

RISA-3D

Rapid Interactive Structural Analysis

Version 22- General Reference



RISA Tech, Inc.

27442 Portola Pkwy, Suite 200

Foothill Ranch, California 92610

(949) 951-5815

(949) 951-5848 (FAX)

risa.com

Copyright

Copyright © 2024 RISA Tech, Inc. All Rights Reserved. RISA is part of the [Nemetschek Group](#). No portion of the contents of this publication may be reproduced or transmitted in any means without the express written permission of RISA Tech, Inc. RISA and the RISA logo are registered trademarks of RISA Tech, Inc.

We have done our best to ensure that the material found in this publication is both useful and accurate. However, please be aware that errors may exist in this publication, and that RISA Tech, Inc. makes no guarantees concerning accuracy of the information found here or in the use to which it may be put.

Table of Contents

Before You Begin	1
Overview	1
Minimum System Requirements	2
Program Limits	2
License Agreement	3
Main User Interface	4
Start a Model	4
Title Bar	5
Ribbon Toolbar	7
Quick Access Toolbar	8
Properties Panel	12
Explorer Panel	18
3D View	19
Shortcut Keys and Hot Keys	26
Status Bar	30
Windows Behavior	32
Modes	35
File Menu	36
Home Ribbon	39
Modify Ribbon	45
View Ribbon	50
Drawing Tools Ribbon	53
Spreadsheet Ribbon	55
Advanced Ribbon	60
Results Ribbon	63
Aluminum	66
Aluminum - Databases	66

Aluminum - Design	82
Boundary Conditions	88
Creating and Modifying Boundary Conditions	88
Generate Soil Springs	90
Boundary Conditions Spreadsheet	91
Boundary Condition Options	93
Cold Formed Steel	97
Cold Formed Steel - Databases	97
Cold Formed Steel - Design	105
Cold Formed Steel Walls - Design Rules	115
Unity Check	115
CFS Wall (Studs)	116
Design Rules - CFS Wall (Fasteners)	117
CFS Walls	118
Cold Formed Steel Walls - Results	135
Concrete Members	148
Concrete Member - Database	148
Concrete Member - Design	154
Concrete Member - Design Results	170
RISACONNECTION Integration	192
Available Connection Types	192
Integrated Connection Design Procedure	194
Seismic Connections	211
Tapered Wide Flange Connections	213

Table of Contents

Troubleshoot RISACONNECTION Integration	214	Drift	288
Integration Error Messages	218	Drift Definitions Spreadsheet	288
RISAFloor & RISA-3D Connection Design Interaction	220	Drift Results	291
Customizing RISA-3D	224	Drift Modeling Tips	294
Save as Defaults	224	DXF Files	296
Application Settings (Preferences)	224	Import a DXF File	296
Member Design Optimization	239	Export a DXF File	298
Wall Design Optimization	239	Merge After a DXF Import	301
Member Design Lists	239	DXF Element Numbering	302
Member Design Rules – Size / U.C. ..	240	DXF File Format	302
Member Design Rules – Deflection ..	241	Dynamic Analysis	305
Member Design Rules – Concrete Rebar	241	Dynamic Analysis - Eigensolution	305
Member Design Rules Editor	242	Dynamic Analysis - Response Spectra	317
Member Optimization Procedure	245	Dynamic Analysis - Time History	336
Optimization Results	246	File Operations	367
Diaphragms	249	Start Off	367
Rigid Diaphragms	249	Append Models	371
Semi-Rigid Diaphragms	252	Import and Export	372
Semi-Rigid Diaphragms (RISAFloorES)	255	Automatic Backup	373
Flexible Diaphragms	262	Generation	374
Diaphragms Spreadsheet - General Tab	264	Circular Arc Generation	375
Diaphragm Modeling Tips	268	Parabolic Arc Generation	376
Flexible Diaphragms - Analysis and Results	272	Grid Generation	378
		General Truss Generation	381
		Geodesic Dome Generator	383
		Cylinder Generation	385

Table of Contents

Cone Generation	388	Split Members	466
Circular Disk Generation	391	Add Nodes to Members	468
Circular Radius Generation	392	Swap Member Ends	470
Rectangular Tank Generation	395	Trim / Extend Members	470
Graphic Display	400	Re-Label Selection Items	473
Window Behavior	400	Merge Model Elements	474
Adjust the Viewpoint (3D View)	405	Match	475
Animate Results	411	Graphic Selection	478
Create a Custom Snapshot of a Model	413	Selection Shortcuts	480
View Only Part of a Model	415	Cycle Selection	480
Save / Retrieve a Model / Snapshot View	415	Selection Modes	480
Delete a Model View	418	Invert Selections	483
Graphic Editing	420	Criteria Selections	483
Draw / Modify Features	420	Lock & Dim Lock	496
Undo / Redo Operations	422	Viewer Mode	498
Project Grid	422	Input Filter	502
Drawing Grid	433	Graphic Selection from Spreadsheets	502
Snap Points	441	Save Selections	503
Lock to Axis	442	Quick Find	506
Lock to Plane	443	Graphic View Settings	508
Import DXF	444	Graphics - Model View Settings	508
Delete Elements	449	Set as Default - checkbox	510
Copy Model Elements	452	Results View Settings	522
Move / Rotate Model Elements	460	Help Options	537
Scale Elements	464	Electronic Help File	537
		Context Sensitive Help	537

Table of Contents

RISA Technology Online	537	Loads - Line Loads	683
Tool-tips	538	Loads - Moving Loads	688
Tutorials	538	Loads - Thermal Loads	700
Hot Rolled Steel - Databases	539	Loads - Surface Loads	702
Select a Database Shape	540	Load Generation - Notional Loads .. 713	
Add a Database Shape	542	Vertical Load Used for Notional Force Calculations	714
Edit a Database Shape	545	Notional Load Generation Window ...	714
Available Database Shapes	548	Notional Load Results	715
Hot Rolled Steel - Databases	553	Load Generation - Notional Loads	716
Hot Rolled Steel - Design	568	Load Generation - Seismic Loads	719
Hot Rolled Steel - Code Check	596	Load Generation - Wind Loads	726
Stainless Steel	617	Material Properties	751
Database Shapes	617	Hot Rolled Specific Material Data	752
Design	619	Cold Formed Specific Material Data ..	753
Limitations	622	Wood Specific Material Data	753
Stainless Code Check Results	622	Concrete Specific Material Data	755
Unity Check (UC) Results	623	Masonry Specific Material Data	756
Loads	624	Aluminum Specific Material Data	757
Self Weight (Gravity Load)	624	Stainless Steel Specific Material Data	758
Draw Loads	625	General Material Data	758
Modify Loads	625	Material Take-Off	759
Delete Loads	626	Members	762
Basic Load Cases	626	Draw Members	762
Load Combinations	634	Modify Members	765
Loads - Nodal Load / Displacement ..	658	Material and Cross Section Properties	769
Loads - Area Loads	662		
Loads - Point Loads	678		

Table of Contents

Members Spreadsheet	770	Member Stress Results	817
Members Spreadsheet - Advanced Data	772	Single Angle Results	819
Tension/Compression-Only Members	774	Member Torsion Results	819
Physical Members	778	Member Deflection Results	821
Local Axes	779	Model Merge	829
Member Orientation	780	Model Merge Options	829
Member End Releases	781	Model Merge Examples	830
Simpson Yield-Link	783	Model Merge Limitations	830
Deflection Ratio Options	788	Model Merge Process	831
Analysis Offset	791	Model Settings	833
Col-Wall Vert Release	800	Project Information	834
Inactive and Excluded Elements	801	Solution	835
Shear Deformations	801	Axis	843
Member Shear Stresses	802	Codes	844
Torsion	803	Concrete	848
Warping	803	Rebar	850
Cardinal Points	808	Seismic	851
Overview	808	Modeling Tips	854
Detailing Input and Modification	810	Applying In-Plane Moment to Plates	854
Visualization of the Detailing Layer ..	811	Modeling a Beam Fixed to a Shear Wall	854
File I/O	811	Modeling a Cable	855
Members - Results	812	Modeling Composite Behavior	856
Number of Reported Sections	812	Modeling Inclined Supports	857
Number of Internal Sections	812	Modeling One Member Over Another	858
Member Force Results	813	Reactions at Nodes with Enforced Displacements	858

Table of Contents

Rigid Links	858	Modify Plates	881
Solving Large Models	860	Plate Information	883
Modeling a "Gap" (Expansion Joint) Between Structures	861	Plate Corner Releases	885
Nodes	863	Inactive and Excluded Plates	887
Define Nodes	863	Rotate Plates	888
Relabel Nodes	863	Plates Spreadsheet - Primary Data ..	902
Round-Off Node Coordinates	864	Plates Spreadsheet - Advanced Data	904
Node Coordinates Spreadsheet	864	Plate Local Axes	905
Node Information	866	Plate/Shell Element Formulation	906
Nodes - Results	868	Plate Modeling Tips	910
Nodes - Tethering Nodes	870	Plate Error Checking	912
P-Delta - Analysis	872	Finite Element Basics	912
P-Delta Procedure	872	Plates/Shells	914
P-Delta Limitations on Standard Nodal Shear Method	874	Plates/Shells - Results	949
Compression Only P-Delta	874	Plates/Shells - Design Tools	954
P-Delta Convergence	874	Plates/Shells - Modeling Examples ..	960
P-Delta Troubleshooting	875	Printing	972
Wall Panels	876	Print Graphics	974
Leaning Column Effect	876	Print Reports	979
P-Little Delta Analysis	877	Print a Spreadsheet	988
P-Little Delta Procedure	877	Print to a File	995
AISC Direct Analysis Method	878	Build a Custom Report	998
ACI Concrete Design	879	Save a Report as a Template	1001
Plates/Shells	880	Results	1007
Draw Plates	880	Save Results	1008
		Results Spreadsheets	1008

Table of Contents

Exclude Results	1008	Seismic Detailing - Design Rules ...	1043
Graphic Results	1011	Seismic Detailing - Results	1052
Clear Results	1011	Seismic Detailing - Detail Reports ..	1063
Internal Force Summation Tool	1012	Shape Databases	1073
Member Detail Report	1013	Database Shape Types	1073
Concrete Member Detail Reports ...	1021	Hot Rolled Shapes	1073
RISAFloor & RISA-3D Integration ..	1022	Cold Formed Shapes	1074
Lateral System Model Generation ..	1022	Concrete Shapes	1074
Diaphragms	1023	Wood Shapes	1074
Gravity Loads	1024	Aluminum Shapes	1074
Wind Loads	1024	General Shapes	1075
Seismic Loads	1024	RISASection Files	1076
Buckling-Restrained Braced Frame (BRBF) System	1024	On-Line Shapes	1079
Detach	1025	Database Files	1080
RISAFoundation Interaction	1026	Solid Elements	1082
RISAFoundation Interaction with RISA-3D	1026	Create Solids	1082
RISAFoundation Interaction with RISAFloor	1029	Modify Solids	1085
Limitations	1030	Solids Spreadsheet	1088
Section Sets	1032	Solid Information	1089
Define a New Section Set	1032	Inactive and Excluded Solids	1090
Section Sets Spreadsheet	1032	Solids Formulation	1091
Seismic Detailing - Design Rules ..	1035	Solid Modeling Tips	1092
General Frames and Columns	1035	Loading	1092
Beams	1038	Verification Examples	1092
Braces	1041	Solid Elements - Results	1095
		Solid (Global) Stress Results	1095
		Solid Principal Stress Results	1096

Table of Contents

Solution	1100
Static Solutions	1102
Dynamic Solutions	1104
Response Spectra Solutions	1105
Spreadsheet Operations	1107
Move and Scroll	1108
Spreadsheet Keyboard Commands	1109
Select Spreadsheet Cells	1109
Undo Operations	1110
Redo Operations	1111
Edit Spreadsheets	1111
Move and Copy Cell Contents	1115
Sorting and Finding in Spreadsheets	1117
Default Spreadsheet Data	1118
Export to Excel Spreadsheet	1119
Special Spreadsheet Functions	1120
Stability	1123
Instability Procedure	1123
Instability Causes	1124
Instability Examples	1124
Testing Instabilities	1130
Unbraced Lengths	1131
Lb Values (Lb, Lu, Le)	1132
Lcomp Values (Lcomp, Le-bend) ...	1133
L-torque	1134

Automated Unbraced Length Commands	1135
Material Specific Behavior	1138
K Factors (Effective Length Factors)	1142
Sway Flags	1144
Units	1145
Standard Imperial Units	1147
Standard Metric Units	1147
Units Specifications	1147
Wall Panels	1149
Drawing Wall Panels	1149
Modifying Wall Panels	1151
Wall Panel Spreadsheets	1153
Wall Panel Editor	1157
Wall Panel Local Axes	1164
Loading	1166
Load Attribution	1166
Meshing the Wall Panels	1169
Wall Panels - General Results	1174
Wall Force Results	1174
General Wall Detail Report	1176
CFS Walls	1179
Cold Formed Steel Walls - Design ..	1179
Type I - Segmented Method	1183
Geometric Requirements and Limitations	1184
CFS Walls	1196

Table of Contents

Cold Formed Steel Walls - Design Rules	1212	Wood Material/Shape Selection	1443
Cold Formed Steel Walls - Results	1216	Wood Database Limitations	1446
Concrete Wall Panel - Design	1228	Custom Wood Sizes	1450
Concrete Wall Panel - Design	1228	Wood - Database	1450
Concrete Wall - Design Rules	1245	Wood - Design	1458
Concrete Wall Results	1249	Warning Log	1477
Concrete Wall - Seismic Design	1266	Sum of reactions is not equal to the sum of the loads (LC xx)! Check for any small rigid links or fixed boundary conditions.	1477
Masonry Wall - Modeling	1278	Appendix A – Redesign Lists	1479
Masonry Wall Input	1278	Creating or Modifying Redesign Lists	1479
Openings / Lintels	1279	Design List Mismatch	1481
Regions	1281	Locating the Design Lists	1481
Optimization / Limitations	1286	File Naming Convention for the Design Lists	1481
Masonry Wall - Modeling	1290	File Format for Redesign Lists	1482
Masonry Wall - Design	1302	Manual Modification	1484
Masonry Wall - Design Rules	1321	Appendix A – Redesign Lists	1484
Masonry Wall Results	1327	Appendix C – STAAD® Files	1490
Masonry Wall - Seismic Design	1346	Appendix E - Interfacing with Other Programs	1499
Masonry Wall - Seismic Design Results	1354	Appendix F – Wood Shear Wall Files	1502
Wood Wall - Design	1359	Appendix G – CFS Shear Wall Files	1514
Wood Wall Input	1359	Technical Support	1518
General Program Functionality and Limitations	1371	Email Support	1518
Wood Wall - Design	1387	Phone Support	1518
Wood Wall - Design Rules	1415		
Wood Wall Results	1420		
Wood - Database	1442		

Before You Begin

Welcome to the RISA-3D General Reference manual. Please read this topic prior to installing the program and pay particular attention to the [License Agreement](#). If you are a first time user of RISA-3D you should turn your attention to the [Tutorials](#) (a separate document) which are designed to get you up and running as fast as possible while still exposing you to the important features of the software.

The [Tutorials](#) are designed to be read in two ways. If you are already familiar with structural modeling in general you can skip the supporting text and read only the underlined action items to quickly move through the tutorial. If you want more thorough explanations of the modeling process you may read all or some of the supporting text as you see fit.

After you have gone through the [Tutorials](#), use this General Reference for detailed information on any topic. The topics are arranged in alphabetical order and are thoroughly indexed.

Overview

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

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

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

You may access the features in RISA-3D by using the menu system, or the toolbars. The best way to learn RISA-3D is to go through the [Tutorials](#). The advantage to this is that you are exposed to the tools RISA-3D provides and the ways that you can take advantage of them.

Minimum System Requirements

Software

One of the following operating systems is required:

- Microsoft Windows 11 (64 bit only)
- Microsoft Windows 10 (64 bit only)
- Microsoft Windows 8.1 (64 bit only)

Hardware

The following hardware is required:

- 1 GHz or faster processor (x86-64)
- 1920x1080 monitor resolution recommended (1024x768 minimum)
- 2 (or more) button mouse, mouse wheel recommended
- 8 GB of RAM
- 4 GB of hard disk space

Internet

An internet connection is required to launch the program. The internet connection must be maintained as long as the program is open, although brief internet outages (a few minutes) do not affect the user's ability to keep the program open.

Program Limits

Software Limitations

Joints	500,000
Members	100,000
Plates	250,000
Shapes	100,000
Wall Panels	10,000
Openings/Regions (per Wall Panel)	25/100
Solids	250,000
Section Sets	10,000
Loads (All Types Combined)	300,000
Materials	500
Custom Wood Species	500
Diaphragms	500

Basic Load Cases	1,000
Moving Loads	500
Time History Functions	500
Load Combinations	5,000
Mode Shapes	2,000
Time History Load Functions	500
Drift Definitions	500

Demonstration Version

While you can open and solve a larger model, the demonstration version does not allow saving or printing results. RISACONNECTION results may not be exported back into RISA-3D when using the demonstration version.

License Agreement

For the full license agreement, please visit: risa.com/eula

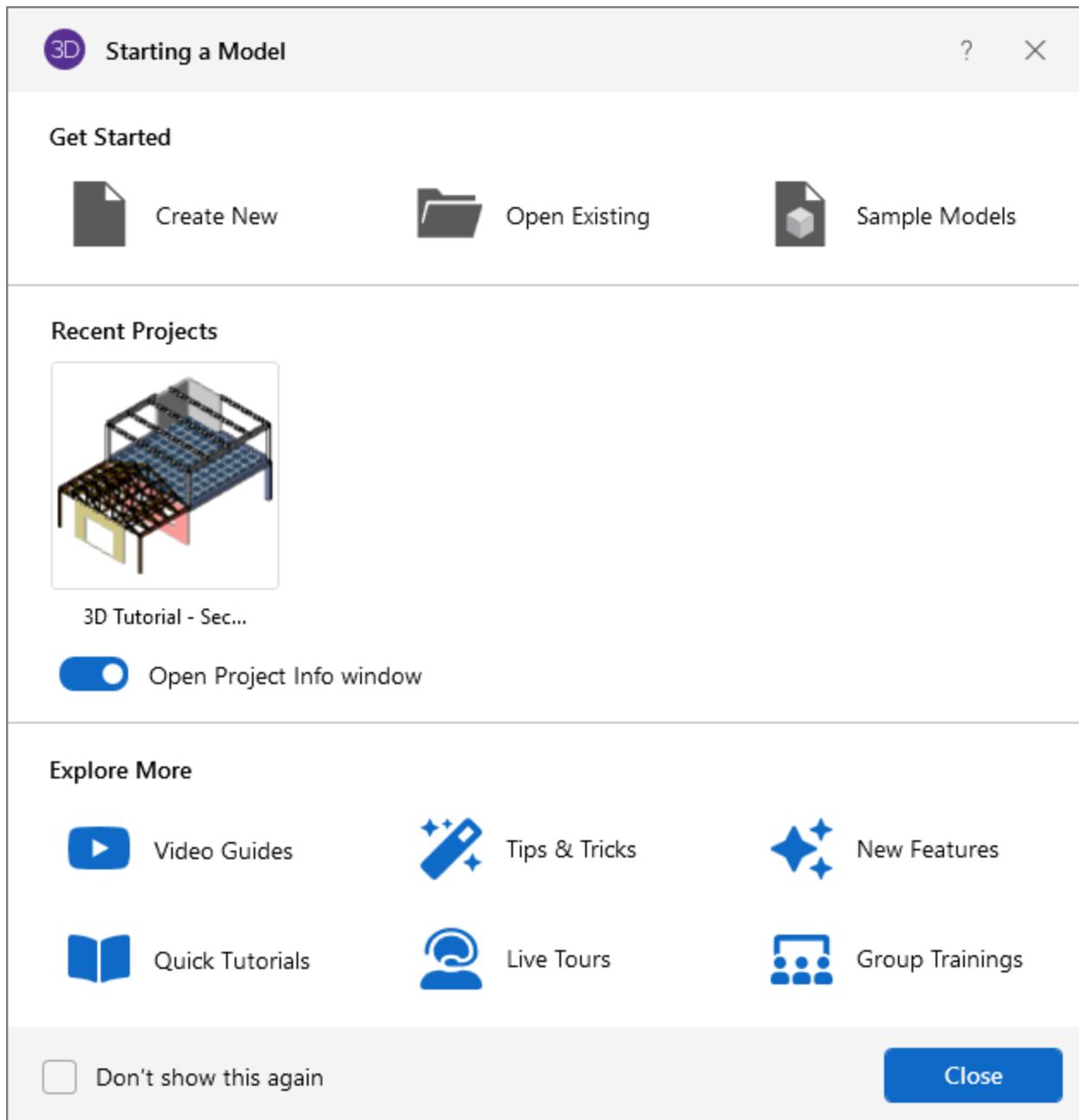
Main User Interface

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

The features that are available to you in RISA-3D may be accessed through the Ribbon toolbar, Quick Access toolbar, Properties panel, Explorer panel, 3D view and shortcut keystrokes. You may use any or all of these vehicles to interact with the software. The Ribbon toolbar has the advantage of containing all of the program options and features. The Quick Access toolbar contain more common options and invoke with one click. The Properties panel will display relevant properties for selected elements within your model and allow you to make quick changes to those elements. The Properties panel is also where you will modify and execute most of the program features. The Explorer panel provides a quick way to access all of the data entry and results spreadsheets. The shortcut menus present options relevant to the task at hand. The shortcut keys provide a fast way to access features should you use the program often enough to make them familiar to you. All of these features are discussed in the sections below. There are many ways to access features and the method that you will use will simply be a matter of personal preference. The good news is that you have the options.

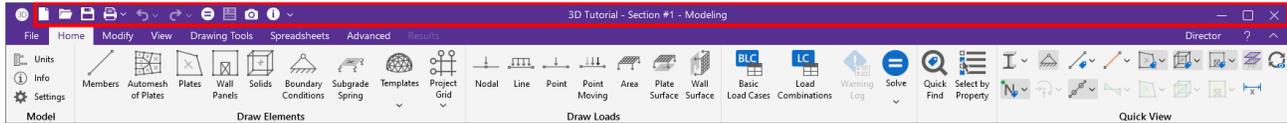
Start a Model

The first dialog that appears when you launch the program is Starting a Model. From this dialog, you can create a new model, open an existing model, or open a sample model which is included with the installation. The **Recent Projects** section displays a preview of recently opened projects. At the bottom, the **Explore More** section contains quick links to learn more about the program like reading the Tutorials or requesting a live Software Tour.



Title Bar

The bar along the top of the screen is called the title bar and contains: the Quick Access toolbar (to the left next to the 3D logo), the name of the file that is currently open (in the middle) and application controls (to the right).



The three buttons on the upper far right side of the title bar are used to control the main window.



- The  (Minimize) icon shrinks the main application window to a button on the taskbar.
- The  (Restore Down) icon shrinks or maximizes the window on your screen.
- The  (Close) icon closes the window, prompting you to save changes if necessary.

You will also see these icons in other windows where they have basically the same functions.

Below these three icons are the Feedback and Director links, and the Help and Minimize Ribbon toolbar icons.



- The **Feedback** link takes you to the RISA Tech Inc. Feedback page where you can view current feedback and leave feedback of your own. This feature provides a quick and easy way for you to communicate program improvements or changes you would like us to consider, or for sending us ideas for new features and enhancements. If you need to report a bug, or require help with an issue you are experiencing with one of our products, please reach out to our Technical Support support@risa.com, as they are better equipped to handle such issues.
- The **Director** link lets you transfer back a RISAFloor or RISAFoundation model back to those programs. See [Director - Program Interaction Tool](#) for more information about using the 'Director' feature.
- The  (Help) icon opens the RISA-3D help file.
- The  (Minimize the Ribbon) icon lets you minimize the contents within the Ribbon toolbar.

To the left of the title bar, next to the 3D logo, is the Quick Access toolbar.



The Quick Access toolbars provides a quick way to access commonly used features and functions in the software. See the [Quick Access toolbar](#) section for more information.

Most of the functions and features that you will be using are within the Ribbon toolbar. Clicking the headings or tabs that are presented in the lower half of the title bar gives you access to different portions of the [Ribbon toolbar](#).

Director - Program Interaction Tool

RISA-3D has a “Director” tool that allows for interaction with other RISA programs (currently RISAFloor, RISAFoundation and RISAConnection). Models must originate from either RISAFloor or RISA-3D. The Director tool lets you send the model to any of the aforementioned programs. Models created in RISA-3D, as well as models integrated into RISA-3D from RISAFloor, can be sent to RISAConnection for connection configuration and solution using the Director tool.

After a model has been analyzed (solved) in RISA-3D, it can be sent to RISAFoundation or RISAConnection for further work in that program. Those programs also have the capability to send the changed model back to RISA-3D, using the Director tools in those programs.

Note that the model must be solved in RISA-3D, before attempting to send it back to its originating program or integrating it into one of the other programs.

Send a Model to Another RISA Program

To send a model to another RISA program:

1. Ensure that you have “Solved” the model in RISA-3D.
2. Click on the **Director** link to display its menu.
3. Click on the name of the program you want to send the model to.

RISA-3D closes and the model is opened in the chosen program. If however you attempt to send the model without solving it first, the program will remind you that it is not solved and will not close RISA-3D or send the model to the other chosen program.

See [RISAFoundation Interaction](#), [RISAFloor-3D-Integration.htm](#) or [RISAConnection Integration](#) for more information on interaction with those programs.

Ribbon Toolbar

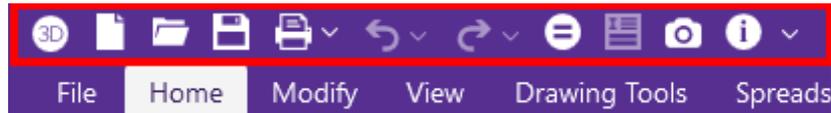
All of the program features may be accessed through the Ribbon toolbar system beginning with the **File** tab on the far left and ending with **Results** tab on the far right.



Clicking on each of the tabs displays options that you can choose from.

Quick Access Toolbar

The Quick Access toolbar provides a quick and easy way to access commonly used features and functions within the software.



The following table provides a description of each Quick Access Toolbar icon.

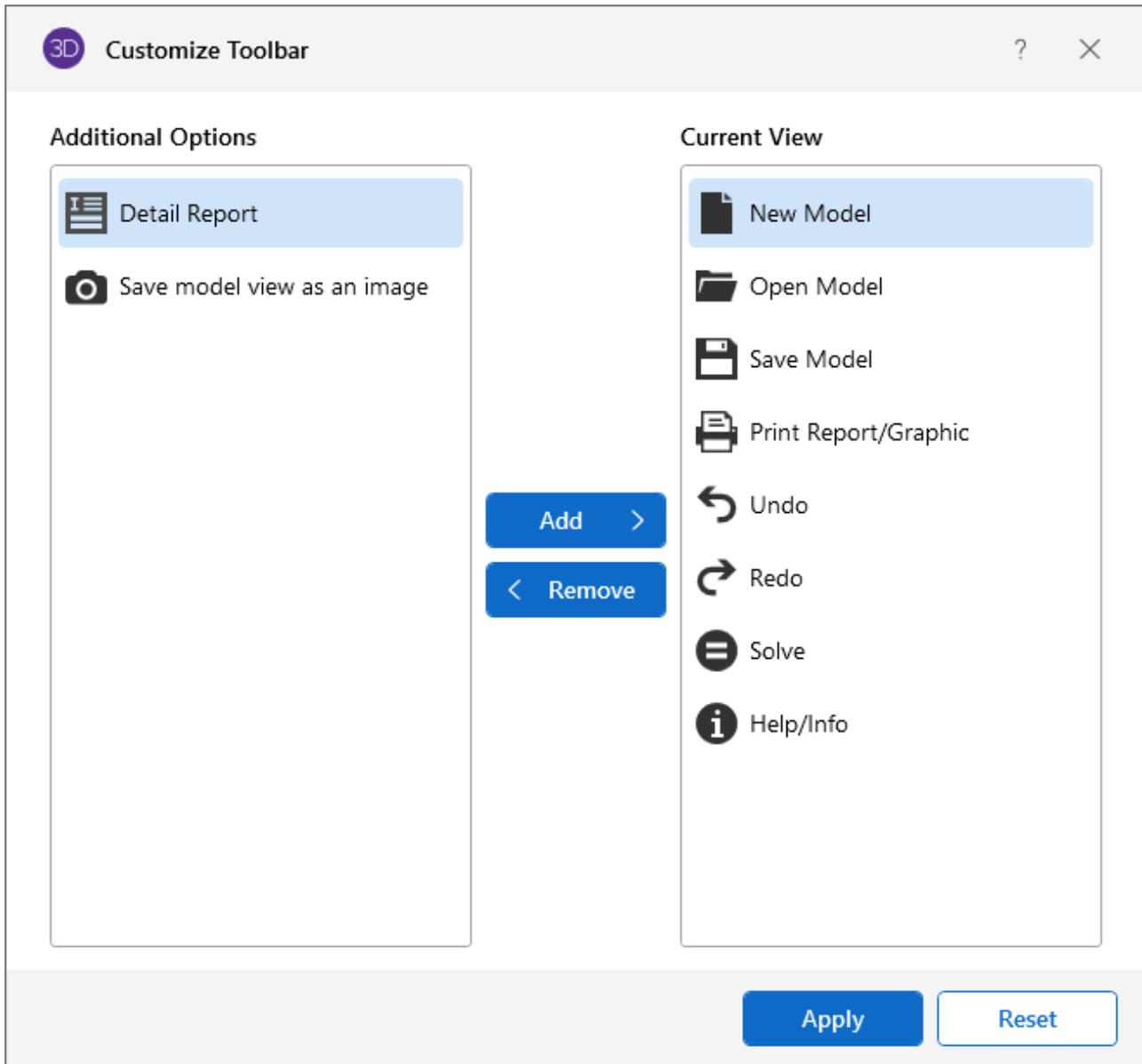
Quick Access Toolbar Icons

Icon	Name	Description
	New Model	Closes the current file (prompting for saving if necessary) and opens a new blank file.
	Open Model	Closes the current file (prompting for saving if necessary) and opens an existing file.
	Save Model	Saves the current file, prompting for a name if necessary.
	Print Report/Graphic	Opens the graphics print preview, report printing setup or spreadsheet printing window.
	Undo	Undoes the last edit that was applied to the model whether it was graphical or in the spreadsheets. You can click the small down arrow to the right of the icon, to review the modeling history and revert the model back through several undo steps at once.
	Redo	Reverse the last undo that was applied to the model. You can click the small arrow to the right of the icon to review the modeling history and jump forward to a point in time.
	Solve	Shows solution choices to solve the model.
	Detail Report	Activates the detail report feature. See the Results topic for more information.
	Save model view as an image	Allows you to capture an image of the 3D view, which can be added to your report. See Printing for more information.
	Help/Info	Provides information about the software licensing and RISA-3D version.
	Change Quick Access Toolbar	Opens a menu that allows you to do one of the following:

Icon	Name	Description
		<ul style="list-style-type: none">• Open the Customize Toolbar window in which you can choose what icons you want displayed in the Quick Access Toolbar.• Move the Quick Access Toolbar below the Ribbon Toolbar.• Move the Quick Access Toolbar above the Ribbon Toolbar.

Customize the Quick Access Toolbar

In RISA-3D you have the ability to choose the icons that appear in the Quick Access Toolbar in the Customize Toolbar window. You can add icons you want displayed and remove those that you don't want displayed. You can also change the position of any icon currently displayed icon.



Add an Icon You Want Displayed

To add an icon to the Quick Access Toolbar:

1. In the **Additional Options** pane, click on the icon.
2. Click the **Add** button.

Remove an Icon You Don't Want Displayed

To remove an icon from the Quick Access Toolbar:

1. In the **Current View** pane, click on the icon.
2. Click the **Remove** button.

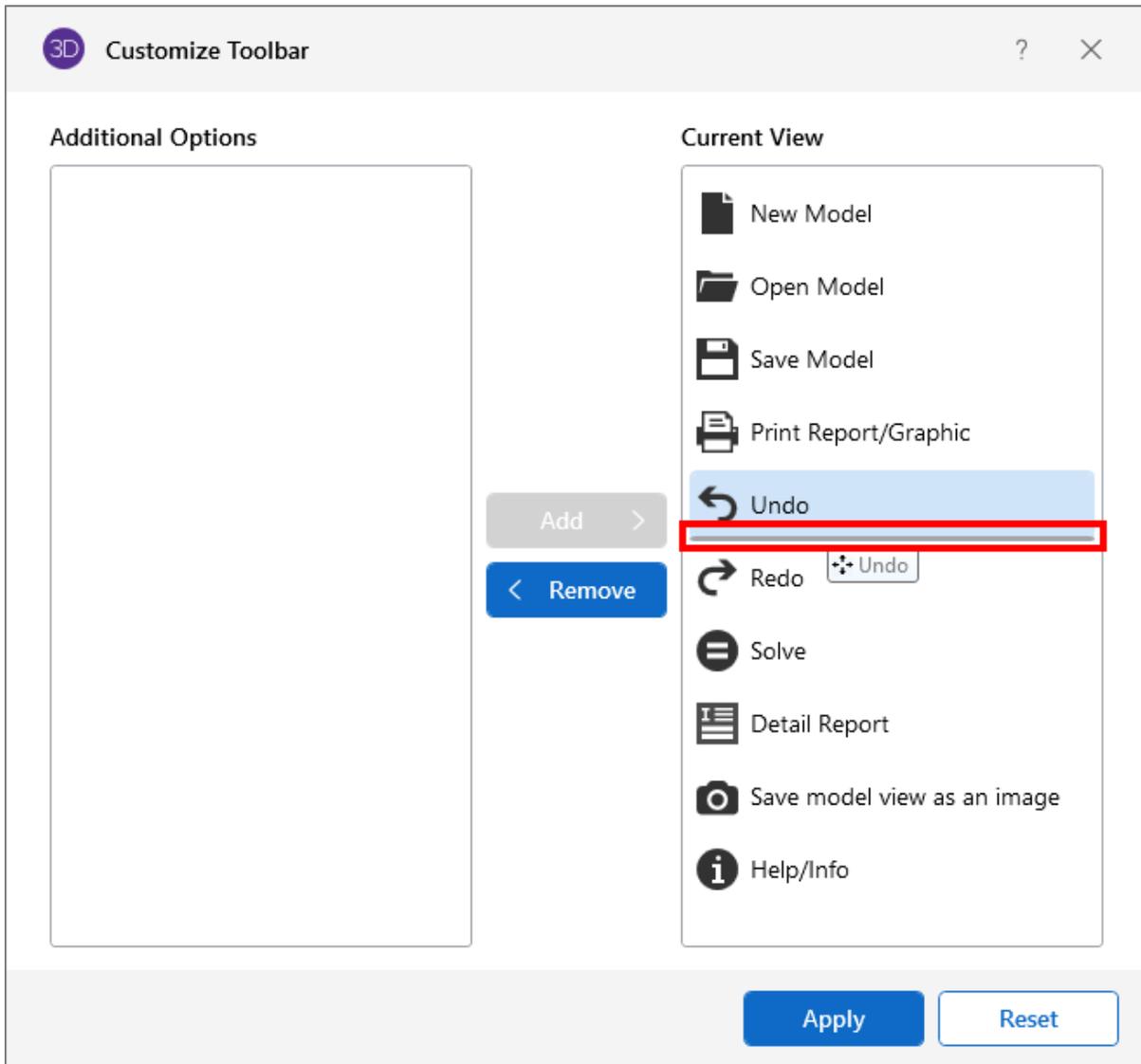
Rearrange Icons

You can rearrange the icons in the Quick Access Toolbar to whatever positions you want them to be appear in.

To rearrange the position of an icon:

1. In the Current View pane, click and hold the mouse button down on an icon.
2. Begin dragging icon up or down in the Current View panel.

A line appears above or below the icon (as shown in the following image), indicating that the direction you have started to move the icon.



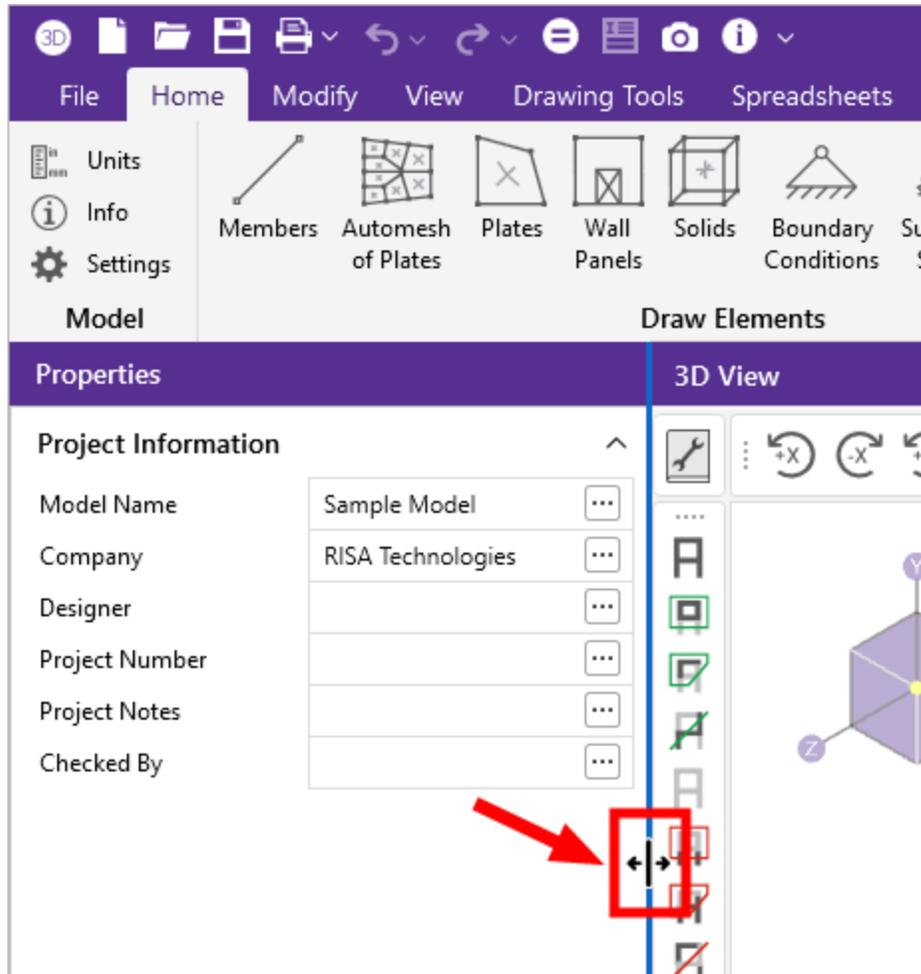
Dragging up moves the icon to the left on the Quick Access Toolbar, while dragging down moves it to the right.

3. Repeat for each icon you want to move.
4. Click **Apply**, after you have moved the icon(s) into the position(s) you want them in.

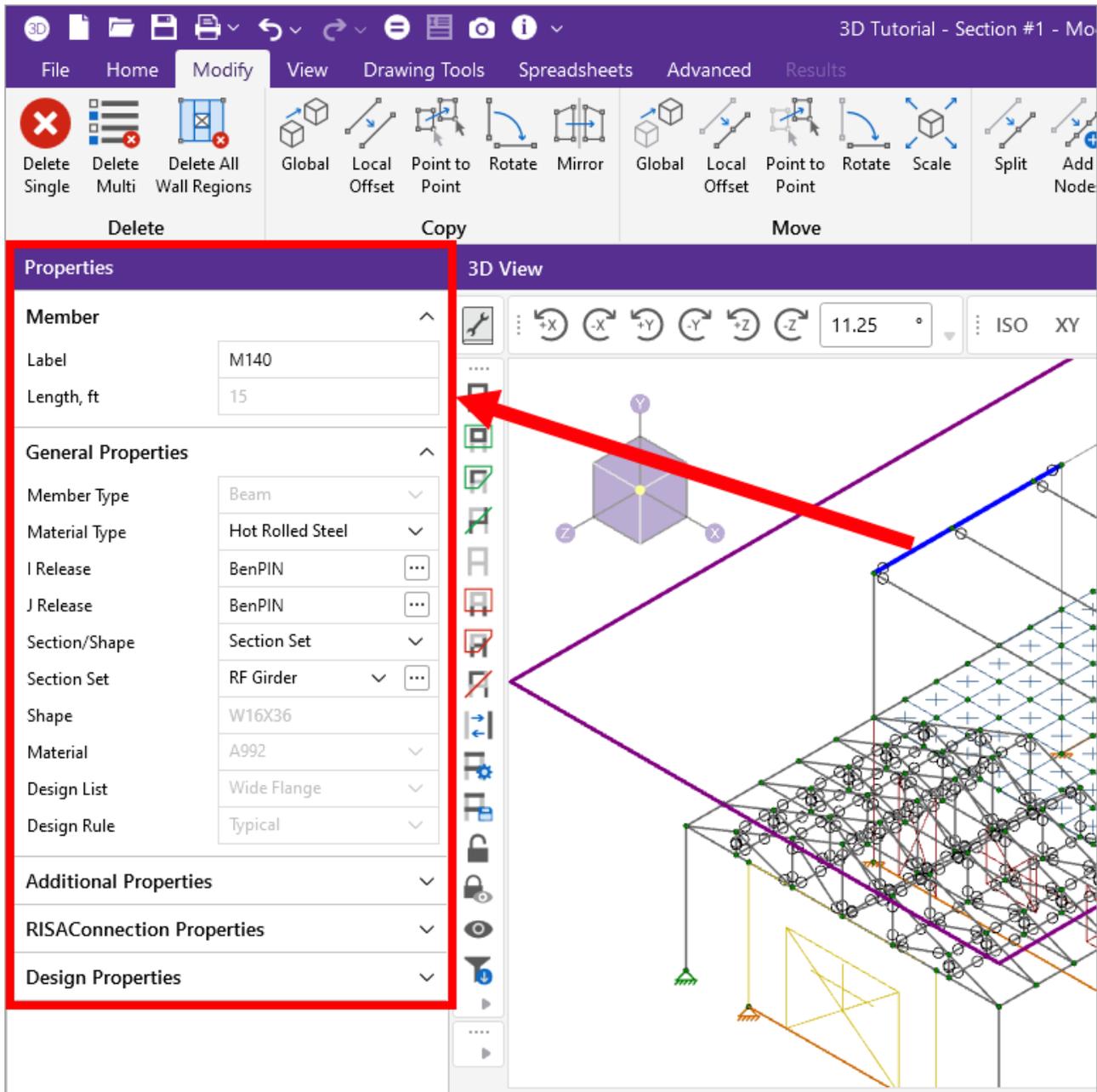
Properties Panel

The **Properties** panel is an affixed panel on the left side of the program interface. This panel serves different functions depending on what elements are selected or what function is activated. When there are no features activated or no elements selected, the **Properties** panel displays the [Project Information](#) entered for the current model. Note that hovering your mouse over the right edge of the

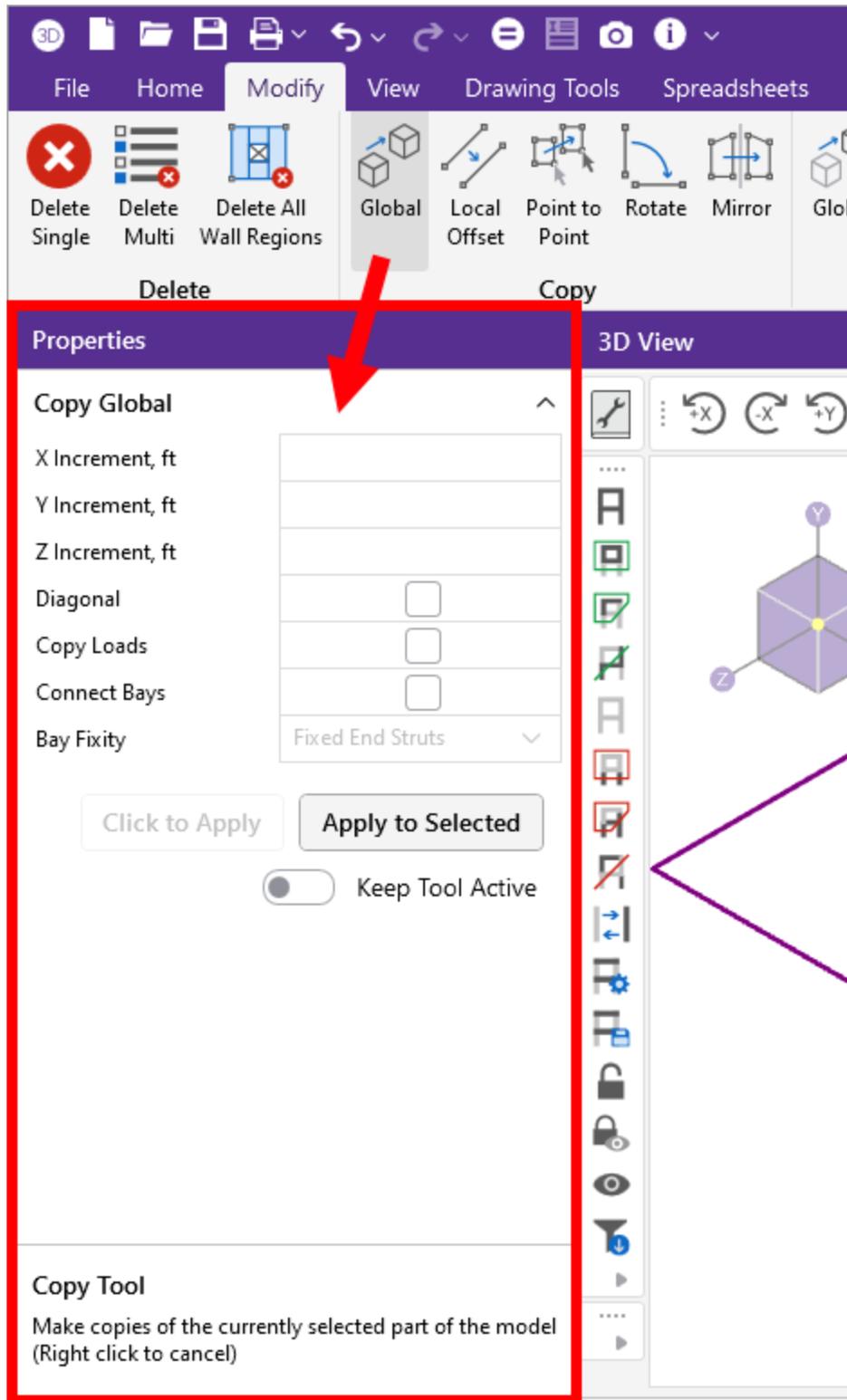
Properties panel (as shown in the following image) allows you to drag and expand or shrink the panel horizontally.



When you select an element within the model, the **Properties** panel displays all of the information related to the selected element(s). You can review and modify the element information and parameters directly from the **Properties** panel.

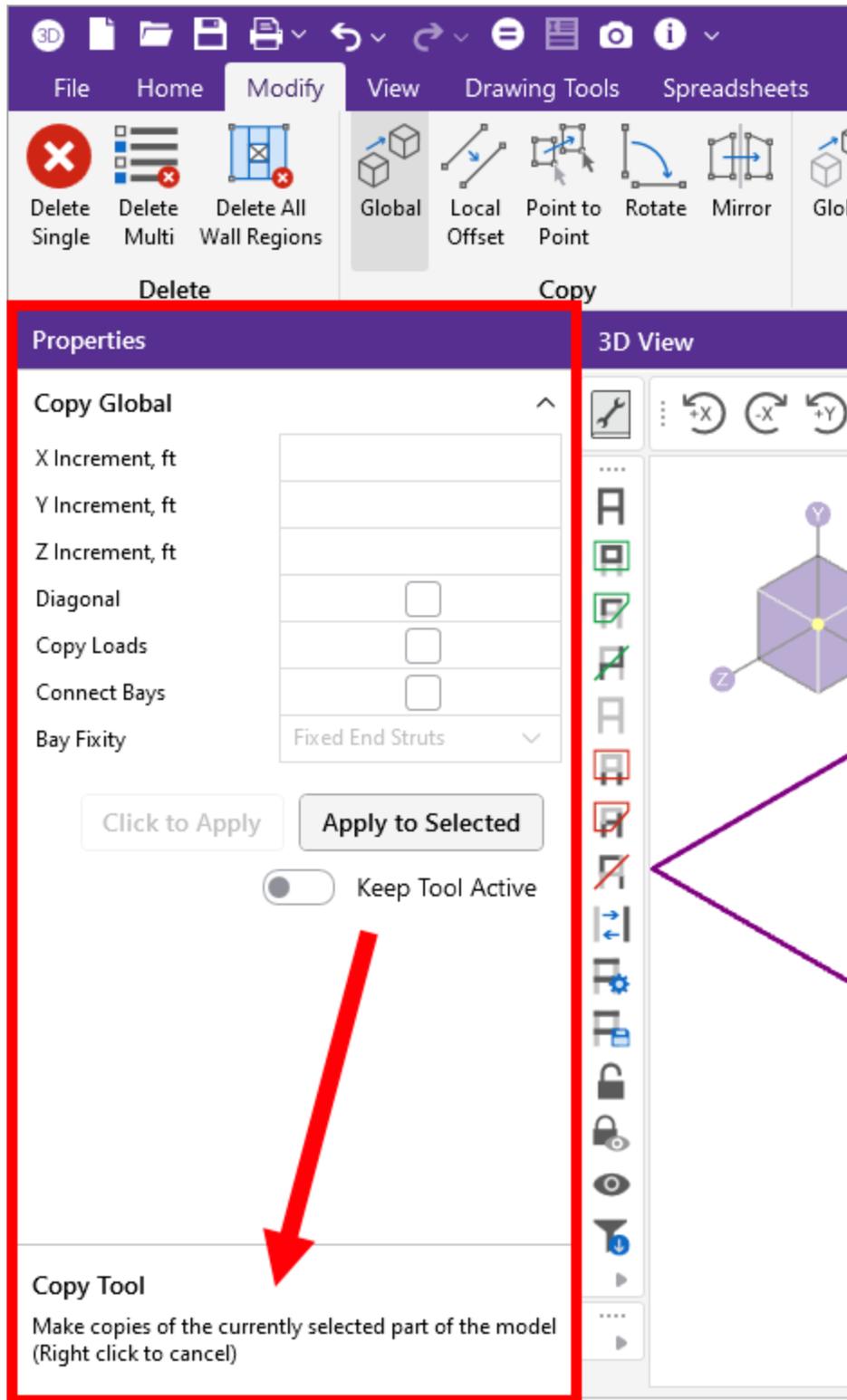


When activating a feature within the program such as adding a load or a copy/move tool, the Properties panel will serve as the method to specify and apply that feature's parameters.



You can then use the **Click to Apply** to option to activate your parameters and click the elements you would like to apply the feature to. Or you can click **Apply to Selected** in order to automatically activate and apply the feature to applicable selected elements in your model.

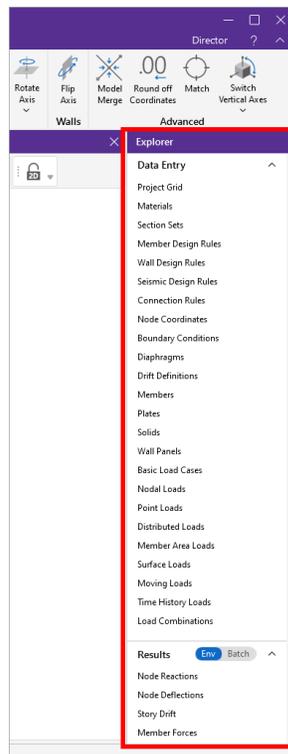
When navigating through a spreadsheet or inputs for a specific function, you will find a short description at the bottom of the **Properties** panel.



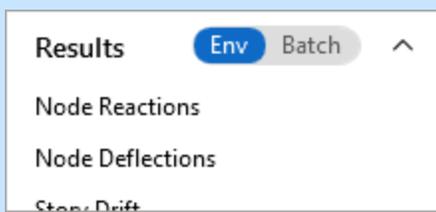
This can be very useful when you are not sure what a specific parameter means.

Explorer Panel

The **Explorer** panel is an affixed panel on the right side of the program interface. It contains a list of all the **Data Entry** spreadsheets. When a solution is present, it also displays the **Results** spreadsheets. Similar to the Properties panel, you can hover your mouse over the left edge of the panel to drag expand or shrink the panel horizontally. You can click the caret next to the **Data Entry** or **Results** title to collapse or expand the spreadsheet list.

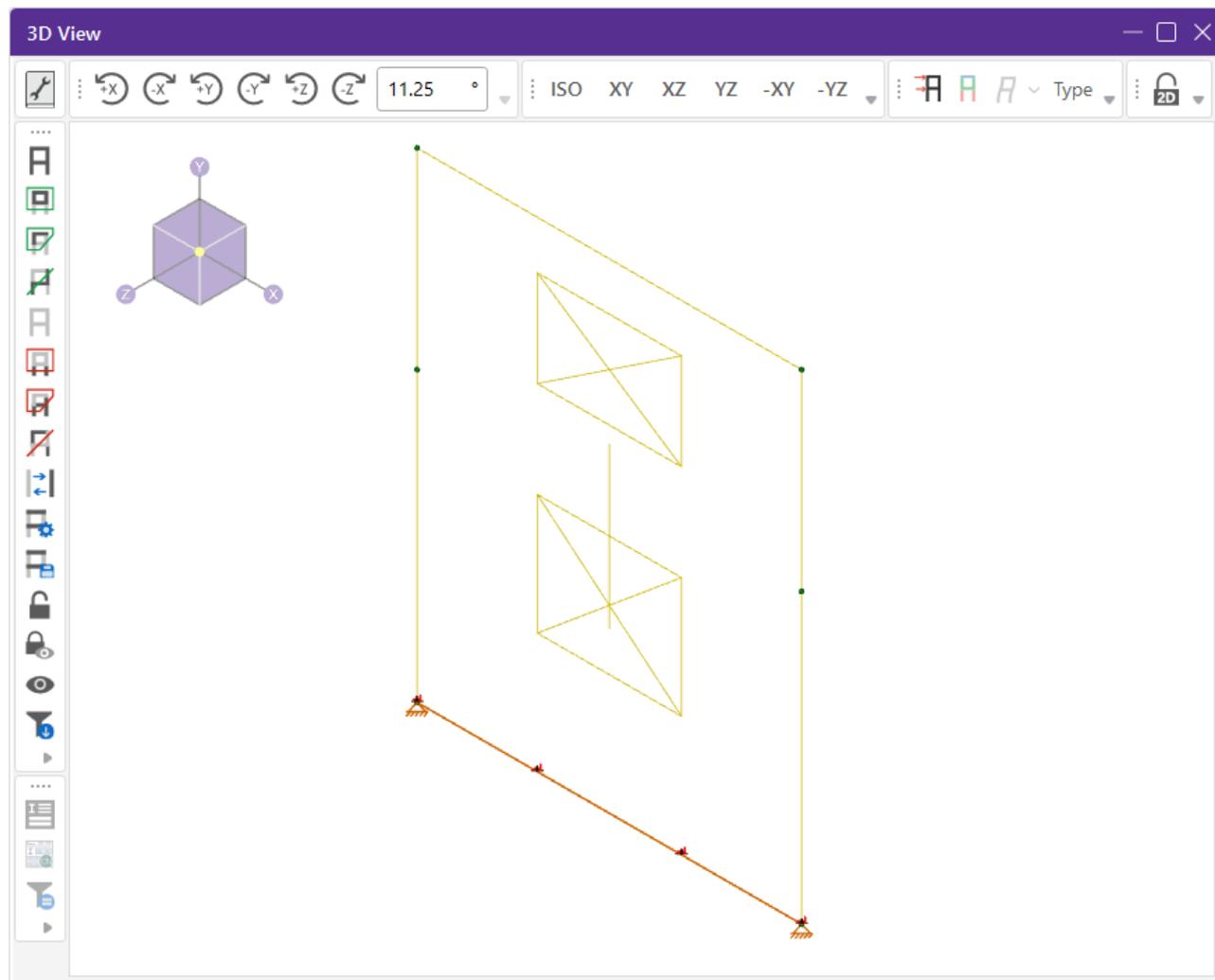


Note: If the solution run is a **Batch with Envelope** solution, then two sets of results spreadsheets are available in many cases, Envelope and Batch. The Env/Batch toggle in the Results toolbar will determine which spreadsheet opens. For example, with **Env** selected, pressing the **Member Forces** button will open the **Envelope Member Forces** spreadsheet. If **Batch** is selected, pressing **Member Forces** will open the **Batch Member Forces** spreadsheet.



3D View

The 3D View is where you do the graphic editing of your model file and where your actual structure can be viewed.



Notice that there are several toolbars located within the 3D View panel. You can rearrange these toolbars to your liking, by clicking on the 4 dots  and dragging the toolbar to any space along the window's perimeter.

You can close this window at anytime by clicking the  (Close icon) to the top right of the 3D View panel.

See the [Graphic Display](#) section for additional details on working with 3D View windows.

Window Toolbar

The toolbar at the top of the 3D View provides controls to adjust the current 3D view.



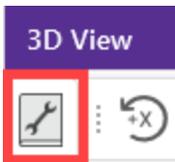
Other 3D view windows that are open are not affected, so that each may show different information. For more information on this, see the [Adjust the Viewpoint \(3D View\)](#) section of Graphic Display.

Custom Bins

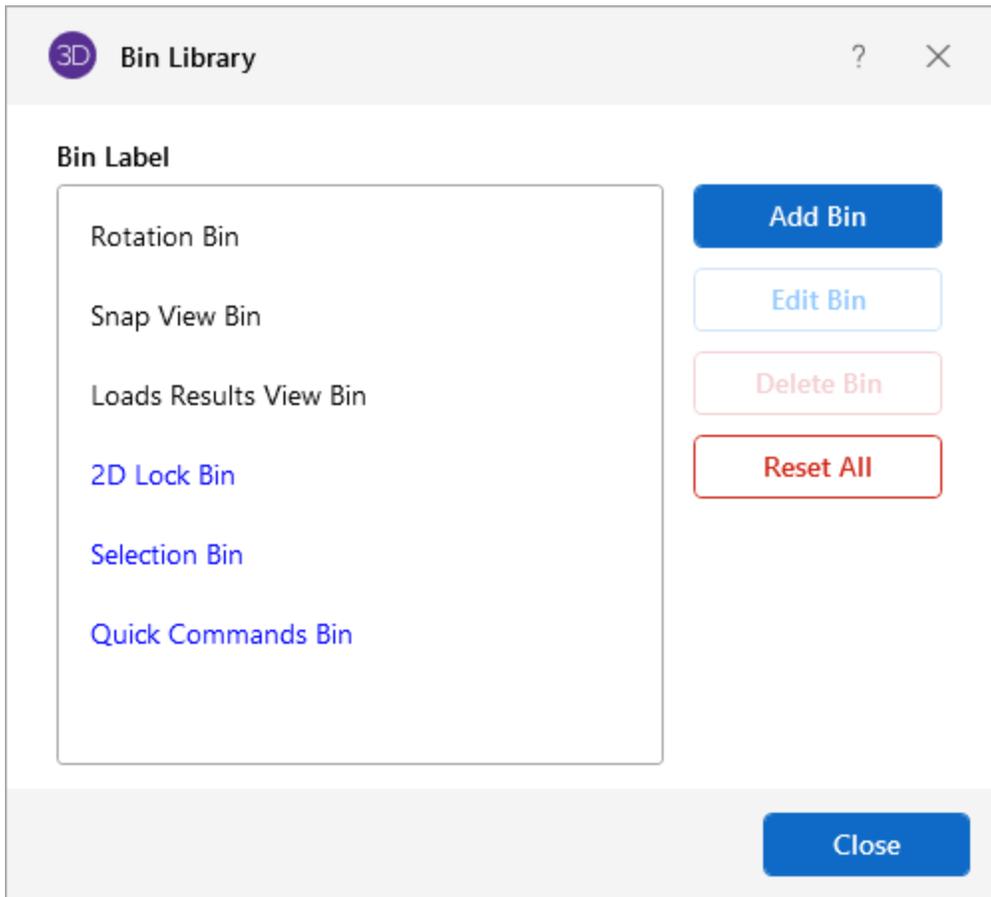
Each grouping of quick buttons in the 3D View panel is called a **Bin**. You can customize your bins to have your most-used tools easily accessible.

To customize your bins:

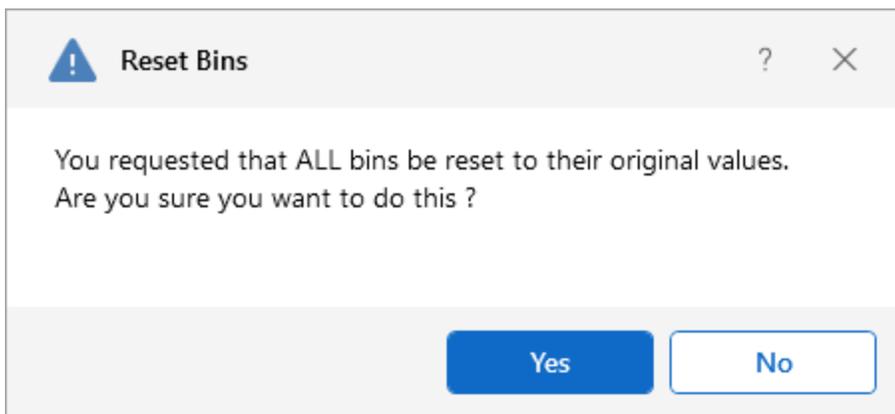
1. Click the **Bin Library** icon in the upper left corner of the 3D View panel, as shown in the following image.



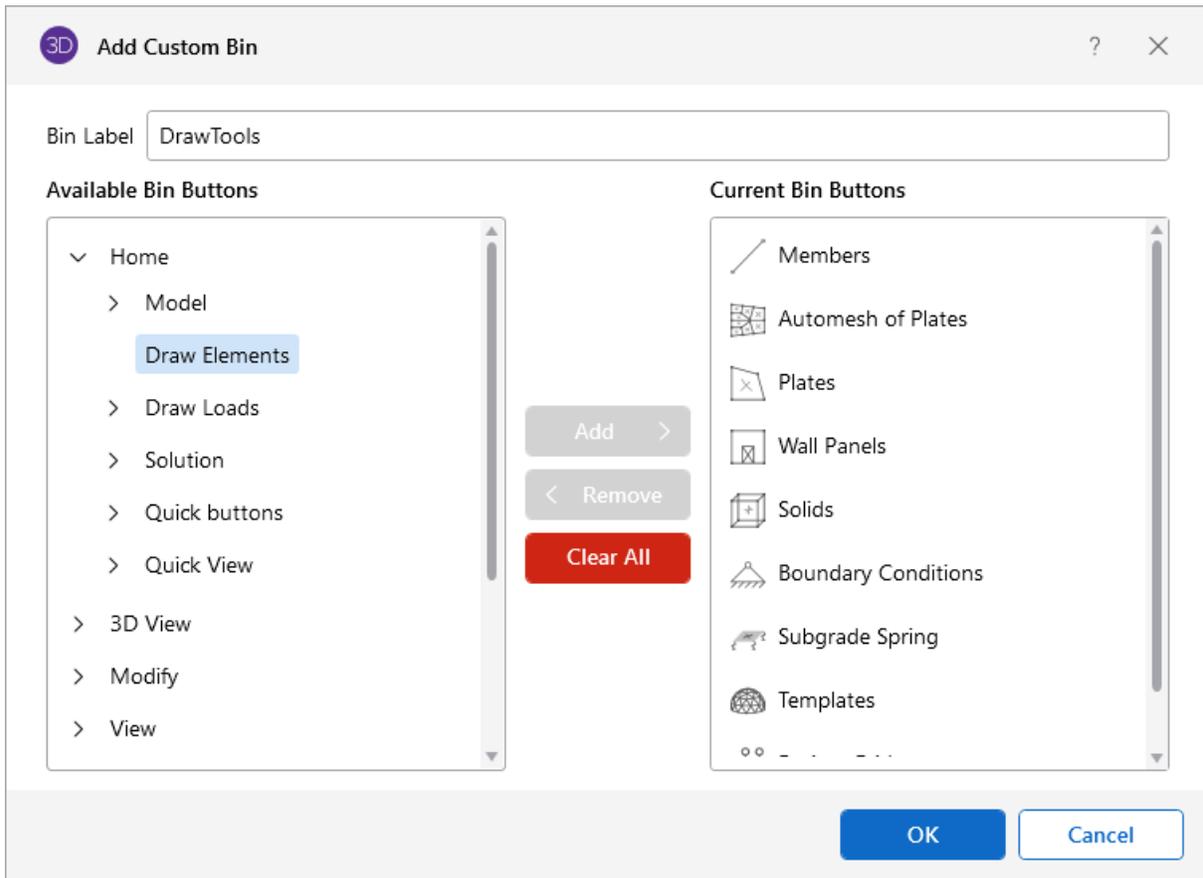
The Bin Library window opens.



In this window you can add your own bins and customize the quick buttons within each bin. You can also edit and delete bins. Note though, that the default bins can not be deleted (Rotation Bin, Snap View Bin, Loads Results View Bin).



If adding a custom bin, the Add Custom Bin window opens after you click the 'Add Bin' button.



This window allows you to name, add and remove bin buttons for your custom bin.

2. (Optional) Click the **Reset All** button to reset all your bins to their default configuration.
3. Click **OK** in the Add Custom Bin window, to save it and return to the Bin Library window.
4. Click **OK** in the Bin Library window if you are finished customizing bins.

Selection Toolbar

The vertical toolbar on the left side of the 3D View is the **Selection Toolbar**. This toolbar is only available when the active window is a model view. The buttons on this toolbar help you select and deselect items in the model, in order to help you build and modify the model or view results. See [Graphic Selection](#) for more information.



Results Toolbar

The vertical toolbar on the left side of the screen below the Selection Toolbar is the **Results Toolbar**.

