# Overview

## Table of Contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Installation</td>
<td>1</td>
</tr>
<tr>
<td>Disclaimer</td>
<td>1</td>
</tr>
<tr>
<td>License Agreement</td>
<td>1</td>
</tr>
<tr>
<td>Technical Support</td>
<td>2</td>
</tr>
<tr>
<td>Maintenance</td>
<td>2</td>
</tr>
<tr>
<td>Basic Load Cases</td>
<td>4</td>
</tr>
<tr>
<td>Boundary Conditions</td>
<td>5</td>
</tr>
<tr>
<td>Configuration</td>
<td>9</td>
</tr>
<tr>
<td>Diaphragms</td>
<td>11</td>
</tr>
<tr>
<td>Distributed Loads</td>
<td>15</td>
</tr>
<tr>
<td>Story Drift</td>
<td>18</td>
</tr>
<tr>
<td>DXF Files</td>
<td>20</td>
</tr>
<tr>
<td>Dynamic (Modal) Analysis</td>
<td>25</td>
</tr>
<tr>
<td>Generation</td>
<td>31</td>
</tr>
<tr>
<td>Global Information</td>
<td>42</td>
</tr>
<tr>
<td>Help Options</td>
<td>44</td>
</tr>
<tr>
<td>Load Combinations</td>
<td>45</td>
</tr>
<tr>
<td>Material Properties</td>
<td>50</td>
</tr>
<tr>
<td>Material Take-Off Results</td>
<td>51</td>
</tr>
<tr>
<td>Members</td>
<td>51</td>
</tr>
<tr>
<td>Member Results</td>
<td>58</td>
</tr>
<tr>
<td>Model Merge</td>
<td>63</td>
</tr>
<tr>
<td>Modeling Tips</td>
<td>66</td>
</tr>
<tr>
<td>Moving Loads</td>
<td>73</td>
</tr>
<tr>
<td>Nodes</td>
<td>76</td>
</tr>
<tr>
<td>Node Loads - Enforced Displacements</td>
<td>79</td>
</tr>
<tr>
<td>P-Delta Analysis</td>
<td>80</td>
</tr>
<tr>
<td>Plate/Shells</td>
<td>83</td>
</tr>
<tr>
<td>Point Loads</td>
<td>94</td>
</tr>
<tr>
<td>Printing</td>
<td>95</td>
</tr>
<tr>
<td>Response Spectra Analysis</td>
<td>97</td>
</tr>
<tr>
<td>Sections Sets</td>
<td>104</td>
</tr>
<tr>
<td>Shape Database</td>
<td>107</td>
</tr>
<tr>
<td>Slaving Nodes</td>
<td>111</td>
</tr>
<tr>
<td>Stability</td>
<td>112</td>
</tr>
<tr>
<td>Steel Design</td>
<td>117</td>
</tr>
<tr>
<td>Steel Design Code - Optimization</td>
<td>132</td>
</tr>
<tr>
<td>Surface Loads</td>
<td>134</td>
</tr>
<tr>
<td>Technical Support</td>
<td>137</td>
</tr>
<tr>
<td>Thermal Loads</td>
<td>140</td>
</tr>
<tr>
<td>Timber Design</td>
<td>142</td>
</tr>
<tr>
<td>Torsion</td>
<td>152</td>
</tr>
<tr>
<td>Units</td>
<td>156</td>
</tr>
<tr>
<td>STAAD File Translation</td>
<td>158</td>
</tr>
</tbody>
</table>
Installation

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

To install RISA-3D please follow these instructions:

1) Put the RISA-3D disk 1 in your computers “A” (3.5inch) drive.
2) Click the Windows Start button and select Run.
3) In the Run dialog box type “a:\setup” and then click the OK button.
4) Follow the on-screen instructions.

After you have installed RISA-3D, be sure to send us your registration card!

Disclaimer

We intend that the information contained in this manual and the RISA-3D computer program 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 RISA-3D and any related software programs.

RISA-3D is intended for use by professional engineers and architects who possess an understanding of structural mechanics.

In no event will RISA Technologies or its officers be liable to anyone for any damages, including any lost profits, lost savings or lost data. In no event will RISA Technologies or its officers be liable for incidental, special, punitive or consequential damages or professional malpractice arising out of or in connection with the usage of the RISA-3D computer program or any other computer programs or documentation, even if RISA Technologies or its officers have been advised of or should be aware of the possibility of such damages.

If you do not agree with the terms of this disclaimer or the terms of the License Agreement, return this manual and the program disk to RISA Technologies within thirty days for a full refund. You also must delete any installations or copies of RISA-3D.

License Agreement

Copyright © 1985-1999 by RISA Technologies. All rights reserved.
This document may not be duplicated in any way without expressed written consent of RISA Technologies. Making copies of this document or any portion of it is in violation of United States copyright laws and various international treaties.

The computer software program RISA-3D provided to you on electronic media is protected by United States copyright laws and various international treaties. You are specifically limited to the use of this program on no more than one CPU at any given time. The network edition of RISA-3D is licensed for simultaneous use on a certain maximum number of network stations; this maximum number of stations may vary on a per license basis.

As part of the license to use RISA-3D, the program user acknowledges the reading, understanding and acceptance of all terms of the Disclaimer listed above.

RISA-3D may not be reviewed, compared or evaluated in any manner in any publication without expressed written consent of RISA Technologies.

You may not disassemble, decompile, reverse engineer or modify in any way the RISA-3D executable modules.

The terms of this license agreement are binding in perpetuity.

**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 the Technical Support section in the Help File for a list of your support options.

The “life of the program” is defined as the time period for which that version of the program is the current version. In other words, whenever a new version of RISA-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.

**Maintenance**

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

The first year of program maintenance is included in the purchase price. After the first year, you will be given the opportunity to continue program maintenance on an annual basis. You are under no obligation to continue program maintenance, of course, but if you decide to discontinue maintenance you will no longer receive RISA-3D program upgrades.

If you let your maintenance lapse and decide later that you wish to reinstate it, you will have to “catch up” on the maintenance fees with the following terms: If your maintenance expired more than a year ago, you will need to pay the
maintenance fee for the previous year, plus the maintenance fee for the coming year, for a total of two years, to get reinstated. You would then receive maintenance for the next year. If your maintenance expired less than a year ago, you would only need to pay the maintenance fee for the current year to get reinstated, but your maintenance would expire again once the current year is up.
Basic Load Cases

When loads are defined they are grouped into separate sets of basic load cases (BLC’s). These are the basic building block 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 box. 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.

You are allowed up to 1000 separate Basic Load Cases.

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

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 listed below.

The Basic Load Case Spreadsheet has nine columns. 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.

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 box and 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 that 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 four 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 load that it represents. For example, if you had nine joint loads as part of BLC 2 you would see the number 9 in the second line of the Joint Loads column. Clicking on the number 9 with the mouse will open the Joint Loads spreadsheet and display the loads for BLC 2.
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.

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 . You will be presented with a dialog from which you may 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.

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 restraining or partially restrained with a spring. You can define a spring support as only having stiffness in one direction with tension-only or compression-only springs.

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. For help on an item, click and then click the item.

3 You may choose to apply the boundary condition to a single node at a time or to an entire selection of nodes.
To apply the boundary condition to a few nodes choose **Apply Entries by Clicking Items Individually** and click **Apply**. Click on the nodes with the left mouse button.

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

**Note**
- To apply more boundaries with different conditions, press CTRL-D to recall the Boundary Conditions dialog.
- You may also specify or edit boundary conditions in the Boundary Spreadsheet.

**To Display Boundary Conditions**
- Toggle the display of boundary conditions by clicking  on the Window Toolbar.
- You may specify what is displayed by clicking Plot Options  on the Window Toolbar.

**Boundary Spreadsheet**

Boundary conditions are recorded on the **Boundary** spreadsheet. The first column contains the label of the node that is restrained. The remaining columns record the boundary conditions that apply to the node. There are six degrees of freedom for each node (3 translation, 3 rotation), so there are six columns. 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.

There are two fixed options that are essentially the same. Both provide full restraint for the indicated direction. The difference between the two is that for the second one no reaction is calculated. This condition actually removes the degree of freedom from the solution. The "Reaction" condition retains the degree of freedom in the solution. If you aren't interested in reactions, using the removing the degree of freedom will result in a slightly smaller model and less output.

There are three types of springs, all requiring an accompanying spring constant. Compression-only springs have stiffness for negative displacements and NO stiffness for positive displacements. Compression-only springs are useful as soil springs when analyzing foundations that may have uplift. Tension-only springs have stiffness for positive displacements and NO stiffness for negative displacements.

When a node is slaved, it is linked to another nodes (the "master" node). The slave and master nodes actually share the same degree of freedom for the direction of slaving. The slaving can be for any or all of the global degrees of freedom. A node can be slaved to more than one master (in different directions). Any number of nodes may be slaved to the same master, but the master node itself may not be slaved to another master.
Boundary Conditions

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. For models with lots of one-way springs, this can take a quite a bit longer than a regular static solution.

- You can enter the first letter of the code ("R" for Reaction, "TS" for Tension Only 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 Nodes
The entry "ALL" may be entered in the Node Label field. The boundary conditions entered on this line will be applied to ALL the nodes not listed otherwise. 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, θx Rotation and θy Rotation. See the following figure:

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 nodes 1 and 2 in the figure above also have the Fixed code in the Z translation, θx Rotation and θy Rotation fields.

Reaction Boundary Condition
The "R" code (Reaction) specifies full restraint for the indicated direction. No movement will be allowed in the indicated direction for this node. Further, by entering an "R", you are specifying that the reaction be calculated at this node, for this direction.

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

Spring Boundary Condition
The "Snnn" code models a spring attached to the node in the indicated direction. The "nnn" is the spring 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 node, for that node you would enter S 1000 for the X direction boundary condition.

Compression Only Springs
The "CSnnn" code models a one way “compression-only” spring attached to the node in the indicated direction. This spring has stiffness for negative displacements and NO stiffness for positive displacements. The “nnn” is the spring 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 500 Kips per Inch were desired in the Y direction at a certain node, you would enter

   CS 500

for that nodes Y direction boundary condition. This means that all displacements at this node in the negative Y direction will be resisted with a stiffness of 500 Kips per Inch. The node is, however, free to translate in the positive Y direction.

Tension Only Springs
The "TSnnn" code models a one way “tension-only” spring attached to the node in the indicated direction. This spring has stiffness for positive displacements and NO stiffness for negative displacements. The “nnn” is the spring 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 500 Kips per Inch were desired in the Y direction at a certain node, you would enter

   TS 500

for that nodes Y direction boundary condition. This means that all displacements at this node in the positive Y direction will be resisted with a stiffness of 500 Kips per Inch. The node is, however, free to translate in the negative Y direction.

Story Drift Nodes
The Boundary spreadsheet is also used to record nodes to be used for story drift calculation. For example, to indicate that a particular node is to represent the fourth story level for X direction drift, you would enter “STORY 4” for the X direction boundary condition for the node. These STORY entries may only be made in the translational degrees of freedom.
When RISA-3D begins, it reads a configuration file called "risa3dw.ini" to pass in start-up information. RISA-3D looks in the current (default) directory for risa3d.ini, regardless of where RISA-3D is installed. If risa3dw.ini is not found, RISA-3D will terminate.

If you need to make changes to the risa3d.ini file, you may use any text editor or word processing program that can read and write ASCII text files. A good program for this would be the NOTE PAD program that comes with Windows. You must restart RISA-3D to then see the effects of any changes. Here is what risa3d.ini contains:

**Shapespath=drive\directory**

This record is used to tell RISA-3D where the steel shape database files are to be found. Normally the shape database files are stored in the same directory as RISA-3D, but if you would like to store these files elsewhere use this record to tell RISA-3D where the shape database files are located. If nothing is entered RISA-3D expects the file to be in the current directory, whatever that directory is.

**woodpath=drive\directory**

Similar to the “shapespath”, this record is used to tell RISA-3D where the wood shape database files are to be found. If nothing is entered RISA-3D expects the file to be in the current directory, whatever that directory is.

**r3dpath=drive\directory**

Here you can tell RISA-3D what the default directory for the R3D data files should be. If you put an entry here this entry will appear in the "Look In" field when you go to the File-Open dialog, and this directory will be automatically scanned for R3D files. If nothing is entered RISA-3D looks in the current directory, whatever that directory is.

**Miscpath=drive\directory** (Used for Network Version Only)

This record, only used with the network version of RISA-3D, directs RISA-3D to the numerous auxiliary files it uses during execution. These files would be the help files, the error message files and others. This should point to the drive:<path> where the program was installed. If nothing is entered for this record, RISA-3D expects the auxiliary files to be in the current directory, whatever that directory is.

**mlpath=drive\directory**

This record tells RISA-3D where to find the moving load library file (ML_LIB.FIL). If nothing is entered RISA-3D expects the file to be in the current directory, whatever that directory is.

**Scratchpath=drive\directory**
Sometimes when executing RISA-3D finds it necessary to create temporary disk files. This record tells RISA-3D where to put those temporary files. These scratch files will be erased when RISA-3D is finished with them. Normally the only reason you would want to put an entry for this record is that the current disk does not have sufficient space and you need to direct RISA-3D to another drive.

These scratch files can be as large as 30+ Mbytes and can cause overall network performance to be seriously degraded if placed on the network. It will also cause the execution time for the problem solution to increase substantially, given that the user’s hard drive data transfer rate is generally faster than the average network data transfer rate. Also note that if multiple users have their “scratchpath” set to the same location and they run multiple copies at the same time, the temporary files for each copy will be overwritten, resulting in unpredictable program behavior.

**spectrapath=drive\directory**

This record tells RISA-3D where to find the response spectra library files (RSPECTRA.RDS and RSPECTRA.NDX). If nothing is entered for this record, RISA-3D expects the files to be in the current directory, whatever that directory is.

**savepath=drive\directory**

Use this entry to specify a path for the automatic backup file. If nothing is entered RISA-3D stores the file in the current directory, whatever that directory is. This needs to be a place where the user has write access. This location can be on the network, as long as it is unique for each user.

**autosave=minutes**

This entry is used to control the timing of the RISA-3D automatic backup feature. If nothing is entered here, the automatic backup occurs every 15 minutes. If you want a different timing, just put the number of minutes (from 1 to 60) after this record. You can also use this entry to turn the automatic backup feature off. To do this, enter a timing of "0".

**staadpath=drive\directory**

This record tells RISA-3D where to find the STAAD files when you choose Import from the File menu. If nothing is entered RISA-3D expects the files to be in the current directory, whatever that directory is.

**dxf=drive\directory**

This record tells RISA-3D where to find the dxf files when you choose Import from the File menu. If nothing is entered RISA-3D expects the files to be in the current directory, whatever that directory is.
Diaphragms

RISA-3D has two types of rigid diaphragms, a rigid membrane and a rigid plane. The two different types of diaphragms are provided to handle different modeling situations. The plane diaphragm option is rigid in all 6 degrees of freedom. It is has a very large stiffness in-plane and out-of-plane. The membrane diaphragm option is only rigid in the plane of the diaphragm. It has NO stiffness out-of-plane.

Diaphragm Applications

The rigid diaphragm feature can be used to aid the engineer in quickly modeling the transfer of lateral forces into resisting elements such as shear walls, braced frames, and columns. These loads can be of a static or dynamic nature. Thus, lateral loads can be applied where they really occur on a structure, and the effects of the center of applied force being different from the center of stiffness will be accounted for automatically in the static stiffness solution. Dynamic mass can also be applied where it actually occurs on the structure, and the differences between the center of mass and the center of stiffness will be accounted for automatically as part of the dynamic solution.

To Define a Diaphragm

A rigid diaphragm is defined by a master node and a global plane to be made rigid. Note that only planes parallel to the global axes (XY, YZ, or ZX) may be made rigid. For example, say you enter joint 10 as the master node and specify ZX as the plane. This means a plane passing through node 10 and parallel to the global ZX axis will be established. The actual location of the plane relative to the Y-axis will be the Y coordinate of joint 10. Any joints having the same Y coordinate as joint 10 will be on the plane and will be rigidly linked together. Note that the tolerance for other nodes to be on the same plane is 0.01 ft. You may create partial diaphragms by taking advantage of this tolerance.

You should not apply a boundary condition to a node that is part of a Rigid Diaphragm. This will cause numerical problems in the solution and will probably give erroneous results. The stiffness method requires that there be some meaningful deflection in the model to obtain results. Rigid Diaphragms and boundary conditions are both rigid; these elements should not be attached to one another. If these elements are attached together, nothing is left to deflect properly, and any deformation is likely to be the result of numerical round off.

At this time RISA-3D does not display the center of rigidity and center of mass for a diaphragm (Although it does display the center of gravity for all applied vertical loads along with the reaction results). 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 will have to model any accidental torsion.
DIAPHRAGM SPREADSHEET

The input for diaphragms is straightforward. Enter the node to define the location of the diaphragm and then specify a global plane (XZ, XY or YZ) that is to be parallel to the diaphragm.

The “Rigidity Option” field is used to specify the type of rigid diaphragm. The two options are a fully rigid Plane (enter “P”) or a diaphragm that is rigid only for Membrane action (enter “M”).

The Inactive field can be used to turn off the rigid diaphragm without actually deleting the entry. Enter “Y” to inactivate the diaphragm.

To quickly verify that a diaphragm has been assigned to the right node and in the right plane, view the plot of the structure with the diaphragms activated in the Set Plot Options dialog.

INACTIVE DIAPHRAGMS

Making a diaphragm inactive allows you to analyze the structure without the diaphragm, without having to delete the information that defines it. This leaves data intact so the diaphragm may be easily reactivated. This is handy if you want to solve a model with and then without certain diaphragms, without having to actually delete the data.

Putting a “y” in the Inactive field makes the diaphragm inactive, i.e. the diaphragm is not included when the model is solved or plotted.

MEMBRANE DIAPHRAGMS

The rigid membrane diaphragm can be used in situations where the engineer wants to model a rigid floor slab that will carry the in-plane loads, but still allow frame action in the beam and columns (i.e., the beam and column joints in the diaphragm are free to rotate out-of-plane). The lateral stiffness for such a model is simply the combined beam and column frame stiffness, with very little contribution from the diaphragm. (There will be a slight contribution to the lateral stiffness from the diaphragm, since the beams cannot deform axially within the plane of the diaphragm).

The membrane diaphragm is also useful if you want to combine out-of-plane loads (typically vertical dead or live loads) with lateral loads (typically seismic or wind). One limitation of the membrane option is that beams or plate elements are needed to provide any out-of-plane stiffness or “frame” type lateral stiffness. For example, a one-story structure composed of 4 columns, fixed at their bases, and a membrane type diaphragm at their tops, will have the lateral stiffness of 4 cantilevers, since there are no beams to prevent tip rotation of the columns.

The behavior of the rigid plane diaphragm in RISA-3D is defined by observing three “rules”. The first rule for the membrane is that after a model has been solved, all the nodes on the diaphragm will have equal in-plane rotations. The out-of-plane rotations will be based on the stiffness of the attached members and
will most likely not be equal. The second rule is that the absolute distances between all nodes on the diaphragm must be kept the same at all times. We say absolute distances because the horizontal projected in-plane distances can change due to the different out-of-plane rotations. Following the above two rules will cause a third rule to be satisfied as well. The in-plane translation of any node on the diaphragm is the sum of the absolute in-plane translation of the diaphragm itself, plus the in-plane diaphragm rotation times the projected in-plane distance between the node and the diaphragm center of stiffness.

**Plane Diaphragms**

The rigid plane diaphragm can be used to model situations such as shear buildings where the floor system is much stiffer than the columns, and in-plane frame action is not desired or unimportant. This option has the advantage of not needing any floor beams or plate elements to provide out-of-plane stiffness. In fact, any existing floor beams or elements will get absorbed into the stiffness of the rigid plane. A rigid plane diaphragm can be defined solely with just nodes to connect it to the lateral force resisting elements. A limitation of the plane diaphragm is that lateral loads cannot be directly combined with vertical loads if moment frame action is desired to occur within the diaphragm (i.e., since the diaphragm is rigid in all 6 degrees of freedom, the connection joints between columns and beams cannot rotate). If it is desired to model these effects, you must use the membrane option. To model composite action between the beams and the floor you must use a mesh of plate elements to model the diaphragm.

The behavior of the rigid plane diaphragm in RISA-3D is defined by observing two rules. The first rule is that after a model has been solved, all nodes on the diaphragm will have equal rotations. The second rule is that the distances between all nodes on the diaphragm must be kept the same at all times. Following the above two rules will cause a third rule to be satisfied as well. The translation of any node on the diaphragm is the sum of the absolute translation of the diaphragm, plus the diaphragm rotation times the distance between the node and the diaphragm center of stiffness.

**How Diaphragms Work**

Internally both types of diaphragms are implemented by connecting a series of rigid and weightless members between all the nodes on the diaphragm plane. There is no slaving involved so no degrees of freedom are lost.

**Diaphragms vs. Slaving?**

It is not accurate to use nodal slaving to try to create a rigid diaphragm. While the nodal rotations can be slaved and the correct diaphragm behavior maintained, slaving the in-plane translational degrees of freedom will produce incorrect diaphragm behavior. A diaphragm that has a load applied to a location other than the center of stiffness should experience both a translation and a rotation. A
diaphragm that is created by slaving in-plane translations will not rotate under such a loading, it will only translate, and be much stiffer than it should be.

**PARTIAL DIAPHRAGMS**

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. One way this can be done is to offset the elevations of the nodes 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 nodes to be on the same plane as the master node) This works because the rigid diaphragm feature will only rigidly connect nodes that are at the same elevation as the master node. The other nodes, which are on the flexible portion of the floor and are now at a different elevation than the master node, will not be incorporated into the diaphragm.

