length	: 11	ft
Hor Bar Size	: #6		Steel fy	: 60	ksi	Blk Grouting	: Partially Grouted	
Vert Bar Size	: #6		Steel E	: 29000	ksi	Grout Spacing	: 72"	
No of Ten Bars	: 1		Blk Material	: Conc 115 pcf		Blk Nom Width	: 10"	
Effective Depth	: 128	in	Grt Weight	: 105 pcf		1.5 Shear Factor	: No	

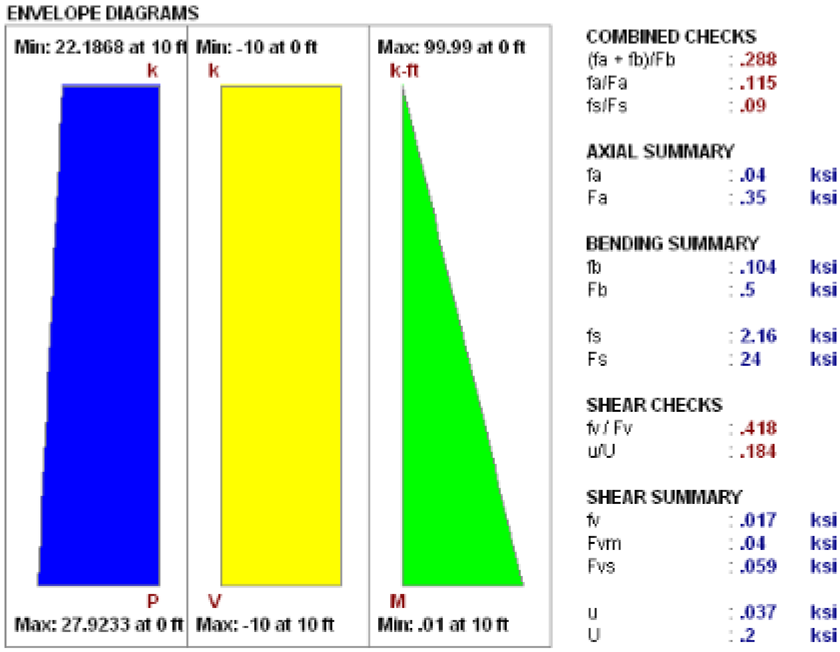
Criteria	Description
Code	Gives the code used to design your wall panel.
Special Inspection	Indicates whether special inspection is required for your wall. Special inspection is normally required for all walls designed to the MSJC / IBC codes. The UBC 1997 is the one code that allows the user to decide whether special inspection is required.
Horizontal Bar Size	Indicates the bar size to be used to resist shear forces.
Vertical Bar Size	Indicates the bar size to be used to resist the tensile stress due to moment.
Number of Tension Bars	Indicates the number of vertical bars present in the boundary region of the wall.
Effective depth	This gives you the distance from the compression face of the wall to the centroid of tension reinforcement.

The **Materials** column can be mostly modified under the Materials button on the Data Entry toolbar. The block material and the grout weight are specified under the Design Rules>>Masonry Wall tab under the Self Weight column.

The **Geometry** column gives the basic geometry of the wall. Block grouting and the grout spacing are specified in the Wall Panel Editor. The block nominal width is input under the Design Rules>>Masonry wall tab. The 1.5 shear factor indicates whether the shear wall will be designed for an additional 1.5 safety factor. This may be required by section 2107.1.7 of the 1997 UBC depending on the seismic zone or seismic design category for the shear wall.

In Plane Masonry - Force Diagrams and Code Check Summary

The next section of the detail report displays the envelope axial shear and moment diagrams as well as a summary of the code checks performed on the shear wall.

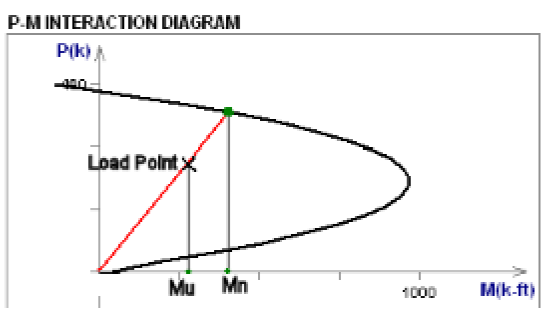


The Combined Check Summary gives maximum overall code checks considering the effects of both bending and axial stresses. This section is not reported for MSJC strength design.

The Axial Summary and Bending Summary give the values used in the axial and flexural code checks at the location which controls the combined check (fa+fb)/Fb for ASD design.

For ASD design, the maximum bending stress in the flexural reinforcement is reported as (fs), and the allowable steel bending stress as (Fs). These values are given for the Load Combination and section that produce the maximum code check (fs/Fs).

For Strength design the axial check is compared to the Pn value for pure axial force only and is based primarily on the h/r ratio. The Mu and Mn values shown in the bending summary for strength design are based on the axial - moment interaction diagram for the wall as shown below.



The Shear Summary reports the maximum shear demand on the beam as fv or Vu. The allowable shear stresses are then reported as (Fvm) and (Fvs), where Fvm is the allowable of the masonry alone and Fvs is the allowable considering the effects of the shear reinforcement.

The bond stress is reported as (u), and the allowable bond stress as (U). This Bond Check is always performed at the section location where the shear check is maximum. This check will not often control the final design, but is reported for completeness. Refer to Section 2107.2.16 of the 1997 UBC for more information on this type of check.

In Plane Masonry - Axial Details

DESIGN DETAILS
AXIAL DETAILS

Max Axial	: 27.923	k
Location	: 0	ft
Load Comb	: 1	
Rad gyration r	: 3.34	in
h/r	: 35.928	
Red Factor R	: .934	

Axial Details	Description
Maximum Axial Force	The maximum axial force in the wall.
Location	The location along the height of the wall which produces the maximum axial force.
Load Combination	The load combination that produced the maximum axial force.
Radius of Gyration (r)	The out of plane radius of gyration for the wall.
h/r	Slenderness ratio of the wall
Reduction Factor R	Used to reduce the allowable stresses for walls where slenderness and P-Delta become a design consideration. For example: R = 1-(h/140r)^2 for the MSJC-05 code if h/r < 99.

In Plane Masonry - Bending Details

BENDING DETAILS

Max Moment	: 0	k-ft
Location	: 0	ft
Load Comb	: N/A	
a	: 0	in
c	: 0	in
d	: 0	in

Ultimate Strength Design

BENDING DETAILS

Max Moment	: 99.99	k-ft
Location	: 0	ft
Load Comb	: 1	
Sect Mod S	: 15237.288	in3
Tension St Asv	: .442	in2
Per of steel p	: .000657796	
k*d	: 75.39	in
j	: .8	

Working Stress Design (ASD)

Bending Details	Description
Maximum Moment	The maximum moment in the wall.
Location	The height of the wall where the maximum bending moment is located.
Load Combination	The load combination that produced the maximum bending moment.
Section Modulus (S)	The uncracked section modulus. This is based on the effective thickness and the length of the wall.
Tension Steel Asv	The area of tension steel in this region of the wall panel.
Percentage of	The reinforcement ratio in this region of the wall panel.

Bending Details	Description
Steel (p)	
k*d	The length of the compression block.
j	The ratio of the distance between the centroid of the compressive and tensile forces (j).
a	Depth of compression block.
c	Depth of Neutral Axis.
d	Depth of section from compression fiber to centroid of tensile reinforcement. Note that for boundary zones, we assume that the tension force acts at the centroid of the steel in the boundary zone and calculation of stresses is based on that. For that reason, long boundary zones are not recommended. For a reference on this, see "Masonry Structures Behavior and Design" by Drysdale, Hamid, and Baker, copyright 1999.

In Plane Masonry - Cracked Section Details

```
CRACKED SECT ANALYSIS
fm = fa + fb     : .144     ksi
C                : 28.493   k
T                : .57      k
```

In ASD design, if the axial compressive stress (fa) is greater than or equal to the bending stress (fb), then minimum jamb steel is required as vertical reinforcement. Since it is not needed to resist the forces, the stress in this steel is assumed to be zero.

If the bending tension stress exceeds the compressive stress, then the wall is assumed to crack and an iterative analysis is performed to determine the section properties of the cracked wall. The maximum masonry stress (fm) is obtained by solving the moment equilibrium equation as a quadratic equation of kd. Each iteration of the steel area is based on the amount of steel needed to create a 0.005 ksi difference in the calculated bending stress (fb). The iterative process is carried out until the calculated value of required reinforcement is less than the reinforcement provided. The final values of fm (fa+fb), C (0.5*fmtkd), T (C-P) are then displayed on the detailed report. A good reference for this iterative procedure is Design of Reinforced Masonry Structures by Narendra Taly and published by McGraw-Hill.

In Plane Masonry - Shear Details

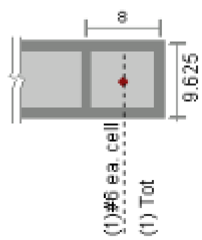
```
SHEAR DETAILS
Max Shear        : .085     k
Location         : 0        ft
Load Comb        : 1

Corresponding M  .2473      k-ft
M / (V*d)        : 1.0916
Shear St Area    : Not Reqd.
Shear Spacing    : N/A
Peri of bars     : N/A
```

Shear Details	Description
Moment	The moment corresponding to the maximum shear check is reported along with the M/Vd ratio.
Shear Steel Area	The shear area per bar (Av) is reported along with its center to center spacing and the perimeter of the reinforcing bars used for the calculation of allowable bond stress. • If the shear can be taken by the masonry block alone, we will report "Not Reqd." • If shear reinforcement is required, we will report "Yes."

Shear Details	Description
	• If the wall maximum shear the wall can take is reached, we will report "Over allowable."
Shear Bar Spacing	The spacing of shear steel in the region of the wall panel.
Perimeter of Bars	Used in the Bond Stress Checks for ASD design.

In Plane Masonry - Cross Section Detailing



NOTE: All units are in "in."

This section of the detail report is meant to provide a visual confirmation to the user of the boundary zone width and reinforcement.

Out of Plane Masonry Detail Reports

Out of Plane Walls - Input Echo

The top section of the detail report echoes back the input information used in the design of the wall region or lintel. This information is summarized below:

CRITERIA		MATERIALS		GEOMETRY	
Code	: MSJC05	Masonry f'm	: 1500 psi	Total Height	: 10 ft
Special Insp	: Yes	Masonry Em	: 1.35e+9psi	Eq Sld Thickness	5.247"
Type of Design	: ASD	Steel fy	: 60000 psi	Blk Grouting	: Partially Grouted
Reinforced	: Yes	Steel E	: 29000 psi	Grt/Bar Spacing	: 72"
Vertical Bar Size	: #6	Blk Material	: Conc 115 pcf	Parapet Height	: 0 ft
End Face Dist	: 4.813 in	Grt Weight	: 105 pcf		

The information in the Criteria section is mostly described in detail in the section on In Plane Detail Report. The End Face Dist is the distance from the edge of wall to centroid of vertical bar.

The **Materials** column can be mostly modified under the Materials button on the Data Entry toolbar. The block material and the grout weight are specified under the Design Rules>Masonry Wall tab under the Self Weight column.

The **Geometry** column gives the basic geometry of the wall. Block grouting and the grout spacing are specified in the Wall Panel Editor. The block nominal width is input under the Design Rules>Masonry wall tab.

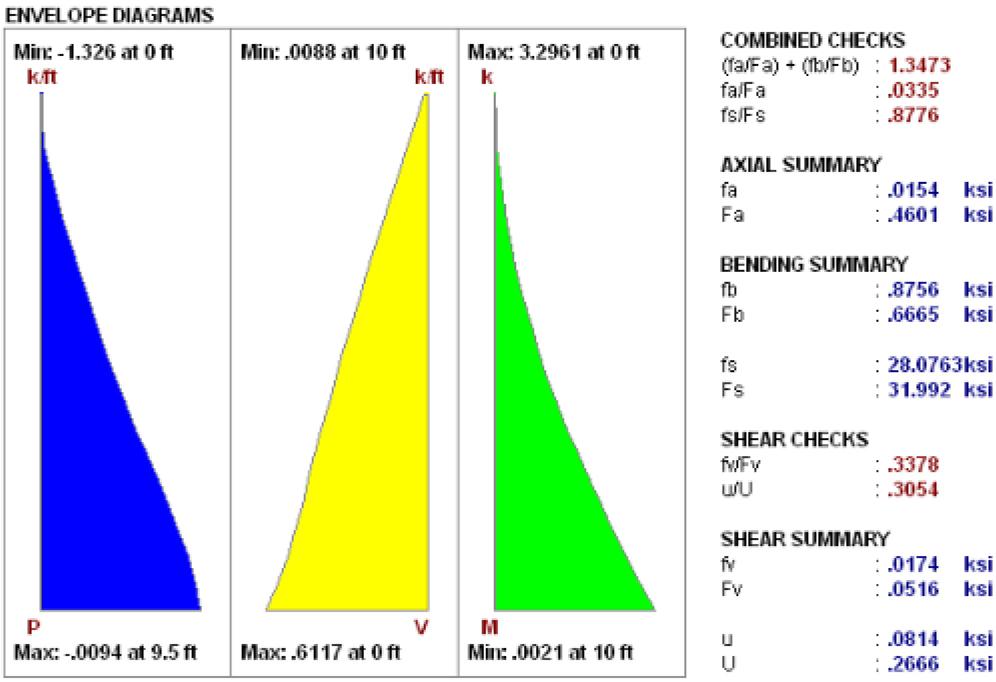
The **Equivalent Solid Thickness** is obtained by dividing the volume of the solid material in the wall by the face area of the wall. This is used in for calculating the axial stress in the wall. A good reference for this value is the Reinforced Masonry Engineering Handbook, 5th ed. by James Amrhein, specifically, Tables B-3a and B-3b.

Out of Plane Walls - Force Diagrams and Code Summary (ASD)

The next section of the detail report displays the envelope axial shear and moment diagrams as well as a summary of the code checks performed on the transverse wall.

Note:

- The forces given in the detail report are given on a per foot basis. Thus these forces are averaged over the width of a given region. For specialty loading conditions, you may need to create much smaller regions to get more accurate local load conditions.



The Combined Check Summary gives maximum overall code checks considering the effects of both bending and axial stresses. This section is not reported for MSJC strength design.

The Axial Summary and Bending Summary give the values used in the axial and flexural code checks at the location which controls the combined check (fa/Fa)+(fb/Fb).

For ASD design, the maximum bending stress in the flexural reinforcement is reported as (fs), and the allowable steel bending stress as (Fs). These values are given for the Load Combination and section that produce the maximum code check (fs/Fs).

The Shear Summary reports the maximum shear demand on the beam as fv or Vu. The allowable shear stresses are then reported as (Fvm) and (Fvs), where Fvm is the allowable of the masonry alone and Fvs is the allowable considering the effects of the shear reinforcement.

The bond stress is reported as (**u**), and the allowable bond stress as (U). This Bond Check is always performed at the section location where the shear check is maximum. This check will not often control the final design, but is reported for completeness. Refer to Section 2107.2.16 of the 1997 UBC for more information on this type of check.

DESIGN DETAILS

AXIAL DETAILS			BENDING DETAILS			SHEAR DETAILS		
Max Axial	: 1.941	k/ft	Max Moment	: .454	k-ft/ft	Max Shear	: .173	k/ft
Location	: 5.25	ft	Location	: 5.25	ft	Location	: 0	ft
Load Comb	: 1		Load Comb	: 1		Load Comb	: 1	
Rad gyration r	: 2.19	in	k	: .406		Width for Shear	: 16	in
h/r	: 57.534		d	: 3.813	in			
			j	: .865				

Design Details	Description
Radius of Gyration (r)	The out of plane radius of gyration for the wall.
h/r	Slenderness ratio of the wall.
k	This value multiplied by d gives the depth to the neutral axis.
d	The depth between the extreme face of masonry in compression and the center of the tension reinf.
j	This value multiplied by d gives the moment arm between the compression and tension resultants.
Width for Shear	This is the value used to calculate the shear capacity. For fully-grouted walls, this is simply the center to center spacing of vertical reinforcement. For partially-grouted walls, this value is conservatively taken as the width of the grouted cell plus the thickness of the web and end wall on either side. An example of this can be seen in Example 6.3 (p6.65) of "Design of Reinforced Masonry Structures" by Taly, copyright 2001.

T-Section Analysis

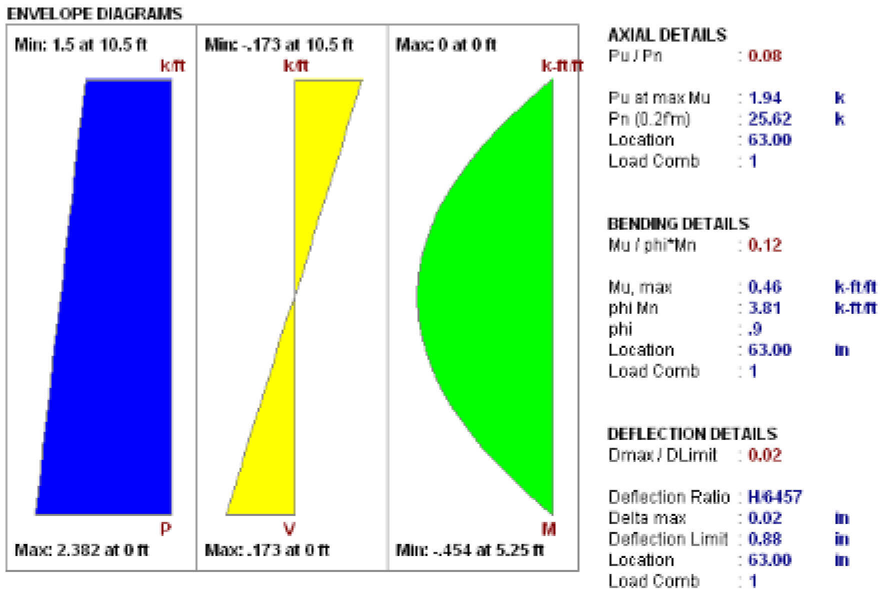
If you are using a partially grouted wall where the neutral axis passes through the webs of your masonry, then RISA will do a T-section analysis to define the section properties. We use a similar analysis as if you were doing a t-beam analysis for a concrete tee section. For more information on this, see "Design of Reinforced Masonry Structures" by Narendra Taly, copyright 2001, example 6.3, P6.61.

T-SECTION ANALYSIS

kd	: 1.609	in
Flange thickness	1.375	in
Web width	: 8.5	in

Out of Plane Walls - Force Diagrams and Code Summary (Strength)

The next section of the detail report displays the envelope axial shear and moment diagrams as well as a summary of the code checks performed on the out-of-plane loaded slender wall.



The **Axial Details** provide the axial code check for pure compression forces. The Pn (0.2f'm) value reported is the maximum allowable axial force based on Section 3.3.5.4 of MSJC 05.

The **Bending Details** are explained below:

- Mu / phi*Mn represents the ratio of applied moment to moment capacity of the wall
- Mu, max is the out-of-plane bending moment at the controlling location of the wall.
- phi Mn is the out-of-plane moment capacity of the wall at the controlling location. It is calculated per Section 3.3.5.4 of the MSJC-05. For partially grouted walls an effective width is calculated for b, per Section 2.3.3.3.
- phi is the strength reduction factor specified in Section 3.1.4 of the MSJC-05
- Location is the elevation of the wall which resulted in the highest (Mu / phi*Mn) ratio. It is the location at which Mu and Pu are used to calculate phi Mn.
- Load Combination is the load combination which resulted in an Mu and Pu which yielded the highest (Mu / phi*Mn) ratio.

Deflection Details

Dmax/Dlimit is a code check result. Thus, if the value is larger than one, your wall fails deflection criteria. **Delta max** is the calculated deflecting using the iterative slender wall design procedure detailed below. The **Delta max** is the allowable deflection from Section 3.3.5.5 from the MSJC 05.

Note:

- Both the Service and the Masonry flags must be checked on the Design tab of the Load Combinations for the deflections to be checked for that load combination.
- The forces in the region are given on a per foot basis and are an average of the forces over the width of the region.

Sectional Properties (Strength)

SECTIONAL PROPERTIES

ALL RESULTS PER C/C OF REINFORCEMENT

Total Width	: 16	in	Agross	: 101.6	in2	As	: .442	in2
Eff Width	: 16	in	Igross	: 985.794	in4	Ase	: .442	in2
Flange thick tf	: 1.375	in	Crack Moment Mcr	: 1.965	k-ft	Rho Provided (%)	: .006	
Effective Thick te	: 7.163	in	Icracked	: 112.595	in4	Rho Maximum (%)	: .005	

The center to center distance between reinforcing (Total width) and the effective width (which accounts for partially grouted walls) are both reported in this section. The flange thickness (tf) refers to the thickness of the face shell whereas the effective thickness (te) refers to the overall effective depth of the block.

The gross area and moment of inertia are reported along with the cracking moment and the cracked moment of inertia.

The area of steel (As) and the effective area of steel (Ase) are reported, with the effective area based upon As + Pu/fy per the design procedure given in the code.

The gross steel ratio (rho gross) is calculated as the area of steel divided by the total area of the section.

The structural steel ratio (rho struct) is calculated as the area of steel divided by the total width times the effective depth (As/bd). This is limited to a maximum value of 0.5 * rho balanced per the UBC-97 Section 2108.2.3.7. This provision was implemented in future codes as well.

The Sectional Properties gives a number of the properties used during the Iterative Slender Wall Design Procedure. All of these properties are reported based on a section of wall equal to the center to center spacing of the reinforcement.

Slender Wall Design Procedure

The P-Delta analysis that RISA uses during the Finite Element solution does not account for P-Delta effects on plate elements or wall panels themselves. Instead, the program uses a generalized design procedure very similar to that provided in the UBC and MSJC slender wall design provisions.

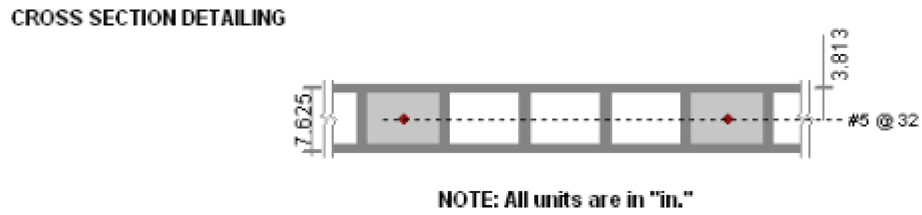
Moment and Deflection Amplification

1. The deflections and rotations at both supported ends of the wall are taken directly from the FEM solution.
2. The program then splits the wall into 20 segments and uses a moment area method to come us with elastic moments and deflections at each of the end points of those segments.
3. The program adds in the P-Delta moments based on these original elastic moments and deflections.
4. The program then amplifies the deflection for each section based on the ratio between the average section moment and the cracking moment of the wall. This is very similar to the deflection amplification equations in the MSJC and UBC provisions for slender walls.
5. New deflections and moments are calculated and compared to the results from the previous iteration. The program iterates the procedure until the results have converged within 0.5% of the previous iteration, or until it has iterated 100 times.

Note

- This procedure is intended only to determine the localized P-Little Delta amplification of Moments and Deflections within the wall itself. This procedure does NOT amplify the shear forces, nor does it contribute to the "leaning wall" effect by pushing on the rest of the structure.

Out of Plane Masonry - Cross Section Detailing



This section of the detail report is meant to provide a visual confirmation to the user of the spacing and location of reinforcement.

Masonry Detail Reports - Lintels

Lintels - Criteria / Materials / Geometry

The first section of the detail report echoes back the basic input parameters (Criteria, Materials, Geometry) entered by the user. An example is shown below:

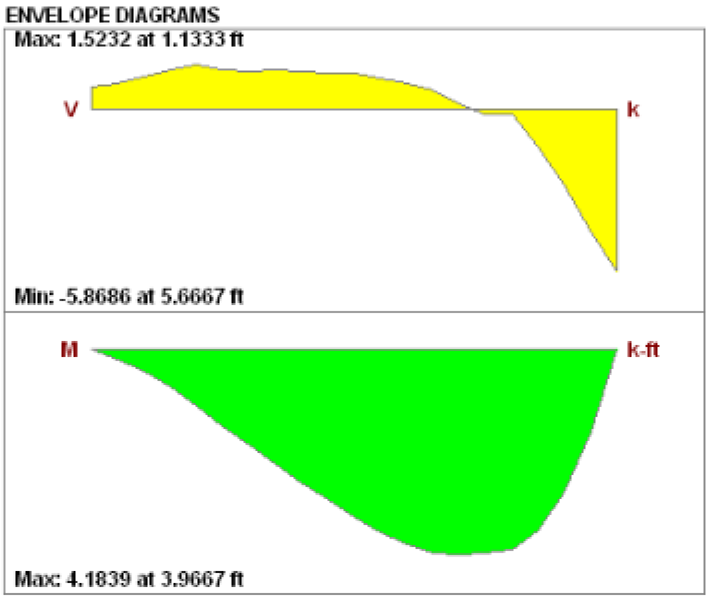
CRITERIA		MATERIALS			GEOMETRY		
Code	: MSJC05	Masonry f'm	: 1.5	ksi	Top of Wall	: 5.5	ft
Special Insp	: Yes	Masonry Em	: 1350	ksi			
Type of Design	: Strength	Steel fy	: 60	ksi	Eff Length	: 5.667	ft
		Steel E	: 29000	ksi	Eff Width	: 9.625	in
Stirrup Size	: #4	Beam Dead Wt	: .084	ksf	Eff depth	: 12.5	in
Flex Steel	: 1-#6	Wall Dead Wt	: .084	ksf	Total Depth	: 16	in

The geometry portion and the dead weight of the wall warrant further explanation

Attribute	Description
Beam Dead Weight	This is based on the user defined density for the Lintel / Opening.
Wall Dead Weight	This is based on the wall self weight or the self weight of the region immediately above the opening.
Top of Wall	This is the distance from the bottom of the lintel to the top of the wall and is used in the calculation of the arching action of the lintel loads.
Effective Length	The Center to Center distance between beam supports. Assumed to be length of the opening plus half of the bearing area on each side.
Effective Width	The thickness of the block used to define the lintel.
Effective depth	Equivalent to the "d" distance, the distance from the effective compression face to the centerline of the tension reinforcing.
Total Depth	This is the total depth of the fully grouted portion of the lintel.