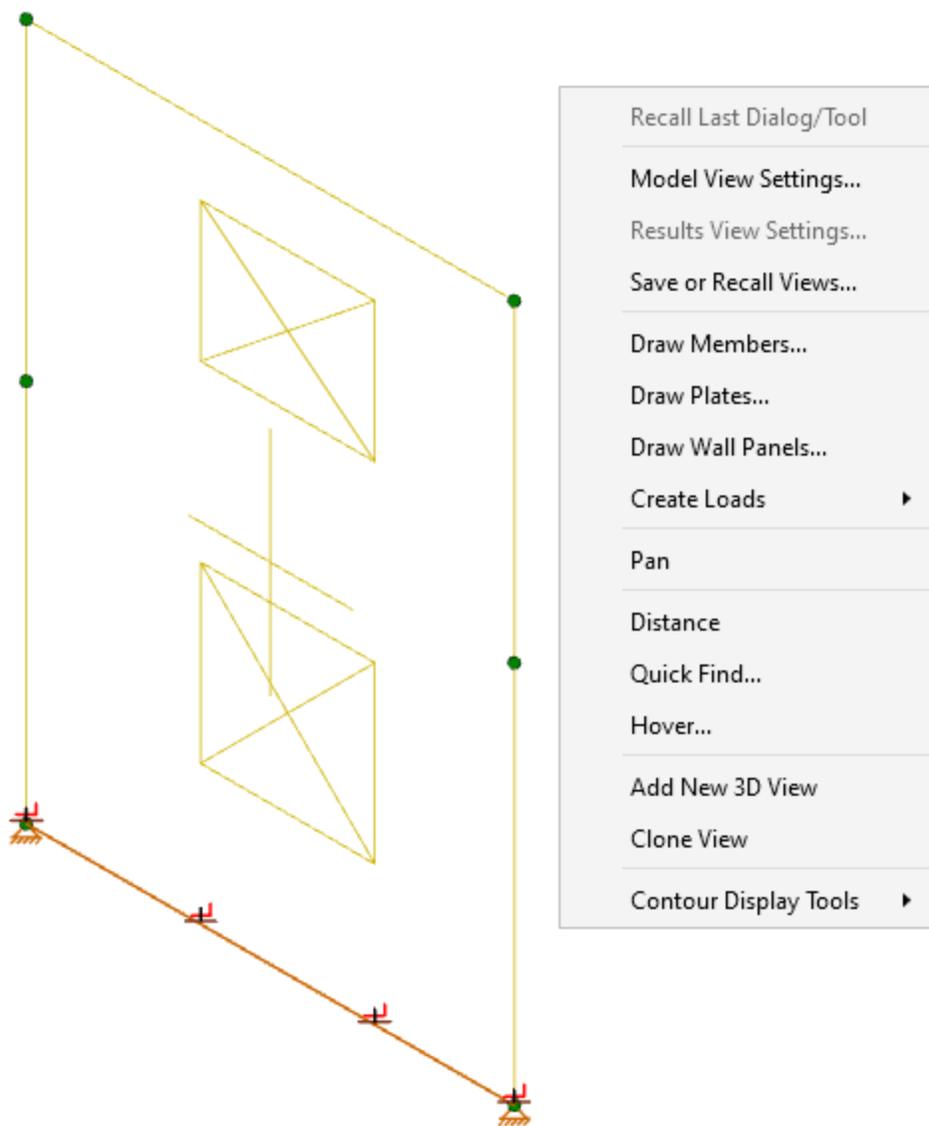


The tools in this toolbar become active when a solution is present. They provide quick access to **Result** tools that would otherwise be selected in the **Results** tab in the **Ribbon** toolbar. For more information see the [Results](#) topic.

Shortcut Menu

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

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



Dynamic View Controls

When your current window is a graphical model view, you can use the mouse wheel to dynamically zoom, pan, or rotate the graphical image.

Mouse Action	Model View Function
Roll the Wheel Forward	Zoom In
Roll the Wheel Backward	Zoom Out
Click and hold the Wheel Button	Grab the image and pan in the direction of mouse movement

Mouse Action	Model View Function
Click and hold the Wheel button while pressing the Shift key	Dynamically rotate the structure in the direction of mouse movement

The following table describes how to use the mouse to perform specific tasks.

Task	What to do
Dynamic Pan	Click and hold the mouse wheel button to trigger the Dynamic Pan feature, which allows you to pan or drag the view to the limit of the scroll bars.
Dynamic Zoom	Rotate the mouse wheel button forward to zoom in and backward to zoom out.
Dynamic Rotate	Click and hold the mouse button while holding the Shift key down, to trigger this feature. The rotational movement is based on the how you drag the mouse cursor over the screen and the projection of global axis on the screen. For rotation about X axis, drag the cursor perpendicular to the projection of the global X axis. The same logic applies for Y or Z axis rotations. When rotation is initiated, the system locks for rotation about that axis, until you release the middle mouse button.
Zoom Previous/Next	The function keys F3 and F4 are associated with Zoom Previous and Zoom Next respectively. The system holds a doubly linked list of zoom info. This list has 10 zoom-states in the list. Press the F3 or F4 key to move the active pointer forward or backward on the list. Each window has its own zoom list.
Dynamic Distance Tool	Press the F5 function key to trigger the Dynamic Distance Tool, then pick up two points on the screen. On the status bar, the system displays the total and partial distance between the points.

Shortcut Keys and Hot Keys

Shortcut Keys and **Hot Keys** allow you to use the keyboard to quickly access features. The difference between the two is simply that the shortcut keys are related to a specific window and will only work in that window while the hot keys will perform at most any time. See [Keyboard Application Settings](#) for more information on how you can customize the Shortcut and Hot Key combinations for various functions in RISA-3D.

General

Key Combination	Function
General Hot Keys	
F1	Help on the active window
F5	Activates the Distance Tool
Ctrl + F1	Main Help topics
F7	Opens solution choices
Ctrl + F7	Solve (Batch + Envelope Solution)
Ctrl + N	Start a new file
Ctrl + O	Open an existing file
Ctrl + S	Save the current file
Ctrl + P	Print
Ctrl + Z	Undo
Ctrl + Y	Redo
Ctrl + Shift + I	Image Capture
Draw Elements Shortcut Keys	
M	Draw members
P	Draw plates
W	Draw wall panels
B	Draw boundary conditions
Loading Shortcut Keys	
L	Display loads
Shift + N	Draw nodal loads
Shift + L	Draw line loads
Shift + C	Draw point loads
Shift + A	Draw area loads
Shift + P	Draw plate surface loads
Shift + W	Draw wall surface loads
Model View Shortcut Keys	
0	Isometric view
Ctrl + 0	Reset view
F2	Open Model View Settings dialog
Ctrl + 5	Open Results View Settings dialog
Double click mouse wheel	Zoom extents
Selection Shortcut Keys	
Ctrl + A	Select all model elements

Key Combination	Function
Ctrl + U	Unselect all model elements
Ctrl + I	Invert model element selection state
Ctrl + L	Lock - view only selected model elements
Ctrl + Shift + L	Dim Lock - view only selected elements with the rest of the model shown in a dimmed state
O	Viewer mode
Drawing Tool Shortcut Keys	
G	Toggle Drawing Grid display
Shift + G	Toggle Project Grid display
Z	Toggle Global Axis Lock
X	Toggle Global Plane Lock
8	Toggle universal snap points
Results Shortcut Keys	
R	Detail Report
K	Toggle display of graphical results
D	Toggle display of deflected shape
I	Internal Force Summation tool
Solution Shortcut Keys	
=	Solution Choices dialog
Ctrl + =	Solve model (Batch + Envelope)
Ctrl + 3	Warning Log
Quick View Shortcut Keys	
F	Quick Find
Ctrl + Shift + F	Select By Property
, (Comma)	Render Elements
. (Period)	Node Display
/	Member Labels
Shift + /	Member Color Codes
' (Apostrophe)	Member Forces
[Plate Labels
Ctrl + Shift + P	Plate Contours
]	Wall Labels
Ctrl + Shift + W	Wall Contours
\	Diaphragms
Modify Shortcut Keys	
C	Global Copy

Key Combination	Function
V	Global Move
S	Split member
A	Add Nodes (along members)
E	Extend member
T	Trim member
Ctrl + Shift + M	Model Merge
Ctrl + Shift + C	Match Properties
Del	Delete

Shortcut Keys available for Specific Windows

Key Combination	Model View Window	Spreadsheet
Ctrl + D	Open last graphic editing dialog (opens in Properties panel)	
Ctrl + A	Select all model elements	Select all cells
Ctrl + U	Unselect all model elements	Unselect all cells
Ctrl + F		Block Fill
Ctrl + C		Copy to clipboard
Ctrl + V		Paste from clipboard
Ctrl + M		Block Math
F2	Open Model View Settings	Start/Stop cell edit
F3		Insert line
F4		Delete Line
F5	Initiates the distance tool	Find/Replace
F8		Repeat current line
Shift + I		Filter Input
Shift + R		Filter Results

Hot Keys that open Spreadsheets

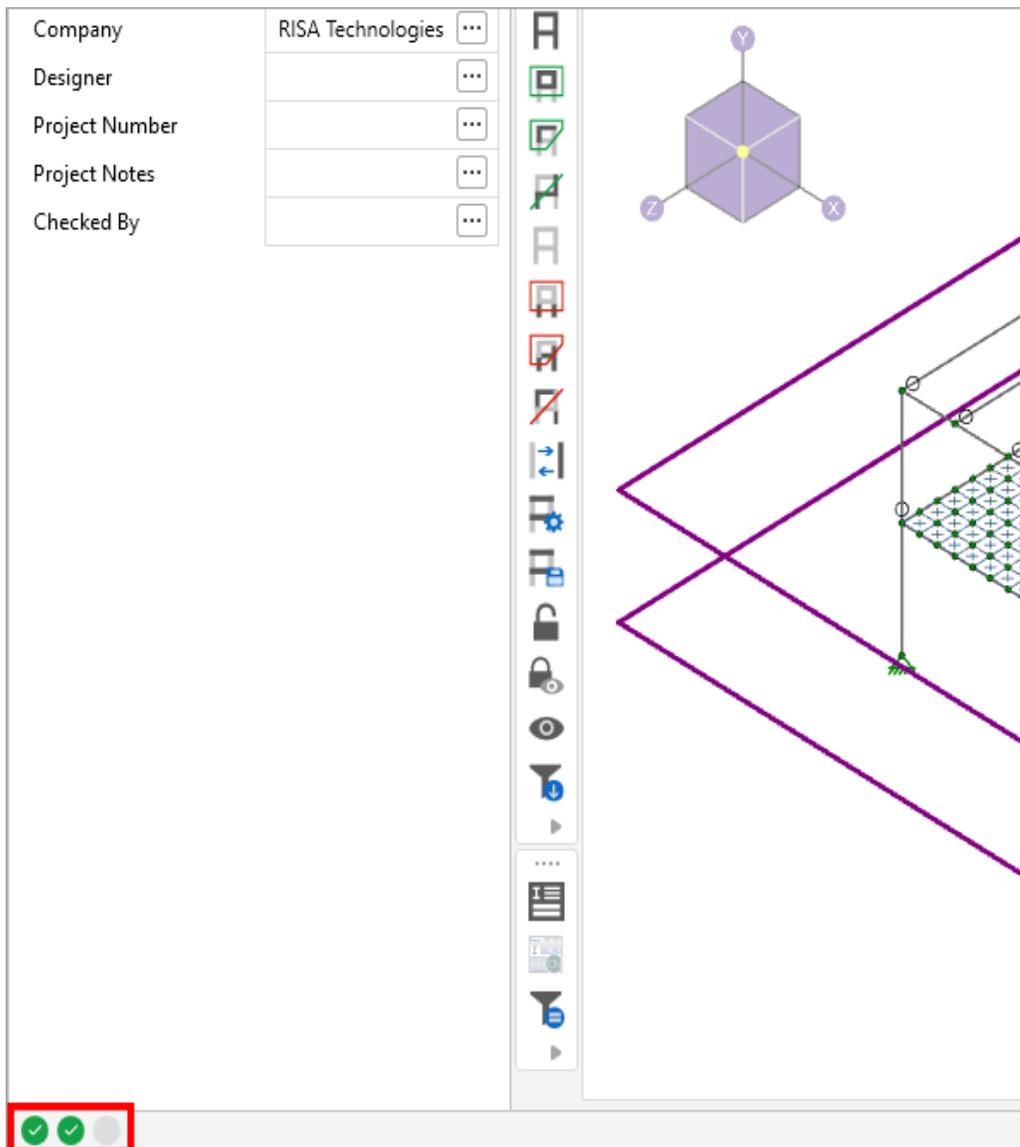
Key Combination	Unsolved Model	Solved Model
Ctrl + Alt + B	Basic Load cases	
Ctrl + Alt + C	Node Coordinates	Corner Forces
Ctrl + Alt + D	Member Distributed Loads	Node Deflections
Ctrl + Alt + E	Member Primary Data	Deflection Results
Ctrl + Alt + F		Member Forces
Ctrl + Alt + G	Model Settings	Model Settings
Ctrl + Alt + H		Suggested Shapes
Ctrl + Alt + I	Diaphragms	Member Torsion
Ctrl + Alt + J	Wall Panel Rules	Solid Principal Stress
Ctrl + Alt + K		Solid Stresses
Ctrl + Alt + L	Load Combinations	Plate Forces
Ctrl + Alt + M	Materials	Material Take Off
Ctrl + Alt + N	Joint Loads	Concrete Reinforcing
Ctrl + Alt + O	Boundary Conditions	Boundary Conditions
Ctrl + Alt + P	Member Point Loads	Plate Stresses
Ctrl + Alt + R	Design Code Check	Node Reactions
Ctrl + Alt + S	Section Sets	Member Stresses
Ctrl + Alt + T	Time History Loads	Story Drift
Ctrl + Alt + U	Seismic Design Rules	Code Check
Ctrl + Alt + V	Moving Loads	
Ctrl + Alt + X	Surface Loads	
Ctrl + Alt + Y	Dynamics Settings	
Ctrl + Alt + Z	Area Loads	
Ctrl + Alt + 4	Plates	

Status Bar

The **Status Bar** shows you the solution status of your model. It is comprised of three solution flags located along the very bottom of the application window, just beneath the workspace.



The first flag corresponds to a static analysis while the second flag corresponds to a dynamic analysis (eigensolution) and the third flag corresponds to a Response Spectra analysis. When the flags are gray (as shown in the image above), this means that a solution is not present. The flag turns green when a solution is present. In the following example, a static solution and eigensolution are present but a Response Spectra analysis solution is not available.



When the static solution flag turns yellow, this indicates that a static solution is present however instability have been detected.



For more information on instabilities see the [Stability](#) topic.

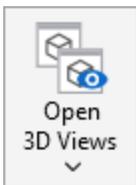
Windows Behavior

Modeling the structure takes place within model views and spreadsheets, each in their own window that may be moved around the workspace and sized as you wish. The ability to have multiple model views and multiple spreadsheets open at one time is a powerful feature. You can move the windows around by clicking the 3D View tab and dragging as desired.

These windows all contain the **x**  in the upper left corner to help you close the window. There are also scroll boxes to help you view information that is outside of the window viewing area. Click the scroll bar buttons or drag the scroll box to advance the display in one direction or another.

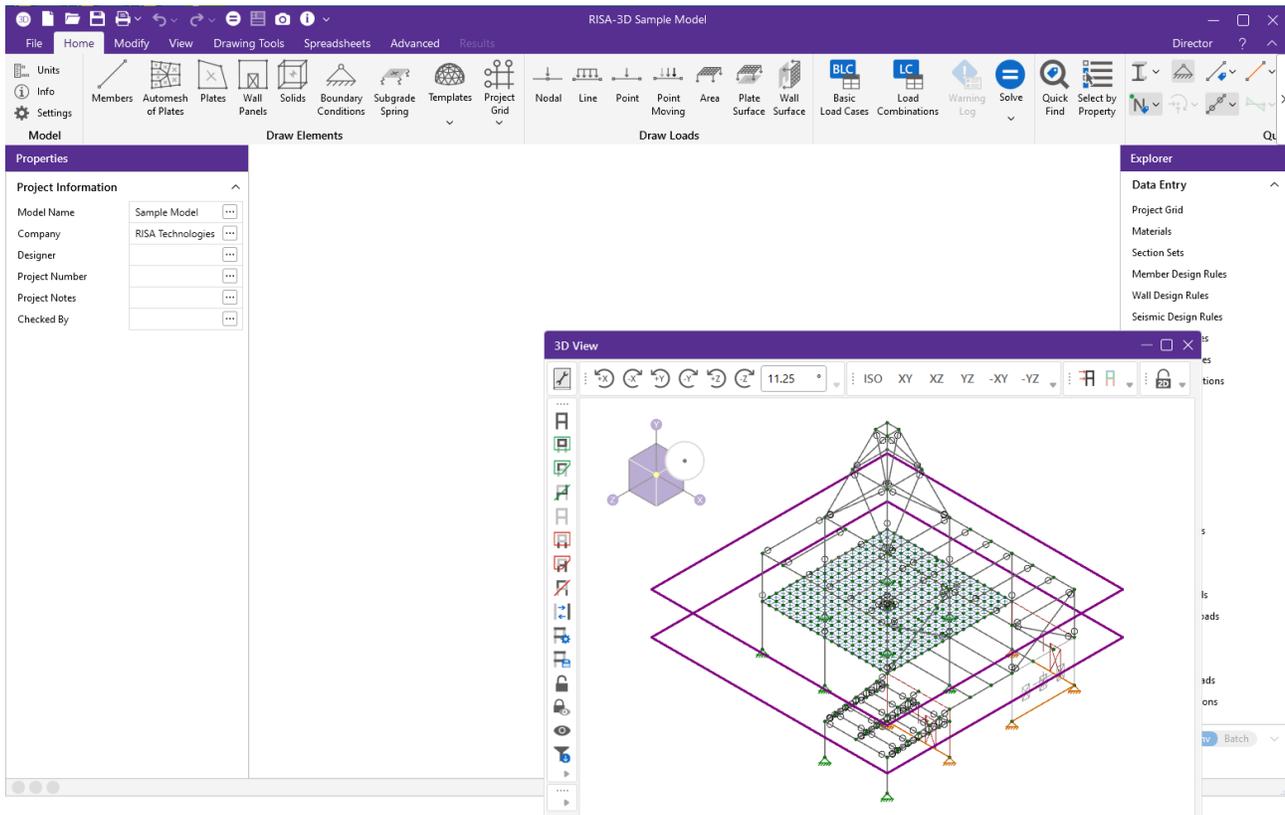
Model Views

Model View windows show a graphic view of the model. Open a new view with the **Open 3D Views** button (shown below).



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

One thing to remember is that the toolbars that are displayed depends upon what window is active. The active window is the one with the highlighted purple bar. An inactive window will be grayed out as shown below where 3D View 2 is the active window.



For more information on this view [Window Behavior](#) under Graphics Display.

Spreadsheets

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

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

Dialogs

A **Dialog** is a third type of window and is used to access a specific function within the program. Another powerful feature is that most of the dialogs may be left open while you edit the model, making it easy to make adjustments as you work. You will find that dialogs are very easy to work with. There are **Help** buttons that will bring you directly to the relevant topic in the help file.

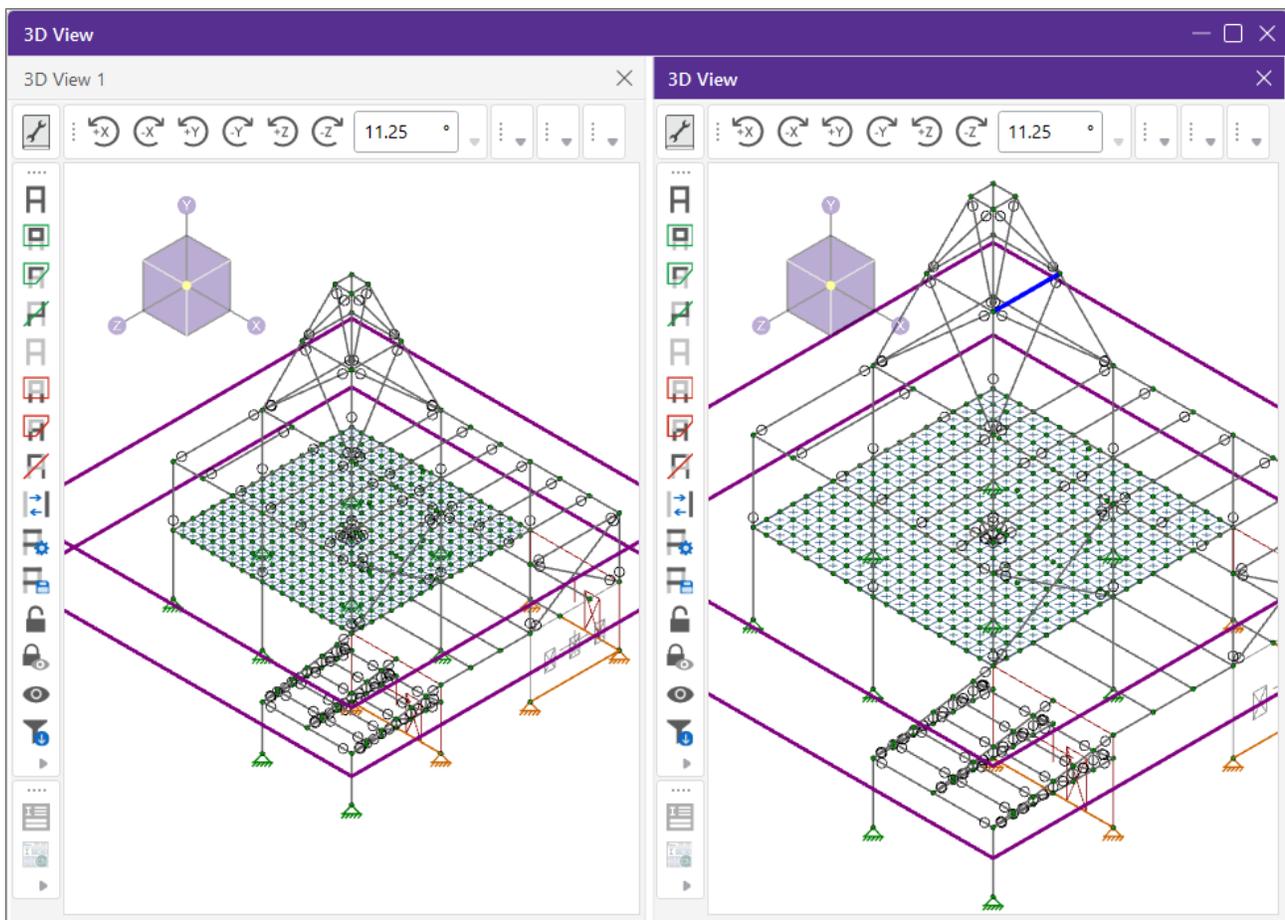
Window Tiling

Standard window tilings help you set up your workspace. With any 3D view or spreadsheet, you can click the tab and drag the window into a configuration that best suits your preferences. When dragging a window you will see a circle with four arrows.



You can place the windows over each of these arrows to designate where to place the window. Placing the window over the center dot will make the window cover the entire workspace between the **Properties** and **Explorer** panels.

By placing windows over the dot within other windows, the windows tab together. See below example where 3D View 2 and 3D View 3 have been tabbed together.



The best way to learn how the tiling works within the program is to try it out.

For additional information on working with viewpoint windows, see the [Windows Behavior](#) section under Graphic Display.

For information about working with multiple views, see the [Multiple Windows](#) section under Graphic Display, Window Behavior.

Modes

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

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

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

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

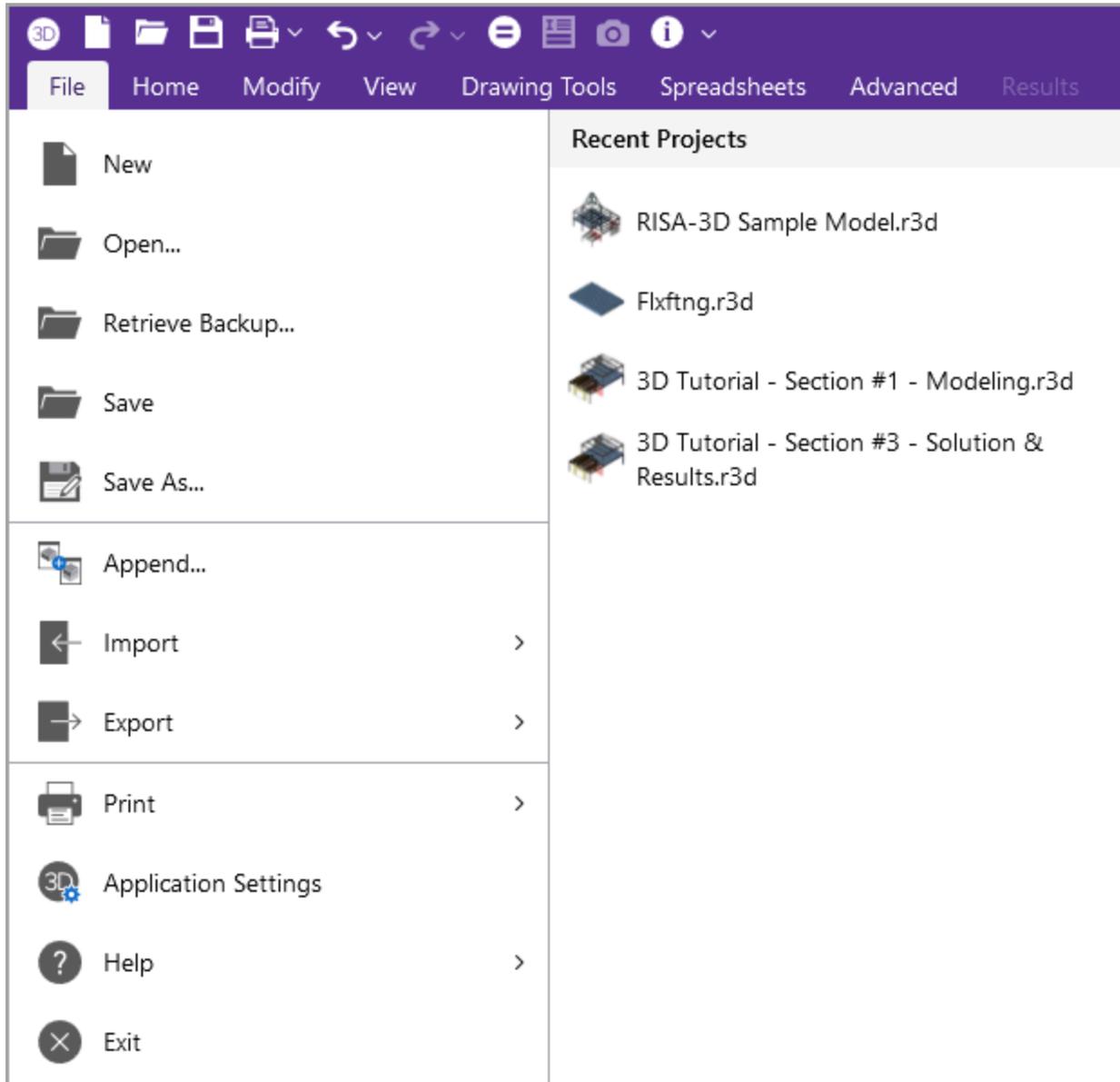
Edit Mode is the lower level mode that allows you to graphically edit the model. You may make selections and adjust the view while in the edit mode such that when you are finished selecting you will be returned to the **Edit Mode**. Some **Edit Mode** features have options on how you apply the edit.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



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

File Menu

Clicking the File tab will expand the File menu.



To the right of the File Menu are recently opened files with a preview of the model file. Selecting one of these files will close the current file, prompting for saving if necessary, and will open the selected file.

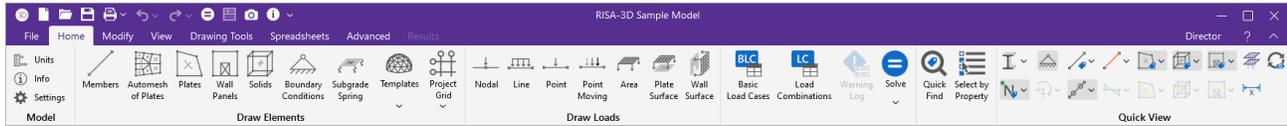
The following table provides descriptions of all the menu options.

File Menu Icons / Options

Icon	Option	Description
	New	Allows you to close the current file, prompting you to save it, and opens a new file.
	Open	Allows you to close the current file, prompting you to save it, and opens an existing file.
	Retrieve Backup	Allows you to close the current file, prompting you to save it, and allows you to open an available backup file.
	Save	Allows you to save the current file, prompting you for a name if it's a new file.
	Save As	Allows you to save a copy of the current file as a new file, prompting you for a name.
	Append	Inserts another RISA-3D model into the current model.
	Import	Allows you to close the current file, prompting you to save it if applicable, and opens an existing RISA-2D (R2D), DXF, STD, Autodesk Revit, Tekla Structures, tnxTower, or SAF file. For more information on how RISA-3D interacts with other programs, see Appendix E - Interfacing with Other Programs .
	Export	Allows you to export the current file as a DXF, ADAPT Builder, SAF, SDS2 End Reactions, or SDNF file. For more information on the interaction between RISA and other programs refer to Appendix E .
	Print	Allows you to access RISA-3D printing options.
	Application Settings	Allows you to modify several program options. Refer to the Customizing RISA-3D topic for more information.
	Help	Opens a menu that displays RISA-3D Help options.
	RISA-3D Manual	Available from the 'Help' sub-menu, this icon allows you to open a PDF copy of the RISA-3D Manual.
	RISA-3D Tutorial	Available from the 'Help' sub-menu, this icon allows you to access all available RISA-3D tutorials.
	Licensing	Available from the 'Help' sub-menu, this icon provides license information for subscription users and provides the option to Borrow or Return a license. Additional subscription licensing information can be found here: Subscription Licensing .
	Check for Updates	Available from the 'Help' sub-menu, this icon allows you to run an internal check for possible program updates. If your program is up to date, you receive a message saying you are up to date. If you are not using the current version of the software, the check offers you options of how to update.

Icon	Option	Description
	About RISA-3D	Available from the 'Help' sub-menu, this icon allows you to view your RISA-3D version and hardware key information.
	Switch User Credentials & Exit	Available from the 'Help' sub-menu, this icon allows you to log out your Subscription credentials and closes RISA-3D, prompting you to save if applicable. The next time the program is opened you need to re-enter your subscription license username and password.
	Exit	Allows you to close RISA-3D, prompting you to save if applicable.

Home Ribbon



The **Home** ribbon offers various tools for setting up your personal preferences in the application interface and the basics on models. The following table provides a short description of each tool.

Icon	Tool	Description
Model		
	Units	Allows you to define the units to be used for the input and output data of the model. See Units for more information.
	Info	Allows you to enter descriptive information such as a title for the particular model being defined, the name of the designer and a job number. Enter descriptive information such as company and designer name. This information displays when printing report and graphic output. Some information can also be saved using the 'Set As Default' icon. See Model Settings for more information.
	Settings	Allows you to modify settings specific to the model which is currently open. Designate the code standard to use for the analysis, input criterion for concrete and seismic analysis and adjust parameters used in the solution such as, mesh size, global axis control and processor utilization. Information can also be saved using the Set As Default button. See Model Settings for more information.
Draw Elements		
 Members	Members	Allows you to graphically draw members within your model. See Members for more information. Designate general properties such as material and shape, and specify design parameters such as unbraced length. Then, draw the member in your model at the desired location.
 Automesh of Plates	Automesh of Plates	Allows you to draw a polygon that will automatically submesh into smaller <u>quadrilateral</u> plate elements. See Submeshing Plates for more information. Select a general material, thickness and maximum edge dimension to assign the plates. More advanced plate settings may also be selected. Then, draw the plates in your model at the desired locations.

Icon	Tool	Description
 Plates	Plates	<p>Allows you to graphically draw plates in your model. See Plates for more information.</p> <p>Select a general material and thickness to assign the plates. More advanced plate settings for end releases and plate formulation may also be selected. Then, draw the plates in your model at the desired locations.</p>
 Wall Panels	Wall Panels	<p>Allows you to graphically draw wall panels in your model. See Wall Panels for more information.</p> <p>Select a wall material and thickness. Wall design rules may also be selected. Then, draw walls in your model at the desired locations. Double-click the wall panel to modify openings and boundary conditions.</p>
 Solids	Solids	<p>Allows you to extrude plate elements into solid elements. See Solids for more information.</p> <p>Solids are created by extruding an existing plate element in any of the three global directions (X, Y, Z). Enter your extrude dimension and then apply to one or all selected plates.</p>
 Boundary Conditions	Boundary Conditions	<p>Allows you to apply boundary conditions in your model. See Boundary Conditions for more information.</p> <p>All models must be attached to some external point(s) of support. The support point(s) can be completely restrained or partially restrained, by using a Spring with a specified stiffness.</p>
 Subgrade Spring	Subgrade Spring	<p>Allows you to generate compression-only springs in the vertical direction at all plate joints in the selected group of plates.</p> <p>Designate a subgrade modulus to assign to the plates. Then, draw the springs in your model at the desired locations. See Generate Soil Springs for more information.</p>
 Templates ▼	Templates	<p>Allows you to automatically generate typical structures or portions of structures to start a new model or add to an existing model. See Generation for more information.</p> <p>Enter specific geometric parameters based on the template. Specifying member or plate material properties is optional. When the member or plate option is unchecked, only joints will be generated.</p>
 Project Grid ▼	Project Grid	<p>Allows you add project grids to your model file.</p> <p>Project grids are used as a reference to draw new elements in the model. Columns and walls are attached to project grids and move with the project grid. See Project Grid for more information.</p>

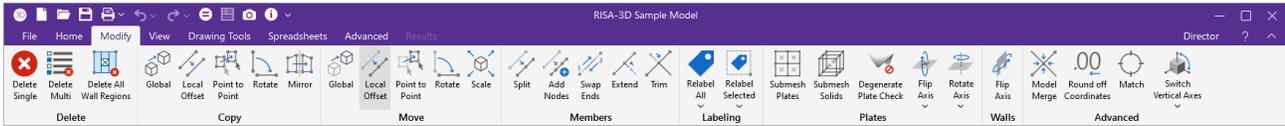
Icon	Tool	Description
Draw Loads		
 Nodal	Nodal	<p>Allows you to graphically draw concentrated loads at nodal points.</p> <p>Nodal loads may be applied at node locations only. If nodal load is being added to any member span location, a new node must be added to that location prior to applying nodal load. See Nodal Load for more information.</p>
 Line	Line	<p>Allows you to graphically draw distributed loads in your model.</p> <p>Click on the member or wall panel edge to apply a line load. The load can be applied to the entire length or partial length and can vary in magnitude from start to end. See Distributed Load for more information.</p>
 Point	Point	<p>Allows you to graphically draw concentrated loads on member elements in your model.</p> <p>point loads may be applied to a specific location along a member length or wall edge. A node is not required at the point of application. The point location may be entered in terms of length or as a percentage of the element length. See Point Loads for more information.</p>
 Point Moving	Point Moving	<p>Allows you to create point loads based on a specific time step of a defined moving load pattern in your model.</p> <p>Successive point loads are applied to the member at the specified increment and pattern designated in the Moving Loads spreadsheet. See Moving Loads for more information.</p>
 Area	Area	<p>Allows you to graphically draw area loads in your model.</p> <p>Click on points to define the area to apply the area load. Nodes are not required at the application point. The load direction determines how the load is distributed to supporting elements. See Area Loads for more information.</p>
 Plate Surface	Plate Surface	<p>Allows you to graphically draw plate surface loads.</p> <p>Apply surface loads to plates by entering in the load direction and magnitude. Apply the load to all selected plates or individually click on the plates in the model to apply the load to. See Surface Loads for more information.</p>
 Wall Surface	Wall Surface	<p>Allows you to graphically draw wall surface loads.</p> <p>Apply surface loads to wall panels by entering in the load direction and magnitude. Apply the load to all selected wall panels or individually click on the wall panels in the model to apply the load to. See the Wall Surface Load topic for more information.</p>

Icon	Tool	Description
Miscellaneous		
These tools allow you to access load cases and combinations, and run a solution which will reveal errors if any, in a warning log.		
	Basic Load Cases	<p>Opens the Basic Load Cases spreadsheet.</p> <p>The Basic Load Case spreadsheet groups and summarizes the loads applied to the model. See Basic Load Cases for more information.</p>
	Load Combinations	<p>Opens the Load Combinations spreadsheet.</p> <p>The Load Combination Spreadsheet allows you to solve models for combined basic load cases with factors applied. See Load Combination for more information.</p>
	Warning Log	<p>Opens the Warning Log spreadsheet if warnings are available after solution. See the Warning Log topic for more information.</p>
	Solve	<p>Performs the analysis and design calculations to solve the current model file.</p> <p>Solution options include single load combinations, all selected load combinations (batch), only the governing cases (envelope), designing connections via RISACONNECTION (if applicable) and/or performing a dynamic analysis. See Solution for more information.</p>
	Quick Find	<p>Allows you to search for specific elements within the model.</p> <p>Quick Find performs a quick search to find specific elements in your model by label and add them to the selection state. See Graphic Selection for more information.</p>
	Select by Property	<p>Allows you to select items within the model based on specific criteria.</p> <p>Select by Property selects or unselected elements based on various parameters for either nodes, members, plates, wall panels, solids, loads or coordinates. See Graphic Selection for more information.</p>
Quick View		
These tools can be used to change the graphical display of the model. You can use these icon to turn on member and node labels, and view color-coded properties and results. Additional view options can be specified in the View Settings on the 'View' ribbon.		
	Render	<p>Allows you to toggle through wireframe, color coded wireframe and rendering of model elements.</p> <p>Rendering is dependent on material type. The color coding</p>

Icon	Tool	Description												
		<p>criteria can be set in the Model View Settings. By default, the elements are designated based on section set.</p> <table border="1"> <thead> <tr> <th>Icon</th> <th>Option</th> <th>Description</th> </tr> </thead> <tbody> <tr> <td></td> <td>WireFrame</td> <td>Allows you to display elements as wireframe.</td> </tr> <tr> <td></td> <td>Color Coded</td> <td>Allows you to display elements as wireframe and color coded based on the Model Settings.</td> </tr> <tr> <td></td> <td>Rendered</td> <td>Allows you to render elements based on material type.</td> </tr> </tbody> </table> <p>See Model View Settings for more information on rendering.</p>	Icon	Option	Description		WireFrame	Allows you to display elements as wireframe.		Color Coded	Allows you to display elements as wireframe and color coded based on the Model Settings.		Rendered	Allows you to render elements based on material type.
Icon	Option	Description												
	WireFrame	Allows you to display elements as wireframe.												
	Color Coded	Allows you to display elements as wireframe and color coded based on the Model Settings.												
	Rendered	Allows you to render elements based on material type.												
	Show Node Labels	<p>Toggles through the display of nodes and node labels.</p> <p>Node labels and corresponding node locations can be seen in the Node Coordinates spreadsheet. You can also relabel all nodes from the Node Coordinates spreadsheet.</p>												
	View Boundary Conditions	<p>Allows you to toggle the boundary conditions.</p> <p>Click to turn the graphical display of applied boundary conditions on or off.</p>												
	Node Reactions	<p>Displays model reactions at nodes with boundary conditions.</p> <p>View the model reactions for the specified load combination or enveloped solution. Display options include the global X, Y, Z directions as well as X-axis, Y-axis and Z-axis moments.</p>												
	Member Labels	<p>Allows you to display member labels when viewing members in the wireframe.</p> <p>Label options include member properties such as shape and material type, as well as design properties such as unbraced length.</p>												
	Member Ends	<p>Displays various member end parameters graphically.</p> <p>Display options include member end offsets, end releases, connection rules, and I/J labels.</p>												
	Member Color Codes	<p>Allows you to toggle the display of the various member parameters using color coding options.</p>												
	View Member Forces	<p>Displays the member force diagrams.</p> <p>Click to turn on the graphical display of the member force</p>												

Icon	Tool	Description
		diagrams. Display options include none, axial, shear, torque, and bending moment diagrams.
	Plate Labels	Allows you to toggle the display of plate labels. View the plate labels as text. Other label display options include plate properties such as material, thickness and local axis.
	Plate Contours	Allows you to toggle the display of plate contours. Use the Contour Legend in the top right of the Model View to view plate forces or stresses. See Plates for more information.
	Solid Labels	Allows you to toggle the display of solid labels. View the solid labels as text. Other label display options include solid material type as well as identifiers such as label and number.
	Solid Contours	Allows you to toggle the display of solid stress contours. Use the Contour Legend in the top right of the Model View, to view solid stresses. See Solids for more information.
	Wall Panel Labels	Allows you to toggle the display of wall panel labels.
	Wall Panel Contours	Allows you to toggle the display the wall panel force results as color contours. Use the Contour Legend in the top right of the Model View to view wall panel forces and stresses. See Wall Panels for more information.
	Diaphragms	Toggles the display of diaphragms.
	Distance Measure Tool	Allows you to measure the distance between two points. Click on two points in the model to measure the distance. The measurement result is displayed in the bottom-left of the Properties Panel.
	Reset View	Clears the currently displayed options and sets the model view back to the default wireframe view.

Modify Ribbon



The **Modify** ribbon offers various tools to modify items and elements in a model. The following table provides a short description of each tool.

Icon	Tool	Description
Delete		
These tools provide a way to delete specific elements in your model. See Deleting Elements in Graphic Editing for more information.		
 Delete Single	Delete Single	Allows you to delete a single item in the model. Delete a single item by clicking on it individually in the model view. Alternatively, you can select an element and press the DELETE key on your keyboard
 Delete Multi	Delete Multi	Allows you to delete multiple items in the model at once. Deleted selected items based on elements types such as, nodes, unattached nodes, members, zero length members, plates, wall panels, displayed loads, solids or wall boundary conditions and straps.
 Delete All Wall Regions	Delete All Wall Regions	Allows you to delete all existing wall items in the model at once. This tool clears all existing wall regions in the model that have been manually drawn in or that have been generated automatically by the program.
Copy		
These tools provide various ways to copy elements within the model. See Copying Elements in Graphic Editing for more information.		
 Global	Global	Allows you to copy elements in the global X, Y and/or Z direction. Specify a distance to copy an elements in the X, Y and/or Z direction. Click diagonal to apply the distances simultaneously and/or click Copy Loads to also copy associated member loads in the duplicate.

Icon	Tool	Description
 Local Offset	Local Offset	Allows you to copy elements by specifying an off-set position. Specify the direction along a global or local axis and offset distance to copy elements accordingly.
 Point to Point	Point to Point	Allows you to copy selected elements by specifying two points for distance and direction. Specify a start point and end point for copying selected elements, in the direction and the distance between the two specified points.
 Rotate	Rotate	Allows you to copy elements by specifying a rotation. Specify the axis of rotation, angle and how many increments to create rotational copies of the element.
 Mirror	Mirror	Allows you to copy elements by mirroring them to a specific plane. Specify a plane to mirror selected elements and plane location to create a mirror image copy of the elements.
Move		
These tools provide various way to move or rotate elements within the model. See Moving Elements in Graphic Editing for more information.		
 Global	Global	Allows you to move elements in the X, Y and/or Z direction. Specify the distance to move an element in the global X, Y and Z direction.
 Local Offset	Local Offset	Allows you to move elements by specifying an off-set position. Specify a distance to move an elements along the local x, y and z axes.
 Point to Point	Point to Point	Allows you to move selected elements by specifying two points for distance and direction. Specify a start point and end point for moving selected elements in the direction and the distance between the two specified points.
 Rotate	Rotate	Allows you to move elements by specifying a rotation. First, specify a rotation axis and angle of rotation. Then, specify an original by clicking a node in the model or by entering specific coordinates.
 Scale	Scale	Allows you to scale elements based on specific factors. Specify the factors used to scale the elements in your model. Then, specify an origin by clicking a node in the

Icon	Tool	Description
		model or by entering specific coordinates.
Members		
These are graphical tools that provide ways to modify members within the model. See Trim/Extend , Swap Member Ends , and Split Members for more information.		
 Split	Split	Allows you to physically split a member into smaller members.
 Add Nodes	Add Nodes	Allows you to add additional nodes along the length of a member. Specify the spacing type and number of segments or location to add nodes.
 Swap Ends	Swap Ends	Allows you to swap the member I End with the member J End.
 Extend	Extend	Allows you to extend the member in relation to another member. First, select the member you want to extend. Then, select the member you want to extend to.
 Trim	Trim	Allows you to trim the member in relation to another member. First, select the side of the member you want to trim. Then, select the boundary member to trim it back to.
Labeling		
These tools provide ways to control the labels of elements within the model. See Graphic Editing for more information.		
 Relabel All ▼	Relabel All	Allows you to relabel all elements (Nodes, Members, Plates, Walls, Panels and Solids). Use the drop-down menu to select which type of element to relabel.
 Relabel Selected ▼	Relabel Selected	Allows you to relabel all selected elements (Nodes, Members, Plates, Walls, Panels and Solids). Use the drop-down menu to select which type of element to relabel and apply this to only selected elements.
Plates		
These tools provide ways to modify both plates and solid elements. See Plates and Solids for		

Icon	Tool	Description
more information.		
 Submesh Plates	Submesh Plates	<p>Allows you to refine the existing plate mesh by choosing a submesh method: auto, quad or tri.</p> <p>Select a submesh method to apply to existing plates. The original plate will be replaced with the refined submesh consisting of smaller plate elements.</p>
 Submesh Solids	Submesh Solids	<p>Allows you to refine the existing solid elements into smaller solid elements.</p> <p>Select the number of pieces along each side of the solid to specify how to submesh the existing solid elements.</p>
 Degenerate Plate Check	Degenerate Plate Check	<p>Allows you to check for problematic plates that cause errors during solution.</p> <p>The option 'Show Problematic Plates' identifies the plates that have poor geometry or are not planar. The second option, 'Split Problematic Plates', creates four triangular plates out of any non-planar plates.</p>
 Flip Axis ▼	Flip Axis	<p>Allows you to switch the plate vertical local axis orientation.</p> <p>Plates are oriented by the direction and sequence in which their corner nodes were defined. Switching the vertical axis may be useful to better align the plate local axes.</p>
 Rotate Axis ▼	Rotate Axis	<p>Allows you to rotate the plate local axis from its default position.</p> <p>Use this tool to rotate the plates local axis about its local z axis.</p>
Walls		
 Flip Axis	Flip Axis	<p>Allows you to flip the wall panel local axes, which reverses the direction of the wall panel's local x and z-axes. You can choose to flip the wall and the openings or flip just the wall's local axis without affecting the openings or regions. This feature is useful for aligning the positive z-direction for a group of wall panels that all receive the same surface load. See Walls for more information.</p>
Advanced		
 Model Merge	Model Merge	<p>Allows you to merge elements in your model based on your merge tolerance. See Model Merge for more information.</p> <p>Model Merge detects unconnected joints along member spans, unconnected crossing members and duplicate joints, members and plates. Model Merge can help detect and fix modeling errors.</p>

Icon	Tool	Description
 Round off Coordinates	Round off Coordinates	<p>Allows you to round off the node coordinates in your model.</p> <p>Use this tool to indicate the decimal precision used for node coordinates. This is especially useful when importing a model from another program, to reduce the possibility of element misalignment due to excessive decimal precision.</p>
 Match	Match	<p>Allows you to match the properties of one member to another. See Match for more information.</p> <p>Select a base member whose properties you want copied to additional members. This tool is useful when making mass modifications to the model.</p>
 Switch Vertical Axes ▼	Switch Vertical Axes	<p>Allows you to change the vertical axis of the model.</p> <p>The program defaults to use Y as the global vertical axis, although you may use this tool to switch this at anytime. If you change the global vertical axis, the orientation of all elements and loads will also be rotated.</p>

View Ribbon

The **View** ribbon offers various tools to change the model view, display of loads and results, and active windows.



The following table provides a short description of each tool.

View - Icons

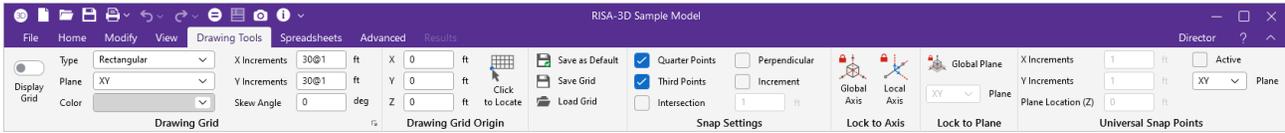
Icon	Tool	Description
 Project Grid	Project Grid	Allows you to toggle the display of the project grid on or off. The Project Grid is part of the model and can be allowed to move model elements as grid lines are moved. Create new project gridlines from the Home tab under Draw Elements. Modify existing project gridlines using the Project Grid spreadsheet. Columns and walls are attached to project grids and move with the project grid. See Project Grid for more information.
3D View		
Tools for snapping your model to a specific view.		
 Isometric	Isometric	Allows you to snap your model to an isometric view.
 XY	XY	Allows you to snap your model to an XY planar view.
 XZ	XZ	Allows you to snap your model to an XZ planar view.
 YZ	YZ	Allows you to snap your model to an YZ planar view.
View Settings		
These tools let you specify how to plot nodes, members, plates, loads and the information that is to accompany them.		
 Model	Model	Allows you to specify model display settings. Set display settings for the model display using dialog selection options. Customize the display settings for members, plates, wall panels or solids. See Model View Settings for more information.

Icon	Tool	Description
 Results	Results	<p>Allows you to specify result display settings.</p> <p>Set display settings for the results of members, plates, wall panels or solids. In addition, the display of reactions or deflections can be adjusted. See Results View Settings for more information.</p>
Animate		
 Results	Results	<p>Allows you to view an animation of the model's deflected shape, mode shapes, or time history loads.</p> <p>A new model view window will open displaying a video animation of the model's behavior. See Animate Results for more information..</p> <p>Additional controls are included for the animation of Time History motion. See Animate Time History Deflection for more information.</p>
 Moving Loads	Moving Loads	<p>Allows you to animate Moving Loads in the current model view.</p> <p>It displays the model view with video animation that includes the moving loads.</p> <p>See Animate a Moving Load for more information.</p>
Zoom		
 Zoom Extents	Zoom Extents	<p>Allows you to reset the view to display the full model. See Zoom Extent - tool for more information.</p>
 Zoom In	Zoom In	<p>Allows you to zoom the view in for a closer view of the model.</p> <p>This tool sets the Zoom to "In" so that each time you click you mouse button, the model view is reduced. See Zoom In - tool for more information.</p>
 Zoom Out	Zoom Out	<p>Allows you to zoom the view out for a wider view of the model.</p> <p>This tool sets the Zoom to "Out" so that each time you click you mouse button, the model view is diminished. See Zoom Out - tool for more information.</p>
 Zoom Box	Zoom Box	<p>Allows you to zoom in on a specific area of the view using a box selection. See Zoom Box - tool for more information.</p>
Miscellaneous		

Icon	Tool	Description
 Image Capture	Image Capture	<p>Allows you to create custom snapshot images of model views and detail reports that can be added to a printed report. See Capture a Custom Snapshot for more information.</p> <p>See Printing for more information on printing your image captures.</p>
 Saved Views	Saved Views	<p>Allows you to save or recall specific selection states for a model view. See View a Saved Snapshot for more information.</p>
Window		
 Open 3D Views	Open 3D View	<p>Allows you to add a new 3D View window or choose from a list of all other 3D View windows that are currently open. See Open a New 3D View for more information.</p>

For more details about the various options on the View ribbon, see [Graphic Display](#) .

Drawing Tools Ribbon



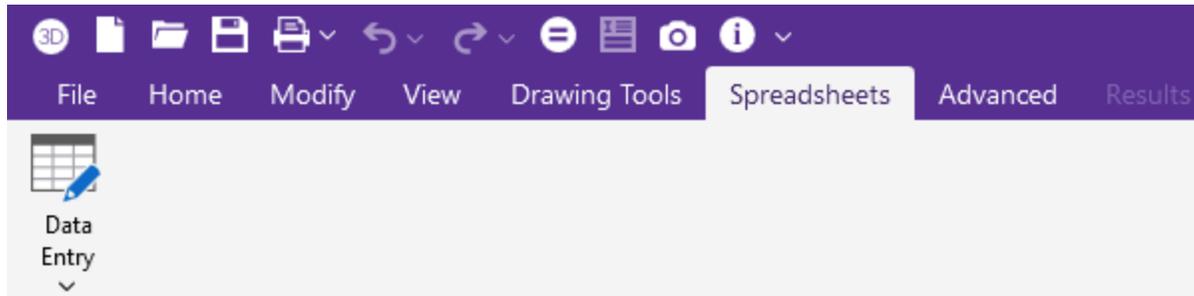
The **Drawing Tools** ribbon provides options to aid in drawing the elements in the model.

Icon	Tool	Description
Drawing Grid		
	Display Grid	Allows you to toggle the display of the drawing grid.
	Import .dxf Grid	Allows you to import a DXF file to be used for a Drawing Grid. Change the drawing grid type to DXF Underlay to utilize this feature.
	Color/Layer Options	Allows you to select which layers of the DXF file to display and what colors to display them in. Change the drawing grid type to DXF Underlay to utilize this feature.
Drawing Grid Origin		
	Click to Locate	Allows you to change the origin of the drawing grid by selecting a new point.
Drawing Grid Options		
	Save as Default	Allows you to save the existing drawing grid as the default grid.
	Save Grid	Allows you to save the existing drawing grid.
	Load Grid	Allows you to load a previously saved drawing grid.
Snap Settings		
<input type="checkbox"/>	Quarter Points	Allows you to create snap points at quarter intervals along a member. A check indicates that this feature is turned on. To check or uncheck, click inside the Quarter Points box.
<input type="checkbox"/>	Third Points	Allows you to create snap points at third intervals along a member.

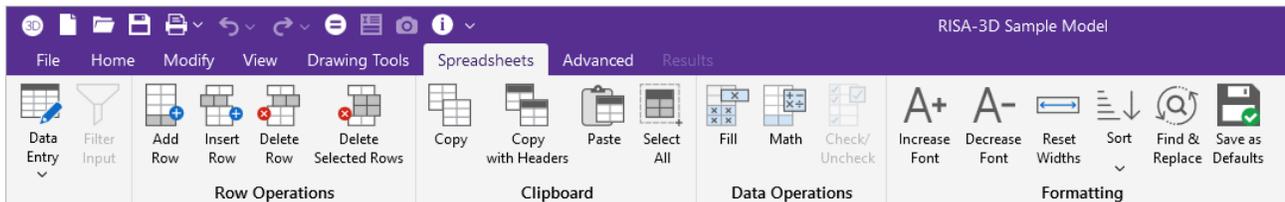
Icon	Tool	Description
		A check indicates that this feature is turned on. To check or uncheck the box, click inside it.
<input type="checkbox"/>	Intersection	Allows you to create a snap point where elements intersect. A check indicates that this feature is turned on. To check or uncheck the box, click inside it.
<input type="checkbox"/>	Increment	Allows you to create snap points based on distance specified. A check indicates that this feature is turned on. To check or uncheck the box, click inside it.
<input type="checkbox"/>	Perpendicular	Allows you to create a snap point to draw perpendicular members. A check indicates that this feature is turned on. To check or uncheck the box, click inside it.
Lock Axis		
 Global Axis	Global Axis	Allows you to lock drawing tools to the global axis specified. When using drawing tools and clicking on an initial point in the model, subsequent click points will only be available within the global axis specified.
 Local Axis	Local Axis	Allows you to lock drawing tools to the local axis specified. When using drawing tools and clicking on an initial point in the model, subsequent click points will only be available within the local axis specified.
Lock to Plane		
 Global Plane	Global Plane	Allows you to lock drawing tools to the global plane specified. When using drawing tools and clicking on an initial point in the model, subsequent click points only be available within the global plane specified.
Universal Snap Points		
<input type="checkbox"/>	Active	Allows you to activate snap points in the model space based on universal snap point options. A check indicates that this feature is turned on. To check or uncheck the box, click inside it.

For more details about the various options on the Drawing Tools ribbon, see [Graphic Editing](#).

Spreadsheet Ribbon



The **Spreadsheets** ribbon provides a way to access all the Data Entry spreadsheets in the model, as well as provide various tools to manipulate spreadsheet values. If you do not have a spreadsheet open, the only option on this ribbon is the Data Entry button, which allows you to open a spreadsheet. Once you open a spreadsheet, you have various tools, pertinent to that spreadsheet, available to you. For example, if you open the Materials spreadsheet, the Spreadsheets ribbon looks like the following image.



The following table provides a list of the various tools on the Spreadsheets ribbon (when a spreadsheet is opened) and their descriptions.

Icon	Tool	Description
 Data Entry	Data Entry	Allows you to select to open one or more spreadsheets from a list of all the Data Entry spreadsheets. The tool displays a list of all the Data Entry spreadsheets and whether or not they are currently open. A check in the checkbox indicates that the spreadsheet is open.
 Filter Input	Filter Input	Allows you to filter graphically selected elements in the Data Entry spreadsheets. Choose to filter selected or unselected elements from the Data Entry spreadsheets. This is useful when you want to focus on the inputs for a specific portion of the model.

Row Operations

These tools are available only if a Data Entry spreadsheet is open.

Icon	Tool	Description
 Add Row	Add Row	Allows you to add a single row. The tool adds a new, single row to the bottom of the spreadsheet.
 Insert Row	Insert Row	Allows you to insert a single row. The tool inserts the row above the currently selected row in the spreadsheet.
 Delete Row	Delete Row	Allows you to remove a single row. The tool deletes the selected, single row from the spreadsheet.
 Delete Selected Rows	Delete Selected Rows	Allows you to remove several rows at once. The tool deletes all selected rows from the spreadsheet.
Clipboard		
These tools are available only if a spreadsheet is open.		
 Copy	Copy	Allows you to copy spreadsheet information from one or more cells. The tool copies the information in the selected cells which can then be pasted into other cells or outside of the program.
 Copy with Headers	Copy with Headers	Allows you to copy spreadsheet information and with the column header descriptions. Choose this tool to include the column header descriptions with the spreadsheet information when pasting outside of the program.
 Paste	Paste	Allows you to paste spreadsheet information into selected cells. Once data has been copied to the clipboard, the data can be pasted into the selected cells or outside of the program.
 Select All	Select All	Allows you to select all cells at once. Choose this tool to select all data in the spreadsheet.
Results		
These tools are only available if a Results spreadsheet is open.		

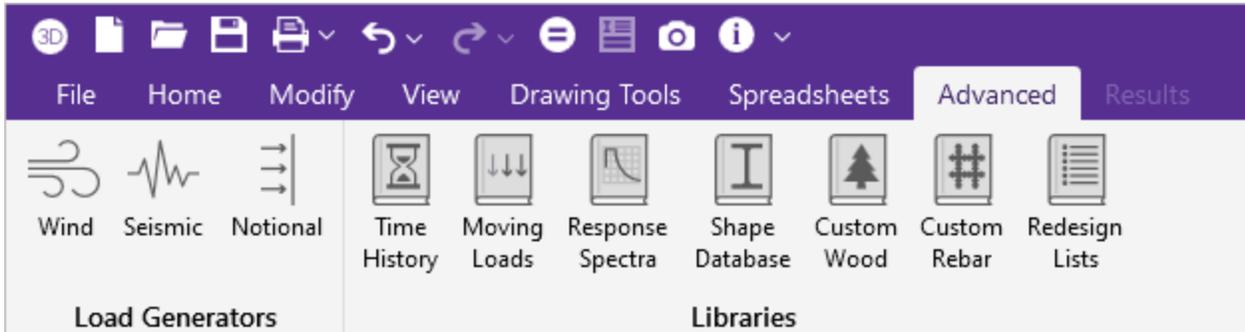
Icon	Tool	Description
 Ignore After	Ignore After	Allows you to exclude all results after the current items. The tool removes spreadsheet information for the results after the current item selected in the active spreadsheet, and reduces the size of the results output. These elements are still present in the analysis.
 Show All	Show All	Allows you to includes all results in the spreadsheet. The tool includes all previously excluded results in the spreadsheet.
 Clear	Clear Results	Allows you to delete all solution results.
Data Operations These tools are available only if a Data Entry spreadsheet is open.		
 Fill	Fill	Allows you to fill all selected cells with a common value. Select the cells you want to fill, then type in the value to fill them with.
 Math	Math	Allows you to perform math on the currently selected cell(s). You can add, subtract, multiple or divide the cell(s) by a specific operand.
 Check/ Uncheck	Check/Uncheck	Allows you to check or uncheck selected cells in a Results spreadsheet. Select any cells containing a checkbox in the spreadsheet to check/uncheck. The tool reverses the existing condition.
Formatting These tools are available only if a spreadsheet is open.		
 Increase Font	Increase Font	Allows you to increase the size of the font in the spreadsheet.
 Decrease Font	Decrease Font	Allows you to decrease the size of the font in the spreadsheet.
 Reset Widths	Recent Widths	Allows you to reset the widths of the columns in the spreadsheet. When selected, this tool resets the widths of the columns in the spreadsheet to the program defaults.

Icon	Tool	Description
 Sort ▼	Sort	<p>Allows you to sort information/data by a selected column in the spreadsheet.</p> <p>Select a column header in a spreadsheet and click and then choose high to low or low to high, to reorganize the spreadsheet based on the sort selection in that column.</p>
 Find & Replace	Find & Replace	<p>Allows you to search for and replace selected information.</p> <p>The tool searches for the data in the current spreadsheet and replaces it with the selected data.</p> <p>This tool is available only when working with a Data Entry spreadsheet.</p>
 Find	Find	<p>Allows you to find the row containing a specific joint, member, or wall panel in the results spreadsheet.</p> <p>Open a spreadsheet containing results to activate the tool. Enter the joint, member, or wall panel label; the containing row is highlighted.</p> <p>This tool is available only when working with a Results spreadsheet.</p>
 Save as Defaults	Save as Defaults	<p>Allows you to save common entries as the default values in a spreadsheet, so that they are readily accessible for new, future models.</p>
Graphical Highlight		
These tools are available only when you select one or more rows in a spreadsheet.		
 Select	Select	<p>Allows you to select marked lines in the current model view.</p> <p>You can quickly select elements in the model view that correspond to the marked lines in the spreadsheet.</p>
 Unselect	Unselect	<p>Allows you to unselect the marked lines in the current model view.</p> <p>You can quickly unselect elements in the model view that correspond to the marked lines in the spreadsheet.</p>
BLC		
These tools are available only when the Basic Load Cases spreadsheet is opened.		
 Copy Loads	Copy Loads	<p>Allows you to copy loads from one Basic Load Case to another.</p> <p>you can choose to copy all load types or only some load types. Unchecking a box excludes that load type from the selection.</p>

Icon	Tool	Description
 Clear Loads	Clear Loads	Allows you to clear all load types from a Basic Load Case.

For more details about the various options on the Spreadsheets ribbon, see [Spreadsheet Operations](#)

Advanced Ribbon



The following table provides a list of the various tools on the Advanced ribbon and their descriptions.

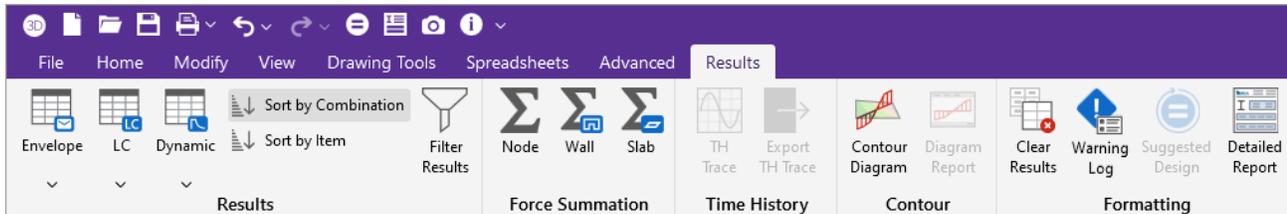
Icon	Tool	Description
Load Generators		
 Wind	Wind	<p>Opens the Wind Load Generator, that allows you to generate wind loads for the lateral design of a building structure.</p> <p>Wind loads are based on the specified code requirements, length/width of the diaphragm and tributary height to the diaphragm. The loads are applied to automatically generated nodes located in the plane of each diaphragm.</p> <p>See the Wind Loads topic for more information.</p>
 Seismic	Seismic	<p>Opens the Seismic Load Generator, that allows you to generate seismic loads for the lateral design of a building structure.</p> <p>Seismic loads are based on the tributary weight of each diaphragm, as well as the seismic parameters set in the Model Settings. The loads are applied to automatically generated nodes located in the plane of each diaphragm.</p> <p>See the Seismic Loads topic for more information.</p>
 Notional	Notional	<p>Opens the Notional Load Generator, that allows you to generate notional loads for the stability design of a building structure.</p> <p>Notional loads generate a de-stabilizing load for each diaphragm/floor level at the center of mass, based on a percentage of applied mass.</p> <p>See the Notional Loads topic for more information.</p>

Icon	Tool	Description
Libraries		
 Time History	Time History	<p>Allows you to view the library of Time History functions/patterns, which may be generated or imported into the program.</p> <p>See Time History for more information.</p>
 Moving Loads	Moving Loads	<p>Allows you to view the Moving Loads pattern database, which defines the magnitude, direction and distance used when applying a moving load. Add a new pattern or edit an existing pattern to use in the model.</p> <p>See Moving Loads for more information.</p>
 Response Spectra	Response Spectra	<p>Allows you to view the Response Spectra database, which provides spectra for existing building codes.</p> <p>View an existing spectra or add a new spectra. Input options include period and frequency, as well as the three spectral values: acceleration, velocity, and displacement.</p> <p>See the Dynamics topic for more information.</p>
 Shape Database	Shape Database	<p>Allows you to view the Shape database.</p> <p>The shape database contains hot rolled steel, cold formed steel, wood, aluminum, stainless steel, general shapes and their properties.</p> <p>View, edit, or create a shape to use in the model.</p> <p>See Shape Database topic for more information.</p>
 Custom Wood	Custom Wood	<p>Allows you to view the properties for Custom Wood species.</p> <p>You can modify customized wood properties in this database, which can then be used as the Custom Wood material found in the Materials > Wood spreadsheet.</p> <p>See Wood Design for more information.</p>
 Custom Rebar	Custom Rebar	<p>Allows you to view Custom Rebar layouts.</p> <p>You can create custom flexural or shear rebar layouts to be used in the analysis of concrete columns or beams.</p> <p>See Concrete Design for more information.</p>
 Redesign Lists	Redesign Lists	<p>Allows you to view Member Redesign Lists.</p> <p>Redesign lists are available for all material and shape types. You can customize the member optimization, by viewing and editing the member redesign lists. In addition, you can edit the existing lists or create a new list.</p>

Icon	Tool	Description
		See Redesign List for more information.

Results Ribbon

The **Results** ribbon is visible only after you solve the model and a solution is present. This ribbon provides tools for viewing the analysis result. Results that display in spreadsheet format can also be found in the [Explorer](#) panel under the *Results* section.



The following table provides a short description of each tool.