**DIAPHRAGM STIFFNESS**

You may alter the stiffness of the diaphragm, though this value should almost never be changed. Arbitrarily changing the diaphragm stiffness without understanding the ramifications on the stiffness solution can produce solution results that are inaccurate. Having said all that, the stiffness of the diaphragm may be adjusted from the Diaphragm spreadsheet by clicking the button on the Window Toolbar. The default value is 1.E+7. The exponent of the internal stiffness of the diaphragm is twice this displayed value (1.E+14 in the default case) This value should only be adjusted for 2 reasons.

The first reason is that the lateral force resisting elements in your model are so stiff that they are causing the rigid diaphragm to behave semi-rigidly (i.e. the rotations are not all the same for all nodes on the diaphragm.). In this case you could try increasing the diaphragm stiffness to 1.E+8, however the internal diaphragm stiffness of 1.E+16 starts to encroach on the boundary condition reaction stiffness which is internally modeled as 1.E+20. A diaphragm stiffness value greater than 1.E+8 can produce unpredictable results that are characterized by ghost reactions, meaning some of the nodes in the diaphragm begin to behave as Reaction boundary conditions.

The second reason that you might adjust the diaphragm stiffness is because your dynamics solution will not converge. In this case, you will want to reduce the diaphragm stiffness to 1.E+6 or even lower if necessary. As you lower the stiffness, you will need to watch the nodal rotations for joints on the diaphragm to insure that you getting, or at least approximating, rigid diaphragm action. The nodal rotations should be the same for all joints in a rigid diaphragm.
Distributed Loads

Distributed loads are loads that are spread across all or part of a member and can be of uniform or of varying magnitude (triangular or trapezoidal). Distributed loads may be defined by a pattern on the Distributed Load Patterns spreadsheet and a magnitude on the Distributed Loads spreadsheet.

Note

- If the member being loaded has offsets defined, the offset distances will NOT be loaded. The locations are still relative to the I node, 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 node distance minus the I end and J end offset distances.

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 help on an item, click and then click the item. If the load is not uniform you will need to define the load pattern on the Distributed Load Patterns spreadsheet if you have not already done so.

3. You may choose to apply the load to a single member at a time or to an entire selection of members.

   To apply the load to a few members choose Apply Entry by Clicking Items Individually and click Apply. Click on the members 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 dialog.
- You may also specify or edit point loads in the Distributed Load Spreadsheet.

Distributed Load Spreadsheet

The first column contains the label of the member to assign a distributed load. The pattern label in the second column represents the load pattern described on the Distributed Load Patterns spreadsheet.

The multiplier is recorded in the third column and is applied to the load pattern. Typically the pattern is normalized to a magnitude of 1.0 and the multiplier is used to adjust the load from member to member. For area loads, another option is to give the pattern the area load value and the multiplier for the member could then be the tributary area that the member yielding the proper distributed load for the member.
Distributed Load Patterns

A distributed load may be defined by a load pattern and a multiplier. Typically the distributed loads applied to a structure vary in magnitude but vary little in terms of pattern. For instance, most of the distributed loads on a model are usually full length uniform, with only the magnitude of the uniform load varying. Taking advantage of the similarities in distributed loads on a model can greatly reduce the amount of data that needs to be entered.

A distributed load pattern is defined by entering a label for the pattern, a direction, start and end magnitudes, and start and end locations.

These patterns are independent of the Basic Load Cases; they may be used to define distributed loads for any Basic Load Case. Entering a pattern does NOT automatically cause the load pattern to be applied to the model. The pattern label must be called out on the Distributed Loads spreadsheet for the pattern to be applied.

The pattern label is used to later reference the particular pattern. This label can be anything you wish, so long as it's unique relative to the labels entered for the other patterns.

The next entries on the pattern spreadsheet are the pattern magnitudes. A convenient way to use these patterns, particularly for uniform loads, is to enter magnitudes of "1" or "-1". Then, when defining the distributed loads, the pattern multiplier can be made equal to the load magnitude. These magnitudes do not have to be equal, of course, and they can even have different signs. Any type of trapezoidal load may be described.

Next are the load start and end locations. These are the starting and ending locations for the load, relative to the member's I node. If the start and end locations are both 0., the load will be applied across the full length of the member! If you do wish to define a partial length load, just enter the start and end locations for the load. You may enter a start location other than 0. and leave the end location equal to 0. to have a load applied from any start point to the end of the member.

Since these patterns will likely be applied to members of different length, it may not be convenient to enter specific start and end distances. For this reason, you may define the start and end locations as a percentage of member length. To do this, enter the character "%" followed by the percent value. For example, say you wish the load to be applied to the middle 2 quarters of the members. You would enter a start location of "%25" and an end location of "%75". You can't enter a percentage greater than 100, and entering an end location of "%100" is equivalent to leaving the end location equal to 0.
DISTRIBUTED LOAD DIRECTIONS

The direction code indicates how the load pattern is to be applied. Following are the valid entries:

<table>
<thead>
<tr>
<th>Entry</th>
<th>Load Direction</th>
</tr>
</thead>
<tbody>
<tr>
<td>x, y or z</td>
<td>Applied in the member’s local x, y or z axis</td>
</tr>
<tr>
<td>X, Y or Z</td>
<td>Applied in the global X, Y or Z axis</td>
</tr>
<tr>
<td>T (or t)</td>
<td>Thermal (temperature differential) load</td>
</tr>
<tr>
<td>V</td>
<td>Projected load in the global Y-axis direction</td>
</tr>
<tr>
<td>L</td>
<td>Projected load in the global X-axis direction</td>
</tr>
<tr>
<td>H</td>
<td>Projected load in the global Z-axis direction</td>
</tr>
</tbody>
</table>

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 member’s local axis directions, so their direction relative to the rest of the model changes if the member 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 full length Y direction load of 1 kip/foot applied to a 10 foot member inclined at 45° 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 "V" 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.

**Story Drift**

You may calculate and report inter-story drift information based on the calculated joint displacements. All that is necessary is for you to 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.

**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 is skipped (not defined), then there will be no calculations for both that story and the following story.
- If a story 0 is NOT defined, the base height and displacement values are assumed to be zero (0.).

**Drift Calculation**

To calculate inter-story drift for a particular direction, RISA-3D subtracts the previous story displacement 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 node 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 nodes are used to calculate heights.

Since the drift and height value for each story is dependent upon the previous story, the question arises as to how this is handled for the first story (story 1).
There are two options here. If you wish, you may define a "STORY 0" node. If defined, this story 0 node’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 node 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 node. 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.

**Drift Results**

Access the Story Drift spreadsheet by selecting it from the Results Menu.

This drift report lists the inter-story drift for all defined stories. Drift calculations can be done for the X, Y or Z directions, so stories can be defined for all three directions. Stories are defined on the Boundary Conditions spreadsheet. Enter a boundary condition of "Story nn" to designate the particular joint as representing story "nn" for the particular direction. If you wish, you may define a "Story 0" to set the base value for story height calculations. If Story 0 is not defined, the base height is assumed to be 0.

For stories defined for the X and Z directions, the node Y coordinate value is used to calculate story heights. If you define stories for the Y direction, the node Z coordinate values are used to define story heights. For example, suppose for X direction drift, node 1 is designated as Story 0, node 5 is Story 1 and node 7 is Story 2. This means you would enter "STORY 0" for the X translation boundary code for node 1, "STORY 1" for node 5 and "STORY 2" for node 7. Let’s say the Y coordinate values for nodes 1, 5 and 7 are 10., 22. and 36. The story height for story 1 will be (22. -10.), or 12. The story height for story 2 will be 14. If we did not define node 1 as Story 0, the story height for story 1 would be 22. The actual drift value is the difference in displacements for the nodes defining the current and previous stories. Again, for story 1, the previous story displacement will be taken as 0. if a "Story 0" is not defined.

Continuing the example, say the X displacements for nodes 1, 5 and 7 are 1",3" and 4". The drift for story 1 will be 2", and for story 2 will be 1". If Story 0 isn’t defined, the drift for story 1 would be 3". The "% of Ht" is simply the ratio of drift to story height, expressed as a percentage. This usually should be kept less than 5%. The example was for X direction drift. We could use the same three nodes, or three different nodes, to calculate Z direction drift, if desired. Just enter the story designations as the Z translation boundary codes.
DXF Files

You may read and write 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 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
- It’s always a good idea to do a Model Merge on any model created from a DXF file.

Reading DXF Files

You may translate POINT’s, LINE elements and 3DFACE’s. POINT’s are converted into nodes, 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.

Translate Layer Names to Section Database Shapes
This is a Yes/No choice. Choosing “Yes” will treat each entity layer name as a database shape label. Section sets will be created for each unique layer name. If a layer name is not a valid database shape label, a section set will still be created, but the database shape label field will be left blank. All LINES/MEMBERS on same layer in the DXF file will then be assigned to the same section set in the model, and will also be assigned the same database shape.

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.
Writing DXF Files

Only the node, member, 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:

Node Layer
Type the name of the layer for the node 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”.

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 nodes 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 nodes at each member endpoint which will separated by the standoff distance)

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.
Translate Section Set Database Shapes to Layer Names

This is a Yes/No choice. If you choose “Yes”, layers will be created in the DXF file corresponding to the section set database shape labels in the RISA model. Members will be assigned to layers based on the their section set database shape. If a section set does not have a database shape label, the section set label will be used instead. A “Yes” choice here overrides any layer name entered for the member layer.

For example, let’s say you have designed a structure with W14X132 columns, W24X68 primary members, W14X26 secondary members and TU4X4X5 miscellaneous tube framing. If you type in a layer name such as "STEEL" then all members, regardless of size, will appear on a layer named "STEEL". However, if you choose “Yes” for this option, all W14X132’s will appear on a layer named "W14X132", W24X68’s on a layer named “W24X68”, etc.

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 member such as a C10X15.3 in your model will translate that decimal point into an underscore character resulting in a layer name of "C10X15_3". This translation is necessary because AutoCAD does not allow periods to be included as a part of a layer name. 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 “-”.

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 (UCS).

The default vertical axis in RISA is usually the positive Y-axis and may be specified on the Global dialog. Once you’ve read in a DXF file, you can rotate your geometry to match the vertical axis.

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 nodes 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 doing a model merge. This feature will also deal with intermediate nodes along member spans, a common problem in models created from DXF drawings and members that cross, but do not have nodes at their intersection point.

**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 nodes, 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 its 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 starts with an integer group code. RISA recognizes the following group codes:

<table>
<thead>
<tr>
<th>Group Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>Identifies the following overall keywords: SECTION, ENDSEC, and EOF. Within the ENTITIES section it also identifies POINT, LINE, and 3DFACE.</td>
</tr>
<tr>
<td>2</td>
<td>Identifies a section name (i.e., ENTITIES)</td>
</tr>
<tr>
<td>8</td>
<td>Identifies a layer name.</td>
</tr>
<tr>
<td>10, 11, 12, 13</td>
<td>Identifies the X coordinate of the 1st, 2nd, 3rd and 4th points of an item.</td>
</tr>
<tr>
<td>20, 21, 22, 23</td>
<td>Identifies the Y coordinate of the 1st, 2nd, 3rd and 4th points of an item.</td>
</tr>
<tr>
<td>30, 31, 32, 33</td>
<td>Identifies the Z coordinate of the 1st, 2nd, 3rd and 4th points of an item.</td>
</tr>
</tbody>
</table>

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 (Modal) Analysis**

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. See the Response Spectra Analysis section for more information.

You may calculate up to 300 modes for a model. The process used to calculate the modes is called an eigensolution. The frequencies and mode shapes are also referred to as eigenvalues and eigenvectors.

**To Perform a Dynamic Analysis**

1. You may wish to solve a static analysis first to verify that there are no instabilities.
2. Select **Dynamics** from the **Solve** menu.
3. Specify the number of modes to solve for and the load combination to use as the mass.

**Note**

- You may view the mode shapes graphically by choosing this option in the Set Plot Options dialog.

**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 that in order to know how much mass is participating you first have to do a dynamic analysis! So this becomes a trial and error process. First pick an arbitrary number of modes (5 to 10 is usually a good starting point) then do 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.

**Dynamic Mass**

The eigensolution is based on the stiffness characteristics of your model and also on the mass distribution in your model. In order to calculate the amount and location of the mass contained in your model, RISA takes the loads contained in the load combination you specify and converts them to mass using the gravity factor. The masses are lumped at the nodes and are applied in all three global directions (X, Y and Z translation). There must be mass assigned to be able to perform the dynamic analysis.

**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* window. Unless you specify otherwise, the Y-axis is considered to be the vertical axis.
- 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.

**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 will have to model any accidental torsion, which may be done by taking advantage of the rigid diaphragm feature.

If you have modeled the dynamic mass at the center of mass only, then you may simply move the nodes that specify the center of mass. For example, if the required accidental eccentricity is 5% of the building dimension then move the nodes that distance, perpendicular to the applied load. 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 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.
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 supporting column shears and brace forces below the floor and the sum of the supporting column shears and brace forces 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 not be conservative 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 node 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.

**EIGENSOLUTION AND CONVERGENCE**

The eigensolution procedure for dynamic analysis is iterative, i.e. a guess is made at the answer and keeps improving its guess until the guess from one iteration closely matches the guess from the previous iteration. The tolerance value specifies 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).

**Note**

- Keep in mind that these mode shapes do not, in and of themselves, represent model deflections. They only represent how the nodes 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.
SAVE THE DYNAMIC SOLUTION

After you’ve done the dynamic solution, you can save that solution to file to be recalled and used later. This option is called Save w/Dynamic Restart and is located on the File menu.

**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 N modes, no extra modes, to make the solution as fast as possible. 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. Selecting Converge Work Vectors calls for N modes plus a few extra modes, so there is a better chance of getting all N of the lowest modes.

If it is detected that frequencies have been missed once N modes have been extracted RISA will automatically reset to converge work vectors and continue with the eigensolution. This saves a lot a time versus stopping the solution, having you the user reset the flag, and starting over from the beginning.

If, when the solution is finished, you see Converge Work Vectors has been selected, leave it selected for any subsequent solutions for the particular model.

Why not converge work vectors for all models? Because if the model does not require the work vectors be solved to get the lowest N modes, the solution will be slower than it could be. You are allowed control this setting because there is a slight penalty in time if RISA has to reset the flag itself. If you know before starting the solution that this particular model requires converging the work vectors, the solution will be faster if you select it from the beginning, versus letting it reset automatically. If in doubt, however, do not choose to converge the work vectors.

DYNAMICS MODELING

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 always 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, you must 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 nodal loads to your free nodes. Or you may want to add more free nodes 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 less modes than actually exist. In this case it may be helpful to include the selfweight 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 nodes 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 from the ground reactions to the elevation of the floor. This will give you at least 3 free nodes between the ground reaction and the floor.

When you perform a dynamic analysis of beam structures, such that you are trying the capture the flexural vibrations, (IE, the beams are vibrating vertically or in the transverse direction), you must make sure that you have at least 3 free nodes 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 nodal loads to the free nodes.
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. This is most easily accomplished using the ALL code on the Boundary Conditions spreadsheet. For example, if the global Z direction is out-of-plane, you would enter “ALL” for a node number and enter “R” for the Z-translation.

**Modal Frequency Results**

Access the Modal Frequency spreadsheet by selecting it from the Results Menu. 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.

**Mode Shape Results**

Access the Modal Shape spreadsheet by selecting it from the Results Menu. 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.

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.

**Note**

- You can plot and animate the mode shape of the model by using the Plot Options dialog.

**Dynamics Troubleshooting – Localized 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 very weak in the out-of-plane
direction and thus is vibrating out of plane. Localized modes are not immediately obvious from looking at the frequency or numeric mode shape results, but they can usually 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.

If you have localized modes in your model, always try a Model Merge before you do anything else.

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 nodal 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.

Quite often, localized modes are due to modeling errors (erroneous boundary conditions, members not attached to plates correctly, etc.).

### 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 Dialog 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.
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 is entered as the axis of rotation (X, Y or Z) and the arc will be in the plane normal to the axis of rotation. 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.

There are entries for a section set for the arc members. You may also have unique labels assigned to the generated members, by entering a start label.

GRID MEMBERS 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 a starting point X, Y and Z coordinates 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 1D or 2D grid. Negative increments are OK. Joints are created at all the grid intersection points.

If you have one or more series of equal increments it is not necessary to enter each increment value individually. Instead, use the "@" symbol to denote a series
of equal increments. For example, the entry "3@12" means 3 equal increments of 12 units each.

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.

**Cylinder Generation**

The cylinder generation is used to make cylinders 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 cylinder by specifying the start and sweep angles.

In the figure Y is the vertical, X is the axis of rotation.

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 be 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 members parallel to the axis of rotation so these two sets of members can be different sizes. 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-node better serves this purpose.

Finally, you may enter a material set and thickness and label for plates to be defined.
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.

In the figure Y is the vertical, X is the axis of rotation.

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 be 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 members parallel to the axis of rotation so these two sets of members can be different sizes. 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-node better serves this purpose.

Finally, you may enter a material set and thickness and label for plates to be defined.

CIRCULAR GRID GENERATION

The circular grid generation enables you to generate a full or partial circular grid comprised 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. 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.

**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.

If you have one or more series of equal increments it is not necessary to enter each increment value individually. Instead, use the "@" symbol to denote a series of equal increments. For example, the entry "3@12" means 3 equal increments of 12 units each.

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 desired can be entered. The Fluid Load BLC provides a drop down list of all the Basic Load Cases. If you’ve filled 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.

**Plate Disk Generation**

The disk generation enables you to generate a full 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 length 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 may enter a material set and thickness to have plates defined. Only quadrilateral plates are generated. You may also specify labels assigned to the plates.

**WARREN TRUSS GENERATION**

The Warren truss generation is used to make box trusses and various forms of Truss Joists. The truss model will be comprised of joints and optional top chord, bottom chord, vertical strut member, and diagonal brace member.

In the figure Y is the vertical, X is the truss axis, and the truss plane is the XY plane.

Specify the start point for the truss generation. This defines the first point of the bottom chord.

You must enter the truss height and the panel width. Note that the panel width is 2 bays wide so that a diagonal brace in each direction can be created.

Enter the number of panels to be generated. The number of panels times the panel width will define the total length of the truss.
The truss axis defines the global axis that will be parallel to the chords. The truss plane defines the plane in which the truss will generated. For example, if you have the XY plane as your generation plane, and the Y axis as your truss axis, the truss chords will run along the Y axis and the height of the truss will toward the positive X axis.

You may optionally enter a Joint label prefix which will be used for all the joints generated as part of the truss. This is useful if you want a way to track individual parts of your model by looking at 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 may assign an independent section set, "x-axis rotate angle", and member label prefix. Note that you must mark the checkbox for a member type before you can set any of the values and also 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-node better serves this purpose in some instances.

**X Brace Truss Generation**

The X Brace truss generation is used to make box trusses. The truss model will be comprised of joints and optional top chord, bottom chord, and diagonal brace members.

![X Brace Truss Diagram](image)

In the figure Y is the vertical, X is the truss axis, and the truss plane is the XY plane.

Specify the start point for the truss generation. This defines the first point of the bottom chord.

You must enter the truss height and the panel width.

Enter the number of panels to be generated. The number of panels times the panel width will define the total length of the truss.

The truss axis defines the global axis that will be parallel to the chords. The truss plane defines the plane in which the truss will be generated. For example, if you have the XY plane as your generation plane, and the Y axis as your truss axis, the truss chords will run along the Y axis and the height of the truss will toward the positive X axis.
You may optionally enter a Joint label prefix that will be used for all the joints generated as part of the truss. This is useful if you want a way to track individual parts of your model by looking at joint prefixes.

You may optionally create members for the bottom chord, top chord, and diagonal braces. For each one of these members you may assign an independent section set, “x-axis rotate angle”, and member label prefix. Note that you must mark the checkbox for a member type before you can set any of the values and also 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-node better serves this purpose in some instances.

**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.

![Parabolic Arc Generation Diagram](image)

In the figure Y is the vertical, Z is the rotation axis.

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 which 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.

You may optionally create members for the arc. For these members you may assign a section set, “x-axis rotate angle”, and member label prefix. Note that you must choose a valid section set for member generation to be performed.

The “x-axis rotate” may be used to rotate the local axes to a desired orientation, however you may find that a K-node better serves this purpose in some instances.

**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.
In the figure Y is the vertical, and the beam axis is along the global X-axis.

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 also generating loads, you will also want to select a Basic Load Case for any Uniform Distribute 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-node 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 truss. 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 very first joint (the Start Joint), all the middle joints as a group, and the last joint (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 means that no translations or rotations are restrained. The is useful for the first or last node if you want a cantilever overhang. The “Roller” option will only restrain the vertical translation. (Note that this will be controlled by the Vertical Axis selected on the Global screen.) The “Pinned” option will restrain all the translations. The “Fixed” option restrains all the translations and rotations.

To generate the continuous beam geometry, you will want to assign increments (similar to the other generators like the Member Grid generator). The increments are in the current length units and for multiples of the same number you can use the “@” symbol. E.g., if you had 5 spans of 10’, you could type 5@10 in the first increment row, assuming that feet was the current length unit. If you have different length increments, then you would use multiple rows. If you have more
than 5 different increments, you will to enter the first 5 increments and generate
the first part of your beam, then bring up the Continuous beam generator again
and generate the remaining increments. (Remember to move your Start Point to be
the last node generated!)

With each increment, you can also generate distributed loads by selected a valid
distributed load pattern, 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. These unbraced lengths are the
same as those used on the Design Parameters screen.

**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 as
desired. The tank will be assumed to be pinned and 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.

You must enter a positive value for the tank height, width, and length. These are
in the current length units for the program.

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 optionally choose to have rigid links generated that will model the
composite action of the stiffener and the tank wall due to the center of the shape
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. This option
may only be used for shapes that are defined using the Shape Editor. 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. Note that these shapes can still be used as stiffeners; you just can’t have the composite offset automatically calculated and modeled. 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.

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. 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 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. The default is "None", so you will need to select a section from the list of currently defined section labels to generate your stiffeners.