Lintel Detail Reports - Diagrams and Code Check Summary

The next section of the detail report provides the envelope shear and moment diagrams as well as a summary of the code checks for the Lintel.



SHEAR SUMMARY

Vu/(Vn*phi)	: .46	
Vu	: 5.869	k
Vn,total	: 15.956	k
Vn,masonry	: 15.956	k
Vn,steel	: 0	k
phi	: .8	

BENDING SUMMARY

Mu/(Mn*phi)	: .185	
Mu	: 4.184	k-ft
Mn	: 25.077	k-ft
Mn,masonry	: 3.802	k-ft
Mn,steel	: 21.275	k-ft
phi	: .9	

Lintel Detail Reports - Design Details

The next section of the detail report gives further details for the design checks performed on the lintel.

DESIGN DETAILS

BENDING DETAILS

Max Moment	: 4.184	k-ft	Steel Area As	: .442	in2
Location	: 3.967	ft	Per of steel p	: .004	
Load Comb	: 1				

COMPRESSIVE STRESS BLOCK DETAILS

Depth of Block	: 2.295	in	beta 1	: .8	in
N.A. Location	: 2.869	in	alpha	: .8	in

Mm	: 9.1057	k-ft	k	: .3261
Ms	: 9.8442	k-ft	j	: .8913

SHEAR DETAILS

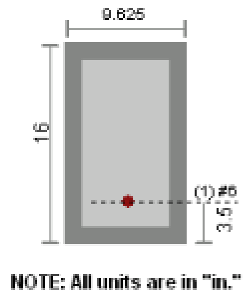
Max Shear	: 5.869	k
Location	: 5.667	ft
Load Comb	: 1	
Tie Spacing	: Ties Not Reqd	

This **Bending Details** show the magnitude, location and load combination that correspond to the maximum moment used in the design as well as the area of flexural steel required (As) and the steel ratio (rho).

The **Compressive Stress Block Details** show the parameters used in calculating the strength of the lintel.

The **Shear Details** show the magnitude, location and load combination that correspond to the maximum shear used in the design as well as the required spacing of ties (if applicable).

Lintel Detail Reports - Cross Section Detailing



This section of the detail report is meant to provide a visual confirmation to the user of the basic geometry and reinforcement provided in the lintel.

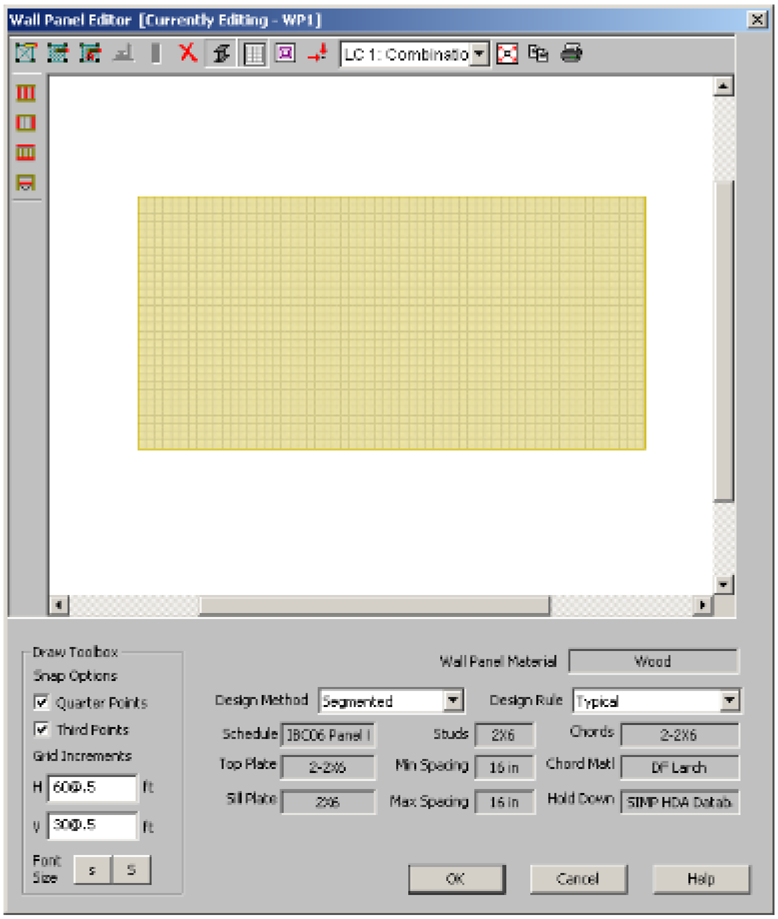
Wood Wall - Design

The wood wall panel element allows you to easily model, analyze and design wood walls for in plane loads. Here we will explain the wood specific inputs and design considerations. For general wall panel information, see the [Wall Panels] topic. For wood wall results interpretation, see the [Wood Wall Results] topic.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Wood Walls**.

Wood Wall Input

The **Wall Panel Editor** gives some specific information and options for modeling/analysis of wood walls.

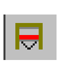


Wood View Controls

 **Toggle Wall Studs Display** allows you to turn the display of the studs on and off.

 **Toggle Wall Chords Display** allows you to turn the display of wall panel region chords on and off.

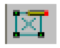
**Toggle Top/Sill Plate Display** allows you to turn the display of the top/sill plates on and off.

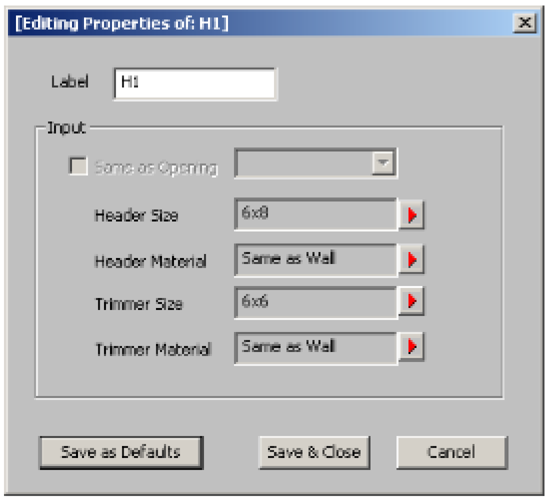
 **Toggle Opening Headers Display** allows you to turn the display of headers on and off.

Design Rules

You must set up design rules for the stud/chord sizes, as well as make database selection for shear panels and hold-downs. This is done in the Design Rules spreadsheet in the**Wood Wall (Studs)** and **Wood Wall (Fasteners)**tabs. See the Wood Wall - Design Rules topic for more information.

Creating Openings in Wood Walls with Headers

Within the **Wall Panel Editor**, you have the option of adding rectangular openings to wood wall panels. To draw an opening, select the **Create New Openings**  button and then select two nodes or grid intersections which make up the two diagonal corners of your opening. When an opening is drawn a header beam is automatically created above the opening. To view or edit the properties of a header beam, double-click inside the boundary of the drawn opening. This will bring up the Editing Properties window for that particular header beam.



Label - This defines the name of this header and shows up in the results output for this header.

Same as Opening - This allows you to define this header with the same properties as another header already defined in this wall panel.

Header Size - This allows you to define the size of the header member. The program will do design checks based on this size.

Header Material - This allows you to define the header material.

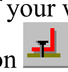
Trimmer Size - This allows you to define the trimmer size for this opening. This value is only used for the material take off for the wall.

Trimmer Material - This allows you to define the material for the trimmers.

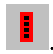
Hold Downs and Straps

Hold-downs and straps are automatically added to your walls in their required locations. For a **Segmented** design, you must have hold downs at the bottom corners of each of your design segments. For **Perforated** and **Force Transfer Around Openings (FTAO)**, hold downs are only allowed at the two far corners of the wall panel. The program will not permit the drawing of hold downs at locations where hold downs are not allowed.

If there are custom locations that you want to add hold-downs, you can do this within the **Wall Panel Editor**.

Hold downs represent the anchorage of your wall to the foundation. To add hold downs to the base of your wall, first select the **Add New Wood Hold Downs** button .

Hold downs must be added after regions are created and can only be added at the corners of regions. Hold down requirements depend on the type of wall design you are performing.

Straps represent the anchorage of the current wall panel to a wall panel below. To add straps to the base of your wall, first select the Add New Wood Straps button .

Straps also can only be added after regions are created and can only added at the corners of regions. Strap requirements also follow the same logic as hold downs as to where they must be defined as far as regions are concerned.

Note:

- All boundary conditions for wall panels should be defined in the wall panel editor. Adding external boundary conditions can create problems.
- The locations of hold-downs and straps define where the program will calculate tension forces in your walls.
- If you have applied your hold downs for the Segmented design with openings in your wall, then hold downs will be required at the interior of the wall panel. However, running Perforated or FTAO does not require hold downs at the interior of the wall panel. Thus, if you toggle between Segmented to Perforated or FTAO, then the hold downs you drew will be removed at the interior of the walls. If you switch back to Segmented the interior hold downs will come back again.

The output for straps and hold downs will show up on the detail report for the wall panel. More information on this can be found in the Wood Wall Results topic.

General Requirements for Shear Walls

The design of wood shear walls within the framework of the NDS requires that many criteria are satisfied before a wall can be considered adequate. For RISA to work within this framework we require that certain modeling practices be followed. Outlined below are many general wall modeling practices and limitations. Also included are specific requirements for each of the three design procedures for wood wall design with openings: segmented, force transfer around openings and perforated.

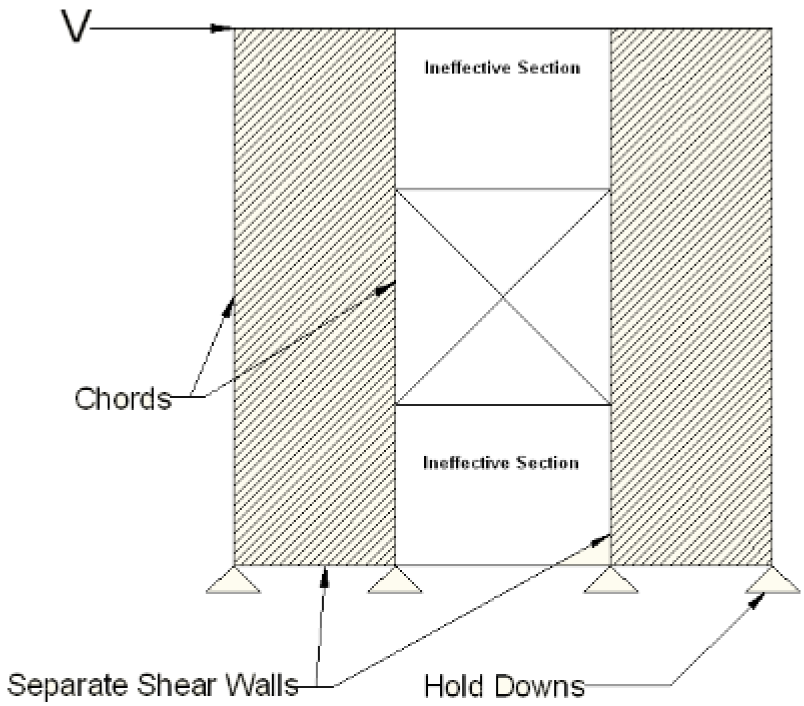
The three different types of shear walls are defined in Section 4.3 of the NDS *Special Design Provisions for Wind and Seismic*.

Segmented Method

Where there is a wall panel with openings, the area above and below the openings is disregarded and the wall is designed as being made up of separate, smaller shear walls.

Like all wall panels, the segmented wood wall is broken into a series of meshed plate elements to represent the overall wall. The portions of the segmented shear wall that are considered "ineffective" in resisting shear are modeled with a plate elements that have a significantly reduced shear stiffness so that they will not receive any significant moment or shear from the FEM analysis.

See the diagram below for more information:



In addition, the out of plane stiffness and in plane stiffnesses of the segmented wood wall are modeled separately based on different assumed plate thicknesses. This is done to insure that the shear stiffness is based entirely on the properties of the sheathing and is not influenced by the out-of-plane stiffness of the wall studs.

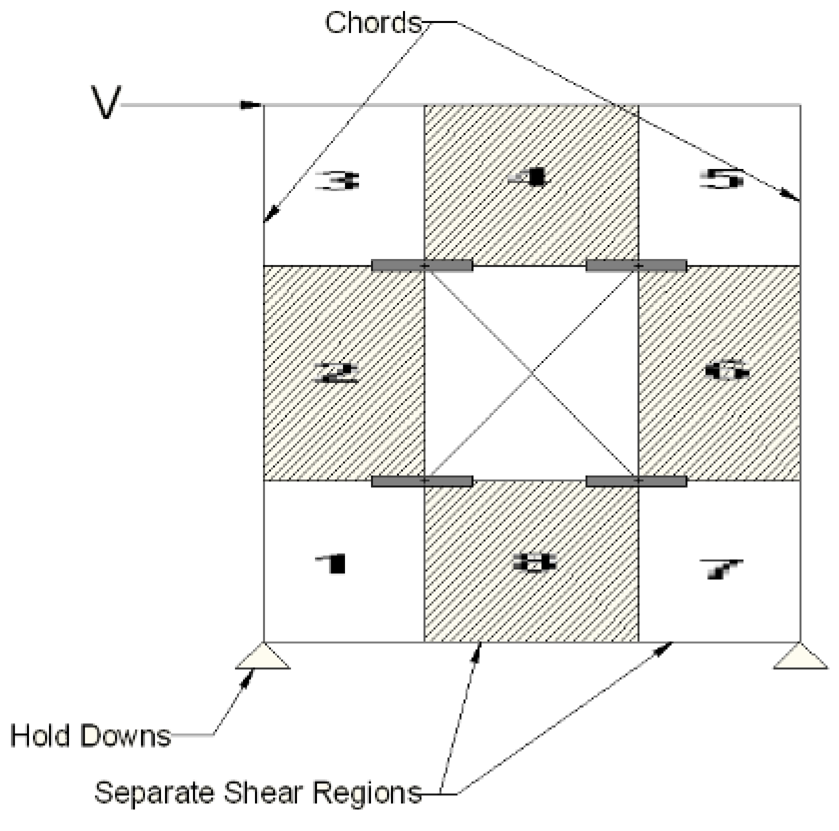
Note:

- If you have several stacked Segmented wall panels with misaligned openings, you will receive the Warning Message shown below upon solution. This message means that RISA-3D has assumed that the strap force from the above wall panel will be spread out across the region directly under it. Therefore, you need to be aware of this assumption and detail the wall panel
accordingly.

- A shear panel design will be chosen for the worst-case region in a segmented wall. That panel will then be used for all regions in that wall. The worst-case region is the one that has the highest Shear UC value.

Force Transfer Around Openings Method

This method is based on a rational analysis of the wall. The assumption being that straps and blocking can added at the corners of the openings to transfer the sheathing forces across these joints. This method essentially allows you to use the entire area of the wall (minus the opening) to resist the shear in the wall.



The basic assumptions made in the shear wall analysis are the following:

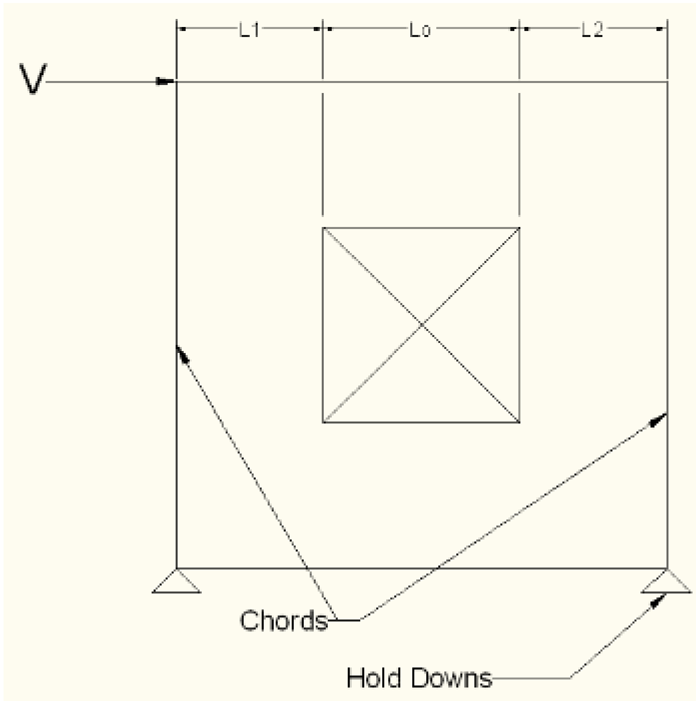
- The sheathing resists the shear forces. The average shear force in each block of the wall (numbered 1~8 as shown in the image above) is used as the controlling shear force in that location. The maximum shear in each of these locations will control the design of the wall. The program uses an area weighted average of the Fxy plate forces to determine the average shear for each block.
- The moment at the edge of each block that is above or below an opening is assumed to be transmitted across the opening interface by horizontal tension straps or compression blocks as shown in the image above. The required force is reported to the user, but the design and length of these elements is left to the engineer.
- The moment at the edge of each block that is to the right or left of an opening is assumed to be transmitted across the opening by tension straps or compression blocks. Since it is likely that the sheathing and king studs will be capable of transmitting these forces, these elements are not shown in the image above. However, these forces are reported so that the design of the studs and sheathing in these regions can be checked by the engineer to consider these effects.

Note:

- The program is limited in the automatic generation of regions for walls with multiple openings that are not aligned. Therefore, it is recommended that complex walls will multiple openings be simplified based on engineering judgement to facilitate easier detailing of the force transfer around these openings.

Perforated Method

This method for design of wood shear walls with openings may end up being the most cost effective. It only requires hold downs at the corners of the wall, yet it does not require straps or blocking around the openings. A perforated shear wall design approach is, however, subject to a number of code constraints about when it can be used.



The basic design procedure for perforated walls is to essentially ignore the portions of the wall that do not have full height sheathing and treat the wall instead as a significantly shorter wall. This amplifies the chord and hold down design forces significantly while at the same time increasing the design unit shear as shown in the equations below:

$$T := \frac{V \cdot h}{C_o \cdot \Sigma L_i} \qquad v_{max} := \frac{V}{C_o \cdot \Sigma L_i}$$

Where:

$$\Sigma L_i := L1 + L2$$

Note:

- The perforated method of design also has many caveats that are given in Section 4.3.5.3 of the NDS 2005 *Special Design Provisions for Wind & Seismic*. The program will not allow the design of wall panels that do not follow these provisions.
- For multi-story perforated shear walls, the amplified hold down forces required per the code become difficult to interpret for the lower walls. Therefore, when RISA calculates the hold down forces for the lower wall it assumes that the reduction coefficients for the upper wall are identical to the values for the lower wall. This will result in conservative hold down forces when the lower wall has more openings than the upper wall. But, it may be un-conservative for situations where this is not the case. This assumption does NOT affect the shear design of either wall, nor does it affect the strap force calculations in the upper wall.

Shear Capacity Adjustment Factor, Co:

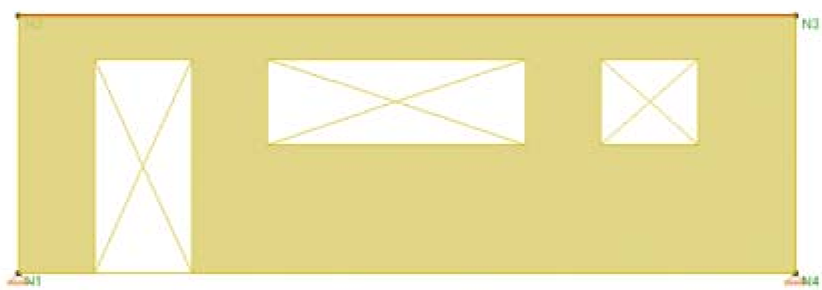
The NDS *Special Design Provisions for Wind & Seismic* lists Effective Shear Capacity Ratio (Co) values that are used in calculating the nominal shear capacity of perforated shear walls. Because the tabular values are limited to wall heights of 8' & 10', RISA instead uses the equation (from the 2008 NDS SDPWS) to calculate the Co factor for any height wall.

$$Co = \left(\frac{r}{3 - (2 * r)}\right) * \frac{Ltot}{\Sigma Li}$$

Where,

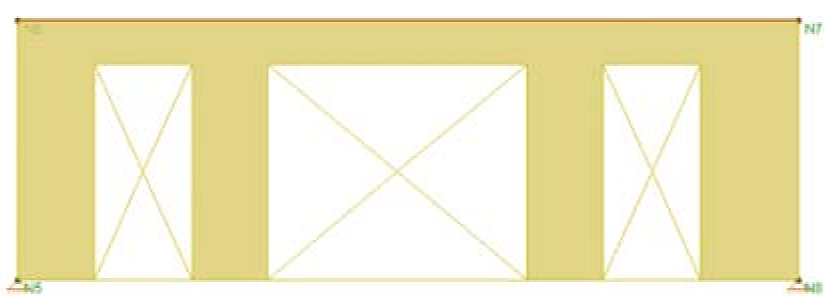
$$r = \frac{1}{1 + \left(\frac{Ao}{h * \Sigma Li}\right)}$$

When using these equations, RISA takes Ao as the true area of the openings. However, Table 4.3.3.4 of the 2005 NDS SDPWS references Co values based on all opening heights equal to the maximum opening height. Therefore if you want the program to calculate Co equal to that in Table 4.3.3.4 of the NDS, you must draw all openings as equal to the maximum height. Please see the image below for reference.



Co = 0.84

vs.



Co = 0.53

(Full height openings assumption- matches NDS Table 4.3.3.4)

Note:

- When the assumption is made that all opening heights are equal to the maximum opening height then the equation produces values of Co that are within 1% for all the values shown in the NDS table.

C_o Limitation for Stacked Wall Panels

For wall panels that are stacked on one another, each wall panel has its own C_o value. These C_o values will be different if the openings in each wall are different. This is the source of a limitation in the program.

The program determines the design forces in each wall panel separately using the finite element solution. We then take those design forces and factor them for the C_o values. The problem is that the design forces for the bottom wall are affected by the C_o value for the upper wall. Thus, the forces that come down on a lower wall from an upper wall have been factored for the $C_o$from the upper wall. RISA-3D does not consider this. The program takes the forces from the finite element solution and uses the C_o value only for the wall in question.

Thus, the highest level in a stacked wall configuration will always use C_o in a correct manner. The lower wall(s), however, will be conservative if they have more openings than the upper wall(s) and can be unconservative if the upper wall(s) have more openings than the lower wall(s).

This limitation must be considered when designing Perforated walls that are stacked.

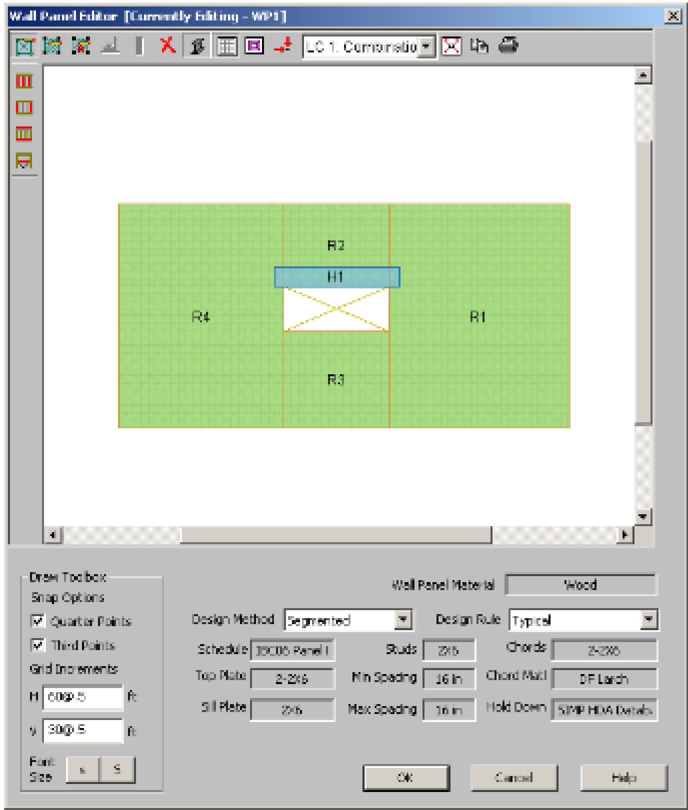
General Program Functionality / Limitations

RISAFloor and RISA-3D Interaction

When using RISAFloor and RISA-3D in combination, the interface transitions nicely between the two programs. Here is a quick walk through of this interaction.

Input Interface

1. Model the entire building (gravity and lateral members) within RISAFloor. Be sure to model all openings and regions for all of the wall panels in the model.



Note:

- You can not modify your openings or regions in RISA-3D. All region and opening modifications must be taken back to RISAFloor to be done.

2. Add loading and solve the model.
3. Take the model into RISA-3D via the Director tool.
4. Once in RISA-3D you must add your hold downs and straps to your wall panels.