Results- Icons

Icon	Tool	Description
Results		
 Envelope	Envelope	Allows you to view enveloped analysis results. The tool displays enveloped (max and min values only) analysis results in the selected spreadsheets.
 LC	LC	Allows you to view analysis results on a per-load combination basis. The tool displays results corresponding to each load combination solved in the analysis. This is useful when investigating the results for a specific load combination.
 Dynamic	Dynamic	Allows you to view dynamic analysis results. The tool displays the natural frequencies or mode shapes, after performing a dynamic analysis.
	Sort by Combination	Allows you to sort the spreadsheet results of a batch solution by load combination. Results are listed in chronological order, based on load combination. For example, all the member forces under Load Combination 1 are listed first. The forces under Load Combination 2 are listed next.
	Sort by Item	Allows you to sort the spreadsheet results of a batch solution by item. For example, you can group results from each load

Icon	Tool	Description
		combination under a particular member.
 Filter Results	Filter Results	<p>Allows you to filter selected elements from the spreadsheet results.</p> <p>Choose to filter out the results for selected or unselected elements from the results spreadsheets. This is useful when you would like to focus on the results for a specific portion of the model.</p>
Force Summation		
 Node	Node	<p>Allows you to sum the internal forces across selected elements.</p> <p>Select two points to define a cutting plane and then select a third point to locate the plane. The internal force summation tool then sums the forces of every selected element that it cuts through and reports the totals.</p>
 Wall	Wall	<p>Allows you to sum internal plate forces between two points within a wall.</p> <p>Select two points in a wall panel to sum the internal forces between the two points. This tool is useful for investigating forces within specific regions in a wall panel. This can also be done across multiple walls that are in the same plane.</p>
 Slab	Slab	<p>Allows you to sum internal plate forces between two points within a slab.</p> <p>Select two points in a wall panel to sum the internal forces between the two points. This tool is useful for investigating forces within specific regions in a wall panel. This can also be done across multiple walls that are in the same plane.</p> <div style="border: 1px solid black; padding: 5px; margin-top: 10px;">  Note: This tool is available only when the model originates in RISAFloor and has Semi-Rigid Diaphragms. </div>
Time History		
 TH Trace	TH Trace	<p>Allows you to view Time History results at a specific node location.</p> <p>Select a node to view displacement, velocity, acceleration or reaction results over time for the specified node. You can choose to display results for the global X, Y and Z direction as well as rotational directions RX, RY and RZ.</p> <p>See Time History for more information.</p>

Icon	Tool	Description
 Export TH Trace	Export TH Trace	Allows you to export a series of Time History traces to a text file. Export a series of time history traces for all selected nodes to an ASCII text file. See Time History for more information.
Contour		
 Contour Diagram	Contour Diagram	Allows you to display a diagram for stresses and/or forces along a defined cut in Plates or Walls. Select two points in a wall panel or mesh of plates to display a diagram representation of color contours for stresses and/or forces along a defined cut in those plates or walls.
 Diagram Report	Diagram Report	Allows you to display an enlarged image of the Contour Diagram. Use this tool to view a graphical report that shows the plate/wall contours as diagrams and numerical data for every section along the length of the diagram.
Formatting		
 Clear Results	Clear Results	Allows you to delete all solution results. The tool clears all solution results to make changes in the model or solve another solution.
 Warning Log	Warning Log	Opens the Warning Log spreadsheet, which provides you with a record of any warnings or errors that occurred during the solution of your model. See the Warning Log topic for more information.
 Suggested Design	Suggested Design	Allows you to solve again using suggested shapes. Resolve the model using suggested shares based on design criteria for a more optimal design.
 Detailed Report	Detailed Report	Allows you to access detailed analysis and design information for elements. Click on an element to view all analysis and design results pertaining to that specific element. View shear, moment and stress diagrams as well as design information and code checks.

For more details about the various options on the Results ribbon, see [Results](#).

Aluminum

Aluminum - Databases

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

Aluminum Databases

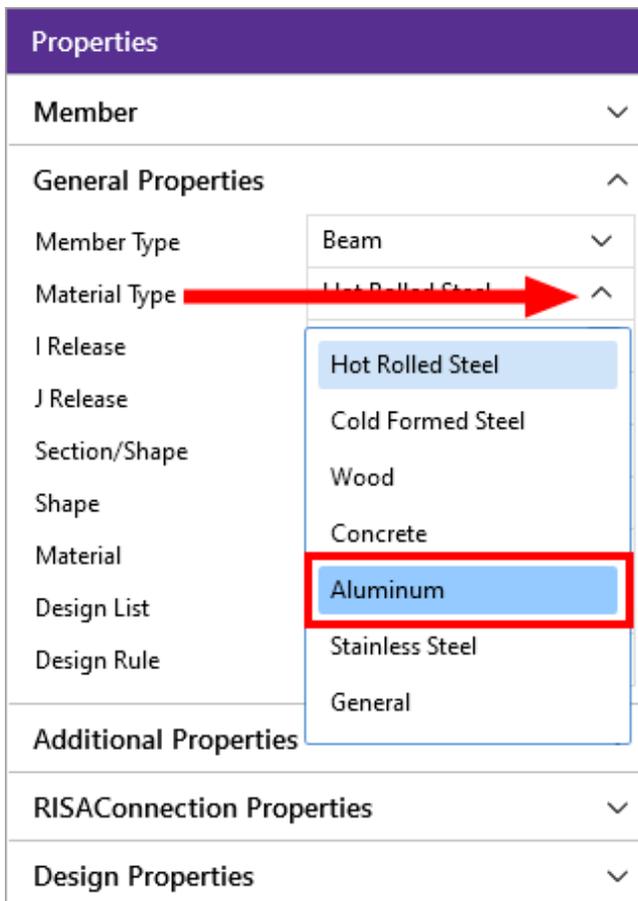
RISA currently supports the following two common Aluminum databases:

- Aluminum US
- Aluminum CAN

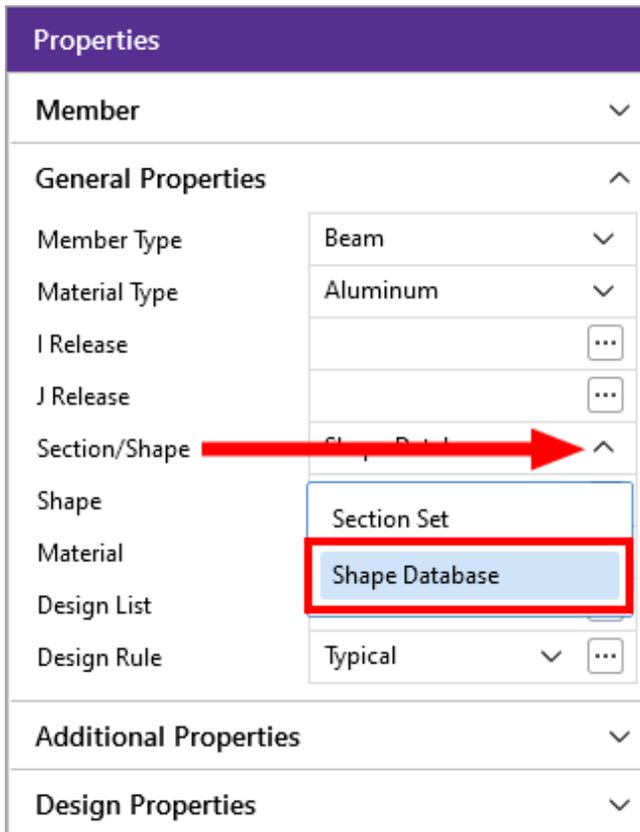
Select an Aluminum Database

To select an aluminum database when drawing a new member:

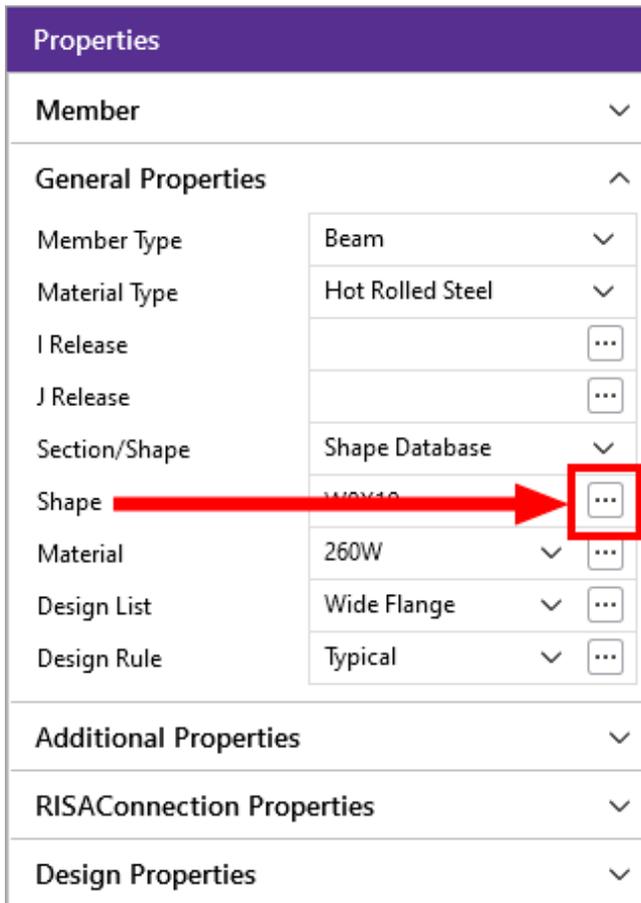
1. Click the **Material Type** ▾ down-arrow in the Properties panel and choose **Aluminum**.



2. Click the **Section/Shape** ▾ down-arrow and choose the **Shape Database**.



3. Click the **Shape** ellipses button to open the Shape Selection window (shown in the following image).



The Shape Selection window opens to the Aluminum tab.

3D Shape Selection
? X


General


Hot Rolled


Cold Formed


Wood


Aluminum


Stainless

Database Aluminum US ▼

Shape Type

WF

Z

T

L

CS Single ▼

Round Tube

Rect Tube

Solid Rectangle

Shape Name

AACS2X0.577

AACS2X1.07

AACS3X1.14

AACS3X1.60

AACS4X1.74

Preview

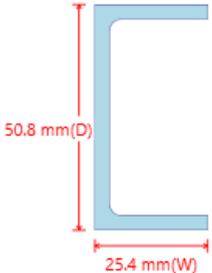
Add

View

Edit

Copy

Delete



Save Database as default

OK

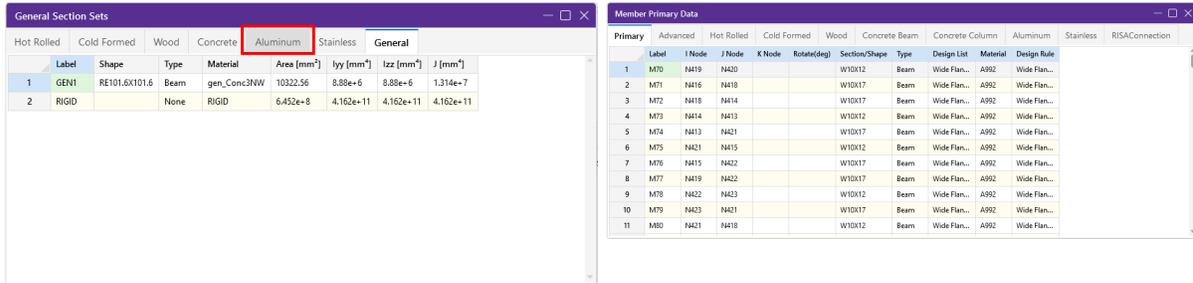
Cancel

To select an aluminum database from the spreadsheets:

1. Click the **Aluminum** tab of the 'Section Sets' spreadsheet or the **Primary** tab of the 'Members' spreadsheet.

Section Sets spreadsheet - General

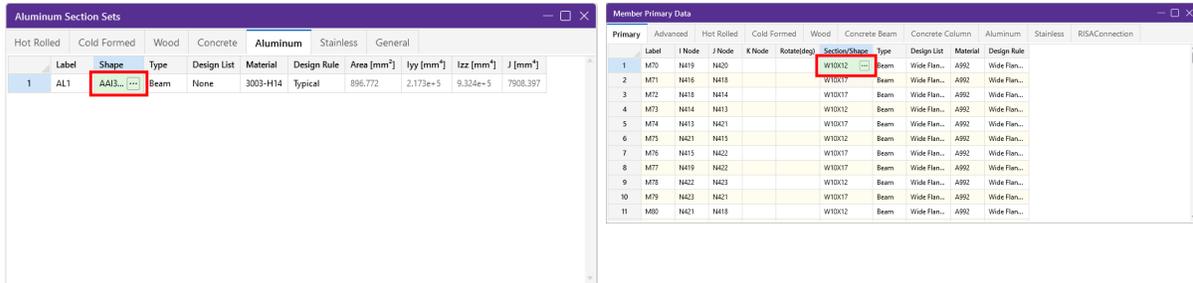
Member spreadsheet - Primary



- Click on the **Shape** or **Section/Shape** cell of the row you want to work with, then click the **...** ellipsis button that appears (shown in the following images).

Section Sets spreadsheet - Aluminum tab

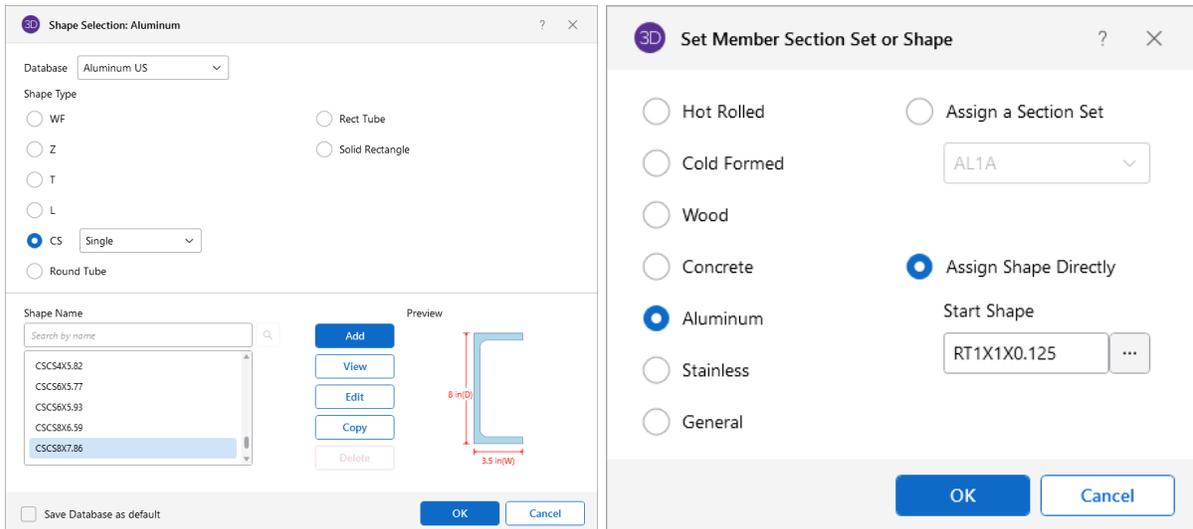
Member spreadsheet - Primary tab



The Shape Selection: Aluminum or Set Member Section or Shape window opens, depending on which spreadsheet you clicked the ellipsis button in.

Shape Selection: Aluminum window

Set Member Section Set or Shape window



Select an Aluminum Shape

To select an aluminum shape from the Shape Selection window:

1. Click the **Database** ▾ down-arrow and choose the database where the shape is stored.
RISA currently supports two common Aluminum databases: Aluminum US and Aluminum CAN.

The screenshot shows a 'Database' dropdown menu with 'Aluminum US' selected. Below it, a 'Shape Type' dropdown menu is open, showing 'Aluminum CAN' and 'Aluminum US' as options. To the left of the 'Shape Type' dropdown are two radio buttons: 'WF' and 'Z', both of which are unselected.

2. Click on the **Shape Type** you want to select from.
If you select **CS**, you also have to click the down-arrow and choose a layout.

The screenshot shows the 'Shape Type' selection window. It contains several radio buttons: 'WF', 'Z', 'T', 'L', 'CS', and 'Round Tube'. The 'CS' radio button is selected. To the right of the 'CS' radio button is a dropdown menu with 'Single' selected. Other options include 'Rect Tube' and 'Solid Rectangle'.

3. In the **Shape Name** list box, click on the name of the shape you want.
To narrow the scope of names appearing in the list box, begin typing the name in the search box.
As you type, the scope of the list is filtered to show only those names that match what you are typing.

The screenshot shows the 'Shape Name' list box. At the top is a search box with the placeholder text 'Search by name' and a magnifying glass icon. Below the search box is a list of shape names: 'CSCS8X6.59', 'CSCS8X7.86', 'CSCS10X8.58', 'CSCS10X9.32', and 'CSCS10X10.1'. The 'CSCS8X7.86' entry is highlighted in blue.

4. Click **OK** to select the shape and close the Shape Selection window.

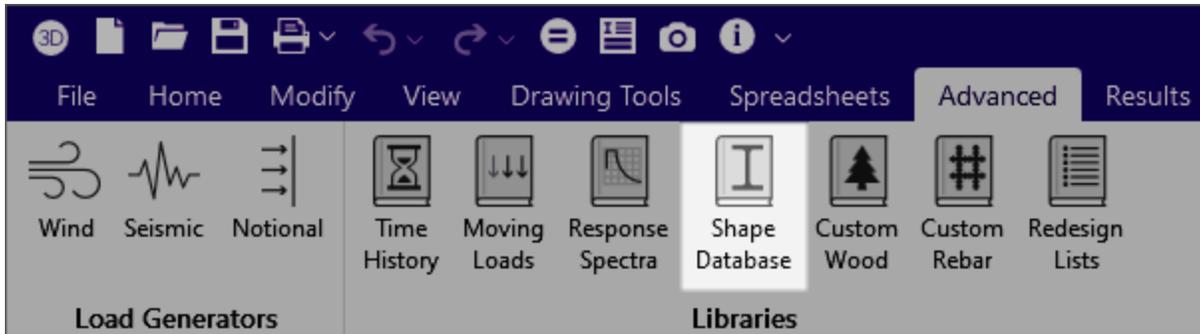
Shape Database Files

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

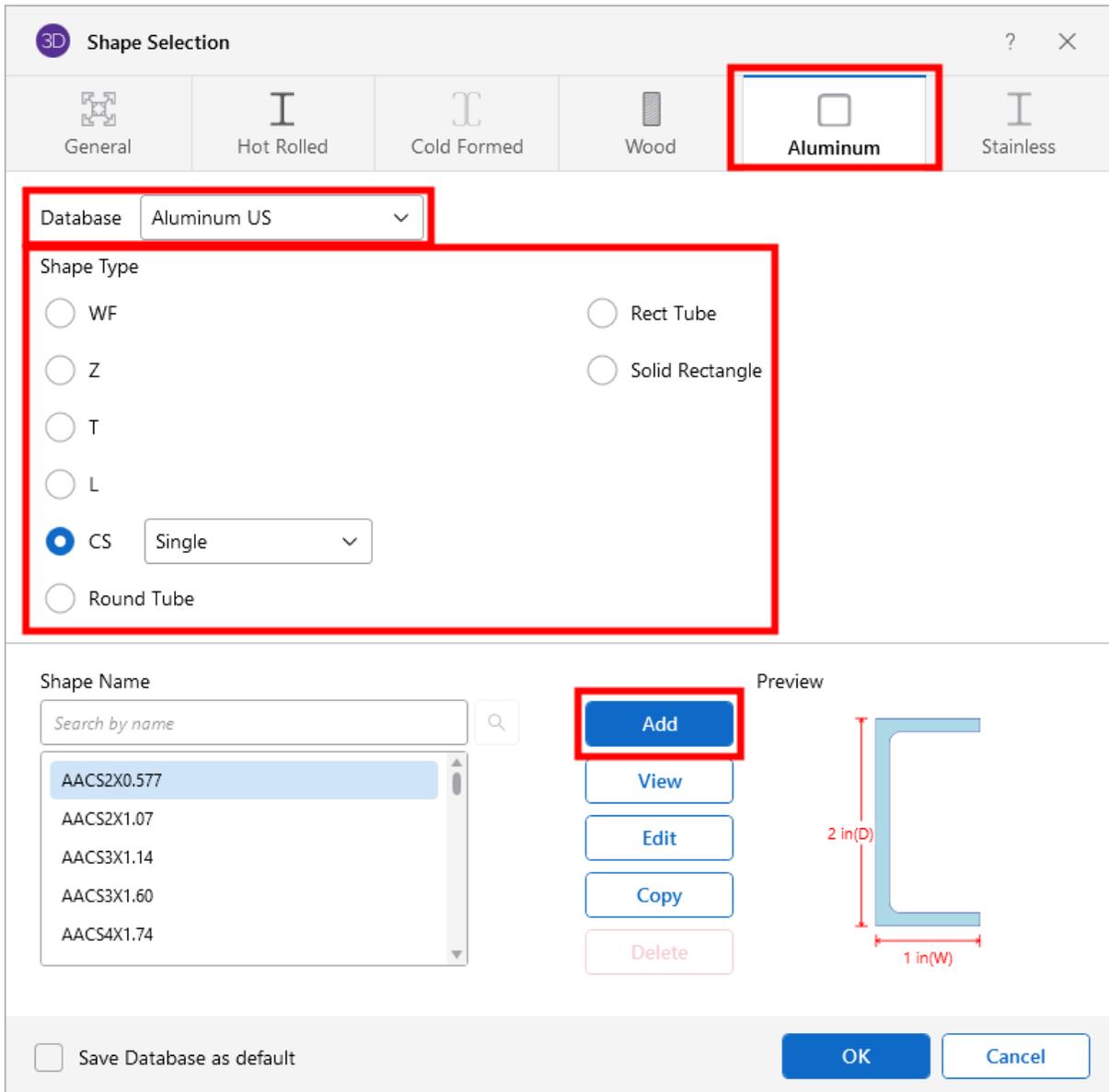
Add a Database Shape

To add a database shape:

1. Click the **Shape Database**  icon, on the 'Advanced' ribbon (as shown in the following image).



The Shape Selection window opens.



2. Click the **Aluminum** tab.
3. Click the **Database** \downarrow down-arrow and choose the database where you want to store the new shape (as shown in the previous image).

RISA currently supports two common Aluminum databases: Aluminum US and Aluminum CAN.

4. Click on the **Shape Type** you wish to add, then click the **Add** button (as shown in the previous image).

The Add Shape window opens.

3D
Add Shape
?
×

Basic Properties

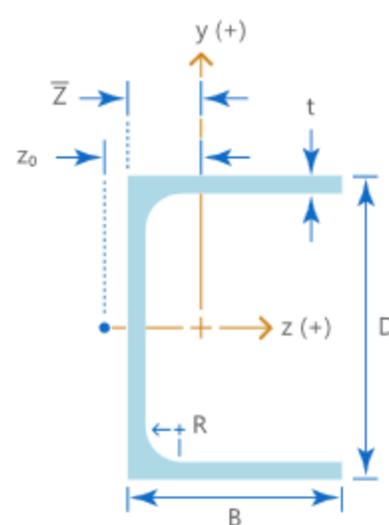
Shape Name

D in

B in

t_f in

t_w in



Calculated Properties

Area	0	in ²	C_w	0	in ⁶	<div style="border: 1px solid red; padding: 5px; background-color: #007bff; color: white; font-weight: bold; margin-bottom: 5px; width: 100px;">Calculate</div> <div style="background-color: #ccc; padding: 5px; width: 100px;">Clear</div>
I_{yy}	0	in ⁴	\bar{z}	0	in	
I_{zz}	0	in ⁴	z_o	0	in	
J	0	in ⁴				
Z_{yy}	0	in ³				
Z_{zz}	0	in ³				

Units

OK

Cancel

5. Type a **Shape Name** for the shape.
6. Enter in the **Basic Properties** (D, B, t_f and t_w).
7. Click **Calculate** to determine the shape properties.

The Calculated Properties section displays the results from the calculation (as shown in the following image).

3D
Add Shape
?
✕

Basic Properties

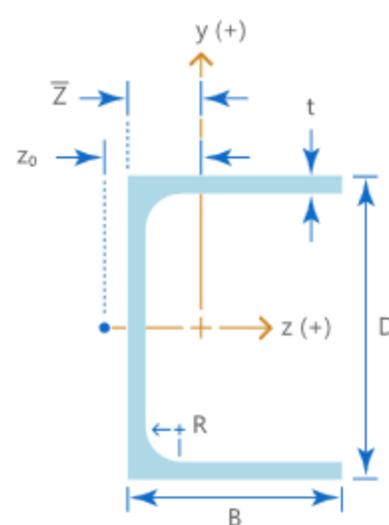
Shape Name

D in

B in

t_f in

t_w in



Calculated Properties

Area	3.438	in ²	C_w	126.218	in ⁶	<input style="width: 100px; height: 20px;" type="button" value="Calculate"/> <input style="width: 100px; height: 20px;" type="button" value="Clear"/>
I_{yy}	11.498	in ⁴	\bar{z}	1.38	in	
I_{zz}	32.955	in ⁴	z_o	3.41	in	
J	0.047	in ⁴				
Z_{yy}	4.373	in ³				
Z_{zz}	9.66	in ³				

Units

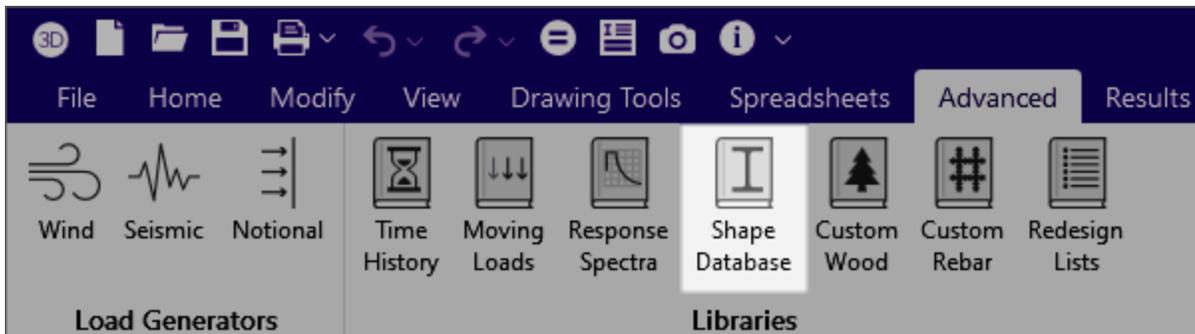
If you are satisfied with the results, save the new shape.

8. Click **OK** to save the shape and close the Shape Selection window.

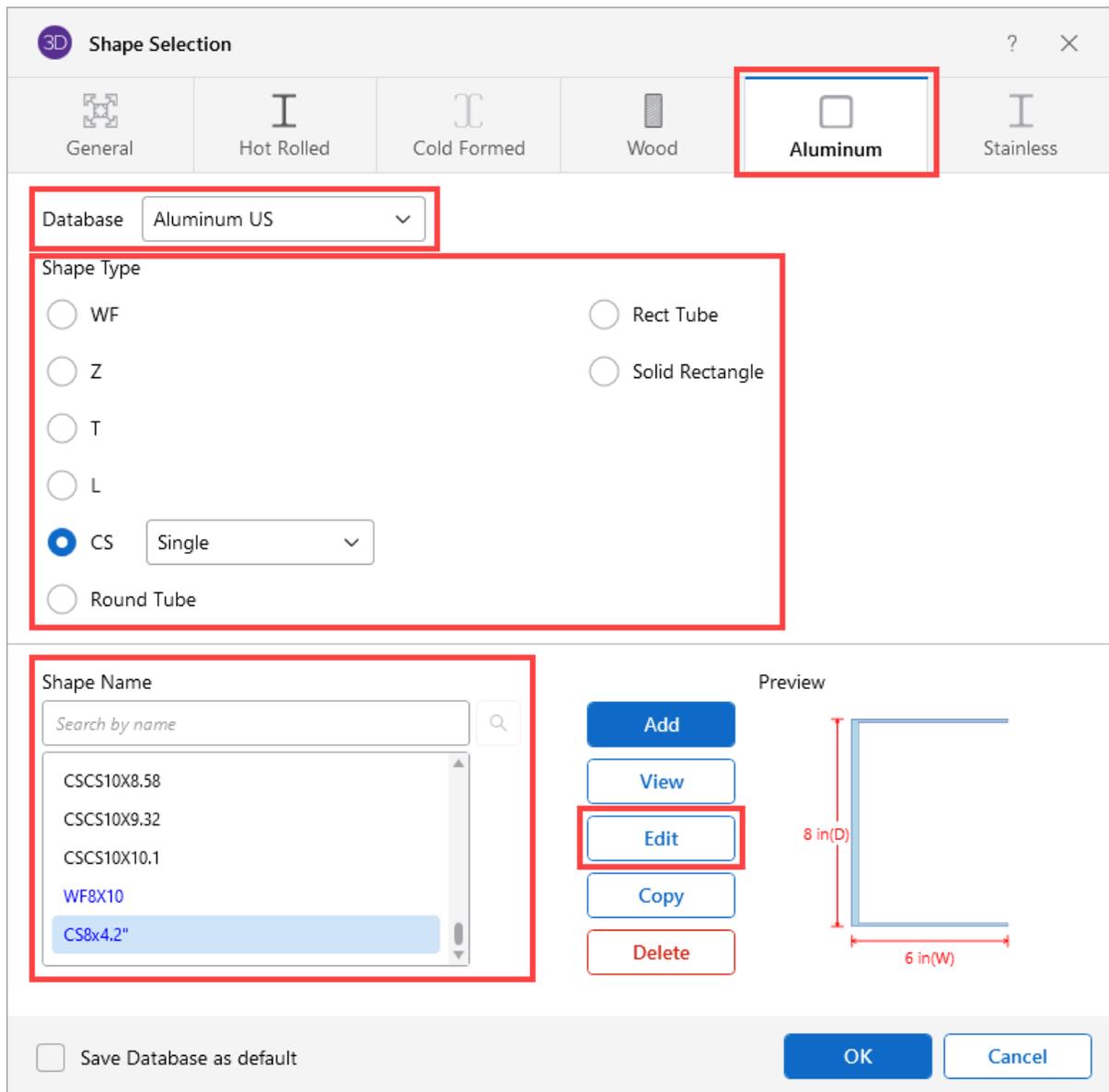
Edit a Database Shape

To edit a database shape:

1. Click the **Shape Database**  icon on the **Advanced** ribbon(as shown in the following image).



The Shape Selection window opens.



2. Click the **Aluminum** tab.
3. Click the **Database** \vee down-arrow and choose the database where you want to edit the shape.
4. Click on the **Shape Type** of the shape you want to edit, if not already selected.
5. Click on the **Shape Name** of the shape you want to edit, then click the **Edit** button.
The Edit shape window opens.

3D Edit Shape

Shape Name CS8x4.2"

Geometric Properties		Section Properties	
D	8 in	Area	3.438 in ²
B	6 in	I _{yy}	11.498 in ⁴
t _f	0.125 in	I _{zz}	32.955 in ⁴
t _w	0.25 in	Z _{yy}	4.373 in ³
		Z _{zz}	9.66 in ³
		C _w	126.2182 in ⁶
		J	0.047 in ⁴
		R	0 in
		r _o	4.956 in
		z _o	3.41 in
		\bar{z}	1.38 in

Print Clear All Re-Calc OK Cancel

You have the option to input 'Geometric Properties' and 'Section Properties' manually, or you may choose to have the program calculate the Section Properties based on values input for the Geometric Properties.

6. Make your **Geometric Properties** changes, then click **Re-Calc** to have the program calculate the **Section Properties** automatically.

3D Edit Shape
?
×

Shape Name

Geometric Properties

D in

B in

t_f in

t_w in

Section Properties

Area in²

I_{yy} in⁴

I_{zz} in⁴

Z_{yy} in³

Z_{zz} in³

C_w in⁶

J in⁴

R in

r_o in

z_o in

\bar{z} in

Print

Clear All

Re-Calc

OK

Cancel

7. Click **OK** to save the shape change(s) and close the Edit Shape window.
You are returned to the Shape Selection window.
8. Click **OK** to save the change to the Shape database and close the Shape Selection window.

**Note:**

- New shapes are added to the bottom of the database in a blue font to show they were not originally from the default database.
- To delete a shape, specify the database and shape type you wish to delete and then click the **Delete** button.
- Only shapes added to the default database (shown in blue) are able to be edited or deleted.

Aluminum Shape Types

There are ten types of aluminum shapes. Names for each shape type follow the convention of the manufacturer for each shape. If you know the shape name, you can type the name directly into the

Shape field on the spreadsheets, or use the search box to type part of the name to narrow the scope of names displayed in the Shape Name list.

WF sections

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

Zee sections (Z)

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

Tee sections (T)

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

Solid Rectangular

The Solid Rectangular sections or bar sections are defined by the user, there are no default shapes.

Angles sections (L)

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

Channel sections (CS)

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

Double Sections

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



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

Round Tube or Pipe (OD or NPS)

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

Rectangular Tube sections (RT)

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

Aluminum - Design

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

- Aluminum Design Manual 2015: Building and Bridge
- Aluminum Design Manual 2010: Building and Bridge
- Aluminum Design Manual 2005: Building and Bridge

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

Design Parameters

The **Aluminum** tab on the **Members Spreadsheet** records the design parameters for the aluminum code checks. These parameters may also be assigned graphically. See [Modifying Member Design Parameters](#) to learn how to do this.

Aluminum Design Parameters												
Primary	Advanced	Hot Rolled	Cold Formed	Wood	Concrete Beam	Concrete Column	Aluminum	Stainless	RISAConnection			
	Label	Shape	Length [ft]	Lb y-y [ft]	Lb z-z [ft]	Lcomp top [ft]	Lcomp bot [ft]	L-Torque [ft]	K y-y	K z-z	Cb	Function
1	M311	RT1X1X0.125	10	Segment		Lbyy						Lateral
2	M312	RT1X1X0.125	10			Lbyy			1	1		Lateral
3	M313	RT1X1X0.065	10			Lbyy						Lateral
4	M314	RT1X1X0.065	10			Lbyy						Lateral
5	M337	RT2.75X2.75X0.125	3.334			Lbyy						Lateral
6	M338	RT3X4X0.188	3.334			Lbyy						Lateral
7	M345	RT3X4X0.188	3.334			Lbyy						Lateral
8	M347	RT3X3X0.125	3.334			Lbyy						Lateral
9	M136	RT4X4X0.125	18.868			Lbyy						Lateral
10	M137	RT4X4X0.125	18.868			Lbyy						Lateral
11	M138A	RT4X4X0.125	18.868			Lbyy						Lateral

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

Label

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

Shape

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

Length

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

Unbraced Length

See the [Unbraced Lengths](#) topic.

K Factors (Effective Length Factors)

See the [Unbraced Lengths](#) topic.

Sway Flags

See the [Unbraced Lengths](#) topic.

Cm - Interactive Bending Coefficient

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

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



Note: Cm only shows up in the spreadsheet for the 2005 codes.

Cb - Bending Coefficients

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

Function for Stiffness Reduction

The **Function** entry may be set to either 'Lateral' or 'Gravity' using the drop down list in the spreadsheet. If the Adjust Stiffness option is set to Yes on the Codes tab of the Model Settings Dialog, then all members with a 'Lateral' Function will be considered for the stiffness reduction required per the Chapter C.2 Calculations of Required Strength.

The **Flexural Stiffness Reduction** of the Direct Analysis Method will be applied to all 'Lateral' members whose member type is set to either 'Beam' or 'Column' on the Primary Tab of the Members Spreadsheet. This process involves evaluating certain conditions:

- **Primary Stiffness Adjustment Requirements**

The program first checks whether a member satisfies the three primary stiffness adjustment requirements:

- The member is defined as "Lateral".
- The stiffness adjustment option is enabled for the specific material being used.
- The chosen material code permits stiffness adjustment for the selected material.

- **Stiffness Adjustment Calculation**

Once the above conditions are met, the software proceeds to calculate the stiffness adjustments for the member. This includes reducing the area and moment of inertia based on predefined multiplication factors:

- Aluminum Material: Area and moment inertia are multiplied by 0.8.

- **Exclusive Criteria**

However, certain exclusion criteria are considered to determine whether to apply the stiffness adjustments or not. The following conditions must be met simultaneously for exclusion:

- The material is Hot Rolled
- The selected steel code is AISC 13th ed.
- The member type is not specified as:
 - Column or v-brace for area reduction.
 - Column or beam for moment of inertia reduction.

If all the exclusion criteria are met, the original area and moment of inertia values will be retained without applying the stiffness adjustment multiplication factors. This ensures that certain types of HR members, when using the AISC code, maintain their original stiffness characteristics.

The program can perform an iterative analysis during the solution depending on the value of τ_b . In this case, the stiffness matrix is recomputed for each iteration until the value of τ_b converges within 1 percent for all 'Lateral' members in compression. In the unlikely event that τ_b is less than zero, the value of τ_b is considered to be 1.e-5.

When the users sets the Adjust Stiffness flag on the [Model Settings](#) to **Yes (Tau =1.0)**, then the program will use a Tau of 1.0 in the stiffness analysis and no iteration of the stiffness matrix is necessary. This option is a good feature for models which take a long time to solve or which have not yet been proportioned to control drift.

The **Axial Stiffness Reduction** of the Direct Analysis Method will be applied to all 'Lateral' members whose member type is set to either 'Column' or 'VBrace' on the Primary Tab of the Members Spreadsheet.



Note: The stiffness reduction required by the Chapter C will be ignored if the Adjust Stiffness option is not selected on the [codes](#) tab of the **Model Settings**, or if the design code chosen does not have an option for stiffness reduction.

Aluminum Code Check

Access the **Code Check** spreadsheet by selecting the **Results** tab ▶ **Envelope** (or **LC**) button ▶ **Code Check**. Alternatively, on the **Results** toolbar you can select **Code Check**.

Envelope AA ADM1-20: ASD - Building Aluminum Code Checks																		
Hot Rolled Steel		Cold Formed Steel		Wood		Concrete Beams		Concrete Columns		Aluminum		Stainless						
Member	Shape	Code Check	Loc[ft]	LC	Shear Check	Loc[ft]	Dir	LC	Pnc/Om[k]	Pnt/Om[k]	Mny/Om[k-ft]	Mnz/Om[k-ft]	Vny/Om[k]	Vnz/Om[k]	Cb	Eqn		
1	M311	RT1X1X0.125	0.639	10	21	0.002	10	y	21	0.203	8.535	0.202	0.202	1.923	1.923	2.179	H.1-1	
2	M312	RT1X1X0.125	0.651	0	21	0.002	0	y	21	0.203	8.535	0.202	0.202	1.923	1.923	2.128	H.1-1	
3	M313	RT1X1X0.065	0.62	0	20	0.002	0	y	19	0.127	4.735	0.126	0.126	1.288	1.288	1.949	H.1-1	
4	M314	RT1X1X0.065	0.053	0	21	0.002	0	y	21	0.127	4.735	0.126	0.126	1.288	1.288	2.202	H.1-1	
5	M337	RT2.75X2.75X0.125	0.98	0	18	0.024	3.334	z	21	22.518	25.528	1.941	1.941	7.308	7.308	1.23	H.1-1	
6	M338	RT3X4X0.188	0.874	3.334	21	0.025	0	y	21	44.64	48.523	4.172	4.941	15.901	11.273	2.367	H.1-1	
7	M345	RT3X4X0.188	0.848	0	21	0.028	3.334	z	21	44.64	48.523	4.172	4.941	15.901	11.273	1.211	H.1-1	
8	M347	RT3X3X0.125	0.91	3.334	21	0.023	3.334	z	18	25.653	28.062	2.295	2.295	8.077	8.077	1.558	H.1-1	
9	M136	RT4X4X0.125	0.418	8.844	21	0.003	18.868	y	19	4.858	37.805	3.746	3.746	11.154	11.154	1.131	H.1-1	
10	M137	RT4X4X0.125	0.443	10.024	19	0.003	18.868	y	21	4.858	37.805	3.746	3.746	11.154	11.154	1.131	H.1-1	
11	M138A	RT4X4X0.125	0.398	8.844	19	0.007	18.868	y	19	4.858	37.805	3.746	3.746	11.154	11.154	1.131	H.1-1	

The final results of the code checking are the code check values **UC Max** and **Shear UC**. So, if this value is less than 1.0, the member passes. If it is greater than 1.0, the member fails. If the value is greater than 9.999 it will be listed as "9.999". The Shear Check is based on f_s/F_s . Note that torsional shear, if any, is also included in this check. The location for the shear check is followed by "y" or "z" to indicate the direction of the shear. The **Loc** field tells at what location the maximum code check occurs measured from the I-node location of the member.

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

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

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

The moving load results are enveloped and the governing load combination and step location is shown for each result value under the "LC" column. The first number is the load combination, the second is the step number: (load combination - step number). See [Moving Loads](#) to learn more.



Note:

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

Aluminum Detail Report

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

AA ADM1-20: ASD - Building Code Check					
Limit State	Gov. LC	Required	Available	Unity Check	Result
Applied Loading - Bending/Axial	21	-	-	-	-
Applied Loading - Shear + Torsion	21	-	-	-	-
Axial Tension Analysis	-	5.179 k	8.535 k	-	-
Axial Compression Analysis	-	0 k	0.203 k	-	-
Flexural Analysis (Strong Axis)	-	0.005 k-ft	0.202 k-ft	-	-
Flexural Analysis (Weak Axis)	-	0.002 k-ft	0.202 k-ft	-	-
Shear Analysis (Major Axis y)	-	0.004 k	1.923 k	0.002	PASS
Shear Analysis (Minor Axis z)	-	0.002 k	1.923 k	0.0008796	PASS
Bending & Axial Interaction Check (UC Bending Max)	-	-	-	0.639	PASS

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

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

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

The **Slender Limit** $\lambda 1$ and $\lambda 2$ are calculated based on the **Gov Eqn**. The Slenderness Ratio are given based on the Design Aids in Tables 2-x ADM Section VI. The **Slender. Ratio** is also based on the **Gov Eq**, for further information refer to the ADM Design Aids.

Assumptions and Limitations

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

- **Single Angles** are not checked for bending in the 2005 and 2010 codes. They are checked for bending in the 2015 and newer codes.
- **r_{ye}** , effective radius of gyration from Eq 4.9.1-1 is used for doubly symmetric sections beams.
- **Welded** regions are not checked in RISA-3D. You can use the welded material properties for the entire member, or create segments that are welded material in order to check the weld properties. For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Welded Aluminum**.
- **Connections** are not checked, including web crippling, fatigue or stiffeners.
- **Kt** in is assumed to be the smaller of **Kyy** or **Kzz**.
- **Double Sections** - The program currently only performs an analysis of double sections and does not perform a code check of any kind.

Special Messages

Aluminum Code Check Not Calculated

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

-Pu was ignored in the calculation of the unity check-

This message is reported in the detail report when the member axial force is low (less than 5% of capacity). This is done so that beam members with very low axial forces will give similar code checks whether in tension or compression.

Boundary Conditions

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

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Boundary Conditions**.

Creating and Modifying Boundary Conditions

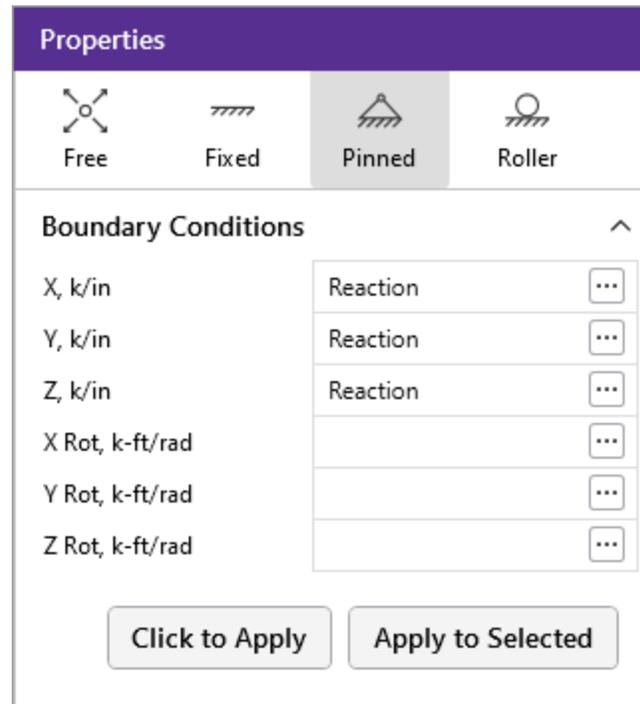
There are a number of ways to create or modify boundary conditions. You may view and edit the data in the **Boundary Conditions Spreadsheet**, you may click a node to view and edit its properties, or you can use the **Boundary Conditions** tool to graphically assign or modify a possibly large selection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

Modify Boundary Conditions for Nodes

The graphical **Boundary Conditions** tool discussed here lets you specify and modify boundary conditions graphically. To use this, you typically specify the new boundary condition, then click on **Click to Apply** and click on nodes one by one or press **Apply to Selected** after having selected the nodes that you want to assign or modify.

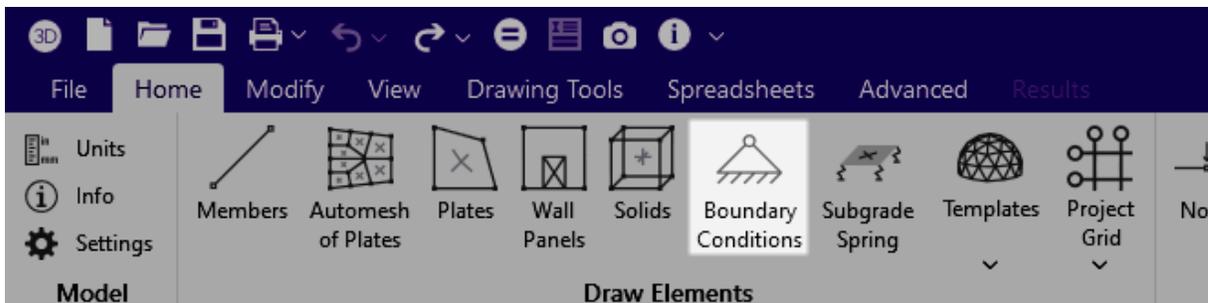
You can modify or assign nodes one at a time by selecting the **Click to Apply** option and then click on the nodes you wish to modify. You may also modify or assign entire selections of nodes by selecting the nodes first and then use the **Apply to Selected** option.

The parameters shown below are the same as those on the **Boundary Conditions Spreadsheet** and are described in [Boundary Condition Options](#). Use the ellipsis button to select the boundary condition.



To Apply Boundary Conditions

1. If a model view is not already open, click the **Open 3D Views** icon on the 'View' ribbon to open a new view.
2. Click the **Boundary Conditions** icon on the 'Home' ribbon (as shown in the following image) and select a boundary condition.



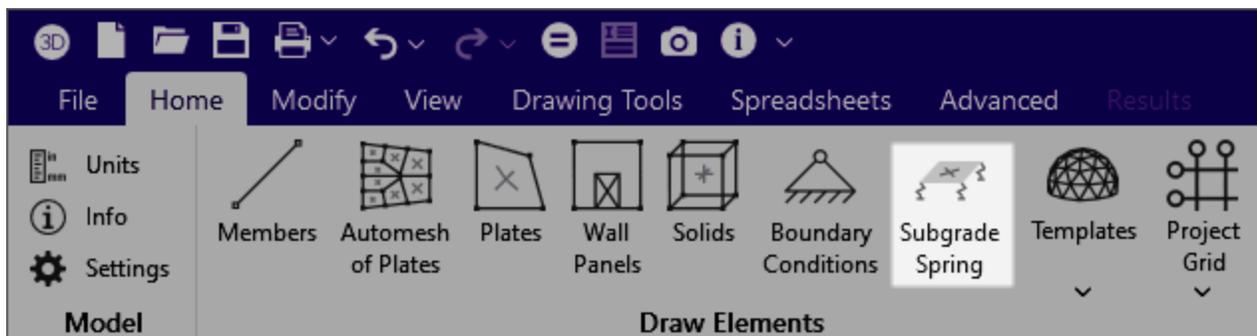
3. You can apply the boundary condition by choosing **Click to Apply** or **Apply to Selected**.
 To choose nodes on the fly, choose **Click to Apply**. Click/Box the nodes with the left mouse button.
 To apply the boundary condition to a selection of nodes, choose **Apply to Selected**.

**Note:**

- To apply more boundaries with different conditions, press the **Boundary Conditions** button once more.
- You may also view and edit boundary conditions by clicking on a node and editing the values in the **Properties** panel.
- You may also specify or edit boundary conditions in the **Boundary Conditions Spreadsheet**.
- You may undo any mistakes by clicking the **Undo**  button.

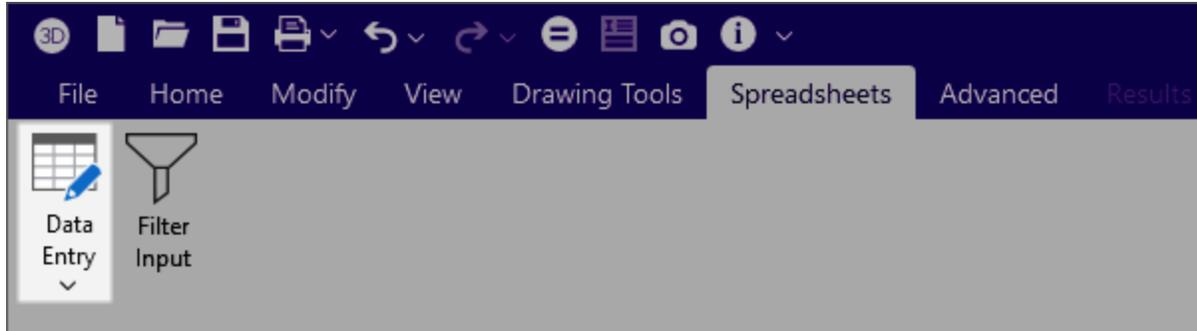
Generate Soil Springs

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



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

Boundary Conditions Spreadsheet



The **Boundary Conditions Spreadsheet** records the boundaries for the nodes and may be accessed by selecting **Boundary Conditions** from the **Data Entry** drop down menu in the **Spreadsheets** tab of the **Ribbon Toolbar** or by selecting **Boundary Conditions** from the **Data Entry** Window.

Node Boundary Conditions							
Primary		Advanced					
	Node Label	X [k/in]	Y [k/in]	Z [k/in]	X Rot [k-ft/rad]	Y Rot [k-ft/rad]	Z Rot [k-ft/rad]
1	N22	Reaction	Reaction	Reaction			
2	N35	Reaction	Reaction	Reaction			
3	N19	Reaction	Reaction	Reaction			
4	N45	Reaction	Reaction	Reaction			
5	N12						
6	N418						
7	N13	Reaction	Reaction	Reaction			
8	N508	Reaction	Reaction	Reaction			
9	N8	Reaction	Reaction	Reaction			
10	N7	Reaction	Reaction	Reaction			
11	N15	Reaction	Reaction	Reaction			

The **Node Label** 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 for degrees of freedom.

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.

Advanced Boundary Conditions

The **Advanced** tab of the boundary conditions spreadsheet contains information on nodal damping.

Node Advanced Boundary Conditions						
Primary	Advanced					
	Node Label	X Damp [k*sec/ft]	Y Damp [k*sec/ft]	Z Damp [k*sec/ft]	XR Damp [k-ft*sec/rad]	YR Dam
1	N22					
2	N35					
3	N19					
4	N45					
5	N12					
6	N418					
7	N13					
8	N508					
9	N8					
10	N7					
11	N15					

Damping for Spring Boundary Conditions

When the user has defined a two-way soil spring to a node and chooses to run a time history analysis using the Direct Integration solution method, then they may manually assign a Damping value to each of these soil springs.



Note:

- If there is not a spring boundary condition for the direction specified, then the user entered damping value will be ignored. You can only manipulate supports in this column if they are simple point supports. Any footings, piles or pile caps are manipulated in the other columns.
- These damping entries are only used when the direct integration method is selected for the time history solution. These values are not editable unless this method is already selected in the **Global Parameters**.

Boundary Condition Options

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



Note:

- Models that contain compression-only or tension-only springs must be iterated until the solution converges. Convergence is achieved when no more load reversals are detected in the springs. During the iteration process, each spring is checked, and if any springs are turned off (or back on), the stiffness matrix is rebuilt and model is resolved. This can take quite a bit longer than a regular static solution.
- With this iteration procedure it is possible to run into a phenomenon known as "clapping". This occurs when a compression-only spring keeps going between a tension force present (where the spring is removed) to a compression force present (where the spring is re-added). This can be somewhat common in non-linear solutions. If you run into this phenomenon, you may see the program give Error 1162. The fix is to very slightly adjust either the loading or the spring/element stiffness to get the offending spring out of the range where it can bounce back and forth. Another option would be to move the location of the soil spring slightly so it's not right at an inflection boundary.
- You can enter the first letter of the option ("R" for Reaction, "S" for Spring, etc.) rather than typing out the entire code. RISA-3D fills in the rest automatically. The exception is the Tether entry, where the full word does have to be entered. After Tether, the primarynode needs to be entered (for example, Tether N2)

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 otherwise listed. This is useful if you should want to lock certain directions of movement for all or most of the nodes. For example, if you are solving a 2D frame defined in the XY plane and you're only interested in the planar action, you could enter "ALL" and put an "F" (for Fixed) for Z translation, X Rotation and Y Rotation. See the following figure:

Node Boundary Conditions							
Primary		Advanced					
	Node Label	X [k/in]	Y [k/in]	Z [k/in]	X Rot [k-ft/rad]	Y Rot [k-ft/rad]	Z Rot [k-ft/rad]
1	ALL			Fixed	Fixed	Fixed	
2	N22	Reaction	Reaction	Reaction			
3	N35	Reaction	Reaction	Reaction			
4	N19	Reaction	Reaction	Reaction			
5	N45	Reaction	Reaction	Reaction			
6	N12						
7	N418						
8	N13	Reaction	Reaction	Reaction			
9	N508	Reaction	Reaction	Reaction			
10	N8	Reaction	Reaction	Reaction			
11	N7	Reaction	Reaction	Reaction			

**Note:**

- If a node 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 nodes NOT otherwise listed on the **Boundary Conditions Spreadsheet**.
- When using the "ALL" command, boundary conditions are no longer graphically displayed.

Reaction Boundary Condition

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

Fixed Boundary Condition

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

Spring Boundary Condition

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

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

Compression-Only Springs

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

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

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

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

Tension-Only Springs

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

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

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

- When a model contains T/C only springs, the program must iterate the solution until it converges. Convergence is achieved when no more load reversals are detected in the T/C only springs. During the iteration process, each T/C only boundary condition is checked. If any springs are turned off (or turned back on), the stiffness matrix is rebuilt and model is

resolved. For models with lots of T/C only elements, this can take a bit longer than a regular static solution.

Tethered Nodes

You may tether any or all of the node degrees of freedom to another node. See [Tethering Nodes](#) for more information.

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 translation degrees of freedom. See [Drift](#) for more information.

Boundary Conditions at Wall Panels

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

Cold Formed Steel

Cold Formed Steel - Databases

Custom vs. Manufacturer Shapes

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

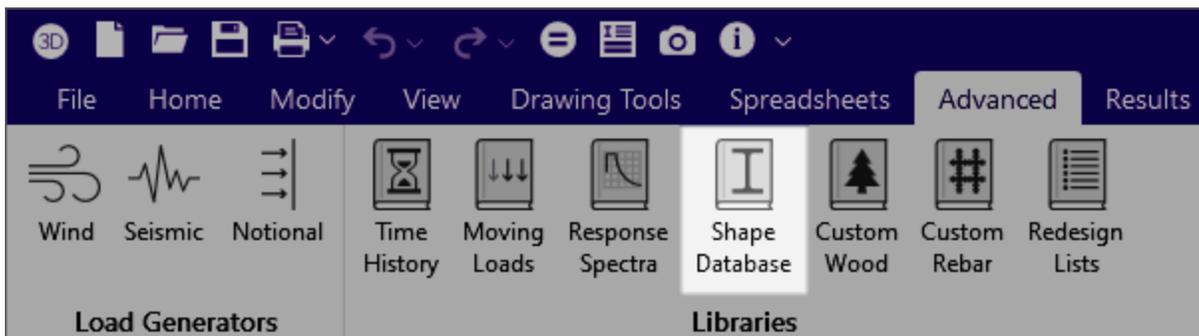
Shape Database Files

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

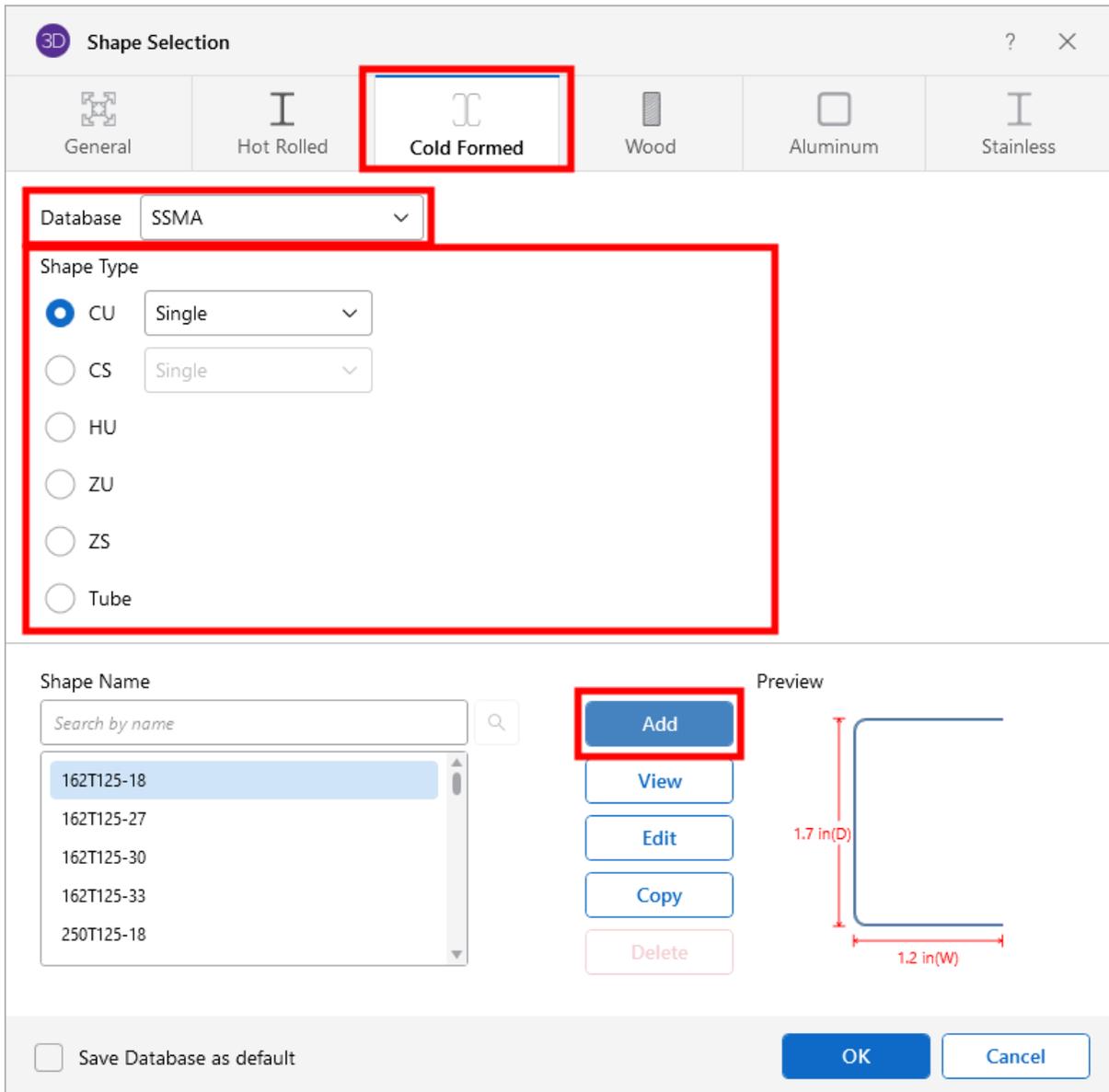
Add a Database Shape

To add a database shape

1. Click the **Shape Database**  icon on the 'Advanced' ribbon (as shown in the following image).



The Shape Selection window opens.



2. Click the **Cold Formed** tab.
3. Click the **Database** \downarrow down-arrow and choose the database where you want to store the new shape (as shown in the previous image).
4. Click on the **Shape Type** you wish to add, then click the **Add** button (as shown in the previous image).

The Add Shape window opens.

3D Add Shape
?
✕

Basic Properties

Shape Name

D in

B in

t in

R in

Calculated Properties

Area	0	in ²	X _c	0	in	<input style="width: 100px; height: 25px;" type="button" value="Calculate"/> <input style="width: 100px; height: 25px;" type="button" value="Clear"/>
I _{yy}	0	in ⁴	m	0	in	
I _{zz}	0	in ⁴	j	0	in	
J	0	in ⁴				
C _w	0	in ⁶				
r _o	0	in				

5. Type a **Shape Name** for the shape.
6. Enter in the **Basic Properties** (D, B, t and R in the above example).
7. Click **Calculate** to determine the shape properties.

The Calculated Properties section displays the results from the calculation (as shown in the following image).

3D
Add Shape
?
×

Basic Properties

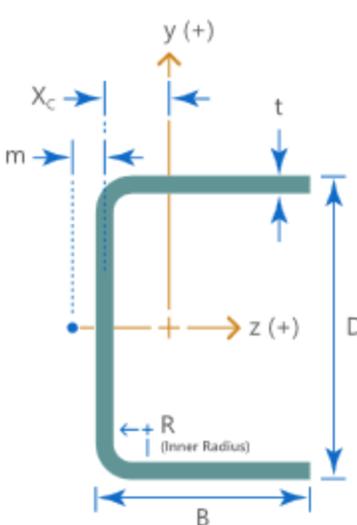
Shape Name

D in

B in

t in

R in



Calculated Properties

Area	5.948	in ²	X_c	0.886	in	<div style="border: 1px solid #4a7ebb; background-color: #4a7ebb; color: white; padding: 5px; margin-bottom: 5px; width: 100px;">Calculate</div> <div style="border: 1px solid #ccc; background-color: #f0f0f0; padding: 5px; margin-bottom: 5px; width: 100px;">Clear</div>
I_{yy}	12.478	in ⁴	m	1.522	in	
I_{zz}	263.853	in ⁴	j	10.39	in	
J	0.096	in ⁴				
C_w	722.419	in ⁶				
r_o	7.229	in				

Units

OK

Cancel

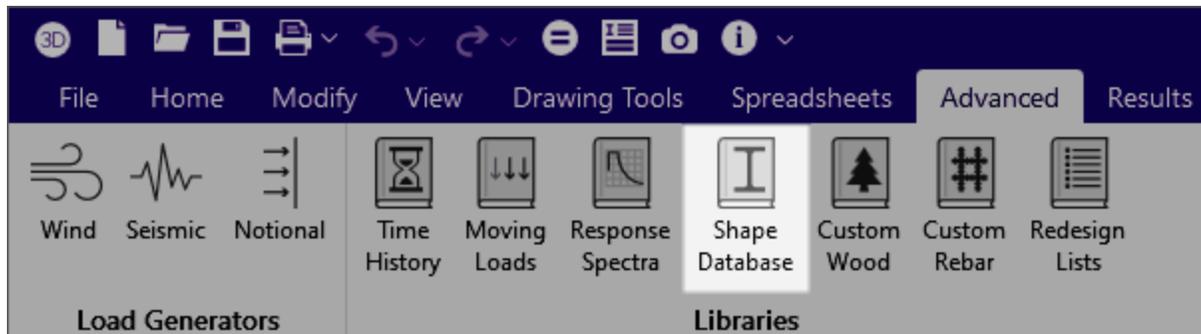
If you are satisfied with the results, save the new shape.

8. Click **OK** to save the shape and close the Shape Selection window.

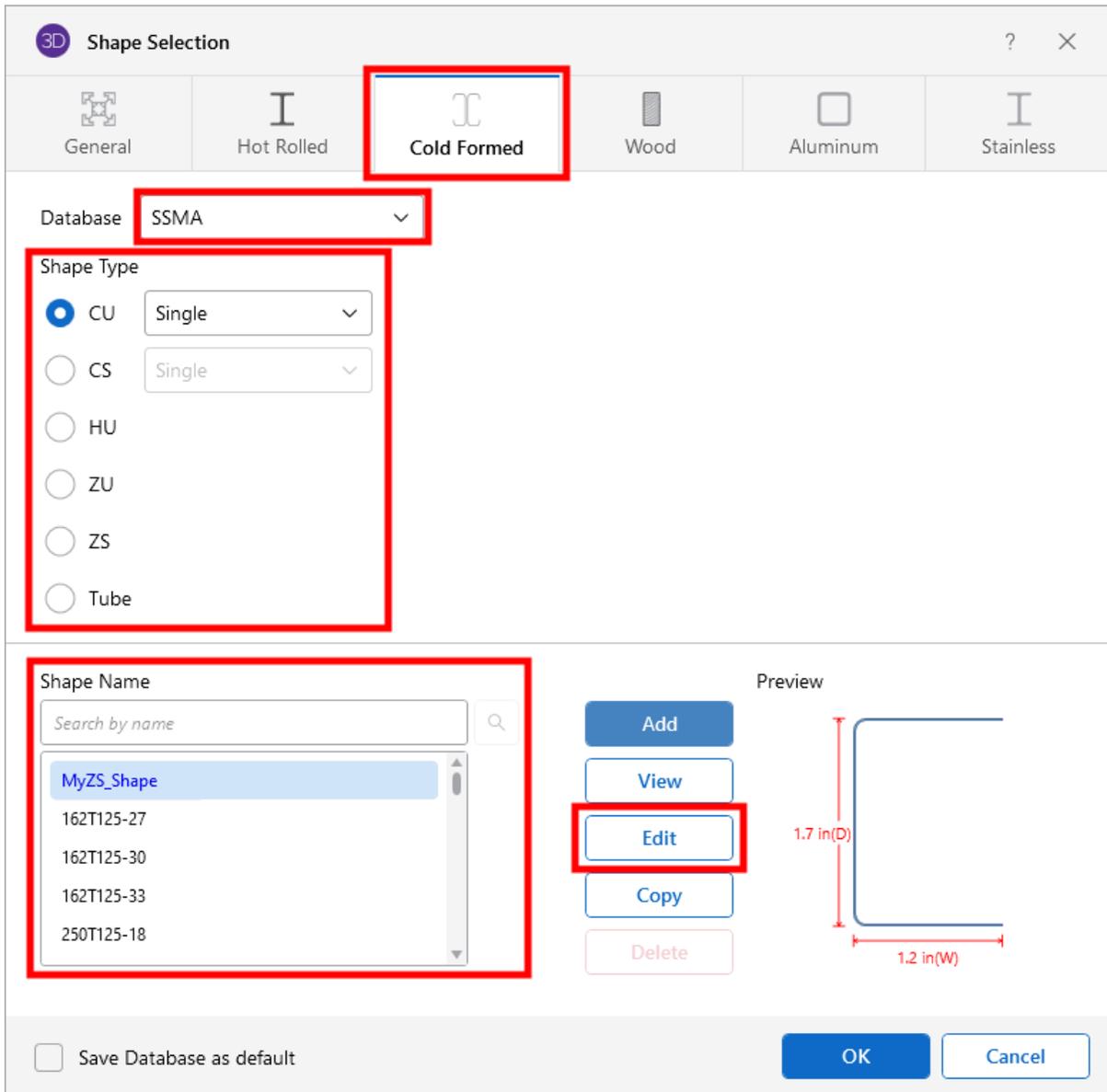
Edit a Database Shape

To edit a database shape:

1. Click the **Shape Database**  icon on the 'Advanced' ribbon (as shown in the following image).



The Shape Selection window opens.