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 due to fluid in the tank as you generate it. The hydrostatic load will be generated using a series of uniform surface loads on your plate elements. The fluid depth is constant along the floor so you will just get uniform value surface loads on your floor plates. The fluid depth will vary along the walls of the tank and these will be modeled as 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 and outward.

The fluid depth is measured in the positive vertical direction from the bottom of the tank.

The default fluid density is for water, but any density desired can be entered. The Fluid Load BLC provides a drop down list of all the Basic Load Cases. If you’ve filled 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.

**Global Information**

The Global dialog, accessed through the **Options** Menu, is used to define information that influences the model and its solution in an overall, or global,
manner. You may save any of the information as the default 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.

The first fields on this window are used to enter descriptive information such as a title for the particular model being defined, the name of the company and the designer. The title may then be printed at the top of each sheet of the output, and on the graphic plot of the model.

Next is the Job Number. This also is simply a description for the model being defined. This field allows up to 15 characters. The Job Number may be also printed on each page of the output.

The next options control steel code checking. The **Steel Design Code** indicates which steel code is to be used in the design of steel database shapes. Currently, the choices are the AISC 9th Edition ASD specifications, the AISC 2nd Edition LRFD specifications, and the Canadian code CAN/CSA-S16.1-94. If you do not want any code checks to be performed you may specify “none”.

The **Steel ASIF** field is used to set the allowable stress increase factor used in the ASD steel code checking. This factor is used for load combinations that are marked for allowable stress increases.

If you want suggested alternate steel shapes after performing the steel code checking, check the **Do Redesign** box. You control the way shapes are selected by setting parameters on the Redesign Spreadsheet.

The **Number of Sections** controls how many places you receive reported member force, stress, torsion, and deflection results. Internally, the program subdivides the member into more sections (usually around 40 total sections) and this is what the member steel and wood code checks are based on. The member force diagrams displayed in the model view and the detail plot are also drawn using the larger internal number of sections.

Check the **Include Shear Deformation** box if shear deformation considerations are to be included in the model solution. This should almost always be checked.

The **Include Warping** box is an option as to whether you wish to consider torsional warping effects when calculating stiffness and stress values for shape types that warp. The only reason you would NOT want to include warping is if you are comparing results with hand calculations or calculations done by another program that does not properly consider warping.

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 $\frac{1}{2}$ of 1 percent (.5%).

The **Vertical Axis** is the default vertical axis. The **Vertical Axis** is important for the dynamic analysis because only loads in the vertical direction may be converted into dynamic mass.
RISA Technologies has put much 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.

**On-line Help File System**

We designed the Help System to help you get your work done as quickly as possible. The help system 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
- Troubleshooters 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

Different types of help are displayed in different types of windows. For instance, task-based procedures are typically in windows that will stay on top of the application window for easier viewing. You can specify that other windows stay on top through the **Options** menu.

**On-Line 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. Context-help is provided in two different ways:

The first method provides specific information on every item that you see in on the screen. For help on an item such as an entry field in a dialog, click **What's This?** and then click the item. This feature is referred to as **“What’s This?”**.

You may also get more detailed help when working in some windows by clicking on the **Help** button at the bottom of the window. This will present 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 On-line**

Our web site address is www.risatech.com and provides many support features that can be useful in verifying our results or working through common problems.

- When a bug is reported it is posted on the web site along with possible work-around procedures or beta version corrections.
- Verification problems are placed on the web site for you
- Answers to frequently asked questions are recorded, providing around-the-clock support.
- Parametric studies on topics such as plate modeling.
TOOLTIPS
Are you uncertain what a toolbar button is for? Simply hold your 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 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 tutorial takes 3 hours to complete and is certainly worth the time and effort.

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.

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.

Note
- You may also solve load combinations by selecting them from the Load Combinations spreadsheet if it is open. To solve a particular load combination, move the cursor to that combination and press on the toolbar. The cursor itself can be anywhere in the line of the combination.

Load Combinations Spreadsheet
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 five fields are for options that you may apply to the load combination. These options are explained further by choosing from the list below.

The next eight sets of fields (BLC, Factor) are for defining what loads are to be part of the combination, along with factors for each. Following are the entries you can use in the BLC field:
Enter in the factor field a multiplier to be applied to the loads being included.

**To Envelope the Results of Multiple Load Combinations**

1. On the Load Combinations spreadsheet, check the box in the Env column for the combinations that you wish to include.
2. Select Envelope from the Solve menu.

**Note**
- The envelope solution is where all combinations with a checkmark in the Env fields 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.

**Load Combinations with RSA Results**

The results from response spectra analyses in the X, Y and/or 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 can retain up to three RSA solutions (one for each direction) simultaneously.

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. So, 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. If you put an "Sn" entry for a direction in which an RSA has not been done, you'll get an error.

**Note**
- 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) 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.
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.

Note
- You can only have one moving load in each load combination. This restriction on moving loads also applies to combinations of load combinations.

NESTED 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.

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 5 flags (on the left side of the spreadsheet) for Load Combination 1 will be ignored.

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

Checking the field labeled W/S indicates that the load combination should be treated as a wind/seismic load combination. Click on the cell to check or clear the field.

If AISC 9th Edition ASD code checking is selected, the allowable stress increase factor is applied to this load combination. Remember,

If LRFD 2nd Edition Code checking is selected, this “W/S” flag indicates whether the seismic provisions for the WF compactness check are to be used (Table 8-1, p. 6-317 of the 2nd ed. LRFD).

Note
- The code selection and the increase factor is specified on the Global dialog.
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.

You may also perform a compression only P-delta analysis. Invoke this option by putting a C in the P Delta field.

Note
- P-Delta analysis is required for LRFD based code checks.
- P-Delta analysis is not performed on plates.

SRSS Combination of Orthogonal RSA Results

This is used to cause all the 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, which is specified on the Response Spectra window.

Load Duration for Timber Design

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 '91 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 section set and material properties are defined on the NDS Based Wood Properties spreadsheet.

Note
- See Table 2.3.2 in the NDS-1991 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.

Setting the BLC Entry

Select a load to combine and then use the Factor column on the spreadsheet to apply a multiplier to the load. Choose from load categories, basic load cases, or moving loads from the drop-down lists. To combine response spectra analysis results you must first run a response spectra analysis. You may also combine
response spectra results for different directions using a square root sum of the 
squares approach by checking the **SRSS** column in the spreadsheet.

**STANDARD 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 selected from a drop down list on the **Window** toolbar once the **Load 
Combinations** Spreadsheet is open and active. If the added load combinations 
are not exactly what you want, you may modify the load combinations after 
adding them.

The codes that are included are recent publications of UBC, BOCA and SBC. 
Allowable Stress Design combinations and Strength Design combinations are 
provided.

All of the design codes have a specification that the most critical effect can occur 
when one or more of the contributing loads are not acting. This really expands 
the list of possible load combinations. An **Expanded** option is provided for all 
combinations that will allow the user to decide if he wants the straight 
combinations as written or all of the possibilities.

The combinations may be expanded in one of three ways. The first is the 
situation mentioned above (inactive loads), the second is sign reversal with wind 
or seismic loads and the third is with special factors that are specific to special 
load categories such as garage loads or roofs that do not shed snow. The 
expanded version will consider all three affects. The unexpanded version 
provides the sign reversals and special load categories but does not consider loads 
not acting.

The following loads are not automatically included in the standard combinations 
but may be added by editing the combinations in the spreadsheet:

**SX, SY, SZ - Response Spectra Results**

**TL** – Long Term Load category

**HL** – Hydrostatic Load category

**FL** – Fluid Pressure Load category

**PL** – Ponding Load category

**EPL** – Earth Pressure Load category

**OL#** - Other Load categories

**ELX, ELY, ELZ** – Directional Earthquake Load categories

**WLX, WLY, WLZ** – Directional Wind 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 A.S.I.F. settings for combinations after you add them.
• You may specify P-Delta options and SRSS combinations for each combination after you have added them.

To Add Standard Load Combinations
1 From the Spreadsheets menu select Load Combinations.
2 Select the standard combinations from the list on the Window toolbar and then click Add.
3 Modify the combinations and options as necessary.

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.
• 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 A.S.I.F. settings for combinations after you add them.
• You may specify P-Delta options and SRSS combinations for each combination after you have added them.

Material Properties

Material properties are defined on the Material spreadsheet and then are referred to as you build sections. 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. All materials except wood database shapes should be defined on this spreadsheet. The entries are explained below.

The material values are for A36 steel are built in and are listed as 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 materials as the default materials by clicking on the button.

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.

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 for you. The equation for “G” is:

\[ G = \frac{E}{(2.0*(1.0+\text{Poisson's Ratio}))} \]
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 (not an error) just to make sure you are aware of that fact.

**Nu** is Poisson’s ratio. Besides being used for the “G” calculation, is also used for shear deformation calculations. The value of Poisson’s ratio may not exceed 0.5.

**Therm**, the coefficient of thermal expansion, is entered per 10^5 (100,000) degrees and is used in the calculation of thermal loads.

**Dens.** 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.

**Fy.** the yield stress, is used only for AISC steel design.

---

### Material Take-Off Results

Access the **Material Take-Off** spreadsheet by selecting it from the **Results** menu.

This spreadsheet shows material takeoff information for each section set. The length listed is the sum of the lengths of all the members assigned this section set. Any offsets defined are deducted from the total, so this length sum is the actual total, including offset considerations. The weight is the sum of the self-weight for all the members assigned to section set. This is calculated as Area * Density * Length for each member, with offset distances deducted. This material takeoff report is independent of the loads applied to the model, i.e. the applied loads do not influence this report. The only values that influence this report are member areas, lengths and weight densities. The database shape (if any) currently assigned to the section set is also listed.

**Note**
- Plates are not included in the Material Take-Off.

### Members

RISA uses a general-purpose beam element. With the use of the member releases this element may be easily converted into a truss element. Member definition data is recorded on the **Members** spreadsheet. The properties of the element are defined by the **Section Set** assigned to the member. Design parameters for steel or timber design are recorded on the **Member Design Parameters** spreadsheet.

### Location of Calculations along Members

The **Number of Sections** that is input in the **Global Parameters** window controls how many places you receive reported member force, stress, torsion, and
deflection results. Internally, the program subdivides the member into more sections (usually around 40 total sections) and this is what the member steel and wood code checks are based on. (Notice that the location of the maximum steel and wood check is always a distance from the I-node in the current length units, not necessarily at a section location).

Note
- The member force diagrams for the overall model view and the plots in the detail report are also drawn using the larger internal number of sections.

**SETTING MEMBER RELEASES**

The **I Release** and **J Releases** are used to designate whether the forces and moments at the ends of the member are considered fixed to or released from the member’s points of attachment (the I and J nodes). Each member has 6 force components at each end (in order, these are axial, y-y & z-z shear, torque, y-y & z-z bending). Any or all of these force components can be released from the member’s point of attachment. If a force component is released, that force is not transferred between the node and the member.

Note
- RISA-3D will not allow you to release the member torsion at both ends. This is because it will be unstable locally, i.e. it will be free to spin about its centerline. For this reason, pinned end conditions should be modeled using the "BenPIN" entry instead of "AllPIN".

**DRAWING FEATURES**

There are several graphic-editing features that make the creation and modification of models quite easy. Use the Drawing Toolbar to access 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 nodes using the Draw New Members and Draw New Plates feature.

To modify multiple members or plates you first graphically select those items that are to be modified and then use the Modify Selected Members and Modify Selected Plates features to actually change their parameters. Alternatively you may double-click any one item to view and edit detailed information about that one item. Other features allow you to move or rotate, copy, rotate and delete existing items.

Once you have created these items you may use other graphic features to load the model and set boundary conditions.

**To Draw Members**

1 If you are not drawing between existing nodes, you will need to create a drawing grid or define nodes on the **Node Coordinates** spreadsheet.

2 Click the **button and set the member properties. For help on an item, click ** and then click the item.
3. Click **OK** to start drawing members by clicking on nodes or grids with the left mouse button. You will notice that the coordinates of the closest node or grid point to your cursor are displayed in the lower right hand corner.

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 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.

4. To stop drawing altogether click the **button**, right click or press the Esc key.

**Note**

- 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.

- Press CTRL-D to quickly recall any of the dialogs and make changes.

- You may also specify or edit members in the Members Spreadsheet.

- You may choose the prefix that is used to label the members.

- 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**.

- To modify members click the **Modify Members button and set the parameters for the new members. Check the Use? Box for the items to apply. For help on an item, click ** and then click the item.

- To modify member design Parameters click the Modify Parameters ** button and set the parameters for the new members. Check the Use? Box for the items to apply. For help on an item, click ** and then click the item.

- To split members click the **Split Members ** button and set the parameters for the new members. For help on an item, click ** and then click the item.

- To relabel members first sort the Members spreadsheet into the desired order then select the **Utility menu and choose **Relabel Members**.

**Members Spreadsheet**

The Members spreadsheet is used to record the beam/truss members of the model. This spreadsheet can be accessed by selecting Members on the Spreadsheets menu. The following are input columns on the spreadsheet which may be used to specify the member label, geometry, orientation, and end conditions:

**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, which will not change even as you add and subtract members. The label can be anything you like so long as it doesn’t start with a number. For example, “BR6” would be a valid label, but “6BR” would not. Each label has to be unique, so if you try to enter the same label more than once you will get an error message.
Member End Nodes
The I-Node and J-Node entries define the start (I-node) and end (J-node) locations of the member.

Note
- The member local axes are defined based on these nodes.

Inactive Members
Making a member inactive allows you to analyze the structure without the member, without having to delete the information that defines the member. This leaves member data intact so the member may be easily reactivated. This is handy if you want to try a frame with and then without certain members, without having to actually delete the member data.

Putting a “y” in the Inactive field makes the member inactive, i.e. the member is not included when the model is solved or plotted.

Another option is to put an “E”. The “E” code means include the member in the solution, but exclude it from the results list. So, 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 you’re not interested in. You don’t have to clutter up the results list with these members and can concentrate on the members you’re most interested in.

Note
- When making members inactive you may need to update the unbraced lengths of the adjacent members.

Member Local Axes
The following diagram illustrates the directions of the member’s local axes that are used to define member forces, member stresses, and member deflections:
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 node towards the J node. The complicated part is defining the orientation for the local y and z-axes. Of course, we only have to define the direction for one of these two (y and z) axes. The third axis direction 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.

**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. 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 defining a **K node** for the member on the **Member** spreadsheet. If a K node is defined the three nodes (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:
**Member Offsets**

Member offsets reflect the fact that the member ends may not be attached at the centerline of the member being attached to. For example, a beam connected to the flange of a column is offset from the centerline of the column. The distance of the offset would be \( \frac{1}{2} \) 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 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 window less 1.
**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 nodes). Each member has 6 force components at each end (in order, these are axial, y-y & z-z shear, torque, y-y & z-z bending). Any or all of these force components can be released from the member's point of attachment. If a force component is released, that force is not transferred between the node and the member.

To specify member releases go to the **I Release** or **J Release** field for the member and click the button. This will open a dialog for you to 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 three (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 "b", the keyword "BenPIN" will be filled in automatically.

**Note**

- **RISA-3D** will not allow you to release the member torsionally at both ends. This is because it will be unstable locally, i.e. it will be free to spin about its centerline. For this reason, pinned end conditions should be modeled using the "BenPIN" entry instead of "AllPIN".

**MEMBER SHEAR DEFORMATIONS**

Including shear deformation in the member formulation 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** dialog.

Shear deformation effects are based on the material shear modulus (G) and the shear area (S.A.).

Including shear deformation causes the member stiffness matrix to be modified by the term \( \Phi \), where

\[
\Phi = 12 * E * I / (G * S.A. * L^2)
\]

The effects of shear deformation are more pronounced for members that are short and deep. 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 \( \Phi \) is the node-to-node member length.
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.

**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.

**Inactive 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.

**Note**
- When making members inactive you may need to update the unbraced lengths of the adjacent members.

**Member Results**

When the model is solved, there are several groups of results specifically for the members. Note that member results (forces, stresses, code checks), are only reported at the section locations. For example, if you set the **Number Of Sections** on the **Global Parameters** window 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 very large 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**
- On the **Global** window adjust the **Number of Sections**

**Note**
- Adjusting the number of sections affects the amount of output.
**Member Force Results**

Access the **Member Section Forces** spreadsheet by selecting the **Results** menu and then selecting **Members ▸ Forces**.

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 on the **Global** window. The number of member segments is this **Number Of Sections** minus 1. The incremental length of each segment is the same. For example, if you specify 5 sections, the member is divided into 4 equal pieces, and the forces are reported for each piece.

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:

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, compressive is positive. For moments, counterclockwise 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:
Since the left side moment is being used, a member under negative moment would have the "holds water" deflected shape, which is contrary to some beam conventions. RISA-3D uses the right hand rule joint convention and is always consistent with this convention.

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. To include a particular Load Combination in the envelope analysis, open the Load Combinations spreadsheet and check the box in the “Env” column.

**Note**
- To view the results for a particular member, use the Find option. To view the maximums and minimums, use the Sort option.
- These results may also be displayed graphically with the Plot Options dialog.

**MEMBER STRESS RESULTS**

Access the Member Section Stresses spreadsheet by selecting the Results menu and then selecting Members ➔ Stresses.

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 on the Global window. The actual number of segments is this Number Of Sections minus 1. The incremental length of each segment is the same. For example, if you specify 5 sections, the member is divided into 4 equal pieces, and the stresses are reported for each piece.

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 \( V/S.A. \), where S.A. is the effective shear area.

The bending stresses are calculated using the familiar equation \( M \times c / I \), 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 z axes. 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. To include a particular Load Combination in the envelope analysis, open the Load Combinations spreadsheet and check the box in the “Env” 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.

- To view the results for a particular member, use the Find option. To view the maximums and minimums, use the Sort option.

- These results may also be displayed graphically by using the Plot Options dialog.
**Single Angle Stresses**

The bending stress calculations for single angle shapes are a special case of member stress. Unlike the other shape types, single angles do not bend about the local geometric axes assigned them in RISA-3D. This diagram illustrates this point:

![Diagram illustrating single angle stresses]

The orientation of the shape is defined using the local y and z axes shown in the above diagram, but 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-3D calculates the angle $\alpha$ and transposes the moments thusly:

- $M_{z'} = M_z \cdot \cos \alpha + M_y \cdot \sin \alpha$
- $M_{y'} = -M_z \cdot \sin \alpha + M_y \cdot \cos \alpha$

The $M_{y'}$ and $M_{z'}$ moments are the moments shown as $M_y$ and $M_z$ 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 $M_z'$ bending moment). The z-top, z-bot stresses are for $M_{y'}$ bending at the extreme fiber locations along the z’ axis.

**Note**
- If you wish to override this and have the stresses for a single angle calculated for the geometric axes (not the principal axes as shown above), enter a value of 0 for the $L_{comp}$ entry on the AISC Parameters spreadsheet for the particular member.

**Member Deflection Results**

Access the Member Section Deflections spreadsheet by selecting the Results menu and then selecting Members ▸ Deflections.

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 window. The actual number of segments is this Number Of Sections minus 1. The incremental length of each segment is the same. For example, if you specify 5 sections, the member is divided into 4 equal pieces, and the deflections are calculated for each piece.

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 $L/n$ 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 \( L/n \) ratios reflect the magnitude of the \( y \) and \( z \) deflections relative to the member node to node length (minus offsets). Expressed as an equation, deflection = \( L/n \), or \( n = L/\text{deflection} \), where \( n \) is what is tabulated in the spreadsheet. The smaller the deflection, the larger \( n \) is. In other words, the value tabulated here is the member length divided by the deflection (\( n = L/d \)). If 'NC' is listed, that means the \( n \) value is greater than 10000. (For a deflection of 0., \( n \) would be infinity). Also, the I-end deflection is used as the base value for the rest of the deflections across the member, so the \( L/n \) value at the I-end is always shown as 'NC'. The minimum \( n \) value that will be shown is 1., so if the actual \( n \) value is less than 1, 1 will be listed here.

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. To include a particular Load Combination in the envelope analysis, open the Load Combinations spreadsheet and check the box in the “Env” column.

**Note**
- To view the results for a particular member, use the Find option. To view the maximums and minimums, use the Sort option.
- 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.

## Model Merge

Model Merge scans through your model and automatically corrects common modeling problems such as joints along member spans that aren’t actually connected to the member, or members that are crossing but not connected to each other.

You can take advantage of this in modeling your structure. You can generate pieces of your model separately and merge the separate pieces together. You can tie newly generated model segments into the currently defined model. You can define members spanning across several nodes or other members and then merge in order to break the members up into node to node pieces. You can submesh plates alongside beam members and then use merge to split the members at the new nodes. The list goes on so be aware of the merge capabilities and use them to your advantage.

### To Perform a Model Merge

1. If there is not a model view already open then click \( \text{[Open View]} \) on the RISA Toolbar to open a new view and click \( \text{[Turn On Drawing Toolbar]} \) 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.

**Model Merge Options**

There are three main options for the model merge.

The **Merge Tolerance** is used as the maximum distance 2 nodes can be apart and still be merged together. It is also used when scanning for crossing members and for unattached nodes 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. This can slow down the merge process significantly for models with many members. If you have cross bracing you may or may not want them to be merged.

Merging only a selected portion of the model is also an option. When checked, limits the merge function to only the parts of your model that are currently selected. This allows you to prevent the program from merging portions of your model where you may have intentionally put nodes at the same location or have two members next to each other.

**Model Merge Examples**

Looking at this frame, consider the column line on the right side, members 1-7 and 7-13. Typically 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. For this example with only two members that isn't a big deal, but imagine if the column line were instead made up of 20 pieces!

Another convenient use of the model merge function is laying out floor plans and being able to draw all the joists right over the main girders. The model merge will take of breaking up the joists and girders up at all the intersecting points.

**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. Also members with point loads, partial length distributed loads, and trapezoidal or triangular loads will not be broken up by the model merge function. Even if such members have
intermediate unattached nodes, or crossing members within their spans, they will not be broken up. These limitations will be addressed in future program releases.