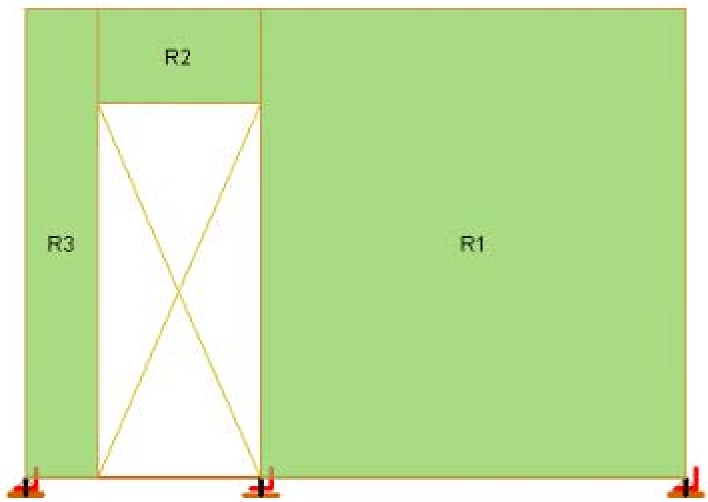
Note:

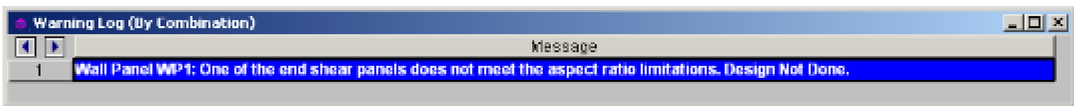
- Hold downs and straps can not be added to wall panels in RISAFloor.
- Hold downs are only allowed to be added to the lower corners of the wall panel for the Perforated or Force Transfer Around Openings design methods.
- Hold downs are required at the corners of all full height regions in the wall panel for the Segmented design method.
- The Design Rule, Design Method and SSAF can be changed in either program at any time.

Modeling Tips- Aspect Ratio

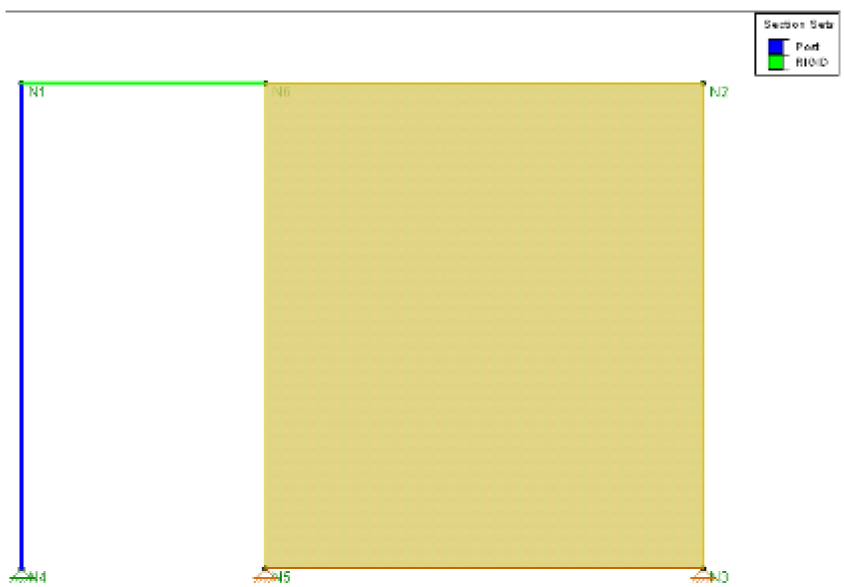
Due to the limitations of aspect ratios for the three design methods, there may be some wall panel models that cannot be designed by RISA-3D. In this case, there are work-around procedures that allow you to modify your model so that it will give a design.

For instance, if your wall panel has an opening very near an edge. This often creates a region (R3 in the image below) whose aspect ratio exceeds the limits of the NDS requirements.





Because you know this region is failing, you can simply omit it from your RISA model in order to obtain the design for the larger, controlling region.



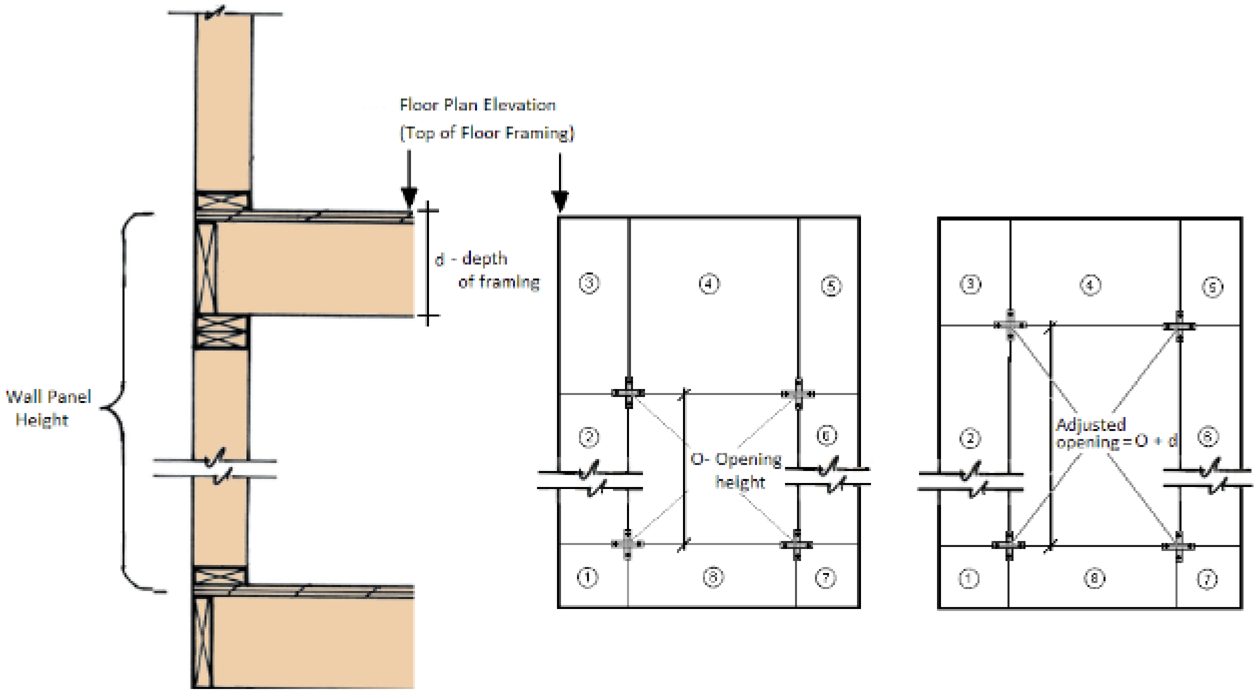
This leaves your model with regions all meeting the aspect ratio limitations, therefore leaving you with a model that RISA-3D can design.

Modeling Tips- Platform Framing

The wood wall height in RISA-3D is measured from the bottom of the sill plate to the top of the floor framing as shown below. However, platform framing causes the wall height to be significantly shorter then a RISA-3D model would represent.

The design method FTAO is the only method that will cause significant problems because the region above openings is assumed to be much larger than it is built. In order to adjust for this framing depth, you can adjust your opening height to include the depth of the floor framing. This will reduce the portion of the wall above the opening thus reducing the amount of area to transfer shear forces.

In Segmented and Perforated design methods, the portion above and below the opening are not used to transfer shear forces.



Limitations for Hold-downs / Straps (Including deformation)

- The program does not make an adjustment to the chord force calculation based on the eccentricities of the chord and hold-down. The program considers the hold down and the chord to be at the same location in plan.
- The program does not currently have a database for continuous tie rod hold-downs. This will likely be added in a future release. However, the design results that are presented for each floor are intended to provide the type of information necessary for the design of these types of hold downs.
- The $d_a$values (deflection at peak load) from our hold-down database are based entirely on the manufacturers listed values. These do not include any allowance for shrinkage (which is often as much as 1/4" or more per floor), or crushing of the sill plate.
- There is not currently a d_a value used in the calculation for strap deformation for upper floor levels. This will be implemented when a strap database is added into the program.

Automatic Boundary Conditions

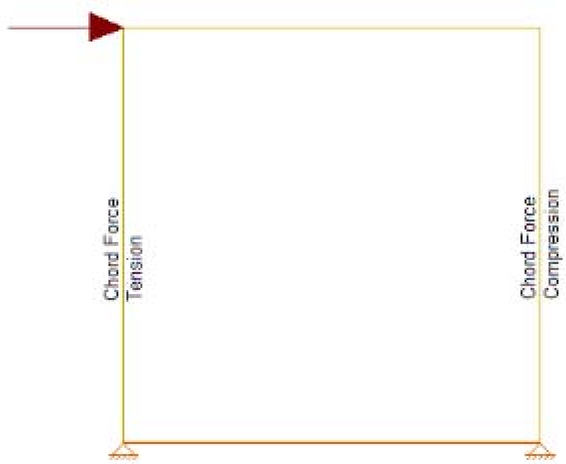
- In RISA-3D, if no boundary conditions/hold downs are defined for wood wall panels at the lowest level of the structure, the program will automatically create hold downs at the corners of the wall panel. If you do not want the hold downs to be automatically created, define a "free" boundary condition at the base of the wall panel in the wall panel editor.

Deflection

- There is currently no code check for drift or deflection for shear wall panels.

Chord Design

The chord design is based on forces that are calculated differently for Compression versus Tension. The tension chord force is calculated including the dead load stabilizing moment as per Section 4.3.6.4.2 of the NDS 2005 *Special Design Provisions of Wind & Seismic*. The compression chord force includes the only the tributary area of one stud spacing in the compression force. For Segmented design, the chord forces are found based on each region, and in FTAO and Perforated design methods the chord forces are determined for the entire wall.



$$\text{Chord Force}_{\text{Tension}} = \frac{M}{L} - \frac{P}{2}$$

$$\text{Chord Force}_{\text{Compression}} = \frac{M}{L} + \frac{P}{\#\ \text{Studs}}$$

M = Max Moment in the wall

L = length of the wall

P = Axial force in the wall

- For Perforated Design, the term M/L found in the above equation is replaced with $Vh/C_o\Sigma L_i$ as indicated in the Equation 4.3-5 of the NDS 2005 *Special Design Provisions for Wind & Seismic*.
- RISA-3D models the shear walls using only the sheathing. The vertical resistance occurs only at the tension and compression chords. Thus, if two wall panels are stacked on top of each other, the load transfer will only happen at the chord locations. Therefore, the lateral analysis should agree very well with hand calculations. However, it also means that gravity load design may be more appropriate in RISAFloor.

Stud Design

Studs are only designed for load combinations which do not contain a wind or seismic load. The maximum axial load, determined as an envelope force from all of the "gravity" load combinations which have been solved, is determined for each region.

The compression capacity for the specified stud size is calculated with the assumption that the stud is fully braced against buckling about its minor axis (within the plane of the wall). This is because the blocking and sheathing are assumed to provide this bracing. The unbraced length for major axis buckling is taken as the wall height, minus the thickness of the top and sill plates.

The program divides the region axial force by the number of studs that would be present in that region for a given stud spacing. An optimal stud spacing is then selected based on the stud capacity, and the parameters defined in the Design Rules.

Force Distribution

- The lateral force distribution between piers is based on the relative stiffness of the sheathing, not on the length of the shear wall. For example, if you have an 8 foot wall and a 4 foot wall, the 8 foot wall will take more than 8/(8+4)*100% of the force. The moment of inertia in the 8 foot wall will allow for a larger proportion of load to go into that pier.

Shear Capacity Adjustment Factors

40% Increase Factor Wind Load Cases

The allowable shear stress values tabulated in Appendix F are intended to be the allowable shear for seismic loads. The solution tab of the global parameters has a checkbox which will automate the 40% shear capacity increase for any load combinations that include Wind Loads.

Allowable Stress Reductions for slender wall segments

The program will modify the allowable stress of the wall based on the 2b/h adjustment factor described in the NDS. This adjustment factor is only applied to the wall for load combinations which include seismic loads. This adjustment factor only affects walls with an aspect ratio between 2.0 and 3.5.

Specific Gravity Adjustment Factor

The NDS defines an adjustment factor associated with using stud material that is less dense than Douglas-Fir-Larch or Southern Pine. The program does NOT automatically account for this factor in the design of Wood Shear Walls.

Unblocked Shear Wall Adjustment Factor

Section 4.3.3.2 of the 2008 NDS Special Design Provisions for Wind and Seismic is ignored in RISA-3D. The program will always assume that the sheathing panel is blocked.

Uplift Limitation

Section 4.4 of the 2008 NDS Special Design Provisions for Wind and Seismic is ignored in RISA-3D. The program currently does not design wall panels for uplift forces.

Stiffness Adjustment Factors

Shear Stiffness Adjustment Factor (to change FEM results)

This stiffness adjustment factor is set on the Wall Panels spreadsheet and is intended as a way to force the FEM stiffness of the wall to more closely resemble the stiffness from the APA / NDS three term deflection equation.

This adjustment affects the stiffness of the entire wall. Therefore, for segmented walls the engineer may be forced to model the piers separately if they need to adjust the pier stiffnesses independently. In a future release, this may become an automated factor.

Green Lumber

The Wood Wall (Studs) tab of the Design Rules spreadsheet contains a Green Lumber checkbox to account for the 50% reduction in the Ga value defined in the NDS footnote.

Stiffness Assumptions

RISA uses an orthotropic plate element to de-couple the vertical and shear stiffness of the wood walls. The vertical stiffness will be based on the E value of the studs and chords. Whereas the shear stiffness will be based on the Ga value designated within the specified nailing schedule.

Note:

- Currently the thickness used, for both the vertical and shear stiffnesses, is based on the minimum thickness for the wall panel nailing schedule.

Wood Wall Self Weight

The program will calculate wood wall self weight as a sum of all the weights of the components. The material density is used to calculate the self weight of the studs, chords, top plates, sill plate, and sheathing. These are all then summed together to give the self weight of the entire wall.

Note:

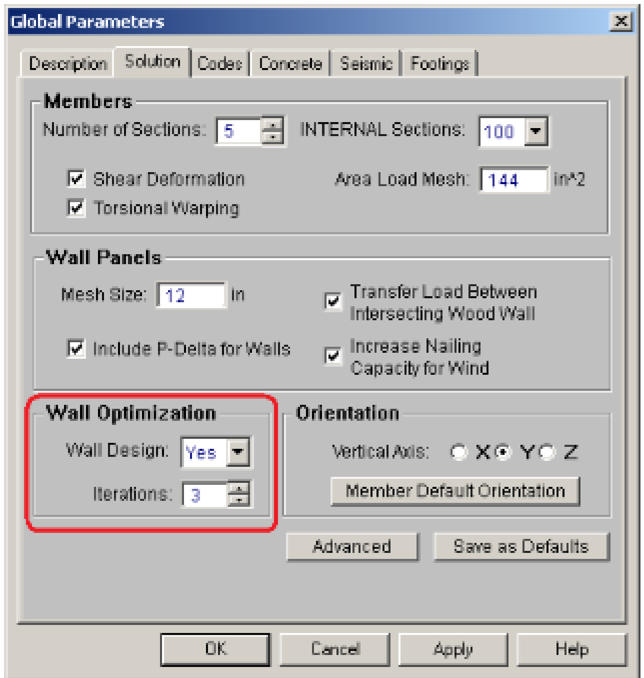
- For this calculation, stud height equals wall height minus the thickness of the sill plate and the top plate.
- The number of studs is calculated using the stud spacing specified in Design Rules.

Wood Wall Optimization

The program will optimize wood walls based on the required demand forces. The program can optimize:

- Sheathing for in-plane shear design.
- Stud spacing for axial design.
- Hold-downs for in plane design.

Of these optimizations the two that substantially modifies the stiffness of the wall are the sheathing and the stud spacing. To properly adjust the stiffness requires an iterative solution that updates the stiffness of the model. This includes updating the strength properties of the wall as well as the stiffness. This optimization/iteration can be done automatically (by choosing **Yes**) or can be done manually (by choosing **No**) in the Global Parameters - Solution tab.



To update the stiffness portion of the wall, the program must re-solve your model with these updated stiffnesses as this will change the distribution of forces through the model. By choosing **Yes** you are telling the program to re-solve automatically. Thus, the program will start with it's initial stiffness parameters and solve the model. It will then optimize the wall to meet strength criteria. Another solution will then be run with the new stiffnesses and the program will again optimize the wall to meet strength criteria. This procedure will continue to occur until you reach the number of **Iterations** set or until all wall panel results match those of the previous solution.

By choosing **No** the program will only run the solution once and the results will be based on the original configuration. You can then manually optimize your walls using the Suggested Design spreadsheet.

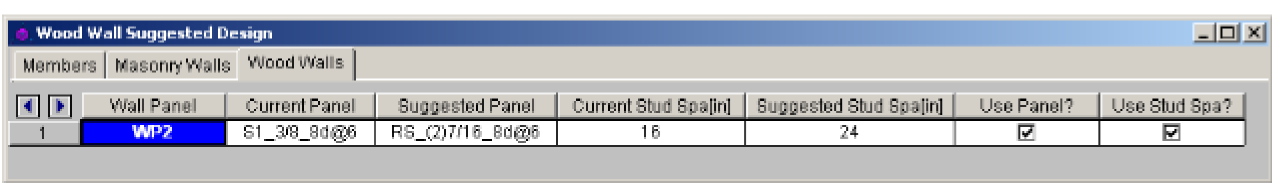
After the solution is run (with or without optimization) the design results are based on the stiffness used in the last iteration (by stating **No** a single iteration is run). The program will then compare the design of the last iteration with the stiffness used in that last iteration. If the two are the same the results shown are the final results. If the two are not the same the program will then provide these two different results in the **Suggested Design** spreadsheet.

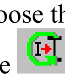
The program will always present results in the output that coincide with the stiffness used in the final solution.

Note:

- The updating of the stiffness for the model is only required for the sheathing and stud spacing optimization. Thus, hold-down call-outs are optimized automatically.

Suggested Design



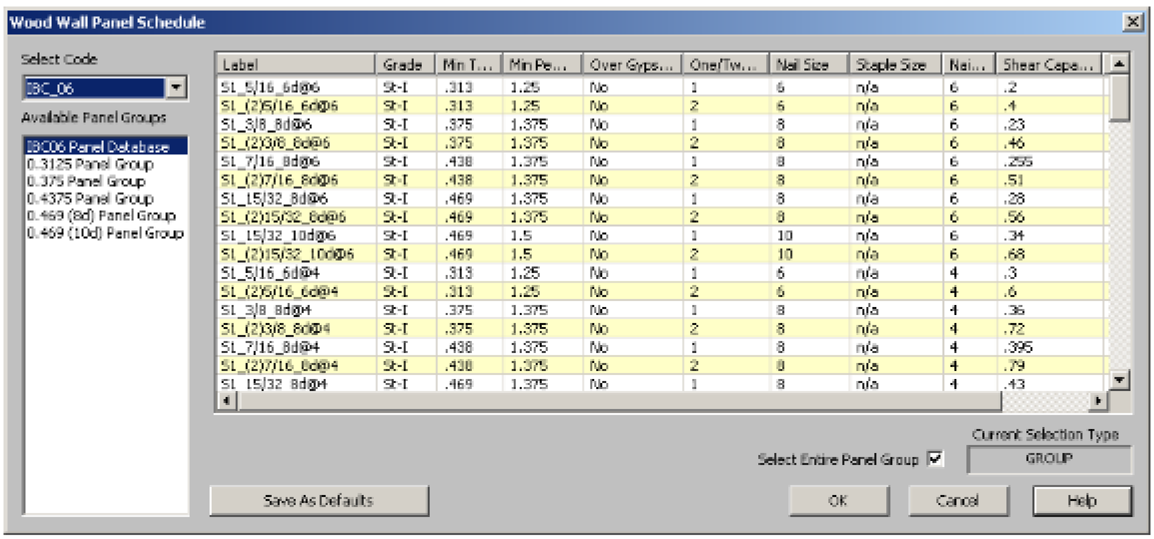
In the **Suggested Design** spreadsheet you will get a list of wall panels in your model that are not yet fully optimized, showing the panel and stud spacing of the last iteration and the program optimized values. From here you have the ability to **Use Panel?** or **Use Stud Space?** which means that you want to re-run the solution with the suggested design. You can choose this for each suggestion for each wall panel individually. Once you have these checkboxes checked appropriately press the 

button. After this the stiffness matrix is re-formulated and may cause some redistribution of loads through the model. Because of this the suggested design may also update and you may need to **Use Panel?**and/or **Use Stud Space?** multiple times to converge on a solution.

Note:

- If the wall does not show up in the Wood Wall Suggested Design spreadsheet then the current wall panel settings used are the optimal ones.
- If either the stud spacing or the panel fields are blank that means that the current selections are the optimal ones.
- For more information on masonry wall optimization see the Masonry Wall - Design topic.
- For more information on member optimization see the Design Optimization topic.
- Concrete walls do not show up here because the reinforcement optimization does not affect the stiffness of the wall.

Panel Optimization



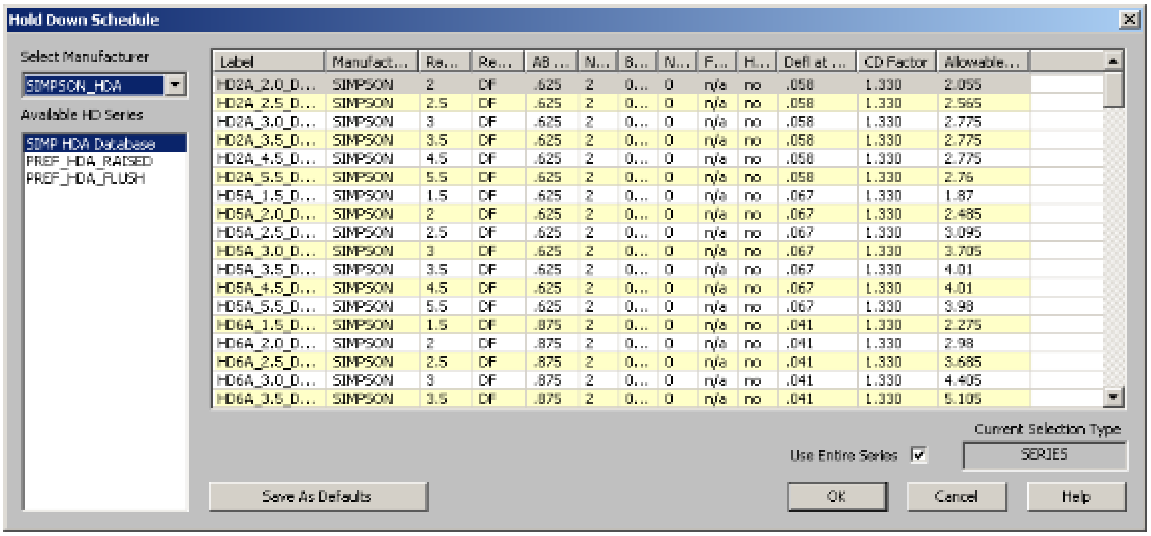
Label	Grade	Min T...	Min Pe...	Over Gyps...	One/Tw...	Nail Size	Staple Size	Nai...	Shear Capa...
SL_5/16_6d@6	St-I	.313	1.25	No	1	6	n/a	6	.2
SL_(2)5/16_6d@6	St-I	.313	1.25	No	2	6	n/a	6	.4
SL_3/8_8d@6	St-I	.375	1.375	No	1	8	n/a	6	.23
SL_(2)3/8_8d@6	St-I	.375	1.375	No	2	8	n/a	6	.46
SL_7/16_8d@6	St-I	.438	1.375	No	1	8	n/a	6	.255
SL_(2)7/16_8d@6	St-I	.438	1.375	No	2	8	n/a	6	.51
SL_15/32_8d@6	St-I	.469	1.375	No	1	8	n/a	6	.28
SL_(2)15/32_8d@6	St-I	.469	1.375	No	2	8	n/a	6	.56
SL_15/32_10d@6	St-I	.469	1.5	No	1	10	n/a	6	.34
SL_(2)15/32_10d@6	St-I	.469	1.5	No	2	10	n/a	6	.68
SL_5/16_6d@4	St-I	.313	1.25	No	1	6	n/a	4	.3
SL_(2)5/16_6d@4	St-I	.313	1.25	No	2	6	n/a	4	.6
SL_3/8_8d@4	St-I	.375	1.375	No	1	8	n/a	4	.36
SL_(2)3/8_8d@4	St-I	.375	1.375	No	2	8	n/a	4	.72
SL_7/16_8d@4	St-I	.438	1.375	No	1	8	n/a	4	.395
SL_(2)7/16_8d@4	St-I	.438	1.375	No	2	8	n/a	4	.79
SL_15/32_8d@4	St-I	.469	1.375	No	1	8	n/a	4	.43

The procedure that RISA uses for design optimization is fundamentally based upon the assumption that there is a 'cost' to shear capacity, and therefore the ideal panel design would have as little shear capacity as possible to meet code requirements. Once the program has determined the shear demand on the wall it will choose the most economical panel configuration based on that which has a **Shear Capacity** closest to, but not exceeding the shear demand.

Note:

- The shear force listed in the XML spreadsheet is tabulated for the seismic values. These values will automatically be multiplied by 1.4 for wind forces if theWind ASIFfunction is enabled.
- Panels with a label containing the characters "_W" together will be ignored during design optimization.

Hold Down Optimization



The procedure that RISA uses for hold-down optimization is fundamentally based upon the assumption that there is a 'cost' to allowable tension in a product, and therefore the ideal hold-down would have as little tensile capacity as possible to meet code requirements. Once the program has determined the tensile force required to hold-down the wall it will choose the most economical hold-down product based on that which most closely matches (but does not exceed) the tension demand. The program looks to the **Allowable Tension** field of the hold-down schedule to choose the design.