2. Click the **Cold Formed** tab.
3. Click the **Database** \vee down-arrow and choose the database where you want to edit the shape.
4. Click on the **Shape Type** of the shape you want to edit, if not already selected.
5. Click on the **Shape Name** of the shape you want to edit, then click the **Edit** button.
The Edit Shape window opens.

3D Edit Shape
?
×

Shape Name

Geometric Properties

D in

B in

t in

R in

Section Properties

Area in²

I_{yy} in⁴

I_{zz} in⁴

J in⁴

C_w in⁶

r_o in

X_c in

m in

j in

You have the option to input 'Geometric Properties' and 'Section Properties' manually, or you may choose to have the program calculate the Section Properties based on values input for the Geometric Properties.

6. Make your **Geometric Properties** changes, then click **Re-Calc** to have the program calculate the **Section Properties** automatically.

Edit Shape
? X

Shape Name

Geometric Properties

D in

B in

t in

R in

Section Properties

Area in²

I_{yy} in⁴

I_{zz} in⁴

J in⁴

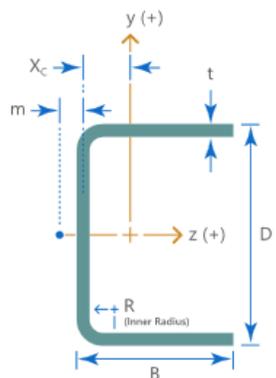
C_w in⁶

r_o in

X_c in

m in

j in



7. Click **OK** to save the shape change(s) and close the Edit Shape window.
You are returned to the Shape Selection window.
8. Click **OK** to save the change to the Shape database and close the Shape Selection window.

**Note:**

- New shapes are added to the bottom of the database in a blue font to show they were not originally from the default database.
- To delete a shape, specify the database and shape type you wish to delete and then click the **Delete** button.
- Only shapes added to the default database (shown in blue) are able to be edited or deleted.

Cold Formed Shape Types

There are five types of shapes. Names for each shape type follow the convention of the manufacturer for each shape. If you know the shape name, you can type the name directly into the **Shape** field on the spreadsheets, or use the search box to type part of the name to narrow the scope of names displayed in the Shape Name list.

C sections without lips (CU)

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

C sections with lips (CS)

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

Z sections without lips (ZU)

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

Z sections with lips (ZS)

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

Hat sections without lips (HU)

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

Closed Box Members (Tube)

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

Double Sections

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

Cold Formed Steel - Design

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

- The 1996 edition of the AISI code with 1999 Supplement (AISI-99 ASD and LRFD)
- The 2001 edition of the AISI code (AISI NAS-2001 ASD and LRFD)
- The 2001 edition of the Mexican code (CANACERO-2001 ASD and LRFD)
- The 2001 edition of the Canadian code (CSA S136-01 LSD)
- The 2004 Supplement of the AISI code (AISI NAS-2004 ASD and LRFD)
- The 2004 Supplement of the Mexican code (CANACERO-2004 ASD and LRFD)
- The 2004 edition of the Canadian code (CSA S136-04 LSD)
- The 2007 edition of the AISI code (AISI S100-2007 ASD and LRFD) including Supplement No.1 (August 09)
- The 2007 Supplement of the Mexican code (CANACERO-2007 ASD and LRFD) including Supplement No.1 (August 09)
- The 2007 edition of the Canadian code (CSA S136-07 LSD)
- The 2010 edition of the AISI code (AISI S100-10 ASD and LRFD contained in the Supplement No. 2 February 2010)
- The 2010 Supplement of the Mexican code (CANACERO-2010 ASD and LRFD contained in the Supplement No.2 February 2010)
- The 2010 edition of the Canadian code (CSA S136-10 LSD contained in the Supplement No.2 February 2010)
- The 2012 edition of the AISI code (AISI S100-12 ASD and LRFD)
- The 2012 edition of the Canadian code (CSA S136-12 LSD)
- The 2012 edition of the Mexican code (CANACERO-2012 ASD and LRFD)
- The 2016 edition of the AISI code (AISI S100-16 ASD and LRFD)
- The 2016 edition of the Canadian code (CSA S136-16 LSD)
- The 2016 edition of the Mexican code (CANACERO-2016 ASD)
- The 2020 edition of the AISI code (AISI S100-16(2020) w/S2-20 ASD and LRFD)
- The 2020 edition of the Canadian code (CSA S136-16(R2021) LSD)
- The 2020 edition of the Mexican code (CANACERO-2020 ASD)

**Note:**

- Back to Back and Face to Face member checks are available for AISI S100-07 and newer.
- Unless otherwise specified, all code references below are to AISI S100-16.

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

Design Parameters

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

Cold Formed Steel Design Parameters														
Primary	Advanced	Hot Rolled	Cold Formed	Wood	Concrete Beam	Concrete Column	Aluminum	Stainless	RISACONNECTION					
	Label	Shape	Length [ft]	Lb y-y [ft]	Lb z-z [ft]	Lcomp top [ft]	Lcomp bot [ft]	L-Torque [ft]	K y-y	K z-z	Cb	R	a [ft]	Function
1	M24	8ZS2.75X105	9			Lbyy								Lateral
2	M196	2-4CS2X059-BB	9			Lbyy							0	Lateral

These parameters are defined for each cold formed member. The entries are explained in the following table.

Design Parameters

Parameter	Description
Label	You may assign a unique Label to all of the members. Each label must be unique, so if you try to enter the same label more than once you will get an error message. You may relabel at any time with the Relabel options on the Tools menu.
Shape	The member Shape or Section Set is reported in the second column. This value is listed for reference only and may not be edited as it is dictated by the entry in the Section/Shape column on the Primary tab.
Length	The member Length is reported in the third column. This value may not be edited as it is dependent on the member end coordinates listed on the Primary Data tab. It is listed here as a reference for unbraced lengths which are discussed in the next section.
Unbraced Length	See the Unbraced Lengths topic.
K Factors (Effective Length Factors)	See the Unbraced Lengths topic.
Sway Flags	See the Unbraced Lengths topic.
Cm - Interactive Bending Coefficient	'Cm Coefficients' are described in Section C5 of the AISI code. If these entries are left blank, they will be automatically calculated.

Parameter	Description
	The C_m value is influenced by the sway condition of the member and is dependent on the member's end moments, which will change from one load combination to the next, so it may be a good idea to leave these entries blank.
C_b - Bending Coefficients	For the cold formed codes, C_b Coefficients are used in the calculation of the nominal flexural strength, M _n . If this entry is left blank, it will be calculated automatically.
R Value	<p>The R Value for cold formed steel design is described in Section 16.2.1 of the AISI code and is used to calculate the moment capacity of beams that have one flange fastened to deck or sheathing. This value only applies to C or Z members and can vary from 0.4 to 0.7 based on the depth of the member (See Table 16.2.1-1 in the AISI Supplement for the actual values).</p> <p>If a value is entered by the user, that value will be used by the program in the moment capacity calculation of the member. There are a number of restrictions that must be met to use this section of the code for moment capacity and the user is responsible to check that these restrictions are satisfied.</p> <div data-bbox="553 1035 1409 1213" style="border: 1px solid black; background-color: #e6f2ff; padding: 10px; margin-top: 10px;"> <p> Note:</p> <ul style="list-style-type: none"> If the R value is entered, the program will use Section 16.2.1 to check moment capacity calculations. </div>
a - Connector Spacing	For double sections, the connector spacing 'a' is used in the calculation of the KL/r. There are also limitations on the connector spacing length that are checked.
Function	The stiffness of gravity members will not be adjusted.

AISI Steel Code Check Results

Access the **Code Check** spreadsheet by selecting the **Results** tab ▶ **Envelope** (or **LC**) button ▶ **Code Check**. Alternatively, on the **Results** toolbar you can select **Code Check**.

Member AISI S100-20: ASD Cold Formed Steel Code Checks (By Combination)																
Hot Rolled Steel		Cold Formed Steel				Wood	Aluminum	Stainless	Concrete							
	LC	Member	Shape	UC Max	Loc[ft]	Shear UC	Loc[ft]	Dir	Pn/Om...	Tn/Om...	Mnyy/Om[k-ft]	Mnzz/Om[k-ft]	Vn...	Vnz...	Cb	Eqn
1	18	M24	8ZS2.75...	0.008	0	0	9	y	14.238	51.707	2.577	10.498	12.1...	10.0...	1.667	H1.1-1
2	18	M196	2-4CS2X...	0.004	0	0	9	y	14.164	35.437	NC	3.705	7.687	-0.6...	1.667	H1.1-1
3	19	M24	8ZS2.75...	0.05	9	0	9	y	14.238	51.707	2.117	7.689	12.1...	10.0...	1.667	H1.2-1
4	19	M196	2-4CS2X...	0.068	9	0	9	y	14.164	35.437	NC	3.287	7.687	-0.6...	1.667	H1.2-1
5	20	M24	8ZS2.75...	0.023	9	0	9	y	14.238	51.707	2.117	7.689	12.1...	10.0...	1.667	H1.2-1
6	20	M196	2-4CS2X...	0.046	9	0	9	y	14.164	35.437	NC	3.287	7.687	-0.6...	1.667	H1.2-1
7	21	M24	8ZS2.75...	0.023	9	0	9	y	14.238	51.707	2.117	7.689	12.1...	10.0...	1.667	H1.2-1
8	21	M196	2-4CS2X...	0.047	9	0	9	y	14.164	35.437	NC	3.287	7.687	-0.6...	1.667	H1.2-1

The final results of the code checking are the code check values **UC Max** and **Shear UC**. These values represent a factored ratio of actual to allowable load for ASD or ultimate load to design strength for LRFD or LSD, based on the provisions of Section H1. Section H2 is also used to check combined bending and shear. So, if this value is less than 1.0, the member passes. If it is greater than 1.0, the member fails. If the value is greater than 9.999 it will be listed as "9.999". The **Shear Check** is based on section C3.2.1. Note that torsional shear, if any, is also included in this check. The location for the shear check is followed by "y" or "z" to indicate the direction of the shear. The **Loc** field indicates at what location the maximum code check occurs measured from the I-joint location of the member. See [Model Display Options – Members](#) to learn how to view the code check results graphically.

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

For **ASD**, the available strengths P_n/Ω , T_n/Ω , M_{nyy}/Ω , M_{nzz}/Ω , V_{ny}/Ω , and V_{nz}/Ω are calculated for the member.

For **LRFD** or **LSD**, the values for factored compression ΦP_n , factored tension ΦT_n , factored moments ΦM_{nyy} and ΦM_{nzz} , and factored shears ΦV_{ny} and ΦV_{nz} are displayed.

P_n is calculated according to the provisions of AISI 2016, Chapter E. **T_n** is based on Chapter D. The **M_n** values are calculated based on Chapter F. The **V_n** values are calculated based on Chapter G. Note that for RISA-3D, "zz" corresponds to "xx" in the AISI code, i.e. RISA-3D substitutes M_{nzz} for M_{nx} , to maintain consistency with the member local axis system.

C_b will be calculated automatically if not specifically entered by you, which is conservative. For AISI S100-12, the C_m coefficients described in Section C5 are also listed. These also are influenced by the sway flag settings.

The final column **Eqn**, lists the controlling equation for the code check. This will be one of the equations from Section H1 or H2.

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

The moving load results are enveloped and the governing load combination and step location is shown for each result value under the "LC" column. The first number is the load combination, the second is the step number: (load combination - step number). See [Moving Loads](#) to learn more.

Cold Formed Steel Detail Report

The image below is the last portion of a detail report for a Cold Formed Steel member showing the member warnings, code checks, and section properties.

AISI S100-20: ASD Code Check					
Combined Shape Design does not consider weak axis 'My' Moments, nor weak axis 'Vz' Forces.					
Limit State	Gov. LC	Required	Available	Unity Check	Result
Applied Loading - Bending/Axial	19	-	-	-	-
Applied Loading - Shear + Torsion	18	-	-	-	-
Axial Tension Analysis	-	0 k	35.437 k	-	-
Axial Compression Analysis	-	0.963 k	14.164 k	-	-
Flexural Analysis (Strong Axis)	-	0 k-ft	3.287 k-ft	-	-
Shear Analysis (Major Axis y)	-	0.000939 k	7.687 k	0.0001222	PASS
Bending & Axial Interaction Check (UC Bending Max)	-	-	-	0.068	PASS

Below describes the Cold Formed Steel specific parameters. Refer to the [Member Detail Report](#) for more information on the values used to perform the code check.

Cw- Torsional warping constant. This will only be shown for Back-to-Back and Face-to-Face shapes. See [Unbraced Lengths](#) topic to understand how the bracing affects this calculation.

a (conn. Spacing)- distance between connections locations of built-up shape. This spacing is checked by code prescribed limits and is used in the calculation of KL/r . This will only be shown for Back-to-Back and Face-to-Face shapes.

A eff. (Fy)- The effective Area used in determining the nominal strength of the section, where the maximum stress is the yield stress.

A eff. (Fn)- The effective Area calculated at stress F_n , where F_n is the nominal buckling stress.

Iy eff. – The effective Moment of Inertia about the y-axis.

Iz eff.– The effective Moment of Inertia about the x-axis.

Sy eff(L) - Effective section modulus about the Y axis for the extreme left fiber.

Sy eff.(R) Effective section modulus about the Y axis for the extreme right fiber.

Sz eff(T) - Effective section modulus about the Y axis for the extreme top fiber.

Sz eff.(B) - Effective section modulus about the Y axis for the extreme bottom fiber.

The effective properties listed above are based on the governing values used to determine the M_{ny} and M_{nz} . The labels on the effective properties will always state "Sy eff" however depending on the

governing value it might be the fully unreduced section (S_f), effective section calculated relative to the extreme compression fiber at F_c (S_c) or effective section calculated relative to the extreme compression or tension fiber at F_y (S_e).

When the Combined Tensile Axial load and Bending unity equations govern, the Detail Report will display the S_f . When the Combined Bending and Shear unity equations govern, the Detail Report will display the S_e . When the Combined Compressive Axial load and Bending unity equations govern, the display depends if governing equations are: local yielding (S_e), Lateral Torsional Buckling (S_c) or Distortional Buckling (S_f).

- See [Spreadsheet Operations](#) to learn how to use **Find**, **Sort** and other options.
- See [Results View Settings - Members](#) to learn how to plot member results.

Assumptions and Limitations

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

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

Torsional warping effects are not included. Torsion stiffness and stress are calculated as pure torsion only. Web crippling is not considered.

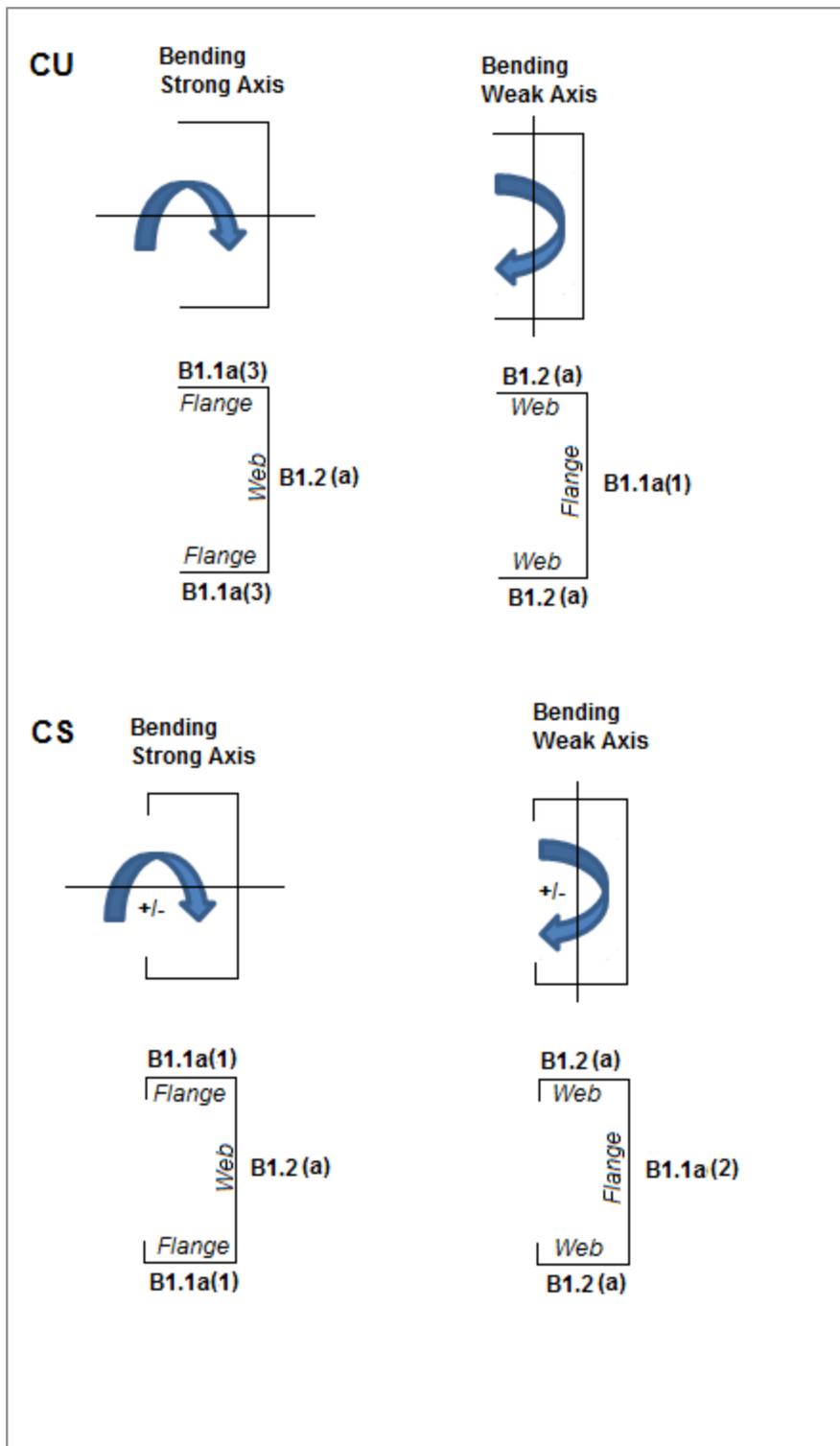
See the [Unbraced Lengths](#) for Cold Formed Steel section for the full details.

Kt in Section F2.1 is assumed to be 1.0. Effects of shear stiffeners for Section G2 are not considered.

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

H Shapes – Hat sections in bending about the y-y axis such that the brims are in compression are assumed braced such that the brims cannot each fail in lateral torsional buckling independently. For AISI S100-12 and older, lateral-torsional buckling is not checked for HU shape.

Slenderness Limitations - The w/t limits of Section B4 are enforced. However, the shear lag effects (Section B4.3) are not enforced. Below is an example showing the slenderness checks (AISI S100-12) for a CU and CS shape:



Double Sections

The distance between connector spacing is checked per s_{\max} (AISI Eq I1.1-1) however the spacing is considered uniform. The concentrated loads are not checked (AISI Eq I1.1-2) and there is no covered plate sections available (AISI Section I1.3).

The **Slenderness ratio** is altered based on the intermediate fastener spacing when applicable (AISI Eq I1.2-1). The Detail Report will display the modified KL/r if it governs. This modification is applied if the spacing meets the code limits that states a/r_i does not exceed one-half the governing slenderness ratio of the member. The welds length and/or weld/fastener strengths are not checked in RISA (Section I1.2b,c). If the fastener spacing does not satisfy the aforementioned conditions, the global buckling stress of the built-up member is calculated based on the section properties of the individual stud members.

For Back to Back sections, the **Weak Axis** moments (M_y) or shear forces (V_z) for back-to-back shapes are not considered in the code check and this is noted in the Detail Report. The program will give you analysis information in the Detail Report and spreadsheets.

For Face to Face sections, the Torsional Constant (J) and Torsional warping constant (C_w) are calculated based on a tube shape if the Connector spacing is set to fully connected ($a=0$). If the connector spacing is greater than zero ($a>0$), the J and C_w calculation will be based on the summation of each element.

Special Messages

AISI Code Check Not Calculated

This message is displayed when the member is not defined with a database shape, or a steel code is not specified on the **Model Settings**, or no units were specified.

Can't do code check, stiffener $D/w > 0.7$ (Table B4.1-1)

The ratio D/w exceeds the limiting criteria listed in Table B4.1-1 (AISI S100-16) or Section B4.2 (AISI S100-12) for simple lip stiffeners. ("D" and "w" are length of the stiffener and the flat length of the flange as defined in B4.2)

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

The angle (γ) for a simple lip stiffener must be greater or equal to 40 degrees or less than or equal to 140 degrees per the criteria in Commentary Section 1.3 (AISI S100-16) or Section B4.2 (AISI S100-12). The angle γ for this shape is outside this range.

Can't do code check, flange $w/t > \text{limit}$ (Table B4.1-1)

The ratio w/t exceeds the limiting criteria listed in Table B4.1-1 (AISI S100-16) or Section B1.1 (AISI S100-12) for flanges. A value of 60 is used per the AISI code for unstiffened elements and

elements stiffened with simple lips.

Can't do code check, web $h/t > 200$ (Table B4.1-1)

The ratio h/t exceeds the limiting criteria listed in Table B4.1-1 (AISI S100-16) or Section B1.2 (AISI S100-12) for webs. The program currently considers all webs as unreinforced, so a value of 200 is used as the limit.

Can't do code check, Connector spacing 'a' $> L/6$ per I1.1

The connector spacing "a" exceeds the limiting criteria listed in Section I1.1 (AISI S100-16) or Section D1.1 (AISI S100-12). The "L" in this equation is the beam span (full length of the member).

Cold Formed Steel Walls - Design Rules

The CFS wall panel element allows you to easily model, analyze and design CFS walls for in plane loads. Here we will explain the CFS specific inputs and design considerations. For general wall panel information, see the [Wall Panels](#) topic. For information on CFS wall design considerations, see the [CFS Wall - Design](#) topic. For CFS wall results interpretation, see the [CFS Wall Results](#) topic.

Unity Check

Wall Panel U.C. Parameters					
Masonry Lintel	Wood Wall (Studs)	Wood Wall (Fasteners)	CFS Wall (Studs)	CFS Wall (Fasteners)	
Unity Check	Concrete Wall (Rebar)	Concrete Wall (Misc)	Masonry Wall	Masonry In	Masonry Out
	Label	Max Bending Chk	Max Shear Chk		
1	8-in CFS Wall	1	1		
2	6-in CFS Wall	1	1		

Max Bending Chk

Wall Panel Unity Check Spreadsheet Columns

Column Header	Description
Max Bending Chk	The Max Bending Chk column defines the hold-down maximum code check value. Currently, hold-down design/optimization is not available for CFS walls. However, this will be made available in a future release.
Max Shear Chk	The Max Shear Chk column defines the shear panel selection maximum code check value. Typically this is always 1.0, but it allows the option of having the program pick out a shear panel layout that is not at maximum capacity. Putting a value of 0.8 chooses a panel thickness and nailing that is at 80% of capacity.

CFS Wall (Studs)

CFS Wall Panel Parameters								
Unity Check		Concrete Wall (Rebar)		Concrete Wall (Misc)		Masonry Wall	Masonry In	Masonry Out
Masonry Lintel		Wood Wall (Studs)		Wood Wall (Fasteners)		CFS Wall (Studs)		CFS Wall (Fasteners)
	Label	Top Plate	Sill Plate	Studs	Stud Space [in]	Header Size		
1	8-in CFS Wall	600T125-33	600T125-33	600S137-33	12	2-600S137-43-BB		
2	6-in CFS Wall	600T125-33	600T125-33	600S137-33	12	2-600S137-43-BB		

CFS Wall Panel Parameters Spreadsheet Columns

Column Header	Description
Top Plate	The Top Plate column lets you specify the member to be used as a top plate for your wall. A top plate is a member that runs continuously along the top of the wall studs.
Sill Plate	The Sill Plate column lets you specify the member to be used as a sill plate for your wall. A sill plate is a member that runs continuously along the bottom of the wall studs.
Studs	The studs column lets you specify the member to be used for studs in your wall. Studs are vertical members in the wall, attached to the sill plate at the bottom and the top plate at the top. Note that you can use back-to-back or toe-to-toe options as well.
Stud Space	<p>The Stud Space column lets you specify the spacing between the studs within the wall.</p> <p>You may specify a minimum and maximum spacing of wall studs. The program can then optimize the stud spacing based on axial design only. For information on how the optimization works, see the Wood Wall - Design topic.</p> <ul style="list-style-type: none"> • If you specify the maximum and minimum stud spacing as the same value, then we will use that value exclusively. • Out of plane design is not performed for wood walls, so any optimization is based only on axial forces in the studs.
Header Size	The Header Size column lets you define the default header size for all openings. Note that the program does not provide a design for CFS headers at this time.

Design Rules - CFS Wall (Fasteners)

Additional CFS Wall Panel Parameters													
Unity Check	Concrete Wall (Rebar)	Concrete Wall (Misc)	Masonry Wall	Masonry In	Masonry Out	Masonry Lintel	Wood Wall (Studs)	Wood Wall (Fasteners)	CFS Wall (Studs)	CFS Wall (Fasteners)			
	Label	Type of Wall	Sheathing Material	Schedule	Design Method	Min Panel Thick [in]	Max Panel Thick [in]	Fastener Min Edge Space	Fastener Max Edge Space	HD Chords	Eccentricity	Ecc Dist [in]	HD Deflection [in]
1	8-in CFS Wall	Type I - Segmented	Steel Sheet sheathing	AISI 2020 Steel	Table	0.003	2	2-in.	6-in.	6005137-33	<input type="checkbox"/>	0	1
2	6-in CFS Wall	Type I - Segmented	Steel Sheet sheathing	AISI 2020 Steel	Table	0.003	2	2-in.	6-in.	6005137-33	<input checked="" type="checkbox"/>	1.5	1

Design Rules - CFS Walls (Fasteners) Spreadsheet Columns

Column Header	Description
Type of Wall	The Type of Wall column shows the design methodology of the CFS wall panel. Currently, CFS walls can only be designed using the Type I - Segmented method.
Sheathing Material	The Sheathing Material column lets you choose which sheathing material you would like to be used for lateral design optimization. The specific sheathing material that you choose automatically populates the corresponding sheathing database in the adjacent Schedule column.
Schedule	<p>The Schedule column lets you choose the sheathing database and Panel Group you would like to use for design optimization.</p> <p>By selecting the Select Individual Panel option, an individual panel type can be assigned.</p> <p>For information on how the optimization works, see the CFS Wall - Design topic. For more information on this schedule, as well as information on how to edit or create your own custom schedule, see Appendix G-CFS Design Databases.</p>
Design Method	<p>The Design Method column lets you choose the method in which the lateral capacity is determined by the program. If you choose the Table option, the program references tabulated shear capacities in the AISI S240 and AISI S400 manual. The Effective Strip Method is only available for the Steel Sheet sheathing material option.</p> <p>For more information on this see the CFS Wall - Design topic.</p>
Min Panel Thick	<p>The Min Panel Thick column lets you set minimums for the thickness of the sheathing to be designed.</p> <p>If the same value is set for both max and min, then that is the thickness used.</p>
Max Panel Thick	<p>The Max Panel Thick column lets you set maximums for the thickness of the sheathing to be designed.</p> <p>If the same value is set for both min and max, then that is the thickness used.</p>
Fastener Min Edge Space	The Fastener Min Edge Space column lets you set minimums for the spacing of the nails that fasten the sheathing to the boundary members

Column Header	Description
	(top plate, sill plate, hold down chords). Note that a 12" spacing is assumed for all field nailing (nails fastening the sheathing to the internal studs).
Fastener Max Edge Space	The Fastener max Edge Space column lets you set maximums for the spacing of the nails that fasten the sheathing to the boundary members (top plate, sill plate, hold down chords). Note that a 12" spacing is assumed for all field nailing (nails fastening the sheathing to the internal studs).
HD Chords	The HD Chords columns lets you choose what member size you would like to use for the Hold Down Chords (Posts) at both ends of the wall panel.
Eccentricity	The Eccentricity checkbox lets you control whether the eccentricity distance of the hold-down (input into the Ecc Dist column) is used in the calculation of the hold-down force. For more information, see the Hold Down force calculation section in the CFS Wall - Design topic.
Ecc Dist	The Ecc Dist column lets you input an eccentricity distance to be used in the calculation of the hold down force. This distance should be measured from the centerline of the HD chord to the actual hold-down anchorage.
HD Deflection	The HD Deflection column lets you input the manufacturer hold down deflection which is used in the reported shear wall design deflection calculation.

CFS Walls

Cold Formed Steel Walls - Design

The CFS wall panel element allows you to easily model, analyze and design CFS walls for in-plane loads. Here we will explain the specific inputs and design considerations. For general wall panel information, see the [Wall Panels](#) topic. For CFS wall results interpretation, see the [CFS Wall Results](#) topic.

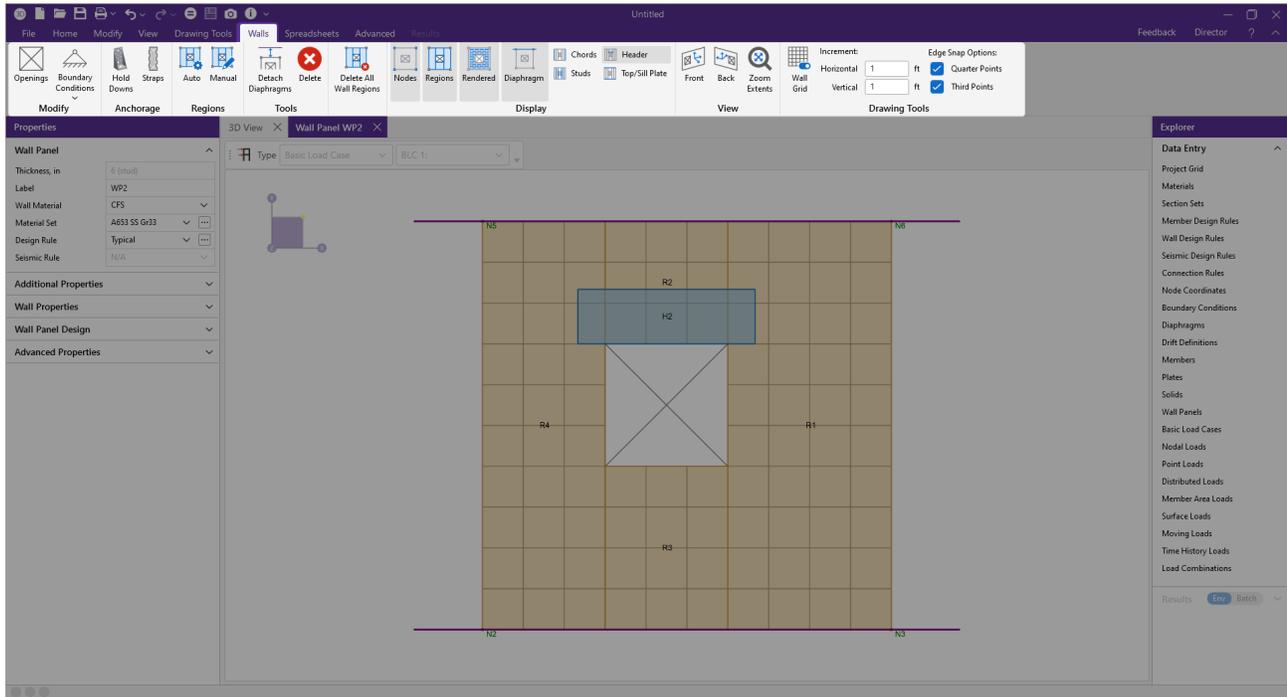
For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **CFS Walls**

Full code checking and design can be performed on the panel sheathing, studs, and chords based on the following codes:

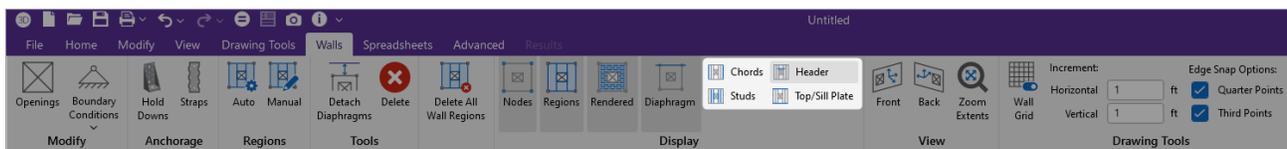
- The AISI S240-15 CFS Structural Framing
- AISI S400-15 w/S1-16 Seismic Design of CFS Structural Systems

CFS Wall Input

The **Walls** ribbon gives some specific information and options for modeling/analysis of CFS walls. It can be accessed by double clicking on a wall panel in the 3D view window.



CFS View Controls



Button/Icon	Name	Description
Chords	Chords	Allows you to turn the display of wall panel region chords on and off.
Studs	Studs	Allows you to turn the display of the studs on and off.
Header	Header	Allows you to turn the display of the studs on and off.
Top/Sill Plate	Top/Sill Plate	Allows you to turn the display of the top/sill plates on and off.

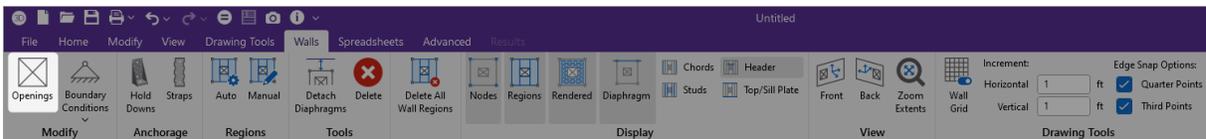
Design Rules

You must set up design rules for the stud/chord sizes, as well as make a database selection for shear panels. This is done in the **Wall Design Rules** spreadsheet in the **CFS Wall (Studs)** and **CFS Wall (Fasteners)** tabs. See the [CFS Wall - Design Rules](#) topic for more information.

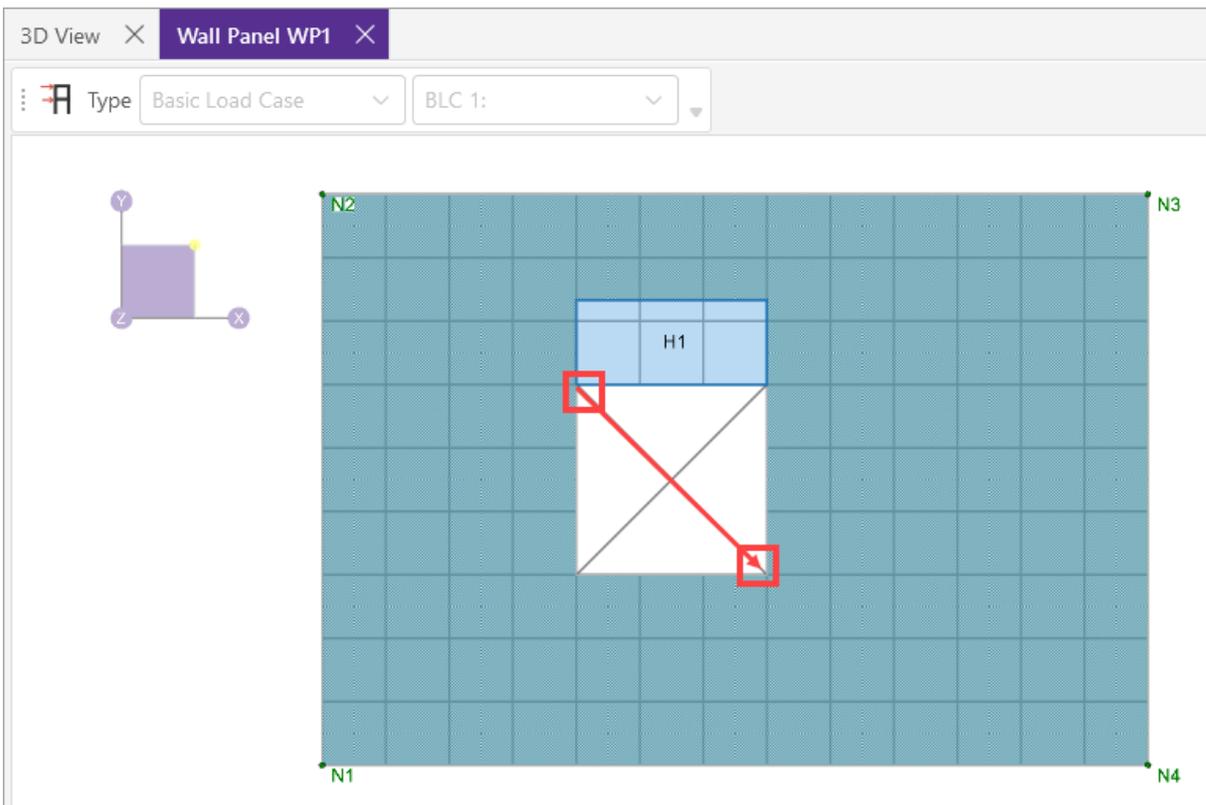
Create Openings in CFS Walls

In the **Walls** ribbon, you have the option of adding rectangular openings to CFS wall panels. To draw an opening:

1. Click the **Openings** icon.



2. Click on two nodes or grid intersections which make up the two diagonal corners of your opening.



Note: Currently header design is not available for CFS walls.

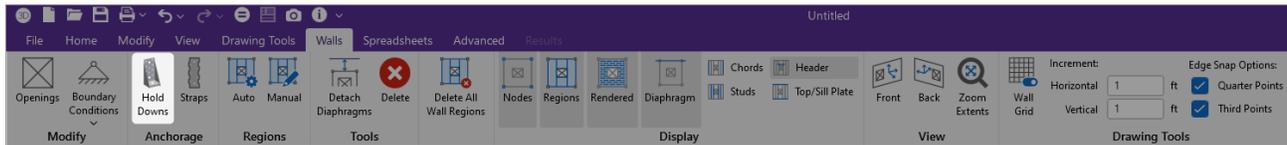
When an opening is drawn, a header beam is automatically created above the opening (as indicated by the H1 block in the image).

Hold-Downs and Straps

Hold-downs and straps are automatically added to your walls in their required locations. For a Type I - Segmented design, you must have hold-downs or straps at the bottom corners of each of your design segments. The program will not permit the drawing of hold-downs or straps at locations where they are not allowed.

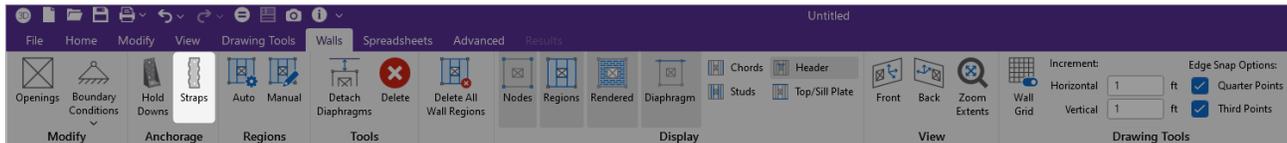
If there are custom locations that you want to add hold-downs or straps, you can do this from the **Walls** ribbon .

Hold downs can represent the anchorage of your wall to the foundation or the connection of shear wall chords between floors. To add hold-downs to the base of your wall, first select the **Hold Downs** icon in the 'Walls' ribbon.



Hold downs must be added after regions are created and can only be added at the corners of regions. Hold down requirements depend on the type of wall design you are performing.

Straps represent the connection of the current wall panel to a wall panel below. To add straps to the base of your wall, first select the **Straps** icon in the 'Walls' ribbon.



Straps also can only be added after regions are created and can only be added at the corners of regions. Strap requirements also follow the same logic as hold downs as to where they must be defined as far as regions are concerned.

**Note:**

- Hold-downs and straps are not designed for CFS Walls. However, the required forces will still be reported. More information on this can be found in the CFS Wall Results topic.
- All boundary conditions for wall panels should be defined in the wall panel editor. Adding external boundary conditions can create problems.
- The locations of hold-downs and straps define where the program will calculate tension forces in your walls.
- By default, the program will automatically add hold-downs to the base of walls that have boundary conditions applied to them and will apply straps to walls that have walls below.
- All Straps and Hold-downs are placed at the base of walls only. So, if you would like to put a strap or hold-down between floors, you would apply it to the base of the upper wall.

General Requirements for Shear Walls

The design of CFS shear walls per the available design codes requires that many criteria are satisfied before a wall can be considered adequate. For RISA to work within this framework, we require that certain modeling practices be followed. Outlined below are many general wall modeling practices and limitations as well as specific requirements for the Type I - Segmented method that is available for CFS Walls in the program.



Note: RISA-3D does not currently support Type I with FTAO or Type II walls.

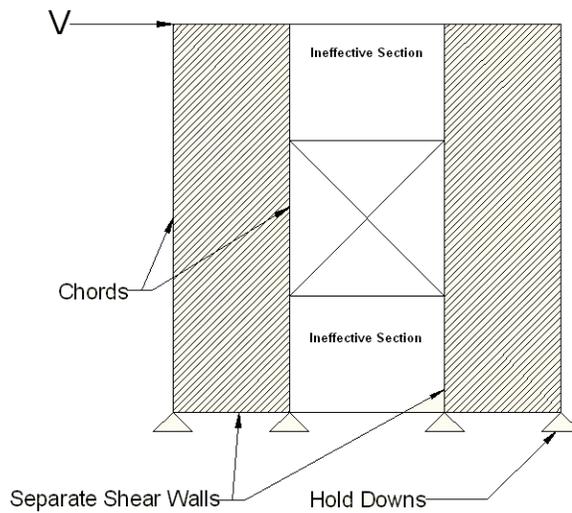
Type I - Segmented Method

The Type I - Segmented methodology is defined in Section B5.2 of the *AISI S240-15 North American Standard for Cold-Formed Steel Structural Framing*.

Where there is a wall panel with openings, the area above and below the openings is disregarded and the wall is designed as being made up of separate, smaller shear walls.

Like all wall panels, the segmented CFS wall is broken into a series of meshed plate elements to represent the overall wall. The portions of the segmented shear wall that are considered "ineffective" in resisting shear are modeled with a plate elements that have a significantly reduced shear stiffness so that they will not receive any significant moment or shear from the FEM analysis.

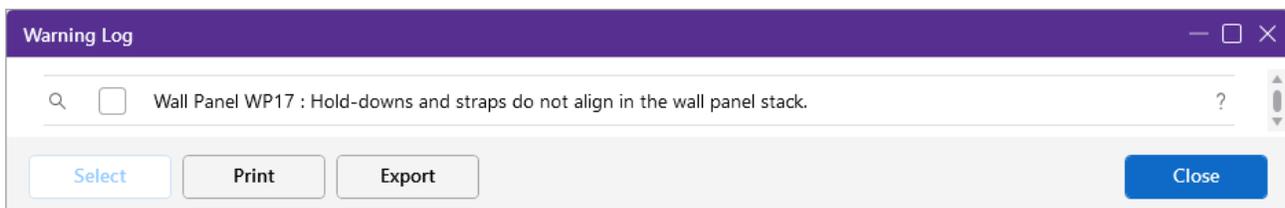
See the diagram below for more information:



In addition, the out of plane stiffness and in plane stiffnesses of the segmented CFS wall are modeled separately based on different assumed plate thicknesses. This is done to insure that the shear stiffness is based entirely on the properties of the sheathing and is not influenced by the out-of-plane stiffness of the wall studs.



Note: If you have several stacked Segmented wall panels with misaligned openings, you will receive the Warning Message shown below upon solution. This message means that RISA-3D has assumed that the strap force from the above wall panel will be spread out across the region directly under it. Therefore, you need to be aware of this assumption and detail the wall panel accordingly.



- A shear panel design will be chosen for the worst-case region in a segmented wall. That panel will then be used for all regions in that wall. The worst-case region is the one that has the highest Shear UC value.

Geometric Requirements and Limitations

The program will check the geometric requirements per AISI S240-15 B5.2.1.1 and AISI S400-E1.4 as required unless otherwise noted below. The sections noted below are assumed to be satisfied and are not checked.

**Note:**

- AISI S400-E1.4 parts (d) - (g) and (k) - (r)
- AISI S240 B5.2.2.3.2.1.1 parts (c) and (e)
- AISI S400 E2.3.1.1.1.1 parts (c) and (e)

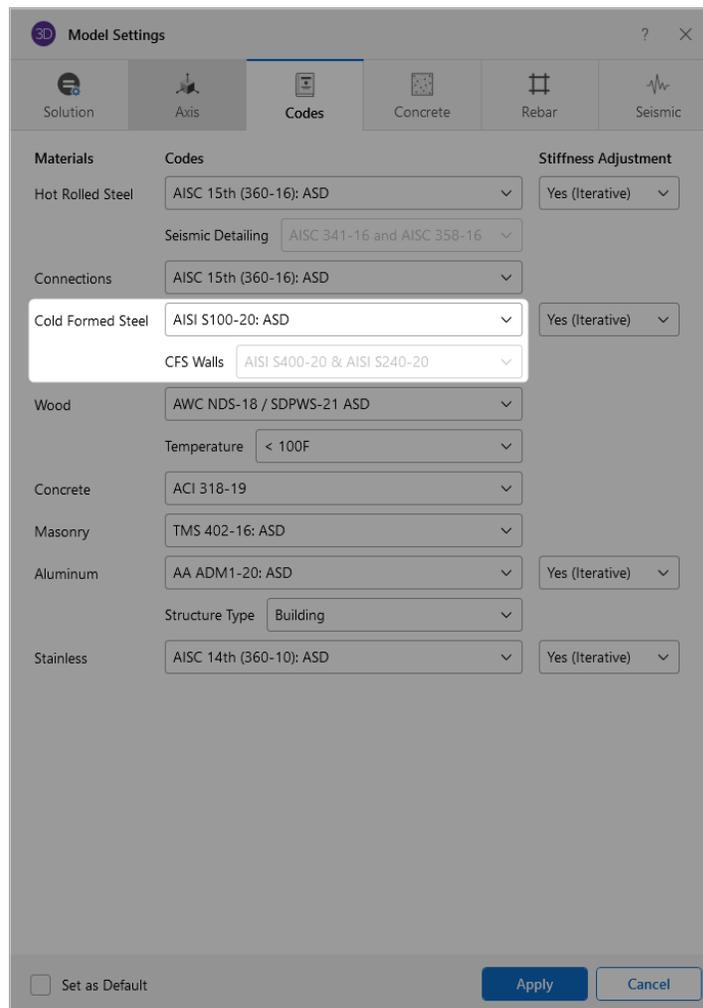
General Program Functionality and Limitations

Assigning Code and Design Methodology

CFS wall designs will be based on the *AISI S240-15 North American Standard for Cold-Formed Steel Structural Framing* and *S400-15 w/S1-16 North American Standard for Seismic Design of Cold-Formed Steel Structural Systems*. You can select these codes in the CFS Walls dropdown within the Codes section of the Model Settings. If you set this code to “None”, the program will not perform design for the CFS walls.

The design properties of CFS stud and chords within the wall are referenced from the AISI S100 specification referenced in the Cold Formed Steel dropdown.

In this manner, the design methodology (ASD or LRFD) for the entire wall panel (stud, chord and sheathing design) is determined by the design method selected in the Cold Formed Steel dropdown. For example, if the AISI S100-16: ASD code is selected as shown below, the CFS walls will use ASD factors in the stud, chord and sheathing designs.

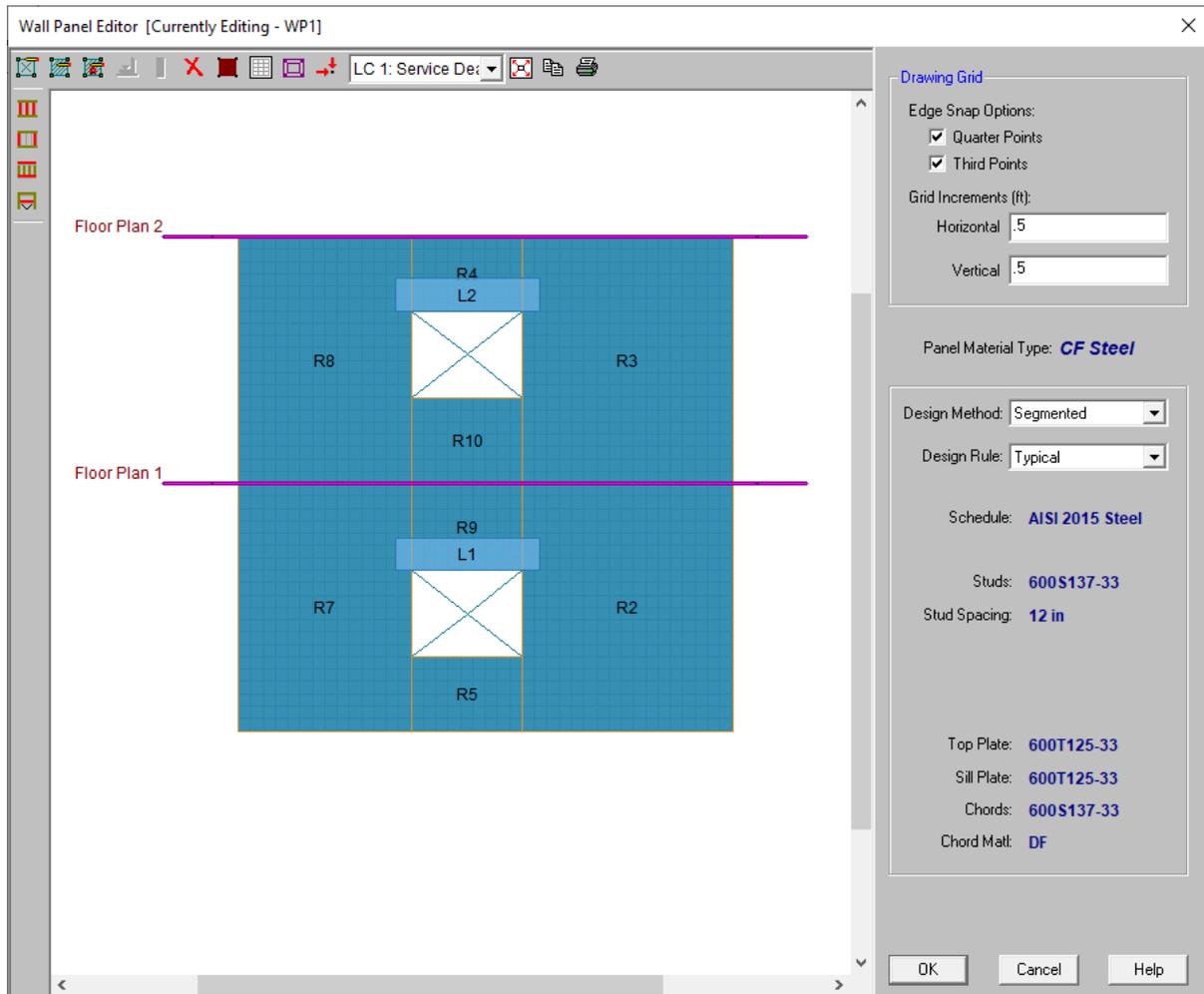


RISAFloor and RISA-3D Interaction

When using RISAFloor and RISA-3D in combination, the interface transitions nicely between the two programs. Here is a quick walk through of this interaction.

Input Interface

1. Model the entire building (gravity and lateral members) within RISAFloor.
Be sure to model all openings and regions for all of the wall panels in the model.



2. Add loading and solve the model.
3. Take the model into RISA-3D via the Director tool.
4. Once in RISA-3D you must add your hold downs and straps to your wall panels.

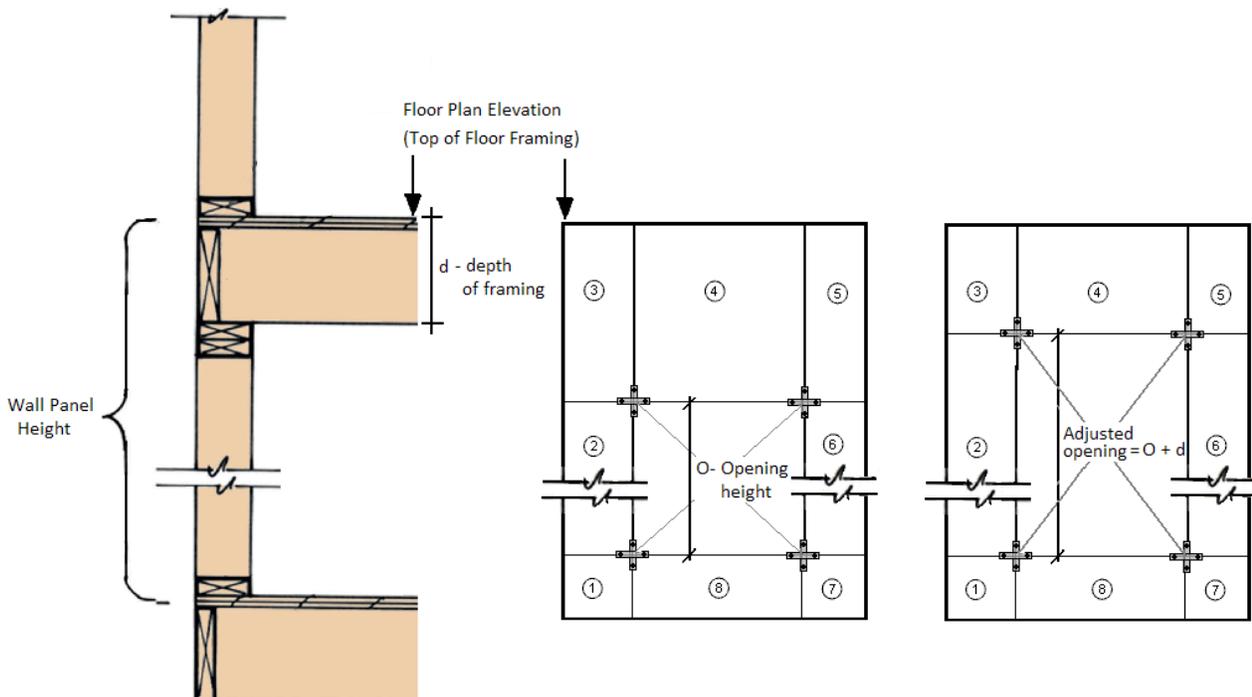
**Note:**

- You can not modify your openings or regions in RISA-3D. All region and opening modifications must be taken back to RISAFloor to be done.
- Hold downs and straps can not be added to wall panels in RISAFloor.
- Hold downs are required at the corners of all full height regions in the wall panel for the Segmented design method.
- The Design Rule and Design Method can be changed in either program at any time.

Modeling Tips- Platform Framing

The CFS wall height in RISA is measured from the bottom of the sill plate to the top of the floor framing as shown below. However, platform framing causes the wall height to be significantly shorter than a RISA model would represent.

This is especially problematic for the FTAO design method where the region above openings is assumed to be much larger than it is built. However, currently only the Type I - Segmented method is available and the portions above and below the opening are not used to transfer shear forces.



Limitations for Hold-downs / Straps (Including deformation)

- The program does not currently have a database for or optimize hold-downs/straps for CFS walls.
- The manufacturer listed or user defined hold-down deformation can be input in the Wall Design Rules spreadsheet for inclusion in the design deflection calculation. See the CFS Wall - Results topic for more information.

Automatic Boundary Conditions

- In RISA-3D, if no boundary conditions or hold downs are defined for CFS wall panels at the lowest level of the structure, the program will automatically create hold downs at the corners of the wall panel. If you do not want the hold downs to be automatically created, define a "free" boundary condition at the base of the wall panel in the wall panel editor.

Deflection

The program will report the design deflection for CFS shear walls that have wood or steel sheathing.

- There is currently no code check for drift or deflection for CFS shear wall panels.

Stud Design

Studs are only designed for load combinations which do not contain a wind or seismic load. The maximum axial load, determined as an envelope force from all of the "gravity" load combinations which have been solved, is determined for each region.

Studs are checked per AISI S240-15 B3.2: Wall Stud Design. The program currently only uses the All Steel Design methodology in which case bracing from the attached sheathing or structural bracing is ignored. Therefore the unbraced length for the buckling of the stud is taken as the full wall height. The top and bottom track thicknesses are considered negligible and are not deducted from the total height.



Note:

- The program currently does not optimize the stud size or spacing.
- The program will use an effective length factor, K , equal to 1.0 for the design of the studs.

Stud force demand and number of studs calculation:

The program first divides the maximum axial force in the region by the length of the region. Then it will multiply this distributed force by the stud spacing to get a maximum stud demand.

The program calculates the number of studs based on the following procedure:

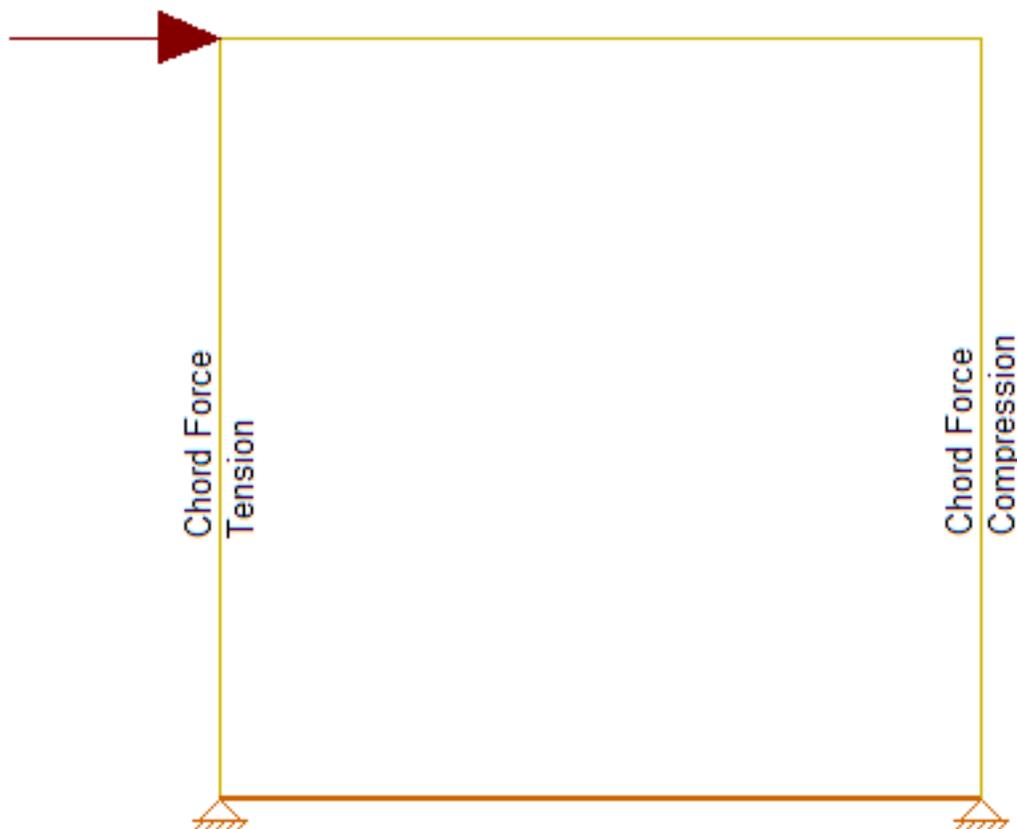
1. Determines the number of spaces between studs by dividing the region length by the stud spacing.
2. Rounds this number up to the nearest whole number.
3. Determines the number of studs by adding one to this number.

Openings

- Currently, the program does not perform header design. This will be available in an updated version.

Chord Design

The chord design is based on forces that are calculated differently for Compression versus Tension. The tension chord force is calculated including the dead load stabilizing moment. The compression chord force includes the only the tributary area of one stud spacing in the compression force. For Segmented design, the chord forces are found based on each region.



Chord forces are calculated per the following equations:

$$\text{Compression Force} = \frac{M}{L} + \frac{P}{n}$$

$$\text{Tension Force} = \frac{-M}{L} + \frac{P}{2}$$

Where:

- M = Moment at the base of the wall
- L = Length of the wall
- P = Axial force at the base of the wall
- n = Number of Studs = Length/Stud Spacing



Note:

- In RISA-3D, the edge of the CFS walls represent the centerline of the chord member. Therefore, the depth of the chord member will not affect the chord force calculation since the centerline of the chord member will always be assumed at the edge of the wall.
- The eccentricity checkbox in the Wall Design Rules spreadsheet will not affect the chord force calculation.
- RISA-3D models the shear walls using only the sheathing. The vertical resistance occurs only at the tension and compression chords. Thus, if two wall panels are stacked on top of each other, the load transfer will only happen at the chord locations. Therefore, the lateral analysis should agree very well with hand calculations. However, it also means that gravity load design may be more appropriate in RISAFloor.

Hold-Down Force

Hold-Down forces are calculated per the following equations:

Note that the hold-down force calculation depends upon whether the eccentricity checkbox has been checked, and the value entered in the eccentricity field.

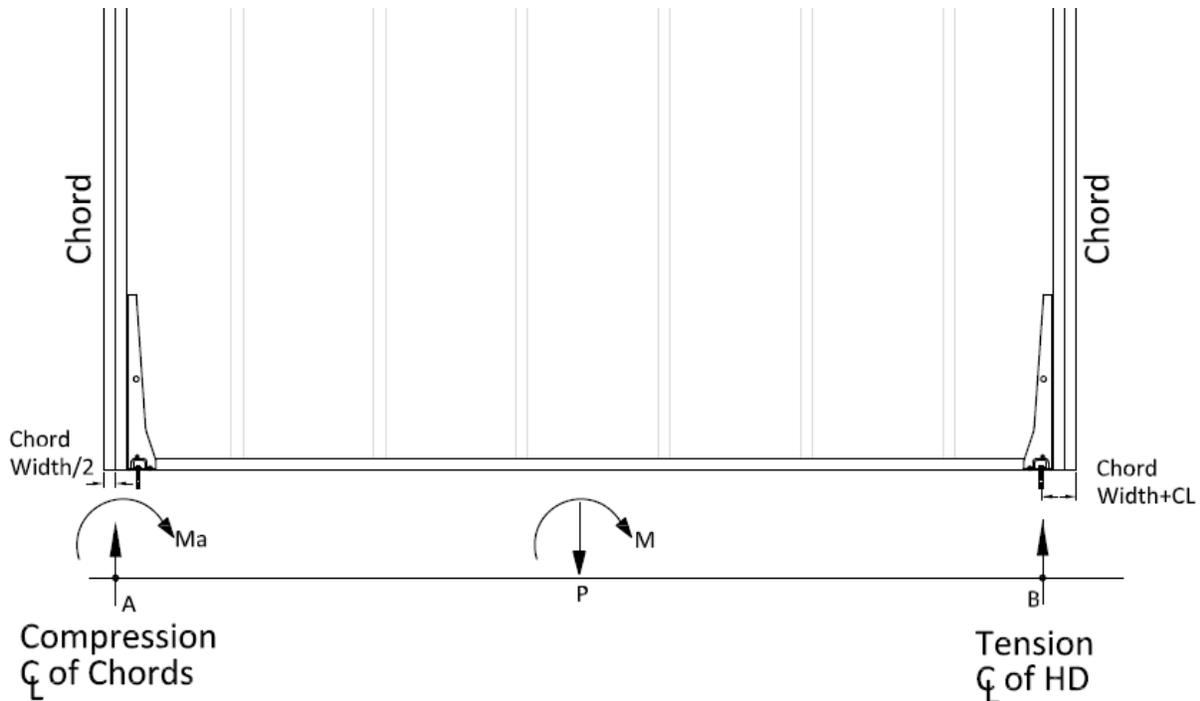
- When the "[Eccentricity](#)" check-box is unchecked in the CFS Wall Design Rule (Fasteners tab), the hold-down force is exactly equal to the chord force indicated above.

Additional CFS Wall Panel Parameters													
Unity Check	Concrete Wall (Rebar)	Concrete Wall (Misc)	Masonry Wall	Masonry In	Masonry Out	Masonry Lintel	Wood Wall (Studs)	Wood Wall (Fasteners)	CFS Wall (Studs)	CFS Wall (Fasteners)			
Label	Type of Wall	Sheathing Material	Schedule	Design Method	Min Panel Thick [in]	Max Panel Thick [in]	Fastener Min Edge Space	Fastener Max Edge Space	HD Chords	Eccentricity	Ecc Dist [in]	HD Deflection [in]	
1	12" [---]	Type I - Segmented	Steel Sheet sheathing	AISI 2020 Steel	Table	0.003	2	2-in.	6-in.	600S137-33	<input type="checkbox"/>	0	1
2	6"	Type I - Segmented	Steel Sheet sheathing	AISI 2020 Steel	Table	0.003	2	2-in.	6-in.	600S137-33	<input checked="" type="checkbox"/>	1.5	1

- When the "Eccentricity" check-box is checked in the CFS Wall Design Rule (Fasteners tab), an eccentricity distance should be indicated in the "Ecc Dist" column. This distance should be the distance between the hold-down and the centerline of the chord member.

Additional CFS Wall Panel Parameters													
Unity Check	Concrete Wall (Rebar)	Concrete Wall (Misc)	Masonry Wall	Masonry In	Masonry Out	Masonry Lintel	Wood Wall (Studs)	Wood Wall (Fasteners)	CFS Wall (Studs)	CFS Wall (Fasteners)			
Label	Type of Wall	Sheathing Material	Schedule	Design Method	Min Panel Thick [in]	Max Panel Thick [in]	Fastener Min Edge Space	Fastener Max Edge Space	HD Chords	Eccentricity	Ecc Dist [in]	HD Deflection [in]	
1	12" [---]	Type I - Segmented	Steel Sheet sheathing	AISI 2020 Steel	Table	0.003	2	2-in.	6-in.	600S137-33	<input type="checkbox"/>	0	1
2	6"	Type I - Segmented	Steel Sheet sheathing	AISI 2020 Steel	Table	0.003	2	2-in.	6-in.	600S137-33	<input checked="" type="checkbox"/>	1.5	1

The Hold-Down force is calculated by finding the Tension chord force. In order to accommodate an unsymmetrical vertical load on the wall, the program adopted a more accurate approach by calculating the moment at the compression side first and then finding the resulting reaction on the tension side.



The resulting equation to determine the hold-down force is as follows:

$$\text{Moment at Point } a = M_a = -M + \left(P * \left(\frac{L}{2} - \frac{b_c}{2} \right) \right)$$

$$\text{Reaction at Point } b = R_b = \frac{M_a}{L - 1.5 * b_c - CL}$$

Where:

- M_a = moment at point A(CL of compression chord)
- M = moment at base of the wall
- P = Axial load at the base of the wall
- L = length of the wall
- b_c = width of chord member
- R_b = reaction at point B = hold down force
- CL = Hold down eccentricity distance (per selected hold down)

Lateral Force Distribution

- The lateral force distribution between piers is based on the relative stiffness of the sheathing, not on the length of the shear wall. For example, if you have an 8 foot wall and a 4 foot wall, the 8 foot wall will take more than $8/(8+4)*100\%$ of the force. The moment of inertia in the 8 foot wall will allow for a larger proportion of load to go into that pier.

Shear Panel Material Capacities

Shear capacity of the wall will be dependent on the Sheathing Material and Design Method that was chosen in the Design Rules spreadsheet - CFS Wall (Fasteners) tab. When design values are pulled from design tables, the program will use the appropriate capacity from the table based on the height-to-width ratio of the wall, member thickness of the stud members and type of loading used.