**Note**
- There is an order of precedence for the section sets based on the order of the section sets on the Sections spreadsheet, i.e., the first set of properties takes precedence over all subsequent section sets. The second set takes precedence over all section sets except the first section set, etc. The merging of duplicate members can result in member properties being changed for merged members depending on which member is kept and which member is removed.

## 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 node 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 and 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 (the column lines are actually right on top of each other).

For step 1, elimination of duplicate nodes, we go 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 via 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 nodes of overlapping members will be merged in Step 3)

Step 3 is where the members are scanned for intermediate span nodes. This takes us from diagram B to diagram C. Referring to diagram B, member 1-2 has two intermediate nodes (5 and 6), member 6-7 has one intermediate node (node 2), and so on. So all the members with intermediate nodes are broken up, shown in diagram C.

For step 4, the duplicate members created in step 2 will be merged. This takes us from diagram C to diagram D. Looking at diagram C, the duplicate members are shown as the double lines. When duplicate members are merged, the lower numbered member is kept and the higher numbered member is deleted. Any loads applied to the higher numbered member are transferred to the lower numbered 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**

- There is an order of precedence for the section sets based the order of the section sets on the **Sections** spreadsheet, i.e. the first set of properties takes precedence over all subsequent section sets. The second set takes precedence over all section sets except the first section set, etc. The merging of duplicate members can result in member properties being changed for merged members depending on which member is kept and which member is removed

---

**Modeling Tips**

**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 node at the wall because the plate/shell element does not model in-plane stiffness. A fairly simple work around is to use rigid links to transfer the bending moment from the node at the wall as shear forces to the surrounding nodes in the wall. Note that 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 node to the interior wall nodes without having the rigid links affect the stiffness of the shear wall. Notice from the figure that the I-node for all the links is the wall node connected to the beam element, while the J-nodes are the ends that extend into the shear wall. The J-ends of all the rigid links should have their \( x \), \( M_x \), \( My \), and \( Mz \) degrees of freedom released. Only the \( y \) and \( z \) degrees of freedom should be connected from the J-ends to the interior wall nodes. (Remember that all member end releases are in the local member axes. 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 COMPOSITE BEHAVIOR**

Occasionally you may want to model a structure with composite behavior included. 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. An example of a plate/beam model with composite action included is shown below:

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.
The beam member and plates are broken up with extra intermediate nodes so that better shear transfer between the elements can be modeled. You want more connection locations than just at the end-points. As shown above, a rigid link is used to connect each set of nodes between the beam and the plates. This rigid link is fixed to each node and therefore has no member end releases.

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 nodes down to the centerline elevation of the beams.

Next you would lay out your beams along the existing nodes at the beam centerline elevation. (Note: don’t assign the members between each node, just assign them from end-node to end-node, the next step will break up the member at all the intermediate nodes) Now you can do a Model Merge, which will break up the beams at all the intermediate locations.

Next you will want to delete out all the extra interior nodes that were copied down from the plate elevation by using the Delete Selection option to delete all the selected, unattached joints. Lastly you will want to build your rigid link member and then connect it between all the plate and corresponding beam nodes. A good way to connect all the links is to generate grid members. You can also draw a row of them in “dot to dot” fashion with the Draw Members command and then copy them to other adjacent rows (This assumes the spacing on each row is equal).

Reactions at Nodes with Enforced Displacements

The reaction at an enforced displacement can be obtained by inserting a very short (.02’ or so) rigid link between the node with the enforced displacement and any attached members. The member forces in this rigid link will be the reactions at the node 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.

Modeling One Member Crossing 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 node at the point of intersection. The distance between the nodes would be half the depth of the top beam plus half the depth of the bottom beam. There are then 2 ways to connect the top and bottom nodes (nodes “A” and “B”). The first method is to simply slave the translational degrees of freedom from node A to node B (The rotations can be slaved as well if desired). Slaving is quick, but it causes the secondary moments to be neglected that are due to the difference in the beam elevations. The second, better way, to connect node A to node 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. Don’t release the bending degrees of freedom at both A and B or the shears will not be transferred correctly through the rigid link.

**Modeling Inclined Supports / Reactions**

You may model inclined supports by using a short rigid link to span between a node which is restrained in the global directions and the item to receive the inclined support. See the figure below:

![Diagram of inclined supports](image)

The rigid link should be “short”, say no more than 0.1 ft. The member end releases for the rigid link at node “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.

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 nodes “A” and “B”. You can also rotate the rigid link to the proper angle.

**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 (undeformed) 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.

A way to model a 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 nodal coordinates for your nodes at a trial deflected shape for the cable. Usually you can use just one member in between concentrated nodal loads. If you are trying to model the effects of cable self-weight, you will need to use at least 7 nodes 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 node at the approximate location of the “final” deflected shape position. Next you will connect your members to your nodes 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 nodal 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 nodes. If the new location is more than a few percent from the original guess, you should move the node to the midpoint of the trial and new location. You will need to do this for all your nodes. You will repeat this procedure until the nodes 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 nodal coordinates is to use the Block Math operation. That way you shift many nodes up or down by a small amount in one step.

**Applying In-Plane Moment to a Plate**

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 nodal 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**.

### Modeling a Plate Corner Moment Release

To model a moment release between 2 adjacent plate elements, insert a short (0.02 ft.), and very stiff rigid link member between each node that connects the plates. You will need to add a new node to do this, since each plate will now be connected to a separate node. See the figure below:

For each pair of adjacent nodes, fix one end of the link member to the first plate and pin the other end to the second plate. Don’t pin both ends or you won’t be able to transfer shear forces. Another way to model the plate moment release is to use Nodal Slaving instead of the rigid links. You would still need to add a new node (1 for the plate and 1 for the other plate or beam), but then you would slave only the translations between the 2 nodes. Slaving the nodes is faster to model than using the rigid link method, and it works well in most cases since there aren’t any secondary moments to account for.

You can follow the same procedure to model a moment release between a plate element and a beam or release other forces.
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 height of the columns in the stiffness matrix, the number of terms that need to be stored for the stiffness matrix, and the amount of RAM in your computer.

The maximum height of the columns in the stiffness matrix is reported to you at the beginning of the solution process. This matrix height is the number of stiffness terms from the main diagonal up to the last non-zero term. This number will generally be 1000 or less. Matrix heights in excess of 2500 are indicative of a modeling error. Basically, the larger the column height, the longer it will take to solve your model.

The number of terms in the stiffness matrix is reported to you along with the matrix height. The number of terms is important because this tells you how much space will be required to store the stiffness matrix. Each term takes 8 bytes, so the total required space is the number of terms times 8. If you divide this by 1 million, you will roughly have the number of megabytes needed to store the matrix. For example, let’s say you have 32 Mbytes of RAM and you try to solve a model with 10 million terms. This model will take approximately 80 Mbytes to store the stiffness matrix. Your operating system and RISA might together take up about 25MB of RAM, leaving 7MB for the solution. This means that you will only be able to store 10% of the stiffness matrix at one time, which will result in extensive use of your hard drive for swap space and will slow your solution down enormously. This example illustrates how important it is to have as much RAM as possible. If you regularly solve models that have 10 million terms or more, you should consider having at least 64 Mbytes of RAM in your computer. Twice that, or 128MB, will go a long way to solving those large models even faster.

A bandwidth minimizer is used at the beginning of the solution to try to reorder your degrees of freedom to get a reasonably sized column height and number of stiffness terms. Sometimes, however, the bandwidth minimizer can be fooled and will give a very bad matrix column height and a huge number of matrix terms.

If you are getting a very large matrix height (greater than 1500) and you don’t think that you have any modeling errors, you can try a few things to reduce the height. The first thing you can try is to sort your nodes. Typically you will want to sort your nodes on the Coordinates spreadsheet from “Low to High” in the 2 lateral directions and then lastly in the vertical direction. After you sort your nodes, try to solve again and check the matrix height and number of terms. The order of sorting depends on the model, so you might want to try a couple of different combinations and check the height and terms each time. Sometimes, sorting the nodes 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 minimizer starts on the small model and then jumps to the big model. 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, these are: the amount of RAM in your computer, the amount of free hard disk space, the operating system, and the Virtual memory settings in the Windows Control Panel.

If you got an error that states “You have run out of address space....”, you will want to note the amount of address space that was available at that time. This amount is 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 for Windows 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 window, 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.

**Note**
- All of the model view windows 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 Options menu and selecting Preferences. You may then use the Refresh All button on the RISA Toolbar to update the windows manually.

### 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. Assign a label on the left side of the spreadsheet. Moving loads are later included in your analysis by referencing this label.
3. Specify a pattern in the Moving Load Pattern field by selecting it from the drop down list.
4. In the Load Increment field specify the distance for the load pattern to be stepped through the path.
5. 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 co-linear nodes 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 node to the next node in
the load path sequence. The load path taken will usually be the most direct route between
the nodes 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 window.
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 dialog 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 enter a dialog by clicking and select the moving load label from the drop down list
  box.

Note
• You can only have one moving load in each load combination. This restriction on moving
loads also applies to “combinations of load combinations” created by nesting one load
combination in another.

MOVING LOADS SPREADSHEET
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 node all the
way to the last node of the load path. The load is then turned around and the last
node is now treated as the first node in the load path. The load is then run back to
the first node 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 node to the next node in the load path
sequence. The load path taken will usually be the most direct route between the
nodes. If you have a long series of co-linear members, or if there is only one
valid path between your start and end nodes, you usually will only need to specify
the first node and the last node in the series. If there are several members that
Moving Loads

branch from a node that are all part of valid paths to the next node 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 nodes at each intersection point. See the figure below:

In the example moving load path shown, you would need to specify nodes A, B, C, and D as the load path nodes. You would not have to specify the nodes 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 node 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 entered using the "mlpath=" parameter in the configuration file. You can add up to 500 different moving load patterns in the pattern database.

When you add a new pattern, the dialog will come up with all blanks. If you are editing an existing pattern, the pattern data will be displayed in the proper fields. The Pattern Label must be a unique name for your moving load pattern. Your moving load pattern 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 vertical axis direction is set on the Global Parameters window. The Distance is the distance between the loads.

When you’ve finished adding or editing your moving load pattern click OK to save your changes.
**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.

You can specify the load pattern, the size of the increment used to move the loads, and the path that the moving load takes through your structure. You can also specify whether or not to model the moving load both ways, which means that it is moved through the path, turned around, and moved back through the path from the end to the beginning.

**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 longer than usual.

**Moving Load 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.

**Nodes**

Nodes are used to define the ends of members and plate corners. Nodes are also used to specify boundary conditions, diaphragms, story drift locations, and joint loads. Each node 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.

Nodes may be input manually, or they may be created automatically as you draw new members and plates on the drawing grid. The nodes, once defined, may also be edited.

**Note**
- The terms “node” and “joint” are interchangeable and are both used in this manual and in the program.

**To Define Nodes**
- Select the Node Coordinates spreadsheet from the Spreadsheets menu and define the node coordinates and temperature.
**Note**
- You may use cut and paste, block fill and block math to enter and edit nodes.
- You may choose the prefix that is used to label the joints.

**To Relabel Nodes**
- After sorting the Joints spreadsheet into the desired order select the Utility menu and choose Relabel Joints.

**Node Coordinates Spreadsheet**

Joint coordinates and ambient temperatures are recorded in the Node Coordinates spreadsheet.

The first column is used to assign a unique label to every joint. You can then refer to the node by its label, which will not change even as you add and subtract nodes. The label can be anything you like so long as it doesn’t start with a number. For example, “NR6” would be a valid label, but “6NR” would not. 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 node.

The next columns contain the coordinates of the node in each of the global directions. These represent the relative offsets of the nodes from an arbitrary origin. The appropriate units are listed at the top of each column.

The last column is used to define the ambient, no-stress, temperature at the node. Temperature loads are then calculated based on the differential between the ambient temperature interpolated across the member, and the applied thermal load.

Two utilities in the Utility menu are available for manipulating the node labels and coordinates. The first is Relabel Joints. Invoking this utility will bring up a dialog where you can define a prefix to be used for the joint labels. The program will then create 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 utility is Round Off Joint Coordinates. This utility is used to round off all the joint coordinates to a user-defined number of decimal places (1, 2 or 3). This is useful for models that have been created using certain generation function or have been created from a DXF file, etc, where a high number of decimal places is present for some of the coordinates. Maintaining a consistent number of decimal places is a good idea because some functions in the program, such as coplanar checking for plate nodes, may give unnecessary warnings or errors because of differences in coordinate values that are numerically insignificant.

**Joint Displacement Results**

Access the Joint Displacement spreadsheet by selecting it from the Results menu.
These are the displacements for every joint in the structure. Each joint has 6 values calculated, 1 for each of the 6 global degrees of freedom, these 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 (360 degrees = 2*PI radians).

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 “Env” column.

**Note**
- To view the results for a particular member, use the **Find** option. To view the maximums and minimums, use the **Sort** option.
- 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 actual joint displacements are multiplied by this factor to give the deflected shape.
- 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.

**JOINT REACTION RESULTS**

Access the **Joint Reactions** spreadsheet by selecting it from the **Results** menu.

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. 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 (including self-weight, if appropriate) in each direction. To have a reaction calculated at a point of support, define the boundary condition for that point of support with either an "R" (Reaction) code or with a spring (the "S" code). Points of support defined with the "F" (Fixed) code do NOT have a reaction calculated. The units for the reactions are listed at the top of each column, and a total reaction for each direction is calculated and listed at the end of each column.

Also displayed here is the Center of Gravity (COG) for the applied loads. This COG is based on the load components acting in the VERTICAL direction, whichever axis is the vertical axis. 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. To include a particular Load Combination in the envelope analysis, open the **Load Combinations** spreadsheet and check the box in the “Env” column.

**Note**
- To view the results for a particular member, use the **Find** option. To view the maximums and minimums, use the **Sort** option.
• These results may also be displayed graphically by using the **Plot Options** dialog.

• If 'NC' is listed, this means "No Calculation". NC is listed for boundary conditions defined with the 'Fixed' (as opposed to 'Reaction') code, and also for joints with an enforced displacement in the particular direction.

---

**Node Loads - Enforced Displacements**

You may specify node loads and enforced displacements in any of the global degrees of freedom. Loads and displacements may be applied in any non-global direction by defining in terms of the resultant in the global axes.

**Note**

• If you have a boundary condition for the same nodal degree of freedom that you have an enforced displacement assigned, no reaction will be calculated.

---

**JOINT LOAD SPREADSHEET**

Node loads and enforced displacements are recorded on the joint load spreadsheet. When you open this spreadsheet you may view only one basic load case at a time. Use the drop down list box on the toolbar to specify a load case. The current load case is displayed in the title bar at the top of the window.

The first column contains the label of the joint the load or displacement is applied to. The same joint may be entered any number of times.

Next is a flag that indicates the value is a load or an enforced displacement. Enter "L" if it's a load, "D" if it's a displacement.

The direction code indicates which of the global directions the value is applied in. Valid entries are X, Y or Z for the translational directions, or Mx, My or Mz for the rotational directions.

Finally, the value of the load or displacement is entered.

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 or a displacement, and whether the direction is translational or rotational.

You are allowed up to a maximum of 1000 enforced displacements.

**Note**

• If you have a “Reaction” or a “Spring” boundary condition for the same nodal degree of freedom that you have an enforced displacements assigned, that NO reaction will be calculated. See the Modeling Tips section to learn how to obtain a reaction at a node with an enforced displacements.

---

**To Apply Joint Loads 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.
You may choose to apply the load to a single node at a time or to an entire selection of nodes.

To apply the load to a few nodes choose **Apply Entry by Clicking Items Individually** and click **Apply**. Click on the nodes 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** dialog.
- You may also specify or edit joint loads/displacements in the **Joint Load Spreadsheet**.

---

**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.

**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):

\[ P \times \Delta = V \times L, \]
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.

The solution sequence is as follows:

1. Solve the model with original applied loads
2. Calculate V's for every member in the model
3. Add these V's to the original loads and re-solve
4. 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 automatically, before numerical problems develop. Error number 2011 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, so there is a problem. If this happens with your model, the model may be unstable under the given loads, or there may be local instabilities present.

**P-Delta Limitations**

The P\(\Delta\) algorithm is based on end-node displacements and will not automatically account for the effect of interior span forces on members. In the case of a braced frame with interior loading, the end nodes will not be able to displace significantly and very little P\(\Delta\) effect will be calculated. This “micro” or “member” P\(\Delta\) effect on members is often much smaller than the “macro” P\(\Delta\) effect experienced. One case where the “micro” P\(\Delta\) can be significant is truss chords in compression that have suspended point loads between the panel points.

One way to have this “micro” P\(\Delta\) effect included is to insert additional nodes along the member span. These nodes will deflect due to the interior span forces and the P\(\Delta\) effect will be calculated for the member.

P\(\Delta\) effects are not calculated for plates.

**Compression Only P-Delta Analysis**

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 nodes that are not connected to supports or other “stiff” members, the P-Delta algorithm could cause a nodal 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 very large amount of tension that also has extra nodes in-between the panel points.) If you have members with very large tensile forces, and intermediate nodes, 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 .5%. This means that the displacements from one solution to the next must vary by no more than \( \frac{1}{2} \) of 1 percent for the solution to be considered converged. You may adjust this tolerance on the Global dialog. 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.

Troubleshooting P-Delta Convergence

The first step in troubleshooting a P-Delta model that won't converge is to run the load combination without P-Delta analysis. This will help you make sure that degrees of freedom are not locked (IE. no locked nodes appear in the warning box after the solution is finished). If it turns out that degrees of freedom are being locked, this indicates instabilities that you will want to fix.

By far, the most common cause of P-Delta convergence problems is local instabilities. A local instability is when one part of the model, or even one individual member, is unstable while the rest of the model is stable. Local instabilities will cause the P-Delta analysis to diverge, so it is important to locate and correct them (if they exist in your model). 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. If there don’t appear to be locally unstable regions in the animation, the model as a whole may be unstable. See Testing and Correcting Instabilities for more on this.

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. This is most easily accomplished using the ALL code on the Boundary Conditions spreadsheet. For example, if the global Z direction is out-of-plane, you would enter “ALL” for a node number and enter “R” for the Z-translation.

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 it almost impossible to run a P-Delta analysis.
Plate Shells

The plate/shell finite element allows you to easily model shear walls, diaphragms, shells, spread footings, mat foundations, tanks and many other surface structures. We refer to the elements as plate elements, but they are actually plate/shell elements.

**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 nodes, you will need to create a drawing grid or define nodes on the Node Coordinates spreadsheet.
3. Click the button and set the plate properties. For help on an item, click and then click the item.
4. Click OK to start drawing plates by clicking on the grid points with the left mouse button. You must click four points in a clockwise or counter-clockwise order. To create a triangular plate click on the third node twice.
   If in step 3 you chose to click in grid areas then you create plates by clicking between the drawing grids.
5. To stop drawing altogether click the button, right click or press the Esc key.

**Note**
- Press CTRL-D to recall any of the Plate dialogs.
- To submesh quadrilateral plates click the Submesh Quadrilateral button and specify the mesh. Click the Submesh Triangular button.
- Click the Modify Plates button and set the parameters for the new plates. Check the Use? box for the items to apply. You may choose to modify a single plate at a time or to an entire selection of plates. To modify a few plates choose Apply Entry by Clicking Items Individually and click Apply. Click on the plates with the left mouse button. To modify a selection, choose Apply Entries to All Selected Items and click Apply.
- You may also specify or edit plates in the Plates Spreadsheet.

**Plates Spreadsheet**
The Plates spreadsheet is used to record the plate/shell elements of the model. Selecting Plates on the Spreadsheets menu will access this spreadsheet. The following are input columns on the spreadsheet that may be used to specify the plate label, geometry, and material.

**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, which will not change even as you add and subtract plates. The label can be anything you like so long as it doesn’t start with a number. For example, “BR6” would be a valid label, but “6BR” would not. Each label has to
be unique, so if you try to enter the same label more than once, RISA-3D will issue an error message.

**Plate Nodes**
The A, B, C, and D node entries are used to define the 4 corner nodes of a quadrilateral element. (To define a 3-node triangle element, just leave the D node entry blank, or make it the same as the C node.) The nodes must all lie on the same plane and be entered in either a clockwise or counter-clockwise sequence. The direction and sequence in which you define the nodes determines how the elements local coordinate system is set up.

**Plate Material**
The material set label links the plate with the desired material set defined on the Material spreadsheet.

**Plate Thickness**
The thickness field on the Plates spreadsheet is the thickness of the element. This thickness is constant over the entire element.

**Inactive Plates**
Making a plate inactive allows you to analyze the structure without the plate, without having to delete the information that defines the plate. This leaves plate data is left intact so the plate may be easily reactivated. This is handy if you want to try a frame with and then without certain plate, without having to actually delete the plate data.

Putting a “y” in the Inactive field makes the plate inactive, i.e. the plate is not included when the model is solved or plotted.

Another option for the Inactive field is to put an “E”. The “E” code means include the plate in the solution, but exclude it from the results list. So, a plate 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 (plate forces, corner forces, principal stresses). This is useful if there are certain plate whose results you’re not interested in. You don’t have to clutter up the results list with these plate and can concentrate on the plate you’re most interested in.