Optimization Options

For users who are new to wood wall design within RISA, the best procedure is to utilize the full databases, and to limit the potential designs by utilizing the design rules spreadsheets. This results in a design based on the maximum number of options, which is often the most efficient design.

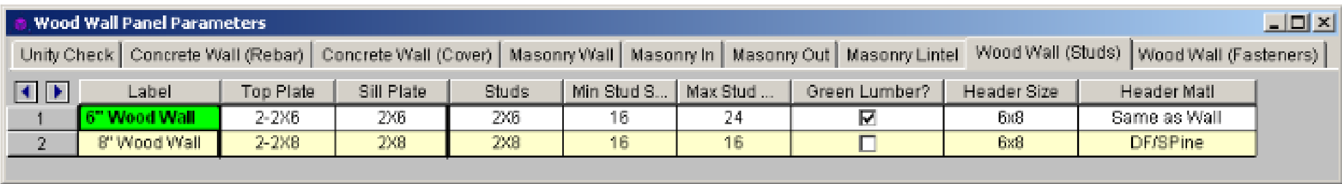
For experienced users who have more specific limitations in terms of the designs they would like to see, user-defined **Groups** (or families) are the solution. For example, an engineer who prefers to use only one sheathing thickness, or one nail type can create a custom Group that contains only the arrangements they want. For more information on creating these custom groups see Appendix F-Wood Shear Wall Files.

For more information on Wood Walls see Wood Wall Results.

Wood Wall - Design Rules

The wood wall panel element allows you to easily model, analyze and design wood walls for in plane loads. Here we will explain the wood specific inputs and design considerations. For general wall panel information, see the Wall Panels topic. For information on wood wall design considerations, see the Wood Wall - Design topic. For wood wall results interpretation, see the Wood Wall Results topic.

Wood Wall (Studs)



	Label	Top Plate	Sill Plate	Studs	Min Stud S...	Max Stud ...	Green Lumber?	Header Size	Header Matl
1	6" Wood Wall	2-2X6	2X6	2X6	16	24	☑	6x8	Same as Wall
2	8" Wood Wall	2-2X8	2X8	2X8	16	16	☐	6x8	DF/SPine

Top Plate

Use the Top Plate column to specify the member to be used as a top plate for your wall. A top plate is a member that runs continuously along the top of the wall studs. Note that you can use multiple plies of nominal lumber, or custom shapes.

Sill Plate

Use the **Sill Plate** column to specify the member to be used as a sill plate for your wall. A sill plate is a member that runs continuously along the bottom of the wall studs. Note that you can use multiple plies of nominal lumber, or custom shapes.

Studs

Use the Studs column to specify the member to be used for studs in your wall. Studs are vertical members in the wall, attached to the sill plate at the bottom and the top plate at the top. Note that you can use multiple plies of nominal lumber, or custom shapes.

Min/Max Stud Space

You may specify a minimum and maximum spacing of wall studs. The program can then optimize the stud spacing based on axial design only. For information on how the optimization works, see the Wood Wall - Design topic.

- If you specify the maximum and minimum stud spacing as the same value, then we will use that value exclusively.
- Out of plane design is not performed for wood walls, so any optimization is based only on axial forces in the studs.

Green Lumber

Check this box if your moisture content is greater than 19%. The program will then multiply the Ga value of the shear panel by 0.5 per Note 5 of Tables 4.3A and 4.3B of the NDS SDPWS.

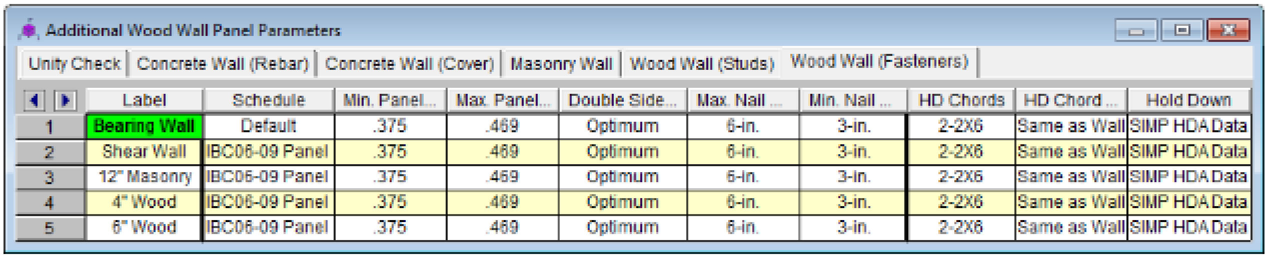
Header Size

This defines the default header size for all openings. Note that this can be modified in the Wall Panel Editor by double-clicking the opening and choosing **Custom**.

Header Matl

By default we will use the same material for the header as we are for the studs, chords, etc. However, you can change the material here.

Design Rules - Wood Wall (Fasteners)



	Label	Schedule	Min. Panel...	Max. Panel...	Double Side...	Max. Nail ...	Min. Nail ...	HD Chords	HD Chord ...	Hold Down
1	Bearing Wall	Default	.375	.469	Optimum	6-in.	3-in.	2-2X6	Same as Wall	SIMP HDA Data
2	Shear Wall	IBC06-09 Panel	.375	.469	Optimum	6-in.	3-in.	2-2X6	Same as Wall	SIMP HDA Data
3	12" Masonry	IBC06-09 Panel	.375	.469	Optimum	6-in.	3-in.	2-2X6	Same as Wall	SIMP HDA Data
4	4" Wood	IBC06-09 Panel	.375	.469	Optimum	6-in.	3-in.	2-2X6	Same as Wall	SIMP HDA Data
5	6" Wood	IBC06-09 Panel	.375	.469	Optimum	6-in.	3-in.	2-2X6	Same as Wall	SIMP HDA Data

Schedule

You can select the Code, and Panel Group you would like to use for design optimization. By unchecking the **Select Entire Panel Group** box an individual panel type may be assigned. For information on how the optimization works, see the Wood Wall - Design topic. For more information on this schedule, as well as information on how to edit or create your own custom schedule, see Appendix F-Wood Design Databases

Min/Max Panel Thick

These values set minimums and maximums for the thickness of the sheathing that will be designed. If the same value is input for both max and min, then that will be the thickness used.

Double Sided

You can choose whether you want the program to force sheathing on only one side of the panel, both sides, or to choose the optimum based on weight.

Max/Min Nail Spacing

These values set minimums and maximums for the spacing of the nails that fasten the sheathing to the boundary members (top plate, sill plate, hold down chords). Note that a 12" spacing is assumed for all field nailing (nails fastening the sheathing to the internal studs).

HD Chords

You can choose what member size you would like to use for the Hold Down Chords (Posts) at both ends of the wall panel.

HD Chord Material

You can specify whether the hold down chords are of the same material as the wall, or another material.

Hold Down Schedule

You can select the Code, and Hold Down Series you would like to use for design optimization. By unchecking the **Use Entire Series** box you my select an individual hold down product to be assigned. For information on how the optimization works, see the Wood Wall - Design topic. For more information on this schedule, as well as information on how to edit or create your own custom schedule, see Appendix F-Wood Design Databases

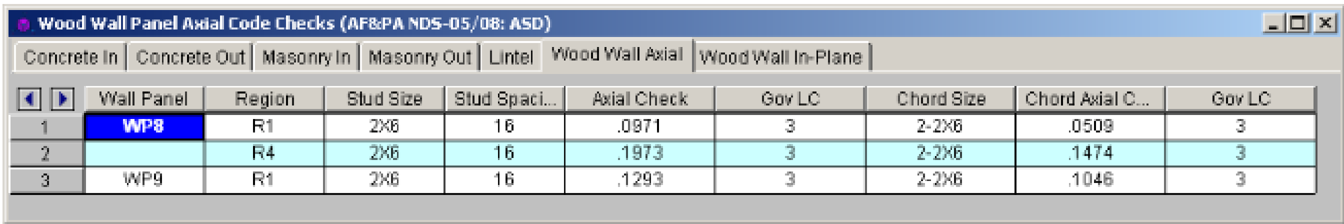
Wood Wall Results

Wood Wall results are presented in the Wall Panel Design Spreadsheet and the detail reports.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Wood Walls**.

Wood Wall Results Spreadsheets

Wood Wall Axial Results


Wood Wall Panel Axial Code Checks (AF&PA NDS-05/08: ASD)

Concrete In | Concrete Out | Masonry In | Masonry Out | Lintel | Wood Wall Axial | Wood Wall In-Plane

	Wall Panel	Region	Stud Size	Stud Spaci...	Axial Check	Gov LC	Chord Size	Chord Axial C...	Gov LC
1	WP8	R1	2X6	16	.0971	3	2-2X6	.0509	3
2		R4	2X6	16	.1973	3	2-2X6	.1474	3
3	WP9	R1	2X6	16	.1293	3	2-2X6	.1046	3

The **Wall Panel** column lists the wood wall panels that you have defined.

The **Region** column lists the wall panel region that the results are based on.

Note:

- For Perforated or Force Transfer Around Openings design methods, regions are not used. Thus, an N/A will be displayed.

The **Stud Size and Spacing** columns show the optimum stud size and spacing chosen for your wall, based on the Design Rules you have defined.

The **Axial Check** value is a code check ratio between the member load and the member capacity. The adjacent **Gov LC** column shows the governing load case for the design.

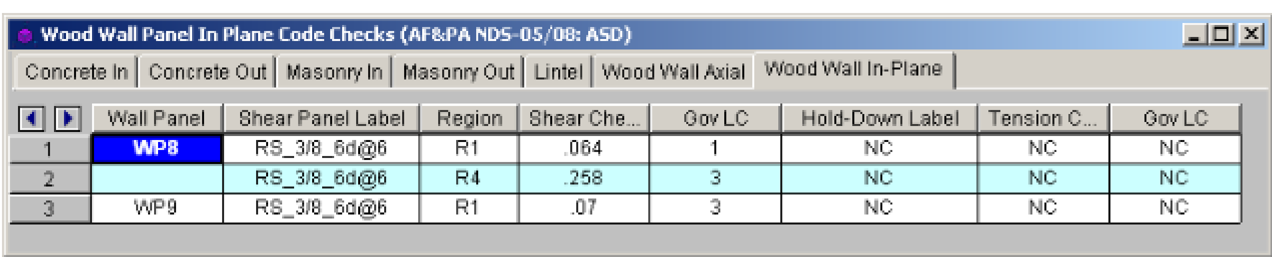
The **Chord Size** column shows the optimum chord size chosen for your wall, based on the Design Rules you have defined. Note that the chords are the vertical hold-down members/posts at the both ends of the wall.

The **Chord Axial Check** value is a code check ratio between the member load and the member capacity. The adjacent **Gov LC** column shows the governing load case for the design.

Note:

- When running the Segmented design method, the wall panel regions above and below the opening are not considered in design. Thus an NC (no calculation) will be displayed.
- If there are some constraints that will not allow a wall to be designed, an NC (no calculation) will be displayed. Check the Warning Log within the program for more information on this.

Wood Wall In-Plane Results


Wood Wall Panel In Plane Code Checks (AF&PA NDS-05/08: ASD)

Concrete In | Concrete Out | Masonry In | Masonry Out | Lintel | Wood Wall Axial | Wood Wall In-Plane

	Wall Panel	Shear Panel Label	Region	Shear Che...	Gov LC	Hold-Down Label	Tension C...	Gov LC
1	WP8	RS_3/8_6d@6	R1	.064	1	NC	NC	NC
2		RS_3/8_6d@6	R4	.258	3	NC	NC	NC
3	WP9	RS_3/8_6d@6	R1	.07	3	NC	NC	NC

The **Shear Panel Label** shows the optimum shear panel arrangement chosen for your wall, based on the Design Rules you have defined.

The **Region** gives the region for which the design values are being displayed.

Note:

- The Perforated and Force Transfer Around Openings methods do not consider regions in their design, thus N/A is displayed.

The **Shear Check** value is a code check ratio between the panel shear load and the panel shear capacity. The **Shear Force** and **Gov LC** show the values and load combination which governed the design.

The **Hold-Down Label** shows the optimum hold-down product chosen for your wall, based on the Design Rules you have defined.

The **Tension Check** value is a code check ratio between the tension load and the hold-down tensile capacity. The **Tie-Down Force**and **Gov LC** show the values and load combination which governed the design.

Note:

- When running the Segmented design method, the wall panel regions above and below the opening are not considered in design. Thus an NC (no calculation) will be displayed.
- If there are some constraints that will not allow a wall to be designed, an NC (no calculation) will be displayed. Check the Warning Log within the program for more information on this.
- If 'default' is shown in the Hold-Down Label column it means that a hold down is not required for this wall/region.

Wood Wall Self Weight

The program will calculate the self weight of a wood wall based on the weights of the individual components. Using the material density, the self weight is calculated for the studs, chords, top plates, sill plates, and sheathing. These are all then summed for the total self weight of the wall.

Wood Wall Detail Reports

The detail report gives detailed information about the wall design. The detail reports are specifically molded to the type of design specified. Here we will walk through how to access the different information for each of the types of design: Segmented, Perforated and Force Transfer Around Openings.

Note:

- RISAFloor only considers the Segmented design method. If using RISAFloor together with RISA-3D, simply taking your model into RISA-3D will open up the Perforated and Force Transfer Around Openings design methods.
- Many of the values for design checks seen below are not performed in RISAFloor as it is strictly a gravity design program.

Accessing the Detail Reports and the Specific Windows

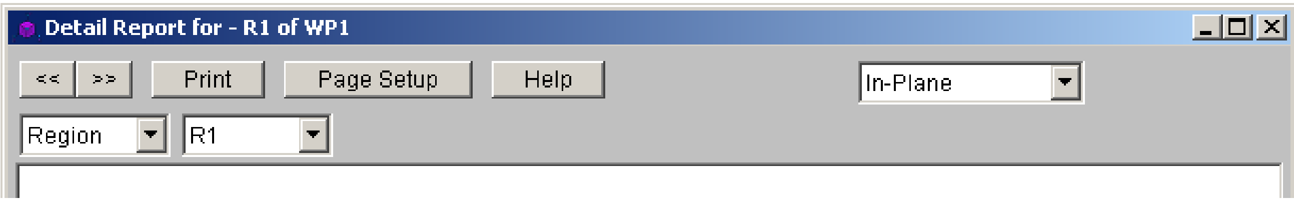
Once you have a solved model, the detail reports become available. They are accessible in two ways:

1. If you have the Wall Panel Design spreadsheet open, there will be a button at the top of the screen: Detail Report for Current Item. This will open up the detail report window.
2. If you are in a graphic view of your model, there is a Detail button that will open up the detail report window.

Note:

- Detail report information is not available for an envelope solution.

Once the detail report window is open, you will see a dialog area at the top.



- 

 - Allows you to click between the different wall panels in your model.
- 

 - This dropdown list allows you to select between the three different parts of the wood wall panel detail report. Below we will explain the importance of each of these sections.
- 

 - This dropdown list allows you to select between different regions or headers defined within the individual wall panel.

The Wall detail report gives an overall summary of your wall, complete with governing code checks and opening information. The Opening detail report gives information to the header design for the opening as well as detailed information for the force transfer around openings method. The Region detail report only applies for segmented walls. Below we have give detailed information on each type of design: Segmented, Perforated and Force Transfer Around Openings.

Segmented Method

The segmented design method uses each of the three detail report windows to give design information.

Wall Window

This window gives an overview of the wall, giving controlling region information and deflection information. Note that this window only gives information on the full-height segments in your wall, as this is the basis of the Segmented method.

Input Echo

GENERAL

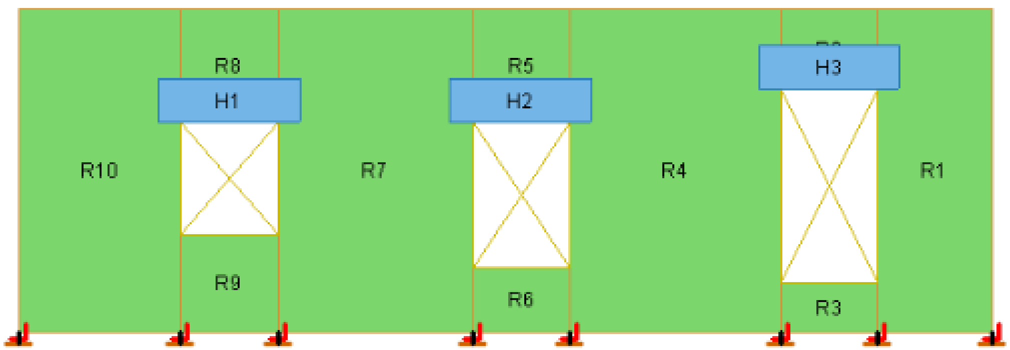
Code : NDS 2005:ASD
Design Method : Segmented
Wall Material : DF Larch
Panel Schedule : IBC06 Panel Data...
Sel. Shear Panel: RS_3/8_8d@6

GEOMETRY

Total Height : 10 ft
Total Length : 30 ft
Max H/W Ratio : 2.86

MATERIALS

Description	Material	Size
Top Pl	DF Larch	2-2X6
Sill	DF Larch	2X6
Wall Stud	DF Larch	2X6
Chord	DF Larch	2-2X6



This lists information about the wall, similar to the Region report and also gives an image of the wall. The image shows the location of hold-downs/straps, regions and headers.

Design Details

DESIGN DETAILS

ENVELOPED RESULTS

Controlling Shear Region	Shear Panel	Shear UC	Shear LC	Controlling Hold-down	Hold-down UC	Hold-down LC	Chord UC	Chord LC	Stud UC	Stud LC
R1	RS_3/8_8d@6	0.933	13 (S)	HD2A_1.5...	0.540	46 (S)	0.364	2	0.426	2

REGION INFORMATION

Full-Height Region Label	H/W Ratio	Shear UC	Shear LC	Hold-down	Hold-down UC	Hold-down LC	Chord UC	Chord LC	Stud UC	Stud LC
R1	2.86	0.933	13 (S)	HD2A_1.5_HF_F	0.418	48 (S)	0.121	10 (W)	0.087	2
R4	1.54	0.768	13 (S)	Not Req'd	NC	NC	0.364	2	0.426	2
R7	1.67	0.804	15 (S)	Not Req'd	NC	NC	0.166	2	0.314	2
R10	2.00	0.693	15 (S)	HD2A_1.5_HF_F	0.540	46 (S)	0.173	10 (W)	0.131	2

OPENING INFORMATION

Headers of openings are not designed for Segmented walls.
Please choose Perforated or FTAO design method to get header design.

DEFLECTION RESULTS

Maximum Region Deflection (in)	Deflection LC	Finite Element Deflection (in)	Shear Stiffness Adjustment Factor (SSAF)
.169 (R7)	15	.211	1

The **Enveloped Results** gives the code checks for all of the controlling elements in the wall and their associated load combinations.

The **Region Information** gives the tabulated results of all of the full-height regions in your wall.

The **Opening Information** simply states that header design can not be completed with the Segmented method. The regions above and below the opening have their shear stiffnesses set to be zero and this causes the header forces to be invalid.

The **Deflection Results** gives both the calculated NDS deflection (Maximum Region Deflection) and the FE deflection for use as a means of comparison. Because the NDS equations are empirical and take into account many non-elastic

considerations such as nail slip these two values may not be the same. The use of the SSAF helps to make these two values similar.

Opening Window

This window defines the design of the header beams across the top of the openings in the wall. The different openings can be chosen from the header drop down list. This report is very similar to a wood member detail report. At the top of the detail report the Criteria, Geometry and Materials section give the input parameters defined for the opening. The Envelope Diagrams and the Design Details are given, which provide code check information and give the information required to verify the program values.

Region Window

This window gives information for your wall on a region by region basis. Note that only full-height regions of the wall panel will have a region detail report. The segmented method only considers these full height segments in the design of the wall.

The region detail report is split into four portions: input echo, diagrams and design, design details, and cross section detailing. Note that in RISAFloor the detail reports are less detailed because RISAFloor does not consider lateral forces which RISA-3D does.

Input Echo

Below is the input echo portion of the detail report.

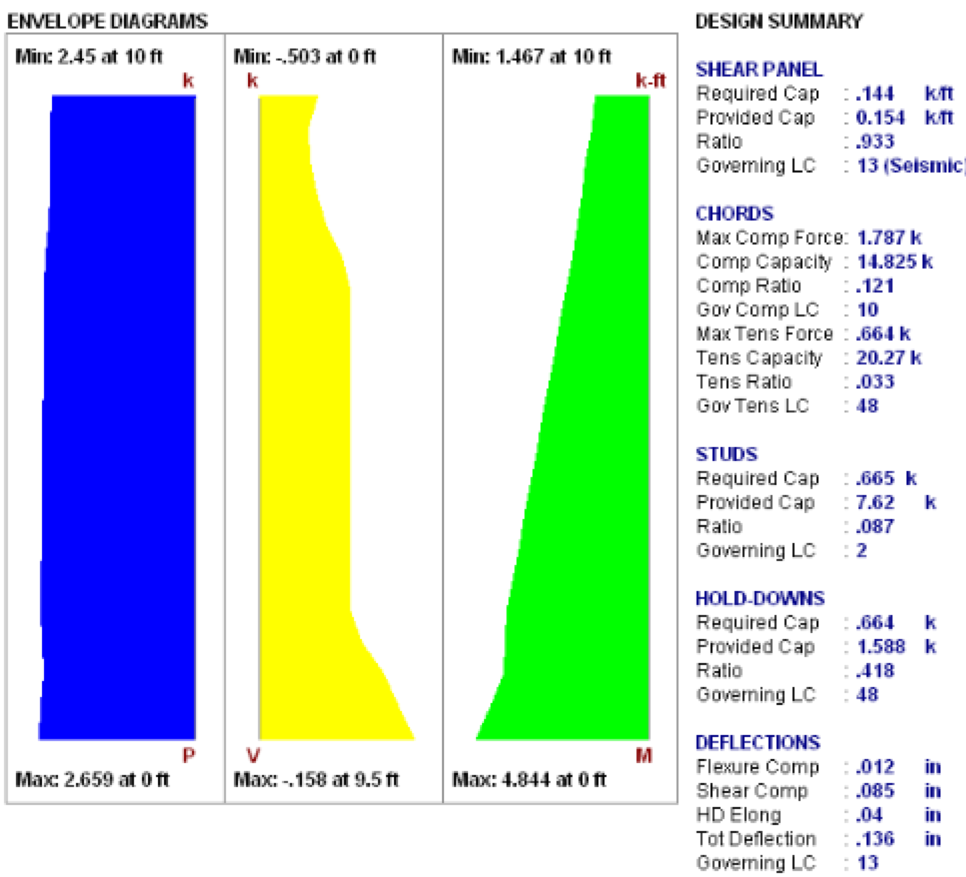
CRITERIA		MATERIALS		GEOMETRY		
Code	: NDS 2005:ASD	Wall Studs	: DF Larch	Total Height	: 10	ft
		Stud Size	: 2X6	Total Length	: 3.5	ft
Wall Material	: DF Larch	Chord Material	: DF Larch	Region H/W	: 2.86	
Panel Schedule	: IBC06 Panel Data...	Chord Size	: 2-2X6	Cap. Adj. (2w/h)	: 1.00	
				Wind ASIF	: 1.4	
Optimize HD	: Yes	Top Pl & Sill	: DF Larch	Stud Spacing	: 16	in
HD Manufacturer:	SIMPSON	Top Pl Size	: 2-2X6			
		Sill Pl Size	: 2X6			

Criteria	Description
Code	Gives the code used to design your wall panel.
Wall Material	Specifies the wood type assigned to the entire wall
Panel Schedule	Specifies the sheathing/nailing schedule database used to optimize panel selection (set in Design Rules)
Optimize HD	Shows whether or not the program needed to optimize the hold-down, or if the user explicitly defined a hold down
HD Manufacturer	Specifies the manufacturer of chosen hold-down

Materials	Description
Wall Studs	Specifies the wood material type assigned to the wall studs
Stud Size	Specifies the member size used for the wall studs
Chord Material	Specifies the wood material type assigned to the chords (vertical members at both ends of the wall)
Chord Size	Specifies the member size used for the chords (vertical members at both ends of the wall)
Top Plate & Sill	Specifies the wood material type assigned to the top and sill plates
Top Plate Size	Specifies the member size used for the top plate
Sill Plate Size	Specifies the member size used for the sill plate

Geometry	Description
Total Height	This is the height of the wall panel region
Total Length	This is the length of the wall panel region
Region H/W Ratio	This is the ratio of wall height to length, using the minimum wall height
Capacity Adjustment Factor	This is an aspect ratio reduction factor for the shear panel strength. This factor applies only for seismic loads, thus it will be 1.0 for wind load combinations. This factor is applied separately for each full-height region in your wall.
Wind ASIF	The code gives a 40% increase in the tables if the lateral load is wind over seismic. For seismic loads this ASIF will be 1.0
Stud Spacing	This is the optimized stud spacing based on your Design Rules

Diagrams and Design



Envelope Diagrams

These diagrams show the axial forces, in-plane shear, and in-plane moments of the wall, as well as the maximum and minimum forces and their locations.

Design Summary

This portion gives you the capacity and strength values at the section in the wall where the combined check is maximum, as well as the governing load combination. Much of this information is also reported in the Wood Wall Panel Design spreadsheets.

Shear Panel

The provided capacity of the shear panel is taken from the **shear capacity** column of the panel database. This is the allowable shear value from Table 2306.4.1 from the 2006 IBC. This capacity automatically considers whether the loading is based on wind or seismic loads. The Governing LC explicitly states if the controlling load combination was based on Wind or Seismic. The program does a unity check for all LCs that are being solved, finds the maximum value and reports that information.