Tabulated shear capacities are based on the *AISI S240-15 CFS Structural Framing* and *AISI S400-15 w/S1-16 Seismic Design of CFS Structural Systems* codes. If the governing lateral load is a seismic load (under the load category EL, ELX, ELY, ELZ, etc) then the capacity will be pulled from the AISI S400-15 code.



Note: Curtain wall considerations per AISI S240-15 are not considered in the program.

Wood Sheathing

Tabulated design values are referenced from AISI S240-15 Table B5.2.2.3-2 and AISI S400-15 Table E1.3-1.

Steel Sheathing

Tabulated design values are referenced from AISI S240-15 Table B5.2.2.3-1 and AISI S400-15 Table E2.3-1.

If the Effective Width Method is used as the design method, this will be referenced from AISI S400-15 Chapter E2.3.



Note: The program will not perform the end distance check in the effective width method. This is assumed to be satisfied.

Fiberboard

Tabulated design values are referenced from AISI S240-15 Table B5.2.2.3-4. AISI S400-15 does not apply for this material.

Gypsum

Tabulated design values are referenced from AISI S240-15 Table B5.2.2.3-3. AISI S400-15 does not apply for this material.

Stiffness Assumptions

Vertical Direction

RISA uses an orthotropic plate element to de-couple the vertical and shear stiffness of the CFS walls. The vertical stiffness will be based on the E value of the studs and chords as specified in the Materials spreadsheet and the thickness of the wall. The thickness is taken as

$$\textit{Thickness} = \frac{\textit{Total Wall Weight}}{\textit{Material Density} * \textit{Wall Length} * \textit{Wall Height}}$$

Where,

$$\textit{Total Wall Weight} = W_{\textit{studs}} + W_{\textit{chords}} + W_{\textit{top plate}} + W_{\textit{sill plate}} + W_{\textit{sheathing panel}}$$

In-Plane Shear

The in-plane shear stiffness will be based on the G_a value designated within the specified nailing schedule divided by the sheathing thickness. Both the G_a value and the sheathing thickness are defined in the specified sheathing schedule.

Out-of-Plane Shear

For the out of plane shear, RISA uses the same Young's Modulus (E) as used in the vertical direction and a thickness that is calculated from the out of plane moment of inertia. This becomes:

$$Thickness = \left(\frac{I_{chord} + I_{stud} + I_{panel}}{Wall\ Length} \right)^{1/3}$$

CFS Wall Self Weight

The program will calculate CFS wall self weight as a sum of all the weights of the components. The material density is used to calculate the self weight of the studs, chords, top plates, sill plate, and sheathing. These are all then summed together to give the self weight of the entire wall.

**Note:**

- For this calculation, stud height equals full wall height.
- The number of studs is calculated using the stud spacing specified in Design Rules.

CFS Wall Optimization

The program will optimize CFS walls based on the required demand forces. The program can optimize sheathing for in-plane shear design.

This optimization can substantially modify the stiffness of the wall. To properly adjust the stiffness requires an iterative solution that updates the stiffness of the model. This includes updating the strength properties of the wall as well as the stiffness. The program will handle this optimization/iteration automatically.

To update the stiffness portion of the wall, the program must re-solve your model with these updated stiffnesses as this will change the distribution of forces through the model. Thus, the program will start with it's initial stiffness parameters and solve the model. It will then optimize the wall to meet strength criteria. Another solution will then be run with the new stiffnesses and the program will again optimize the wall to meet strength criteria. This procedure will continue to occur until all wall panel results match those of the previous solution.

The program will always present results in the output that coincide with the stiffness used in the final solution.

Panel Optimization

Cold Formed Wall Panel Schedule

Select Code: CFS Steel Sheeting

Available Panel Groups:

- AISI 2020 Steel
- AISI 2015 Steel

Current Selection Type: Use All Panels

	Label	Type Of Wall	Sheathing Material	Up 4 to 1?	Sheet Thickness	Fastener Spacing	Stud Block
1	0.018_#8@6/12_33	Type I - Segmented	Steel Sheet sheathing	No	0.018	6	No
2	0.027_#8@4/12_43	Type I - Segmented	Steel Sheet sheathing	No	0.027	4	No
3	0.027_#8@3/12_43	Type I - Segmented	Steel Sheet sheathing	No	0.027	3	No
4	0.027_#8@2/12_43	Type I - Segmented	Steel Sheet sheathing	No	0.027	2	No
5	0.027_#8@4/12_43_TALL	Type I - Segmented	Steel Sheet sheathing	Yes	0.027	4	No
6	0.027_#8@3/12_43_TALL	Type I - Segmented	Steel Sheet sheathing	Yes	0.027	3	No
7	0.027_#8@2/12_43_TALL	Type I - Segmented	Steel Sheet sheathing	Yes	0.027	2	No
8	0.027_#8@6/12_33	Type I - Segmented	Steel Sheet sheathing	No	0.027	6	No
9	0.027_#8@4/12_33	Type I - Segmented	Steel Sheet sheathing	No	0.027	4	No
10	0.027_#8@3/12_33	Type I - Segmented	Steel Sheet sheathing	No	0.027	3	No
11	0.027_#8@2/12_33	Type I - Segmented	Steel Sheet sheathing	No	0.027	2	No
12	0.027_#8@6/12_33_TALL	Type I - Segmented	Steel Sheet sheathing	Yes	0.027	6	No
13	0.027_#8@4/12_33_TALL	Type I - Segmented	Steel Sheet sheathing	Yes	0.027	4	No
14	0.027_#8@3/12_33_TALL	Type I - Segmented	Steel Sheet sheathing	Yes	0.027	3	No
15	0.027_#8@2/12_33_TALL	Type I - Segmented	Steel Sheet sheathing	Yes	0.027	2	No

OK Cancel

The procedure that RISA uses for design optimization is fundamentally based upon the assumption that there is a 'cost' to shear capacity, and therefore the ideal panel design would have as little shear capacity as possible to meet code requirements. Once the program has determined the shear demand on the wall it will choose the most economical panel configuration based on that which has a **Shear Capacity** closest to, but not exceeding the shear demand.

The program will also only pick shear panels that qualify based on the region height-to-width ratio and minimum stud thickness requirements.

The default shear panels in the program come straight from published tables in the AISI S240-15 and AISI S400-15.



Note:

- The shear forces listed in the XML spreadsheet are nominal shear strength values. The force capacity will be adjusted by the appropriate factor depending on if the ASD or LRFD design method is selected in the Cold Form Steel code dropdown.
- Panels with a label containing the characters "_W" together will be ignored during design optimization.

Cold Formed Steel Walls - Results

CFS Wall results are presented in the **Wall Panel Design** Spreadsheet and the detail reports.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **CFS Walls**.

CFS Wall Results Spreadsheets

CFS Wall Axial Results

CFS Wall Panel Axial Code Checks (AISI S100-20: ASD)											
Concrete In		Concrete Out		Concrete Seismic		Masonry In		Masonry Out		Masonry Lintel	
Masonry Seismic		Wood Wall Axial		Wood Wall In-Plane		Wood Header		CFS Wall Axial		CFS Wall In-Plane	
	Wall Panel	Region	Stud Size	Stud Spacing[in]	Axial Check	Gov LC	Chord Size	Chord Axial Check	Gov LC		
1	WP1	R1	600S137-33	12	0	N/A	600S137-33	NC	NC		
2		R4	600S137-33	12	0	N/A	600S137-33	NC	NC		
3	WP2	R1	600S137-33	12	0	N/A	600S137-33	NC	NC		
4		R4	600S137-33	12	0	N/A	600S137-33	NC	NC		
5		R7	600S137-33	12	0	N/A	600S137-33	NC	NC		
6	WP3	R1	600S137-33	12	0.013	1	600S137-33	0.169	1		

The following table provides descriptions of the columns in the spreadsheet.

CFS Walls Axial Results Spreadsheet

Column Header	Description
Wall Panel	The Wall Panel column lists the CFS wall panels that you have defined.
Region	The Region column lists the wall panel region that the results are based on.
Stud Size	The Stud Size column shows the size of the stud chosen for your wall based on the Design Rules you have defined.
Stud Spacing	The Stud spacing column shows the spacing between stud chosen for your wall, based on the Design Rules you have defined.
Axial Check	The Axial Check column displays the code check ratio between the member load and the member capacity.
Gov LC	The Gov LC column adjacent to the Axial Check column, shows the governing load combination for the design.
Chord Size	The Chord Size column shows the optimum chord size chosen for your wall, based on the Design Rules you have defined. Note that the chords are the vertical hold-down members/posts at the both ends of the wall.
Chord Axial Check	The Cord Axial Check column displays the code check ratio between the member load and the member capacity.
Gov LC	The Gov LC column adjacent to the Chord Axial Check column, shows the governing load combination for the design.

**Note:**

- When running the [Segmented](#) design method, the wall panel regions above and below the opening are not considered in design. Thus an NC (no calculation) is displayed.
- If there are some constraints that will not allow a wall to be designed, an NC (no calculation) will be displayed. Check the “Warning Log” within the program for more information on this.

CFS Wall In-Plane Results

Concrete In		Concrete Out		Concrete Seismic		Masonry In		Masonry Out		Masonry Lintel	
Masonry Seismic		Wood Wall Axial		Wood Wall In-Plane		Wood Header		CFS Wall Axial		CFS Wall In-Plane	
	Wall Panel	Shear Panel Label	Region	Shear Check	Shear Force[k/ft]	Gov LC	Hold-down Force[k]	Gov LC			
1	WP1	0.018_#8@6/12_...	R1	0	0	N/A	NC	NC			
2		0.018_#8@6/12_...	R4	NC	0	N/A	NC	NC			
3	WP2	0.018_#8@6/12_...	R1	0	0	N/A	NC	NC			
4		0.018_#8@6/12_...	R4	NC	0	N/A	NC	NC			
5		0.018_#8@6/12_...	R7	NC	0	N/A	NC	NC			
6	WP3	0.018_#8@6/12_...	R1	0.091	0.022	1	NC	NC			

The following table provides descriptions of the columns in the spreadsheet.

CFS Walls In-Plane Results Spreadsheet

Column Header	Description
Wall Panel	The Shear Panel Label shows the optimum shear panel arrangement chosen for your wall, based on the Design Rules you have defined.
Shear Panel Label	The Shear Panel Label column displays the name of the panel in the model.
Region	The Region gives the region for which the design values are being displayed.
Shear Check	The Shear Check value is a code check ratio between the panel shear load and the panel shear capacity.
Shear Force	The Shear Force column displays the value that governed the design.
Gov LC	The Gov LC column adjacent to the Shear Force column, displays the load combination that governed the design.
Hold-down Force	The Hold-Down Force show the maximum value of hold-down/strap that produced the highest strap force. This maximum force could occur on either end of the wall/region.

Column Header	Description
Gov LC	The Gov LC column adjacent to the Hold-down Force column, shows the load combination that produced the highest strap force.

**Note:**

- When running the [Segmented](#) design method, the wall panel regions above and below the opening are not considered in design. Thus an NC (no calculation) will be displayed. If running a combined RISAFloor and RISA-3D model use RISAFloor to get header results.
- If there are some constraints that will not allow a wall to be designed, an NC (no calculation) will be displayed. Check the “Warning Log” within the program for more information on this.
- If “default” is shown in the ‘Hold-Down Label’ column it means that a hold down is not required for this wall/region.

CFS Wall Self Weight

The program will calculate the self weight of a CFS wall based on the weights of the individual components. Using the material density, the self weight is calculated for the studs, chords, top plates, sill plates, and sheathing. These are all then summed for the total self weight of the wall.

CFS Wall Detail Reports

The detail report gives detailed information about the wall design. The detail reports are specifically molded to the type of design specified.

**Note:**

- Only the **Segmented** design method is considered for CFS walls at this time.
- Many of the values for design checks seen below are not performed in RISAFloor as it is strictly a gravity design program.

Accessing the Detail Reports and the Specific Windows

Once you have a solved model, the detail reports become available. They are accessible in two ways:

- If you have the **Wall Panel Design** spreadsheet open, you can right-click on a line in the spreadsheet and select **Detailed Report**.
- If you are in a graphic view of your model, there is a  button that will open up the detail report window.



Note: Detail report information is not available for an envelope solution.

Once the detail report window is open, you will see a dialog area at the top.

Detail Report for - WP1
— □ ×

<
Wall Panel Label WP1
>
Wall ▾

Option	Name	Description
<div style="display: flex; align-items: center; gap: 10px;"> < Wall Panel Label WP1 > </div>	Prior/Next	The left and right arrows let you cycle between the different wall panels in your model.
<div style="border: 1px solid #ccc; padding: 5px; display: inline-block;">Print</div>	Print	Lets you print the Detail Report (found at the bottom of the Detail Report window).

The Wall detail report gives an overall summary of your wall, complete with governing code checks and opening information.

Segmented Method

The **Segmented** design method uses each of the three detail report sections to give design information.

Wall Window

This window gives an overview of the wall, giving controlling region information and deflection information. Note that this window only gives information on the full-height segments in your wall, as this is the basis of the **Segmented** method.

Input Echo

Detail Report for - WP1
— □ ×

< Wall Panel Label WP1
> Wall ▾

Detail Report: WP1

CFS Wall

GENERAL		GEOMETRY		MATERIALS	
Code:	AISI 20 Code	Total Height (ft):	10	Top PI	A653 SS Gr50/1 600T125-54
Design Method:	Segmented	Total Length (ft):	15	Sill	A653 SS Gr50/1 600T125-54
Wall Material:	A653 SS Gr50/1	Max H/W Ratio:	0.67	Wall Stud	A653 SS Gr50/1 600S137-54
Panel Schedule:	AISI 2020 Steel			Chord	A653 SS Gr50/1 600S137-54
Sheathing Material:	Steel Sheet sheathing				
Sel. Shear Panel:	0.033_#8@2/12_33_EQ				
Stud Design Method:	All Steel				
Shear Design Method:	Table				

ENVELOPED RESULTS

Controlling Shear Region	Shear Panel	Shear UC	Shear LC	Chord UC	Chord LC	Stud UC	Stud LC
R1	0.033_#8@2/12_33_EQ	0.958	6 (S)	4.367	3 (W)	N/C	N/C

REGION INFORMATION

Region Label	H/W Ratio	Shear UC	Shear LC	Hold-Down	Hold-Down UC	Hold-Down LC	Chord UC	Chord LC	Stud UC	Stud LC
R1	0.67	0.958	6 (S)	NC	NC	NC	4.367	3 (W)	N/C	N/C

DEFLECTION RESULTS

Maximum Region Deflection (in)	Deflection LC	Finite Element Deflection (in)	Shear Stiffness Adjustment Factor
0.136 (R1)	6 (S)	0.008	1

Print
 Add to Full Report
Close

This lists information about the wall, similar to the [Region](#) report and also gives an image of the wall. The image shows the location of hold-downs/straps, regions and headers.

Design Summary

Detail Report for - WP1

Wall Panel Label WP1 Wall

ENVELOPED RESULTS

Controlling Shear Region	Shear Panel	Shear UC	Shear LC	Chord UC	Chord LC	Stud UC	Stud LC
R1	0.033_#8@2/12_33_EQ	0.958	6 (S)	4.336	3 (W)	0.695	11

REGION INFORMATION

Region Label	H/W Ratio	Shear UC	Shear LC	Hold-Down	Hold-Down UC	Hold-Down LC	Chord UC	Chord LC	Stud UC	Stud LC
R1	0.67	0.958	6 (S)	NC	NC	NC	4.336	3 (W)	0.695	11

DEFLECTION RESULTS

Maximum Region Deflection (in)	Deflection LC	Finite Element Deflection (in)	Shear Stiffness Adjustment Factor
0.136 (R1)	6 (S)	0.008	1

Print Add to Full Report Close

Design Summary Sections

Section	Description
Envelope Results	The Enveloped Results gives the code checks for all of the controlling elements in the wall and their associated load combinations.
Region Information	The Region Information gives the tabulated results of all of the full-height regions in your wall.
Opening Information	The Opening Information simply states that header design cannot be completed with the Segmented method. The regions above and below the opening have their shear stiffnesses set to be zero and this causes the header forces to be invalid.
Deflection Results	The Deflection Results gives both the calculated AISI code deflection (Maximum Region Deflection) and the FE deflection for use as a means of comparison. Because the AISI equations are empirical and take into account many non-elastic considerations such as nail slip, these two values may not be the same.

Region Window

This window gives information for your wall on a region by region basis. Note that only full-height regions of the wall panel will have a region detail report. The **Segmented** method only considers these full height segments in the design of the wall.

The region detail report is split into four portions: [input echo](#), [diagrams](#), [design details](#), and [cross section detailing](#). Note that in RISAFloor the detail reports are less detailed because RISAFloor does not consider lateral forces which RISA-3D does.

Input Echo

Below is the input echo portion of the detail report.

Criteria table

Criteria	Description
Code	Gives the code used to design your wall panel.
Design Method	Gives the design method that was used.
Wall Material	Specifies the CFS type assigned to the entire wall.
Panel Schedule	Specifies the sheathing material used to optimize panel selection (set in Design Rules).

Materials table

Material	Description
Wall Studs	Specifies the CFS material type assigned to the wall studs.
Stud Size	Specifies the member size used for the wall studs.
Chord Material	Specifies the CFS material type assigned to the chords (vertical members at both ends of the wall).
Chord Size	Specifies the member size used for the chords (vertical members at both ends of the wall).
Top Plate & Sill	Specifies the CFS material type assigned to the top and sill plates.
Top Plate Size	Specifies the member size used for the top plate.
Sill Plate Size	Specifies the member size used for the sill plate.

Geometry table

Geometry	Description
Total Height	This is the height of the wall panel region.
Total Length	This is the length of the wall panel region.
Region H/W Ratio	This is the ratio of wall height to length, using the minimum wall height.
Cap. Adj (2w/h)	This is an aspect ratio reduction factor for the shear panel strength per AISI S240-15 B5.2.2.1. This factor applies only for wall panels that have an aspect ratio greater than 2 but less than or equal to 4. This factor is applied separately for each full-height region in your wall.

Geometry	Description
Stud Spacing	This is the optimized stud spacing based on your Design Rules.
K	This is the effective length, K Factor used for stud and chord compression design.
HD Eccentricity	This is the eccentricity of the hold-down connection. This is based on user input (set within the Design Rules).

Envelope Diagrams

These diagrams show the axial forces, in-plane shear, and in-plane moments of the wall, as well as the maximum and minimum forces and their locations.

Detail Report for - R1 of WP1
— □ ×

< Wall Panel Label WP1
> Region
R1
In-Plane

Detail Report: WP1 (Region R1)

CRITERIA	GEOMETRY	MATERIALS
Code: AISI 20 Code	Total Height (ft): 10	Top PI: A653 SS Gr50/1 600T125-54
Design Method: Segmented	Total Length (ft): 15	Sill: A653 SS Gr50/1 600T125-54
Wall Material: A653 SS Gr50/1	Region H/W: 0.67	Wall Stud: A653 SS Gr50/1 600S137-54
Panel Schedule: AISI 2020 Steel	Aspect Ratio: 1.00	Chord: A653 SS Gr50/1 600S137-54
Sheathing Material: Steel Sheet sheathing	HD Eccentricity (in): 0	

ENVELOPE DIAGRAMS

k
P

Min: 10.5 at 10 ft
Max: 10.783 at 0 ft

k
V

Max: 7.501 at 5 ft

k-ft
M

Min: -75.002 at 0 ft
Max: -0.004 at 10 ft

Print
 Add to Full Report
Close



Note:

The diagrams are not actually enveloped.

- The **Axial** diagram shown is the diagram from the governing load combination for *Stud* design.
- The **Shear and Moment** diagram is the diagram from the governing load combination for *Shear Panel* design.

Design Details

This portion gives you the capacity and strength values at the section in the wall where the combined check is maximum, as well as the governing load combination. Much of this information is also reported in the **CFS Wall Panel Design** spreadsheets.

Detail Report for - R1 of WP1
— □ ×

< Wall Panel Label WP1 >
Region ▾ R1 ▾
In-Plane ▾

DESIGN SUMMARY

STUDS			
Required Cap (k):	0.719	Governing LC:	11
Provided Cap (k):	1.034	Stud Spacing (in):	12
Ratio:	0.695	Studs in Region:	16
		K:	1.00

CHORDS			
Max Comp Force (k):	5.578	Gov Comp LC:	3
Comp Capacity (k):	1.286	Max Tens Force (k):	1.534
Comp Ratio:	4.336	Tens Capacity (k):	15.389
		Tens Ratio:	0.1
		Gov Tens LC:	5

SHEAR PANEL			
Selected Shear Panel:	0.033_#8@2/12_33_EQ	Ratio:	0.958
Required Cap (k/ft):	0.5	Governing LC:	6 (Seismic)
Original Cap (k/ft):	1.305	Panel Thick (in):	0.033
Adjusted Cap (k/ft):	0.522	Fastener Spacing (in):	2
		Number of Sides:	One
		Screw Size:	#8

DEFLECTIONS			
Component 1 (in):	0.018	Component 3 (in):	0
Component 2 (in):	0.019	Component 4 (in):	0.1
		Tot Deflection (in):	0.136
		Governing LC:	6

Print
 Add to Full Report
Close

Design Details Sections

Section	Description
Studs	<p>The provided capacities of the Studs members are calculated using the standard provisions for tension/compression members. These members are assumed to be fully braced in the weak axis, and unbraced in the strong axis.</p>
Chords	<p>The provided capacities of the Chords members are calculated using the standard provisions for tension/compression members. These members are assumed to be fully braced in the weak axis, and unbraced in the strong axis.</p> <p>For more information on the chord force calculations, see the CFS Wall - Design topic.</p> <div data-bbox="492 758 1409 905" style="border: 1px solid black; padding: 5px; background-color: #e6f2ff;">  Note: For chord results, the tension/compression capacity is computed using the reduced cross-sectional properties caused by the hold-down bolt hole. </div>
Shear Panel	<p>The Selected Shear Panel section shows the call-out from the shear panel database. The information below it, is the information describing the call-out.</p> <p>The provided capacity of the shear panel is taken from the Shear Capacity column of the panel database. This is the adjusted capacity based on tables within the AISI S240-15 and S400-16. The Governing LC explicitly states if the controlling load combination was based on Wind or Seismic. The program does a unity check for all LCs that are being solved, finds the maximum value and reports that information.</p> <p>This section of the report echoes the database information for the selected shear panel. For more information on these properties refer to Appendix G-CFS Shear Wall Files.</p>

Section	Description
	 <p>Note:</p> <ul style="list-style-type: none"> The displayed "Shear Capacity" takes the given nominal shear capacity from the design code and multiplies or divides it by the appropriate code factor and aspect ratio factor when applicable. A shear panel design will be chosen for the worst-case region in a segmented wall. That panel will then be used for all regions in that wall. The worst-case region is the one that has the highest Shear UC value. Because wind and seismic loading allows for different design capacities, the highest shear in the wall may not be the governing shear (if that highest shear was due to wind).
Deflections	<p>The Deflections section shows the Detail Report based on an approximation from the design code.</p> <p>See the following Deflection section for more information.</p>

Deflection

DEFLECTIONS			
Component 1 (in):	0.018	Component 3 (in):	0
Component 2 (in):	0.019	Component 4 (in):	0.1
		Tot Deflection (in):	0.136
		Governing LC:	6

The deflection listed in the detail report is based on an approximation from the design code:

AISI Design Deflection

Per the AISI S240-15 North American Standard for Cold-Formed Steel Structural Framing Equation B5.2.5-1:

$$\delta = \frac{2 \cdot v \cdot h^3}{3 E_S \cdot A_C \cdot b} + \omega_1 \cdot \omega_2 \cdot \frac{v \cdot h}{\rho \cdot G \cdot t_{sheathing}} + \omega_1^{\frac{5}{4}} \cdot \omega_2 \cdot \omega_3 \cdot \omega_4 \cdot \left(\frac{v}{\beta}\right)^2 + \frac{h}{b} \cdot \delta_v$$

Refer to the AISI 240-15 standard page 52 for detail definitions for each term in the equation.

The first component of the above equation determines the Bending Component of the deflection.

The second component of the above equation determines the Shear Component of the deflection.

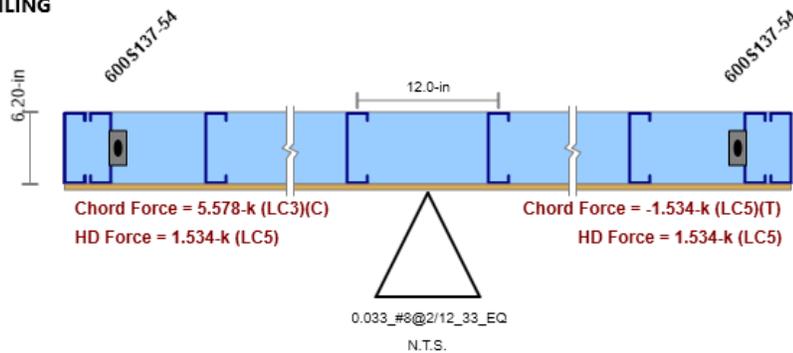
The third component of the above equation determines the Hold-Down Elongation, which causes additional deflection. The δ_v term refers to vertical deformation of the hold-down. This is a user input within the Wall Design Rules.



Note: This is the theoretical deflection of the wall. This may differ from the deflection of the wall as performed by finite element analysis within RISA. Therefore, this deflection value may not coincide with the reported deflection value in the deflections spreadsheets.

Cross Section Detailing

CROSS SECTION DETAILING



The last section of the detail report consists of the wall detailing information. This information is provided as a visual confirmation of the wall design. The wall thickness, and stud spacing are shown as dimensions. The triangle shows sheathing on one side of the wall, with the abbreviated panel designation. The chord sizes/forces are shown at either end.



Note:

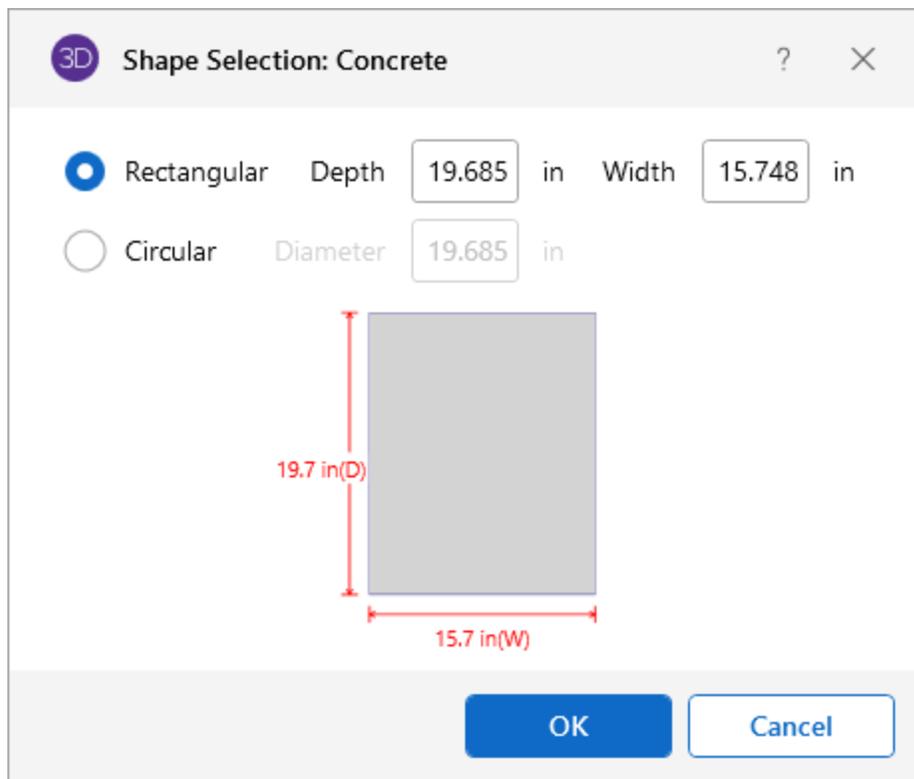
- The graphic will show hold-downs regardless if one is required or not.
- The chord members will always be shown as CS members face-to-face despite what is specified in the Design Rules. This is just a graphical representation of the chord member placement.

Concrete Members

Concrete Member - Database

There are two types of concrete shapes currently supported, Rectangular and Circular. If you're familiar with the shape definitions, you can type the name directly into the appropriate field.

Alternately you can click the  (ellipsis) button to have the program generate the desired shape definition for you.



Rectangular Sections

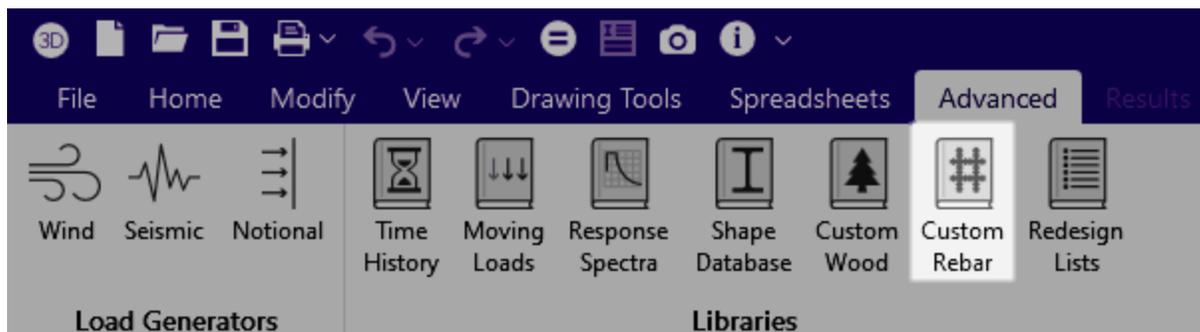
Rectangular sections are defined using a parametric shape code since a rectangular shape may be any depth or width. The code is `CRECT'depth'X'width'`, where 'depth' and 'width' are the values in the current dimension units. For example, if you wanted a beam that was 18" deep and 12" wide, you would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

Circular Sections

Circular/Round sections are also defined using a parametric shape code since a round shape may have any diameter. The code is `CRND'diameter'`, where 'diameter' is the value in the current

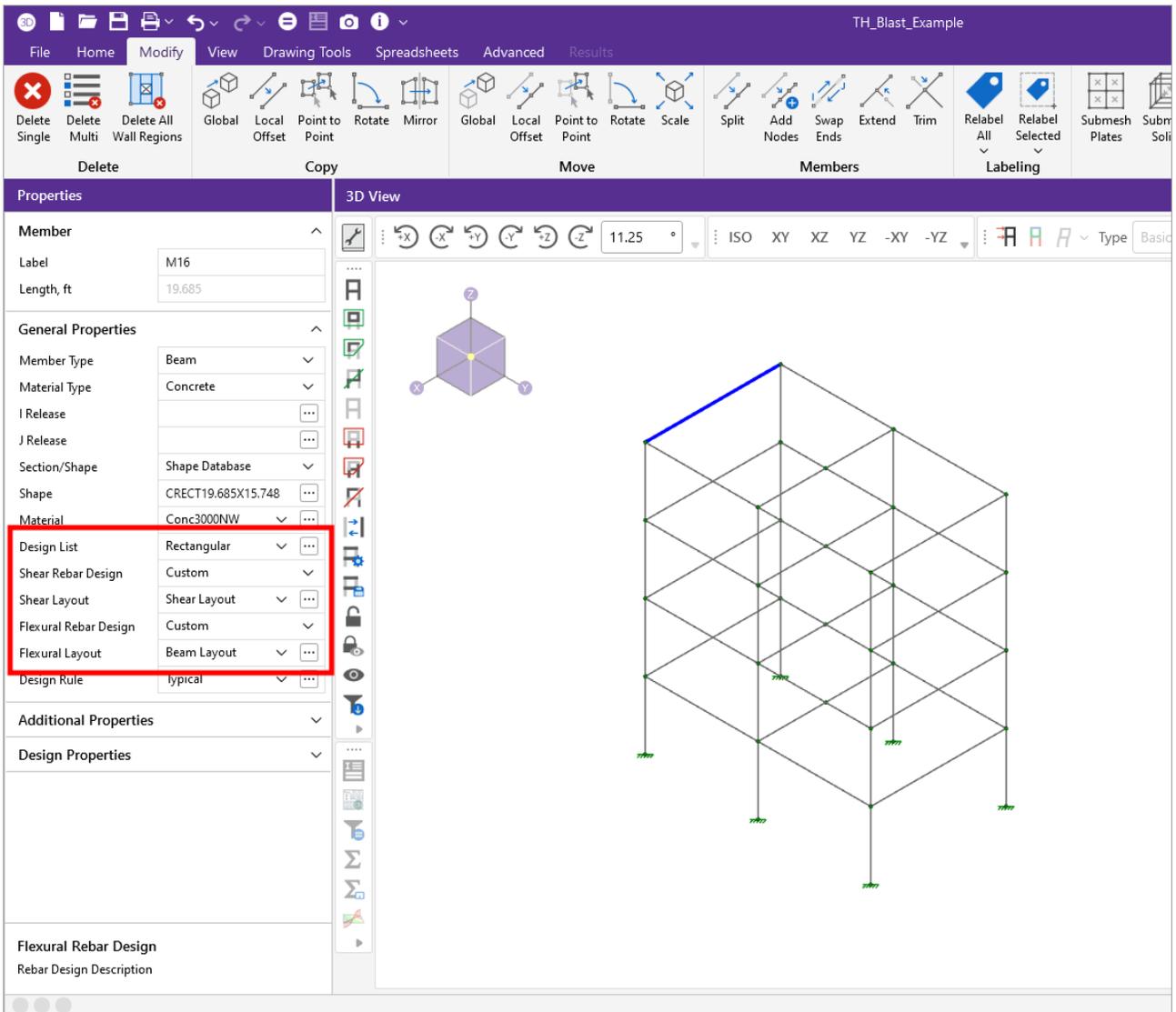
dimension units. For example, if you wanted a column that was 14" in diameter, you would enter "CRND14". Note that the dimension can also be a decimal value like "14.5".

Custom Rebar Layout Database



On the **Advanced** tab of the ribbon, the **Custom Rebar**  button opens the database that is used for creating and storing custom rebar layouts. This allows you to create multiple layers of bars and add in compression reinforcement or unusual bar arrangements.

These reinforcement layouts can be assigned to beam or column members in the same way as the other concrete design parameters are assigned. This can be done from the **Concrete Beam** (or **Column**) tabs of the **Members Spreadsheet**, or the **Rebar Design** dropdown in the **Properties Panel**.



Concrete Beam Design Parameters												
Primary	Advanced	Hot Rolled	Cold Formed	Wood	Concrete Beam	Concrete Column	Aluminum	Stainless	RISAConnection			
	Label	Shape	Length [ft]	B-eff Left [in]	B-eff Right [in]	Slab Thk Left [in]	Slab Thk Right [in]	Icr Factor	Flexural Rebar Design	Flexural Layout	Shear Rebar Design	Shear Layout
1	M1	girder	19.685						Design Rule	Use Design Rule	Design Rule	Use Design Rule
2	M2	girder	19.685						Design Rule	Use Design Rule	Design Rule	Use Design Rule
3	M3	girder	19.685						Design Rule	Use Design Rule	Design Rule	Use Design Rule
4	M4	girder	19.685						Design Rule	Use Design Rule	Design Rule	Use Design Rule
5	M7	girder	19.685						Design Rule	Use Design Rule	Design Rule	Use Design Rule
6	M8	girder	19.685						Design Rule	Use Design Rule	Design Rule	Use Design Rule
7	M9	girder	19.685						Design Rule	Use Design Rule	Design Rule	Use Design Rule
8	M10	girder	19.685						Design Rule	Use Design Rule	Design Rule	Use Design Rule
9	M13	girder	19.685						Design Rule	Use Design Rule	Design Rule	Use Design Rule
10	M14	girder	19.685						Design Rule	Use Design Rule	Design Rule	Use Design Rule
11	M15	girder	19.685						Design Rule	Use Design Rule	Design Rule	Use Design Rule

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Custom Rebar Layouts**.

Beam Rebar Layouts

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

Custom Rebar Layout

Beam | Column | Shear

Label: Beam Layout

Rebar Set: ASTM A615 Fy: 60 ksi

Buttons: New, Delete, Copy

	Placement	y [in]	No of Bars	Size	Start [(ft, %)]	End [(ft, %)]
1	From B...	1	1	#5	0	%100

Diagram: 3D view of a beam cross-section showing rebar placement at the top and bottom.

Buttons: OK, Cancel

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

Rectangular Column Rebar Layouts

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

Normally, column bars are arranged in **layers**. One 'Top' and one 'Bottom' horizontal layer must **always** be defined, each containing at least two bars. These layers, as well as any additional

horizontal layers, are specified by entering a **y1** value to specify the depth from the top or bottom fiber to the centerline of the reinforcing steel. The number and size of the bars must then be entered. The **z1** and **z2** values dictate where the first and last bar in that layer are located. Additional bars in that layer are placed so that they are evenly spaced in that layer.

3D Custom Rebar Layout
? X

Beam
Column
Shear

Label Column Layout Shape Rectangular

Rebar Set ASTM A615 Fy 60 ksi

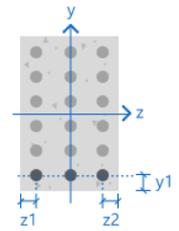
New

Delete

Copy

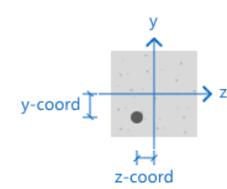
Custom Bar Groups

	Placement	y1 [in]	z1 [in]	z2 [in]	Size	No of Bars	Start [(ft, %)]	End [(ft, %)]
1	From B... v	1	1	1	#7	1	0	%100



Custom Single Bars

	Size	z-coord [in]	y-coord [in]	Start [(ft, %)]	End [(ft, %)]
No data available for this spreadsheet. Press a key or click in this area to start...					



OK
Cancel

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

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

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

Circular Column Rebar Layouts

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

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

3D Custom Rebar Layout
? X

Beam
Column
Shear

Label Column Layout Shape Circular

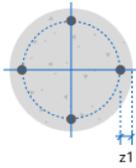
Rebar Set ASTM A615 Fy 60 ksi

New Delete

Copy

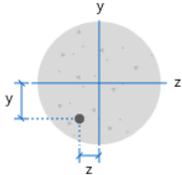
Custom Bar Groups

	z1 [in]	Size	No of Bars	Start [(ft, %)]	End [(ft, %)]
1	1	#3	1	0	%100
2	1	#3	1	%30	%70



Custom Single Bars

	Size	z-coord [in]	y-coord [in]	Start [(ft, %)]	End [(ft, %)]
No data available for this spreadsheet. Press a key or click in this area to start...					



OK Cancel

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

Shear Rebar Layouts

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

3D Custom Rebar Layout
? X

Beam
Column
Shear

Label Shear Layout ▾

Rebar Set ASTM A615 ▾

Legs/Stirrup 2 ▾

Fy 60 ksi

New

Copy

Delete

Custom Bar Groups

	Size	Spacing [in]	Start [(ft, %)]	End [(ft, %)]
1	#8	2	0	%100



OK Cancel

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

Concrete Member - Design

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

- The 2022, 2019, 2014, 2011, 2008, 2005, 2002 and 1999 Editions of ACI 318
- The 1997 Edition of the British code (BS 8110)
- The 1992 EuroCode (EC2) and the British publication of the 2014 and 2004 Eurocode (BSEN)
- The 1994 and 2004 Editions of the Canadian code (CSA A23.3)
- The 2000 Edition of the Indian code (IS 456)
- The 2001 Edition of the Australian code (AS 3600)
- The 1995 Edition of the New Zealand code (NZS 3101)

- The 2004 Edition of the Mexican code (NTC-DF)
- The 2007 Edition of the Saudi Building Code (SBC 304)

**Note:**

- Unless otherwise specified, all code references below are to ACI 318-14.
- Beams and Columns designed in RISA meet all of the requirements for Ordinary Moment Frames except for the additional requirements indicated in ACI 318-14 Section 18.3. Those provisions should be checked by hand outside of RISA.
- ACI 318-19 (22) re-approves previous ACI 318-19 without any technical changes, therefore all references in the help documentation and in the program continue to use ACI 318-19.

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

Apply a Concrete Design Code

To Apply a Concrete Design Code

1. On the **Codes** tab of 'Model Settings' window.
2. Choose the **Concrete** code from the drop down list.
3. Click **Apply** or **OK**.

Concrete Spans

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

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



Note:

- The quickest way to create new joints at beam / column intersections is to run a Model Merge.
- Clear spans must be greater than 1 ft for the program to design the member. This considers half the depth of the columns or beams framing into the ends of the member.
- The program's ability to recognize spans is important because it gives you more relevant span to span information without overwhelming you with independent design results for each finite element segment that comprises your physical member.
- For continuous beam members, the program evaluates the framing to determine which beams elements are supporting other beam elements so that only supporting members are treated as supports and not vice versa.
- Currently, members of type HBrace, VBrace and None do not affect the span distances. Nor do any arbitrary joints within each span along a member.

Concrete Design Parameters - Columns

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

Concrete Column Design Parameters																
Primary	Advanced	Hot Rolled	Cold Formed	Wood	Concrete Beam	Concrete Column	Aluminum	Stainless	RISACorrection							
	Label	Shape	Length [ft]	Lu-yy [ft]	Lu-zz [ft]	K y-y	K z-z	Cm y-y	Cm z-z	y sway	z sway	Icr Factor	Rebar Design	Flexural Layout	Rebar Design	Shear Layout
1	M1	Exterior Column	12							<input type="checkbox"/>	<input type="checkbox"/>		Design Rule	Use Design Rule	Design Rule	Use Design Rule
2	M2	Exterior Column	12							<input type="checkbox"/>	<input type="checkbox"/>		Design Rule	Use Design Rule	Design Rule	Use Design Rule
3	M3	Exterior Column	12							<input type="checkbox"/>	<input type="checkbox"/>		Design Rule	Use Design Rule	Design Rule	Use Design Rule
4	M195	Exterior Column	12							<input type="checkbox"/>	<input type="checkbox"/>		Design Rule	Use Design Rule	Design Rule	Use Design Rule
5	M383	Exterior Column	12							<input type="checkbox"/>	<input type="checkbox"/>		Design Rule	Use Design Rule	Design Rule	Use Design Rule
6	M384	Exterior Column	12							<input type="checkbox"/>	<input type="checkbox"/>		Design Rule	Use Design Rule	Design Rule	Use Design Rule
7	M385	Exterior Column	12							<input type="checkbox"/>	<input type="checkbox"/>		Design Rule	Use Design Rule	Design Rule	Use Design Rule

The following parameters can be defined for each concrete column.

Label

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

Shape

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

Length

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

Unbraced Length

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

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

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

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



Note:

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

For additional advice on this topic, please see the RISA Tips & Tricks website: www.risa.com/post/support. Type in Search keywords: **Unbraced Lengths**.

K Factors (Effective Length Factors)

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

If a value is entered for a **K Factor**, that value is used for the entire length of the physical member. If an entry is not made (left blank), the value internally defaults to '1' for that member. See ACI 318-14 Section R6.2.5 (ACI 318-11 Section R10.10.1) for an explanation of how to calculate K Factors.

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

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



Note: This is an approximation of K-values and is NOT based on the Jackson and Moreland Alignment Charts presented in ACI 318-14 Section R6.2.5.

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

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

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

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

Limitation:

RISA-3D currently neglects 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 neglects the beam member when

it calculates the K factor for the column and vice versa. The effect is, the ends of the members at that joint are seen as “pinned” and not “fixed” for the K-factor approximation.

Sway Flags

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

C_m – Equivalent Moment Correction Factor

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

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

Flexural and Shear Rebar Layout

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

Icr Factors (Cracked Moment of Inertia Factors)

The **Icr Factor** is used to reduce the bending stiffness of concrete columns per ACI 318-14 Table 6.6.3.1.1(a) (ACI 318-11 Section 10.10.4.1). If this entry is left blank, default values of 0.35 for beams and 0.70 for columns are used.



Note:

- The **Icr Factor** is ignored if the “**Use Cracked Sections**” box is not checked on the **Concrete** tab of the **Model Settings** dialog.
- The alternative calculations in ACI 318-14 Table 6.6.3.1.1(b) (ACI 318-11 Equations 10-8 and 10-9) are not considered.
- The sustained load reduction of ACI 318-14 Section 6.6.3.1.1 (ACI 318-11 Section 10.10.4.2) is not considered.

Service Level Stiffness - Columns

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

For typical applications, ACI 318-14 Section 6.6.3.1 (ACI 318-11 Section 10.10.4) requires that member stiffness be reduced to account for the cracking that occurs when a member is subjected to ultimate level loads. As described in the previous section, RISA uses the **Icr Factor** to account for this stiffness reduction. However, for service level analysis, the level of cracking is significantly less. Therefore, the stiffness used in your analysis should be representative of the reduced loading and reduced cracking. Per the ACI 318-14 Section R6.6.3.2.2 (ACI 318-11 Section R10.10.4.1), the program accounts for this increased stiffness by applying a factor of 1.43 to the cracked section properties for any load combination that has the “**Service Load**” flag checked on the **Design** tab of the **Load Combinations Spreadsheet**.

Concrete Design Parameters - Beams

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

Concrete Beam Design Parameters												
Primary	Advanced	Hot Rolled	Cold Formed	Wood	Concrete Beam	Concrete Column	Aluminum	Stainless	RISAConnection			
	Label	Shape	Length [ft]	B-eff Left [in]	B-eff Right [in]	Slab Thk Left [in]	Slab Thk Right [in]	Icr Factor	Flexural Rebar Design	Flexural Layout	Shear Rebar Design	Shear Layout
1	M4	CRECT12X8	18						Design Rule	Use Design Rule	Design Rule	Use Design Rule
2	M5	CRECT8X8	18						Design Rule	Use Design Rule	Design Rule	Use Design Rule
3	M6	CRECT8X8	18						Design Rule	Use Design Rule	Design Rule	Use Design Rule
4	M7	CRECT8X8	18						Design Rule	Use Design Rule	Design Rule	Use Design Rule
5	M8	CRECT8X8	18						Design Rule	Use Design Rule	Design Rule	Use Design Rule
6	M9	CRECT8X8	18						Design Rule	Use Design Rule	Design Rule	Use Design Rule
7	M10	CRECT8X8	18						Design Rule	Use Design Rule	Design Rule	Use Design Rule
8	M11	CRECT12X8	18						Design Rule	Use Design Rule	Design Rule	Use Design Rule
9	M12	Conc Joist	18	2.5	2.5	6	6		Design Rule	Use Design Rule	Design Rule	Use Design Rule
10	M13	Conc Joist	18	2.5	2.5	6	6		Design Rule	Use Design Rule	Design Rule	Use Design Rule
11	M14	Conc Joist	18	2.5	2.5	6	6		Design Rule	Use Design Rule	Design Rule	Use Design Rule

The following parameters can be defined for each concrete member.

Label

You can assign a unique **Label** to all of the members. Each label must be unique, so if you try to enter the same label more than once you receive an error message. You can **Relabel** options on the **Tools** menu.

Shape

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

Length

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

Flexural and Shear Rebar Layout

You can choose to manually create the reinforcement layout for the beam. This must be done if the user wishes to take advantage of compression steel, or multiple layers of reinforcement. See [Concrete Database - Rebar Layouts](#) for more information. If *Use Design Rule* is specified, then the program designs for one layer of reinforcing and may vary that reinforcing based on ACI minimums, maximums, and the moment and shear demand at each section along the span using the Member Design Rules as the parameters of the reinforcement selection. If you define your own rebar layout, and compression reinforcement is defined, then the program considers the compression reinforcement in the analysis.

Icr Factors (Cracked Moment of Inertia Factors)

The **Icr Factor** is used to reduce the bending stiffness of concrete beams.

For ACI and Canadian codes (ACI 318-14 Section 6.6.3.1 and A23.3-04 section 9.2.1.2), if this entry is left blank, the Icr factor defaults to a value of 0.35 for beams and 0.70 for columns.

For Australian and New Zealand codes (per section 6.6.2 of AS3600-2001), this defaults to a value of 0.4 for beams and 0.8 for columns.

For Indian and Saudi codes, this entry defaults to a value of 1.0 for beams and columns.



Note: The **Icr Factor** is ignored if the “**Use Cracked Stiffness**” box is not checked on the **Concrete** tab of the **Model Settings** dialog.

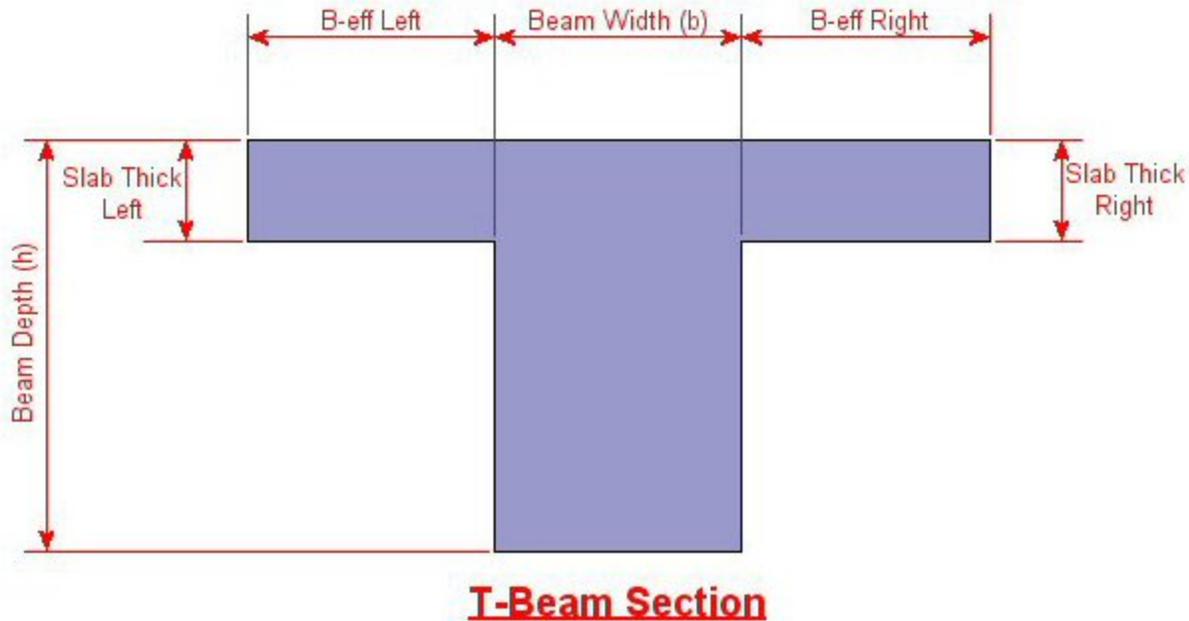
Service Level Stiffness - Beams

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

For typical applications, ACI 318-14 Section 6.6.3.1 (ACI 318-11 Section 10.10.4) requires that member stiffness be reduced to account for the cracking that occurs when a member is subjected to ultimate level loads. As described in the previous section, RISA uses the **Icr Factor** to account for this stiffness reduction. However, for service level analysis, the level of cracking is significantly less. Therefore, the stiffness used in your analysis should be representative of the reduced loading and reduced cracking. Per ACI 318-14 Section R6.6.3.1.1 (ACI 318-11 Section R10.10.4.1), the program accounts for this increased stiffness by applying a factor of 1.43 to the cracked section properties for any load combination that has the “**Service Load**” flag checked on the **Design** tab of the **Load Combinations Spreadsheet**.

T-beam & L-beam Sections

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



RISA-3D automatically trims the effective slab widths, **B-eff Left** and **B-eff Right**, to the maximum values indicated in ACI 318-14 Table 6.3.2.1 (ACI 318-11 Sections 8.12.2(a) and 8.12.3(a) & (b)) if the value entered by the user is greater than that allowed by the code. It should be noted that RISA-3D does not check ACI 318-14 Table 6.3.2.1 portions referring to adjacent framing (ACI 318-11 Sections 8.12.2(b) and 8.12.3(c)) because no adjacent framing checks are performed.

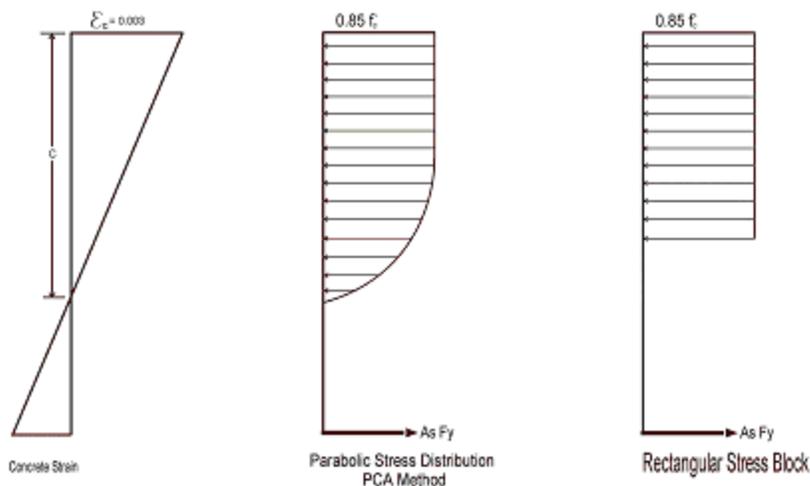
If the values of either **B-eff Left** or **B-eff Right** are left blank, a value of zero is assumed, indicating no additional slab width beyond 1/2 the beam width on that side.



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

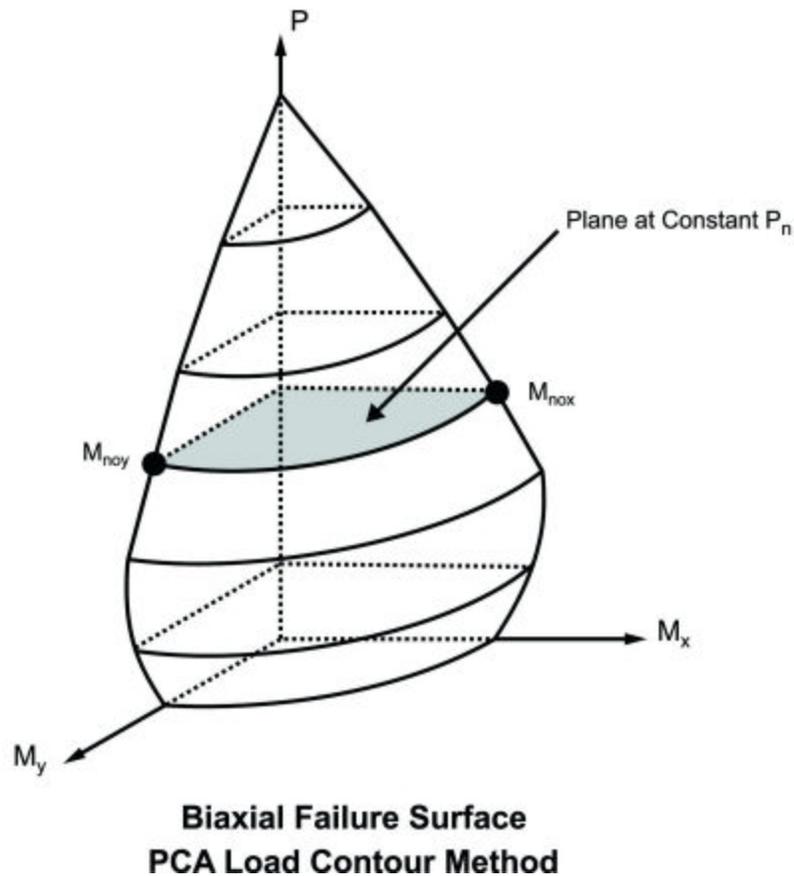
Parabolic vs. Rectangular Stress Blocks

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



Biaxial Bending of Columns

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



British Eurocode Design Parameters (BS EN 1992-1-1: 2004)

General

- f_{ck} – Can not be more than 50 MPa (7252 psi) for normal strength concrete
- α_{cc} is assumed to be 1 (recommended value) : See 3.1.6
- Effective length of T and L: $L_o = .7 \cdot \text{span length}$ and $b_{eff,i} = L_o / 10$
- $\Phi_{concrete} = 1.5$
- $\Phi_{rebar} = 1.15$
- Maximum bar spacing for beams = 300 mm

Tension Development Length

- $\alpha_{ct} = 1$ (assumed in Eq 3.16)
- $\eta_1 = \eta_2 = 1$ (assumed in Eq 8.2 to calculate bond stress)
- $\lambda_3, \lambda_5, \lambda_4 = 1$ (assumed in Eq 8.4)

- C_d in Table 8.2 assumed to be 1 bar diameter rebar spacing
- Development length when hooks are provided uses same assumptions as BS 8110-1: 1997

Shear Capacity of Concrete

To compute the shear capacity of concrete the following recommended values are being used:

- $C_{Rd,c} = 0.18/\gamma_c$ for Eq 6.2.a
- $v_{min} = 0.035 k^{3/2} f_{ck}^{1/2}$
- $v = .6 * [1 - f_{ck}/250]$
- V_{max} is calculated from Eq. 6.5.
- θ is assumed 45 degrees in Eq 6.8.

Slender Column Design

- Biaxial column design done using Eq. 5.39
- Design based on nominal curvature
- $\lambda_{lim} = 20 A B C/n^{1/2}$ (A=.7, B= 1.1, C=0.7 for unbraced)
- $K_\phi = 1$ in Eq 5.34; the effect of creep is neglected.

Limitations

Limitations - General

Torsion – Beams and columns ignore torsion with respect to the design of shear reinforcement. A message is shown in the detail report to remind you of this. You can turn the warning messages off on the **Concrete** tab of the **Model Settings Dialog**. However, when using the 2002 and newer ACI 318 code the program does check the torsion on the member against the Threshold Torsion value (see ACI 318-14 Section 22.7.4.1 and ACI 318-11 Section 11.5.1). A warning is produced, implying that the shear reinforcement has to be designed by the engineer for torsion.

Creep / Long Term Deflections – No considerations are taken in the analysis to account for the effects of creep or long term deflections.

Beam Design – Beams are not designed for weak axis y-y bending, weak axis shear, or axial forces. A message is shown in the detail report to remind you of this. You can turn the warning messages off on the **Concrete** tab of the **Model Settings Dialog**. Beams currently do not consider any compression steel in the calculation of the moment capacity. Beam "skin reinforcement" per the requirements of ACI 318-14 Section 9.7.2.3 (ACI 318-11 Section) 10.6.7 for beams with "d" greater than 36" is currently not specified by the program. The provisions in ACI 318-14 Section 9.9 (ACI 318-11 Section 10.7) for deep beams are not considered.

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

Limitations - ACI

Shear Design –When ACI 318-19 is selected, the shear strength of concrete (V_c) uses equations in Table 22.5.5.1. Note that for members meeting the minimum shear reinforcement requirement ($A_v \geq A_{v,min}$), V_c is taken as the larger of the results calculated by the equations (a) and (b) in the table. ACI 318-19 code suggests ρ_w can be taken as the sum of the areas of longitudinal bars located more than two-thirds of the overall member depth away from the extreme compression fiber. Therefore, RISA calculates ρ_w as the sum of the areas of longitudinal bars on the tension face.

When other ACI 318 editions are selected, the shear strength of the concrete alone is limited to the standard $2\lambda\sqrt{f_c}$ equation from ACI 318-14 Section 22.5.5.1 (ACI 318-11 Section 11.2.1.1) and does not use the more detailed calculations of ACI 318-14 Table 22.5.5.1 (ACI 318-11 Section 11.2.2). Also, note that for members with significant axial tension (greater than 0) the program designs the shear reinforcement to carry the total shear per ACI 318-14 Section R22.5.7.1 (ACI 318-11 Section 11.2.1.3).

Deep Beam Design – The program does not design deep beams as defined in ACI 318-14 Section 9.9.1.1(a) (ACI 318-11 Section 10.7).

Threshold Torsion - The program does not adjust the threshold torsion value for the presence of axial force in a beam, though it does do this for columns.

Limitations - Canadian Code

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

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

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

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

Limitations - Australian and New Zealand Codes

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

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

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

Shear Strength in Beams – In AS code, when calculating the shear strength of a beam β_2 , β_3 are always assumed to be unity. This is always conservative for beams with little axial load, or beams in compression. But, may be unconservative for members subjected to significant net tension.

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

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

Development Length – Development length in NZS is based on NZS 7.3.7.2 where α is conservative assumed to be 1.3 (top bars) for all cases. For the AS code, it is assumed that $K_1=1$ and $K_2=2.4$ in clause 13.1.2.1 of AS 3600:2001.

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

Limitations - British

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

Cracked Sections – $I_{cracked}$ defaults to 1.0 for the British code. But, you can enter a value if you want. Service level stiffness is assumed to be 1.43 times the strength stiffness, but is not allowed to exceed I_{gross} .

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

Limitations - Euro

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

Cracked Sections – $I_{cracked}$ defaults to 1.0 for EuroCode. But, you can enter a value if you want. Service level stiffness is assumed to be 1.43 times the strength stiffness, but is not allowed to exceed I_{gross} .

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

Limitations - Indian

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

Cracked Sections – Icracked defaults to 1.0 for the Indian code. But, you can enter a value if you want. Service level stiffness is assumed to be 1.43 times the strength stiffness, but is not allowed to exceed Igross.

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

Limitations - Saudi Code

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

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

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

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

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

Special Messages

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

No Load Combinations for Concrete Design have been run.

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

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

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

Warning: No design for spans less than 1 ft.

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

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

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

Warning: Slender Compression Failure ($P_u > .75P_c$). No Slender calculations done.

Since RISA-3D allows you to specify a starting column size, it's possible that for slender columns under substantial axial load you'll exceed the critical buckling load used in the slenderness equations in ACI 318-14 Section 6.6.4.5.2 (ACI 318-11 Section 10.10.6). Design results are still shown so the suggested shapes can be used to pick a new suggested column size that will not have this problem. Note that the design results shown are NOT valued because the slender moment effects have NOT been considered.

Warning: $KL/r > 100$ for this compression member. See ACI 318-05 Section 10.11.5

Members that violate the KL/r limit still have design results calculated and shown. If you're using the redesign feature, the next suggested shape should resolve this problem. Note this is only checked for the 2005 code and older. Newer codes require a P-Delta analysis and omit this consideration.

Warning: Exact Integration selected but PCA method used

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

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

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

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

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

P-Delta analysis required for all ACI 318-14/11/08 Load Combinations

A second order analysis is required as of the 2014, 2011 and 2008 editions of the ACI 318 code. A code check is only be given if [P-Delta](#) is turned on in the [Load Combinations](#) spreadsheet, or if this requirement is intentionally waived in the [Application Settings](#).

Concrete Member - Design Results

Access the **Code Check** spreadsheet by selecting the **Results** tab ▶ **Envelope** (or **LC**) button ▶ **Code Check**. Alternatively, on the **Results** toolbar you can select **Code Check**. Unlike wood and steel, concrete results are different for beams and columns so they each get their own results spreadsheet. Note also that concrete results are always based on envelope results, even if you've run a batch solution.

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

Beam Results

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

Design Results Spreadsheet

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

Envelope Concrete Beam Design Results														
Hot Rolled Steel	Cold Formed Steel	Wood	Concrete Beams			Concrete Columns		Aluminum		Stainless				
Member	Shape	UC Max Top	Loc[ft]	UC LC	UC Max Bot	Loc[ft]	UC LC	Shear UC	Loc[ft]	UC LC	Phi*Mnz Top[k-ft]	Phi*Mnz Bot[k-ft]	Phi*Vny...	
1	M4	CRECT12X8	0.792	18	2	0.984	6	2	0.364	11.625	2	44.327	34.262	6.59
2	M5	CRECT8X8	0.464	17.438	2	0.276	6	2	0.344	13.5	2	13.772	13.772	4.1
3	M6	CRECT8X8	0.398	0.563	2	0.342	12	2	0.43	4.313	2	13.772	13.772	4.12
4	M7	CRECT8X8	0.221	17.438	2	0.112	6	2	0.245	2.063	2	13.772	13.772	4.109
5	M8	CRECT8X8	0.227	0.562	2	0.122	10.5	2	0.155	4.125	2	13.772	13.772	4.114
6	M9	CRECT8X8	0.186	17.25	2	0.097	7.875	2	0.183	2.438	2	13.772	13.772	4.11
7	M10	CRECT8X8	0.246	0.375	2	0.123	10.5	2	0.208	17.063	2	13.772	13.772	4.101
8	M11	CRECT12X8	0.924	17.625	2	0.852	7.125	2	0.583	10.688	2	34.262	24.816	5.862
9	M12	CRECT12X8	0.81	17.625	2	0.912	6.75	2	0.595	3.188	2	44.281	25.55	5.862
10	M13	CRECT12X8	0.934	17.438	2	0.721	7.313	2	0.595	10.875	2	34.235	25.55	5.862
11	M14	CRECT12X8	0.898	17.625	2	0.726	8.625	2	0.591	5.063	2	24.803	25.55	5.862

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

Beam Bending Reinforcement Spreadsheet

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

Concrete Beam Bending Reinforcement									
Column Shear		Concrete Wall		Masonry Wall			Masonry Lintel		
Beam Bending			Beam Shear			Column Bending			
	Member	Shape	Span	Left Top	Left Bot	Mid Top	Mid Bot	Right Top	Right Bot
1	M4	CRECT12X8	1	2 #5			2 #6	2 #7	
2	M5	CRECT8X8	1	2 #5			2 #5	2 #5	
3	M6	CRECT8X8	1	2 #5			2 #5	2 #5	
4	M7	CRECT8X8	1	2 #5			2 #5	2 #5	
5	M8	CRECT8X8	1	2 #5			2 #5	2 #5	
6	M9	CRECT8X8	1	2 #5			2 #5	2 #5	
7	M10	CRECT8X8	1	2 #5			2 #5	2 #5	
8	M11	CRECT12X8	1	2 #5	2 #5		2 #5	2 #6	
9	M12	CRECT12X8	1		2 #5		2 #5	2 #7	
10	M13	CRECT12X8	1	2 #5			2 #5	2 #6	

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

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

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

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

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

**Note:**

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

Beam Shear Reinforcement Spreadsheet

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

Concrete Beam Shear Reinforcement						
Column Shear		Concrete Wall		Masonry Wall		Masonry Lintel
Beam Bending			Beam Shear			Column Bending
	Member	Span	Region 1	Region 2	Region 3	Region 4
1	M4	1	18 #4...			17 #4...
2	M5	1	7 #4 @...			21 #4...
3	M6	1	20 #4...			7 #4 @...
4	M7	1	4 #4 @...			
5	M8	1	16 #4...			9 #4 @...
6	M9	1	8 #4 @...			14 #4...
7	M10	1	16 #4...			
8	M11	1	7 #4 @...			18 #4...
9	M12	1	6 #4 @...			20 #4...
10	M13	1	10 #4...			17 #4...

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

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

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

The **Region 1**, **Region 2**, **Region 3**, and **Region 4** entries record the number, size, and spacing of shear reinforcement ties/stirrups that are required in each of the **Reinforcement Regions**. The first

number of each entry indicates the total number of ties/stirrups that are required in that region of the beam span. The second number, preceded by the '#' sign, indicates the size of reinforcement bars used. The third number, preceded by the "@" symbol, indicates the spacing of the ties/stirrups in that region of the beam span.