**Plate Local Axes**
The A, B, C, and D node entries are used to define the 4 corner nodes of a quadrilateral element. (To define a 3-node triangle element, just leave the D node entry blank, or make it the same as the C node.) The nodes must all lie on the same plane and be entered in sequence, in either a clockwise or counter-clockwise direction.
The direction and sequence in which you define the nodes determines how the elements local coordinate system is set up. The following diagrams illustrate how the elements local coordinate system is related to the node numbering sequence and direction:

![Diagram of 4 Node Quad Local Axes](image1)

![Diagram of 3 Node Triangle Local Axes](image2)

The local x-axis is defined as positive from the D node towards the C node for 4 node elements and from C towards B for 3 node elements. The local y-axis is then placed as close to pointing towards the A-node as possible. Note that for triangular elements, the y-axis will probably not pass through the A-node. For 3 node elements, the y-axis is “towards” the A-node 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 NATURAL COORDINATE SYSTEM**

The force and stress output is for point (R, S) on the element. The R, S system is the “natural” coordinate system for the element, and is defined such that the values of R and S vary from -1 to +1 within the element. The following diagrams illustrate how the “natural” coordinate system relates to the local axes:

![Diagram of 4 Node Quad Natural Coordinate System](image3)

![Diagram of 3 Node Triangle Natural Coordinate System](image4)

For the 4-node quad, the local x-axis is always parallel to the R-axis and the local y-axis is always parallel to the S-axis. For the 3-node triangle, the local x-axis is always parallel to the R-axis. Note that for 3 node elements, the S-axis always passes through the A-node and may not be perpendicular to the R-axis. The local y-axis may also not be parallel to the S-axis for a 3-node element.

For example, element joint A would have the (R,S) coordinates (-1,+1). Element joint B would have the (R,S) coordinates (+1,+1). The A-node for the 3 node element has coordinates (0, +1). The center of both element types is at (0,0) and this is where the stresses are calculated by default. If you want the results for the
element calculated at any point other than the center, enter the appropriate (R,S) coordinates on the Plates spreadsheet.

**Note**
- Although you may retrieve stresses from any location on the element, in general you should try to leave the (R, S) location at the element center (0,0). The reason for this is that a finite element will produce its most accurate stress/force values near the center of the element. So the farther you get from the center, the less accurate your results become. The exception to this rule is the corner force output. The forces produced by the corner force output are as accurate as the results obtained at the center of the element. The corner force output option is different than the results you would get by setting the (R, S) point to be at a corner! When you set the (R, S) point to a location other than at the center, the stress results are obtained at that location by interpolating the results from the center point and the “gauss” integration points. The corner forces for the corner force output are obtained by multiplying the global nodal displacements against the local element stiffness matrix.

**Plate Stress Results**

Access the Plate Stresses spreadsheet by selecting the Results menu and then selecting Plates ▶ Stresses.

The plate stresses are listed for the top and bottom of each active plate. The principal stresses sigma1 (σ1) and sigma2 (σ2) are the maximum and minimum normal stresses on the element at the specified (R,S) location. The Tau Max (τ\text{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 σ1 stress (in radians). The Von Mises value is calculated using σ1 and σ2, but not σ3, 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:

\[
\sigma_1 = \frac{(\sigma_x + \sigma_y) \cdot \left( \frac{(\sigma_x - \sigma_y)^2 + \tau_{xy}^2}{4} \right) \cdot \sigma_0}{2}
\]

\[
\sigma_2 = \frac{(\sigma_x + \sigma_y) \cdot \left( \frac{(\sigma_x - \sigma_y)^2 + \tau_{xy}^2}{4} \right) \cdot \sigma_0}{2}
\]

\[
\tau_{\text{max}} = \frac{(\sigma_1 - \sigma_2)}{2}, \quad \phi = \frac{1}{2} \arctan \left( \frac{\tau_{xy}}{\sigma_x - \sigma_y} \right)
\]

\[
\text{Von Mises} = \sqrt{\sigma_1^2 + \sigma_2^2 + \sigma_3^2 - \sigma_1 \sigma_2 - \sigma_2 \sigma_3 - \sigma_3 \sigma_1}
\]

The R,S coordinate system is the "natural" coordinate system for the element and is defined such that the value of both R and S vary from +1 to -1 within the boundaries of the element. For example, element joint A would have the (R, S) coordinates (-1,+1). Element joint B would have the (R,S) coordinates (+1,+1). The center of the element is at the (R, S) coordinates (0,0). If you want the results for the element calculated at any point other than the center, enter the appropriate (R, S) coordinates on the Plates spreadsheet. Keep in mind that the element calculations are most accurate at the location R=0 and S=0.
The angle, \( \phi \), is the angle in radians between the maximum normal stress and the local x-axis. The direction of the maximum shear stress, \( \tau_{\text{max}} \), 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 \( \sigma_x \), \( \sigma_y \), and \( \tau_{xy} \) 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. To include a particular Load Combination in the envelope analysis, open the Load Combinations spreadsheet and check the box in the “Env” column.

**Note**
- To view the results for a particular plate, use the Find option. To view the maximums and minimums, use the Sort option.
- These results may be displayed graphically with the Plot Options dialog.

**PLATE FORCE RESULTS**

Access the Plate Forces spreadsheet by selecting the Results menu and then selecting Plates \( \rightarrow \) Forces.

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 point (R,S) on the natural coordinate system.
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 $\sigma_x$ face) $Q_y$ is positive in the $z$-direction on the element face whose normal vector is in the positive y-direction. (The $\sigma_y$ 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. $M_x$, then, can 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, the total $M_x$ moment could be obtained by multiplying the given $M_x$ force by the length of side BC. (I.e., the distance from node B to node C) The total $M_y$ force can be calculated in the same way by instead using the length of side DC (I.e., the distance from node D to node C).

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 (although conservative) 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.

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. These forces are reported as a force/unit length, so 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 $F_x$ force could be obtained by multiplying the given $F_x$ force by the length of side BC. (I.e., the distance from node B to node 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 the shear force. For example, 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. (I.e., the distance from node B to node C)

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 $Q_x$ force of 20K would have a total shear 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. To include a particular Load Combination in the envelope analysis, open the Load Combinations spreadsheet and check the box in the “Env” column.

**Note**
- To view the results for a particular plate, use the Find option. To view the maximums and minimums, use the Sort option.
- These results may be displayed graphically with the Plot Options dialog.

**PLATE CORNER FORCE RESULTS**

Access the Plate Corner Forces spreadsheet by selecting the Results menu and then selecting Plates > Corner Forces.

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 the Plate Model Examples in the Plate section of the Reference
Manual for examples of 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 at each location 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 “Env” column.

**Note**
- To view the results for a particular plate, use the **Find** option. To view the maximums and minimums, use the **Sort** option.
- These results may be displayed graphically with the **Plot Options** dialog.

### 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 these subjects before, 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 used commonly 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 ($F_x$, $F_y$) and the in-plane shear stresses ($F_{xy}$). There would be no plate moments or out-of-plane shears generated.

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 only a uniform load or no 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.

One other comment on plate results is to point out the convention used for moments in plates. With beams, the $M_y$ moment describes the moment about the local y-axis. However, with a plate element, the $M_y$ moment is the moment that produces stresses in the local y-direction. The $M_y$ moment in a plate is actually about the local x-axis.

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 nodes 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.

Finite elements are also affected by geometric distortion. The “best” shape for the 4-node 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°, and for accurate results, shouldn’t even be approaching 180°.

One last item is that the element used by RISA-3D, like most 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 nodal moment to one of the internal nodes 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 nodal reactions and the element stresses. See Applying In-Plane Moment to a Plate if you need to work around this limitation.

PLATE/SHELL ELEMENT FORMULATION

The element used is a mixed interpolation 4 node quadrilateral element. 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 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-node triangle element that can be used to build transitional meshes. The stress characteristics of the triangle are not as accurate as the 4-node quad and use of the triangle should be limited. It is not recommended that the stresses from the 3-node triangle be used at all. RISA-3D provides a way to convert your triangular plates to quadrilaterals. {button, JumpID(Plate Submeshing)}  How?

**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 utilities. 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 Data” 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:

![Plate Distortion Diagram](image)

**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 refine a mesh of existing quadrilaterals, use the Sub Mesh Quads command. This will submesh all the elements that are selected on the model plot into a mesh of smaller elements, with the mesh ranging from a 2x2 to a 9x9. Performing a Model Merge afterwards will insure that all the new
elements get connected to existing beam elements and that duplicate nodes get merged.

What if you have some triangle elements, and now would like to get stress values from them, or just a more accurate mesh? To submesh existing triangle elements into a mesh of quadrilateral (4 node) elements, use the Sub Mesh Tris command. This feature will sub-mesh each selected triangle into 3 quadrilateral elements.

Note
- Remember to perform a Model Merge after using this feature.

Plate Generation
A fast way to build a new mesh of finite elements is with the generation features. RISA-3D currently provides several High Level Generation features to quickly build common structural shapes. These features provide easy ways to create cylinders, cones, grids, radial grids and disks of plates. The quickest way to see what these features do is to experiment with them. The Arc/Cylinder can be used for generating circular tanks and silos. This feature also allows you to linearly vary the radius to build tapered (conical) tank structures as well. The Radial Grid can be used to generate a “ring” of plate elements (like a donut). It can also generate a partial ring, which would resemble a curved arch. The Grid (Plates) can be used to generate straight grids of elements like shear walls or horizontal diaphragms. The Plate Disk can be used to generate a circular plate mesh.

Another quick way to build a finite element model is to draw large elements to be your floor slabs and shear walls with the graphical utilities. Then use the Sub Mesh Quads command to refine the mesh.

Note
- Before sub-meshing, make sure that any adjacent “large” elements connect at their corner nodes. That way, any subsequent sub-mesh operations will produce element meshes that automatically connect at all the intermediate nodes.

Plate Model Examples
Our website (www.risatech.com) contains 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.
Point Loads

Point loads are concentrated member loads applied along the span of a member and are recorded on the Point Loads Spreadsheet.

To Apply Member 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. For help on an item, click and then click the item.

3. You may choose to apply the load to a single member at a time or to an entire selection of members.
   - To apply the load to a few members choose Apply Entry by Clicking Items Individually and click Apply. Click on the members 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 dialog.
- You may also specify or edit point loads in the Point Load Spreadsheet.

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

Point Load Spreadsheet

The first column contains the label of the member to assign a point load.

The direction in the second column represents the direction of the load, and whether the load is a translational force or a moment.

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 unaffected by any member offsets and is the distance from the I-node of the member.

Note
- The location of the load can alternately be defined as a percentage of member length. To define the distance from the I-node 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 joint coordinates and you wish to have the load some proportional distance from the I end.

Printing

You may print graphics as they appear on the screen or print all or part of the tabulated results. If the current window is a spreadsheet and you click the print button the print report dialog is presented. If the current window is a graphic plot, and you select the print button, the Graphic Printing dialog is opened.

PRINTING REPORTS

The Report Printing Options box helps 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.

You may specify the text color as black or blue. You may choose to have every other line shaded to enhance readability or turn this off so that all the lines print on a white background. You may also 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 dialog 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.

The Item Options button takes you to the Item Printing Options sub-dialog where you can select member and plate related output options. Member output such as code checks or forces you may specify that you want the data to be listed for each section cut in the member or just for the member ends that can be useful for connection design. For plates you may indicate which surface forces you desire.

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.

Note

- If you are in a graphic window the print view dialog is opened.
SELECTING A PRINTING OPTION

Choose to print the current spreadsheet, a selected portion of the spreadsheet, or multiple sections by printing a report.

To print selected lines you must first select the lines by clicking and dragging on the row headings to the left in order to mark the rows in yellow.

The Report Printing Options box helps 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 PRINT GRAPHICS

- While in a graphic window click on the Print button and choose from the options.

Note
- If you are in a non-graphic window the print report dialog is opened instead.

GRAPHICS PRINTING

This dialog is used to control graphics printing. First, indicate whether the print is to be in portrait or landscape mode. For portrait mode, the height exceeds the width. Landscape is the reverse.

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.

The “Title Bar” is an informational bar listing various things such as the model title, the designer and company name, the date and time, etc. You have the option of drawing or not drawing this title bar. A check means the bar will be drawn. You can also enter a comment in the field provided that will be printed in the title bar.

Click Page Setup to set the page margins.

Once you have everything the way you want it choose Continue. This will bring you to the Print dialog that is specific to your printer. The choices are also printer specific 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.

Printing to a File
• While in a spreadsheet click on the Print button and choose to print a report. On the report printing dialog click Write Flat File.

Note
• A flat file is a file with no headings and is useful for importing and parsing into spreadsheets.
• To select multiple sections, in the Report Printing Options dialog, hold down the CTRL key and then click each item you want to select.
• To select a contiguous group of sections to print, click on the first section and hold the mouse button down while you drag the mouse to the last section.

Response Spectra Analysis

Response spectra analysis is the follow-up to the dynamic analysis that results in forces, stresses and deflections being calculated. 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. These modes are calculated via the dynamic analysis.

To Perform a Response Spectra Analysis
1 You need to solve a dynamic analysis first.
2 Select Response Spectra from the Solve menu.
3 Select the spectra to be used and specify the other parameters in the dialog. For help on an item, click and then click the item.

Note
• 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.

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 include more than one RSA solution you can also have RISA-3D combine the multiple RSA results using an SRSS summation, so as to predict the maximum combined RSA results. 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.
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, and "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. The 1997 UBC requires a dynamics based procedure for some structures. The response spectra method satisfies this dynamics requirement. The UBC allows you to use this method for ALL structures, if you wish so there really isn’t any reason to use the static procedure. The response spectra method is easier, faster and more accurate than 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

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 (the higher the participation factor, the more important the mode). The participation factor itself is the percent of the model’s total mass that is deflecting in the shape described by the particular mode. Thus, the sum of all the participation factors in a given direction cannot exceed 100%.

Generally, the amount of participation for the mode reflects how much the mode moves in the direction of application of the spectra. For example, if the 1st mode of the model 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.

**Note**
- For the RSA to be considered valid per the UBC, 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. This really isn’t an option; 90% participation is required by the UBC for the RSA to be considered valid (UBC ’94 Sect. 1629.5.1). Another useful feature of this participation report is that you can isolate which modes are important in which directions; i.e. if you solve for 10 modes but only 3 show significant participation, if you redo the RSA you can specify to just use those three modes. This will speed up the RSA significantly.

- You may also want to move the mass to account for accidental torsion.

**Modal Combination**

There are three choices for combining your modal results: CQC, SRSS, or Gupta. Generally, 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 (if this is the case, you also should set the previous field (Mode No. for Signs) to blank because it’s very unlikely the older program offers that feature).

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, 1990*, 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. It does satisfy the UBC ’94 Section 1629.5.2 requirements for combining modes.

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 (The periodic part is combined as per the CQC method). The Gupta method is fully documented in the reference, *Response Spectrum Method*, by
This method also satisfies the ASCE 4 Standard criteria for combining modal responses.

The Gupta method defines lower \( f_1 \) and upper \( f_2 \) frequency bounds for modes containing both periodic and rigid content. Modes which are below the lower bound are assumed to be 100\% periodic. Modes which are above the upper bound are assumed to be 100\% rigid. See the Reference Manual for the frequency bounds calculation.

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, so the final combined results are all positive!

The purpose of the RSA is to predict the maximum response of the model, and this it does, but if the results are all positive, it can be misleading to combine an RSA solution with a static solution, since for the static solution some of the maximum response values will be negative.

For a typical model the RSA for each global direction will be dominated by a single mode. If we base the signs for the final combined RSA results on the signs for the RSA for this single dominant mode, we’ll have a much better approximation to combine with a static solution. This isn’t perfect, of course, but it’s a lot better than using all positive RSA results!

The only trick is determining which mode is the dominant mode. Fortunately, this turns out to be easy to do. If you look at the participation factors for each mode for a particular direction, the mode with the highest participation factor is the dominant mode. That is the mode number to use here.

**Cut Off Frequency**

This is the “rigid frequency” used by the Gupta method to calculate the upper frequency 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. This is the value assumed for the UBC spectra. 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 not uncommon problem you may encounter in dynamic analysis are 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 are not immediately obvious from looking at the frequency or numeric mode shape results, but they can usually 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, 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.

To get rid of localized modes that are not the result of modeling errors, use boundary codes 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. Quite often, however, 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. If you try the model merge and are still having trouble, see Dynamics Modeling for more help on resolving local modes or getting 90% mass participation for your model.

RSA SCALING FACTOR

Probably the most difficult part of the entire RSA procedure is calculating the scaling factor to be used when including the RSA results in a load combination. The UBC places certain requirements on the value of the design stresses and forces when using a response spectrum analysis. (All UBC references are for the 1997 UBC.)

The 1997 UBC uses a particular “shape” for it’s spectra (See Figure 16-3), but the new parameters Ca (see Table 16-Q) and Cv (see Table 16-R) make it specific to a particular site. However, the UBC imposes several requirements regarding the minimum design values. UBC Section 1631.5.4 specifies that for regular structures using a response spectra constructed from UBC Figure 16-3, the model base shear from the RSA must be at least 90% of the base shear calculated using the UBC static procedure (UBC Sect. 1630.2.1). For irregular buildings, also using response spectra constructed from UBC Figure 16-3 the RSA must be at least 100% of the static base shear. In addition, there is the requirement that the design values not be less then Elastic Response values divided by the value ‘R’ (See UBC Table N and Table P for ‘R’ values.) The Elastic Response values are simply the results you would get when you include the Response Spectrum Analysis results in a load combination with a scaling factor of 1.0.
The static base shear \((V)\) is calculated using the equations in UBC Sect. 1630.2.1:

\[
V = (Cv \times I \times R \times T) \times W \quad (eq. 30-4)
\]

where

- \(Cv\) = Seismic Coefficient, UBC Table 16-R
- \(I\) = Importance Factor, UBC Table 16-K
- \(R\) = Coefficient from UBC Tables 16-N and 16-P
- \(W\) = Structure Weight (UBC Sect. 1630.1.1)
- \(T\) = Structure Fundamental Period (UBC Sect. 1630.2.2)

Note that there are limiting values for the minimum static base shear in UBC section 1630.2.1 (See equations 30-6 and 30-7). The static base also need not exceed the value computed by UBC section 1630.2.1 (eq. 30-5).

So 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 UBC) 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 on the Dynamics dialog for the dynamic analysis). 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 based on \(T\) calculated in this manner, see the first paragraph regarding Method B in UBC section 1630.2.2)

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. For example, again assuming we're looking at a Z direction RSA, enter “SZ” in the BLC field and for the Factor enter "1". Leave all the other BLC fields blank. Solve the load combination. The Z direction reaction total is the unscaled RSA base shear.

Now, to get the correct scaling factor, solve this equation:

\[
\text{Scale Factor} = \left( \frac{V}{\text{Unscaled RSA base shear}} \right) \times K
\]

If this is a "regular" structure, the UBC only requires you to scale to 90% of \(V\), so \(K\) would equal 0.9. For an "irregular" structure, \(K\) equals 1.0.

You would do this calculation to obtain the scaling factors for all the directions of interest (X, Y and/or Z). Remember, the fundamental period for each direction probably is different (unless the model is perfectly symmetric), so be sure to use the proper value for "\(T\)" for the direction being considered.

Note: The UBC has additional requirements for vertical seismic components. (See UBC section 1630.11 and 1631.2.)
UBC 97 Seismic Coefficients

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).

Response Spectra Database

You may add your own spectra to the database and edit and delete them once they are created. All of the spectra that are in the database are shown in the drop-down list above. A tripartite plot of the selected spectra can also be generated.

Note
- The spectra database should be stored in the directory that is specified by the configuration file.

Adding and Editing Spectra

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 in the dialog. For help on an item, click ? and then click the item.

Note
- Zero values are not allowed in the data.

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 is as follows:
\[ T = \frac{1}{f} \]
\[ S_v = 2 \pi f S_d = \frac{S_a}{2 \pi f} \]
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 \( S_v \) values (labeled on the left side) and the diagonal axes plot the \( S_a \) (lower left to upper right) and \( S_d \) (upper left to lower right) values.
The spectra data itself is represented with the thick red line. Therefore, to determine the \( S_v \), \( S_a \) or \( S_d \) 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 \( S_v \), \( S_a \) and \( S_d \) values from their respective axes. Remember, all the axes are logarithmic!

**Saving the Response Spectra Solution**

After you’ve done the response spectra solution, you can save that solution to file to be recalled and used later. This option is called a **save w/Dynamic Restart** and is located on the **File** menu.

**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.

**Sections Sets**

The cross section data for the members is recorded on the **Sections** spreadsheet. The **Section Set** is then referenced on the **Members** spreadsheet to assign properties to a member.

Cross sectional properties can be entered manually or may be retrieved from one of the shape databases. Currently the databases include steel shapes for AISC, Canadian, and Trade ARBED and wood shapes for the NDS specification.

**To Define a New Section Set**

1. On the **Spreadsheets** menu click on **Section** to open the spreadsheet.
2. 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.

   You may choose from steel sections by clicking on the arrow in the **Shape** column.

   To define timber sections you first need to define them on the **Wood** spreadsheet and then choose that material in the **NDS Label** column.

   To define a rectangular, circular or pipe shape you should use the on-line shape definition to automatically calculate the properties.
Section Property Sets

Note
- When you choose a shape from the database the properties are automatically filled in for you. You may change these properties but they will only remain for the current session. If you save the file and open it later it will use the original database properties.

Sections Spreadsheet

The Sections spreadsheet is used to record cross section data for the members in the model. Accesses this spreadsheet by selecting Sections on the Spreadsheets menu. The following are input columns on the spreadsheet that may be used to specify the section set label, geometry, orientation, and end conditions:

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 Material
The Material field is used to enter the label of the desired material. The label must be either one of the labels defined on the Materials spreadsheet or one of the labels defined on the Wood Properties spreadsheet.

Section Database Shape
The Shape field is used to obtain properties from the shape databases. To use a database shape, simply enter the shape name in this field and the shape properties will be filled in automatically. Click to see a list of available database shapes.

If you don’t want to use a database shape and wish to enter the shape properties directly, just leave this field blank. Note that for Wood sections, this field will be filled in automatically when you assign a Wood set in the Material Label field.

Note
- For rectangular, circular, and pipe shapes you can use the On-Line shapes to automatically define the shape properties.

Cross Section Properties
The cross section properties will be filled in automatically if you use a database shape. Otherwise, you may enter these directly. Iyy and Izz 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 be replaced with the original values when the cursor moves over the Database Shape field or the file is read again from the disk.