Note:

- A shear panel design will be chosen for the worst-case region in a segmented wall. That panel will then be used for all regions in that wall. The worst-case region is the one that has the highest Shear UC value.
- Because wind and seismic loading allows for different design capacities, the highest shear in the wall may not be the governing shear (if that highest shear was due to wind).

Chords, Studs

The provided capacities of these members are calculated using the standard provisions for tension/compression members. These members are assumed to be fully braced in the weak axis, and unbraced in the strong axis. For more information on the chord force calculations, see the Wood Wall - Design topic.

Note:

- For chord results, the tension/compression capacity is computed using the reduced cross-sectional properties caused by the hold-down bolt hole.

Hold-Downs

The provided capacity of the hold-down is taken from the **Allowable Tension** column of the hold-down database. This is information supplied by the manufacturer. Note that we are modifying the Cd value for the hold-down based on taking a ratio of the assumed Cd values from the database and the Cd called for in the Load Combinations spreadsheet.

Deflections

The deflection listed in the detail report is based on an approximation from the NDS 2005/08 *Special Design Provisions for Wind and Seismic*, Equation 4.3-1:

$$\delta_{sw} = \frac{8vh^3}{EAb} + \frac{vh}{1000G_a} + \frac{h\Delta_a}{b}$$

b = Shear wall length, ft

Δ_a = Total vertical elongation of wall anchorage system (including fastener slip, device elongation, rod elongation, etc.) at the induced unit shear in the shear wall, in (This value is taken from the hold down database and scaled per the actual tension force; hence you multiply this value by the holddown ratio given in the output)

E = Modulus of elasticity of end posts (chords), psi

A = Area of end post (chord) cross-section, in^2

G_a = Apparent shear wall shear stiffness from nail slip and panel shear deformation, kips/in. (taken from shear panel database)

=

h =Shear wall height, ft

ν =Induced unit shear, lbs/ft

δ_{sw}Total shear wall deflection determined by elastic analysis, in

=

The first term of the above equation determines the Bending Component of the deflection.

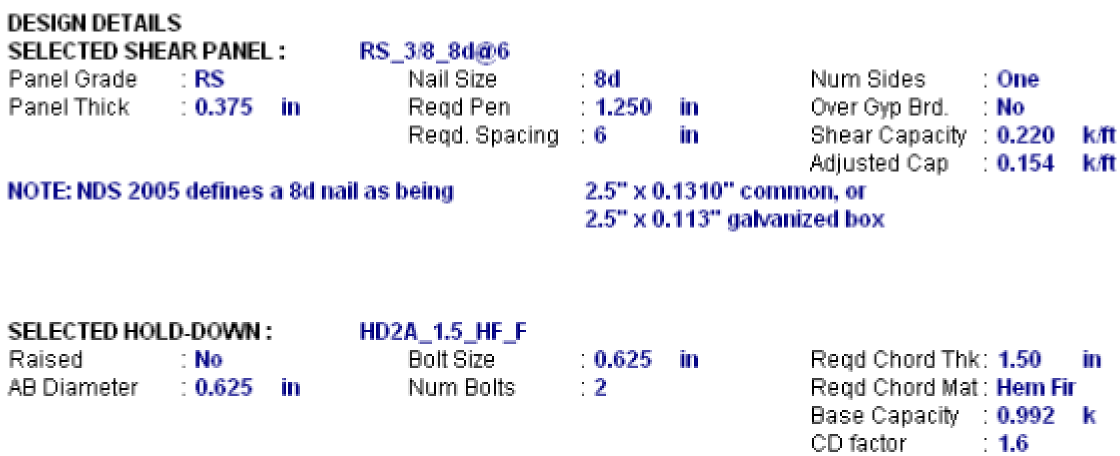
The second term of the above equation determines the Shear Component of the deflection.

The third term of the above equation determines the Hold-Down Elongation, which causes additional deflection.

Note

- This is the theoretical deflection of the wall. This may differ from the deflection of the wall as performed by finite element analysis within RISA. Therefore, this deflection value may not coincide with the reported deflection value in the deflections spreadsheets. For information on making the FEM deflections similar to the reported deflections from the NDS calculated deflections, see the Shear Stiffness Adjustment Factor information.
- For Perforated design deflections, the total deflection is to be divided by $C_o$per section 2305.3.8.2.9 of the IBC 2006. Because the unit shear values have already been amplified by C_o, the only portion of the deflection that needs to be divided is the hold-down portion.
- The hold-down deflection is reported for the maximum shear LC, which may not result in the largest hold-down component, but typically results in the highest total deflection.

Design Details


DESIGN DETAILS
SELECTED SHEAR PANEL : RS_3/8_8d@6
Panel Grade : RS
Panel Thick : 0.375 in
Nail Size : 8d
Reqd Pen : 1.250 in
Reqd. Spacing : 6 in
Num Sides : One
Over Gyp Brd. : No
Shear Capacity : 0.220 k/ft
Adjusted Cap : 0.154 k/ft
NOTE: NDS 2005 defines a 8d nail as being 2.5" x 0.1310" common, or 2.5" x 0.113" galvanized box

SELECTED HOLD-DOWN : HD2A_1.5_HF_F
Raised : No
AB Diameter : 0.625 in
Bolt Size : 0.625 in
Num Bolts : 2
Reqd Chord Thk : 1.50 in
Reqd Chord Mat : Hem Fir
Base Capacity : 0.992 k
CD factor : 1.6

The **Selected Shear Panel** gives the call-out from the shear panel database. The information below it is the information describing the call-out.

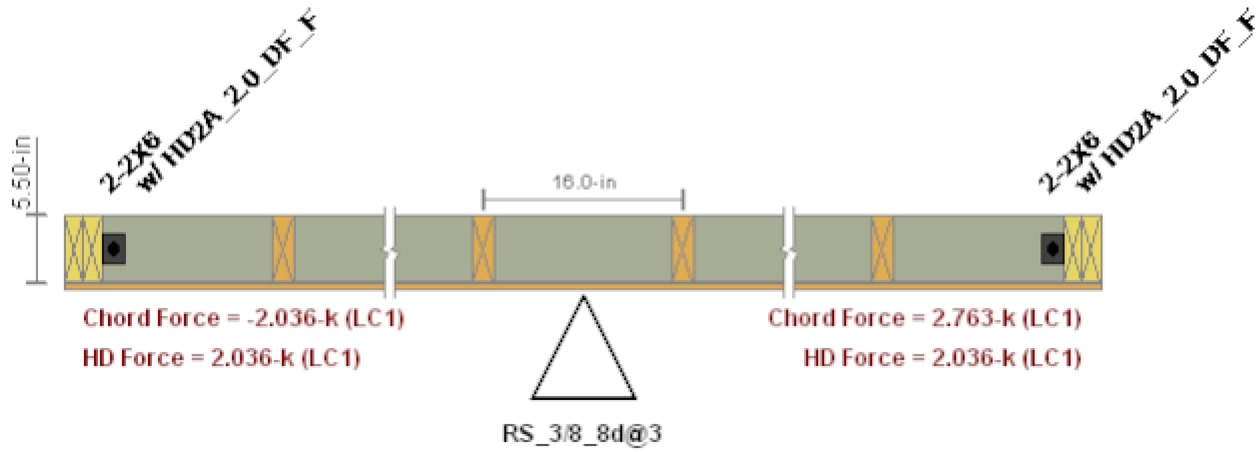
Note:

- There will also be an "Adjusted Shear Capacity" that takes the given shear capacity from the design code and divides it by any appropriate adjustment factors. One factor is the $2b_s/h$ factor from section 2305.3.8.2.2.3 of the IBC 2006 that is applicable only to seismic forces. If a wind load combination controls, the capacity will see the 1.4 increase.

The **Selected Hold-Down** gives the call-out from the hold-down database. The information below it is the information describing the callout. The "Base Capacity" is the capacity from the manufacturer divided by the assumed Cd value from the database. The "CD factor" that is displayed is the value from the load combinations spreadsheet for the controlling load combination. The actual capacity of the hold-down is the Base Capacity*CD factor.

The above section of the report echoes the database information for the selected shear panel and hold-down. For more information on these properties refer to Appendix F-Wood Shear Wall Files.

Cross Section Detailing



The last section of the detail report consists of the wall detailing information. This information is provided as a visual confirmation of the wall design. The wall thickness, and stud spacing are shown as dimensions. The triangle shows sheathing on one side of the wall, with the abbreviated panel designation. The chord sizes/forces and hold down designations/forces are shown at either end. If either chord is only experiencing a compression force, the hold down will not be drawn.

Note:

- The displayed chord force is not necessarily the force used in the hold-down design because hold-down optimization only considers the governing tension LC.

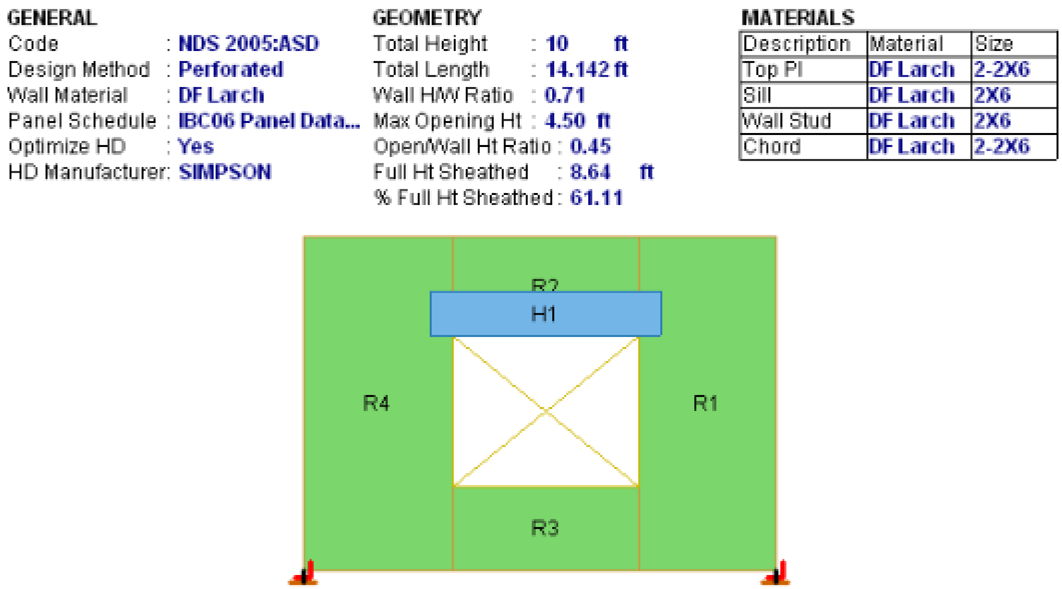
Perforated Method

Opening Window

This window is identical to the Segmented Window information.

Wall Window

This is where the majority of the information is located for the perforated method.



The **General** information gives some of the important parameters in designing the wall.

The **Geometry** section gives dimensions and ratios for the wall panel.

- The Wall H/W Ratio is checked against the aspect ratio limits given in Table 4.3.4 of the NDS 2005 *Special Design Provisions for Wind & Seismic* (SDPWS).
- The Max Opening Ht is used in the calculation of $\mathbf{C_o}$.
- The % Full Ht Sheathed is used in the calculation of $\mathbf{C_o}$.
- The Full Ht Sheathed is the Sum Li value for $\mathbf{C_o}$.

The **Materials** information just gives the sizes of the members that are not explicitly talked about in the detail report.

Note:

- The top plate, sill plate and trimmer sizes are used only for the Material Take Off.

DESIGN DETAILS

Shear Stiffness Adjustment Factor	: 1.00	Shear Capacity Adjustment Factor (Co):	0.88
Wall Capacity Adjustment Factor (2w/h):	0.83	Total Area of Openings (Ao)	: 24.75 ft^2
		Sheathing Area Ratio (r)	: 0.78

WALL DEFLECTIONS

Elastic:	: .017 in
HD:	: .126 in
Shear:	: .116 in
Total:	: .259 in

WALL RESULTS:

Governing LC	: 14 (Seismic)
Total Shear	: 3.889 k
Max Unit Shear	: .511 k/ft
Shear Ratio	: .995

SELECTED SHEAR PANEL : RS_(2)15/32_10d@6

Panel Grade	: RS	Nail Size	: 10d	Num Sides	: Two
Panel Thick	: 0.469in	Reqd Pen	: 1.250in	Over Gyp Brd.	: No
		Reqd. Spacing	: 6 in	Shear Capacity	: 0.620 k/ft
				Adjusted Cap	: 0.514 k/ft

NOTE: NDS 2005 defines a 10d nail as being: 3.0" x 0.1480" common, or 3.0" x 0.122" galvanized box

SELECTED HOLD-DOWN : HD8A_3.5_HF_F

Raised	: No	Bolt Size	: 0.875in	Reqd Chord Thk:	3.50 in
AB Diameter	: 0.875in	Num Bolts	: 3	Reqd Chord Mat :	Hem Fir
				Base Capacity	: 4.774 k
				CD factor	: 1.6

CHORDS		STUDS		HOLD-DOWNS	
Max Comp Force:	14.024 k	Required Cap	: 3.418 k	Required Cap	: 7.467 k
Comp Capacity	: 14.055 k	Provided Cap	: 7.62 k	Provided Cap	: 7.639 k
Comp Ratio	: .998	Ratio	: .449	Ratio	: .978
Gov Comp LC	: 22	Governing LC	: 2	Governing LC	: 46
Max Tens Force	: 7.467 k	Gov Region	: 1		
Tens Capacity	: 19.217 k	Spacing	: 16 in		
Tens Ratio	: .389				
Gov Tens LC	: 46				

The **Design Details** section gives adjustment factors and some other values used to calculate C_o. See the Wall Panels topic for more information on the Shear Stiffness Adjustment Factor. See the Wood Wall - Design topic for more information on C_o.

The **Deflections** section gives the calculated deflection for the three term shear wall equation from the NDS. See above for more information on deflections.

The **Wall Results** section gives:

- The **Governing LC** is the load combination that produced the highest allowable code check. This will also state if this load combination was a wind or seismic LC.
- The **Total Shear** is the total shear in the wall for the governing LC.
- The **Max. Unit Shear** is the maximum shear in the shear panel and it is what is used to optimize the sheathing/nailing selection from the shear panel database. Note that the Max Unit Shear may not be the absolute maximum, because shear walls governed by wind are allowed a 40% stress increase. Thus, a wind Max Unit Shear would need to be 40% higher than a seismic Max Unit Shear to govern.
- The **Shear Ratio** is a ratio of the Max Unit Shear over the Shear Capacity of the shear panel selected from the database.

The **Selected Shear Panel** and the **Selected Hold-Down** section gives the same hold down and shear panel information that was given in the Segmented region report.

There is then the chord, stud and hold-down design section that has the same information as given in the Segmented region report.

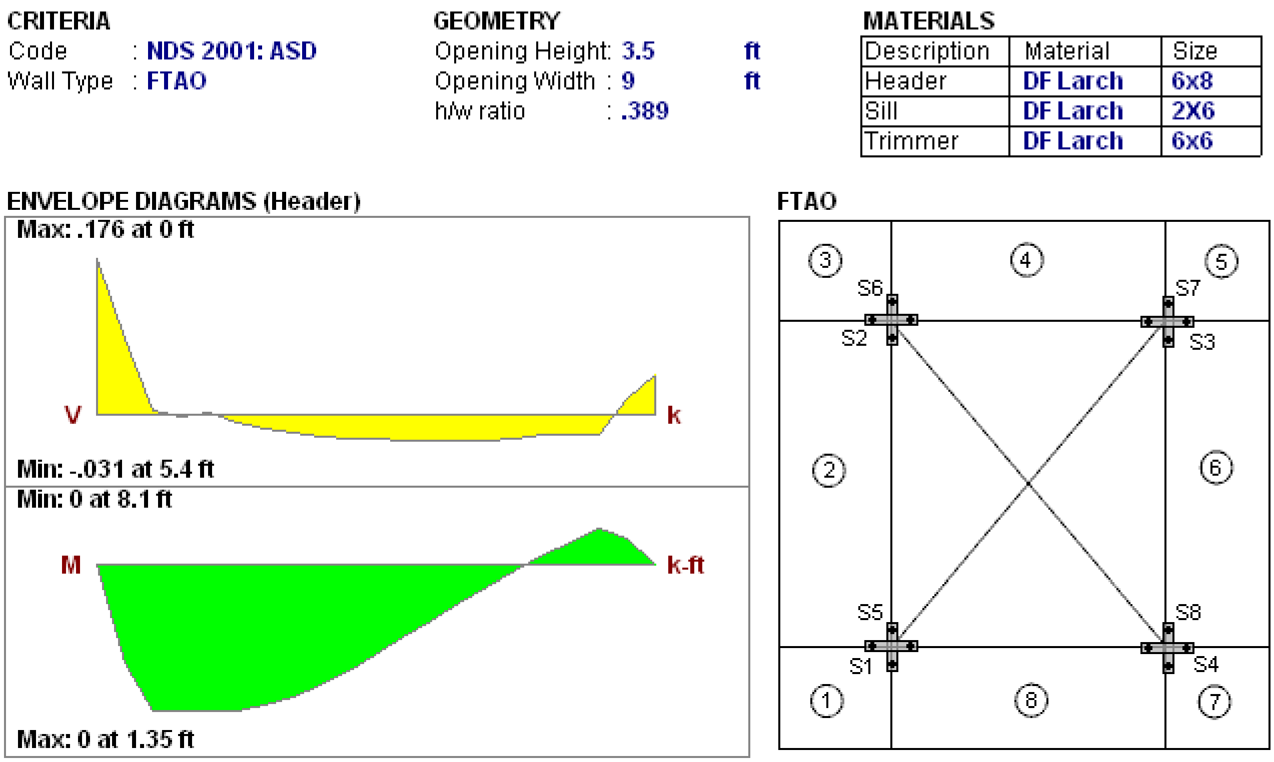
The **Cross Section Detailing** section gives a detailed view of the wall. For more information see the Segmented section.

Force Transfer Around Openings (FTAO)

Wall Window

This is the overall wall information and is essentially identical to the Perforated method wall window information. There is some geometry information that is not necessary for FTAO that is omitted.

Opening Window



The **Criteria** section gives the code being used and which design method used.

The **Geometry** section gives the opening dimensions and the h/w ratio.

The **Materials** section gives dimensions for some of the members in the wall.

The **Envelope Diagrams** give the enveloped shear and moment diagrams for the header beam above the opening.

The **FTAO** graphic shows the design block numbers around the wall panel opening along with the strap numbers.

DESIGN DETAILS

OPENING STRAPS

Name	Location	Direction	Req'd Cap (k)	Gov LC
S1	Bottom, Left	Horizontal	-0.7	1
S2	Upper, Left	Horizontal	0.3	1
S3	Upper, Right	Horizontal	-1.7	1
S4	Bottom, Right	Horizontal	0.9	1
S5	Bottom, Left	Vertical	-0.1	1
S6	Upper, Left	Vertical	-0.0	1
S7	Upper, Right	Vertical	-0.2	1
S8	Bottom, Right	Vertical	0.1	1

ANALYSIS SUMMARY

Block #	Unit Shear (k/ft)	h/w Ratio
1	-0.111	0.917
2	-0.206	0.583
3	-0.156	0.500
4	-0.292	0.333
5	-0.317	0.600
6	-0.190	0.700
7	-0.058	1.100
8	-0.174	0.611

The **Design Details** section is split into two different tables: Opening Straps and Analysis Summary. The opening straps information gives the location, direction, force in the strap and the load combination that caused that force.

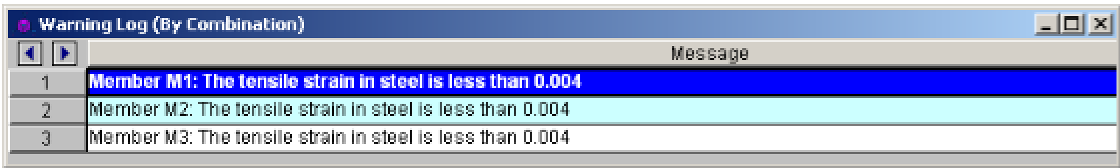
Note:

- For more information on how the strap forces and unit shears are calculated, see the Wood Wall - Design topic.
- The program does not design the straps around the opening, just presents the forces.

The analysis summary gives the unit shear and the h/w ratio for each of the blocks.

The information below is the code check information for the header member. This is identical to the information given for Segmented Window information.

Warning Log



The **Warning Log Spreadsheet** provides you with a record of any warnings or errors that occurred during the solution of your model. The log should be reviewed for warnings or errors that would affect the design of your structure. These messages are, in general, self-explanatory. The error log reports back the item label for which the error occurred.

You can view the warning log by clicking on the Spreadsheets menu item and then clicking on the Warning Log selection.

Below are a few of the common warnings which may require some additional explanation:

Sum of reactions is not equal to the sum of the loads (LC xx)! Check for any small rigid links or fixed boundary conditions.

When the solution is complete, the program checks the sum of the reaction forces in each direction if this is more than 0.1% different from the sum of the applied loads, then this warning message will be displayed. The most common causes for this warning message are the following:

- A joint instability which has been automatically LOCKED by the program. If the joint is locked then the reaction at that location is not computed. Once you rectify the instability then this warning log message should also go away. You may also need to uncheck the "Lock isolated ROTATIONAL instabilities without notification" box as this may conceal some instabilities. See the Stability section for more information.
- A user assigned boundary condition which was used the "Fixed, reaction will not be calculated" option rather than the "Reaction" option. See the Boundary Conditions section for more information. Using the term "Fixed" suppresses the reaction output in the Joint Reactions and thus the applied loads does not equal joint reactions. In this case the Warning Log could likely be ignored, as there is a valid reason for this discrepancy.
- A "ghost reaction" may have developed where one or more rigid elements (Diaphragms, rigid end offsets, or rigid links) have become so stiff that they actually became stiffer than the internal stiffness used by the program to define boundary conditions. In this rare case forces may be leaving the model at locations other than boundary conditions. Having a combination of a rigid diaphragm, rigid links, rigid end offsets, top of member offsets, etc., all in a localized place in the model could cause these.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Sum of Reactions**.

Member is Unsupported or Cannot Establish Span Information

The program assumes that each beam will be supported vertically by two columns, plates or boundary conditions. It also assumes that each column will be supported laterally by two beams, plates, or boundary conditions.

The program uses this information to determine "span" information for beams and columns.This span information is then used for concrete rebar design. For non-concrete members, the bad framing messages can be ignored.

If you would like to suppress these warnings in the future, just go to the **Concrete** tab of **Global Parameters Dialog** and uncheck the **Bad Framing Warnings** box.

Span has fewer than 5 analysis sections. Increase number of sections.

In order to perform concrete rebar design, the program needs to have a good representation for what sort of moment and shear diagrams exists in the beam or column. When there are less than 5 internal sections in a beam or span, the program will

warn you that it did not have enough information to perform an adequate concrete design. Increasing the number of internal sections on the **Global Parameters Dialog** may correct this problem.

Since, the program wants to perform shear design at a distance "d" from the face of support, it may not be possible to get rebar design for beam or columns with a large depth and a relatively short span. Keep in mind that this will only affect rebar design. Deflection and force results are unaffected.

Span length is less than 1 foot. No design for this span.

When a beam span is very short (less than a foot) the program will not be able to perform concrete design. Typical cases where this may occur are curved concrete members or grade beams.

Factored torsional moment Tu exceeds the threshold torsion

The torsion present in this concrete member exceeds the threshold below which torsion may be ignored for the design of shear reinforcement. The shear reinforcement must be designed by the engineer to accommodate the torsion in the member.

Internal Results Corrupted - Call RISA Technologies

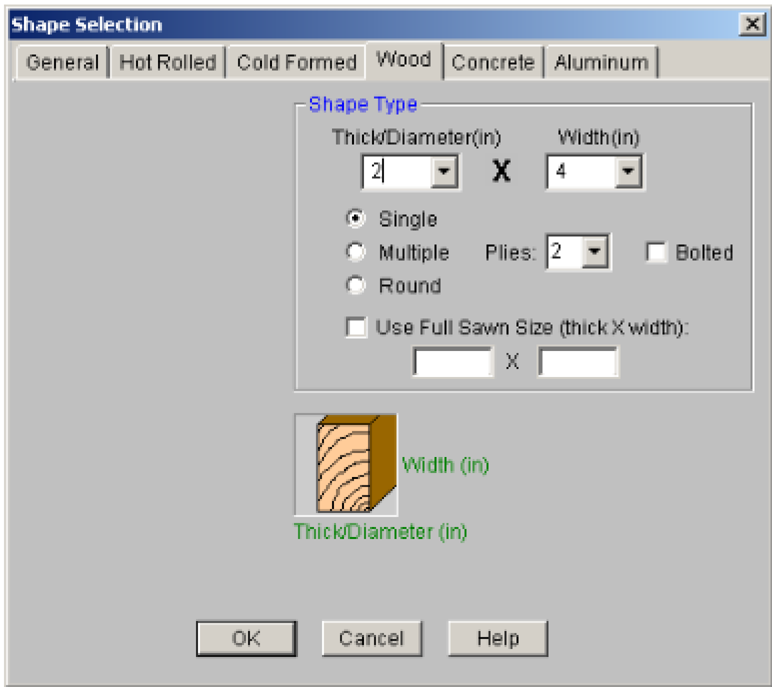
The program has detected an internal error while processing the solution results. Please contact RISA Technologies immediately.

Member Mxx at (length): The rebar does not exist. No Capacity Calculated."

This message means that a user defined reinforcement profile has resulted in a member with zero capacity (and zero reinforcement) at the described location.

Wood - Database

The Wood Database may be accessed from the **Wood** tab of the **Section Sets** spreadsheet by clicking in the **Shape** field and then clicking , or by clicking the **Shape Database** button on the **RISA** toolbar and clicking the **Wood** tab.