**Note:**

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

Column Results

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

Design Results Spreadsheet

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

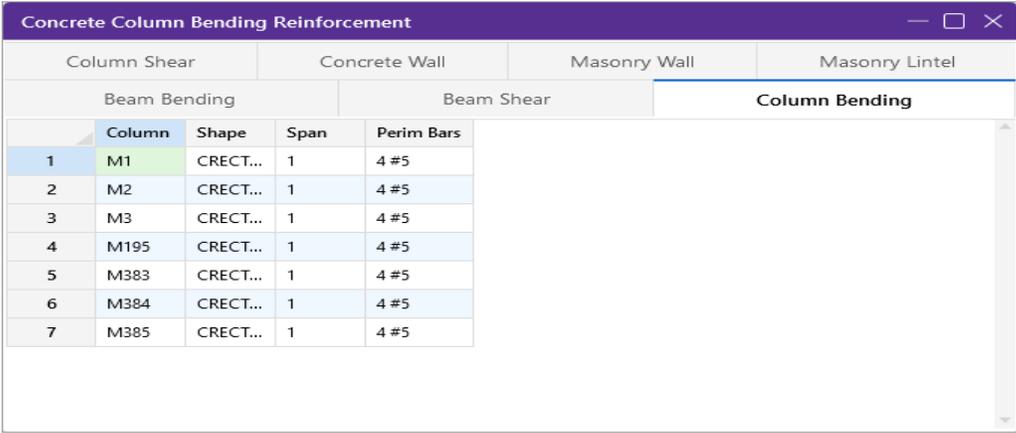
Envelope Concrete Column Design Results															
Hot Rolled Steel	Cold Formed Steel	Wood	Concrete Beams	Concrete Columns						Aluminum	Stainless				
Column	Shape	UC Max	Loc[ft]	UC LC	Shear UC	LC	Loc[ft]	Dir	Phi used	Pn[k]	Mny[k-ft]	Mnz[k-ft]	Vny[k]	Vnz[k]	
1	M1	CRECT12X10	0.635	11.625	2	0.355	2	11.625	y	0.9	36.609	31.342	39.386	54.736	52.245
2	M2	CRECT12X10	0.454	0	2	0.016	1	11.5	y	0.65	301.201			36.597	31.35
3	M3	CRECT12X10	0.376	11.625	2	0.035	2	11.625	z	0.65	193.393	39.166	2.621	37.035	31.767
4	M195	CRECT12X10	0.36	11.625	2	0.038	2	11.625	y	0.9	29.544	29.71	37.253	34.484	29.338
5	M383	CRECT12X10	0.885	12	1	0.031	2	12	y	0.65	301.201			45.211	39.553
6	M384	CRECT12X10	0.815	12	1	0.029	2	12	y	0.65	301.201			45.211	39.553
7	M385	CRECT12X10	0.88	12	1	0.024	2	12	z	0.65	301.201			45.211	39.553

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

The moving load results are enveloped and the governing load combination and step location is shown for each result value under the "**UC LC**" column. The first number is the load combination, the second is the step number: (load combination - step number). See [Moving Loads](#) to learn more.

Column Bending Reinforcement Spreadsheet

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



Concrete Column Bending Reinforcement					
Column Shear		Concrete Wall		Masonry Wall	Masonry Lintel
Beam Bending		Beam Shear		Column Bending	
	Column	Shape	Span	Perim Bars	
1	M1	CRECT...	1	4 #5	
2	M2	CRECT...	1	4 #5	
3	M3	CRECT...	1	4 #5	
4	M195	CRECT...	1	4 #5	
5	M383	CRECT...	1	4 #5	
6	M384	CRECT...	1	4 #5	
7	M385	CRECT...	1	4 #5	

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

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

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

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

**Note:**

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

Column Shear Reinforcement Spreadsheet

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

Concrete Column Shear Reinforcement							
Beam Bending		Beam Shear				Column Bending	
Column Shear		Concrete Wall		Masonry Wall		Masonry Lintel	
	Column	Span	Region 1	Region 2	Region 3	Region 4	
1	M1	1	12 #4...			6 #4 @...	
2	M2	1	14 #4...				
3	M3	1	14 #4...				
4	M195	1	14 #4...				
5	M383	1	14 #4...				
6	M384	1	14 #4...				
7	M385	1	14 #4...				

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

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

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

The **Region 1**, **Region 2**, **Region 3**, and **Region 4** entries record the number, size, and spacing of shear reinforcement ties/stirrups that are required in each of the **Reinforcement Regions**. The first number of each entry indicates the total number of ties/stirrups that are required in that region of the

column span. The second number, preceded by the '#' sign, indicates the size of reinforcement bars used. The third number, preceded by the '@' symbol, indicates the spacing of the ties/stirrups in that region of the column span.



Note: If '2' shear regions are selected on the Rebar tab of the Model Settings dialog, the Region 2 and Region 3 entries in this spreadsheet will be left blank.

Concrete Detail Reports

The **Concrete Detail Reports** allow you to see the overall force, stress, and deflection state for any particular member. Detail reports for concrete members are not based on individual load combinations as they are for steel or wood members. Instead, they are based on an envelope of the solved load combinations. Concrete columns are the exception to this in that the columns are solved for all load combinations and then the resulting required steel is enveloped. The detail reports for concrete Column member types are also different than those for concrete Beam member types in terms of the design information that is shown below the force diagrams.

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



Note: The  **New Floor Plan** button will allow you to take a snapshot of the current detail report you are viewing so that it can be added to a report. View the [Printing](#) topic for more information.

Beam Detail Reports

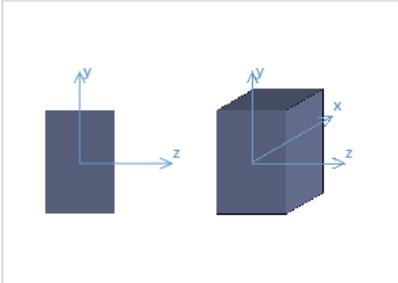
The image below is the first portion of a detail report for a concrete beam member showing the member information, warnings, force diagrams, code checks, and span information.

Detail Report for - M11 — □ ×

< Member Label M11 >
Envelope ▾
Options

Detail Report: M11

Load Combination: Envelope Code check: 0.924 (LC 2)



Input Data			
Shape:	CRECT12X8	I Node:	N23
Member Type:	Beam	J Node:	N24
Length (ft):	18	I Release:	Fixed
Material Type:	Concrete	J Release:	Fixed
Design Rule:	Typical	I Offset:	N/A
Internal Sections:	97	J Offset:	N/A
Design Code:	ACI 318-19	T/C Only:	Both Way

Material Properties			
Material:	Conc4000NW	Therm. Coeff. (1e ⁵ F ^o e ⁻¹):	0.6
E (ksi):	3644	Density (k/ft ³):	0.145
G (ksi):	1584	f _c (ksi):	4
Nu:	0.15	λ:	1
		Flex Steel (ksi):	60
		Shear Steel (ksi):	60

Shape Properties			
D (in):	12	W (in):	8

Design Properties			
B-eff Left:	N/A	Cracked Sections Used:	Yes
B-eff Right:	N/A	Cracked "I" Factor:	0.35
Slab Thk Left:	N/A	Effective "I" (in ⁴):	403.2
Slab Thk Right:	N/A	Effective "I"(Service) (in ⁴):	576.576
Concrete Stress Block:	Parabolic	Flex Rebar Set:	ASTM A615
		Shear Rebar Set:	ASTM A615
		Top Cover (in):	1.5
		Bottom Cover (in):	1.5
		Side Cover (in):	1.5
		Legs/Stirrup:	2



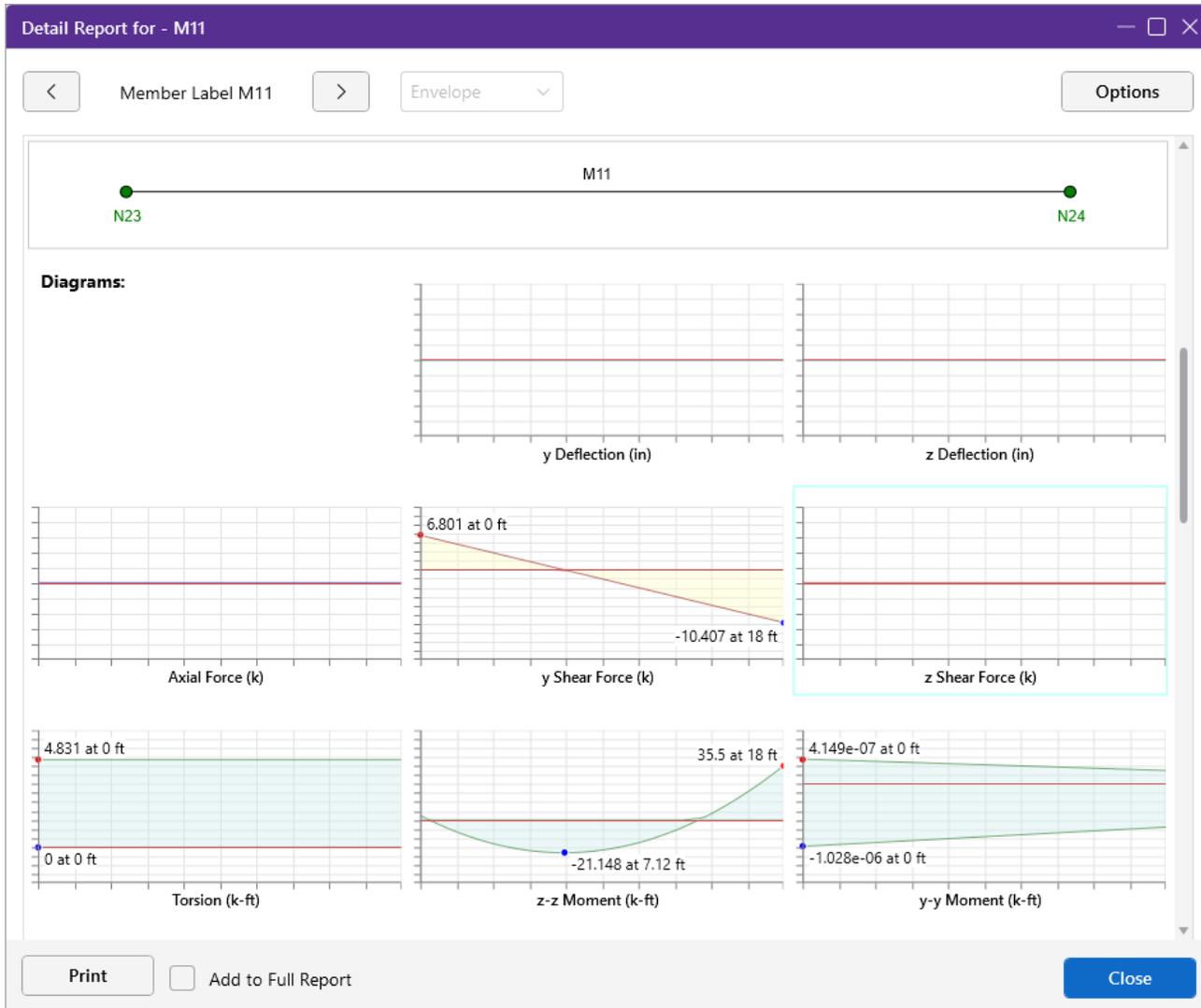
Diagrams:

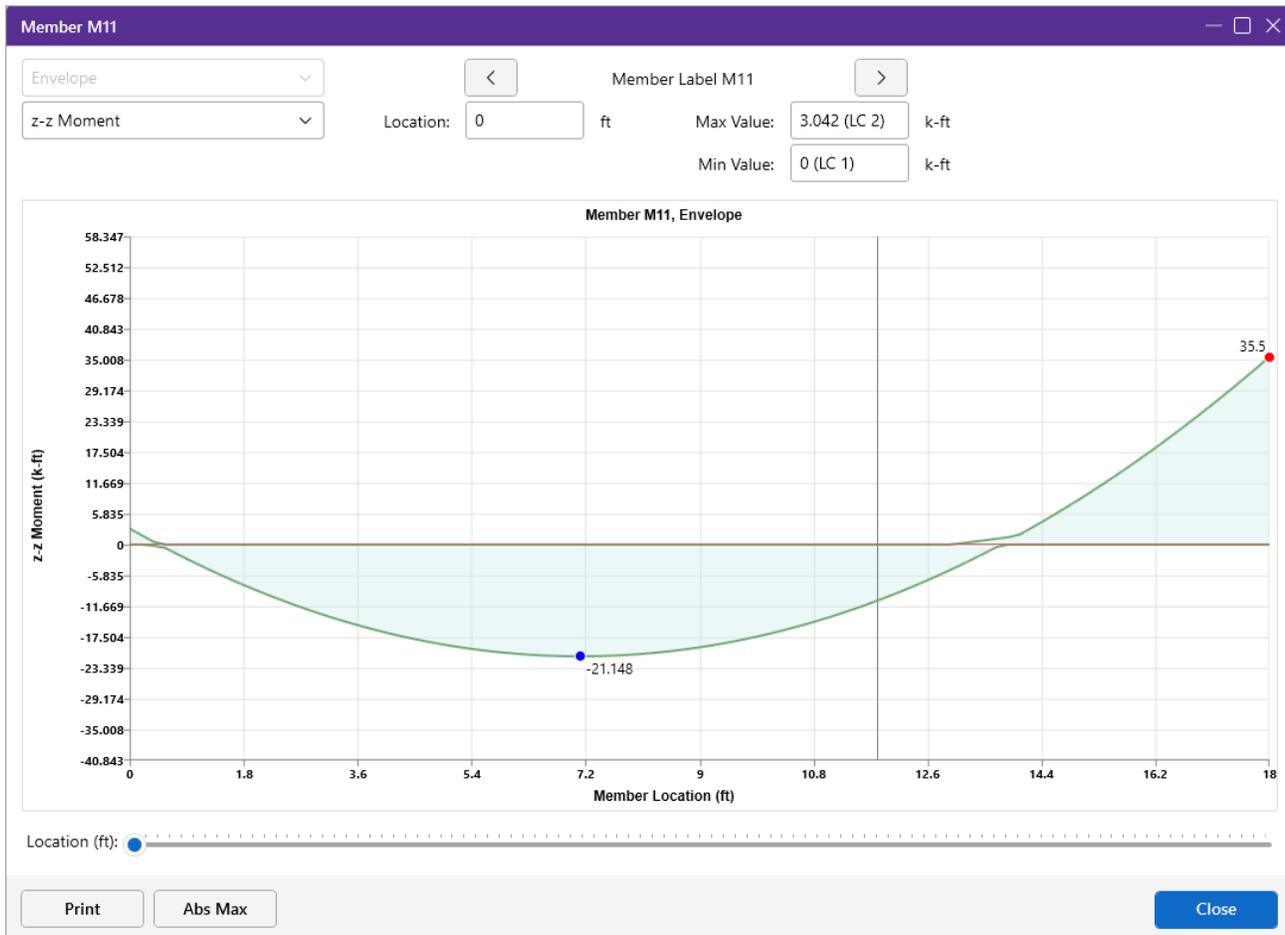
Print
 Add to Full Report
Close

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

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

The next section of the detail report contains the **Member Force Diagrams**. The diagrams shown are envelope diagrams of all solved load combinations. Any **Unused Force Warnings** or critical **Design Warnings** will be shown directly below the force diagrams in the detail report.





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

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

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

There is also general concrete, reinforcement, and bar cover information about the section provided which you would need if you were doing a hand check. **Concrete Type** (Normal Weight vs Light Weight) is automatically determined from the **Concrete Weight** density per the ACI code. λ is taken from the [Materials](#) spreadsheet. The **E_Concrete** value shown here is either the value entered on

the **Concrete** tab of the **Materials Spreadsheet** or is the calculated value based on the given f'_c and weight density (if the 'E' value was left blank on the Materials Spreadsheet).

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

The next portion of the detail report shown below contains detailed information for the placement of the **Bending Steel** and the **Bending Span Results** for each span. The bending capacity for the governing section in each span is shown as **Mnz**, the nominal moment strength. **Rho Min** and **Rho Max** are the minimum and maximum required reinforcement ratios at each location. These values are based on the minimum and maximum reinforcing requirements for flexural members as described in ACI 318-14 Sections 9.6.1 and 21.2.2/9.3.3.1 respectively (ACI 318-11 Sections 10.5.1 and 10.3.3/10.3.5 respectively). **Rho** is the ratio of reinforcement corresponding to the area of steel provided at each location, **As Prvd**. The **As Req** value is the area of steel required at each location.



Note: Per ACI 318-14 Section 9.6.1 (ACI 318-11 Section 10.5.3), the reinforcement ratio (ρ) chosen by the program can be less than ρ_{min} when A_s Provided exceeds A_s Required by more than 33%

Detail Report for - M11

Member Label M11 Envelope Options

ACI 318-19 Code Check

Beam Design does not consider any 'T' & 'My' Moments, nor 'A' & 'Vz' Forces.
Factored torsional moment T_u exceeds the threshold torsion per ACI 318-19 22.7.4.1

Limit State	Gov. LC	Required	Available	Unity Check	Result
Applied Loading - Bending/Axial	-	-	-	-	-
Applied Loading - Shear	-	-	-	-	-
Top Bending Steel Requirements	2	1.838 bars	2 bars	-	PASS
Top Flexural Minimum	2	0.581 bars	2 bars	-	PASS
Top 4/3 Requirement	2	-	-	-	-
Top Bending Check	2	31.665 k-ft	34.262 k-ft	0.924	PASS
Bottom Bending Steel Requirements	2	1.705 bars	2 bars	-	PASS
Bottom Flexural Minimum	2	0.842 bars	2 bars	-	PASS
Bottom 4/3 Requirement	2	-	-	-	-
Bottom Bending Check	2	21.148 k-ft	24.816 k-ft	0.852	PASS
Threshold Torsion	-	4.831 k-ft	0.911 k-ft	5.304	FAIL
Shear Design Strength	2	3.416 k	5.862 k	0.583	PASS
Shear Reinforcement Required	2	3.652 k	-	-	-
Minimum Development Length	-	-	-	-	-
Hook Development Length	-	-	-	-	-

Print Add to Full Report Close

The next portion of the detail report, shown above, contains detailed information for the placement of the **Shear Steel** and the **Shear Span Results** for each span. Shear results are shown by region within each span. The number of regions used is a function of the shear diagram, with the maximum number of regions being taken from the **Shear Regions** setting on the **Concrete** tab of the **Model Settings Dialog**. The number, size, and spacing of reinforcing bars is given for each region. Also indicated is the nominal shear strength, **V_n**, in each region. The portion of the nominal shear strength provided by the concrete and the steel, **V_c** and **V_s** respectively, is given for each region. The area of steel required, **As Req'd**, and the area of steel provided, **As Prvd**, are also given for each shear region and are reported as 'area of steel per unit dimension', i.e. in²/in or mm²/mm.

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

The last section of the detail report shows the **Beam Reinforcement Detailing Diagrams**. The **Rebar Detailing** portion of the report shows elevation views of the beam complete with top and bottom flexural reinforcement indicated for the left, middle, and right portions of each span. The number and size of bars required in each section is indicated on the top middle of each drawn

bar. The required length of each bar is indicated on the bottom middle of each drawn bar in parenthesis. Development lengths are shown in parenthesis at one end of each bar and is represented by a dashed line. For bars at the ends of the beam, hook lengths are given in addition to the development lengths and are shown in brackets.



Note: Development lengths are calculated per ACI 318-14 Sections 25.4.2 and 25.4.3 (ACI 318-11 Sections 12.2.2 and 12.5.2). No additional factors are used, aside from the lightweight modification factor.

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

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

Detail Report for - M11 — □ ×

< Member Label M11 >
Envelope ▾
Options

Span	Span Length (ft)	I-Face Dist. (in)	J-Face Dist. (in)
1	0 - 18	4	4

Bending Steel

Span	Loc	Top/Bot	Bars Provided
1	Left	T	2 #5
	Left	B	2 #5
	Mid	T	-
	Mid	B	2 #5
	Right	T	2 #6
	Right	B	-

Shear Steel

Span	Region (ft)	Bars Provided
1	1.1 - 3.4	7 #4 @4in
	-	-
	10.9 - 16.9	18 #4 @4in

Rebar Detailing, face of support to face of support of each span(Units: in)

The diagram shows a horizontal span of 18 feet. The top reinforcement consists of 2 #5 bars at the left end (hooked at 7.4 inches), 2 #5 bars in the middle (hooked at 23.7 inches), and 2 #6 bars at the right end (hooked at 28.5 inches). The bottom reinforcement consists of 7 #4 bars spaced at 4 inches from 1.1 to 3.4 feet, and 18 #4 bars spaced at 4 inches from 10.9 to 16.9 feet. A tie start distance of 2 inches is indicated at the left end.

Cross Section Detailing(All Bars Equally Spaced, Units: in)

The diagrams show three cross-sections: 'Start', 'Middle', and 'End'. The 'Start' section has a height of 12 inches and a width of 8 inches, with 2 #5 bars at the top and bottom. The 'Middle' section has a height of 23 inches and a width of 8 inches, with 2 #5 bars at the top and bottom. The 'End' section has a height of 15 inches and a width of 8 inches, with 2 #5 bars at the top and bottom. Shear ties are shown as U-shaped stirrups with 1.5-inch clear cover on the top and sides.

Print
 Add to Full Report
Close

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

Column Detail Reports

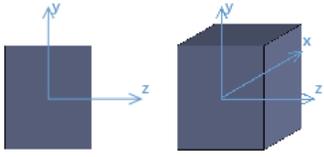
The image below is the first portion of a detail report for a concrete column member showing the member information, warnings, force diagrams, code checks, and span information. As can be seen, the concrete column results are very similar to the beam results with just a few additions and differences.

Detail Report for - M195
— □ ×

< Member Label M195 >
Envelope ▾
Options

Detail Report: M195

Load Combination: Envelope Code check: 0.360 (LC 2)



Input Data			
Shape:	CRECT12X10	I Node:	N13
Member Type:	Column	J Node:	N14
Length (ft):	12	I Release:	Fixed
Material Type:	Concrete	J Release:	Fixed
Design Rule:	Typical	I Offset:	N/A
Internal Sections:	97	J Offset:	N/A
Design Code:	ACI 318-19	T/C Only:	Both Way

Material Properties			
Material:	Conc3000NW	Therm. Coeff. (1e ⁵ F ⁻¹):	0.6
E (ksi):	3156	Density (k/ft ³):	0.145
G (ksi):	1372	f _c (ksi):	3
Nu:	0.15	λ:	1
		Flex Steel (ksi):	60
		Shear Steel (ksi):	60

Shape Properties			
D (in):	12	W (in):	10

Design Properties			
C _{m y-y} :	N/A	z sway:	No
C _{m z-z} :	N/A	Concrete Stress Block:	Parabolic
K _{y-y} :	1	Cracked Sections Used:	Yes
K _{z-z} :	1	Cracked "I" Factor:	0.7
y sway:	No	Effective "I" (in ⁴):	1008
		Effective "I" (Service) (in ⁴):	1441.44
		Flex Rebar Set:	ASTM A615
		Shear Rebar Set:	ASTM A615
		Side Cover (in):	1.5
		Legs/Stirrup:	2



Diagrams:

--	--

Print
 Add to Full Report
Close

Detail Report for - M195

Member Label M195 Envelope Options

ACI 318-19 Code Check

Column Design does not consider any Torsional Moments
Warning: Exact Integration selected but PCA method used

Limit State	Gov. LC	Required	Available	Unity Check	Result
Applied Loading - Bending/Axial	-	-	-	-	-
Flexural Reinforcement	2	1.2 in ²	1.227 in ²	-	PASS
Axial Capacity	2	26.59 k	26.59 k	-	PASS
Bending Unity Check	2	-	-	0.36	PASS
y Shear Design Strength	2	0.98 k	25.863 k	0.038	PASS
z Shear Design Strength	2	0.804 k	22.003 k	0.037	PASS
Threshold Torsion	-	0 k-ft	0 k-ft	0	PASS

Print Add to Full Report Close

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

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

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

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

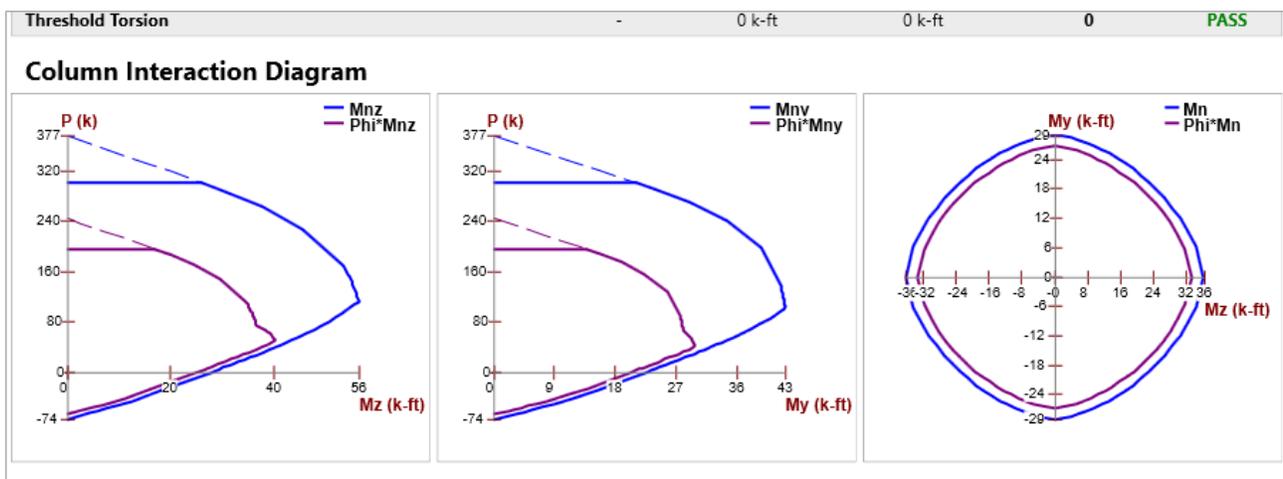
There is also general concrete, reinforcement, and bar cover information about the section provided which are useful for hand calculation verification. **Concrete Type** (Normal Weight vs Light Weight) is automatically determined from the **Concrete Weight** density per the ACI code. The **E_Concrete** value shown here is either the value entered on the **Concrete** tab of the **Materials Spreadsheet** or is the calculated value based on the given $f'c$ and weight density (if the 'E' value was left blank on the Materials Spreadsheet).



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

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

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



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

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

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

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

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

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

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

The **Bending Span Results** show the calculated eccentricities, **ecc. y** and **ecc. z**, due to the ultimate moment about each axis divided by the ultimate axial load. The neutral axis depth for bending about the local y-axis and z-axis are listed as **NA y-y** and **NA z-z** respectively. These neutral axis locations are always given with respect to the geometric center of the column. Also shown in this section are the unreduced nominal moment capacities, **Mny** and **Mnz**, for each span of the column. If the **PCA Load Contour Method** is used, **Mnoy** and **Mnoz** are given, representing the maximum allowable moment for uniaxial bending at the nominal axial strength, **Pn** (see [Biaxial Bending of Columns](#)). If the **Exact Integration Method** is used, these values will be left blank.

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

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



Note: ACI 318-19 requires the consideration of shear interaction in y and z directions when both $V_{uy}/\phi V_{ny}$ and $V_{uz}/\phi V_{nz}$ are larger than 0.5. In this case, the program recalculates the shear UC based on Equation 22.5.1.11. If the UC considering shear interaction happens to govern, a new section is added in Detail Report to present this check:

y Shear Design Strength	1	100 k	136.515 k	0.733	PASS
z Shear Design Strength	1	120 k	136.515 k	0.879	PASS
Interaction of Shear in Y and Z Direction	1	-	-	1.074	FAIL
Span 1 Enveloped Forces					
Shear in Y direction					
$V_{uy} = 100\text{ k}$	Governing shear force				
$\phi V_{ny} = 136.515\text{ k}$	Allowable shear strength				(22.5.1.1)
Shear in Z direction					
$V_{uz} = 120\text{ k}$	Governing shear force				
$\phi V_{nz} = 136.515\text{ k}$	Allowable shear strength				(22.5.1.1)
Shear check considering iteration in Y and Z directions					
$UC_{Shear} = \left(\frac{V_{uy}}{\phi V_{ny}} + \frac{V_{uz}}{\phi V_{nz}}\right) \div 1.5 = 1.074$	Shear unity check considering interaction				(22.5.1.11)

- In the Code Check spreadsheet the direction (Dir) becomes “y,z” to indicate the governing load combination is considering shear interaction.

Envelope Concrete Column Design Results															
Hot Rolled Steel		Cold Formed Steel		Wood		Concrete Beams		Concrete Columns		Aluminum		Stainless			
Column	Shape	UC Max	Loc[ft]	UC LC	Shear UC	LC	Loc[ft]	Dir	Phi used	Pn[k]	Mny[k-ft]	Mnz[k-ft]	Vny[k]	Vnz[k]	
1	M1	CRECT24X24	3.888	0	1	1.074	1	17	y,z	0.864		1436.044	1436.044	182.02	182.02

Magnified Moments / Slenderness Effects

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

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

For Non-Sway frames, the assumption is that $EI = 0.25 \cdot E_c \cdot I_g$. This is equivalent to setting B_{dns} to 0.6 in ACI 318-14 Equation 6.6.4.4a (ACI 318-11 Equation 10-15). For sway frame columns with a KL/r value greater than 22, the moment amplification is applied to the total moment rather than the "non-sway" portion of the moment.

Warning Log Messages will be produced when the following occurs:

- If the KL/r for the column exceeds 100 per Section 10.11.5 of ACI 318-05. Note this message only occurs for 2005 and older codes.

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

RISAConnection Integration

RISAConnection can be used with RISAFloor and/or RISA-3D to design hot-rolled steel connections. The integration will send the member geometry, loads, and connection type information into RISAConnection. You can then open RISAConnection to edit/adjust the details of the connection. When satisfied with an appropriate connection design, you can then send the connection design results back into RISAFloor and RISA-3D for results presentation.

Available Connection Types

The list of available connection types depends on what version of RISAConnection you are integrating with. The following table lists the available connection types from the current version of RISAConnection.

Available Connections

Connection	Description
Beam to Column Shear Connections	(Shear tab, clip angle, end plate) <ul style="list-style-type: none"> Columns may be Wide Flanges, HSS tubes, or HSS pipes (shear tab only). Beams may be Wide Flanges or Channels (not for end plate). Double-sided connections available for clip angle shear connections into the column web.
Beam to Girder Shear Connections	(Shear tab, clip angle, end plate) <ul style="list-style-type: none"> Girders may only be Wide Flanges. Beams may be Wide Flanges or Channels (not for end plate). Double-sided connections available for clip angle shear connections into the girder web.
Shear Splices	(Beam and column) <ul style="list-style-type: none"> Columns may only be Wide Flanges. Beams may only be Wide Flanges.
Beam to Column Flange Moment Connections	(Direct weld, extended end plate, flange plates, flush end plate) <ul style="list-style-type: none"> Direct Weld and Flange Plate moment connection columns may be Wide Flanges or HSS Rectangular Tubes. Extended End Plate and Flush Plate moment

Connection	Description
	connection columns may only be Wide Flanges. <ul style="list-style-type: none"> • Beams may only be Wide Flanges.
Beam to Column Web Moment Connections	(Direct weld, flange plates) <ul style="list-style-type: none"> • Columns may only be Wide Flanges. • Beam may only be Wide Flanges.
Beam to Column Web Moment Connections	(Continuous beam over column) <ul style="list-style-type: none"> • Columns may only be Wide Flanges, HSS Tubes, or HSS pipes. • Beam may only be Wide Flanges.
Moment Splices	(Beam and column, direct weld, end plate, flange plate) <ul style="list-style-type: none"> • Columns may only be Wide Flanges. • Beams may only be Wide Flanges.
Vertical Brace Connections	(Chevron, Diagonal Brace, and Knee Brace) <ul style="list-style-type: none"> • Columns may be Wide Flanges, HSS Tubes, or HSS Pipes. • Beams may be Wide Flanges, HSS Tubes (Chevron and Knee Brace connections only), or HSS Pipes (Chevron and Knee Brace connections only). • Chevron and Diagonal Braces may be Wide Flanges, HSS Tubes, HSS Pipes, Channels, WTs, Double Angles, or Single Angles. • Knee Braces may be HSS Tubes, HSS Pipes, Channels, WTs, Double Angles, or Single Angles.
Single Column Base Plate Connections	<ul style="list-style-type: none"> • Column may be a Wide Flange, Tube, or Pipe.
Brace to Base Plate Connections	<ul style="list-style-type: none"> • Column may be a Wide Flange, Tube, or Pipe.
Brace to Column Base Plate Connections	<ul style="list-style-type: none"> • Column may be a Wide Flange, Tube, or Pipe. • Brace may be HSS Tube, HSS Pipe, Single Angle, Double Angle, WTs, or Channels.
Truss HSS T-Connections	<ul style="list-style-type: none"> • Chord member must be a HSS Tube. • Branch member must be a HSS Tube.

See [Troubleshooting](#) for more assistance with connection type limitations.

Integrated Connection Design Procedure

The steps required to design connections using this integration are as follows:

1. [Complete the Full Structural Model](#)
2. [Define the Connection Rules](#)
3. [Assign the Connection Rules](#)
4. [Assign Load Combinations](#)
5. [Design Connections](#)
6. [Connection Design Results](#)
7. [Multiple Round Trips Between RISAFloor/RISA-3D and RISAConnection](#)

1. Complete the Full Structural Model

You must first draw your model in RISA-3D or RISAFloor. It is important that you use the [Member Type](#) (Column, Beam, VBrace, HBrace) designation properly. Otherwise your connections will not be designed. Keep in mind that connection design will only work for hot rolled connections at this point. Future versions will add to the current RISAConnection connection type capabilities.

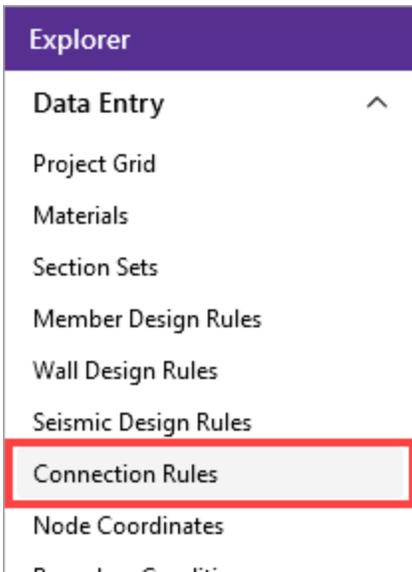
2. Define Connection Rules

The second step is to define Connection Rules. These rules allow you to specify which types of connections you want to design in your model. You must have a rule for each type of connection you want considered.

You may also want to create two separate rules for the same connection Type, so you can manipulate these groups separately.

To open the Connection Rules spreadsheet:

1. Click on **Connection Rules** in the 'Explorer' panel, as shown in the following image.



The Connection Rules spreadsheet opens.

Connection Rules						
Hot Rolled						
	Label	Conn Type	Type	Beam Conn	Col/Girder Conn	
1	Col/Bm Clip Angle	Shear	Column/Beam Clip Double Angle Shear	Welded	Bolted	
2	Col/Bm Shear Tab	Shear	Column/Beam Shear Tab Shear	Bolted	N/A	
3	Girder/Bm Clip Angle	Shear	Girder/Beam Clip Single Angle Shear	Welded	Bolted	
4	Girder/Bm Shear Tab	Shear	Girder/Beam Shear Tab Shear	Bolted	N/A	
5	Flange Plate Moment	Moment	Column/Beam Flange Plate Moment	Bolted	N/A	
6	End-Plate Moment	Moment	Column/Beam Extended End-Plate Moment	N/A	N/A	
7	Col Shear Splice	Shear	Column Shear Tab Splice	N/A	N/A	
8	Col Moment Splice	Moment	Column Moment Plate Splice	N/A	N/A	
9	Diagonal Brace	Brace	Diagonal Vertical Brace	N/A	N/A	
10	Chevron Brace	Brace	Chevron Vertical Brace	N/A	N/A	
11	Knee Brace	Brace	Knee Brace	N/A	N/A	

2. Create rules for each type of connection that you want to be designed through RISAConnection.

The spreadsheet entries are explained in the following table.

Connection Rules Spreadsheet Columns

Column	Description
Label	Allows you to identify your Connection Rule in other areas in the program. Each Connection Rule must have a unique 'Label'.

Column	Description
Conn Type	Allows you to filter the list of 'Types' based on the a specific connection type. Available Connection Types are Shear, Moment, Brace, Baseplate or Truss.
Type	<p>Allows you to choose the Type of connection from a list of available types currently considered in RISAConnection.</p> <p>You need to set up at least one connection rule for each connection type in your project. When you solve the connections in RISAConnection, the member ends assigned to an individual Connection Rule are grouped together.</p> <div data-bbox="610 737 1409 951" style="border: 1px solid black; padding: 5px; margin-top: 10px;">  Note: You may have many double angle shear connections in your project. However, some are bolted with A325 bolts and some use A490 bolts. You should define these as separate Connection Rules. </div>
Beam Conn	<p>Allows you to define how the beam is connected to the supporting pieces (plate, clip angle, etc.). When you go to RISAConnection, all connections grouped in this Connection Rule default to this type of connection.</p> <div data-bbox="610 1129 1409 1239" style="border: 1px solid black; padding: 5px; margin-top: 10px;">  Note: Many connections do not have options for these, so "N/A" is shown instead. </div>
Col/Girder Conn	<p>Allows you to define how the supporting member (column or girder) is connected to the supporting pieces (plate, clip angle, etc.). When you go to RISAConnection, all connections grouped in this Connection Rule default to this type of connection.</p> <div data-bbox="610 1451 1409 1560" style="border: 1px solid black; padding: 5px; margin-top: 10px;">  Note: Many connections do not have options for these, so "N/A" is shown instead. </div>
Eccentricity	<p>This value has no effect on the RISAConnection integration. Instead, it is used only for analysis purposes.</p> <p>In models integrated with RISAFloor, beam/column shear connections add an automatic eccentricity equal to half the column's depth automatically (see the Model Settings - Solution tab for more information on this). However an <i>additional</i> eccentricity away from the column face may be entered in the Connection Rules spreadsheet. This column allows you to enter that value here. The value entered</p>

Column	Description
	<p>should be equal to the distance between the face of column and the resultant beam end reaction (i.e. centroid of bolt/weld group on beam web).</p> <ul style="list-style-type: none"> • This column is only available in RISAFloor and in RISA-3D if you came in from RISAFloor. • This eccentricity is only applied to beams which connect to columns and which have the Connection Rule specified in the Connections tab of the Beams spreadsheet in RISAFloor.

3. Assign Connection Rules

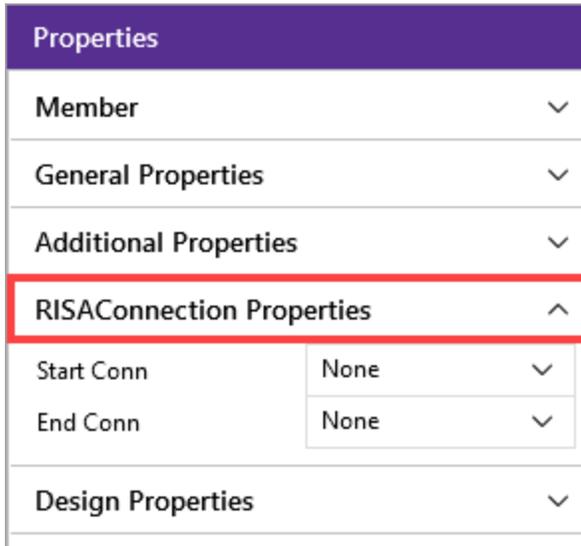
Once the **Connection Rules** are defined they must be applied to the member ends in your model.

- For Column/Beam connections the rule is assigned to the beam end but not the column.
- For Girder/Beam connections the rule is assigned to the beam end but not the girder.
- For Column/Beam/Brace connections the rule is assigned to the beam and brace(s) ends but not the column.
- For Chevron Brace connections the rule is assigned to the brace ends but not the beam.
- For Knee Brace connections the rule is assigned to the brace ends but not the supporting beam or column.
- For Splice connections, the rule is assigned to both of the Column or Beam ends.

Assigning Connection Rules in the Properties Panel

To Assign Connection Rules:

1. Click on a member and expand the **RISAConnection Properties** section in the 'Properties' panel, as shown in the following image.



- For both the **Start** and **End** of the member, click the down-arrow and choose the proper Connection Rule.

You can apply a Connection Rule to one member at a time, or select multiple members to assign a Connection Rule to multiple members.

Assigning Connection Rules in the Members Spreadsheet

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

RISAConnection									
Primary	Advanced	Hot Rolled	Cold Formed	Wood	Concrete Beam	Concrete Column	Aluminum	Stainless	RISAConnection
	Label	Shape	Start Conn	End Conn	Start Release	End Release			
1	M70	W10X12	None	None	Pinned	Pinned			
2	M71	W10X17	Col/Bm...	Col/Bm...	Fixed	Fixed			
3	M72	W10X17	None	None	Fixed	Fixed			
4	M73	W10X12	None	None	Pinned	Pinned			
5	M74	W10X17	End-Plat...	None	Fixed	Pinned			
6	M75	W10X12	None	None	Pinned	Pinned			
7	M76	W10X17	None	None	Pinned	Fixed			
8	M77	W10X17	None	None	Fixed	Fixed			
9	M78	W10X12	None	None	Pinned	Pinned			
10	M79	W10X17	None	None	Fixed	Pinned			
11	M80	W10X12	None	None	Pinned	Pinned			

RISAConnection Spreadsheet Columns

Column	Description
Label	Displays the name of the member for all Hot rolled Steel members

Column	Description
	in the model.
Shape	Allows you to view what shape is assigned to the member.
Start/End Conn	Allows you to select a rule from the 'Connection Rules' spreadsheet. You need to know which end of the member is the Start (I end) and the End (J end), which you can view graphically.
Start Release/End Release	Displays the end release for the beam. This is useful in verifying that the chosen connection for the beam is valid for its end fixity (i.e. shear connection for pinned-end beams).



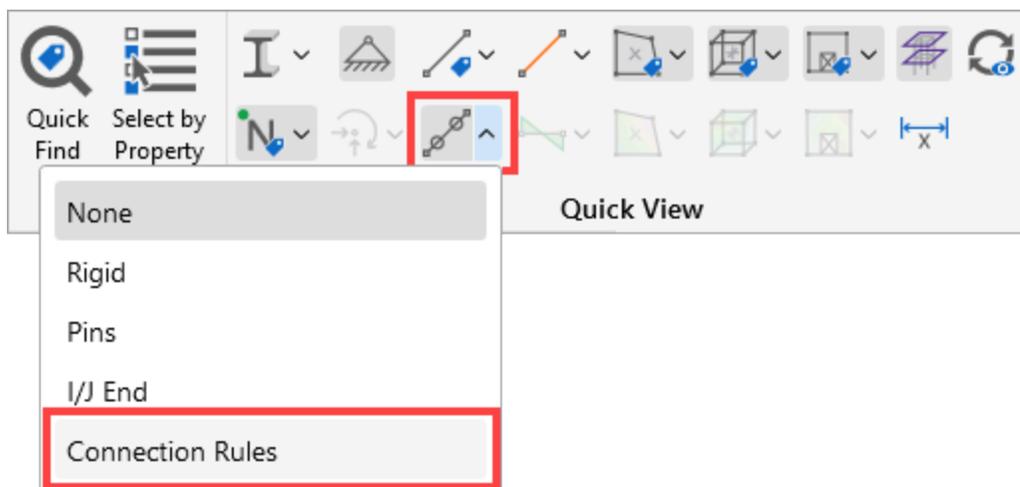
Note: Connections can only be designed for Hot Rolled Steel members, so only those members are shown on the RISAConnection tab in the spreadsheet.

Viewing Connection Rules Graphically

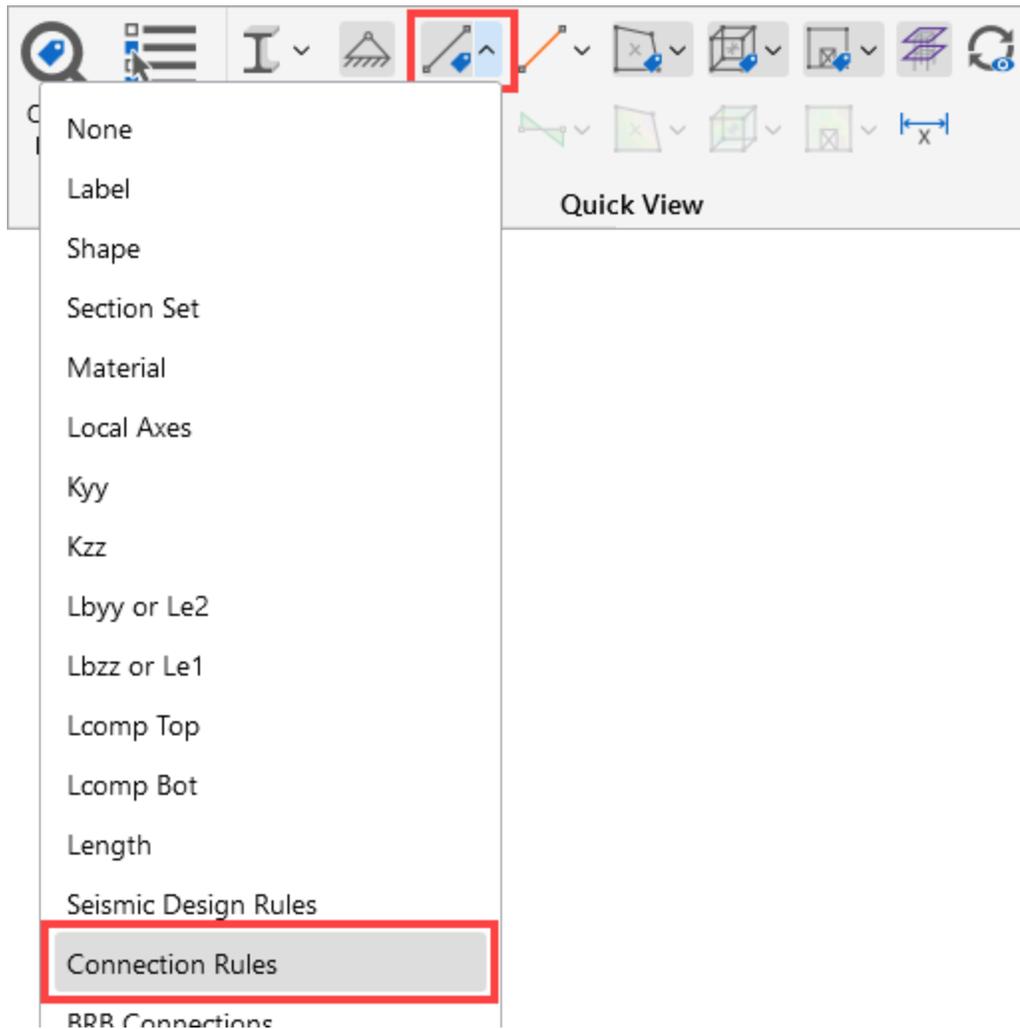
The 'Quick View' toolbar can be used to view the assigned connection rules directly on your model.

To view the assigned Connection rules using the Quick View toolbar:

1. Click the **Member Ends** icon's down-arrow on the Quick View toolbar.

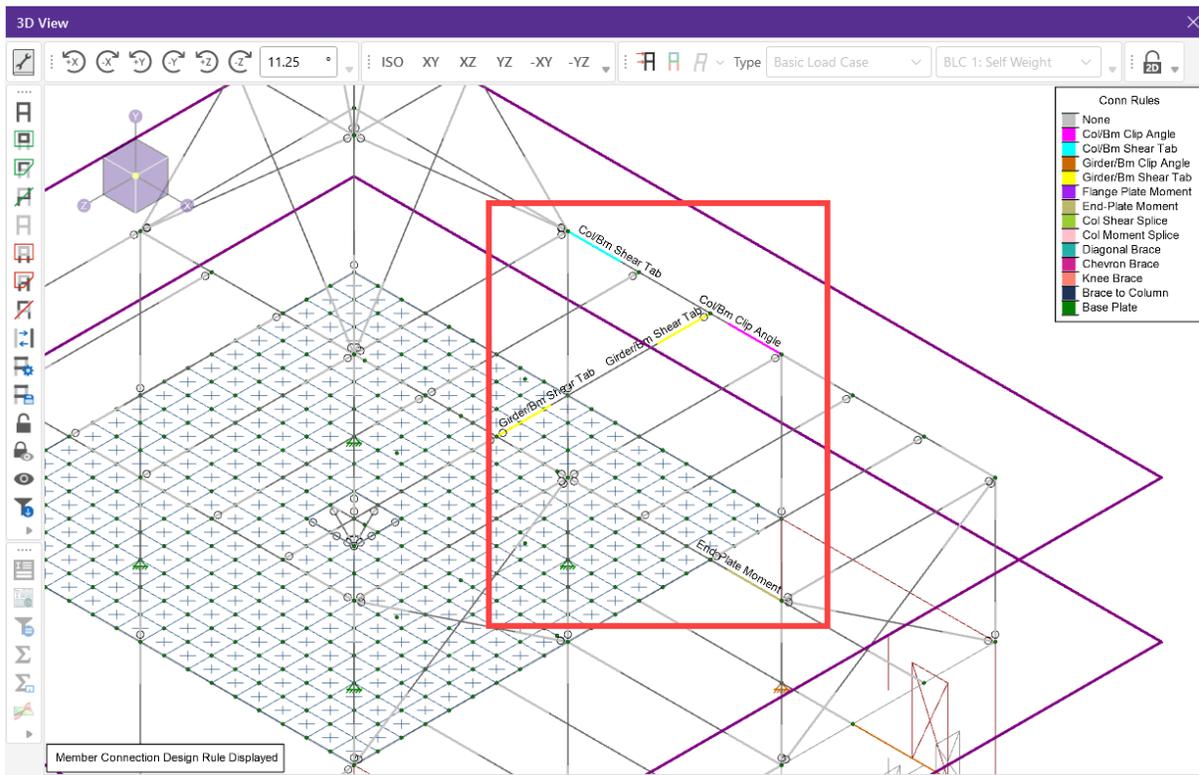


2. Choose **Connection Rules** from the list that appears.
3. Click the **Member Label** icon's down-arrow on the Quick View toolbar.



4. Choose Connection Rules from the list that appears.

The following image shows how the members with connection rules applied appear in color-coded view, along with their labels.

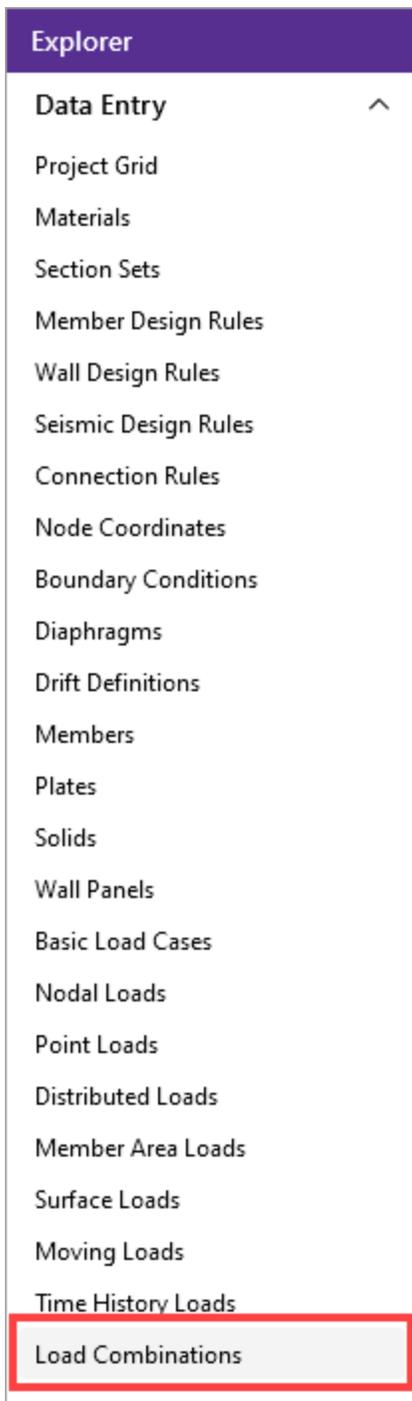


4. Assign Load Combinations

Once you have all of your Connection Rules assigned properly, you must define which load combinations you wish to use for connection design.

To assign load combinations:

1. Click on **Load Combinations** in the Explorer panel, as shown in the following image.



The Load Combinations spreadsheet opens to the 'Combinations' tab.

2. Click on the **Design** tab to view the Load Combination Design spreadsheet, as shown in the following image.

	Description	ASIF	CD	Service	Hot Rolled	Cold Formed	Wood	Concrete	Masonry	Aluminum	Stainless	Connection
1	IBC 16-1			<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
2	IBC 16-2 (a)			<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
3	IBC 16-3 (a)			<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
4	IBC 16-3 (b) (a)			<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
5	IBC 16-3 (b) (d)			<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
6	IBC 16-3 (b) (m)			<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
7	IBC 16-3 (b) (s)			<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
8	IBC 16-3 (b) (t)			<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
9	IBC 16-3 (b) (v)			<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>

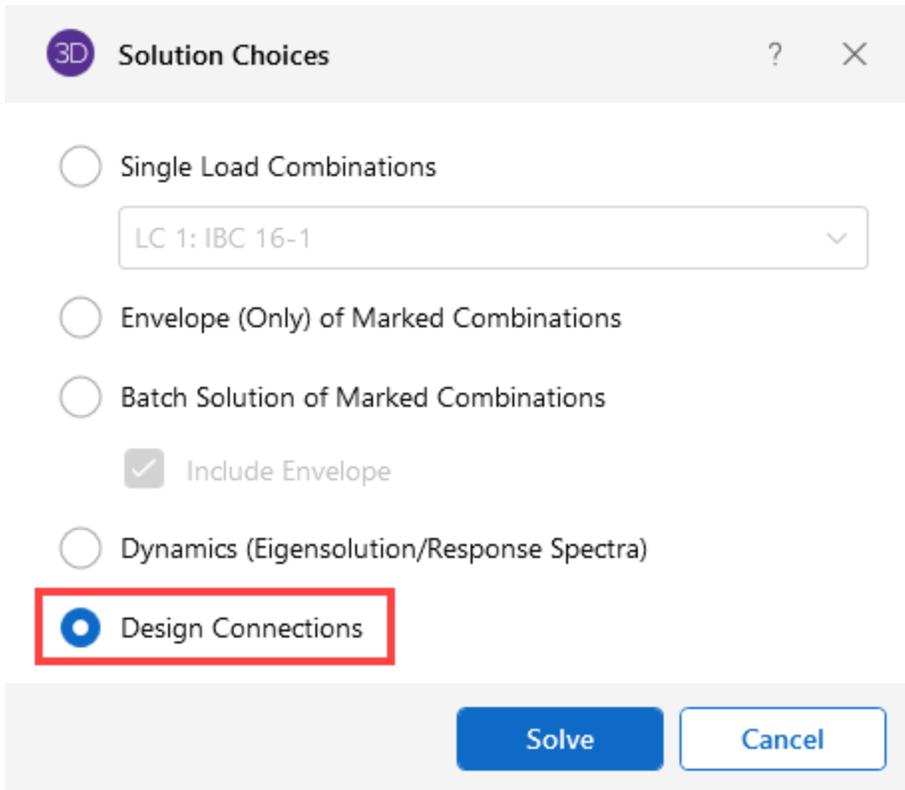
The checkboxes in the ‘Connections’ column allow you to define whether or not you want your connections designed for each LC.

There are times where the member design may be designed by LRFD methods and the connections designed by ASD methods. The ‘Connection’ check box allows for that flexibility.

5. Design Connections

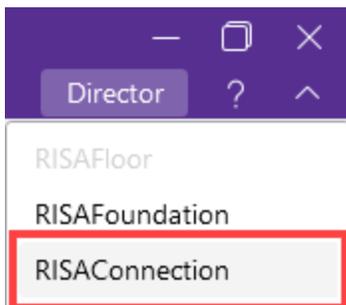
To design connections you must first have either a Single LC or Batch solution present. Once there is a solution, there are two ways to design connections.

- Choose **Solve - Design Connections**.



This option does not automatically open RISAConnection. Instead, it runs in the background, using whatever default or previous settings are in RISAConnection. Once you've done this the **Connection Results** browser is populated. To fully export into RISAConnection, use the **Director** tool, as shown next.

- Choose **Director - RISAConnection**.



This option automatically launches RISAConnection where you can modify the connection parameters, such as number of bolts, clip angle size, clearances, etc., for all the valid connections in your project (see the RISAConnection Help file for more information on how to design connections using RISAConnection). You can then solve your model in RISAConnection and then send the information back to RISA-3D to populate the **Connection Results** browser.

RISAConnection File Creation and Workflow

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

Once this RISAConnection file is created you can:

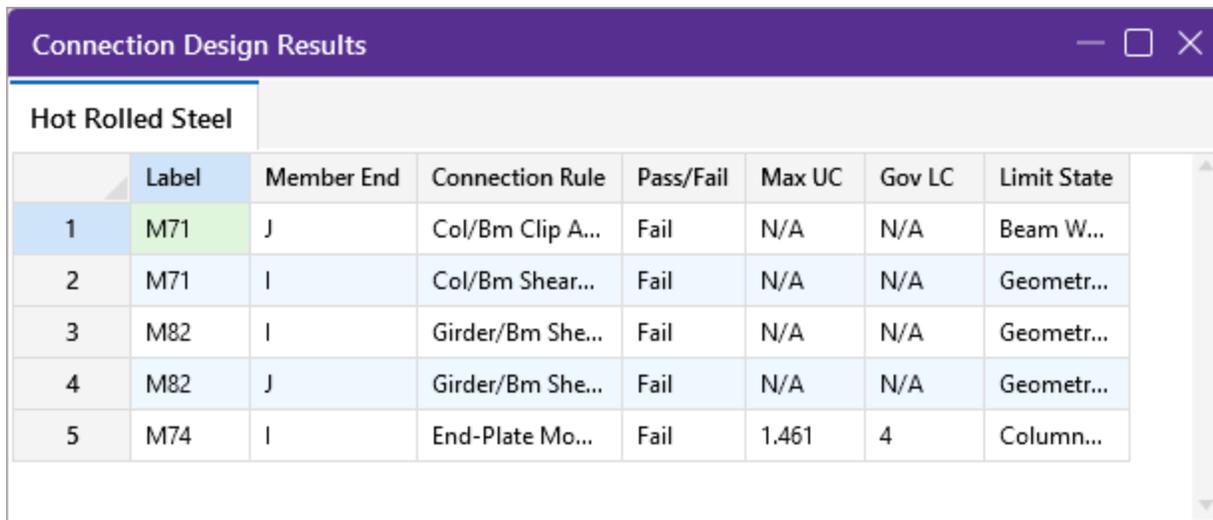
- Use the file separately to make connection changes
- Transfer the file to another machine and work on it separately
- Modify items (bolt criteria, weld criteria, connector sizes, clearances, edge distances, etc.) related to the connections
- Modify or design your connections so that they now work. Move the file back to the location where the RISA-3D model is located. Then, redesign connections in RISA-3D. Your changes are then considered.

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

6. Connection Design Results

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

Connection Results Spreadsheet



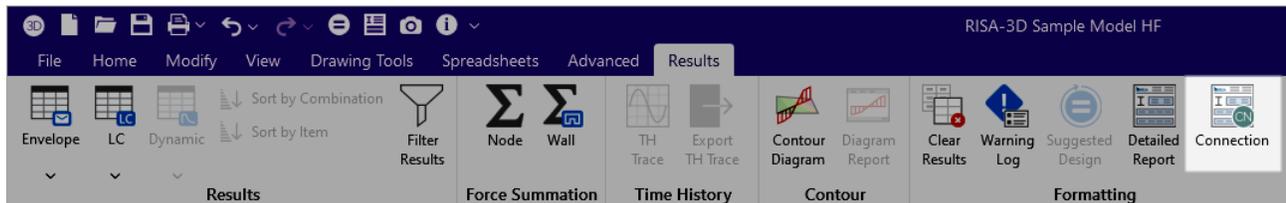
	Label	Member End	Connection Rule	Pass/Fail	Max UC	Gov LC	Limit State
1	M71	J	Col/Bm Clip A...	Fail	N/A	N/A	Beam W...
2	M71	I	Col/Bm Shear...	Fail	N/A	N/A	Geometr...
3	M82	I	Girder/Bm She...	Fail	N/A	N/A	Geometr...
4	M82	J	Girder/Bm She...	Fail	N/A	N/A	Geometr...
5	M74	I	End-Plate Mo...	Fail	1.461	4	Column...

The following table provides descriptions of the columns in the spreadsheet.

Connection Design Results for Hot Rolled Steel

Column	Description
Label / Member End	These two columns display the location in the model that this connection results corresponds to.
Connection Rule	Displays the Connection Rule assigned to this location.
Pass/Fail	Indicates whether or not the connection passes ALL connection checks.
Max UC	Displays the maximum unity check for the worst case LC.
Gov LC	Displays which LC provided the Max UC.
Limit State	Displays the governing Limit State which produced the worst case code check or failing criteria.

When viewing this spreadsheet there is a **Connection** icon in the 'Results' ribbon toolbar, as shown in the following image.



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

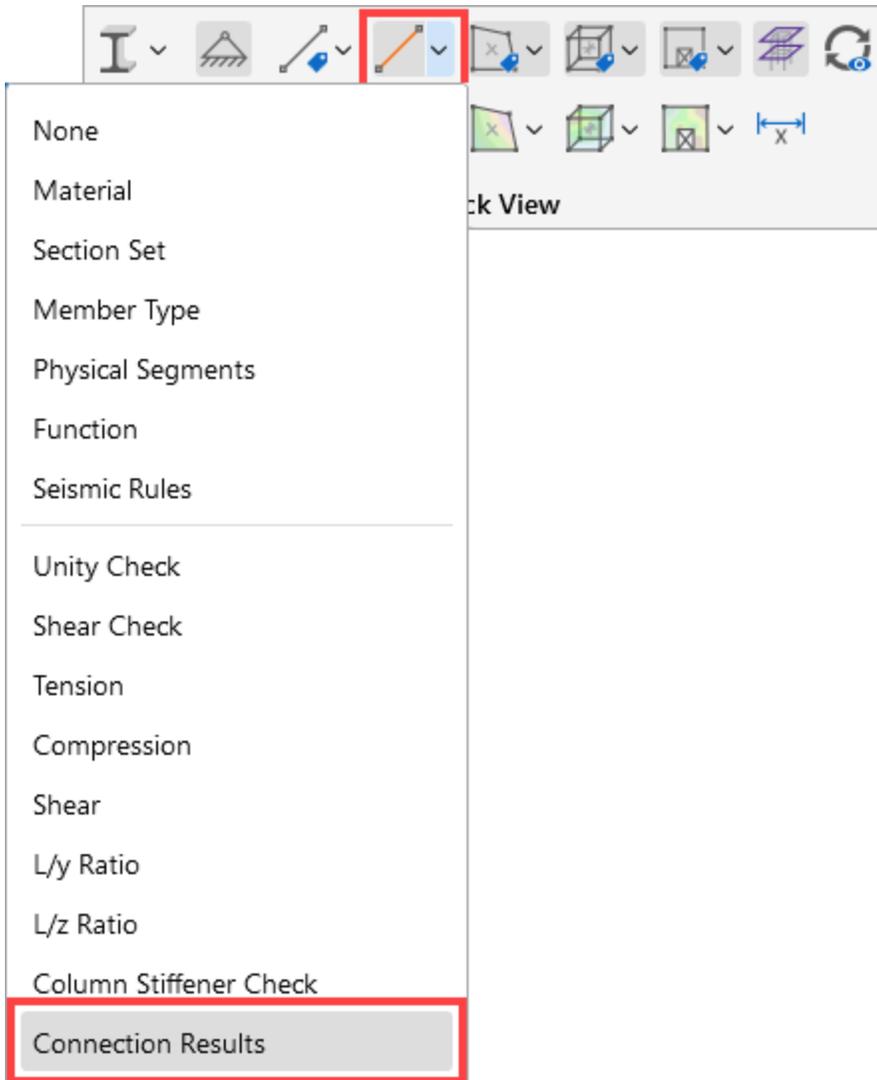


Note: The Limit State field also gives notes for any connections that were not able to be designed in RISAConnection. See [Troubleshooting](#) for more information.

Viewing Results Graphically

Use the **Quick View** toolbar to view the connection results directly on your model.