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 dialog.
Section Shear Area

The SA coefficients are used to calculate the effective shear area for shear force applied parallel to the member’s local y-axis (SAyy) and local z-axis (SAzz) and for shear deformations.

T/C Sections

The T/C field is used to indicate that a section is to be Tension or Compression only. When a section set is flagged as “C”, any members that reference this section set will only be able to take compressive loads. The member will have no stiffness to resist tensile loads. When a section is flagged as “T”, the member will primarily take only tension loads, however it will also take some compression load, up to its Euler buckling load.

Wood Sections

For Timber code checking to be performed on a member, the member must be defined with a NDS database shape. The Section Set assigned to the member must have a Wood Material Label in the Material field on the Sections spreadsheet. This is because there are parameters needed for the code checking that are obtained from the shape database, such as section modulus and allowable stresses. When you reference a label from the Wood Parameters spreadsheet, you are actually referencing material AND shape properties.

You will notice that when you reference a Wood label, that the “Shape Database” field is automatically filled in with the Wood NDS Species Database Label and the shape properties are automatically assigned. The only properties that need to be changed are the SA factors, which should be set to 1.5 for rectangular shapes, and 1.1 for round shapes. (The SA factor affects the shear area for the shear code check)

Note

- If you try to edit the Shape Database field of a section set that references a Wood label, the changes will not be saved, and the referenced wood shape will be rewritten to the field. If you change the displayed shape properties, these values will only be used in the current session. They will not be saved and the database values will be restored the next time that you open the file.

Shear Area

The shear area for the particular direction is calculated as A/SA, where ’A’ is the effective cross sectional area in the appropriate direction.

For example, for the local y direction of a Wide Flange shape, the y direction shear area would be \( (d \times tw) / SAA_{yy} \), where \( d \) = depth and \( tw \) = web thickness. For the local z direction, the shear area would be \( (2 \times bf \times tf) / SAA_{zz} \), where \( bf \) = flange width and \( tf \) = flange thickness. Similar calculations would be used for other shape types. In other words the unique characteristics of the various database shape types are recognized (WF, Tube, Tee, etc.) and taken into consideration when calculating shear area.
The default value for SA is 1.2, which is the correct value for WF shapes when the average area used is the area of the web or flanges. Other common shapes and their values for SA are as follows. For a Solid Circular shape (BAR), SA is approximately 1.1. For a Hollow circular shape (PIPE), SA is approximately 2.0. For Solid Rectangular shapes (RE), SA should be 1.5.

The effect of the As coefficient is to reduce the actual area to reflect the fact that the shear stress is distributed in a parabolic fashion, thus the maximum shear stress is greater than Fv/A (Fv = shear force). By reducing "A", we gain a better approximation of what the maximum shear stress actually is. The effective shear area is also used in the shear deformation calculations for beam elements.

**Tension Only and Compression Only Sections**

Members defined with Compression Only sections will have no stiffness to resist tensile loads. Members defined with Tension Only sections will primarily take only tension loads, however it will also take some compression load, up to its Euler buckling load.

Letting the tension only members take a little compression helps model convergence immensely. Compressive loads in excess of the Euler buckling load will cause the member to be turned "off" and the member will cease to have any stiffness to compressive loads. You can control the amount of compression that a tension only member takes by artificially 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 the AISC Parameters spreadsheet. The larger the Kl/r ratio, the less compression that a tension only member will take.

**Note**
- 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, and if any members are turned off (or 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.

**Shape Database**

There are several databases of common structural shapes such as Wide Flanges, Tee’s, Tubes, etc. You may type in the names directly, select shapes from these databases or add your own shapes.

**To Select a Database Shape**

1. On the Sections spreadsheet move the cursor to the Shape field and click <button>
2. Specify the database and shape type you wish to use and then select from the list of available shapes by clicking on <button>
Note
- You may type the first few characters of the shape name and the drop down list will scroll to relevant selections. For example, to quickly see the list of W12 wide flange shapes type W12 and the list will show the W12 shapes you may choose from. You may have to click the find button if you are typing characters that don't match the specified shape type.

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.

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. If you would like the added shapes to appear with similar types (i.e. New wide flanges displayed with the existing wide flanges) you may sort the database. To sort a database click , select the database, select a shape type, and click Sort.
- 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 Properties. 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.

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.

Database Shape Types
There are ten types of shapes including Arbitrary and On-Line shapes. Names for each shape type follow a syntax so that they may be typed directly into the Database Shape field on the Sections spreadsheet. Alternately you may click the button to look up a shape and select it.

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. (You would need to make sure you had Trade Arbed database loaded first, or you will get an error!) Canadian shapes use the same format as the AISC shapes, but there values are metric (I.e. the depth is called out in millimeters and the mass per length is kg/meter).

**Tube Database Shapes**

Tube shapes are hollow rectangular shapes. The prefix for older AISC tube shapes is "TU". Tubes using this prefix will always have their design wall thickness be the same as their 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).

The new HSS tube shapes 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, a HSS12X10X8 would be a 12" deep, 10" wide tube, and 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.

**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 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 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 nodes 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 nodes.

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%.

• 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 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 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 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.

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. By entering the "d" values (the distances to the extreme fibers), you'll give enough information to calculate stresses at the extreme fibers of any shape.

AISC code checks are not calculated for arbitrary shapes and arbitrary shapes cannot be optimized. Everything else will be calculated for them (forces, deflections, stresses). The max thickness (Max t) value for the cross section is used to determine the pure torsional shear stress for the shape.

To enter an arbitrary shape in the database click , select the Arbitrary Shape Type and click Add. Enter the shape name and properties.

Database Files

The shape databases are stored in the files risadb32.fil, arbed32.fil and canada32.fil. These files are read to bring in the list of valid shape names.

Note

- The path to these files is controlled by the configuration file record "shapespath".

Slaving Nodes

When a node is "slaved", it is linked in one or more directions to other nodes (the "master" nodes). The slave and master nodes actually share the same degree of
freedom for the direction of slaving. The slaving can be for any or all of the
global degrees of freedom. A node can be slaved to more than one master (in
different directions). Any number of nodes may be slaved to the same master, but
the master node itself may not be slaved to another master.

An example of the use of node slaving is in X bracing,
where the bracing members overlay and are pinned to
each other. This pinning will force the brace midpoints
to deflect together for translations, but still leave them
free to rotate independently.

In this diagram, brace 1 to 3 is modeled as members 1
to 5 and 5 to 3. Brace 2 to 4 is modeled as 2 to 6 and 6
to 4. Nodes 5 and 6 are at the same location where the
braces overlap and are pinned together.

We would slave node 6 to node 5 for the translation directions. To define a slave
node, use the Boundary Conditions spreadsheet. Enter the number of the node
to be slaved as the "Joint No." , and for the directions in which it is slaved, enter
SLAVE nnn, where nnn is the number of the master node.

For this X brace example, we would enter this on the Boundary Conditions
spreadsheet:

6  SLAVE 5  SLAVE 5  SLAVE 5  (blank)  (blank)  (blank)

where we have "SLAVE 5" entered for the X, Y and Z translations. The rotation
fields are left blank because node 6 is free to rotate independently. We don't have
to enter anything for node 5.

Note
- Slave nodes should NOT be used to build rigid diaphragms. Slaving the translations will
  not give the desired diaphragm behavior. If you need a rigid diaphragm, you should use
  our Rigid Diaphragm feature.

Stability

Model instabilities occur whenever a nodal degree of freedom can deflect
continuously without any resistance. Instability can also occur when there is very
little stiffness to resist deflection, as opposed to zero stiffness.

If the degree of freedom that represents this deflection is not loaded, then no
deflection will occur, but just the possibility of such an unrestrained deflection
cannot be ignored. These instabilities are detected in the model as zeroes on the
diagonal of the global stiffness matrix and when left alone they prevent a solution.

INSTABILITY PROCEDURE

Instabilities will be locked at the beginning of the solution process by applying a
FIXED code to each degree of freedom that exhibits zero or very little stiffness.
By "lock the instability" we mean the unstable degree of freedom is given a very large stiffness value such that it cannot displace.

At the end of the solution you will be informed that joints have been locked and that you may view the affected joints in a model view window. These locks will also be reported in the Reactions spreadsheet, which will be displayed when the solution is finished. This allows the solution to proceed to conclusion.

**Note**
- The locked joints should **NOT** be ignored. Each locked joint should be examined to determine whether the instability being fixed is a real problem or just a minor numerical problem.

**Instability Examples**

For three-dimensional models, torsional instabilities are not uncommon. A "torsional" instability is where a member, or a series of co-linear members, is free to spin about its centerline (local x) axis. This is because a member that is modeled with "AllPIN" member end releases is released rotationally about all three of its local axes at that end. This diagram illustrates such a situation:

Members 1,2 and 3 are "locally" stable, meaning they are fixed torsionally at one or both ends. The columns and beams framing in at the ends of member 2 (members 4,5,7,8) are pinned so they are not providing any torsional stiffness to the 1,2,3 series. The member series (1, 2 and 3) as a whole is torsionally unstable. This is a very simple example; in a large, complex model, it would be very difficult, if not impossible, for the user to identify and address all instabilities such as this.

Although some instabilities 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.

Another common mistake that results in model instabilities is illustrated in this diagram:

At first inspection, this model looks fine. The problem can arise in the definition of the horizontal beam. If the horizontal beam is defined as a single member with an I-node of 5 and a J-node of 6, the member is not stable. It’s “floating” in space. Even though the beam passes through nodes 2 and 3, it is NOT attached to these nodes. The proper modeling of this beam would be as three separate members, 5-2, 2-3 and 3-6.

Another example of a potential local instability is X-bracing with a center node 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 node 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.

Finally, the model not being continuously connected will cause instabilities. (IE, beams or plates are close to each other, or even touching, but not actually connected).

**Testing and Correcting Instabilities**

Often, the model not being continuously connected will cause instabilities. (IE, beams or plates are close to each other, or even touching, but not actually connected). A quick way to try to fix this condition is to run a Model Merge. A special note is in order for models with both beam and plate elements. All connections to the plates must be made at their corner joints. The model merge will NOT break up plate elements which have unattached nodes in their middle or along their edges.

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.

**A good general diagnostic technique** for locating instabilities is to do a static solution of the model with self-weight applied in all 3 directions, and with factors of 1.0. Then, magnify the deflected shape and animate it. Very often, any parts of the model with very low stiffness will be immediately obvious, as they will deflect much more than the surrounding parts of the model. If you have instabilities in more than 1 direction, you may need to run a static solution of the model with the self-weight applied in 1 direction at a time.

This technique will help to identify instabilities that are caused by a very small stiffness, as opposed to a zero stiffness. Zero stiffnesses are almost always found and locked by the program.

Once you’ve identified areas of your model that may be unstable, now you need to do something about it. If there are parts of your model that are not connected, then you will need to connect them to the rest of your model. Other possible causes of model instabilities include; the boundary conditions, the member end releases, members with very tiny shape properties, or very weak material properties. These items will each be discussed in the following paragraphs.

Examine your Boundary Conditions and make sure that you have provided adequate restraint for your model. Make sure the model has some restraint against moving in all global directions (X,Y,Z) as well as restraint against rotating about the global axes (MX, MY, MZ).

You should replace all “ALLpin” member releases, with “BENpin” member releases. You will only use the “ALLpin” release in very special circumstances. If
you think that a member needs to have an “ALLpin” release, then only apply it at one end of the member. The other end of the member should have the “BENpin” code.

A typical problem with boundary conditions and member end releases is the case where a joint gets “over released”. This commonly happens where a column member connects to a joint that is “Pinned” (IE. it only has boundary condition restraints in the translational directions) and the column member also has a “BENpin” release code at that same joint. This joint will have no stiffness for moments about 2 of the global axes. The solution is to either remove the member end release code, or to apply boundary condition restraints for the global rotations at that joint. Another common case where this happens is when all the members that connect at a joint have the “BENpin” end release code. The joint again has no stiffness for global rotations. The solution is removing the “BENpin” code for one (1) of the members, which will give the joint some stiffness. Note that all the members connecting at the joint will still be “pinned” and will not transfer any moment.

If you have members with very tiny shape properties or weak material properties any joints connected to these items may have very small stiffness. You will want examine why these elements have such small properties and if they can truly handle any loading. If you are using members with small “I” properties to try to model a cable, you will want to see the Modeling Tips section, “Modeling a Cable”.
Full code checking can be applied to standard steel shapes and the shape may then be optimized. Based on the AISC 9th Edition ASD (Allowable Stress Design) or AISC 2nd Edition LRFD (Load & Resistance Factor Design) criteria the calculations encompass all the code requirements including Appendix B, for standard beam/column shapes. All square and rectangular tubes (HSS) code checks using the LRFD code will be done using the HSS Specification. The Canadian CAN/CSA 216.1-94 code is also supported for standard beam/column shapes.

**To Apply a Steel Design Code**

1. On the **Global Parameters** window, select the steel code from the drop down list box
2. To optimize the shapes check the **Do Redesign** box
3. On the **AISC Parameters** 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 **Sections** spreadsheet.

**Member Design Parameters**

Parameters controlling steel and wood member 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. Notice that the steel AISC parameters are the initial column labels. The secondary column labels are for the NDS wood parameters.

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. For the Canadian CAN/CSA 216.1-94 code design parameters see Canadian Steel Design Parameters. For NDS design see NDS Wood Parameters.

**AISC Steel Design Code: Unbraced Lengths**

The unbraced lengths are $L_{byy}$, $L_{bzz}$ and $L_{comp}$. The $L_b$ values represent the unbraced length for the member with respect to bending about its local $y$ and $z$ axes, respectively. These $L_b$ values are used to calculate $KL/r$ ratios for both directions, which in turn impact the calculation of $F_a$, allowable axial stress, for ASD code checking or $P_n$, the axial strength, for LRFD code checking. 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 value is the unbraced length of the compression flange. This may be the same as the Lbyy value, but not necessarily. The Lcomp value is used in the calculation of Fb, allowable bending stress, for ASD or Mn, bending strength, for LRFD. Refer to Chapter F of either 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.

These unbraced lengths all default to the member’s full length if left blank, except if Lbyy is entered and Lcomp is left blank. Lcomp will default to the entered value for Lbyy. Note that there is currently only one Lcomp value used for both the top and bottom flanges. For many framing conditions, the moment in a beam will reverse, and for the same member, the top and bottom flanges will be in compression for different portions of the member span. A future enhancement to RISA-3D will be the addition of another Lcomp field so that the top and bottom unbraced lengths can be specified separately.

K Factors (Effective Length Factors)
The "K" factors are also referred to as effective length factors. Kyy is for bending about the member’s local y-y axis and Kzz is for bending about the local z-z axis. The K-values, if not entered (left blank), will default to 1.0. See the ASD or LRFD code commentary for Chapter C for an explanation of how to calculate the K values.

RISA-3D is able to approximate the K values for a member, based on that member’s sway condition and end release configuration. The K-factor approximation is based on Table C-C2.1, found on page 5-135 of the ASD code or page 6-184 of the LRFD code. Following are the values used for various conditions:

<table>
<thead>
<tr>
<th>Table Case</th>
<th>End Conditions</th>
<th>Sidesway?</th>
<th>K-Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>(a)</td>
<td>Fixed-Fixed</td>
<td>No</td>
<td>.65</td>
</tr>
<tr>
<td>(b)</td>
<td>Fixed-Pinned</td>
<td>No</td>
<td>.80</td>
</tr>
<tr>
<td>(c)</td>
<td>Fixed-Fixed</td>
<td>Yes</td>
<td>1.2</td>
</tr>
<tr>
<td>(d)</td>
<td>Pinned-Pinned</td>
<td>No</td>
<td>1.0</td>
</tr>
<tr>
<td>(e)</td>
<td>Fixed-Free</td>
<td>Yes</td>
<td>2.1</td>
</tr>
<tr>
<td>(f)</td>
<td>Pinned-Fixed</td>
<td>Yes</td>
<td>2.0</td>
</tr>
</tbody>
</table>
Any configuration not described here would get 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 by directly entering the value that you want in the appropriate field. If you have RISA-3D perform another approximation the manually input values will be overwritten.

**Limitations**

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. 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 that 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.

**AISC Steel Design Code: Cm Coefficients (Interactive Bending Coefficients)**

Chapter H of the ASD code describes the Cm coefficients. 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 Comb to the next so it may be a good idea to leave these entries blank.

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. Code checks will not be performed without P Delta analysis.

**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.
AISC Steel Design Code: Bending Coefficients

This coefficient is discussed in Chapter F of the ASD code and is used in the calculation of allowable bending stress (Fb). If this entry is left blank, it will be calculated automatically for ASD code checks. 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 leave the calculations to be done internally. Note that the same Cb factor is used for all the interaction equations in Chapter H. The special provisions for Cb which apply to equation H1-1, in Chapter F.3 of the ASD, are therefore conservatively applied to the Chapter H interaction equations.

For LRFD code checking, if Cb is left blank, a value of 1.0 will be used (Cb is not calculated automatically for LRFD at this time). This is a conservative assumption.

AISC Steel Design Code: Sway Flags

These 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 that the member is subject to sway for the particular direction, or leave the entry blank if the member is braced against sway. These sway flags influence the calculation of the K values and also the Cm and Cb factors (for ASD).

AISC Steel Design Code: Bending Coefficients (Bending Coefficients)

Tapered Wide Flanges - For Tapered WF members, the Cb field is used for the “B” value. B is described in Appendix F7.4 of the ASD and LRFD codes. The Cb term used in the Chapter F equations is always calculated internally for Tapered WF members and will be shown on the Member Detail report and in the Code check spreadsheet. If the Cb 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 automatically.)

Wind/Seismic Increase

If the WS field is checked in the Load Combinations spreadsheet for the load combination being solved, the ASIF factor entered on the Global window is applied to the allowable stresses for ASD code checking, in accordance with section A5 of the ASD code. The default for the ASIF factor is 1.333, representing a 1/3 increase. The ASIF factor also is applied to the Euler values (F'e). This "Wind/Seismic" increase in allowable stresses is allowed because these forces are considered to be transient.

Even if the W/S increase is being used, the final code check value should still be compared to 1.0. This ASIF factor is brought into the code check calculations directly.

For LRFD code checking, setting the W/S flag will cause the use of the seismic provisions of Table 8-1 when checking the compactness of WF shapes. The ASIF factor is not used in any other way with LRFD code checking.
AISC STEEL CODE CHECK RESULTS

Access the Steel Code Check spreadsheet by selecting the Results menu and then selecting Members ▶ Steel Code Checks.

The location of the steel design code checks is based on the Number of Sections specified on the Global Parameters window. The maximum of these section calculations is listed, along with the location where the maximum occurred.

The final result of the code checking is a code check value. This value represents a factored ratio of actual to allowable stresses for ASD or ultimate load to design strength for LRFD, based on the provisions of Chapter H. So, if this code check value is less than 1.0, the member passes. If it is greater than 1.0, the member fails. The code checks are summarized in the Code Check spreadsheet. If a code check is greater than 9.999, it will be listed here as "9.999". The Loc field tells at what location the maximum code check occurs measured from the I-node location of the member. You may also view a Detail Report for each member or view the code checks graphically.

The Shear Check is the maximum ratio of actual to allowable shear stress or, for LRFD, force. The location for the shear check is followed by "y" or "z" to indicate the direction of the shear.

If you are doing ASD code checking, Fa, Ft, Fbyy and Fbzz are the allowable stresses calculated for the member. Fa is calculated according to the provisions of AISC 9th Edition, Chapter E. Ft is based on Chapter D. The Fb values are calculated based on Chapter F. If necessary, RISA-3D will consider the provisions of Appendix B5. Note that for RISA-3D, "zz" corresponds to "xx" in the AISC code, i.e. RISA-3D substitutes Fbzz for Fbx, to maintain consistency with the member local axis system.

For LRFD 2nd Edition, the factored compression Phi*Pnc, factored tension, Phi*Pnt, and factored moment strengths Phi*Mnyy and Phi*Mnzz values are displayed. For tension Pnt, the value is fy * area, per chapter D. Compression Pn is calculated per Chapter E and Appendix B. The Mn values are calculated per Chapter F and Appendix B. 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) LRFD code checks won't be done. All square or rectangular tube shapes (HSS) will designed per the HSS Specification.

The Cb coefficient is calculated based on the description presented in Chapter F, Section F1, Subsection 3 (page 5-47) for ASD checks. 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. For LRFD, Cb is set to 1.0 if not specifically entered by the user, which is conservative. LRFD Eqn. F1-3 is currently not implemented. The Cm coefficients, used only for ASD checks, described in Chapter H, Section H1 (page 5-55) 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 Chapter H (for ASD and LRFD) or section 7 for the HSS code.
Note that the requirements of Section H2 of the LRFD code are satisfied since RISA-3D calculates and includes torsional warping effects. Appendix H is not considered.

For enveloped results the combination that produced the listed code 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. To include a particular Load Combination in the envelope analysis, open the Load Combinations spreadsheet and check the box in the "Env" column.

**Tapered Wide Flanges – ASD** - 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 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**
- For intermediate values used to arrive at the code check values see the Member Detail Report.
- To view the results for a particular member, use the Find option. To view the maximums and minimums, use the Sort option.
- These results may be displayed graphically with the Plot Options dialog.

**General Limitation**

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

**ASD Limitations**

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. Single angles are only checked for axial loading, flexural effects are not considered in the code check calculation.

**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.
LRFD LIMITATIONS

LRFD 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. Single angles are only checked for axial loading, flexural effects are not considered in the code check calculation. Single angles are only checked for Euler buckling. They are not checked for Flexural-Torsional buckling. 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 as taken as 16.5ksi, as for a welded shape.

TAPERED MEMBER LIMITATIONS

ASD 9th edition code checks can be done on a tapered member which can have 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. 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 the Member section for more information on the calculation of single angle stresses.

SPECIAL MESSAGES

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:

AISC Code Check Not Calculated
This is that general message displayed when code checks were not performed for a member. This could be because the member is not defined with a database shape, code checking is turned off, consistent units were specified, etc. Also, for LRFD code checking, if the steel yield stress is less than 10.ksi, you’ll get this message.