To Select a Wood Database Shape

1. On the **Wood** tab of the **Section Sets Spreadsheet**, move the cursor to the **Shape** field and click .
2. Specify the shape type you wish to use (single, multiple, or round), then select from the lists of thicknesses and widths by clicking on .

Note

- If the value that you need is not given on the drop down list, you may directly enter any whole number for the thickness or width.
- Enter the nominal dimensions (or round diameter) in inches, regardless of what units system you are using. These will be automatically adjusted to the actual (dressed) dimensions for stiffness and stress calculations. For example, if you enter “2X4” as the size, the calculated properties are based on an actual size of “1.5 in. X 3.5 in.”. For manually entered numbers above 16 it is still assumed that 3/4" is removed for dressing.
- Multiple-ply lumber is assumed to be fastened sufficiently to achieve full composite behavior between plies. Therefore a two-ply member will have 8 times greater minor-axis flexural stiffness than a single ply member (as opposed to double the stiffness)
- If using multiple plies there is a **Bolted** checkbox. If this box is unchecked it is assumed that the plies are nailed together. If bolted is checked then it is assumed the plies are bolted together. This affects the Kf value from section 15.3.2 of the NDS 2005. If you define the multi-ply section as bolted a **B** will appear after the shape name.

Round Shapes

Per section 3.7.3 of the NDS, the design of a round cross section shall be based on the design calculations for a square shape with the same cross-sectional area. Therefore the moment of inertia values will be calculated per the equivalent square shape, not the round cross section.

Custom Wood Sizes

If you would like to enter explicit dimensions of a member or if the member is "Not Dressed", the member must be designated as a "Full Sawn" member by checking the **Use Full Sawn Size** check box and entering the exact dimensions of the member in the boxes below. This applies to regular wood species, custom wood species, Structural Composite Lumber (SCL), and Glu-Lam members. The member thickness should be entered in the box on the left and the member width should be entered in the box on the right.

Wood - Design

Full code checking can be performed on Dimension Lumber and Post and Timber size wood shapes based on the following codes:

- The 2005/08 edition of the NDS (National Design Specification)
- The 2001 edition of the NDS
- The 1991 / 1997 editions of the NDS

Note

- When the 1991 / 1997 NDS is selected, the 1991 NDS specification will be used with the 1997 stress tables. This is consistent with the requirements of the 1997 UBC.

Glu-Lams

Glu-Lams are treated as any other wood species and may be selected from the list of species on the Wood Tab of the Materials spreadsheet. When a Glu-Lam is selected, the grade will be listed as "na" or not applicable.

All Glu-Lam members should be dimensioned as "Full Sawn" using the format *wXdFS*, where "w" and "d" are the actual width and depth dimensions. If the size is entered as *wXd* without the *FS* designation, then the size will be dressed down as if the member were regular dimensional lumber.

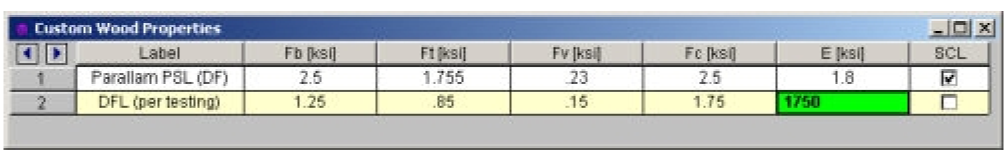
RISA includes two redesign lists for Glu-Lams: Glu-Lam_Western for Western Species and Hardwoods (HW), and Glu-Lam_SouthernPine for Southern Pine (SP/SP).

Note

- Glu-Lams from Table5A are always assumed to have the special tension laminations. Therefore, the Fbx value is not reduced.
- RISA is NOT applying any of the footnotes to Table 5A and 5C at this time except for the values for Fvx and Fvy in Table 5A.
- Glu-Lam design is not supported for the 91/97 NDS code.

Custom Wood Materials & Structural Composite Lumber

To use a custom wood material that is not part of the standard NDS database, you will need to define the design properties of a new / custom wood species.


Custom Wood Properties

	Label	Fb [ksi]	Ft [ksi]	Fv [ksi]	Fc [ksi]	E [ksi]	SCL
1	Parallam PSL (DF)	2.5	1.755	.23	2.5	1.8	☑
2	DFL (per testing)	1.25	.85	.15	1.75	1750	☐

To do this, select **Spreadsheets** ▸ **Custom Wood Species Spreadsheet** from the main menu toolbar. Enter a label for your new custom species, then enter the wood properties (Fb, Fc, etc.) in the columns to the right. This new material will now be included in between the NDS wood species and the glulam types in the **Species** drop down list on the wood materials spreadsheet.

Note

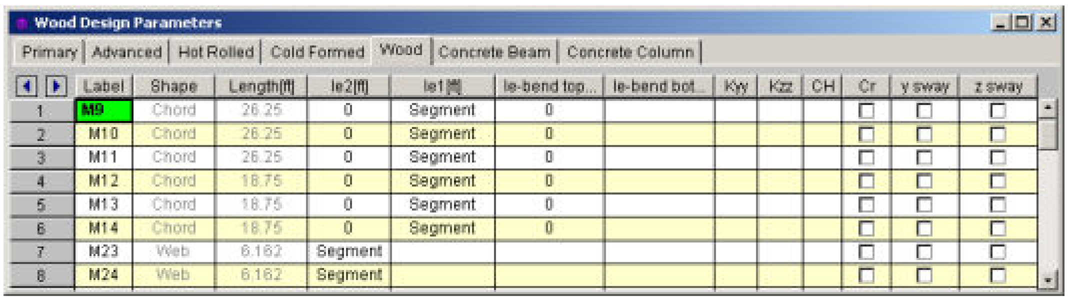
- Fc is the base value for the compressive stress parallel to grain and will be used to calculate the member's ability to resist axial compression.

- The **SCL** checkbox is used to designate whether this new species is Structural Composite Lumber as defined in the 2001/05 NDS. The 2001/05 NDS has a code checking procedure for composite lumber that is slightly different from the procedure used for standard, dimensional lumber.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Custom Wood Species**.

Wood Member Design Parameters

The **Member Design Parameters** spreadsheet records the design parameters for the steel and timber code checks and may be accessed by selecting **Design Parameters** on the **Spreadsheets** menu. These parameters may also be assigned graphically. See Modifying Member Design to learn how to do this.



	Label	Shape	Length[ft]	le2[ft]	le1[ft]	le-bend top...	le-bend bot...	Kyy	Kzz	CH	Cr	y sway	z sway
1	M9	Chord	26.25	0	Segment	0					☐	☐	☐
2	M10	Chord	26.25	0	Segment	0					☐	☐	☐
3	M11	Chord	26.25	0	Segment	0					☐	☐	☐
4	M12	Chord	18.75	0	Segment	0					☐	☐	☐
5	M13	Chord	18.75	0	Segment	0					☐	☐	☐
6	M14	Chord	18.75	0	Segment	0					☐	☐	☐
7	M23	Web	6.162	Segment							☐	☐	☐
8	M24	Web	6.162	Segment							☐	☐	☐

These parameters are defined for each member.

Label

You may assign a unique **Label** to all of the members. Each label must be unique, so if you try to enter the same label more than once you will get an error message. You may relabel at any time with the **Relabel** options on the **Tools** menu.

Shape

The member **Shape** or Section Set is reported in the second column. This value is listed for reference only and may not be edited as it is dictated by the entry in the Section/Shape column on the **Primary** tab.

Length

The member **Length** is reported in the third column. This value may not be edited as it is dependent on the member end coordinates listed on the **Primary Data** tab. It is listed here as a reference for unbraced lengths which are discussed in the next section.

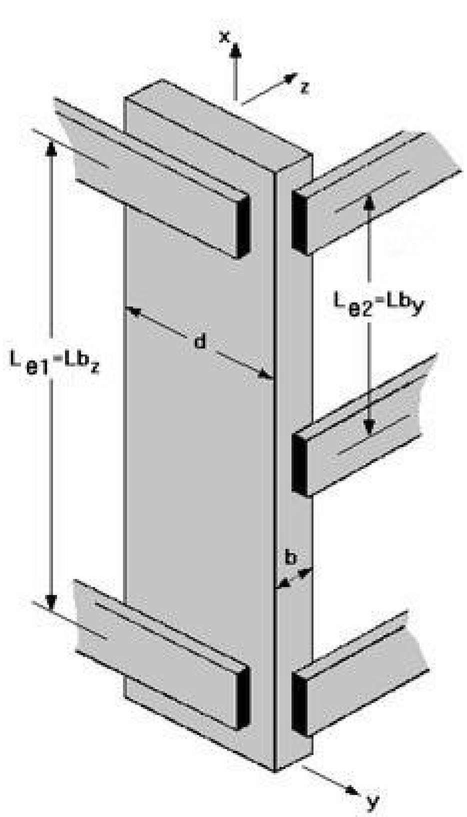
Unbraced Length

You may specify unbraced lengths or have RISA-3D calculate them for you. The unbraced lengths are **Le1**, **Le2**, **Le-bend-top**, and **Le-bend-bot**.

The values **Le1** and **Le2** represent the unbraced length for the member with respect to column-type (Euler) buckling about the member's local z and y axes, respectively. These values are used to calculate Le1/d and Le2/b, which in turn impact the calculation of Cp, the column stability factor. These length to thickness ratios gauge the vulnerability of the member to buckling. Refer to Section 3.7 of the NDS for more information on this. This section also lists the limiting values of the length to thickness ratios.

The **Le-bend** values, **Le-bend-top** and **Le-bend-bot**, are the effective unbraced lengths of the member for bending. This unbraced length is the length of the face of the member that is in compression from any bending moments. This value should be obtained from Table 3.3.3 in the NDS code. The **Le-bend** value is used in the calculation of the slenderness ratio, RB, which is used in the calculation of CL, the beam stability factor. CL is then used to calculate the allowable bending stress. Refer to Section 3.3.3.6 in the NDS for more information on this and note that the value of RB is limited to 50.

For continuous beams the moment will reverse such that the top and bottom faces will be in compression for different portions of the beam span. **Le-bend-top** is the effective unbraced length of the top face and **Le-bend-bot** is the effective unbraced length of the bottom face.



If left blank these unbraced lengths all default to the member's full length. The exception to this is if **Le2** is entered and **Le-bend-top** is left blank, **Le-bend-top** will default to the entered value for **Le2**. Since **Le2** and **Le-bend** are often different in wood design, it is likely you should enter the correct value for **Le-bend**.

For physical members, you can enter the code "**Segment**" in the unbraced length fields and the length of each segment will be used. A "segment" is the distance between the joints that are on the physical member. For example, suppose you have a physical member that is 20 feet in length, and there are two joints along the physical member, one 5 feet from the end and one at 15 feet. An unbraced length of 5 feet will be used for the first segment, then a value of 10 feet will be used in the middle segment, and again a value of 5 feet would be used in the last segment.

Note

- If the intermediate framing members are considered to brace the bottom flange, then you can enter “segment” for Le-bend-bot. When the “segment” command is used ALL intermediate points along the beam are viewed as brace points. Therefore, you may have to delete unused or extraneous points.
- The Top Flange is defined as the flange corresponding to the positive local y axis for the member. For more information on setting local axes refer to the Members section.
- The calculated unbraced lengths are listed on the **Member Detail** report.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keywords: **Unbraced Lengths**.

K Factors (Effective Length Factors)

The **K Factors** are also referred to as effective length factors. **Kyy** is for column type buckling about the member's local y-y axis and **Kzz** is for buckling about the local z-z axis.

If a value is entered for a **K Factor**, that value will be used for the entire length of the physical member. If an entry is not made (left blank), the value will internally default to '1' for that member. See the NDS Appendix G for an explanation of how to calculate K Factors. The **K Factors** are applied to **Le1** and **Le2** to obtain the effective column length. See section 3.7 in the NDS for more on this.

RISA-3D is able to approximate the K values for a member based on the member's sway condition and end release configuration. The K-factor approximation is based on Table G1, found in Appendix G, of the NDS 2005 specification. The following table gives the values used for various conditions.

Table Case	End Conditions	Sidesway?	K-Value
(a)	Fixed-Fixed	No	.65
(b)	Fixed-Pinned	No	.80
(c)	Fixed-Fixed	Yes	1.2
(d)	Pinned-Pinned	No	1.0
(e)	Fixed-Free	Yes	2.1
(f)	Pinned-Fixed	Yes	2.0

RISA-3D will recognize a pinned boundary condition for the K approximation for a full pin, i.e. if all the rotations in the boundary condition are released. If any of the rotations in a boundary condition are restrained, the boundary condition is considered “fixed” for the K approximation.

Any configuration not described here will be given the default value of 1.0.

If any value that influences these K values is changed, the K approximation should be redone. For instance, if you have RISA-3D approximate K values then change some end release designations, you should redo the K approximations.

Remember that the K-values are *approximations,* and you should check to make sure you agree with all K-values RISA-3D assigns. You can always override a K-value after an approximation by directly entering the value that you want in the appropriate field. Keep in mind that a subsequent approximation will overwrite any manually input values so you will need to override the approximation each time it is performed.

Limitation:

RISA-3D will currently neglect the influence of adjoining framing members when those members are connected at a joint that also has degrees of freedom restrained by boundary conditions. For example, suppose a column and beam member connect at a joint that is restrained for translation in all directions (i.e. the joint is “pinned”). The K factor approximation will neglect the beam member when it calculates the K factor for the column and visa-versa. The effect will be that the ends of the members at that joint will be seen as “pinned” and not “fixed” for the K-factor approximation.

Sway Flags

The **Sway Flags** indicate whether the member is to be considered subject to sidesway for bending about its local y and z axes. The **y sway** field is for y-y axis bending and the **z sway** field is for z-z axis bending. Click on the field to check the box and indicate that the member is subject to sway for that particular direction, or leave the entry blank if the member is braced against sway. These sway flags influence the calculation of the K factors.

Timber Design Adjustment Factors

The NDS code has a lot of adjustment factors that you apply to the various allowable stresses, and in some cases, to the Young's modulus (E). All the adjustment factors are summarized in section 2.3 of the NDS-2005 specification. The

following topics help to summarize how adjustment factors are obtained and used. The CT, Ci, and Cb factors are NOT used.

Timber Design CH (Shear Stress Factor)

The Shear Stress Factor entry, **CH**, is the shear stress adjustment factor. This design parameter can be set on the Wood tab of the **Members** **Spreadsheet**. If left blank the program will use a default value of 1.0. See the tables in the NDS supplement for information on other CH factors. Note that only tables 4A, 4B, and 4D are used. The CH factor is only available for the 1991/1997 NDS codes. For other codes, this entry will be ignored.

Timber Design Cr (Repetitive Factor)

The Repetitive Factor field, **Cr**, specifies if the beam is one of a group of repetitive members. This design parameter can be set on the **Wood** tab of the **Members** **Spreadsheet**. If you put a check in the Cr field, a factor of 1.15 will be applied to beam members that are 2" to 4" thick. This flag will be ignored for a NDS shape that is thicker than 4". A value of '1.0' will be used for Wood Products. Different restrictions apply to the use of the Cr factor for Structural Composite Lumber and Glu-Lams.

Timber Design Ct (Temperature Factor)

The Temperature Factor, **Ct**, is calculated internally from the wood temperatures set in the Global Parameters. See section 2.3.3 of the NDS-2005 specification for more information on the temperature factor.

Timber Design Cfu (Flat Use Factor)

The Flat Use Factor, **Cfu**, is automatically applied to the weak axis allowable bending stress of a wood member whenever weak axis moments are present. The flat use factor will only be applied to members that are 2” to 4” thick. See Tables 4A, 4B, 4C, 4F, 5A, 5B, 5C, 5D and the footnotes.

Timber Design CF (Size Factor)

The Size Factor, **CF**, is applied automatically when you assign a wood shape from the NDS shape database. See Tables 4A, 4B, 4D, and 4E in the NDS supplement for information on the CF factor.

Timber Design CV (Volume Factor)

The Volume Factor, **CV**, is applied automatically when you assign a Glu-Lam member from the NDS shape database. The user can override the calculated value by inputting the factor on the **Wood** tab of the **Members** **Spreadsheet**. This entry is only available when using the 2001 or 2005 NDS code.

Note:

- In the calculation of Cv, RISA takes L conservatively as the full length of the member.

Timber Design Cf (Form Factor)

The Form Factor, **Cf**, is applied automatically when designing by the NDS 91/97 or 2001 Specification and a 'Round' shape is selected from the NDS shape database. See section 2.3.8 in the NDS (91/97, 2001) for information on the Cf factor.

Note

- This factor is not applied when design by the NDS 2005 Specification
- This factor is not applied to "diamond" shaped members, which are just rectangular members on edge. This factor is not applied to diamond shapes because any applied moments are transformed internally to the local member axes for the code check calculations, which is the same as applying the “diamond” form factor and NOT transforming the moments.

Timber Design Cm (Wet Service Factor)

The Wet Service Factor, **Cm**, is applied when you check the **Cm** checkbox in the **Materials Spreadsheet**.

Timber Design CP and CL (Column/Beam Stability Factors)

The Column Stability Factor, **CP**, and the Beam Stability Factor, **CL**, are calculated internally. These calculated values are shown on the **Wood** tab of the **Design Results Spreadsheet**, as well as in the **Member Detail Reports**. See NDS 2005 section 3.3.3 for information on the CL factor and NDS 2005 section 3.7.1 for information on the CP factor. The value of Emin used for the calculation of these factors is calculated using equation D-4 from appendix D of the 2005 NDS. For some members (especially for Glu-Lams) this equation may produce a slightly more accurate value of Emin that shown in the NDS tables.

Note:

- The column stability factor, C_p, is affected by the Kf factor from NDS 2005 section 15.3.2, depending on whether multi-ply members are bolted or nailed together. Bolted columns will have a shape name with a **B** after. See the Wood-Database topic for information on how to define bolted vs nailed multi-ply members.

Timber Design CD (Load Duration Factor)

The Load Duration Factor, **CD**, is entered on the **Load Combination Spreadsheet** for each load combination for which you want wood code check results. The CD factor must be entered for each individual load combination because the CD factor is dependent on the types of loads that are applied in each load combination. Therefore, different load combinations could have different CD factors. For example, per the NDS 2005 specification, a load combination that had only dead load, would have a CD factor of “0.9”, while another combination that was comprised of dead load plus wind load would have a CD factor of “1.6”.

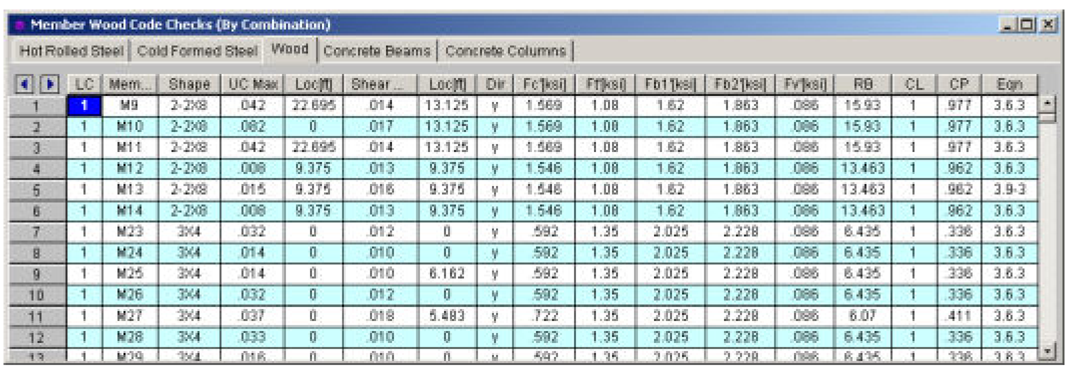
The CD factor will only be applied to wood code checks on wood members. See Table 2.3.2 in the NDS 2005 specification for the CD factors to be applied for typical loads. Appendix B has additional information about the Load Duration Factor.

Note

- The CD factor used for a load combination should be for the load with the shortest load duration in that load combination.

Wood Member Code Check Results

Access the **Wood Code Checks Spreadsheet** by selecting the **Results** menu and then selecting **Members ▸ Design Results** and then clicking the **Wood** tab.



	LC	Mem...	Shape	UC Max	Loc[ft]	Shear ...	Loc[ft]	Dir	Fc'[ksi]	Ft'[ksi]	Fb1'[ksi]	Fb2'[ksi]	Fv'[ksi]	RB	CL	CP	Eqn
1	1	M9	2-2X8	.042	22.695	.014	13.125	y	1.569	1.08	1.62	1.863	.086	15.93	1	.977	3.6.3
2	1	M10	2-2X8	.082	0	.017	13.125	y	1.569	1.08	1.62	1.863	.086	15.93	1	.977	3.6.3
3	1	M11	2-2X8	.042	22.695	.014	13.125	y	1.569	1.08	1.62	1.863	.086	15.93	1	.977	3.6.3
4	1	M12	2-2X8	.008	9.375	.013	9.375	y	1.546	1.08	1.62	1.863	.086	13.463	1	.962	3.6.3
5	1	M13	2-2X8	.015	9.375	.016	9.375	y	1.546	1.08	1.62	1.863	.086	13.463	1	.962	3.9-3
6	1	M14	2-2X8	.008	9.375	.013	9.375	y	1.546	1.08	1.62	1.863	.086	13.463	1	.962	3.6.3
7	1	M23	3X4	.032	0	.012	0	y	.592	1.35	2.025	2.228	.086	6.435	1	.336	3.6.3
8	1	M24	3X4	.014	0	.010	0	y	.592	1.35	2.025	2.228	.086	6.435	1	.336	3.6.3
9	1	M25	3X4	.014	0	.010	6.162	y	.592	1.35	2.025	2.228	.086	6.435	1	.336	3.6.3
10	1	M26	3X4	.032	0	.012	0	y	.592	1.35	2.025	2.228	.086	6.435	1	.336	3.6.3
11	1	M27	3X4	.037	0	.018	5.483	y	.722	1.35	2.025	2.228	.086	6.07	1	.411	3.6.3
12	1	M28	3X4	.033	0	.010	0	y	.592	1.35	2.025	2.228	.086	6.435	1	.336	3.6.3
13	1	M29	3X4	.016	0	.010	0	y	.592	1.35	2.025	2.228	.086	6.435	1	.336	3.6.3

The final result of the code checking is a code check value. This value represents a ratio of actual stress to allowable stress. So, if this value is less than 1.0, the member passes. If it is greater than 1.0, the member fails. If the value is greater than 9.999 it will be listed as "9.999". The **Loc** field tells at what location the maximum code check occurs measured from the I-joint location of the member.

The Shear Check is the maximum ratio of actual to allowable shear stress. The location for the shear check is followed by "y" or "z" to indicate the direction of the shear.

The remaining columns, discussed below, provide some of the values used in the code check with the equation number itself given in the last column. The Member Detail Report described gives more values used to perform the code check. See Plot Options – Members to learn how to view the code check results graphically.

The values (Fc', Ft', Fb1', Fb2', Fv') are the factored allowable stresses. The unfactored allowable stresses are those listed on the **Wood Properties Spreadsheet**. For the bending stresses (Fb), Fb1' is for bending about the local z-z axis (the strong axis) and Fb2' is for bending about the local y-y axis (the weak axis).RB is the adjustment factor described by Eqn. 3.3-5 of the 2005 NDS Specification. This is a slenderness ratio that is not allowed to exceed 50. CL is the beam stability factor calculated using Eqn. 3.3-6 of the NDS Specifications. CP is the column stability factor calculated using Eqn. 3.7-1 of the NDS Specifications.

Finally, the equation controlling the code check is listed, either Eqn. 3.9-1 or 3.9-3. Eqn. 3.9-2 is not checked since this equation includes the tension stress in a beneficial (non-conservative) manner. All other requirements in Section 3.9 are also checked, such as fc < FcE1, etc. To see ALL the adjustment factors and other information used to calculate the factored allowable stresses, please go to a detail report for the member in question. You can do that from this spreadsheet by clicking Detail Report for Current Member.

For enveloped results the combination that produced the listed code and shear checks is given in the column "lc". The other values are the corresponding values and are not necessarily the maximums across all the combinations.

Note

- The Member Detail Report gives more values used to perform the code check.
- See Spreadsheet Operations to learn how to use **Find**, **Sort** and other options.
- See Plot Options – Members to learn how to plot member results.

Special Messages - Wood Design

In some instances code checks are not performed for a particular member. A message explaining why a code check is not possible will be listed instead of the code check value. You may click the cell that contains the message and look to the status bar to view the full message. Following are the messages that may be listed:

NDS Code Check Not Calculated

This is the general message displayed when code checks were not performed for a member.

RB value is greater than 50

Section 3.3.3.7 of the NDS 1991/1997, 2001 and 2005 codes limits the slenderness ratio RB to a maximum of 50. You need to reduce the effective span length, increase the thickness of the shape, or reduce the depth of the shape.

le/d is greater than 50

Section 3.7.1.4 of the NDS 1991/1997, 2001 and 2005 codes limits the column slenderness ratio of Le1/b or Le2/d to a maximum of 50. You need to reduce your effective length by reducing the actual length between supports or changing the effective length factor “K”. You can also use a thicker shape.

fc is greater than FcE1

Section 3.9.3 of the NDS 1991/1997, 2001 and 2005 codes limits the actual axial compressive stress to be less than the term FcE1. This term is approximately the Euler buckling stress for buckling about the strong axis of the member. (Buckling is in the plane of bending)

fc is greater than FcE2

Section 3.9.3 of the NDS 1991/1997, 2001 and 2005 codes limits the actual axial compressive stress to be less than the term FcE2. This term is approximately the Euler buckling stress for buckling about the weak axis of the member. (Buckling is in the plane of bending)

fb1 is greater than FbE

Section 3.9.3 of the NDS 1991/1997, 2001 and 2005 codes limits the actual strong axis bending compressive stress to be less than the term FbE. This term is approximately the lateral buckling stress.