1. Click on the **Member Color** icon down-arrow to open a list of members.

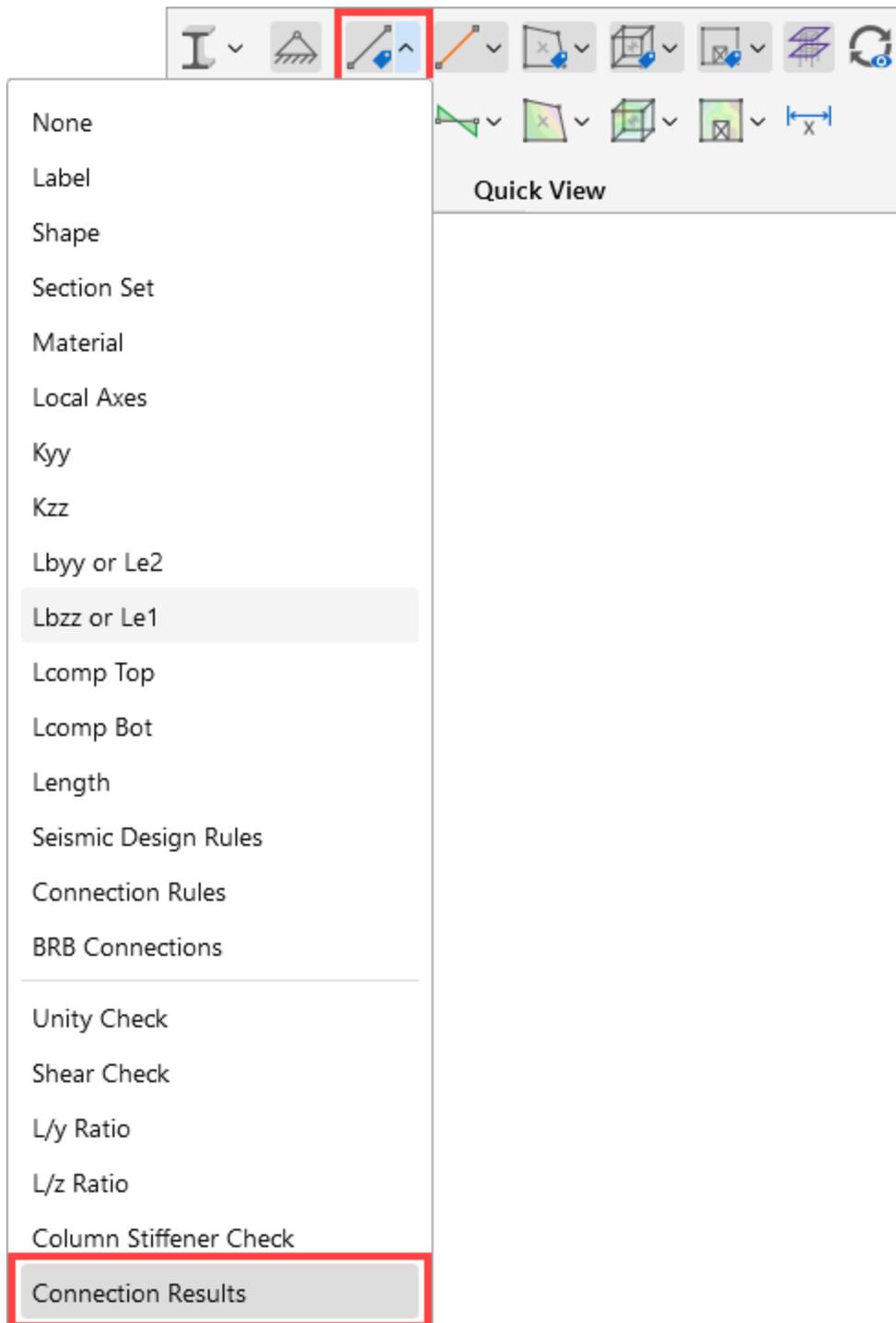


2. Choose **Connection Results** to display the results in a color-coded view (as shown in the image under step 1).

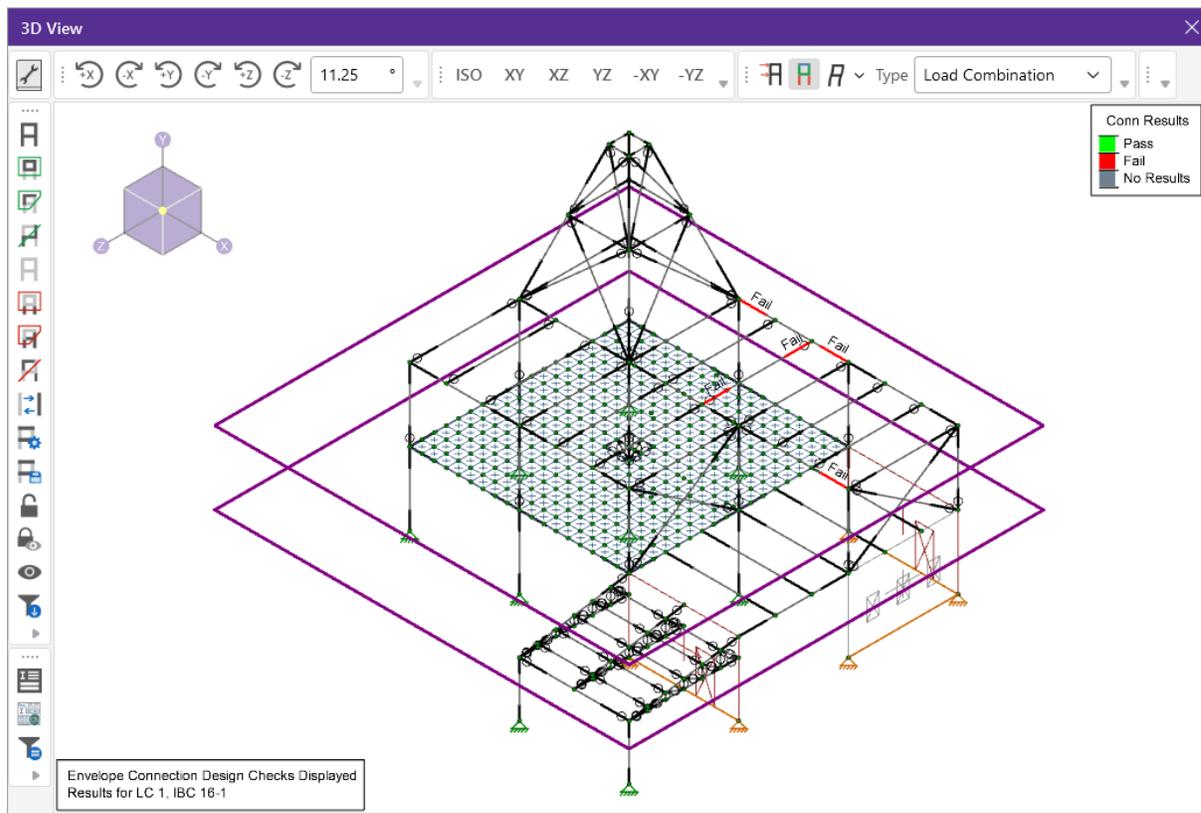
A color legend appears in the 3D View panel, and the connection members are color coded based on that legend.

(See the example after step 4 that shows both the color coded connections and their governing unity check values.)

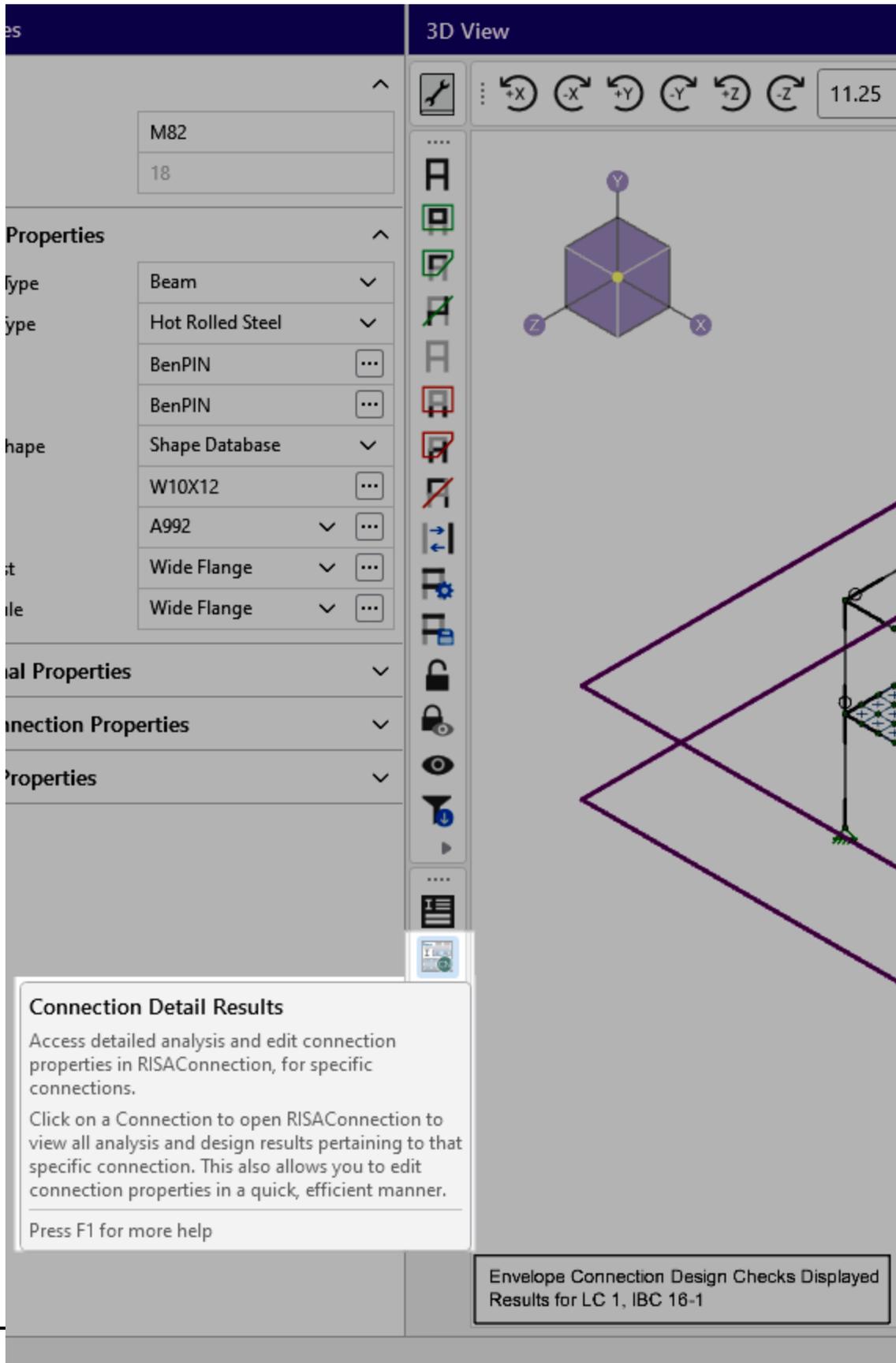
3. Click on the **Member Label** icon.



4. Choose **Connection Results**, to display the governing unity check value, as shown in the following image.



- (Optional) If you are in a graphic view and have valid connection results, you can click the **Connection Detail Results** icon on the left-hand side of the 3D View panel.



This shows a selection cursor and allows you to click on a member end that has a Connection Rule defined, and open RISAConnection to the specific connection that you've clicked on.

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

When round tripping between RISAConnection and RISAFloor/RISA-3D multiple times there are a few items to consider:

- After the first pass into RISAConnection, connection property changes in the RISAConnection file are saved. This means that if the RISAFloor/RISA-3D model is modified and the connections are re-designed you do not lose the connection information that you have modified.



Note: One exception to this is if you make a change to the Connection Rule for that member end. If that occurs, then the basic input of the connection is modified and your connection changes are deleted and replaced with the new connection's default settings.

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

Seismic Connections

With the help of the Seismic Detailing feature within RISA-3D, the program can handle a variety of column/beam moment and vertical brace connections with additional seismic level checks per the *Seismic Design Manual (AISC 341 and AISC 358)*. These connections include:

- Reduced Beam Section (RBS) Moment Connection
- Welded Unreinforced Flange-Welded Web (WUF-W) Moment Connection
- Bolted Extended End Plate Moment Connections (Unstiffened and Stiffened)
- Bolted Flange Plate (BFP) Moment Connection
- Diagonal Vertical Brace Connection.
- Chevron Vertical Brace Connection.

Rules for Export

In order to export a seismic connection, you must do the following:

1. Assign the connection with a **Seismic Design Rule** in RISA-3D.

This defines what level of ductility (OMF, IMF, SMF, OCBF, SCBF) will define your connection and, if it is a seismic moment connection, which predesignated type (RBS, WUF-W, BFP, BEEP, etc.) it shall be designed as.

Label	Frame Type	Column Ductility	Column Overstrength	Beam Ductility	Connection	Beam Overstrength	Z Factor	Hinge Location [in]	Brace Ductility	Brace Overstrength	KL/r
1	OCBF	Minimal	<input checked="" type="checkbox"/>	Minimal	Other/None	<input type="checkbox"/>			Minimal	<input type="checkbox"/>	<input type="checkbox"/>
2	SCBF	High	<input checked="" type="checkbox"/>	High	Other/None	<input checked="" type="checkbox"/>			High	<input type="checkbox"/>	<input checked="" type="checkbox"/>
3	OMF	Minimal	<input checked="" type="checkbox"/>	Minimal	BUEEP	<input type="checkbox"/>		12	N/A	<input type="checkbox"/>	<input type="checkbox"/>
4	IMF	Moderate	<input checked="" type="checkbox"/>	Moderate	BFP	<input type="checkbox"/>		12	N/A	<input type="checkbox"/>	<input type="checkbox"/>
5	SMF-RBS	High	<input checked="" type="checkbox"/>	High	RBS	<input type="checkbox"/>	0.685	14.625	N/A	<input type="checkbox"/>	<input type="checkbox"/>
6	SMF-K...	High	<input checked="" type="checkbox"/>	High	KBB-B	<input type="checkbox"/>		12	N/A	<input type="checkbox"/>	<input type="checkbox"/>
7	SMF-K...	High	<input checked="" type="checkbox"/>	High	KBB-W	<input type="checkbox"/>		12	N/A	<input type="checkbox"/>	<input type="checkbox"/>
8	SMF-B...	High	<input checked="" type="checkbox"/>	High	BSEEP	<input type="checkbox"/>		12	N/A	<input type="checkbox"/>	<input type="checkbox"/>
9	SMF-W...	High	<input checked="" type="checkbox"/>	High	WUF-W	<input type="checkbox"/>			N/A	<input type="checkbox"/>	<input type="checkbox"/>

2. Assign the connection with a **Seismic Moment Connection Rule**, if the connection is a moment frame connection.

Label	Conn Type	Type	Beam Conn	Col/Girder Conn
1	Seismic Moment Connection	Moment	Column/Beam Seismic Moment	N/A

3. Assign the connection with the appropriate **Diagonal Brace Seismic** or **Chevron Brace Seismic** connection type, if the connection is a braced frame connection.

Label	Conn Type	Type	Beam Conn	Col/Girder Conn
1	Seismic Diagonal	Brace	Diagonal Brace Seismic	N/A
2	Seismic Chevron	Brace	Chevron Brace Seismic	N/A

Limitations

- Seismic moment connections must be connected to the flange of the column.
- The model must be solved in both RISAFloor and RISA-3D in order to send the lateral load combination results to RISAConnection.
- Seismic moment splice connections are not currently supported.

- The program does not put any limitations on which RISA-3D load combinations (overstrength or not) are solved and sent to RISAConnection. Therefore, we assume that the user is selecting the correct combinations to include for connection design. Once in RISAConnection, you will be able to scroll through the results from each load combination.

Tapered Wide Flange Connections

Several connections (listed below) can be designed for tapered wide flange members as well as for prismatic wide flanges. RISA-3D requires that the connection occurs at the ends of the members (no mid span connections). This is because RISAConnection imports the section as a prismatic (non tapered) section with the cross-sectional dimensions equal to those of the selected end of the tapered member in RISA-3D.

Available Tapered WF Member Connections

- Beam members in the **Beam Shear Tab Splice Connection**
- Column members in the **Column Shear Tab Splice Connection**
- Column and Beam in the **Column/Beam Clip Angle Shear Connection**
- Column and Beam in the **Column/Beam End Plate Shear Connection**
- Column and Beam in the **Column/Beam Shear Tab Shear Connection**
- Beam member only in the **Girder/Beam Clip Angle Shear Connection**
- Beam member only in the **Girder/Beam End Plate Shear Connection**
- Beam member only in the **Girder/Beam Shear Tab Shear Connection**
- Beam members in the **Beam Direct Weld Moment Splice Connection**
- Beam members in the **Beam Extended End Plate Moment Splice Connection**
- Beam members in the **Beam Moment Plate Moment Splice Connection**
- Column members in the **Column Direct Weld Moment Splice Connection**
- Column members in the **Column Extended End Plate Moment Splice Connection**
- Column members in the **Column Moment Plate Moment Splice Connection**
- Column and Beam in the **Column/Beam Direct Weld Moment Connection**
- Column and Beam in the **Column/Beam End Plate Moment Connection**
- Column and Beam in the **Column/Beam Flange Plate Moment Connection**

Limitations

- Currently this is limited to just doubly-symmetric tapered members (flanges must be equal). In the future, we plan on also allowing singly-symmetric tapered WF sections with unequal flanges.

- Tapered WF connections are only considered if they occur at the member ends. (For example, connections applied to beams who are supported by girders or columns along their mid span are not designed).
- A custom Tapered WF shape must be entered and saved permanently to the shape database through RISA-3D, before it can be accessed for use in RISACONNECTION.

Troubleshoot RISACONNECTION Integration

If you have difficulty getting your RISA-3D or RISAFloor model to integrate into RISACONNECTION, it is possible that you have defined connection or member information incorrectly, or there may be a program limitation which prevents the integration. Please see the following for assumptions, modeling rules, and possible error messages.

Moment Connection Considerations

The following moment connections are currently supported in RISACONNECTION:

- Beam to column flange plate moment connections for wide flange or HSS tube columns and wide flange beams.
- Beam to column direct weld end plate moment connections for wide flange or HSS tube columns and wide flange beams.
- Beam to column extended end plate moment connections for wide flange members.
- Beam to column partially extended end plate moment connections for wide flange members.
- Beam to column flush end plate moment connections for wide flange members.
- Column flange plate moment splice.
- Beam flange plate moment splice.
- Column direct weld moment splice.
- Beam direct weld moment splice.
- Column extended end plate moment splice.
- Beam extended end plate moment splice.

Continuous Beam over Column Connections

In order to properly export a continuous beam over column connection to RISACONNECTION you must obey the following rules:

- A continuous beam over connection rule must exist in the **Connection Rules** spreadsheet.
- That connection rule must be applied at the top of the column where a continuous beam passes over it.
- No other column members from above can frame into the connection.

- Connection rule must be applied to the top floor of the structure.
- The column member must be a prismatic Wide Flange, a HSS Tube, or a HSS Pipe.
- The beam member must be a prismatic Wide Flange.

Shear Connection Considerations

The following shear connections are currently supported in RISAConnection:

- Single or double clip-angle connections for wide flanges
- Shear tab connections for wide flanges
- End plate connections for wide flanges
- Single or double clip-angle connections for channel beams
- Shear tab connections for channel beams
- Double-sided double clip angle connections to the column or girder web.
 - Beams on either side of the web must be within 3 inches (parallel) to be considered a double-sided connection.

Vertical Brace Connections

The following vertical brace connections are currently supported in RISAConnection:

Diagonal Brace Connections

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

- A Diagonal Brace connection rule must exist in the **Connection Rules** spreadsheet.
 - That connection rule must be applied to the end of the VBrace which connects to the gusset
 - That connection rule must also be applied to the end of the beam which connects to the gusset and to the column.
 - There may be one or two VBrace members per connection. If there are two VBrace members, they must frame in from above and below the beam respectively
 - There must be one beam member on the same side as the VBrace per connection.
 - The VBrace(s) may be offset from the beam / column work point. If the offset is along the length of the column then each brace cannot be more than 18 inches from the work point. If the offset is along the length of the beam then the two VBrace members must share a common work point. This eccentricity will be accounted for in the design of the connection. But, the connection design will NOT consider any of the EBF (Eccentrically Braced Frame) requirements of AISC seismic provisions.
 - The beam must be horizontal, and the VBrace cannot have an angle of less than 10 degrees with respect to the Beam or the column.
-

- The column must be a prismatic Wide Flange, a HSS Tube, or a HSS Pipe.
- The beam must be a prismatic Wide Flange.
- The brace(s) must be a single angle, double angle, channel, tube, pipe, WT, or prismatic Wide Flange shape.

Chevron Brace connections

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

- A Chevron Brace connection rule must exist in the **Connection Rules** spreadsheet.
- That connection rule must be applied to the ends of both VBraces which connect to the gusset.
- There must be exactly two VBrace members per connection. They must frame in from the same side (above or below) of the beam.
- The VBraces need not share a common node. However, any eccentricity that exists will not be accounted for in the connection design.
- The beam and VBraces must exist in the same plane.
- The beam must be horizontal, and the VBraces cannot have an angle of less than 10 degree with respect to the beams or the vertical plane.
- The beam must be a prismatic Wide Flange, a HSS Tube, or a HSS Pipe.
 - Note: Currently HSS beams on *seismic* chevron braces are not supported.
- The brace(s) must be a single angle, double angle, channel, tube, pipe, or WT, or prismatic Wide Flange shape.

Knee Brace Connections

In order to properly export a Knee (or Kicker) brace connection to RISAConnection you must obey the following rules:

- A Knee Brace connection rule must exist in the **Connection Rules** spreadsheet.
 - That connection rule must be applied to the end of the VBrace which connects to the gusset. This can either be at the end that connects to a beam or the end that connects to a column.
 - The VBrace cannot have an angle of less than 10 degrees with respect to the Beam or the column.
 - The supporting member (beam or column) must be a prismatic Wide Flange, a HSS Tube, or a HSS Pipe.
 - The Vbrace must be a HSS Tube, HSS Pipe, Channel, WT, Double Angle, or Single Angle
-

shape.

- The Vbrace and its supporting member (beam or column) must be in the same plane.

Splice Connections

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

- For beam to beam column to column splices, both members must be oriented in the same direction.
- You must apply your splice Connection Rule to both sides of the splice.

Base Plate Connections

RISAConnection can design single column base plate connections for shear and/or moment forces. Keep in mind these items when creating your model:

- A Base plate connection rule must exist in the **Connection Rules** spreadsheet.
- The column must be a Wide Flange, a HSS Tube, or a HSS Pipe.
- Base plate connection rules may only be assigned to the bottom of a column member.

Brace to Base Plate Connections

RISAConnection can design vertical brace into a column base plate connections for shear and/or moment forces. Keep in mind these items when creating your model:

- A Brace to Base Plate connection rule must exist in the **Connection Rules** spreadsheet.
- The column must be a Wide Flange, a HSS Tube, or a HSS Pipe.
- The brace must be a HSS Tube, HSS Pipe, Channel, WT, Double Angle, or Single Angle.
- Base plate connection rules must be assigned to the bottom of a column member and the end of the brace that connects to the column.
- The brace end node may be offset along the column center-line up to 24 inches away from the end of the column.

Truss HSS T-Connections

RISAConnection can design HSS T-Connections that have been exported from your RISA-3D model. Keep in mind these items when assigning the connection rule in RISA-3D:

- A Truss HSS T-Connection rule must exist in the **Connection Rules** spreadsheet.
- The branch member (vertical member) must be an HSS Tube section.
- The chord member (horizontal member) must be an HSS Tube section.

- The branch and chord members may be assigned as one of the following Shape Type Combinations:
 - Beam supported by Column
 - Column supported by Beam
 - VBrace supported by Column
 - VBrace supported by Beam
 - HBrace supported by Column
 - HBrace supported by Beam
- The chord member must be continuous under the branch. If the chord member is broken into two members it will not export.
- The branch and chord member must be in the same plane.
- The branch and chord member must be perpendicular to one another.

Integration Error Messages

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

The possible error messages are:

Integration Error Messages

Error Message	Cause(s)
Connection not supported	This message occurs when there is a connection configuration that RISAConnection can not design. An example would be a horizontal brace connection where brace members frame into a beam web.
Invalid or missing supporting connection members	This message occurs if the member(s) that a member is framing into is not of the proper material, shape, member type, etc. Also check that the Member Function is the same (lateral or gravity) for all members for a particular connection.
Invalid connection member material	This message occurs if the member's material is not hot-rolled steel.
Invalid connection member shape	This message occurs if the member's shape is not one of the supported shape types for that connection type.
Invalid member/beam slope (more than 15 degrees)	This message occurs if the member being

Error Message	Cause(s)
	connected has an invalid slope. Currently, beam/column connection design is only for orthogonal connections. If the angle between members is greater than 15 degrees from orthogonal, then this message appears.
Invalid member/brace rotation (more than 15 degrees)	This message occurs if the member being connected has an invalid rotation. Currently, members can only be designed at certain orientations. For example; weak axis beam to column design is not currently supported, so this message appears.
Invalid member skew (more than 15 degrees)	This message occurs if the member being connected has an invalid skew. Currently, beam/column connection design is only for orthogonal connections. If the angle skew between members is greater than 15 degrees from orthogonal, then this message appears.
Connections on skewed members are not supported	This is a similar message to the invalid member skew message.
Invalid vertical brace connection	This message occurs if there was not a valid configuration. If there is not a valid column, beam and vertical brace intersection (at proper orientation, member type, member material, etc) then you may see this message. Note that for integration you must assign BOTH the beam and the brace to the vertical brace Connection Rule.
Brace angle is invalid	For a corner brace or chevron vertical brace connection, the brace angle with the column and beam must be > 10 degrees.
Members have different/missing connection rules	If there is a vertical brace connection and the beam and brace have different Connection Rules, this message appears.
Beam must be a WF in a Chevron brace connection	This message occurs if you have assigned a chevron brace connection type to a tapered WF beam member.
Beam and Column must be WF in Diagonal brace connections	This message will come up if you have assigned a diagonal brace connection type to a tapered WF beam member.
Invalid code. HR Steel and Connection design codes must be the same	This message occurs if the design codes do not match on the Model Settings Codes tab. Seismic design requires that these match.
Invalid design method. IMF/SMF connections must be design with LRFD	This message occurs if an IMF or SMF seismic

Error Message	Cause(s)
	connection is applied to a model that is assigned the AISC 360-10 (14th edition) ASD design code. Section 1.3 of the AISC 358-10 design code requires that seismic connection design must be done per LRFD.
Incomplete definition of connection rule	This message occurs if you have not selected an appropriate "Beam Conn" or "Col/Girder Conn" input in the Connection Rules spreadsheet. To correct this, simply go back to Connection Rules and select the connection type (Bolted or Welded).
Beam must be drawn continuously over the column	This message occurs when utilizing a Continuous Beam over Column connection type, and the beam framing is set to terminate at the column without spanning the entire width of the column. This error serves as a reminder to ensure that the beam is consistently drawn in a continuous manner over the entire top surface of the column.

Connection Types Not Currently Supported

- **Skewed or sloped connections.** If the skew or slope is less than 15 degrees, the program transfers the connection to RISAConnection as an orthogonal connection. If the skew or slope is greater than 15 degrees, the connection is not transferred.
- **Horizontal brace connections.**

RISAFloor & RISA-3D Connection Design Interaction

RISAFloor and RISA-3D both have the ability to have their connections designed with RISAConnection. Thus, there are three possible scenarios for connections between RISAFloor, RISA-3D and RISAConnection:

- Scenario 1: Connections that are only in RISAFloor (Gravity connections in a combined RISAFloor/RISA-3D model).
- Scenario 2: Connections that are in both RISAFloor and RISA-3D (Lateral connections in a combined RISAFloor/RISA-3D model).
- Scenario 3: Connections that are only in RISA-3D. This scenario does not involve RISAFloor, so it is not discussed here.

Scenario 1: RISAFloor Gravity Member Connections

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

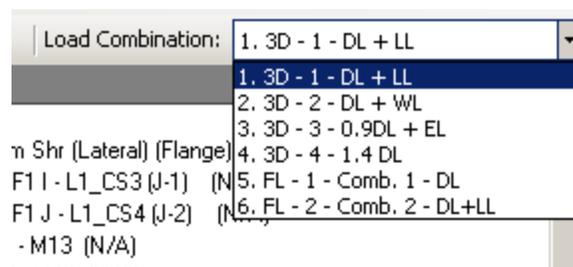
The difference is when you use the Director to go into RISA-3D and then into RISAConnection, the Gravity connections are still in the RISAConnection project. The .rcn file contains BOTH RISAFloor AND RISA-3D connections. Therefore, when entering RISAConnection from RISA-3D, you will see more connections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

Scenario 2: RISAFloor Lateral Member Connections

Additional items to consider are [Load Combinations](#) and [Naming Convention](#), as explained in the following sections.

Load Combinations

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



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



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

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

Connection Design Results							
Hot Rolled Steel							
	Label	Member End	Connection Rule	Pass/Fail	Max UC	Gov LC	Limit State
1	F1_M11 (M11)	I	End-Plate Mo...	Pass	0.151	3	Bolt Shea...
2	F1_CS9_L2 (M7)	I	None	Fail	N/A	N/A	Geometr...
3	F1_M81 (M81)	I	Col/Bm Doubl...	Pass	0.186	2(FL)	Bolt Bear...
4	F1_M87 (M87)	I	Col/Bm Shear...	Pass	0.331	11	Bolt Shea...
5	F3_M20 (M20)	I	Col/Bm Shear...	Pass	0.41	8	Bolt Shea...
6	F1_M60 (M60)	I	Flange Plate M...	Fail	N/A	N/A	Geometr...
7	F1_CS1_L1 (M2)	I	Base Plate	Pass	0.701	9(FL)	Plate Fle...
8	F1_CS4_L1 (M5)	I	Base Plate	Fail	N/A	N/A	Flange W...
9	F1_CS8_L1 (M10)	I	Base Plate	Pass	0.369	3	Plate Fle...
10	F1_CS12_L1 (M15)	I	Base Plate	Pass	0.574	11	Plate Fle...
11	F1_CS17_L1 (M21)	I	Base Plate	Pass	0.57	5(FL)	Plate Fle...
12	F1_M77 (M77)	I	Col/Bm Doubl...	Fail	N/A	N/A	Geometr...
13	F3_J121 (J121)	I	Girder/Bm Do...	Fail	N/A	N/A	Geometr...

Naming Convention

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

RISACONNECTION ALWAYS uses the RISAFloor naming convention. This may be a little confusing. For easier comprehension, the Connection Results spreadsheet in RISA-3D displays BOTH the RISAFloor and RISA-3D labels.

Connection Design Results							
Hot Rolled Steel							
	Label	Member End	Connection Rule	Pass/Fail	Max UC	Gov LC	Limit State
1	F1_M11 (M11)	I	End-Plate Mo...	Pass	0.151	3	Bolt Shea...
2	F1_CS9_L2 (M7)	I	None	Fail	N/A	N/A	Geometr...
3	F1_M81 (M81)	I	Col/Bm Doubl...	Pass	0.186	2(FL)	Bolt Bear...
4	F1_M87 (M87)	I	Col/Bm Shear...	Pass	0.331	11	Bolt Shea...

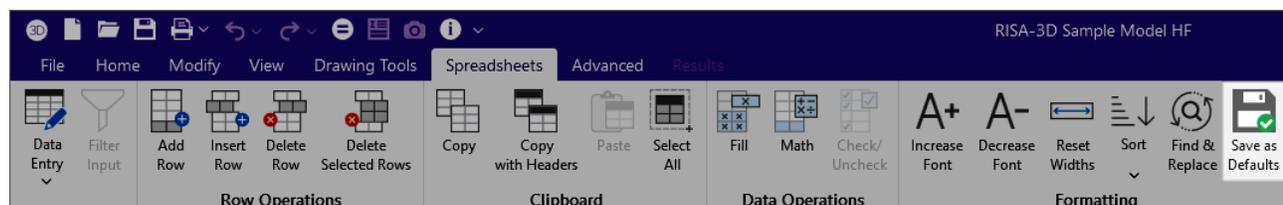
In this image the F1_M and F1_C labels are the RISA-3D labels. The labels in parentheses are the RISAFloor labels.

Customizing RISA-3D

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

Save as Defaults

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



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

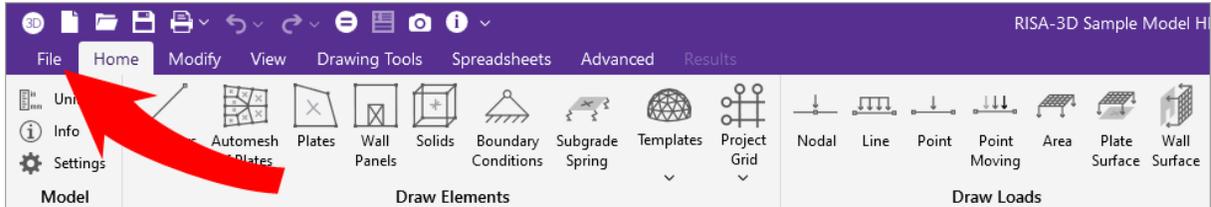
Once you create a new file you may redefine any of the default data and settings for that particular file so the **Save as Defaults** feature may be used to give you a good starting point for new files but won't hold you to those settings.

Application Settings (Preferences)

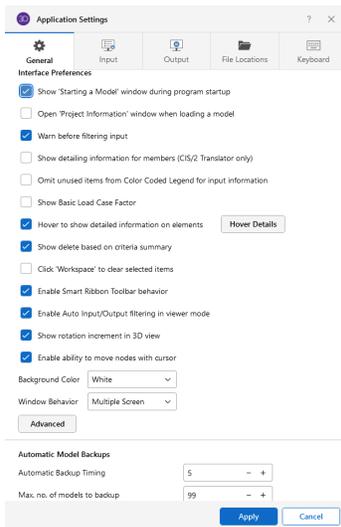
The Application Settings window allows you to choose your program preferences. You can set general preferences such as showing the "Starting a Model" window when you open RISA-3D, opening the "Project Information" window when loading a model, showing a warning before filtering input, and many other options for setting up your application to best suit your liking. For options may be accessed by clicking on **File** in the ribbon toolbar and selecting **Application Settings**.

To open the Application Settings window:

1. Click on **File** in the ribbon toolbar, as shown in the following image.



The Application Settings window opens to the General tab.

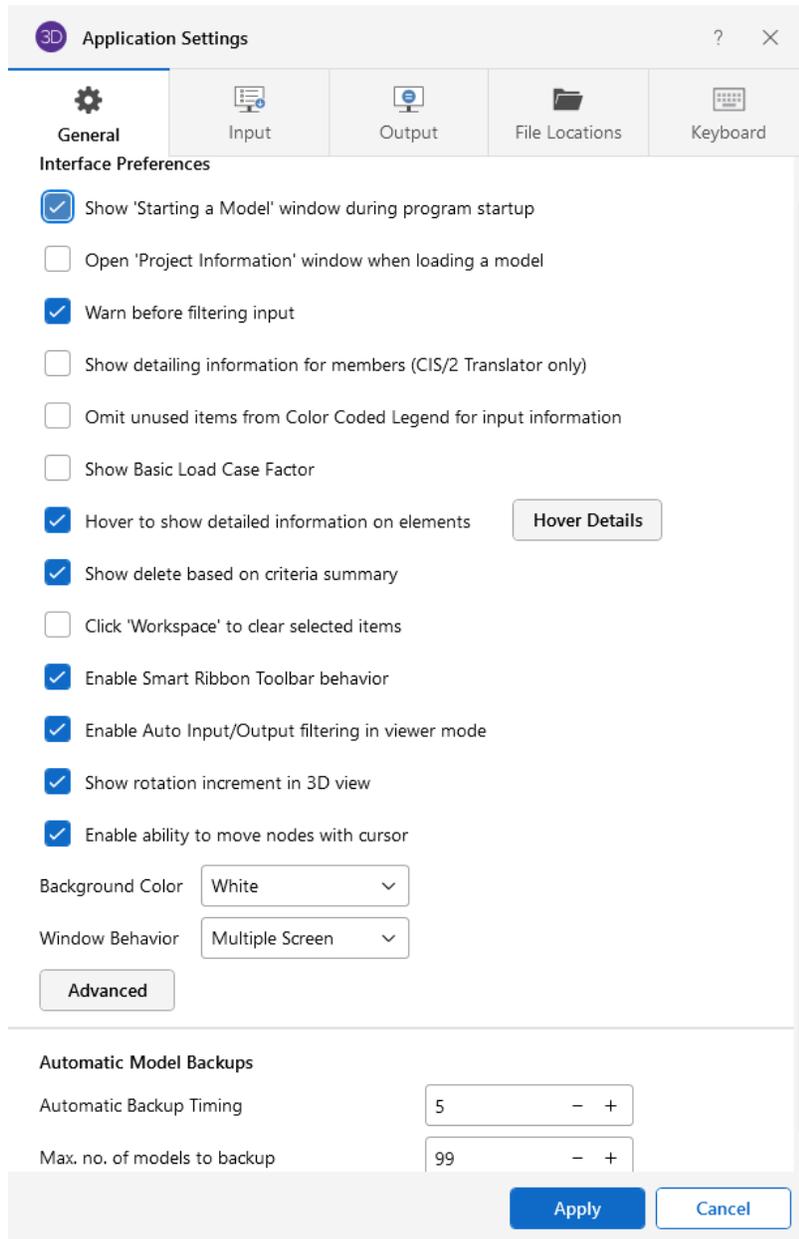


2. (Optional) Click on the tab associated with the preferences you want to set.

Many of the Application Settings themselves are self-explanatory. However, the following sections provide more details on the available preference settings.

General Application Settings

The general Application Settings are straightforward. For help on an item click **?** for more information. You can set the backup timing within the general section. See [Automatic Backup](#) to learn about the backup capabilities of RISA-3D. The **Reset all Program Defaults** button in the [Advanced General Application Settings](#) will clear all of the Application Settings that you have set on any of the tabs.



The following table provides descriptions for the settings.

General Application Settings

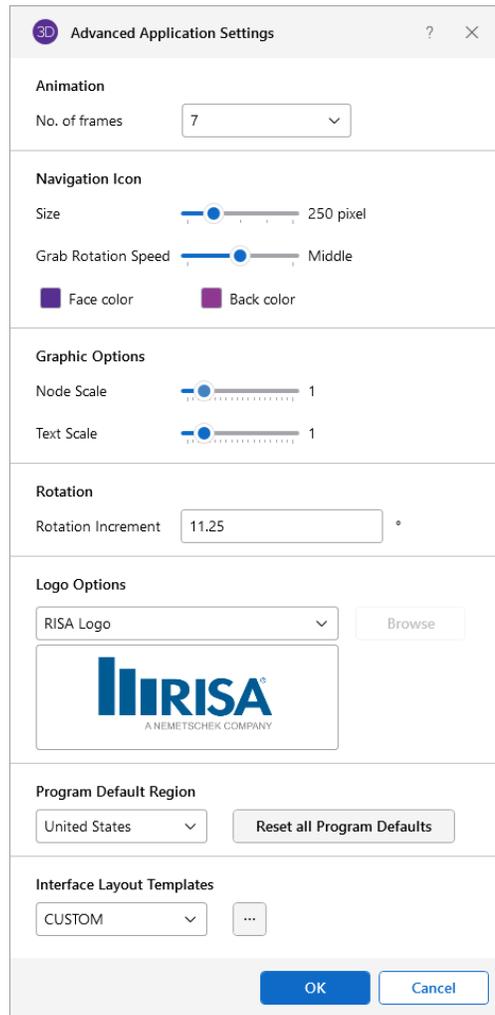
Setting	Description
Show "Starting a Model" Panel when starting a new model	The New Model Dialog will be displayed when opening the program or selecting 'New File' from the File Menu.

Setting	Description
Open Project Information window when loading model	Displays the Project Information settings automatically after loading a file.
Warn before Filtering Input	With this option checked, the program will always warn you before you filter input.
Show Detailing Info for members (CIS/2 Translator only)	Choose to display the Detailing information in the spreadsheets and Properties panel. This information is only applicable for the CIS/2 translator.
Omit unused items from Color Coded Legend for input information	This option will clean up the Color Coded Legend to only display items used in the model.
Show Basic Load Case Factor	This option will allow you to factor your Basic Load Cases. A new column will display in the Basic Load Cases spreadsheet called Basic Load Case Factor. The loads in a particular Basic Load Case will be factored by the value entered into this column.
Hover to show detailed information on elements	Click the Hover Details button to customize the information shown when hovering over an element.
Show Delete Based on Criteria Summary	This provides an option to display a summary of all items that were deleted after deleting base on a specific criteria.
Click Workspace to Clear Selected Items	This option allows you to click the white space in the 3D View to clear your current selection.
Enable Smart Ribbon Toolbar behavior	The Smart Ribbon Toolbar behavior will predict which tab of the Ribbon Toolbar to open based on the actions performed in the program.
Enable Auto Input/Output Filtering in Viewer Mode	This option will automatically filter both input and output spreadsheets based on visible items in Viewer Mode.
Show Rotation Increment in 3D View	This option will show the degree increment field in the Incremental Rotate bin.

Setting	Description
Enable ability to move nodes with cursor	This provides an option to move nodes with cursor by clicking them
Background Color	Choose which color to make the background color in the user interface.
Window Behavior	Choose your window behavior option. For more information, see Window Behavior .
Automatic Backup Timing	Automatic backup of the currently open model occurs at the specified interval. Each backup overwrites the previous one, such that only one backup of a given model exists at any time.
Max. no. of models to backup	This controls how many models are kept in the backup folder. Once the folder becomes "full" according to this setting, the oldest backups will be deleted automatically. Setting this to zero turns off automatic backups.

Advanced Application Settings

Additional program options can be found by clicking the **Advanced** button in the Application Settings window, under the General tab.



The following table provides descriptions for the settings.

Advanced Application Settings

Setting	Description
Number of Animation Frames	This option allows you to specify how many frames are used in the animation to move from the undeflected state of the model to the fully deflected state.
Size	This option controls the size of the Global Axes.
Grab Rotation Coeff	This option controls the rate of rotation when rotating the model with mouse cursor. A small grab rotation coefficient will cause the model to rotate less as your

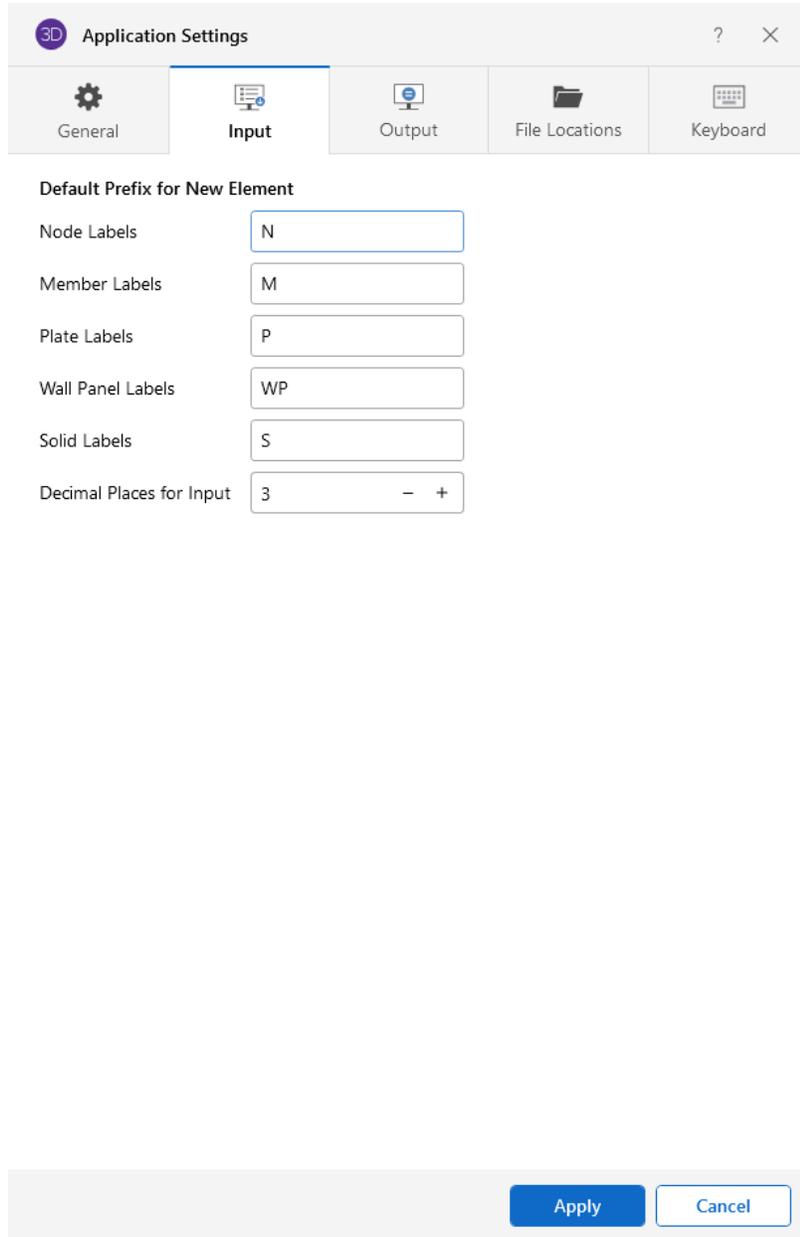
Setting	Description
	rotate it with your mouse. A large grab rotation coefficient will cause the model to rotate at a higher rate as you rotate using your cursor.
Face Color	This option controls the color of the front of the Global Axes icon.
Back Color	This option controls the color of the back face of the Global Axes icon.
Node Scale	This setting controls the size of the nodes displayed in the 3D View.
Text Scale	This setting controls the size of the text displayed in the 3D View.
Rotation	This setting controls the rotational increment for the Click to Rotate function
Interface Layout Template	This setting controls the layout of the Bins in the 3D View.
No Logo	This option will not include a logo on your printed output.
RISA Logo	This option will place the RISA logo on all printed output.
Custom Logo	This option allows you to add a custom company logo to your printed output.
Default Region	Choose the default region for your projects.
Reset all Program Defaults	Choose this to reset all of the Application Settings that you have a set on any of the tabs.

Input Application Settings

Here you can specify the number of decimal places that are displayed. The one exception is the **Node Coordinates**. RISA-3D maintains the coordinates to 15 significant figures and the exact value is always displayed. You may also use the **Round off Coordinates** button in the **Modify** tab to round off node coordinates.



If you wish to use a prefix with your node, member, and plate labels, such as "N" with nodes, you can specify the default prefix. The prefixes may be changed as you build your model.



The following table provides descriptions for the settings.

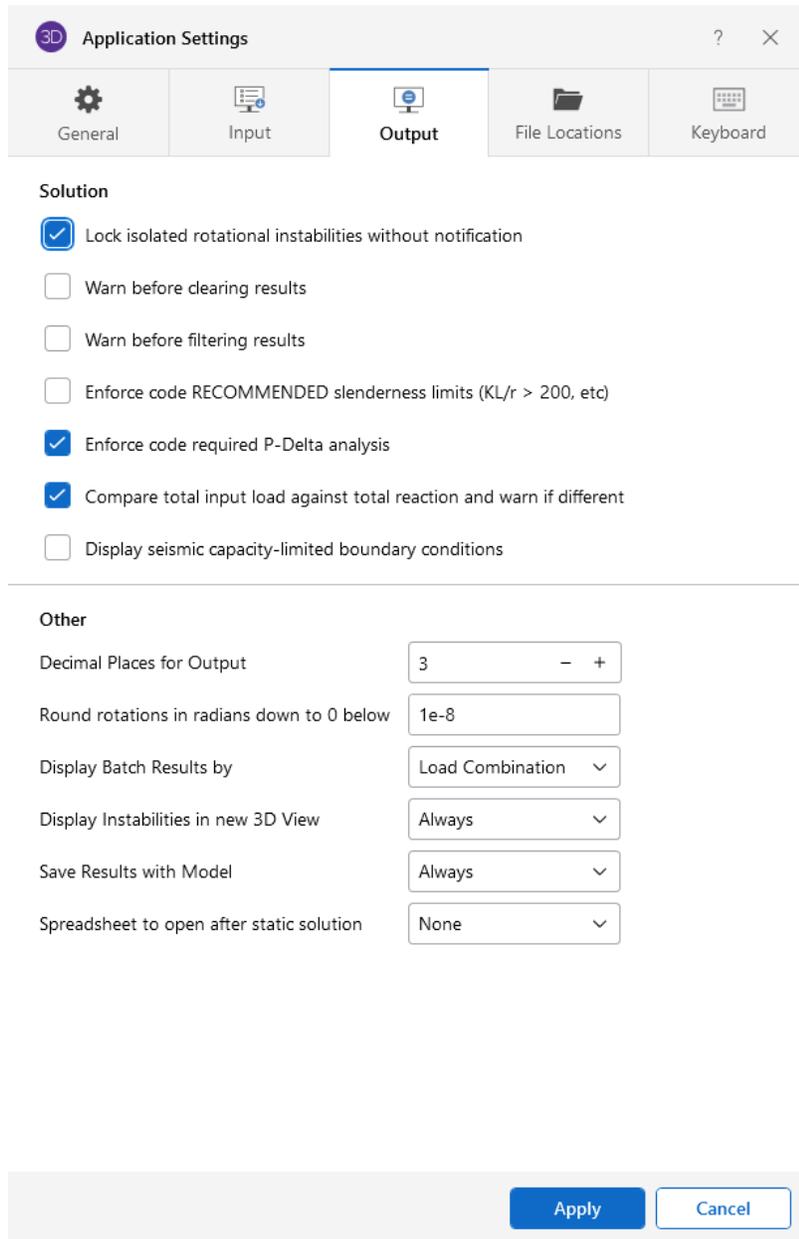
Input Application Settings

Setting	Description
Node Labels	Sets the default prefix to be used in node labels.
Member Labels	Sets the default prefix to be used in member labels.
Plate Labels	Sets the default prefix to be used in plate labels.
Wall Panel Labels	Sets the default prefix to be used in wall panel labels.
Solid Labels	Sets the default prefix to be used in solid labels.
Decimal places for Input	Sets the number of decimal places to display in the data spreadsheets with a maximum of four places.

Output Application Settings

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

RISA-3D can provide a warning when clearing results. You may also specify the number of decimal places that are displayed. The number of figures displayed may not be the actual number. Behind the scenes RISA-3D maintains numbers to numerous decimal places.



The following table provides descriptions for the settings.

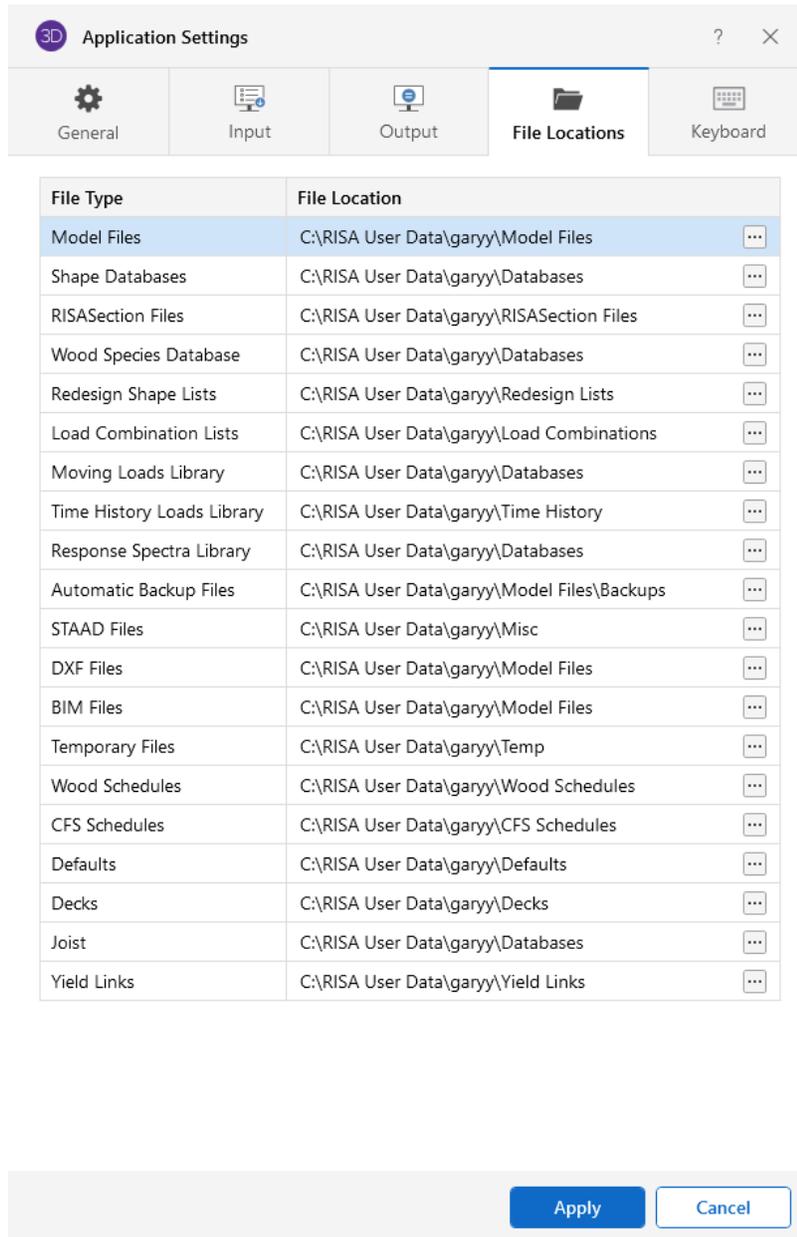
Output Application Settings

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

Setting	Description
	 Note: If you are getting a Warning Log message that states "Sum of reaction is not equal to the sum of the loads", you may need to uncheck this box to look for instabilities in your model.
Warn before clearing results	Verifies that results are to be cleared to edit the model.
Warn before filtering results	With this option checked, the program will always warn you before you filter output results.
Enforce code RECOMMENDED slenderness limits (KL/r > 200, etc)	Unchecking this option waives the slender check for slender members.
Enforce code required P-Delta analysis	Unchecking this option waives the P-Delta analysis requirement for certain design codes.
Compare total input load against total reaction and warn if different	Enables a check in the program to confirm that the total applied load equals the sum of the joint reactions. See the Warning Log topic for more information on this. For additional advice on this topic, please see the RISA Tips & Tricks website: www.risa.com/post/support . Type in Search keywords: Sum of Reactions .
Display seismic capacity-limited boundary conditions	This provides option to display the capacity-limit boundary conditions in seismic analysis
Decimal Places for Output	Sets the number of decimal places to display in the results spreadsheets with a maximum of four places.
Round rotations in radians down to 0 below	Shows 0 for the rotation when smaller than this value.
Display Batch Results by	The results of a batch solution may be grouped by load combination or by item. For example you can group results for all members under each particular load combination or you can group results from each combination under a particular member. The setting here is merely a preference. Once you have solved a model you can switch back and forth using the Sort by Combination or Sort by Item options found on the Results tab.
Display Instabilities in new 3D View	If the model has instabilities, the program can provide an additional 3D view selecting only the nodes that were locked as a result of the instability. This option lets you control if and when this additional view is displayed.
Save Results with Model	These options let you control what is done with the results when saving a file.
Spreadsheet to open after static solution	This option allows you to select the default results spreadsheet to open after solution. You can also choose 'None'.

File Location Application Settings

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

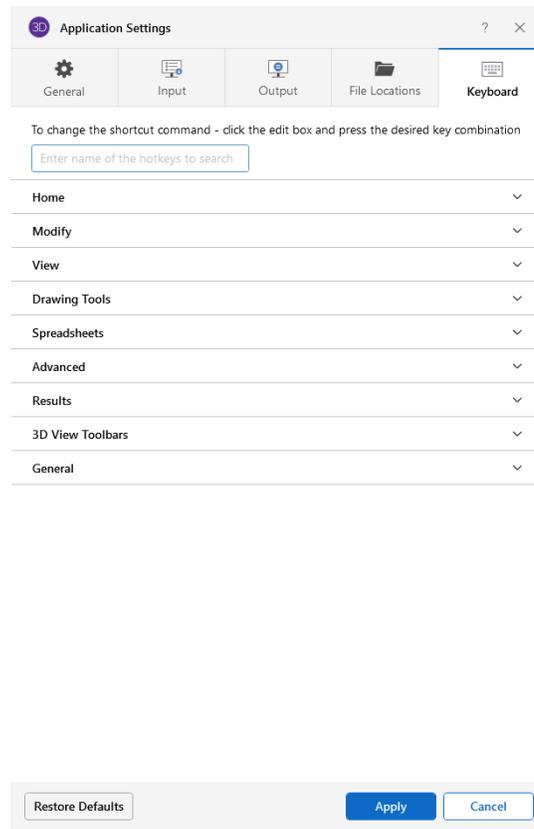


For each file type in the list the current setting is displayed.

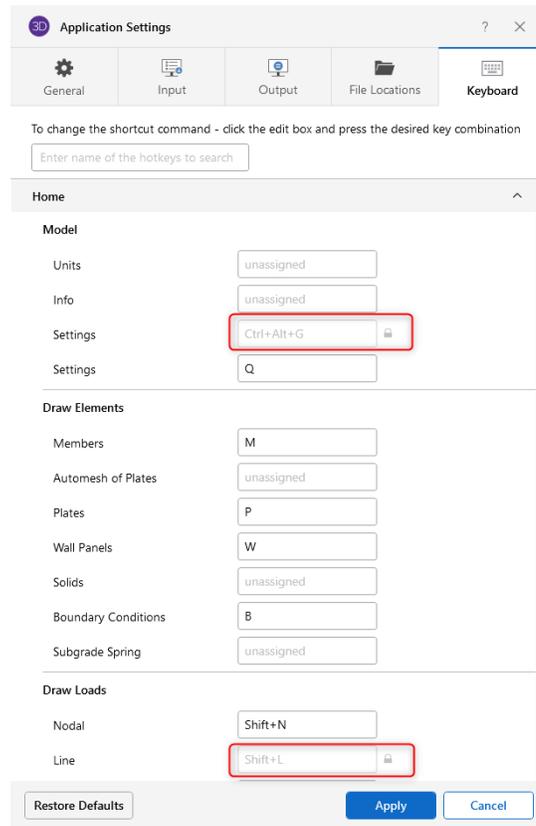
Click the  browse button to choose a different location.

Keyboard Application Settings

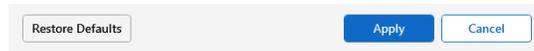
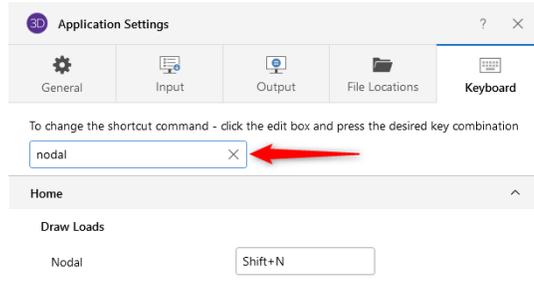
RISA-3D provides options to customize hot keys. All buttons in the ribbon and all input/output spreadsheets can be assigned hot keys for quick access. The hot keys are grouped by the ribbon tabs or by the spreadsheet types as shown below:



The program defines some default hot keys for the most commonly used functions. Note that some hot keys are reserved for specific functions and grayed out with a lock. These hot key can not be modified. You can customize the rest of the keys as needed.



You can type in keywords in the search box to quickly find the hot key for a specific function:



 **Note:** You can quickly access the Keyboard tab by right-clicking in the 3D View and selecting "Hotkeys".

Member Design Optimization

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

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

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

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Redesign**.

Wall Design Optimization

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

- Concrete: [Concrete Wall - Design Rules](#)
- Masonry: [Masonry Wall - Design Rules](#)
- Wood: [Wood Wall - Design Rules](#)

**Note:**

- For the optimization procedure on concrete walls see [Concrete Wall - Design](#).
- For the optimization procedure on masonry walls see [Masonry Wall - Design](#).
- For the optimization procedure on wood walls see [Wood Wall - Design](#).

Member Design Lists

A **Design List** defines a set of member sizes that are considered when the program is suggesting member sizes. You may assign a Design List to a member based on Member Type. The design lists available for Column members may not match the design lists for Beam members, because some shapes are more appropriate for use as beams than as columns. You may edit these lists or create additional custom lists of your own. For more information on these redesign lists, including file format, editing procedure, and user defined lists refer to [Appendix A - Redesign Lists](#).

Member Design Rules – Size / U.C.

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

Size/UC	Concrete Rebar	Deflection (Beams)					
	Label	Max Depth [in]	Min Depth [in]	Max Width [in]	Min Width [in]	Max Axial/Bending Chk	Max Shear Chk
1	Joists	12				1	1
2	Exterior Columns					0.9	1
3	Interior Columns			16	16	0.85	1
4	Girders	24	12			1	1

The spreadsheet has three tabs: **Size/UC**, **Deflection**, and **Concrete Rebar**. The entries for the **Size / UC** tab are explained in the following table.

Size/UC columns

Column Heading	Description
(Design Rule) Label	You must assign a unique label to the design rules. You then refer to the design rule by its label when assigning it to members. The label column is displayed on all tabs of the spreadsheet.
Max/Min Depth	You may enforce depth restrictions by setting either a maximum and/or minimum depth.
Max/Min Width	You may enforce width restrictions by setting either a maximum and/or minimum width.
Max Bending and Shear Check	Enter the maximum bending and shear unity checks. This should usually be specified as "1". If you desire a larger factor of safety, provide a lower factor (i.e. ".95")

Member Design Rules – Deflection

Deflection Design							
Size/UC	Concrete Rebar	Deflection (Beams)					
	Label	LC	Ratio	LC	Ratio	LC	Ratio
1	Joists	1	240	2	360	3	240
2	Exterior Columns	1	240	2	360	3	240
3	Interior Columns	1	240	2	360	3	240
4	Girders	1	240	2	360	3	240

This spreadsheet allows you to define deflection criteria for beams in the model. You must choose specific load combinations from the Load Combinations spreadsheet (up to three) and give the maximum deflection L/y criteria.



Note:

- In the [Load Combination Generator - Gravity tab](#) the program will create deflection load combinations for you automatically to match up with the defaults.
- If you write "None" in the **LC** column the program will ignore this criteria when checking deflections. If all three **LC** columns are defined as "None" then no deflection checking will be done for beams in the model.

Member Design Rules – Concrete Rebar

Concrete Rebar Parameters												
Size/UC	Concrete Rebar	Deflection (Beams)										
	Label	Optimize Rebar ?	Min Flex Bar	Max Flex Bar	Shear Bar	Legs per Stirrup	Top (Column) Cover [in]	Bottom Cover [in]	Side Cover [in]	Top/Bottom Bars	Add'l Side Bars	Shear Bar Spacing [in]
1	Joists	Optimize	#6	#10	#4	2	1.5	1.5	1.5	2	1	12
2	Exterior Columns	Explicit	#6	#10	#4	2	1.5	1.5	1.5	2	1	12
3	Interior Columns	Optimize	#6	#10	#4	2	1.5	1.5	1.5	2	1	12
4	Girders	Optimize	#6	#10	#4	2	1.5	1.5	1.5	2	1	12



Note: If you would like to define specific flexural and shear rebar layouts for beams and columns, either use the **Explicit** option below or see [Rebar Layout Database](#) for Custom Rebar layout.

The entries for the **Concrete Rebar** tab are explained in the following table.

Concrete Rebar

Column Header	Description
Optimize	The options in this column are Optimize and Explicit . Optimize indicates the program will find the most optimal size and spacing of the reinforcement. Explicit indicates the exact number of and size of reinforcement bars. When Explicit is selected the additional columns are editable and allow you to enter the quantity for Top/Bottom Bars, Add'l Side Bars, and Shear Bar Spacing . These columns are grayed out when Optimize is selected.
Flexural Bars	Use the Min Flex Bar and Max Flex Bar columns to restrict bar sizes for your flexural reinforcing. Currently we support the ASTM A615 (imperial), ASTM A615M ("hard" metric, i.e. #8M is an 8mm bar), BS 4449 (British), prENV 10080 (Euro), CSA G30.18 (Canadian), and IS 1786 (Indian) reinforcement standards. You may specify your rebar set in the Model Settings - Rebar . You can force the program to analyze one bar size by setting the Min and Max values to be the same bar.
Shear Ties	Use the Shear Bar column to enter the size of your shear ties. Currently we support the ASTM A615 (imperial), ASTM A615M ("hard" metric, i.e. #8M is an 8mm bar), BS 4449 (British), prENV 10080 (Euro), CSA G30.18 (Canadian), and IS 1786 (Indian) reinforcement standards. You may specify your rebar set in the Model Settings - Rebar .
Legs per Stirrup	Use the Legs per Stirrup column to enter the specific information about how many legs (2 to 6) each of your shear ties is expected to have.
Concrete Cover	The last three columns are used to specify the clear cover measured to the shear reinforcing. Note that the Top Cover is used for all sides of Column members.

Member Design Rules Editor

The **Member Design Rules Editor** provides a way to create or modify member design rules in a graphical way. This editor automatically updates any information input into the **Member Design Rules** spreadsheet.

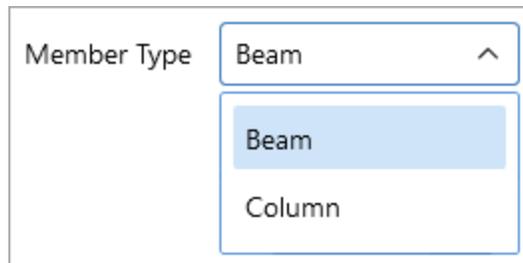


Note:

- Keep in mind that this dialog has three tabs. Make sure to fill out the information on all the tabs before clicking **OK**.
- See the information in the Status Bar for more information on each of these entries.

The **Design Rule Label** is editable and will change the spreadsheet label. It is useful to name the design rule with a descriptive name like "First Floor Columns".

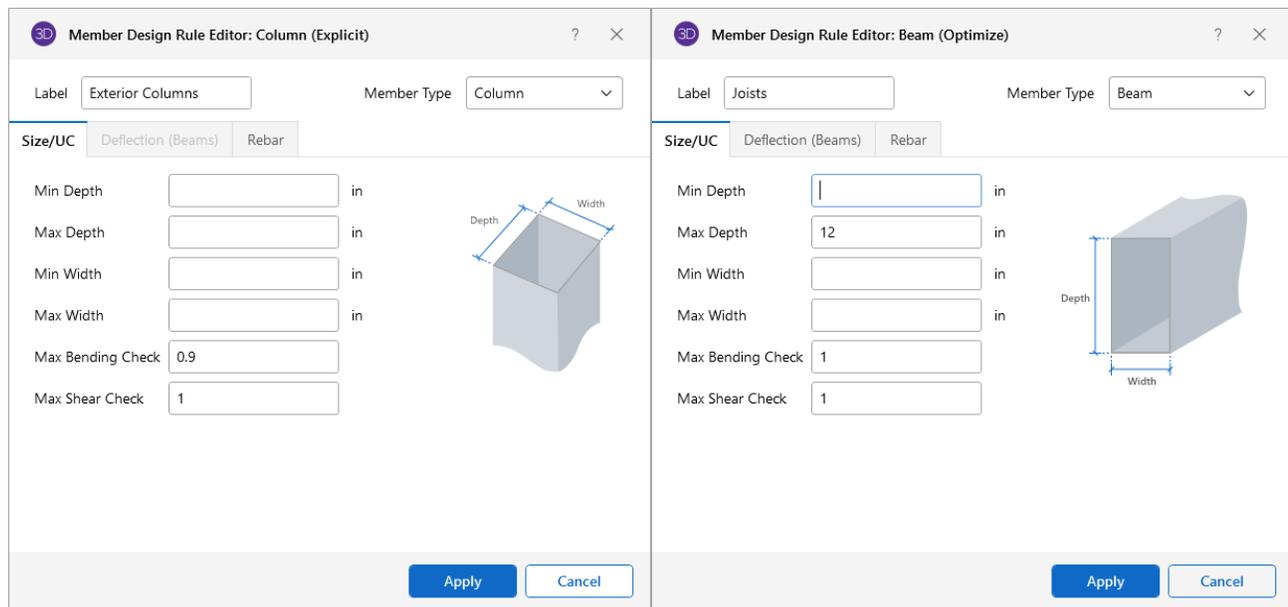
The **Member Type** listed is for reference of the picture only in this dialog. The member type will not control the Member Type of the beam or column which you will need to assign separately when the member is drawn.