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 buckling 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.
Steel Design - Optimization

**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.

**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 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.
CANADIAN AND LRFD STEEL CODE CROSS REFERENCE

This is a partial cross-reference between the Canadian CAN/CSA S16.1-94 steel code and the equivalent section in the AISC LRFD 2nd edition code.

<table>
<thead>
<tr>
<th>AISC LRFD 2nd</th>
<th>CAN/CSA S16.1-94</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>7</td>
<td>Loads and Load Combinations</td>
</tr>
<tr>
<td>B5</td>
<td>11</td>
<td>Width to Thickness Ratios</td>
</tr>
<tr>
<td>B7</td>
<td>10</td>
<td>Maximum Slenderness Ratios</td>
</tr>
<tr>
<td>C</td>
<td>8.6</td>
<td>Frame Stability / 2nd Order Effects</td>
</tr>
<tr>
<td>Chapter C Commentary</td>
<td>Appendix B</td>
<td>K Factors for Compression Members</td>
</tr>
<tr>
<td>D</td>
<td>13.2</td>
<td>Tension Strength</td>
</tr>
<tr>
<td>E</td>
<td>13.3</td>
<td>Compression Strength Double Symmetry.</td>
</tr>
<tr>
<td>F1</td>
<td>13.5/13.6</td>
<td>Moment Strength</td>
</tr>
<tr>
<td>F2</td>
<td>13.4</td>
<td>Shear Strength</td>
</tr>
<tr>
<td>H1</td>
<td>13.8/13.9</td>
<td>Combined Axial and Bending</td>
</tr>
<tr>
<td>H2</td>
<td>15.11</td>
<td>Torsion</td>
</tr>
<tr>
<td>G2</td>
<td>15.4</td>
<td>Plate Girders / This Webs</td>
</tr>
<tr>
<td>Appendix E</td>
<td>Appendix D</td>
<td>Compression Strength Single Symmetry.</td>
</tr>
</tbody>
</table>

CANADIAN STEEL CODE DESIGN PARAMETERS

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 NDS wood parameters.

Unbraced Lengths

The unbraced lengths are Lbyy, Lbzz and Lcomp. The Lb values represent the unbraced length for the member with respect to bending about its local y and z axes, respectively. These Lb values are used to calculate KL/r ratios for both
directions, which in turn impacts the calculation of $C_t$, the axial compressive strength. The $KL/r$ ratios gauge the vulnerability of the member to buckling. Refer to Section 13.3 and Section 13.8.1 in the CAN/CSA S16.1-94 code for more information on this. Also, Section 10.2 lists the limiting slenderness ratios.

The $L_{comp}$ value is the unbraced length of the compression flange. This may be the same as the $L_{byy}$ value, but not necessarily. The $L_{comp}$ value is used in the calculation of $M_{r}$, the strong axis flexural strength. Refer to Section 13.5 and 13.6 of the Canadian code for more information on this.

These unbraced lengths all default to the member's full length if left blank, except if $L_{byy}$ is entered and $L_{comp}$ is left blank, $L_{comp}$ will default to the entered value for $L_{byy}$. Note that there is currently only one $L_{comp}$ value used for both the top and bottom flanges. For many framing conditions, the moment in a beam will reverse, and for the same member, the top and bottom flanges will be in compression for different portions of the member span. A future enhancement to RISA-3D will be the addition of another $L_{comp}$ field so that the top and bottom unbraced lengths can be specified separately.

**K Factors (Effective Length Factors)**

The "K" factors are also referred to as effective length factors. $K_{yy}$ is for bending about the member’s local y-y axis and $K_{zz}$ is for bending about the local z-z axis. The $K$-values, if not entered (left blank), will default to 1.0. See Appendix B and Appendix C in the CAN/CSA S16.1-94 code for more information on how to calculate $K$ factors.

RISA-3D is able to approximate the $K$ values for a member, based on that member's sway condition and end release configuration. The $K$-factor approximation is based on Figure B1 in Appendix B of the Canadian code. Following are the values used for various conditions:

<table>
<thead>
<tr>
<th>Table Case</th>
<th>End Conditions</th>
<th>Sidesway ?</th>
<th>K-Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>(a)</td>
<td>Fixed-Fixed</td>
<td>No</td>
<td>.65</td>
</tr>
<tr>
<td>(b)</td>
<td>Fixed-Pinned</td>
<td>No</td>
<td>.80</td>
</tr>
<tr>
<td>(c)</td>
<td>Fixed-Fixed</td>
<td>Yes</td>
<td>1.2</td>
</tr>
<tr>
<td>(d)</td>
<td>Pinned-Pinned</td>
<td>No</td>
<td>1.0</td>
</tr>
<tr>
<td>(e)</td>
<td>Fixed-Free</td>
<td>Yes</td>
<td>2.0</td>
</tr>
<tr>
<td>(f)</td>
<td>Pinned-Fixed</td>
<td>Yes</td>
<td>2.0</td>
</tr>
</tbody>
</table>
Any configuration not described here would get 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 by directly entering the value that you want in the appropriate field. If you have RISA-3D perform another approximation the manually input values will be overwritten.

**Limitations**

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. 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** - Tapered WF shapes will be designed per the AISC LRFD code. The Kyy factor for Tapered WF shapes can be approximated by RISA-3D, and is the same as that 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 AISC LRFD 2nd Commentary on Appendix F for an explanation of how to calculate the Kzz factor for Tapered members.

**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 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-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 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 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.)

**Canadian Steel Design Code: Sway Flags**

These 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 that the member is subject to sway for the particular direction, or leave the entry blank if the member is braced against sway. These sway flags influence the calculation of the K values and also the w1 and w2 coefficients.

**Canadian Steel Code Check Results**

Access the Steel Code Check spreadsheet by selecting the Results menu and then selecting Members → Steel Code Checks.

The steel design code checks are calculated based on the roughly 40 sections used to check the section. (We say “roughly” because the number of internal sections varies slightly depending on the number of displayed sections chosen on the Global window.)

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-node 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 also are 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. To include a particular Load Combination in the envelope analysis, open the Load Combinations spreadsheet and check the box in the "Env" column.

**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**
- For intermediate values used to arrive at the unity check values see the Member Detail Report.
- To view the results for a particular member, use the Find option. To view the maximums and minimums, use the Sort option.
- These results may be displayed graphically with the Plot Options dialog.

**Canadian Steel Design Code: Limitations**

It is assumed the axial load on the member is occurring through the member’s shear center. This means local secondary moments that may occur if the axial 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 Double angle Limitations
The criteria in the AISC LRFD 2nd code since the Canadian code does not explicitly specify how to calculate the flexural strength of WT and LL shapes.

Tapered WF Member Limitations
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 browser spreadsheet to remind the user of the tension only code check.

Single Angle code check based on Axial Tension ONLY
Please see the Member section for more information on the calculation of single angle stresses.

Canadian Steel Design Code: Special Messages
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: You may also want to look at the AISC special messages

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.

**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 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 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.

**MODIFYING KYY**
Kyy is for bending about the member’s local y-y axis. The K-value, if not assigned, will default to 1.0. See the CAN/CSA S16.1-94 code, Appendix B and C, for an explanation of how to calculate the K values for steel members. The K values for a member may be approximated based on that member’s sway condition and end release configuration.

---

**Steel Design Code - Optimization**
Redesign is performed on standard steel shapes that are listed on the Redesign spreadsheet. Alternate shapes are suggested for each designated section set based on certain redesign parameters.

You can type in the name of the section set in the Section Set field. A quick way to list ALL the section sets in one step is to click the All button. This feature will cause all the section sets listed on the Sections spreadsheet to be listed on the Redesign spreadsheet.

The parameters entered for each section set on the Redesign spreadsheet control which shapes will be considered as alternates. Only shapes of the same type and database as the original shape will be considered as replacements. For example, if a section set is defined initially with an AISC Tube shape, only AISC Tubes will be suggested as alternates. Further, if height and width limits are defined, only shapes falling within those limits will be considered.
If any parameters are left blank, that particular limit will not be considered. For example, if a minimum height of 8" is specified but no maximum is specified, then all shapes with a height greater than (or equal to) 8" will be considered acceptable.

Alternate shapes are selected as follows: All the members assigned the particular 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 forces on these members are then applied to new shapes satisfying the controlling parameters and a code check is calculated. If the calculated code check and shear check falls within the range specified on the Redesign spreadsheet (if a range is specified), the shape is considered to be an acceptable alternate.

After all the applicable database shapes are examined, the four lightest are listed as suggested alternates. If less than four acceptable shapes are found, as many as were found will be listed. If NO acceptable shapes are found, a message to that effect will be listed. If you see the "*" symbol next to the section set, it means the 1st choice alternate is already assigned to the section set, so redesign for that section set is complete.

Note that all the currently loaded databases will be used to consider shapes as design alternates. If you do not want alternate shapes to be recommend from a certain shape database, make sure this database is not loaded in the configuration file.

To Optimize Member Sizes

1. On the Global window specify the Steel Design Code and check Do Redesign.
2. To control which members may be considered specify the parameters on the Redesign spreadsheet.
3. After running the analysis you may view the suggested shapes on the Alternate Shapes spreadsheet.

Note
- For optimization to be performed on a member, the member must be defined with a database shape on the Sections spreadsheet. Only similar shapes will be suggested as alternates.

Viewing Shape Optimization Results

Access the Alternate Shapes spreadsheet by selecting the Results menu and then selecting Members ▶ Alternate Shapes.

These are the suggested alternate shapes resulting from redesign calculations. The shapes are listed in order of weight, lightest first. Up to 4 shapes will be listed for each section set. The controlling member is listed in the second column.

Keep in mind that the current results are based on whatever shapes are currently assigned to the section sets. Changing the sections will change the stiffness matrix, which then needs to be resolved to be sure that the shapes are acceptable. To confirm that these alternate shapes are acceptable, you MUST go back to the
Sections spreadsheet and enter them for the appropriate section sets. Then re-solve the problem and check the results. It may be necessary to cycle through this process a few times until the suggested 1st choice alternate shapes match the shapes actually used.

The symbol "*" is displayed before the shape if the first choice shape is the same as the shape currently assigned to the section set. In other words, the "*" symbol means the section set redesign is complete, you already are using the optimum shape for that particular section set.

If the message "No acceptable shapes found..." is listed, no shapes could be found for the section set that satisfy the redesign criteria defined on the Redesign spreadsheet. If you’ve entered a minimum code check value and the members assigned this section set are lightly loaded, it is possible that no shape generates a code check value high enough to exceed the minimum, resulting in this message. Also, if the section set has not been assigned an initial database shape, no redesign can be done. Shapes entered via on-line shape definitions (RE, PI and BAR) cannot be redesigned.

If the message "Redesign not performed..." is listed, one or more of these conditions are true:

1) A steel code was not specified on the Global window.
2) The flag "Do Redesign" on the Global window is not checked.
3) No section sets are entered on the Redesign spreadsheet.

Note
- You may try the new shapes by clicking the Refill and Rerun button.
- There will be no alternate shapes if you have no sections sets listed on the Redesign spreadsheet.

Surface Loads

Surface loads are the loads that are spread out over the surface of a plate element. RISA-3D allows surface loads directed in the global axes, the local axes, or projected in the direction of the global axes. Loads may be input manually or assigned graphically.

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 Apply 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** dialog.
- You may also specify or edit surface loads in the **Surface Load Spreadsheet**.
- Applied surface loads are converted to equivalent corner nodal loads based on the plate area tributary to each corner node.

**Surface Load Spreadsheet**

The Surface Load Spreadsheet records the surface loads for the plate/shell elements and may be accessed by selecting **Plates** on the **Spreadsheets** menu.

A surface load is defined by the label of the plate to be loaded, a direction, and a magnitude. The first column contains the Plate Label, the second column contains the direction of application, and the third contains the magnitude of the load. The load direction may be in the local plate (x, y, z) axes, global (X,Y,Z) axes, as well as projected surface loads (L,V,H) in the global axes.

The direction code indicates the direction of application for the surface load. Following are the valid choices:

<table>
<thead>
<tr>
<th>Direction Code</th>
<th>Applied Direction</th>
</tr>
</thead>
<tbody>
<tr>
<td>x, y, or z</td>
<td>Element's local x, y, or z direction.</td>
</tr>
<tr>
<td>X, Y, or Z</td>
<td>Global X, Y, or Z direction</td>
</tr>
<tr>
<td>V</td>
<td>Projected in the direction of the global Y axis</td>
</tr>
<tr>
<td>L</td>
<td>Projected in the direction of the global X axis</td>
</tr>
<tr>
<td>H</td>
<td>Projected in the direction of the global Z axis</td>
</tr>
</tbody>
</table>

**Surface Load Options**
This 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°, 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.

Loading options for the plate elements are currently limited to uniform surface loads over the entire surface of the plate. Other plate loading options such as non-uniform and partial surface loads will be added in future program releases.

**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. This support is primarily for help with installation of the program and answering any specific questions about the use, capabilities and solution methods of RISA-3D. This service is not to be used as a way to avoid learning the program, or learning how to do structural modeling in general.

**Hours:** 9AM 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 Reference Manual!** Most questions we get asked about RISA-3D are already answered. Use the index to find specific topics and the appropriate section(s). 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 tutorial when you first get the program.

2.) If you have access to the Internet, you can go to our Web page at WWW.RISATECH.COM and check out our technical support FAQ (Frequently Asked Questions) section. Your question may already be answered there. The FAQ is on our Technical Support page, which is found under “Product Support”. We also post our latest bug reports on the web site under "Product Support".

3.) Make sure you understand the problem, and make sure your question is related to the program or structural modeling. Technical support does not provide free engineering consulting. RISA Technologies does provide a structural modeling service. If you’re 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. We realize that your time is valuable, however our time is valuable also, and our experience has shown that many questions we get asked could be figured out by the user if a few minutes had been taken to experiment and think about the problem. This is especially true with regard to modeling difficulties. You are also more likely to remember the solution to the problem, should it arise again in the future, if you spend some time trying to solve it.

For all modeling support questions, please be prepared to send us your model-input file via e-mail (see below), or postal mail. We often will need to have your model in hand to debug a problem or answer your questions.
Fax Support: (949) 951-5848: This is, by far, the fastest way to reliably get answers to your questions. Most faxed questions are responded to in less than 1 hour after they are received. For a list of questions, or when a sketch would aid a problem description, this is the best way to send your request. Very small model input files can be faxed as well. Please indicate "URGENT" on your fax if you need assistance ASAP. Provided in the help file is a form you can print to FAX your questions to us. Please provide us with a FAX number for our response.

E-Mail: support@risatech.com: This method is second to the fax because, at this point, we only check our e-mail a few times a day. If you send an e-mail request and it's urgent, you should probably send a fax or call to let us know you've sent an e-mail. This method is the BEST way to send us a model you would like help with. Most e-mail packages support the attachment of files, or for small files, you can embed the input within the text of the message. The input file you would send will have a .R3D extension. Make sure you tell us your name, phone number, fax 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.

Snail(Postal) Mail to RISA Technologies: This method works fine as long as you can wait for the postal service. If you don't have e-mail, then this is the only way to send us the model that you need help with. Most people who are in a rush will send a floppy disk via overnight mail (e-mail is a lot cheaper and faster, though).

Phone Support: (949) 951-5815: This method is our least preferred method but if you need a quick answer give us a call.

RISA Technologies has a web site that we use to provide current bug reports on the latest production versions, beta program information, expected future enhancements, etc. The address for the site is "WWW.RISATECH.COM". Take some time and go check it out!
RISA-3D Fax Support Form
You are free to reproduce this page. Copyright restrictions are waived for this page.

Our FAX number: 949-951-5848

Date: ______________

User’s Name: ____________________________________________

Company Name: __________________________________________

Telephone: ______________________________________________

FAX Number: _____________________________________________

RISA-3D Serial Number: ________________________ (This is required!)

If you don’t know your serial number, start RISA-3D and select About in the Help menu.

Your Question: (Please attach additional sheets as needed.)

Our Reply: (Please don’t write below here)
Thermal Loads

You can model the effects of temperature differentials in members. These effects cause the axial expansion or contraction of the member along its length, i.e. axial stress only. The temperature is assumed constant across the member’s depth.

Note
- The internal axial deflections for beam members are the average of the end deflections for thermal loads.

Recording Thermal Loads

The nodal temperatures recorded on the Node 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. 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.

The Coefficient of Thermal Expansion (\(\alpha\)) is entered on the Materials spreadsheet. Note that this value is entered per 100,000 degrees (it is usually listed in tables per 1,000 degrees).

To Apply a Thermal Load

1. From the Spreadsheets menu select the Load Patterns Spreadsheet and define a pattern with a direction code of “T”.
2. Select the members you wish to assign a thermal load.
3. Click \(\mathbb{T}\) to turn on the Drawing Toolbar if it is not already displayed and click the Apply Distributed Load button.
4. Referencing the pattern defined in step 1, specify a magnitude, in temperature, for the load to be applied.
Note

- The Coefficient of Thermal Expansion ($\alpha$) is entered on the Materials spreadsheet. Note that this value is entered per 100,000 degrees (it is usually listed in tables per 1,000 degrees).
- You may also specify or edit thermal loads in the Distributed Load Spreadsheet.
- An easy way to do thermal loadings is to define the node temperatures (on the Joint Coordinates Spreadsheet) as all zero, so any defined thermal loads are the full stress inducing temperatures.
- The actual stress inducing temperature is this input thermal load minus the ambient temperature of the member.

THERMAL FORCE CALCULATION

The nodal temperatures recorded on the Node Coordinates Spreadsheet define the ambient thermal state of the structure. The node 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:

$$F_t = A \times E \times \alpha \times \Delta T$$

Where,

- $F_t =$ Calculated Thermal Force
- $A =$ Member Cross Sectional Area
- $E =$ Elastic Modulus
- $\alpha =$ Coefficient of Thermal Expansion
- $\Delta T =$ Stress Inducing Temperature

Note

- 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.

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 $\Delta T$. Remember, as the model expands (or contracts), the prestress force may be altered.

Note

- Often it is desirable to insert a member to receive the thermal loads, which will then prestress the adjacent member.
Timber Design

Full code checking can be performed on dimension lumber and Post and Timber size wood shapes, based on the NDS-1991 criteria. The wood database values are based on the supplement from the 1997 NDS specification.

The NDS code checking calculations encompass all the code requirements for standard beam/column shapes. The calculations are reported at each section for each member. You can specify from 2 to 10 sections per member on the Global dialog.

To Specify a Timber Section

1. On the Wood Parameters spreadsheet enter a Wood Label of your choice to name the shape in order to refer to it later.
2. In the second column assign wood section and material properties by specifying a NDS Database Shape for each of your wood sections.
3. On the Sections spreadsheet assign each of your wood sections to a section set, by entering your Wood Label in the Material field for each desired wood section. (One wood label for each section set).
4. On the Member spreadsheet assign these Section Sets to the members for which you want wood code checks performed by specifying the desired section set label in the section set field.

Timber Design Parameters

Parameters controlling the wood 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. Notice that the steel AISC parameters are the initial column labels. The secondary column labels are for the NDS wood parameters.

NDS Wood Properties Spreadsheet

If you want to define wood members for the Timber code-checking feature, you need to define the material and section set properties for these members on the Wood Parameters spreadsheet. Wood sections are unique because the NDS Database Shape on the Wood Parameters spreadsheet will define both the material and the section set properties. The wood section Label is then referenced on the Sections spreadsheet to allow the “As” factors to be entered as necessary, but no section set properties are actually assigned on the Sections spreadsheet.

If you're trying to analyze wood sections that are larger than 16", you will need to enter the properties here and define the section set properties on the Sections
Timber Design

spreadsheet. Note that you can only analyze members that you have defined in this way (no wood code checking).

The Wood Set field contains the label that you will use to identify your wood section on the Sections spreadsheet. Each label must be unique and you can have a maximum of 500 wood sets.

The NDS Species Database field is where you specify the species, grade, and size of your wood member. This data is entered in a particular format so that it can be read from our Wood Shape database.

The next 5 fields on the Wood Parameter spreadsheet are the allowable stresses and Young’s Modulus (E) for the wood section. These values are shown to confirm the design values that are used for that wood section. These values can be edited but those edits will be lost if your cursor passes through the field or if the input file is read back in.

The next two fields are check-mark fields that allow you to specify whether the Cm factor (Moisture Content / Wet Service) or the Cr factor (Repetitive Member) 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. 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”.

The next field is the E_Mod field, which is a factor that is applied to Young’s Modulus (E) to reflect the Appendix F criteria. The default for this field is 1.0.

The next 3 fields are Poisson’s Ratio, the Thermal Coefficient of Expansion, and the Weight Density. The default values are typical averages for wood in general and do not change by species. Poisson’s ratio is used for the shear deformation calculations for members. The thermal coefficient is used for any applied thermal loads. The weight density is used to calculate the member’s self weight. These values can all be edited for each wood section.

Timber Design Shear Check

Typically when shear code checks are performed for rectangular wood sections, the average shear stress is multiplied by 1.5 to reflect the higher peak shear stress that occurs in the middle of the section. This is NOT done automatically since this would limit the timber design to only rectangular shapes. To take account of peak shear stresses, you should modify the “As” factors on the Sections spreadsheet. For a rectangular shape, you would set the “As” factors to 1.5. For a round shape, you would set the “As” factors to 1.1.

Timber Shape Database

The Timber Shape database is based on the wood species information contained in the NDS Supplement (1997) in tables 4A, 4B, and 4D. You may choose the species, grade and section from the drop down lists. You may also type in the
designation directly in the **Wood Label** field on the NDS Wood Properties spreadsheet. The NDS database entry is structured thusly:

```plaintext
sssssggg_ttxww
```

The first 5 characters (ssss) are the Species Designation

The next 3 characters (ggg) are the Grade Designation

The next 1 character (_) is for spacing.