Limitations - Wood Design

It is assumed that the load on the member is occurring through the member's shear center. This means secondary torsional moments that may occur if the load is not applied through the shear center are not considered.

Buckling Stiffness Factor - The buckling stiffness factor, CT, is not currently accounted for.

Incising Factor - The incising factor, Ci, is not currently accounted for.

Bearing Area Factor - The bearing area factor, Cb, is not currently accounted for.

Appendix A – Redesign Lists

RISA-3D has Redesign Lists that are used to optimize hot-rolled steel, cold-formed steel, dimensional lumber, and concrete beams and columns. Although, the criteria used for this optimization is the **Design Rules**, the sizes must be chosen from the available sizes in the Redesign List.

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keyword: **Redesign**.

Locating the Design Lists

By default the re-design lists will exist in the C:\RISA\risa__redesign_lists\ directory on the user's hard drive. However, this directory can be changed by selecting Tools - Preference - File Locations from the main menu toolbar.

File Naming Convention for the Design Lists

The design lists must be ASCII text files with a file extension of ASC. In addition, the first 3 characters of the Design indicate which Region the lists will be available for. The region can be changed by selecting Tools - Prefereces - General from the Main Menu Toolbar. Each region coresponds to the following file name prefix.

Region	File Prefix
United States	US_
Canada	CA_
Britain	BS_
Europe	EU_
India	IN_
Australia	AU_
New Zealand	AU_
Mexico / South America	MX_

File Format for ReDesign Lists

The following is an example of a user defined redesign list. You may view these files using any text editor such as Window's Notepad. However, this file must be saved in the risa__redesign_lists folder as an ASCII text file with a file extension ASC. If the file is saved with a txt extension, it will not be read intoRISA-3D. The various entries for the input fields are described below:

The first field [RISA_REDESIGN_LIST] identifies this as a RISA Redesign list. If this line is not present the file will not be recognized by the program.

The second field [NAME] identifies that the next entry will be the name of the Shape Group that will identify this list in the program. In this case, this Shape Group is given the name *Available W24s*.

The next field [MATERIAL_TYPE] identifies that the next entry will be the material type that this shape group belongs to. This material type must be given one of the following designations: Hot Rolled Steel, Cold Formed Steel, NDS Wood, Steel Products, or Wood Products.

The next field [MEMBER_TYPE] identifies that the next entry will be the designation of the member type. This entry must be designated as either a Beam or Column.

The next field [ASSOCIATED DATABASE] identifies that the next entry will be the RISA database name in which the entered shapes are defined.

The next field [UNITS] identifies the units used to define the member sizes in the redesign list. Entries may be Inches, Millimeters, or Centimeters. This field is only applicable to files similar to the concrete and wood lists which have the dimensions built right into the list. For files which are based on database shape names, this field will have no effect.

The next field [END HEADER] identifies the end of the descriptive information and the beginning of the shape list.

The next field [AVAILABLE_SHAPES] identifies that the next series of lines will signify all of the shapes available for this Shape Group. The end of this field is reached when the [SHAPE_END] entry is read. The file below uses an asterisk (*) to specify that W24x68's and W24x76's are NOT available for this particular job.

The next field [OPTIMUM_SHAPES] identifies that the next series of lines will signify the preferred shapes for this Shape Group. The program will attempt to only use optimum shapes for beams that are significantly braced. By significantly braced, we mean that the unbraced length of the top flange is less than or equal to three feet. The end of this field is reached when the [SHAPE_END] entry is read.

The last field [END] identifies the end of the file.

```
[NAME]
Available W24s
[MATERIAL_TYPE]
Hot Rolled Steel
[MEMBER_TYPE]
Beam
[ASSOCIATED_DATABASE]
AISC
[END_HEADER]
[AVAILABLE_SHAPES]
W24X104
W24X103
W24X94
W24X84
*W24X76
*W24X68
W24X62
W24X55
[SHAPE_END]
[OPTIMUM_SHAPES]
W24X55
W24X62
*W24X68
W24X76
[SHAPE_END]
[END]
```

Appendix B – Error Messages

Extensive error checking is performed during data entry and model solution in order to prevent you from having to scan input files for offending data. If an error or questionable entry is detected, an error or warning message will be displayed. Errors are divided into two groups such that data entry error numbers are between 1000-1999 and solution errors are between 2000-2999.

Please refer to the ***Help File*** *in RISA-3D for detailed information about each error message. The Help File may be accessed by selecting* ***Help Topics*** *from the* ***Help Menu****.*

Appendix C – STAAD® Files

STAAD files may be imported into RISA-3D by choosing File ▸ Import and then specifying STAAD as the file type. RISA-3D can translate files produced by the STAAD III or STAAD/Pro programs. (STAAD III and STAAD/Pro are registered trademarks of Research Engineers, Inc.)

The translation process will cause model information including geometry data, member and element properties, load information, some advanced modeling information, and AISC steel code check information to be read into RISA-3D.

Translated geometry data are the joints, members, and plate/shell elements. Supported member and element properties are the material property information, element thickness and member section shape data. Loading information includes joint loads, support displacements, member distributed and point loads, and element surface loads. The advanced modeling information that are translated are such things as the joint boundary conditions, including springs, the member end releases, and the "Truss" members. Note that the STAAD model type (i.e. Space, Plane, etc.) will also be detected and this information will be used to help translate the model. The AISC steel code check information is comprised of all the parameters required to perform code checks for the ASD 9th or LRFD 2nd codes.

The translator has been tested with files as old as STAAD, version 10. If you're having a problem translating an older STAAD file, you may want to read the file into the most recent version of STAAD that you have and then save a new copy out with a new file name.

Note

- "STAAD" is a registered trademark of Research Engineers Inc.

Translation log File

All lines that are not translated, including unsupported shapes, unsupported loads, comment lines, etc., are written out to a log file called 'filename'.TXT (where 'filename' is the prefix of your STAAD filename). A message will pop up and tell you the location of the file and whether any important warnings were written to the file. This file is an ASCII text file that can be viewed with any editor (NotePad, WordPad, etc.) and should be reviewed after each translation.

Supported STAAD Features

The translator supports both the "Single Item per Line" format and "Multiple Item per Line" format for Joint definition and Member/Element Incidences. (The "Single Item per Line" format was an option for older versions of STAAD). The use of the REPEAT keyword or command file data generation functions are not supported. If you have a model with these features, you will need to read the model back into STAAD and save it back out. Saving the model back out of STAAD will expand data specified with the REPEAT keyword or data generation functions.

Most properties, loads, etc. are assigned in STAAD using a "list" of items. RISA-3D supports most of the list format features, including the TO and BY keywords, the line continuation character "-", and the listing of items by "Group" name. We do not support the listing of members by specifying Global Axes for members, or by specifying Global Ranges for joints, members, and elements. If you have a model that uses either Global Axes or Global Ranges to specify item lists, you will need to specify the item lists using one of the other list features that are supported.

All comment lines (lines that start with the "*" character) are skipped and copied to the STAAD log file.

STAAD General Keywords

UNIt statements cause model data to be interpreted in the specified units. All STAAD unit types are supported.

SET Z UP - This statement will cause the vertical axis setting on the G**lobal Parameters** to be set to the Z-axis. (Default vertical in RISA-3D is the Y-axis)

FINish - This keyword is used to mark the end of the STAAD file. Nothing is translated after the FINISH keyword.

STAAD Model Type Keywords

RISA-3D2 recognizes the STAAD model types and uses the information to help translate the model.

PLAne models are assumed to be in the X-Y plane at a Z-coordinate of zero. Thus, only the X and Y coordinates are read and the Z coordinates of all joints are set to zero.

SPAce models are read in as is.

TRUss models cause member end releases to be set for all members so that members will only take axial loads. The member release codes are set to ALLpin on the I-end and BENpin on the J-end. Depending on the model geometry, this may cause RISA-3D to report instabilities when solving. (The instabilities occur if all the members connecting to a joint have the bending rotational degrees of freedom released, the joint then will have no rotational stiffness.) If this happens in a plane truss, you can use the **ALL Boundary Condition** code to apply a very soft spring to the in-plane rotational DOF. For space trusses, you can use the ALL code to apply very soft springs to all the rotational DOF's (MX, MY, MZ) for all free joints. See Stability for more information.

FLOor models are assumed to be in the X-Z plane at a Y-coordinate of zero. Thus only the X and Z coordinates are read and the Y coordinates are all to zero.

STAAD Joint Keywords

JOInt COOrdinates - Only Cartesian coordinates are supported. If you have a model in cylindrical or reverse cylindrical coordinate, you will need to read the model into STAAD and then save it back out. This will cause the coordinates to be converted to the Cartesian format. Repeat keywords and command file data generations are not supported.

JOInt LOAd - All joint forces, moments, and support displacements are read in using the units from the last Units statement. Support displacements that are rotations are converted from the STAAD convention of degrees to the RISA-3D convention of radians.

SUPports - All regular joint support types are available, including spring supports. Inclined supports and automatic spring generation using the Footing or Elastic Mat keywords are NOT supported.

STAAD Member Keywords

MEMber INCidences – Repeat keywords and command file data generation are not supported.

MEMber PROperties- If a type is not specified for Member Properties, AMErican will be assumed.

MEMber PROperties AMErican- Unsupported shapes will cause members that were assigned those shapes to be grouped together by section set with the default section properties. Different section sets will be created for the same unsupported geometric sections with different material properties. Data lines specifying unsupported shapes will be written out to the STAAD log file.

MEMber PROperties CANadian- Unsupported shapes will cause members that were assigned those shapes to be grouped together by section set with the default section properties. Different section sets will be created for the same unsupported geometric sections with different material properties. Data lines specifying unsupported shapes will be written out to the STAAD log file.

PRIsmatic - Shape properties specified using the prismatic keyword are supported. A section set will be created and the section properties will be entered into the Sections spreadsheet. A RISA-3D "arbitrary" database shape will NOT be created, and thus no bending or torsion stresses will be calculated for these sections. Note that just the properties are read in. RISA-3D does try to detect what 'type' of prismatic shape is being specified. The following "property_spec" items are recognized and read in for prismatic sections : **AX, IZ, IY, IX, AY, AZ, YD,** and **ZD**. The section area is calculated as a rectangular section via the **YD** and **ZD** items if they are specified and the area was not already given with the AX spec. Shear area factors are calculated from the specified **AY** and **AZ** values. If not specified, these values are set to 1.2, which is the RISA-3D default.

TABle - Shape properties specified using the AISC American standard table or Canadian standard table of steel shapes are supported. These shapes are matched against the RISA-3D shape database and for matched shapes, full stress calculations and steel code checks are performed.

For American AISC standard shapes, the following "type_spec" words are supported: **ST, RA, LD, SD, T,** and **SP**. All wide flange, channel, WT, single and double angle, and HSS shapes are supported. All pipe shapes, built up box type tube shapes, double channels, and built up plate girders are not supported. For double angles, only specified spacings of 0",3/8", or 3/4" are recognized. The translator will treat double angles with other spacings as unsupported shapes, however these shapes can be later added to the database using the shape editor.

For Canadian shapes as listed in the S16.1-94 standard, the following "type_spec" words are supported: **ST**, and **T**. All wide flange, channel, and WT shapes are supported. The HSS shapes are supported, however, STAAD uses the AISC names for the HSS shapes. All pipe shapes, single angles, double angles, built up box type tube shapes, double channels, and built up plate girders are not supported.

MEMber RELease - All full member end releases are recognized. Partial releases are not supported or translated.

MEMber TRUss - Members which are assigned this property are given an I-end release of ALLpin and a J-end release of BENpin. The member will take moment if a distributed load or self weight load is applied.

MEMber LOAd - Most member loads are supported. Unsupported member loads include projected point loads and projected moments, loads with a shear center offset, distributed moment loads, and triangular loads with the maximum at the center of the member specified using the **LIN** load option.

STArt GROup DEFinition - This feature is used in STAAD to give a frequently used list of member/element items an easier to reference "name". There is a limit of 32,000 groups, and 50,000 total group items that RISA-3D will use when translating the STAAD file.

CONstants - The constant keywords STEEL, CONCRETE, and ALUMINUM are supported. Note that since RISA-3D ties material properties and section properties together by using Section Sets, members with the same geometric properties but different material properties will be assigned to different Section Sets.

DEFine MATerial STArt – This feature is fully supported for ISOTROPIC materials only. NON-Isotropic material definitions will be replaced by the first default material in RISA-3D.

STAAD Element Keywords

ELEment INCidences - Repeat keywords and command file data generation are not supported.

ELEment PROperty - Only uniform element thicknesses are supported. If multiple thicknesses are specified for an element, only the first thickness is read and used as the thickness for the whole element.

ELEment LOAd - Only uniform surface loads are supported.

STArt GROup DEFinition - This feature is used in STAAD to give a frequently used list of member/element items an easier to reference "name". There is a limit of 32,000 groups, and 50,000 total group items that RISA-3D will use when translating the STAAD file.

CONstants - The constant keywords STEEL, CONCRETE, and ALUMINUM are supported. Note that since RISA-3D ties material properties and section properties together by using Section Sets, members with the same geometric properties but different material properties will be assigned to different Section Sets.

DEFine MATerial STArt – This feature is fully supported for ISOTROPIC materials only. NON-Isotropic material definitions will be replaced by the first default material in RISA-3D.

STAAD Load Keywords

LOAding - All Load cases will be translated into Basic Load Cases in RISA-3D. The loads within each Load Case will be translated to the appropriate BLC in RISA-3D. Note that RISA-3D does not solve BLC's, only Load Combinations. If you want to have a particular BLC solved by itself, you should build a Load Combination with only that BLC specified.

LOAd COMbination - All load combinations will be translated into load combinations in RISA-3D. The SRSS feature is not supported for BLC's. The SRSS feature for load combinations in RISA-3D only applies to Response Spectrum loading. If P-Delta analyses are desired, they must be assigned later on the **Load Combination** spreadsheet. RISA-3D has a limit of 8 Basic Load Cases per Load Combination. If a STAAD file is read that has more than 8 LOAD cases per Load Combination, only the first 8 will be used. A warning will be written to the log file.

STAAD AISC Parameters

PARameter - Only the AISC (ASD 9th or LRFD 2nd) codes and the Canadian CAN/CSA S16.1-94 code are supported in RISA-3D . The following parameters are recognized: **KY, KZ, LY, LZ, FYLd, UNL, UNF, CB, SSY, SSZ, CMY,** and **CMZ**. You can check the values that have been translated into RISA-3D on the **Design Parameters** spreadsheet.

Unsupported STAAD Features

In general, you will want to examine your STAAD translation log file to note all lines that were not read in and translated. Typically a line will only be written to the log file if it is not recognized and translated successfully. This will give a good indication of any features that weren't brought into RISA-3D.

STAAD solves LOAD cases and LOAD Combinations; RISA-3D only solves Load Combinations. You will need to have additional load combinations containing only one basic load case per combination to solve your basic load cases.

P-Delta analyses are specified for each load combination in RISA-3D. The P-Delta flag will NOT be set automatically. You will need to go set it for combinations where you want to include P-Delta effects.

RISA-3D does not translate any of the information in the JOB Information block of model files.

Members with K-joint's cause joints to be created at the K-joint coordinates. These joints have their degrees of freedom locked automatically during model solution.

RISA-3D has a limit of 8 Basic Load Cases per Load Combination. If a STAAD model has more than 8 LOAD cases in a Load Combination, only the first 8 will be used and a warning message will be written to the log file.

Any response spectra entered in your STAAD file will need to be entered in RISA-3D's spectra database. STAAD stores each spectra with a particular data file, whereas RISA-3D maintains a library of spectra which are accessible from any data file.

Shapes that are defined using a User defined shape database file will need to be entered into RISA-3D's shape database. RISA can automatically convert these shapes based on a Mapping file defined in the STAAD Mapping File section at the end of this appendix.

RISA-3D doesn't support non-isotropic materials. If you translate a model with a non-isotropic material specified, we will still translate the model, but we'll use the first default material instead.

STAAD User's Overview

Folks who have a lot a structural modeling experience with STAAD can usually come up to speed with RISA-3D fairly quickly. The only thing that'll slow you down is figuring out how to do in RISA-3D what you knew how to do in STAAD. The RISA-3D User's Guide is a great place to start, in spite of the fact that it covers a lot of basic modeling concepts, because it shows you the most common ways to get things done in RISA-3D. In STAAD, you were probably accustomed to generating the model by manually editing the command text file, or maybe starting the model with the graphical pre-processor and then fine tuning the model by hand in the text file. With RISA-3D the steps are similar, except that you won't ever be directly editing the text file. You will do all manual data editing using our spreadsheets. Things like Section Sets and Material Properties are good examples of data that will always be entered via the spreadsheets. The actual model geometry and the application of boundary conditions, loads, and design parameters is usually done quickest using the Model Generation functions or the Graphics Editing functions. Most of these tools will require that you spend a few minutes the first time you use them to study what they can do and how it can help you model. There is a full explanation of all the input parameters for each graphical tool in this Help file under each graphical tool topic.

Many STAAD users who are now using RISA-3D often want to know about the differences in the way modeling is performed between the two programs. You may want to read about some of the differences between RISA-3D and STAAD that we've documented and discussed in the relevant Help file topics.

A big plus for people who've used STAAD for a while and built up a library of models is that RISA-3D can read STAAD input files. All the details about the files translation are covered in the STAAD File Translation topic.

The translation process will cause model information including geometry data, member and element properties, load information, some advanced modeling information, and AISC steel code check information to be read into RISA-3D.

(STAAD III and STAAD/Pro are registered trademarks of Research Engineers, Inc.)

STAAD Differences from RISA-3D

STAAD is a "batch" mode program, where you are building a text input file either by hand or using their pre-processor. The latest STAAD/Pro program is very nearly an interactive program, with the only external programs being the solvers. RISA-3D is completely an interactive program in that we do not write an intermediate file. All input, solution, and results are performed using the same program.

For manual data entry, the input file can be directly edited in STAAD, whereas in RISA-3D you edit your data manually in custom spreadsheets that error check your input as it goes in. RISA-3D also has many built in spreadsheet functions to assist manual editing of the model data. You can cut and paste from other programs and spreadsheet directly into the RISA-3D data spreadsheets. The *.R3D file format uses keyword delimited format that may be edited directly from a text editor such as notepad. See Appendix D for more information on the RISA file format. Directly editing the *.R3D file also bypasses many of the error-checking features that would catch syntactical errors in the model data (having your model data integrity assured before you even run the model will save you lots of time in the long run).

Member Data

RISA-3D uses a Section Set to relate a set of members to a particular shape. The analog in STAAD is their "Groups". The RISA-3D Section Set combines a material and a shape into a one entity, which is then assigned to members. In RISA-3D, steel redesign is performed on a Section Set basis, so the worst-case member in a section set will control the size of all the members in that set.

Load Data

STAAD solves Primary LOAD cases and LOAD Combinations; RISA-3D only solves Load Combinations. If you'd like to run all your Basic Load Cases in addition to your Load Combinations, you will need to set up additional load combinations that only contain one basic load case per combination. RISA-3D has a limit of 8 Basic Load Cases per Load Combination. To learn how to get more than 8 BLC's per Load Combination, see Nesting Load Combinations for more information.

Analysis Types

A P-Delta analysis is a specific analysis option for a STAAD file and usually applies to all the loads in the current data file. In RISA-3D, a P-Delta analysis can be specified by setting a P-Delta flag for each combination on the Load Combination spreadsheet where you would like to include 2nd order effects.

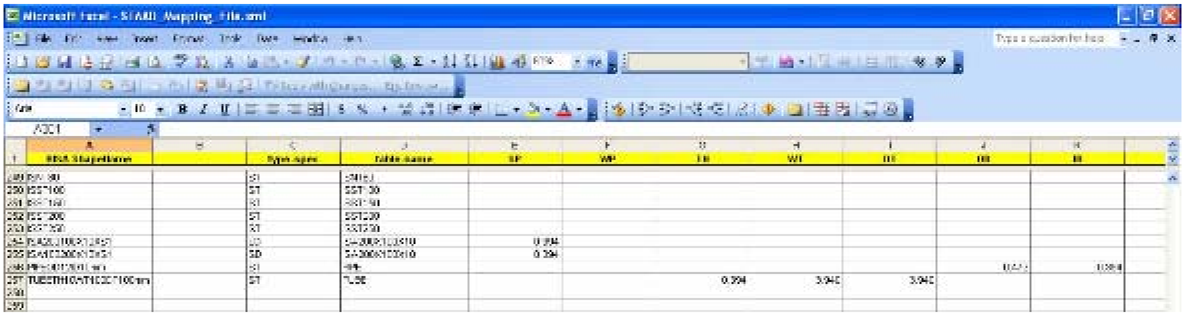
Dynamics/Response Spectrum

Any response spectra used by your STAAD file will need to be entered in RISA-3D's spectra database. STAAD stores each spectra with a particular data file, whereas RISA-3D maintains a library of spectra which are accessible from any data file.

For dynamic analysis in RISA-3D, mass is assigned in the vertical direction and then assumed to act in all three global directions. In STAAD, you have to specify your mass in all the directions in which you want it to act.

STAAD Mapping File

The STAAD importing feature now accepts a mapping file (STAAD_Mapping_File.XML). This file format allows RISA to import User defined database or table shapes from STAAD. All parameters in the tables are assumed to be based on units of kips and inches.



Column Headers:

RISA Shape Name Shape Name in RISA Programs
type-spec Can be any of the following : ST, RA, D, LD, SD, T, CM, TC, BC, TB
table-name Table section name like W6X9, C9X15 etc.
SP Spacing between angles
WP Width of Cover Plate
TH Thickness of plate or tubes
WT Width of tubes
DT Depth of tubes
OD Outer Diameter
ID Inner Diameter
CT Concrete Thickness for Composite Sections
FC Compressive Strength of Concrete for Composite Sections.

Appendix D – File Format

RISA-3D Versions 5.5 and above use a plain text, keyword driven input file. The input data is delimited by sets of keyword labels that divide the input into a number of segments containing semantically related data.

Each input segment may include one or more input records. The input records are text strings that define sets of properties for single input items, e.g., label, coordinates, etc. for nodes. Each record is terminated with a semicolon (;), except for a few project description records (between the [.PROJECT_DESCRIPTION] and [.END_PROJECT_DESCRIPTION] labels). Those records are delimited by their corresponding keyword labels.

Multiple records are used to describe multiple instances of input items. The number of records immediately follows the keyword label and is given in the <> brackets.

Lines starting with a // are ignored by the program and may be used for user comments.

The reading of Label fields (such as member names, or shape names) uses a combination of fixed length fields and field delimiter. The field lengths are set in the Label Length Data. Care should be taken to maintain the proper length for all label fields. If the length of the field is not entered correct, the program will attempt to use the field delimiters (") to read the data. A combination of incorrect field length and the use of (") marks within a shape label itself (e.g. a shape label of: 3/4" rod bracing) will result in the program producing an error during the reading of the file.

*Please refer to the **Help File** in RISA-3D for detailed information about the file format. The Help File may be accessed by selecting **Help Topics** from the **Help Menu**.*

Appendix E - Interface w/ Other Programs

RISA-3D interacts with a number of other RISA programs within the RISA suite (see below) as well as a the 3rd party programs listed in the sections below.

Integration with other RISA programs

For information on the integration between Floor and 3D see RISA-3D Integration.

For information on the integration between RISA-3D and RISASection see Shape Databases.

For information on the integration between RISA-3D and Foundation see RISAFoundation Integration.

For information on the integration between RISA-3D and RISAFoot see RISAFoot Integration - Design.

For information on the integration between RISA-3D and RISAConnection see RISAConnection Integration

Linking your Autodesk Revit Structure model with RISA-3D

AutoDesk® Revit® Structure users can now link directly with RISA-3D and/or RISAFloor. This link is being continuously improved and updated. Therefore, for the most up to date information on this link, please visit the AutoDesk and RISA web sites.

www.risatech.com/partner/revit_structure.asp

www.autodesk.com/revitstructure

For additional advice on this topic, please see the RISA News website: www.risanews.com. Type in Search keyword: **Revit**.

Importing or Exporting CIS/2 Files

RISA-3D and RISAFloor have the ability to import and export files in the CIS/2 file format (.stp file extension). This is a generic file format that allows different pieces of software to communicate with each other. Our three main partners that accept and produce this type of file are:

- SDS/2 Design Data
- Tekla Structures
- StruCad

For more information and to download the CIS/2 translator, visit our website at: www.risatech.com/p_cis2.html

Importing or Exporting DXF Files

For DXF importing / exporting, refer to the DXF Files section of the main manual.

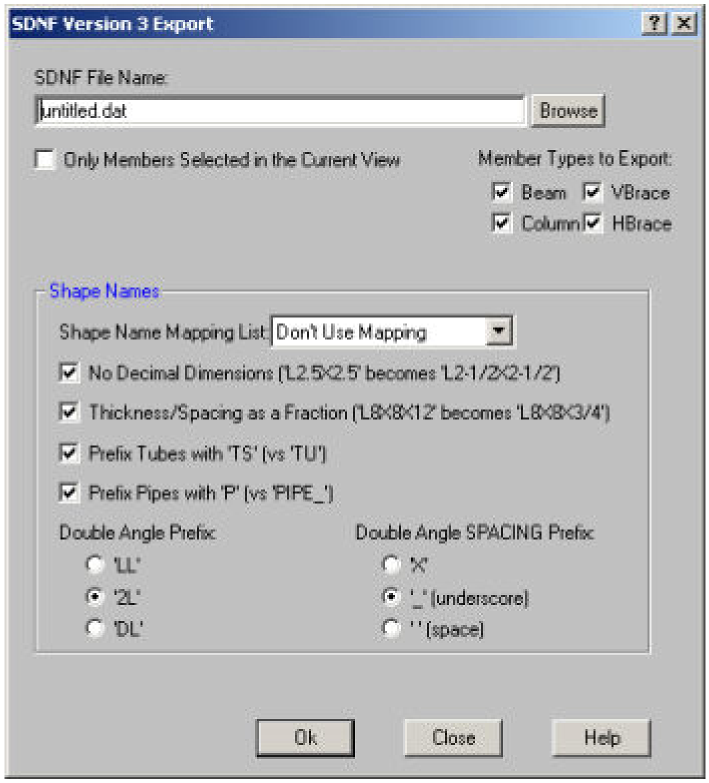
Importing STAAD Files

Refer to Appendix C for more information on importing STAAD files.