Last 5 char (ttxww) are the thickness times the width (nominal dimensions in inches) except for a round shape. For round shapes enter 'Rdddd', where ‘dddd’ is the actual diameter in inches.

For shapes not in the database use Woodsize.

**Note**

- 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.”

<table>
<thead>
<tr>
<th>Examples</th>
<th>Designation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Red Oak, 2x8, No.3</td>
<td>RDOAKNO3_2X8</td>
</tr>
<tr>
<td>Southern Pine, 2x10,Dense Select Structural</td>
<td>MIXSPDSS_2x10</td>
</tr>
<tr>
<td>White Oak,12x12, Select Struc (Post)</td>
<td>WTOAKSEL_12x12</td>
</tr>
<tr>
<td>White Oak,10x16, No.1 (Beam)</td>
<td>WTOAKNO1_10X16</td>
</tr>
<tr>
<td>Western Hemlock, 8.5&quot; (actual) Round, No.2</td>
<td>WEHEMNO2_R8.5</td>
</tr>
</tbody>
</table>

**TIMBER NON-DATABASE SHAPE**

To enter the wood properties (Fb, Fc, etc.) directly for a case where the species/grade is either unknown or not in the database, you will still need to use the database field to define the size. In this case use the Woodsize designation by choosing “None” from the Species and Grade lists and enter the dimensions. You may also type in the designation directly in the **Wood Label** field on the NDS Wood Properties spreadsheet. The Woodsize entry is structured thusly:

```plaintext
WOODSIZE_ttxww
```

Where 'ttXww' (thickness and width) is as previously defined.

For example a 2x4 of undetermined species and grade would be specified **WOODSIZE_2x4**

**Note**

- Enter the nominal dimensions (or round diameter) in inches, regardless of what units system you are using. RISA-3D will automatically adjust these 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.”
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 applied in accordance with Table 2.3.1, which is on page 5 of the NDS-1991 code. The following topics help to summarize how adjustment factors are obtained and used. The CV, CT, and Cb factors are NOT used.

Timber Design Unbraced Length

The unbraced lengths, \( L_{e2} \), \( L_{e1} \), and \( L_{\text{bend}} \), are defined on the NDS Parameters spreadsheet. The \( L_{e2} \) and \( L_{e1} \) values represent the unbraced length for the member with respect to bending about its local y and z axes, respectively. These \( L_e \) values are used to calculate \( L_{e1}/d \) and \( L_{e2}/b \), which in turn impact the calculation of \( C_p \), 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 \( L_{\text{bend}} \) value is the unbraced length 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 \( L_{\text{bend}} \) value is used in the calculation of the slenderness ratio, \( R_B \). \( R_B \) is used to calculate \( C_L \), which is the beam stability factor, which is then used to calculate the allowable bending stress. Refer to Section 3.3.3.6 in the NDS for more information on this. Note that the value of \( R_B \) is limited to 50.

These unbraced lengths all default to the member’s full length if left blank, except if \( L_{e2} \) is entered and \( L_{\text{bend}} \) is left blank. \( L_{\text{bend}} \) will default to the entered value for \( L_{e2} \). Since \( L_{e2} \) and \( L_{\text{bend}} \) are often different in wood design, it is likely you should enter the correct value for \( L_{\text{bend}} \).

Timber Design K Factors

The ”K” factors, also referred to as effective length factors, are entered on the NDS Parameters spreadsheet. \( K_{yy} \) is for bending about the member’s local y-y axis and \( K_{zz} \) is for bending about the local z-z axis. The K-values, if not entered (left blank), will default to 1.0. See the NDS Appendix G for how to calculate the
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 that member’s sway condition and end release configuration. The K-factor approximation is based on Table G1, found in Appendix G, on page 116 of the NDS-1991 code.

Following are the values used for various conditions:

<table>
<thead>
<tr>
<th>Table Case</th>
<th>End Conditions</th>
<th>Sidesway?</th>
<th>K-Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>(a)</td>
<td>Fixed-Fixed</td>
<td>No</td>
<td>.65</td>
</tr>
<tr>
<td>(b)</td>
<td>Fixed-Pinned</td>
<td>No</td>
<td>.80</td>
</tr>
<tr>
<td>(c)</td>
<td>Fixed-Fixed</td>
<td>Yes</td>
<td>1.2</td>
</tr>
<tr>
<td>(d)</td>
<td>Pinned-Pinned</td>
<td>No</td>
<td>1.0</td>
</tr>
<tr>
<td>(e)</td>
<td>Fixed-Free</td>
<td>Yes</td>
<td>2.1</td>
</tr>
<tr>
<td>(f)</td>
<td>Pinned-Fixed</td>
<td>Yes</td>
<td>2.4</td>
</tr>
</tbody>
</table>

Any configuration not described here would get 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 here, then change some end release designations, you should come back and redo the K approximations.

Remember that the K values are approximations, and you should check to make sure you agree with all the K-values that RISA-3D assigns. You can always override a K-value by directly entering the value that you want in the appropriate field. If you have RISA-3D perform another approximation the manually input values will be overwritten.

Limitations

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. 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.

Timber Design CM Factor

Checking the appropriate box on the NDS Wood Properties spreadsheet specifies the CM wet service factor. 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.

**Timber Design Cr Factor**
Checking the appropriate box on the NDS Wood Properties spreadsheet specifies the Cr repetitive member factor. 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”.

**Timber Design CH Factor**
The CH entry shares the same field as the Cm-yy entry on the Member Design spreadsheet. For wood sections, this entry will be interpreted as CH. CH is the shear stress adjustment factor. If this entry is left blank, the default used will be 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 applied to the allowable shear stress Fv.

**Timber Design Sway Flags**
These 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. These are "yes/no" flags, so enter a "y" if the member is subject to sway for the particular direction, or leave the entry blank if the member is braced against sway. These sway flags influence the calculation of the K values.

**Timber Design Temperature Factor**
The temperature factor (Ct) is calculated internally from the nodal temperatures that are assigned on the Node Coordinates spreadsheet. See section 2.3.4 for more information on the temperature factor.

**Timber Design Flat Use Factor**
The flat use factor (Cfu) is 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 table 2.3.1 and the footnotes.

**Timber Design Size Factor**
The size factor (CF) is applied automatically when you assign a wood shape from the NDS shape database. See tables 4A, 4B, and 4D in the NDS supplement for information on the CF factor.

**Timber Design Form Factor**
The form factor (Cf) is applied automatically when you specify a Round shape from the NDS shape database. See section 2.3.8 in the NDS for more information
on the Cf factor. Note that 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 Beam/Column Stability Factor**
The CP factor (column stability) and the CL factor (beam stability) are calculated internally. These calculated values are shown on the Wood Code Checks spreadsheet, as well as on the Member Detail report. See NDS section 3.3.3.8 for more on the CL factor. See NDS section 3.7.1 for more on the CP factor.

**Timber Design Load Duration Factor**
The CD factor is entered on the Load Combination spreadsheet for each load combination that you want wood code check results. The CD factor must be entered for each load combination since the CD factor will change based on the types of loads that are applied in each load combination. Different load combinations would have different CD factors. For example, per the NDS ‘91 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. See Table 2.3.2 in the NDS-1991 code for the CD factors to be applied for typical loads. Appendix B has additional information about the Load Duration factor. Note that the CD factor used for a load combination should be for the load with the shortest load duration in that load combination.

**Timber Design Results**
The final result of the code checking is a code check value. This value represents a factored ratio of actual to allowable stresses based on the equations in section 3.9.1 and 3.9.2 of the NDS-1991 code. So, if this code check value is less than 1.0, the member passes. If it is greater than 1.0, the member fails. The code checks are summarized in the Code Check spreadsheet. You may also view a Detail Report for each member or view the code checks graphically.

**Timber Code Check Results**
Access the Wood Code Check spreadsheet by selecting the Results menu and then selecting Members » Wood Code Checks.

These are the code checks for the timber members per the requirements of the 1991 NDS Specifications. The final code check value represents a ratio represents a ratio of actual stress to allowable stress, thus a value greater than unity (1.0) means the member fails to satisfy the NDS criteria. Any code check that exceeds 9.999 will be listed here as 9.999. The "loc" field reports the location the maximum code check value occurs. 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 controlling shear.

The next values (Fc', Ft', Fb1', Fb2'; Fv') are the factored allowable stresses. The unfactored allowable stresses are listed on the Wood 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 1991 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 (nonconservative) 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 listed are the values calculated for the controlling load combination. They are not necessarily the maximums across all the combinations. To include a particular Load Combination in the envelope analysis, open the Load Combinations spreadsheet and check the box in the “Env” column.

Note
- For other values used to arrive at the code check values see the Member Detail Report.
- To view the results for a particular member, use the Find option. To view the maximums and minimums, use the Sort option.
- These results may be displayed graphically with the Plot Options dialog.

**Timber Design Member Detail**

The member detail report displays most of the intermediate values used to calculate the code check value.

Some of the code check information displayed on the detail report is available on the Wood Code Check spreadsheet. The detail report additionally shows the values for all the adjustment factors used to modify the allowable stresses, as well as the unbraced lengths used, and the sway conditions.

**Timber Design Special Messages**

In the instances where code checks are not performed 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:
**RISA-3D General Reference**

*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 code 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 code 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 code 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 code 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 code limits the actual strong axis bending compressive stress to be less than the term FbE. This term is approximately the lateral buckling stress.

**Timber Design Limitation**
It is assumed that the axial load on the member is occurring through the member’s shear center. This means local secondary moments that may occur if the axial load is not applied through the shear center are not considered.
### Timber Design Species

<table>
<thead>
<tr>
<th>Species</th>
<th>Designation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Aspen</td>
<td>ASPEN</td>
</tr>
<tr>
<td>Balsam Fir</td>
<td>BALFR</td>
</tr>
<tr>
<td>Beech-Birch-Hickory</td>
<td>BEECH</td>
</tr>
<tr>
<td>Coast Sitka Spruce</td>
<td>COAST</td>
</tr>
<tr>
<td>Cottonwood</td>
<td>COTON</td>
</tr>
<tr>
<td>Douglas Fir-Larch</td>
<td>DFLAR</td>
</tr>
<tr>
<td>Douglas Fir-Larch (North)</td>
<td>DFLAN</td>
</tr>
<tr>
<td>Douglas Fir- South</td>
<td>DFLAS</td>
</tr>
<tr>
<td>Eastern Hemlock</td>
<td>EHEML</td>
</tr>
<tr>
<td>Eastern Hemlock-Tamarack</td>
<td>EHTAM</td>
</tr>
<tr>
<td>Eastern Hemlock-Tamarack (N)</td>
<td>EHTNO</td>
</tr>
<tr>
<td>Eastern Softwoods</td>
<td>ESOFT</td>
</tr>
<tr>
<td>Eastern Spruce</td>
<td>ESPRU</td>
</tr>
<tr>
<td>Eastern White Pine</td>
<td>EWPIN</td>
</tr>
<tr>
<td>Hem-Fir</td>
<td>HEMFR</td>
</tr>
<tr>
<td>Hem-Fir (North)</td>
<td>HFNOR</td>
</tr>
<tr>
<td>Mixed Maple</td>
<td>MXMAP</td>
</tr>
<tr>
<td>Mixed Oak</td>
<td>MXOAK</td>
</tr>
<tr>
<td>Mixed Southern Pine</td>
<td>MIXSP</td>
</tr>
<tr>
<td>Mountain Hemlock</td>
<td>MOHEM</td>
</tr>
<tr>
<td>Northern Pine</td>
<td>NOPIN</td>
</tr>
<tr>
<td>Northern Red Oak</td>
<td>NROAK</td>
</tr>
<tr>
<td>Northern Species</td>
<td>NOSPE</td>
</tr>
<tr>
<td>Northern White Cedar</td>
<td>NWCED</td>
</tr>
<tr>
<td>Ponderosa Pine</td>
<td>PONPI</td>
</tr>
<tr>
<td>Red Maple</td>
<td>RDMAP</td>
</tr>
<tr>
<td>Red Oak</td>
<td>RDOAK</td>
</tr>
<tr>
<td>Red Pine</td>
<td>RDPIN</td>
</tr>
<tr>
<td>Redwood</td>
<td>REDWD</td>
</tr>
<tr>
<td>Sitka Spruce</td>
<td>SITKA</td>
</tr>
<tr>
<td>Southern Pine</td>
<td>SOPIN</td>
</tr>
<tr>
<td>Spruce-Pine-fir</td>
<td>SPFIR</td>
</tr>
<tr>
<td>Spruce-Pine-Fir (South)</td>
<td>SPFISO</td>
</tr>
<tr>
<td>Western Cedars</td>
<td>WECED</td>
</tr>
<tr>
<td>Western Cedars (North)</td>
<td>WCEDN</td>
</tr>
<tr>
<td>Western Hemlock</td>
<td>WHEHEM</td>
</tr>
<tr>
<td>Western Hemlock (North)</td>
<td>WHEMN</td>
</tr>
<tr>
<td>Western White Pine</td>
<td>WWPIN</td>
</tr>
<tr>
<td>Western Woods</td>
<td>WEWOD</td>
</tr>
<tr>
<td>White Oak</td>
<td>WTOAK</td>
</tr>
<tr>
<td>Yellow Poplar</td>
<td>YLPOP</td>
</tr>
</tbody>
</table>
TIMBER DESIGN GRADES

<table>
<thead>
<tr>
<th>Grade</th>
<th>Designation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Select Structural</td>
<td>SEL</td>
</tr>
<tr>
<td>No.1</td>
<td>NO1</td>
</tr>
<tr>
<td>No.2</td>
<td>NO2</td>
</tr>
<tr>
<td>No.3</td>
<td>NO3</td>
</tr>
<tr>
<td>Stud</td>
<td>STU</td>
</tr>
<tr>
<td>Construction</td>
<td>CON</td>
</tr>
<tr>
<td>Standard</td>
<td>STD</td>
</tr>
<tr>
<td>Utility</td>
<td>UTI</td>
</tr>
<tr>
<td>Dense Select Structural</td>
<td>DSS</td>
</tr>
<tr>
<td>Non-Dense Select Structural</td>
<td>NDS</td>
</tr>
<tr>
<td>No.1 Dense</td>
<td>N1D</td>
</tr>
<tr>
<td>No.1 Non-Dense</td>
<td>N1N</td>
</tr>
<tr>
<td>No.2 Dense</td>
<td>N2D</td>
</tr>
<tr>
<td>No.2 Non-Dense</td>
<td>N2N</td>
</tr>
<tr>
<td>Dense Structural D86</td>
<td>D86</td>
</tr>
<tr>
<td>Dense Structural D72</td>
<td>D72</td>
</tr>
<tr>
<td>Dense Structural D65</td>
<td>D65</td>
</tr>
<tr>
<td>No.1 &amp; Better</td>
<td>N1B</td>
</tr>
<tr>
<td>Clear Structural</td>
<td>CST</td>
</tr>
<tr>
<td>Select Structural, Open Grain</td>
<td>SEO</td>
</tr>
<tr>
<td>No.1, Open Grain</td>
<td>OG1</td>
</tr>
<tr>
<td>No.2, Open Grain</td>
<td>OG2</td>
</tr>
<tr>
<td>No.3, Open Grain</td>
<td>OG3</td>
</tr>
</tbody>
</table>

Note
- MSR and MEL (mechanically graded) lumber are not included in the database. If you need to use mechanically graded lumber, use the WOODSIZE code as described below.

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.

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:

\[ k = G \times J / L \]

\( G \) = 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:

\[ k = G \times J / \left( A \times \left( \tanh(L/2A) \times \cosh(L/A) - \tanh(L/2A) + L/A - \sinh(L/A) \right) \right) \]

\( G, J \) = as above
\( A = \sqrt{\left( \frac{E \times Cw}{J \times G} \right)} \)
\( Cw = \) 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 torsionally. 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 (it’s stiffer torsionally).

**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.

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.

**Torque Length**
The "torque length" is the length between points of torsional restraint (or release).
This may be equal to or greater than the member’s actual node to node length.
This torque length is calculated automatically by RISA-3D and is used for the
member’s torsional stiffness and stress calculations. Note that since this value
must be calculated independently for each warping member, the solution will be
slower (versus non-warping members), though probably not significantly. Each
member’s torque length is shown on the member detail report (see the section
Detail Report). Note that the calculation of the torque length by RISA-3D can be
“fooled” by curved beams that are modeled by several straight-line segments. The
torque length used for each member will be the length of each straight-line
segment.

**Torsional Warping**
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 window, 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.

**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:

$$\tau = \frac{Mx \times t}{J}$$

$$Mx = \text{Torsion Moment}$$
\[ t = \text{Maximum Thickness of Any Part of the Cross Section} \]

This is the only torsional stress calculated for non-warping shapes. Shapes 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 stresses 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.

**Applied Torsional Loads**

You’ll notice when you’re defining member loads you are not able to input concentrated or distributed torques along the member span. All torques have to be applied as joint loads. The reason for this is that we (RISA Technologies) have not yet found the time to work out the derivatives necessary to properly handle these member torque loads when applied to warping members. The calculations for non-warping members are quite simple but for warping members they’re complex. This may be added in a future version. For warping members, if you try to “fake” a point load by inserting a node within the span of a member and then put a nodal torque at that node, you will NOT get the same results as a point torque. The results you get with a nodal “point torque” will produce conservative results for the rotation, however the warping normal stresses will be unconservative near the quarter points.

**Member Torsion Results**

Access the **Member Torsion Stresses** spreadsheet by selecting the **Results** menu and then selecting **Members ‣ Torsion**.

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** window. The actual number of segments is this number of sections minus 1. The incremental length of each segment is the same. For example, if you specify 5 sections, the member is divided into 4 equal pieces, and the torsional stresses are reported at each section location.

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, tension is negative. Please refer to the Torsion section 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. To include a particular Load Combination in the envelope analysis, open the Load Combinations spreadsheet and check the box in the “Env” column.

Note
- To view the results for a particular member, use the Find option. To view the maximums and minimums, use the Sort option.
- These results may be displayed graphically with the Plot Options dialog.

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 and with the plot of values in the model view.

To Change Units
1 Click on the RISA toolbar.
2 Specify the units you want for each item in the drop down list boxes.

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**.

**Note**
- You may also specify that the units are to be the default units by checking the **Save as default** box.

**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 set property entries such as area and moment of inertia. Force and weight units are Kips, where 1 Kip = 1000 pounds. Stress units are Ksi (Kips per square inch).

**Standard Metric Units**
This units system uses meters for location entries and centimeters for property entries. Force units are 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 kilograms and thermal units are degrees centigrade.

**Consistent Units**
This option means no units considerations are made. It is the program user’s responsibility to insure that all the data is consistent in terms of forces, lengths, densities, etc. For example, if you wish to use inches and pounds, all entries concerned with length units (joint coordinates, section areas, etc.) would be entered in inches, and all force entries (joint forces, weight densities, etc.) would be entered using pounds.

**Note**
- Steel and Timber design is not available when consistent units are used.
UNIT SPECIFICATIONS

The following are the unit specifications and their applications:

<table>
<thead>
<tr>
<th>Measurement</th>
<th>Usage</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lengths</td>
<td>Coordinates, Unbraced Lengths, Load Locations</td>
</tr>
<tr>
<td>Dimensions</td>
<td>Section Set Properties, Plate Thickness, Member Offsets</td>
</tr>
<tr>
<td>Material Strengths</td>
<td>E, Fy, G</td>
</tr>
<tr>
<td>Weight Densities</td>
<td>Material Density</td>
</tr>
<tr>
<td>Forces</td>
<td>Loads, Forces</td>
</tr>
<tr>
<td>Linear Forces</td>
<td>Distributed Loads</td>
</tr>
<tr>
<td>Moments</td>
<td>Loads, Forces</td>
</tr>
<tr>
<td>Surface Pressures</td>
<td>Plate/Shell Surface Loads</td>
</tr>
<tr>
<td>Translational Springs</td>
<td>X Y Z Boundary Conditions</td>
</tr>
<tr>
<td>Rotational Springs</td>
<td>X Rot, Y Rot, Z Rot Boundary Conditions</td>
</tr>
<tr>
<td>Temperatures</td>
<td>Thermal Coefficient, Temperatures</td>
</tr>
<tr>
<td>Deflections</td>
<td>Deflections, Displacements</td>
</tr>
<tr>
<td>Stresses</td>
<td>Allowable and Actual Stresses</td>
</tr>
</tbody>
</table>

STAAD File Translation

STAAD files may be imported into RISA-3D by choosing File-Open 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.) These files are translated automatically when read into the RISA-3D program.

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 thicknesses 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 (i.e. unbraced lengths, K factors, etc.)
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.

**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 box 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 Global window 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-3D 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 the Stability section 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.

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.

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.

STAAD Unsupported Features for File Translation

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-node’s cause joints to be created at the K-node 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.

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 tutorial 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.

If you’re not sure where to start, there is a help topic called How to Build and Solve a Model that can help prompt you if you forget the basic minimum input that is needed to have a viable model.

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.

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 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. We do not encourage directly editing the *.R3D data file as it’s format has been designed for speed in reading and writing and not for ease of user editing. 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". Using Groups in STAAD is optional, whereas using Section Sets in RISA-3D is not. 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. You can see the Load Combinations topic on how to nest Load Combinations and get more than 8 BLC’s per Load Combination.

STAAD allows you to solve multiple load combinations in "batch" mode and then review all the results from all solved combinations. RISA-3D will currently let you review one load combination at a time or envelope multiple combinations. The ability to review multiple load combinations at one time will be added in a near future release.

**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.

**Results**

STAAD allows you to store your results and retrieve them later. RISA-3D will add this feature to a near future release. Note that RISA-3D’s results can be dynamically sorted and separated by graphical view, which makes it a little harder for us to maintain a results file. STAAD’s results are essentially static. RISA-3D will allow you save your dynamic results for future use, allow you still have to run a static solution to review any of the analytical results besides the frequencies and mode shapes.

**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 that you want it to act.