Exporting to an SDNF File Format

The Steel Detailing Neutral File is a file format that is used by a number of CAD and steel detailing packages. This format is superior to a DXF export in that it contains member size and orientation information as well. The SDNF export dialog may be obtained by selecting File - Export - SDNF from the Main Menu Toolbar.

Because the SDNF data exchange is so heavily dependent on naming conventions for the structural steel, there are a number of options related to the steel shape prefixes and naming conventions as shown in the dialog below.



Note

- SDNF exporting includes member information only. It does not export Wall Panel Elements, solids or plates.
- The SDNF exporting does not link to the Member Detailing information. Setting cardinal points and offsets in the Member Detailing will have no effect on this export.

Structural Desktop

Structural Desktop is a 3rd party program that works inside AutoCAD and is capable of importing RISA models into AutoCAD for the purposes of drawing production. Structural Desktop is also capable of exporting RISA files as well.

Structural Desktop also includes a library of steel joists that can be used directly inside of RISA-3D or 2D. Refer to the Shape Database - General Shapes section of the manual for more information.

For more information, see the RISA and Structural Desktop web sites:

www.risatech.com/partner/sdt.asp

www.structuraldesktop.com

Pro-Steel

Files may be exported from RISA-3D into a DXF format, an SDNF format, or a Pro-Steel exchange file. In addition, connection design information may be exported to the Descon connection design program. For more information, refer to the Pro-Steel and RISA web sites:

www.risatech.com/partner/prosteel3d.asp

www.strucsoftsolutions.com

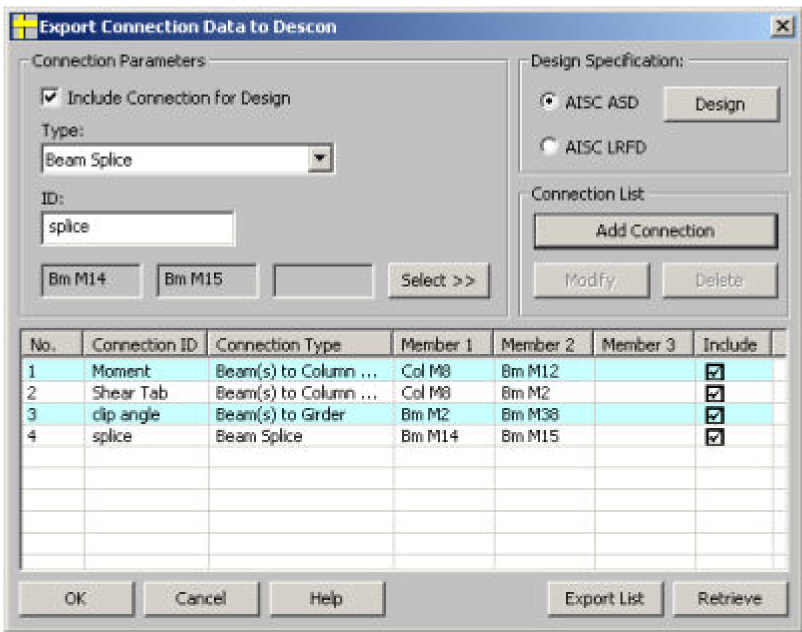
Exporting Connection Data to Descon

The Descon program is a connection design program written by Omnitech Associates, Inc. RISA is capable of exporting Steel shape and force results for multiple connections to the Descon program (vers. 5 or 6) for the purposes of steel connection design.

Note

- Version 6.1 of Descon is not compatible with RISA-3D.
- By default, RISA cannot currently launch the network version of Descon. If you are running a network version please refer to the specific section on that topic below.

The Descon export dialog window may be accessed by selecting File - Export - Descon Connection from the Main Menu Toolbar. This will bring up the dialog displayed below.



Connection Parameters:

This section consists of information related to the design of a single connection.

The **Include Connection for Design checkbox** indicates whether this connection information will be included in the Descon design. Otherwise the connection can be defined and reserved for future use.

The **Type** menu merely lists the type of Descon connection that will be designed. Refer to the Descon program for more information about the definition of each of these connections and how they will be designed.

The **ID** field is a text description that is intended solely to help the users arrange and identify their connections. The user must define a unique ID for each connection in the file.

The 

button initiates the graphical selection of the connecting members. The Status bar will direct the user to select the 1st, 2nd or 3rd member associated with the connection. The primary member (usually a Column for beam to column connections, or the girder for beam to beam connections) should be selected first. The secondary members (beams or joists) should be selected next. If there are only two members in a connection, then the user can right-click the mouse when he/ she is done selecting their two members to terminate the selection tools.

Design Specification:

This section indicates whether the connection design should be performed with the LRFD or ASD design code. When all the connections have been specified, the **Design** button should be clicked to hand the information over to the Descon program. At this point RISA-3D will generate a temporary file according to the latest info available in the memory and feed it to Descon. The format of the generated temporary file is the same as the file that will be generated if the user hits Save/Export List but it is always updated with the latest internal forces. The temporary file will be deleted as soon as the Descon program terminates.

For Descon to design or check a connection, some complementary information may need to be defined at the run time. The user will have the option to save the Descon input file from within the Descon program.

Connection List:

This section of the dialog consists of a grid / spreadsheet with all the currently designated connections and the tools needed to **Add**, **Modify** or **Delete** connections from the list. This list of connections is only stored during the current RISA session and will be cleared upon exiting. Therefore, it is suggested that you use the **Save List** button to save your connections whenever you make any significant changes. That way, you may use the **Retrieve** button to recall all the previously entered connection data.

The **Add Connection** button only becomes available after the user has filled sufficient information for the design of the most basic connection. When the add connection button is pressed, the program checks the connection validity and if all validity procedures pass then the connection info will add to the list. This validity check will perform at the time of adding connection, at the time of modifying info and at the time of launching Descon.

Exporting and Retrieving Connection Data:

The Export List button will save the connection and force data to an ASCII text file. This file can later be retrieved by RISA, or it can be read directly by the Descon program. This Export List option is currently the only way that RISA can work with the network version of Descon.

To retrieve this information directly from Descon, the descon program must be run from a command line (or using the windows Run command) using the name of the exported text file as an argument in the command line. For example the run command may look something like the following (where connection5.txt is the name of the file exported from the RISA program):

C:\Program Files\Descon\DesconWin6 connection5.txt

Working With the Network Version of Descon

The network version of Descon does not populate the Windows registry in the same way that the stand-alone program does. Therefore, RISA is not able to automatically launch the program for connection design. There are two possible solutions to this problem.

First, the user may export the connection to a connection list and launch Descon from a command like as described in the section on Exporting and Retrieving Connection Data above.

Second, the user can attempt to create the proper Descon registry settings manually. For a list of the required registry settings contact either the Descon technical support or RISA technical support.

Appendix F – Wood Shear Wall Files

RISA-3D has design databases for wood shear walls and diaphragms which are used to optimize the nailing and hold downs for the wall. The criteria used for this optimization is specified on the Wood Wall (Fasteners) and Wood Diaphragms tabs of the **Design Rules** spreadsheet. In addition to this basic criteria, the user may specify a subset of the overall database from which the design / optimization must be performed.

Hold Downs

Each database of hold downs is specified by an XML file in the "Hold Downs" sub-directory of the Wood Wall panels directory. The location of this directory is based on the information in the File Locations tab of the Tools - Preferences dialog.

The program comes pre-loaded with three XML files, each of which contain a database of commonly used hold downs: the Simpson HDA hold downs, Simpson HDU hold downs, and the Simpson HTT hold downs. The name of the XML file itself will be used in the list of databases in the Hold Down Schedule Dialog.

The first sheet of the XML file should always be descriptive of the contents of the database (such as **SIMP HTT Database**). This is because the name used here is the name used in the Design Rules spreadsheet. This sheet contains all of the identifier, design and code check information used for each hold down. These entries are described below.

Full Database - Required Fields

The following fields are *required* information. If they are not provided or are left blank, then that hold down will not be available for use in that database.

The **Label** field is used to identify the hold down. This field must be referenced on the sheets that identify families or groups of hold downs.

The **Deflection at Peak Load** entry is used to calculate the deflection of the shear wall per APA / NDS formulas. This deflection is then reported on the shear wall detail report for each wall panel.

The **CD Factor** is the assumed load duration factor that was used as the basis for specifying the listed **Allowable Tension** value for that hold down.

A load combination may be solved with a load duration factor different from the CD Factor described above. When this is the case, the **Allowable Tension** for that hold down will be adjusted based on the difference between the assumed and actual load duration factors.

Full Database - Optional Fields

The following fields are optional. They are not currently used in the design or capacity calculations, but are reported on the detail reports for reference purposes only.

The **Manufacturer** field is an identifier for the hold down. It is provided so that the engineer can more easily identify the callouts for their final design drawings.

The allowable capacity of the hold down will vary based on the Chord Thickness. Therefore, the **Required Chord Thickness** gives the minimum chord thickness that will yield the listed allowable tension load. However, this field is NOT currently used in the calculations. A future revision may provide a warning message if the actual chord thickness provided is less than required.

The allowable capacity of the hold down will vary based on the density of the wood species being used. Therefore, the **Required Chord Density** lists the density assumed for the entered allowable tension. However, this field is NOT currently used in the calculations. A future revision may provide a warning message if the actual chord density provided is less than required.

The **AB Diameter** is not currently used in the design calculations and is reported for display purposes only.

The **Bolt Size** when specified is used to reduce the axial capacity of the hold down chord itself. The only change to the calculation is that the program will perform the allowable tension check on the net area of the chord member rather than the gross area. The **Nail Size**, on the other hand, is NOT assumed to affect the tension capacity of the hold down chord.

The **Number of Bolts** and **Number of Nails** are not used in the design calculation and are reported for reference purposes only.

The **Is Hold Down Raised?** field is not used in the design calculations and is reported for display purposed only. It is assumed that if the hold down deformation is significantly affected by the connection being flushed or raised, then the **Deflection at Peak Load** entry described in the previous section will adjusted instead.

Grouping Hold Down Schedules for Design Optimization

The other sheets allow the user to group hold downs together into families for optimization purposes. These additional sheets CANNOT be the first sheet in the XML file as that first sheet must always be the one where the full database information resides.

The hold down labels specified on these sheets refer only to hold downs that have already been defined on the full database sheet. The information in this sheet need not be organized in a specific order. Instead, they will always be optimized based on the assumption that the hold down cost is directly related to the tension capacity. Therefore, when this group is selected, then the hold down within the group with the code check closest to unity, but still less than 1.0 will get selected during the optimization process.

Panel Nailing Schedules

Each database of wall panels is specified by an XML file in the "Shear Panels" sub-directory of the Wood Wall panels directory. This directory is located based on the information File Locations tab of the Tools - Preferences dialog.

The program comes loaded with two XML files one for the tabularized nailing schedules listed in the 1997 UBC and one for the ones listed in the 2006 IBC.

The first sheet of the XML file should always be descriptive of the contents of the database (such as **IBC 06 Panel Database**). This is because the name used here is the name used in the Design Rules spreadsheet. This sheet contains all of the identifier, nailing, design and code check information for each nailing schedule. These entries are described below.

Full Database - Required Fields

The following fields are *required* information. If they are not provided or are left blank, then that nailing schedule will not be available for use in the database.

The **Label** field is a used to identify the panel schedule and its nailing requirements. This field must be referenced on the sheets that identify families or groups of panels.

Note:

- Panels with a label containing the characters "_W" together will be ignored during design optimization.

The **Min Panel Thickness** is used during the design optimization to limit the selected panels based on the Design Rules chosen by the user. It is also used to help set the elastic stiffness of the wall panel used during the FEM solution.

The **Ga** value is the **Apparent Shear Stiffness** from nail slip and panel deformation as defined in equation 4.3-1 of the NDS' Special Design Provisions for Wind and Seismic. This value (in combination with the Min Panel Thickness defined above) is used to set the elastic stiffness of the wall panel that will be used during the FEM solution.

Note:

- When a family or group of panels / nailing schedules are assigned to a shear wall, the lowest value of Ga and Min Panel thickness will be used to determine the elastic stiffness of the plate elements in the FEM solution.

The **One/Two Sided** field is used during the design optimization to limit the available panels based on the Design Rules specified by the user.

The **Boundary Nail Spacing** field is used during the design optimization to limit the available panels based on the Design Rules specified by the user.

Note:

- The maximum field spacing is never entered in the program but is generally equal to 12 inches for the nailing schedules defined in the 1997 UBC and 2006 IBC databases. If a different nail spacing is present, then the user should add in a new nailing schedule to the existing database with a user defined shear capacity.

The **Shear Capacity** listed in the spreadsheet is the primary value that controls the code checking of the shear wall. This is the seismic capacity, which the program can automatically increase for wind loads if the Wind ASIF function is enabled.

Full Database - Optional Fields

The following fields are optional. They are not currently used in the design or capacity calculations, but are reported on the detail reports for informational purposes only.

The **Panel Grade** and **Min Penetration** fields are identifiers for the engineer, but are not used in the design calculations. They are provided so that the engineer can most easily identify the panels in their design results and drawings.

The **Panel Applied Over Gypsum** field is also a identifier for the engineer that will not be used in the design calculations.

The **Nail Size** listed in the spreadsheet is intended to refer to the Common nail size, but is reported only for reference purposes and are NOT used in the capacity calculations. If the nail size is changed by the user, then the user should also change the Shear Capacity entry accordingly. Below is a reference table for common, box, and sinker nails.

	Diameter				Length	
Penny Weight	Common (in)	Common (Wired Gage)	Box (in)	Sinker (in)	Box and Common (in)	Sinker (in)
6d	0.113	11.5	0.099	0.092	2	1.875
7d	0.113	11.5	0.099	0.099	2.25	2.125
8d	0.131	10	0.113	0.113	2.5	2.375
10d	0.148	9	0.128	0.120	3	2.875
12d	0.148	9	0.128	0.135	3.25	3.125
16d	0.162	8	0.135	0.148	3.5	3.25

The Staple size listed in the database is reported for reference purposes only. If the staple size is entered or changed by the user, then the user should also change the shear capacity entry to the appropriate value.

Grouping Panel / Nailing Schedules for Design Optimization

The other sheets in the database allow the user to organize multiple nailing schedules into groups or families for design optimization purposes. These additional sheets CANNOT be the first sheet in the XML file as that first sheet must always be the one where the full database information resides.

The panel labels specified on these sheets refer only to panel / nailing schedules that have already been defined on the full database sheet. The information in this sheet need not be organized in a specific order. Instead, they will always be optimized based on the assumption that the installed cost is directly related to the shear capacity. Therefore, when a group or family is selected, then the nailing schedule within the group with the code check closest to unity, but still less than 1.0 will get selected during the optimization process.

Diaphragm Nailing Schedules

Each database of diaphragms is specified by an XML file in the "Diaphragms" sub-directory of the Wood Wall schedules directory. This directory is located based on the information on the File Locations tab of the Tools - Preferences dialog.

The program comes loaded with two XML files, both based on the shear values of the 2006 IBC and the stiffness values of the 2005 NDS *Special Design Provisions for Wind and Seismic*. The WSP (Wood Structural Panel) is intended to be a generic database that could be used for plywood or OSB panels. However, it uses the Ga (apparent stiffness) values for plywood because they are generally lower resulting in a more conservative deflection.

Note:

- Diaphragm design is currently only available for flexible diaphragms that were created in RISAFloor and brought into RISA-3D.

The first sheet of the XML file should always be descriptive of the contents of the database (such as IBC 06 OSB Database). This is because the name used here is the name used in the Design Rules spreadsheet. This sheet contains all of the identifier, nailing, design and code check information for each nailing schedule. These entries are described below:

Full Database - Required Fields

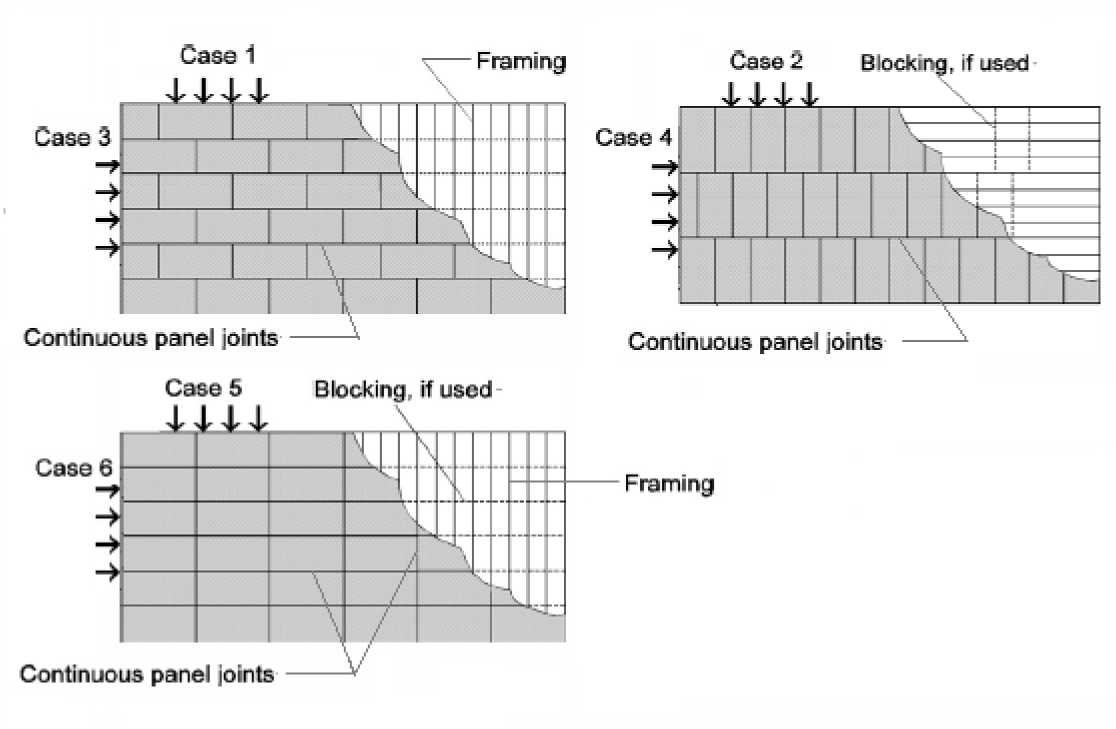
The following fields are required information. If they are not provided or are left blank, then that nailing schedule will not be available for use in the database.

The **Label** field is a used to identify the diaphragm nailing. This field must be referenced on the sheets that identify families or groups of panels.

Note:

- Panels with a label containing the characters "_W" together will be ignored during design optimization.

The **Case** field is used to specify the layout of the shear panels as shown below. Any diaphragm that has a Case 1 layout also has a Case 3 layout, and the same goes for 2/4 and 5/6.



The program considers the deck span defined in RISAFloor to coincide with the long direction of the plywood. Therefore, Cases 2, 3 and 6 are considered parallel to the to the RISAFloor deck span. Whereas, Cases 1, 2 and 5 would be considered perpendicular to the same deck span.

The **Blocked** field specifies whether blocking is used to achieve the associated design strength. It also determines the method by which the diaphragm deflection will be calculated. For more information see Diaphragm Deflection.

The **Panel Grade** field specifies what grade of structural panel is used in the diaphragm. This may be set to either “Structural-I”, "Rated Sheathing" or “Other” and is used as a criteria in the Design Rules.

The **Panel Thickness** field specifies the thickness of the structural panel used for the diaphragm. This is a decimal value that is rounded to four places for reporting convenience. For example, a 15/32” panel is listed as 0.4688

The **Boundary/Cont Edge Spacing** field specifies the nail spacing at the boundary and along any continuous edges. These must be specified as the same value.

The **Other Edge Spacing** field specifies the nail spacing at non-continuous edges.

The **Nail Lines** field specifies the number of lines of nails along each panel edge. This value is greater than (1) for High Load diaphragms. This value is not currently used in design optimization, but is reported on the output for reference purposes only.

The **Strong Shear Capacity** field specifies the shear strength of the diaphragm (lbs/ft) based on its stronger case. For example, while Case 1/3 represents the same panel layout, Case 1 has greater strength than Case 3 (based on load direction). This is the seismic capacity, which the program can automatically increase for wind loads if the Wind ASIF function is enabled.

The **Weak Shear Capacity** field specifies the shear strength of the diaphragm (lbs/ft) based on its weaker case. There are many situations where strong and weak capacities are identical. In these cases the same value must be specified for both fields. This is the seismic capacity, which the program can automatically increase for wind loads if the Wind ASIF function is enabled.

The **Strong Ga** field specifies the apparent shear stiffness (kips/in) of the diaphragm as specified in the NDS document *Special Design Provisions for Wind and Seismic* Since this is the strong direction it will be based on the stronger direction / case for loading. For example, while Case 1/3 represents the same panel layout, Case 1 has greater stiffness than Case 3 (based on load direction). For more information see Diaphragm Deflection.

The **Weak Ga** field specifies the apparent shear stiffness (kips/in) of the diaphragm as specified in the NDS document *Special Design Provisions for Wind and Seismic*.

The **Gt** field specifies the shear stiffness of panel depth. It is always the same for both strong weak directions, hence it does not need to be specified twice. For more information see Diaphragm Deflection.

The **Strong Nail Slip (en)** field specifies the nail slip used for deflection calculations based on the stronger case. For example, while Case 1/3 represents the same panel layout, Case 1 may have less nail slip than Case 3 (based on load direction). For more information see Diaphragm Deflection.

The **Weak Nail Slip (en)** field specifies the nail slip used for deflection calculations based on the weaker case. There are many situations where strong and weak nail slips are identical. In these cases the same value must be specified for both fields. For more information see Diaphragm Deflection.

Note:

- The Gt and Nail Slip fields are ignored for unblocked diaphragms.

Full Database - Optional Fields

The following fields are optional. They are not currently used in the design, capacity or deflection calculations, but are reported on the detail reports for informational purposes only.

The **Framing Width** field identifies the minimum required framing width for the nailing layout. A higher shear capacity can typically be achieved for a diaphragm by using wider supporting framing, thereby reducing the tension perpendicular to the grain of supporting members.

The **Minimum Penetration** field identifies the minimum required nail penetration specified in the IBC/NDS tables.

The **Nail Size** listed in the spreadsheet is intended to refer to the Common nail size, but is reported only for reference purposes and are NOT used in the capacity calculations. If the nail size is changed by the user, then the user should also change the Shear Capacity entry accordingly. The section on shear walls contains a good reference table for common, box and sinker nails.

Grouping Panel / Nailing Schedules for Design Optimization

The other sheets in the database allow the user to organize multiple nailing schedules into groups or families for design optimization purposes. These additional sheets CANNOT be the first sheet in the XML file as that first sheet must always be the one where the full database information resides.

The labels specified on these sheets refer only to nailing schedules that have already been defined on the first (full) database sheet.

Technical Support

Technical support is an important part of the RISA-3D package. There is no charge for technical support for all licensed owners of the **current** version of RISA-3D. Technical support is very important to the staff at RISA Technologies. We want our users to be able to reach us when they are having difficulties with the program. However, this service is not to be used as a way to avoid learning the program or learning how to perform structural modeling in general.

Hours: 6AM to 5PM Pacific Standard Time, Monday through Friday

Before contacting technical support, you should typically do the following:

1. **Please search the Help File or General Reference Manual**. Most questions asked about RISA-3D are already answered in the Help File or General Reference Manual. Use the table of contents or index to find specific topics and appropriate sections. We go to great lengths to provide extensive written and on-line documentation for the program. We do this in order to help you understand the features and make them easier to use. Also be sure to go through the entire User's Guide when you first get the program.
2. If you have access to the Internet, you can visit our website at **www.risatech.com** and check out our **Support** section for release notes, updates, downloads, and frequently asked questions. We list known issues and product updates that you can download. So, if you think the program is in error you should see if the problem is listed and make sure you have the latest release. The **FAQ** **(Frequently Asked Questions)** section may also address your question.
3. Make sure you understand the problem, and make sure your question is related to the program or structural modeling. Technical Support does not include free engineering consulting. RISA Technologies does provide a consulting service. If you are interested in inquiring about this service, please call RISA Technologies.
4. Take a few minutes to experiment with the problem to try to understand and solve it.

For all modeling support questions, please be prepared to send us your model input file via email or postal mail. We often will need to have your model in hand to debug a problem or answer your questions.

Email: **support@risatech.com** This method is the **best** way to send us a model you would like help with. Most email packages support the attachment of files. The input file you would send will have a .R3D extension. Make sure you tell us your name, company name, serial number or Key ID, phone number, and give a decent problem description. If you have multiple members, plates, or load combinations, make sure you specify which ones to look at.

Phone Support:(949) 951-5815: Feel free to call, especially if you need a quick answer and your question is not model specific and therefore doesn't require us to look at your file.

Index

C

D

E

F

G

H

I

M

N

O

P

Q

R

S

T

U

V

W

X

Y

Z