The Size/UC tab:

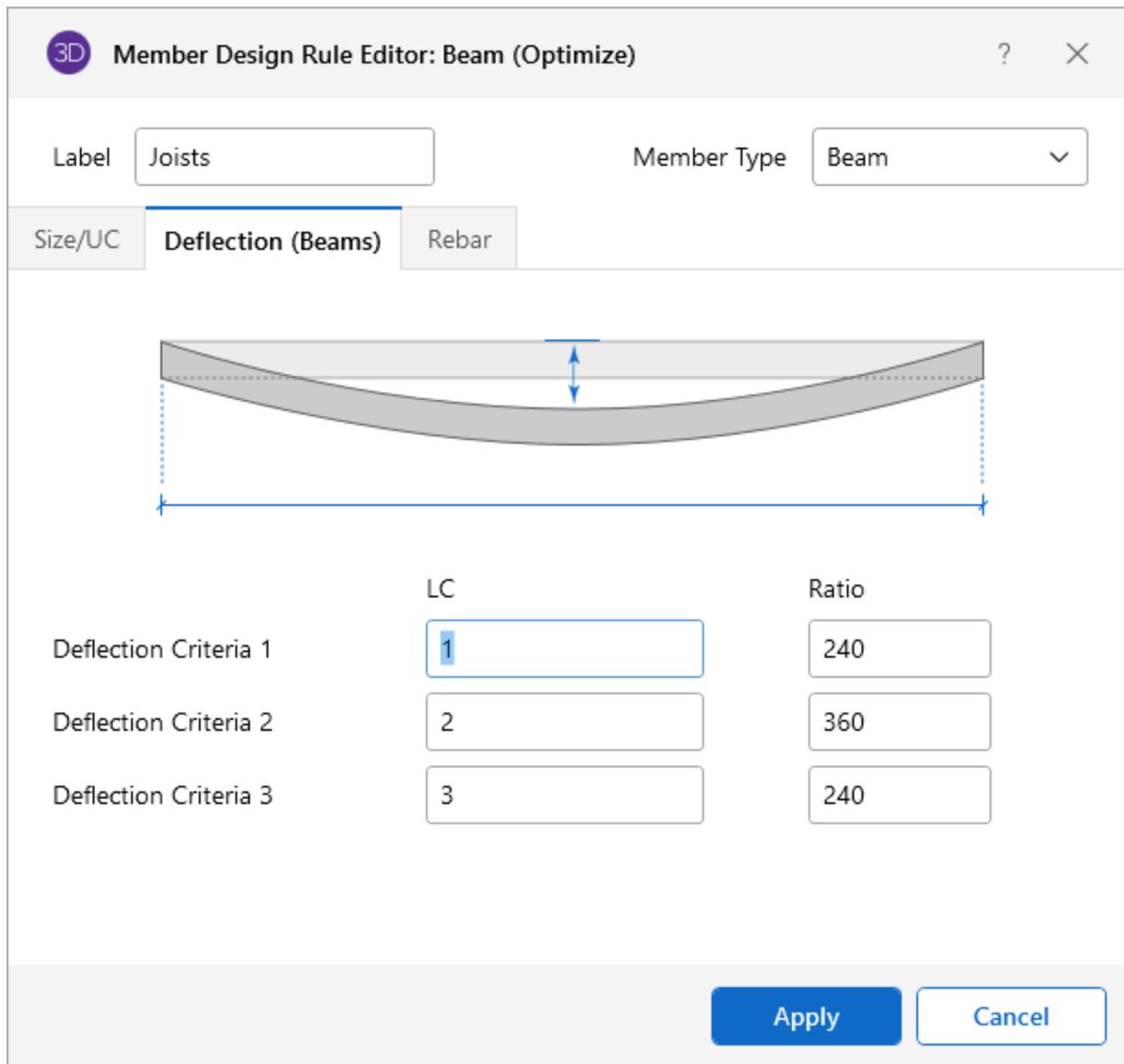
Explicit

Optimize



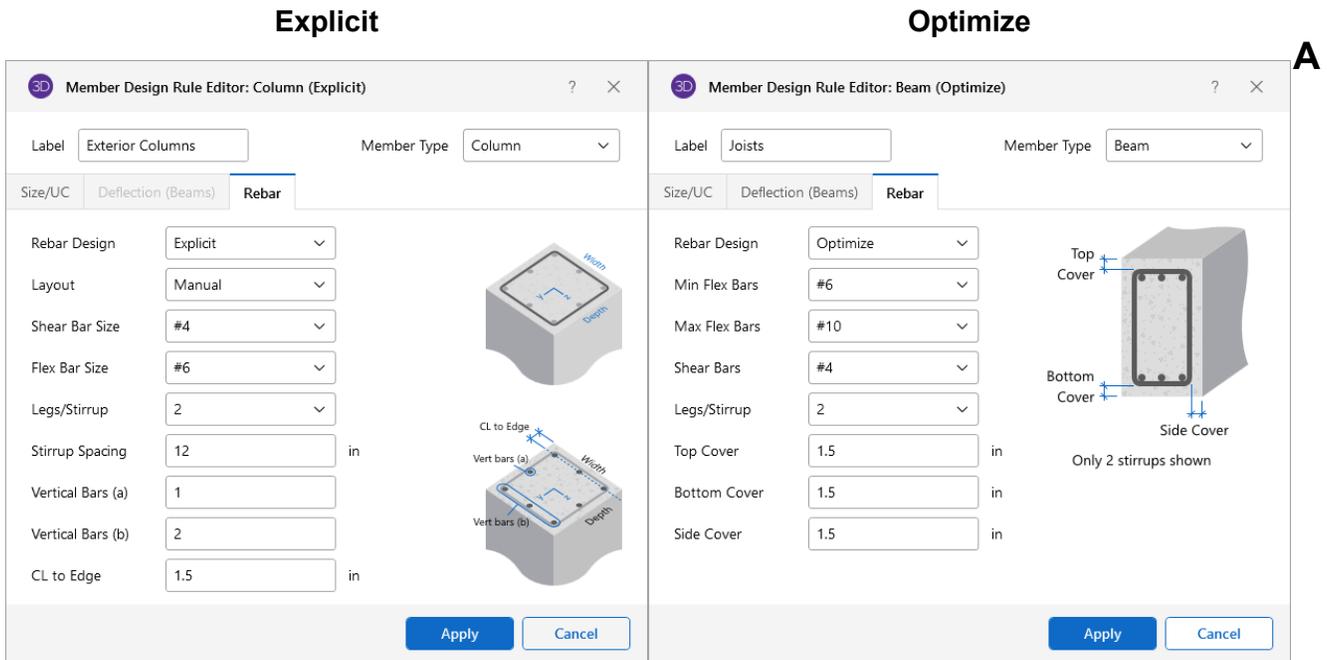
Deflection (Beams) tab:

This only applies to Beam members.



Rebar tab:

You can select **Optimize** or **Explicit** to set the rebar design parameters.



A

C

Accessing the Member Design Rules Editor

To access this editor you can click the three dots on any tab in the **Member Design Rules** spreadsheet (click in the **Label** cell to make the three dots appear).

Design Size and Code Check Parameters			
Size/UC	Concrete Rebar	Deflection (Beams)	
	Label	Max Depth [in]	Min Depth [in]
1	Joists	12	
2	Exterior Columns		
3	Interior Columns		

Member Optimization Procedure

Member optimization is performed both on explicitly defined members and on members defined through the use of Section Sets. Members defined as part of a Section Set are checked to determine which member has the highest code check value and which member has the highest shear check value. These members are considered to be the controlling members for that section set.

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

Optimization Results

Suggested Shapes Spreadsheet

Access the **Member Suggested Shapes** spreadsheet by selecting the **Results** tab ▶ **Envelope** (or **LC**) button ▶ **Suggested Design**. Alternatively, on the **Results** toolbar you can select **Suggested Design**.

Member Suggested Shapes						
Members	Masonry Walls	Wood Walls	CFS Walls			
Re-solve with Suggested Shape(s)/Wall(s)						
	Section Set/Member	Current Shape	Suggested Shape	Controlling Member	Controlling Criteria	Use Suggested?
1	M1	CRECT10X10	CRECT8X8		Strength	<input checked="" type="checkbox"/>
2	M2	CRECT10X10	CRECT8X8		Strength	<input checked="" type="checkbox"/>
3	M3	CRECT10X10	CRECT8X8		Strength	<input checked="" type="checkbox"/>
4	M4	CRECT10X10	CRECT8X8		Strength	<input checked="" type="checkbox"/>
5	M5	CRECT10X10	CRECT16X16		Strength	<input checked="" type="checkbox"/>
6	M6	CRECT10X10	CRECT16X16		Strength	<input checked="" type="checkbox"/>
7	M7	CRECT10X10	CRECT16X16		Strength	<input checked="" type="checkbox"/>
8	M8	CRECT10X10	CRECT16X16		Strength	<input checked="" type="checkbox"/>
9	M9	CRECT10X10	CRECT16X16		Strength	<input checked="" type="checkbox"/>

These are the suggested shapes resulting from the optimization calculations. They are chosen from each member's assigned [Design List](#). The suggested shape is *estimated* to most closely meet the criteria specified in the [Member Design Rules](#) without exceeding them. If the member is a "Beam" member type then the **Controlling Criteria** could be based on either strength criteria or deflection criteria.



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

To confirm that these alternate shapes are acceptable you **MUST** adopt any changes into the model then re-solve and check the results. The suggested shapes are based on the forces for the current model. Keep in mind that the current results are based on the stiffness of the current

shapes. Changing the shapes will change the stiffness, which is why the model needs to be resolved.

It may be necessary to cycle through this process a few times to achieve the best shapes.

You may try the new shapes by right clicking in the Suggested Shapes spreadsheet and selecting **Solve Again Using the Suggested Shapes**. The shapes listed in the **Suggested Shapes** column will only be used to update the model file if the "Use Suggested?" box is checked for that particular member or section set.

	Section Set/Member	Current Shape	Suggested Shape	Controlling Member	Controlling Criteria	Use Suggested?
1	M1	CRECT10X10	CRECT8X8		Strength	<input checked="" type="checkbox"/>
			CRECT8X8		Strength	<input checked="" type="checkbox"/>
			CRECT8X8		Strength	<input checked="" type="checkbox"/>
			CRECT8X8		Strength	<input checked="" type="checkbox"/>
			CRECT16X16		Strength	<input checked="" type="checkbox"/>
			CRECT16X16		Strength	<input checked="" type="checkbox"/>
			CRECT16X16		Strength	<input checked="" type="checkbox"/>
			CRECT16X16		Strength	<input checked="" type="checkbox"/>
			CRECT16X16		Strength	<input checked="" type="checkbox"/>

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

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

enough to exceed the minimum.

- A code was not specified for that material on [Model Settings - Codes](#).

**Note:**

- For masonry wall design optimization see the [Masonry Wall - Design](#) topic.
- For wood wall design optimization see the [Wood Wall - Design](#) topic.

Diaphragms

Diaphragms provide lateral load distribution functions, and are necessary for automatic [Wind](#) and [Seismic](#) load generation. There are three fundamental types of diaphragms in structural modeling; Rigid, Semi-Rigid and Flexible. The following sections provide more details.

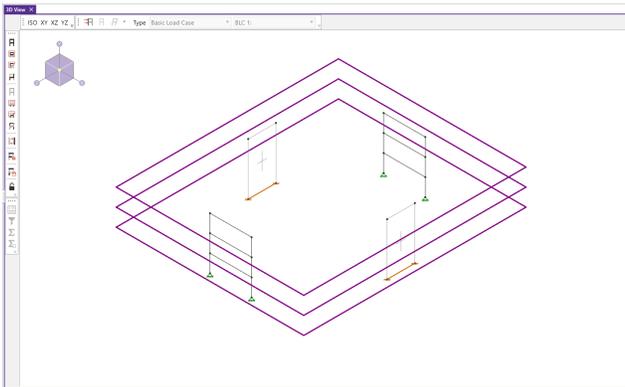
- [Rigid Diaphragms](#)
- [Semi-Rigid Diaphragms](#)
- [Semi-Rigid Diaphragms \(RISAFloorES\)](#)
- [Flexible Diaphragms](#)

**Note:**

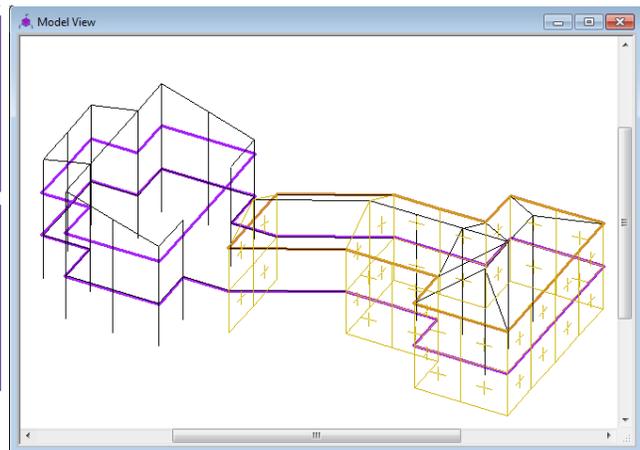
- See the [Diaphragms - Analysis and Results](#) topic for more information on flexible diaphragm analysis and design.
- Wall panels can be detached from a diaphragm from the [Wall Panel Editor](#).

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Diaphragms**.

RISA-3D Rigid Diaphragm



RISAFloor linked with RISA-3D Includes Rigid and Flexible Diaphragms



Rigid Diaphragms

Rigid diaphragms represent a plane of very high rigidity. Rigid diaphragms distribute load to elements which connect to them solely based on the stiffness of the elements. They achieve this by tying all of the nodes within the diaphragm plane together for both translation and rotation, but only within the plane of the diaphragm. This is typical behavior for most slabs and decks, which attribute vertical loads based on the tributary area of their supporting members.

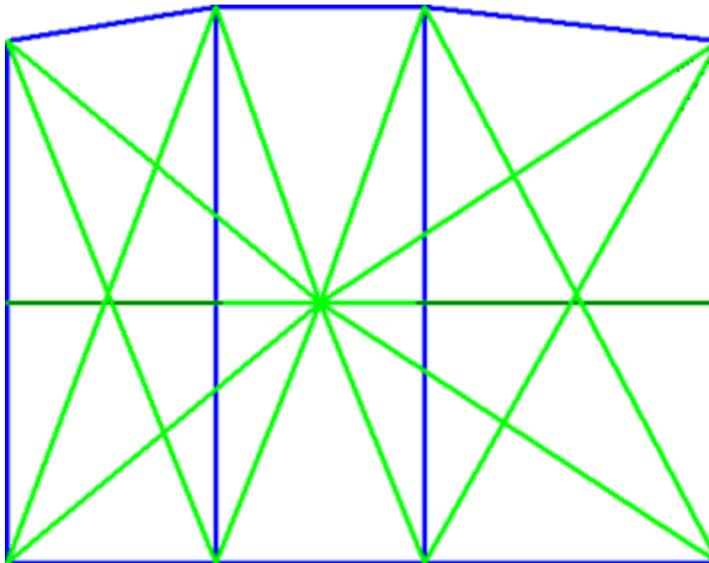
**Note:**

- There is no rigid diaphragm design in the program. This is strictly an analysis tool.
- Sloped rigid diaphragms are not supported. Therefore, rigid diaphragms always exist in a flat horizontal plane.

Load Attribution

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

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



Connectivity

All nodes which fall within the plane of the diaphragm automatically become connected to it. Nodes may be intentionally disconnected from the diaphragm by checking the **Detach From Diaphragm** box in the [Node Coordinates spreadsheet](#). If a [boundary condition](#) exists within the plane of a diaphragm it will act as a restraint for all of the nodes connected to the diaphragm.

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

Rigid Diaphragm Stiffness

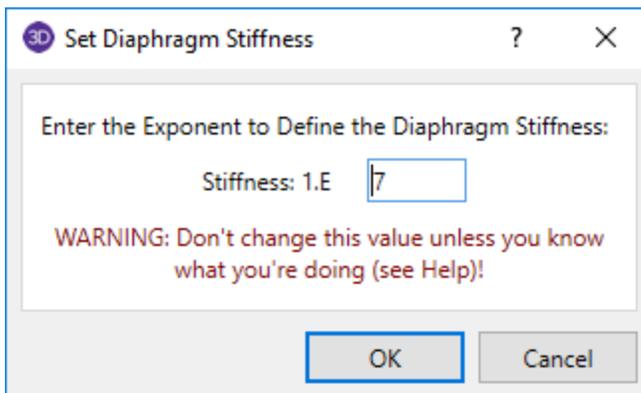
The stiffness of the rigid diaphragm is set to a unitless value of 1×10^7 by default. This value has been calibrated as providing the best behavior for most models. It can be adjusted from within the diaphragms spreadsheet, however adjusting this value is only recommended in the following circumstances:

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

To adjust the diaphragm stiffness:

1. [Open the Diaphragms spreadsheet](#) from the Explorer panel or from the Data Entry list.
2. Right-click in the Diaphragms spreadsheet and choose Set Diaphragm Stiffness from the menu.

You may also modify this value in **Model Settings** ▶ **Solution** tab ▶ **Advanced** options:



Rigid Diaphragms in a RISA-3D Only Model

With RISA-3D's diaphragm feature, a node can be defined within a given plane as the diaphragm node, and all points in that plane will be rigidly connected to each other with internal rigid links. Thus, each node will rotate and translate as one rigid body and exhibit rigid diaphragm behavior. RISA-3D's rigid diaphragm feature is discussed further in the [Diaphragm Modeling Tips](#) section.

Rigid Diaphragms in a Combined RISAFloor/RISA-3D Model

A RISA-3D model that is linked up to RISAFloor has an automatic rigid diaphragm analysis and design. Each individual slab/deck polygon is converted to a rigid diaphragm within RISA-3D. Therefore, it is possible to have multiple independent diaphragms at any given floor elevation.

Semi-Rigid Diaphragms

A semi-rigid diaphragm is one which distributes loads based on both the stiffness of elements which connect to it and on the stiffness of the diaphragm itself. For more details about semi-rigid diaphragms when RISA-3D is integrated with RISAFloor, see [Semi-Rigid Diaphragms \(RISAFloorES\)](#).

Semi-Rigid Modeling Tips for Concrete Floor Slabs

Lateral Elements: When modeling with a Semi-Rigid diaphragm, it's important to make all columns and walls **Lateral**. This will bring all the columns and walls into RISA-3D which will support the slab as the vertical loads are applied. In RISA-3D, the program will apply both lateral and vertical loads to the diaphragm for analysis of the lateral system. This is different than Rigid diaphragms because only the lateral resisting system needs to be transferred into RISA-3D.

Reinforcement Design: RISA-3D will not design the reinforcement for the slab. The reinforcement is designed in RISAFloor and is based on the vertical loads. However, you can find forces in the slab in RISA-3D using the tools listed below.

- **Contours:** The contours will display the global axis forces over the slab. See the [Results View Settings - Diaphragms](#) section of the RISAFloor help, for further information on display of the contours. The contours can be used to find the high and low forces as needed for reinforcement design.
- **Internal Force Summation Tool (IFST):** There is an IFST Slab tool that allows you to find forces from a point on the submesh to another point on the slab. This tool can be used to "cut" across the slab and will find forces along the "cut". This is the same tool that is used internally in RISAFloor to find Auto-Design Cuts inside the Design Strips. For further information on this tool, see [IFST](#) for more details.

Semi-Rigid Modeling Tips for Beam Supported Floors

Lateral Elements: When modeling with a Semi-Rigid diaphragm, it's important to make all columns and walls **Lateral**. This will bring all the columns and walls into RISA-3D which will support the slab as the vertical loads are applied. In RISA-3D, the program will apply both lateral and vertical loads to the

diaphragm for analysis of the lateral system. This is different than Rigid diaphragms because only the lateral resisting system needs to be transferred into RISA-3D.

Vertical Loads: The semi-rigid diaphragm has self weight and out-of-plane stiffness. Internally, the plates are modeled as normal so that there is stiffness out-of-plane. The diaphragm is used for out-of-plane and in-plane load distribution.

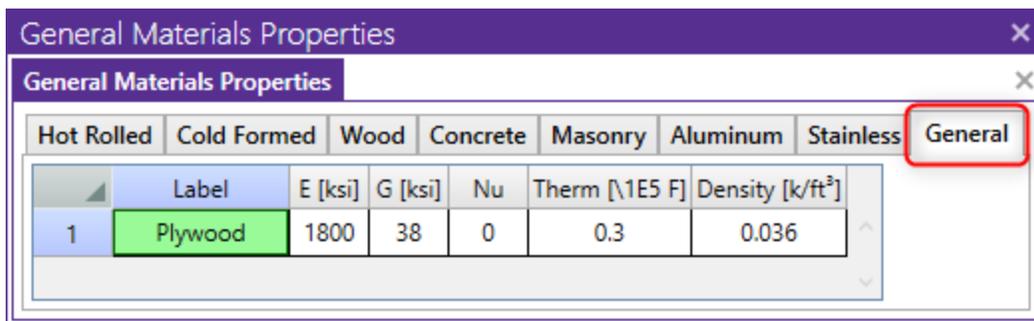
Diaphragm Material: The Semi-Rigid diaphragm material uses the General Materials. There are a list of default General Materials, however you can add to this list to match your diaphragm. See [Materials](#) for further information on creating your own General Material.

Diaphragm Thickness: The Semi-Rigid diaphragm thickness should be selected based on the approximate stiffness you expect from the slab or deck for lateral loading. The semi-rigid diaphragm is modeled using isotropic plates. Therefore, there is no association with the deck direction for the semi-rigid diaphragms. If your diaphragm is truly one-way, you may consider using a Flexible diaphragm.

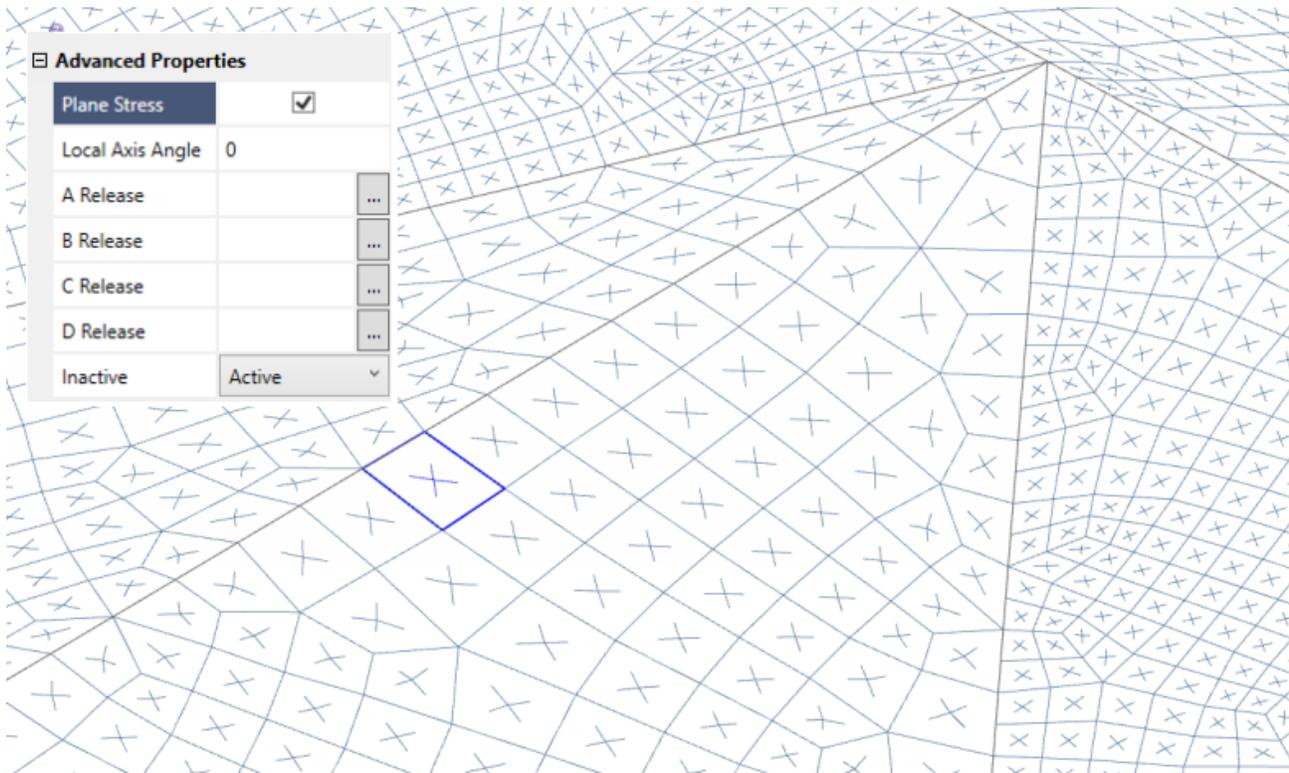
Semi-Rigid Diaphragms for RISA-3D Only Models

RISA-3D does not have the ability to automatically define a Semi-Rigid diaphragm. However, Semi-Rigid diaphragms can be represented in the model using [plates](#).

A semi-rigid diaphragm is modeled with plates, and requires you to model and submesh the plates appropriately. In order to adequately mesh your plates it is good to be familiar with Plate-Member Interaction. Plates are modeled using general materials, so the first step is to set up a material with the material properties of your diaphragm.



Next, model your plates using the specified material, and with an accurate thickness. Be sure to check the “Plane Stress” option. This makes it so that the plates only have stiffness within their own plane, and as a result won’t add any composite bending strength to the beams in the plane of the diaphragm.

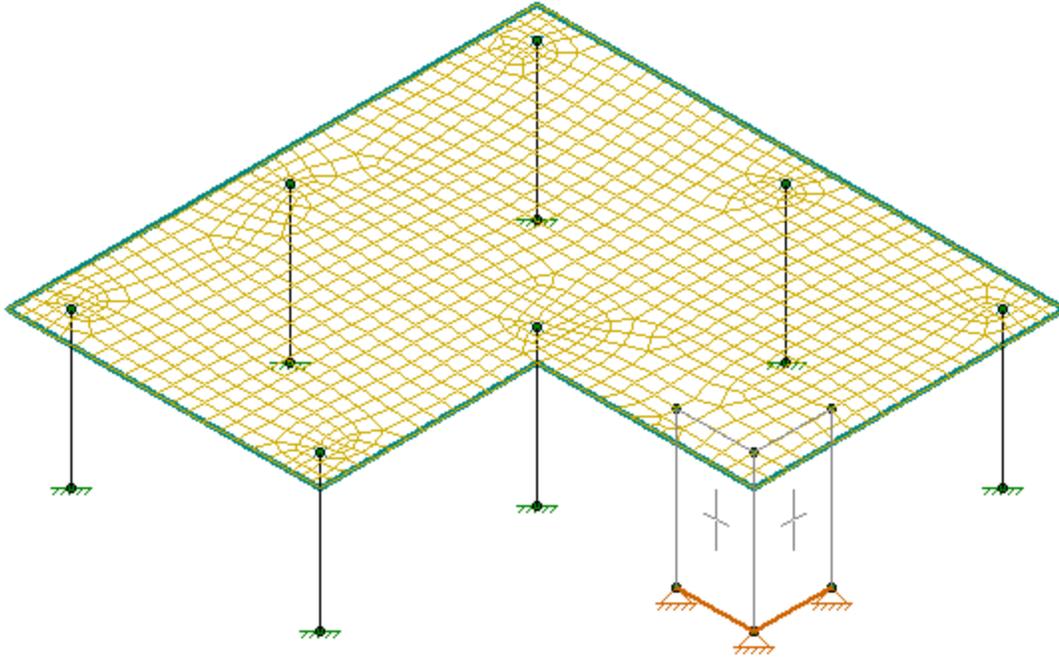


Note: When meshing a floor diaphragm around members, you want to make sure that the corners of your plates meet at members. Otherwise, when applying surface loads to plates, the members will not see that load.

Semi-Rigid Diaphragms in a RISAFloorES Model

The Semi-Rigid diaphragm in RISA-3D helps you distribute the lateral forces based on the stiffness of the slab. When RISA-3D is integrated with RISAFloor, the program can create a Semi-Rigid diaphragm automatically. The Semi-Rigid diaphragm is defined in Diaphragm Parameters in a Beam Supported Floor and the **Slab Definitions** spreadsheet in a Concrete Floor Slab in RISAFloor.

During the integration between RISAFloor and RISA-3D, the program creates a mesh of FEA plate elements within the edges of the slab. The thickness of the plates is defined by the slab/diaphragm defined in RISAFloor. The plate elements are automatically submeshed so that they attach to all vertical elements (columns and walls) as well as any new members or loads added into the RISA-3D model. Below shows an example of a simple L-shaped building with the mesh displayed. For further display information see the [Results View Settings - Diaphragms](#) topic.



When the model is in RISA-3D, the mesh size is controlled by the **Model Settings - Solution Tab - Semi-Rigid Mesh**. For RISAFloor ES models, the slab stiffness can be reduced, based on the *Icr* factor in the RISAFloor **Slabs** spreadsheet and the **Use Cracked Slabs** checkbox in the Model Settings window in RISA-3D. The *Icr* slab stiffness only affects the out-of-plane stiffness. For further information on cracked slabs, see **Elevated Slabs - Analysis** in RISAFloor. The top of the columns are fixed to the diaphragm using links to distribute the forces over a 12"x12" area rather than a single column node. For further information see the **Column Meshing** section of the help in RISAFloor.



Note: Sloped semi-rigid diaphragms are not supported. Therefore, semi-rigid diaphragms always exist in a flat horizontal plane.

Semi-Rigid Diaphragms (RISAFloorES)

If you are running a RISAFloor ES/RISA-3D combined model that has semi-rigid diaphragms then at solution time, the elevated slab will be automatically meshed into quadrilateral plate elements. The automatically generated plate elements are transient in the program and will not be saved in the input file.

The slab meshing is treated similar to analysis results. When the results of an analysis are deleted, the slab mesh is cleared to be re-built during the next solution. When a solution results file is saved, the meshed elements will be included in that file.

**Note:**

- This topic is only applicable for RISAFloor ES/RISA-3D combined models that have semi-rigid diaphragms. For information on semi-rigid considerations for RISA-3D only models see the [Diaphragms](#) topic.
- The program uses the finite element analysis, along with cracking factors (columns, beams and slabs) to come up with the stiffness of each element and the system as a whole. ACI 318-14 Sections 8.11.3, 8.11.4 and 8.11.5 (ACI 318-11 Sections 13.7.3, 13.7.4 and 13.7.5) provide ways to estimate the stiffness of individual elements in the equivalent frame method. These provisions are not used in the program.

RISA-3D Linked With RISAFloor

Floor Diaphragms																	
General																	
Wood Diaphragms																	
	Elevation [ft]	Mass [k]	Mass MOI [k-ft ²]	Center Of Mass [ft]	Plus X Eccentricity [ft]	Minus X Eccentricity [ft]	Plus Z Eccentricity [ft]	Minus Z Eccentricity [ft]	Inactive	Diaphragm	Type	Region	Design Rule	SGAF	Material	Thickness [in]	No Wind/Drift
1	0	74.583	11314.192	87.5, 35	%5	%5	%5	%5	<input type="checkbox"/>	D1	Semi-Rigid		N/A	N/A	gen_Conc3NW	12	<input type="checkbox"/>

The following table lists the General tab settings and their descriptions.

General tab settings

Setting	Description
Elevation	Displays the elevation of the diaphragm. This is the same elevation as the floor which the diaphragm was created on.
Mass	Displays the dynamic mass tributary to the diaphragm. This mass is used to calculate seismic forces for both static and response spectra methods.
Mass MOI	Displays the dynamic mass moment of inertia of the diaphragm. This moment of inertia is used to calculate seismic forces for response spectra analysis.
Center of Mass	Displays the X and Z coordinates of the center of mass of each diaphragm. This is the location at which static (equivalent lateral force) seismic loads are applied for each diaphragm.
Eccentricities	Indicates the percent of length/width of the diaphragm, which are used to place "accidental eccentric" seismic

Setting	Description
	loads for each diaphragm. This only applies to the static (equivalent lateral force) procedure. See ASCE 7-16, Section 12.8.4.2 for more information.
Inactive (checkbox)	When this box is checked, the diaphragm is ignored by the program. It provides a convenient way to disable diaphragms without deleting them.
Diaphragm	Displays the diaphragm label. This label is used for the naming of Diaphragm Regions.
Type	Specifies whether the diaphragm is Rigid (Membrane) or Flexible for a Beam Supported Floor or Rigid (Membrane) or Semi-Rigid for Concrete Floor Slab. If a diaphragm has been defined as Flexible within RISAFloor it can be toggled between Rigid and Flexible in RISA-3D.
Region	Lists the diaphragm regions for each diaphragm. Diaphragm regions are used for the design of wood flexible diaphragms, and are also useful for explicitly defining how flexible load attribution is to be performed.
Design Rule	Specifies the Design Rule which is assigned to each region. Only the information on the Diaphragms tab of the Design Rules Spreadsheet is considered.
SGAF	The specific gravity adjustment factor for the design of wood flexible diaphragms. For more information see AF&PA NDS SDPWS, Table A4.2, Note 2. This value defaults to 1. However, it should be manually changed if the framing supporting the wood flexible diaphragm is not Douglas Fir-Larch or Southern Pine.
Material	The 'Material' column specifies the material properties of the diaphragm (when a semi-rigid diaphragm is selected).
Thickness	The 'Thickness' column indicates the thickness of the diaphragm (when a semi-rigid diaphragm is selected).

Setting	Description
No Wind/Drift	You may designate any floor level as a No Wind/ Drift (i.e. mezzanine) level in order to omit it from the generated wind load calculation and the drift calculations .



Note: For information about the **Wood Diaphragms** tab see the [Diaphragms - Analysis and Results](#) topic.

Mesh Considerations

Mesh Size

The global mesh size for the slab can be input on the [Solution](#) tab of the Model Settings. As a smaller mesh size is more accurate, the mesh size can get too small. The smaller mesh size will lead to a longer solution time and more memory usage.

Graphical Display of the Slab Mesh

By default, the plate elements associated with the slab are not visible to you. The mesh can be turned on using the setting on the [Model View Settings - Diaphragms](#) tab. Listed under the Wireframe selection, the **Show Mesh** check box turns the display of the slab mesh on or off.



Note: The display of the mesh is only available when there are active analysis results.

Point and Line Constraints for the Slab Mesh

Point constraints are the locations within the slab that require connectivity to the meshed plate elements. The program automatically generates point constraints at the following locations:

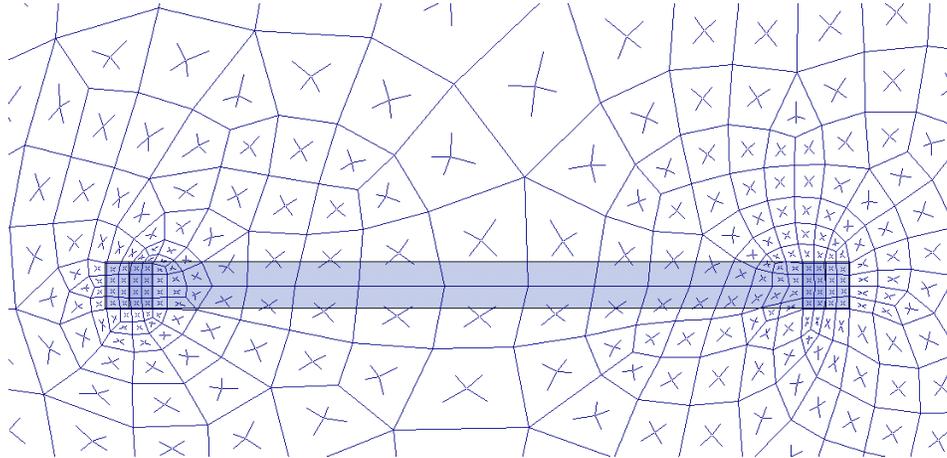
- Support points which include: joints along a wall panel and column joints (see [Column Meshing](#) for full description).
- Beams points which include: the start, end and intermediate joints along the beam.
- Any nodes that land in the plane of the semi-rigid diaphragm that have Joint Loads applied to them.



Note: The [P-Delta Leaning Column](#) effect is not considered for RISAFloor gravity columns in a combined RISAFloor/RISA-3D model.

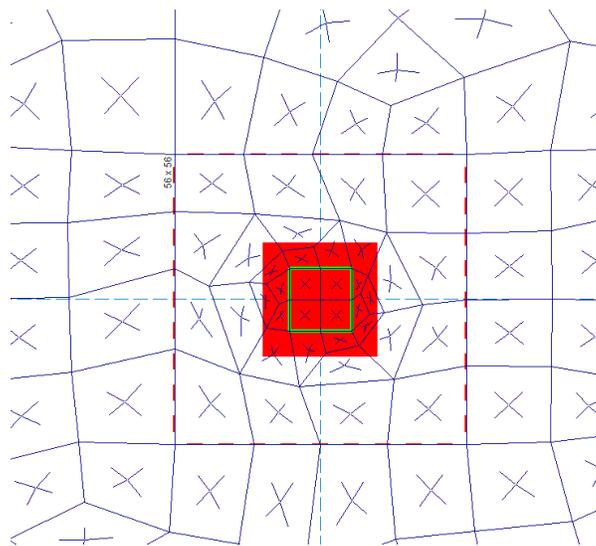
Beam Connectivity to Mesh

Beams drawn on a Two-Way slab are meshed directly to the surrounding plates. The following image shows that the center-line of the beam is meshed to the plates.

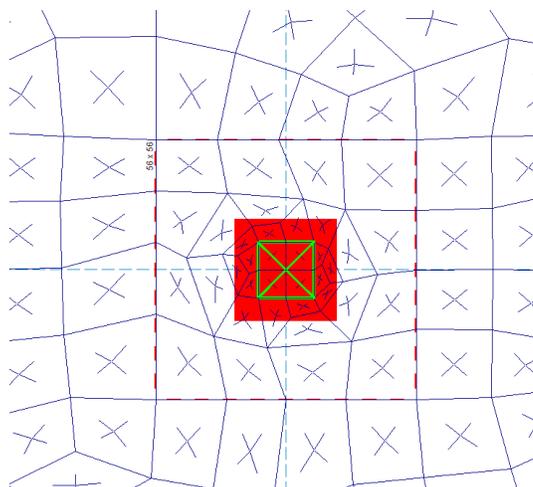


Column Meshing

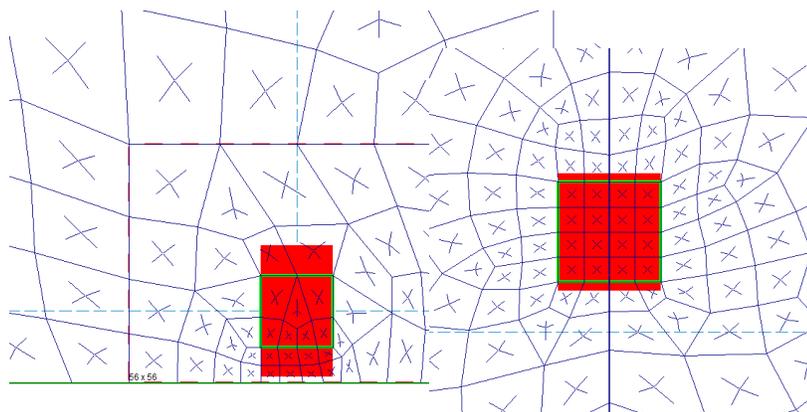
The top of the column will be submeshed and connected to the surrounding plates with rigid links in a 12" x 12" square. This approach is more accurate than modeling the column connection at only one node. When only one node is used the slab will have a higher peak moment and higher deflection at that one point. Instead the rigid links create a rigid region over the top of the column that approximate the stiffness of the column attachment to the slab. The size of the rigid links will not change based on the column dimensions as this is only approximate. In most cases there will be 4 plates inside the rigid links.



The column node will connect to the slab via the additional diagonal rigid links. After the slab is meshed, additional diagonal rigid links are drawn to connect the column node to the square rigid links (shown below). The column node has rotational and translational support so there will be vertical and rotational constraint at the column location.



In most cases, the column node will land inside the 2x2 plate mesh in the same location as a plate corner. However, in certain edge and corner situations, as well as beam intersections, the column node will not align with the plate corners.



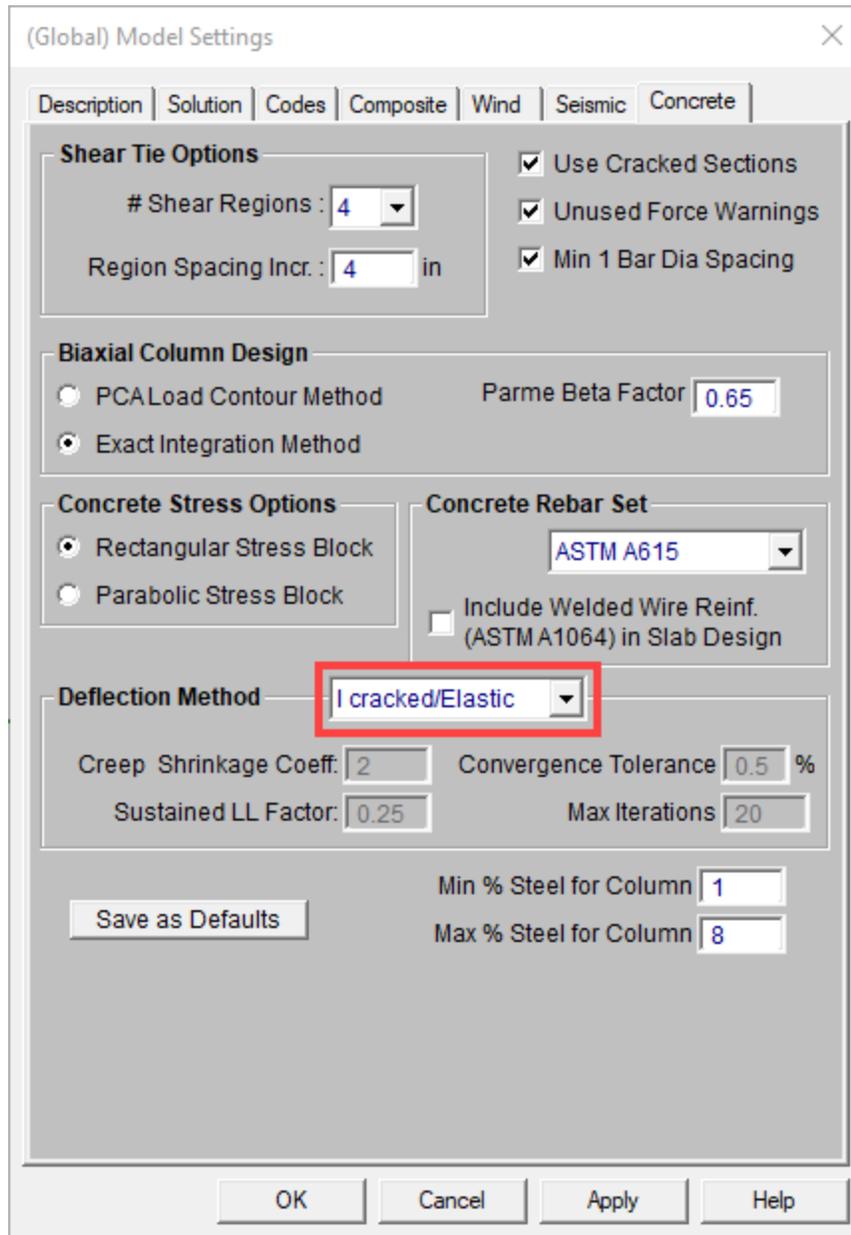
Edge Column

Beam Intersecting a Column

Slab Stiffness Considerations

Cracked Slabs

Selecting a Deflection Method of **I Cracked/Elastic** (available only in RISAFloor) in Model Settings, controls the stiffness reduction based on the I_{cr} factor in the 'Slabs' spreadsheet.



The image shows the "Slabs" window with a table of slab properties. The table has columns for Label, Definition, Thickness[in], Material, Type, and Icr Factor. The first row is highlighted in green.

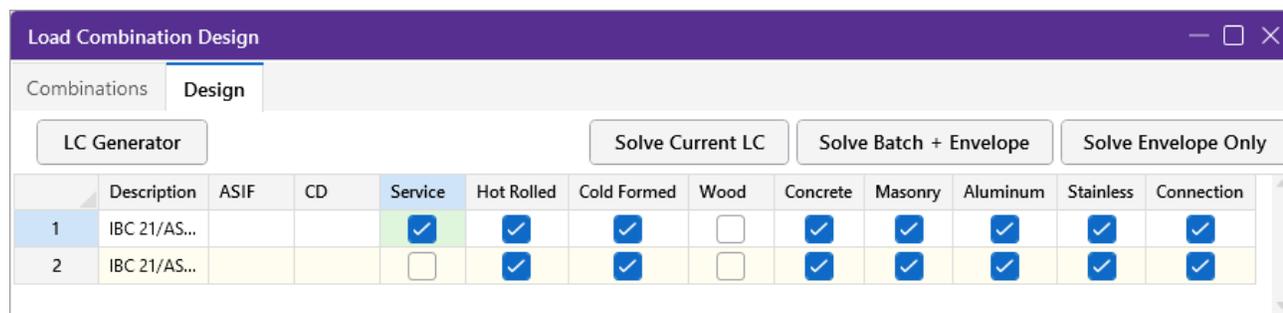
	Label	Definition	Thickness[in]	Material	Type	Icr Factor
1	S1	Semi-Rigid Definitic	12	Conc3000NW	Semi-Rigid	0.25

The Icr factor default is set to 0.25 per ACI 318-14 Section 6.6.3.1.1 (ACI 318-11 Section 10.10.4.1). You can modify this factor with any value between zero and 1.0 by typing in this field. The Icr factor

adjusts the stiffness of the plate elements comprising the slab by altering the thickness during solution. This adjusted thickness does not, however, affect design values (reinforcement calculations).

Service Checkbox

The **Service** checkbox in [Load Combinations- Design](#) tab currently only applies to Concrete Floor Slabs in RISAFloor ES.



The commentary in ACI 318-14, Section 6.6.3.1.1 (ACI 318-11 Section 10.10.4.1) recommends a 1.43 factor to be applied to the cracked moment of inertia for all service loads. The program applies this factor to all Load Combinations that have the **Service** checkbox checked, as long as the **Use Cracked Slabs** is selected in the **Model Settings** in RISAFloor. This results in an effective default slab moment of inertia of $(0.25) \cdot (1.43) \cdot I_g = 0.3575 \cdot I_g$



Note: The Long Term Deflection also gives the cracking analysis for the Service Load Combinations in the graphical display, spreadsheet results and Detail Report.

Flexible Diaphragms

Flexible diaphragms distribute loads to elements which connect to them solely based on the tributary area of the element within the plane of the diaphragm.

Flexible Diaphragms in a RISA-3D Only Model

The diaphragm feature in RISA-3D cannot be used to create a flexible diaphragm. To consider a flexible diaphragm in RISA-3D there are a couple of options. The main option is to manually calculate how much force is going to each frame or shear wall in the lateral system and apply that load directly to those elements as a point or distributed load. Another option is to essentially use a semi-rigid path to use plates with an equivalent thickness and material properties as your diaphragm. You may choose a very small equivalent thickness which will still be semi-rigid, but much closer to flexible than rigid.

Flexible Diaphragms in a Combined RISAFloor/RISA-3D Model

A flexible diaphragm can be defined in a combined RISAFloor/RISA-3D model. For wood diaphragms, the program actually designs the sheathing and nail spacing, incorporating the code specified design tables from the NDS/IBC. See the [Diaphragms - Analysis & Results](#) topic for more information on how the design works and how loads are attributed both for wind and seismic loads. The diaphragm can be defined as flexible when drawing the deck edge or from the Diaphragms spreadsheet.

RISAFloor

RISA-3D

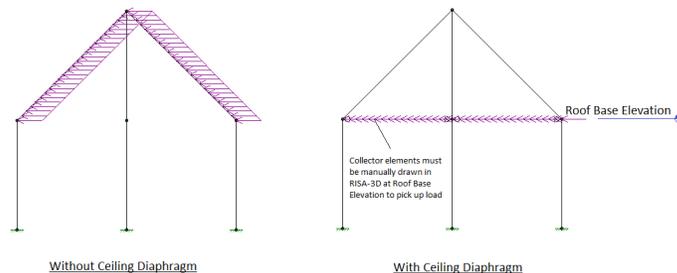
General	Elevation [ft]	Mass [k]	Mass MOI [k-ft ²]	Center Of Mass [ft]	Plus X Eccentricity [ft]	Minus X Eccentricity [ft]	Plus Z Eccentricity [ft]	Minus Z Eccentricity [ft]	Inactive	Diaphragm	Type	Region
1	10	4276.791	2.818e+7	130.261, 110.09	5.5	5.5	5.5	5.5	<input type="checkbox"/>	D2	Rigid	
2	15	3824.793	3.375e+7	118.371, 116.03	5.5	5.5	5.5	5.5	<input type="checkbox"/>	D3	Flexible	

Note: Flexible diaphragms cannot be defined on an elevated slab floor.

Flexible diaphragms within RISAFloor/RISA-3D are used solely as load-attribution devices, and do not exist as elements within the stiffness matrix, unlike rigid or semi-rigid diaphragms. Flexible diaphragms have no stiffness, and are incapable of transferring load from one element within them to another. Thus, members that do not have lateral resistance out of plane are susceptible to stability issues. To increase stability in a model with a flexible diaphragm, it may be necessary to add small springs in the out-of-plane directions on the frames at the tops of the columns. This will stabilize the members while generally having a minimal effect on the analysis of the model.

Ceiling Diaphragm (Sloped Roofs)

When a flexible diaphragm is applied to a sloped roof in RISAFloor, a [setting](#) exists to specify whether a ceiling diaphragm is present.



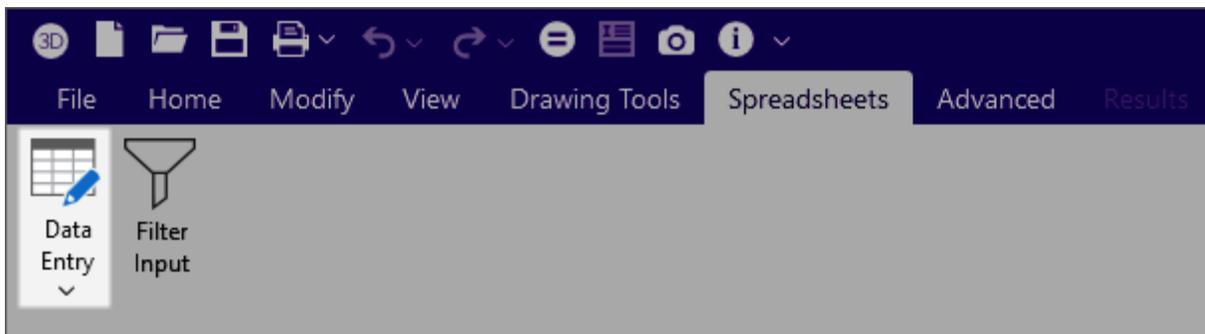
The load attribution for the flexible diaphragm occurs only for Lateral Members from RISAFloor when a ceiling diaphragm is *not* specified. This is illustrated above where the lateral load follows the rafters. When a ceiling diaphragm *is* specified, it is necessary to draw in collector beams at the ceiling elevation so that the load can be carried directly the columns or walls at the base of the sloped members. With a ceiling diaphragm, the sloped members will not experience any lateral load directly. Note that the same total amount of lateral force will be applied regardless of this setting. This only controls whether the lateral load is applied to the sloped members, or in a horizontal flat plane.

Diaphragms Spreadsheet - General Tab

The **Diaphragms** spreadsheet contains diaphragm information. For RISA-3D models there are two different versions of the Diaphragms spreadsheet; one for models which have been created in [RISA-3D](#) and another for models which are linked to [RISAFloor](#).

To open the Diaphragms spreadsheet:

1. Open the **Spreadsheets** ribbon, as shown in the following image.

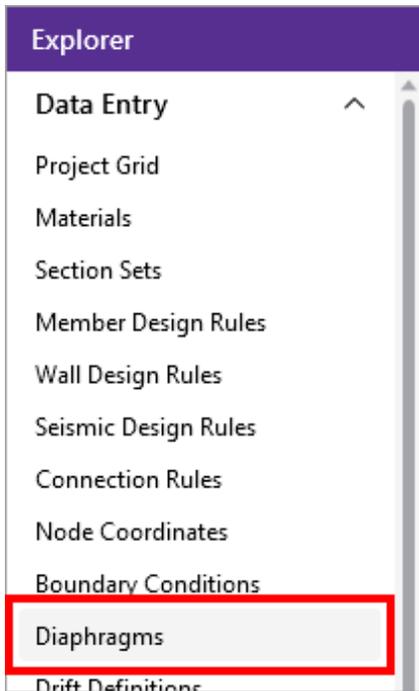


2. Click on **Data Entry**.
3. Click on the Diaphragms **Diaphragms** checkbox, to open the Diaphragms spreadsheet.

Rigid Diaphragms				
	Node Label	Plane	Inactive	No Wind/Drift
1	N2	ZX	<input type="checkbox"/>	<input type="checkbox"/>

Alternatively, you can access the spreadsheet by clicking on **Diaphragms** in the Explorer

panel, as shown in the following image.



RISA-3D Only

	Node Label	Plane	Inactive	No Wind/Drift
1	N14	ZX	<input type="checkbox"/>	<input type="checkbox"/>
2	N419	ZX	<input type="checkbox"/>	<input type="checkbox"/>

Rigid Diaphragms Spreadsheet

Column	Description
Node Label	The 'Node Label' column defines the primary node for the diaphragm. For diaphragms in the ZX plane, the diaphragm will be created at the Y-coordinate of the specified node. Similarly, XY diaphragms will use the node's Z coordinate, and YZ diaphragms will use the node's X coordinate.
Plane	The 'Plane' column indicates which plane the diaphragm is in.
Inactive	When the 'Inactive' checkbox is checked, the diaphragm is ignored by the program. This is a convenient way to disable diaphragms without deleting them.

Column	Description
No Wind/Drift	You may designate any floor level as a No Wind / Drift (i.e. mezzanine) level in order to omit it from the generated wind load calculation and the drift calculations .



Note:

- Only the in-plane rigidity is in play and not the out-of-plane rigidity.
- There are not semi-rigid or flexible options for RISA-3D as a standalone program.

RISA-3D Linked With RISAFloor

The following table lists the General tab settings and their descriptions.

General tab settings

Setting	Description
Elevation	Displays the elevation of the diaphragm. This is the same elevation as the floor which the diaphragm was created on.
Mass	Displays the dynamic mass tributary to the diaphragm. This mass is used to calculate seismic forces for both static and response spectra methods.
Mass MOI	Displays the dynamic mass moment of inertia of the diaphragm. This moment of inertia is used to calculate seismic forces for response spectra analysis.
Center of Mass	Displays the X and Z coordinates of the center of mass of each diaphragm. This is the location at which static (equivalent lateral force) seismic loads are applied for each diaphragm.
Eccentricities	Are the percent of length/width of the diaphragm which are used to place "accidental eccentric" seismic loads for each diaphragm. This only applies to the static (equivalent lateral force) procedure. See ASCE 7-16,

Setting	Description
	Section 12.8.4.2 for more information.
Inactive (checkbox)	When the Inactive box is checked, the diaphragm will be ignored by the program. This is a convenient way to disable diaphragms without deleting them.
Diaphragm	Displays the diaphragm label. This label is used for the naming of Diaphragm Regions.
Type	Specifies whether the diaphragm is Rigid (Membrane) or Flexible for a Beam Supported Floor or Rigid (Membrane) or Semi-Rigid for Concrete Floor Slab. If a diaphragm has been defined as Flexible within RISAFloor it can be toggled between Rigid and Flexible in RISA-3D.
Region	Lists the diaphragm regions for each diaphragm. Diaphragm regions are used for the design of wood flexible diaphragms, and are also useful for explicitly defining how flexible load attribution is to be performed.
Design Rule	Specifies the Design Rule which is assigned to each region. Only the information on the Diaphragms tab of the Design Rules Spreadsheet is considered.
SGAF	The 'Specific Gravity Adjustment Factor' for the design of wood flexible diaphragms. For more information see AF&PA NDS SDPWS, Table A4.2, Note 2. This value defaults to 1. However, it should be manually changed if the framing supporting the wood flexible diaphragm is not Douglas Fir-Larch or Southern Pine.
Material	The 'Material' column specifies the material properties of the diaphragm (when a semi-rigid diaphragm is selected).
Thickness (in)	The 'Thickness' column indicates the thickness of the diaphragm (when a semi-rigid diaphragm is selected).
No Wind/Drift	You may designate any floor level as a No Wind/ Drift (i.e. mezzanine) level in order to omit it from the

Setting	Description
	generated wind load calculation and the drift calculations .

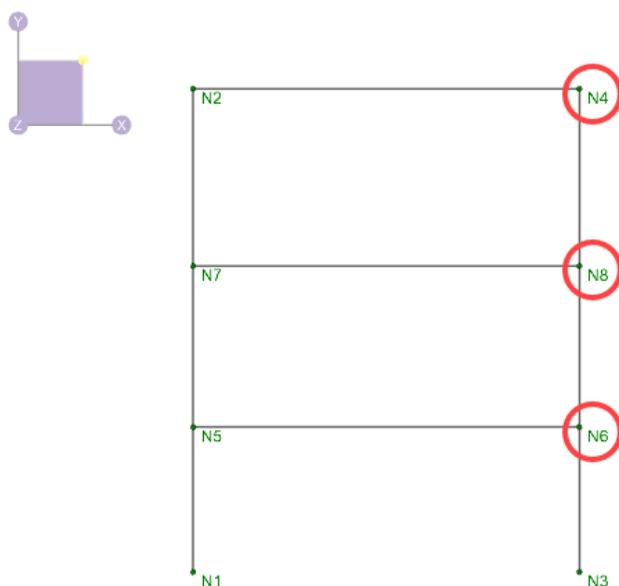
 **Note:** For information about the **Wood Diaphragms** tab see the [Diaphragms - Analysis and Results](#) topic

Diaphragm Modeling Tips

Modeling a Rigid Diaphragm in RISA-3D

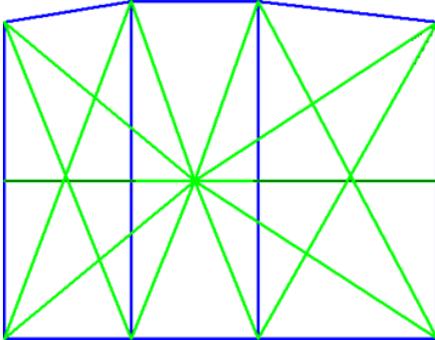
In the **Diaphragms** spreadsheet:

1. Enter the node in the Node Label column that the diaphragm will be defined by.
2. Designate the **Plane** the diaphragm will act.



	Node Label	Plane	Inactive	No Wind/Drift
1	N6	ZX	<input type="checkbox"/>	<input type="checkbox"/>
2	N8	ZX	<input type="checkbox"/>	<input type="checkbox"/>
3	N4	ZX	<input type="checkbox"/>	<input type="checkbox"/>

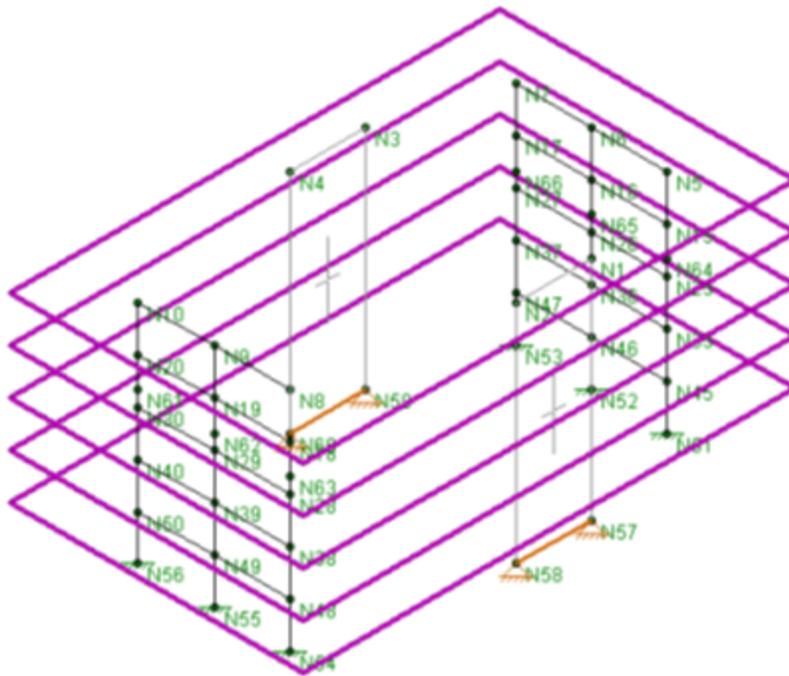
Internally, the Rigid Diaphragm ties all nodes at that elevation together with Rigid links.



To view the diaphragm:

1. Click the  **Toggle Display of Diaphragms** button .

This allows you to visually see the location of the diaphragms. Inactive diaphragms do not show up graphically.



2. (Optional) From here you can run the [Wind](#) and [Seismic](#) load generators.

Axial Load Limitation

Members in the plane of the diaphragm will have no axial load attributed to them. This is because the internal rigid links that are created to achieve rigid behavior take the entire load. In these cases, the axial loads in these members will need to be considered outside of the program.

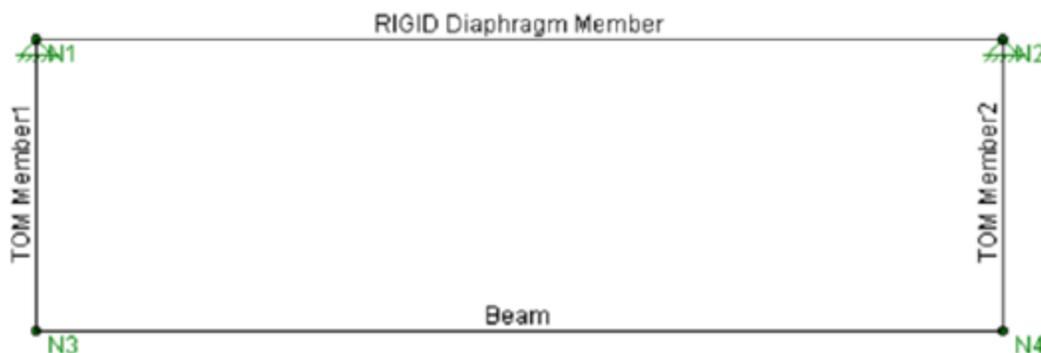
Openings in a RISA-3D Only Model

If there are openings within the defined diaphragm, where portions of the structure are not rigidly connected, it is possible to disconnect those locations from the diaphragm in the **Node Coordinates** spreadsheet.

	Label	X [ft]	Y [ft]	Z [ft]	Temp [F]	Detach From Diaphragm
1	N1	0	0	0		<input type="checkbox"/>
2	N2	0	12	0		<input type="checkbox"/>
3	N3	12	12	0		<input type="checkbox"/>
4	N4	12	0	0		<input type="checkbox"/>
5	N5	0	0	15		<input type="checkbox"/>
6	N6	0	12	15		<input type="checkbox"/>
7	N7	12	12	15		<input type="checkbox"/>
8	N8	12	0	15		<input type="checkbox"/>

Use of Rigid Diaphragms with the Top of Member Offset

Never use Top of Member Offsets with the diaphragm feature. This combination will almost surely make the forces in the member incorrect. This is because a Vierendeel-type truss is created, where the internal rigid links created by the diaphragm feature acts as a top flange and the member acts as the bottom flange. What now is drawn as a single member has multiple internal members influencing these forces.



Partial Diaphragms in a RISA-3D Only Model

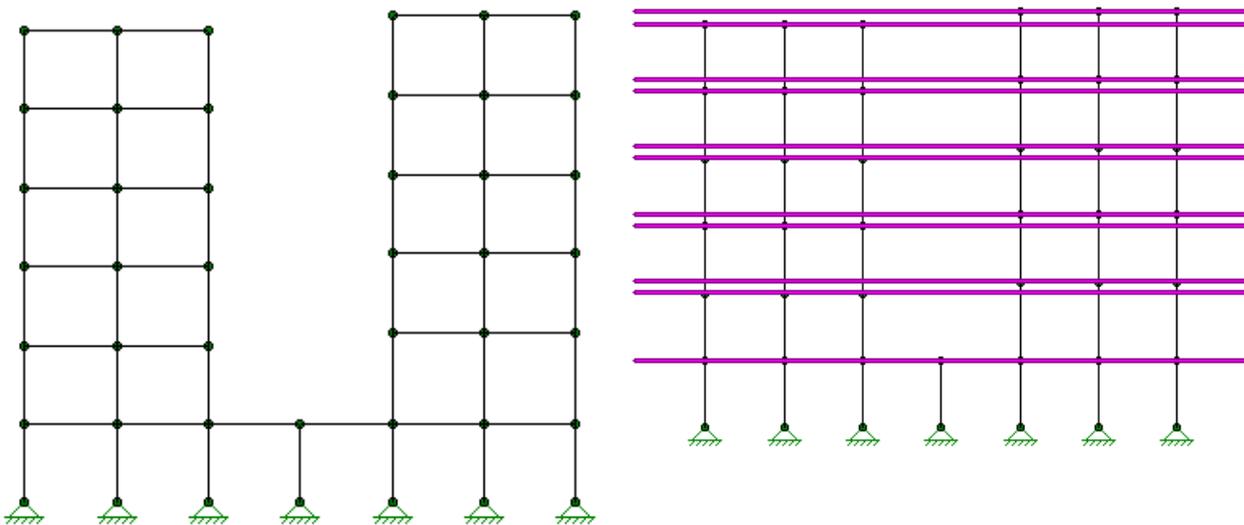
There may be times when you want to model a partial diaphragm, i.e., a diaphragm that extends over only a portion of a floor or plane. For example, let's say you are trying to model a floor that is composed of a relatively rigid section (thick concrete slab) and a relatively flexible section (corrugated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

To accomplish this, specify that a node or group of nodes be detached from the diaphragm.

To detach a node or group of nodes from the diaphragm:

1. Select the **Detach from Diaphragm** option in the Node Coordinates spreadsheet or click a node.
2. Specify the node in the **Information Coordinates** window.

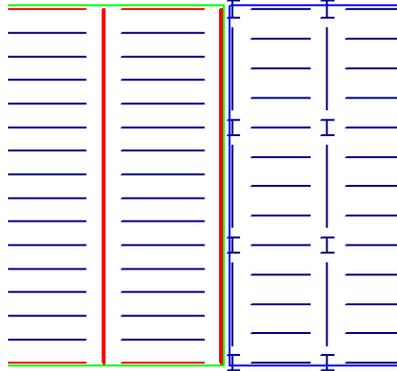
Another way this can be done is to offset the elevations of the nodes that comprise the rigid floor section so that they are all a little higher or a little lower than the surrounding floor. The offset only needs to be slightly larger than the Merge Tolerance, since this is the tolerance for other nodes to be on the same plane as the primary node. This works because the rigid diaphragm feature will only rigidly connect nodes that are at the same elevation as the primary node. The other nodes, which are on the flexible portion of the floor and are now at a different elevation than the primary node, will not be incorporated into the diaphragm. This can also be used for a "twin tower" situation where you want each tower to act independently of the other.



Partial Diaphragms in a Combined RISAFloor/RISA-3D Model

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

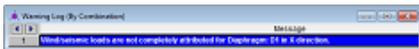
The following image is an example of the gap:



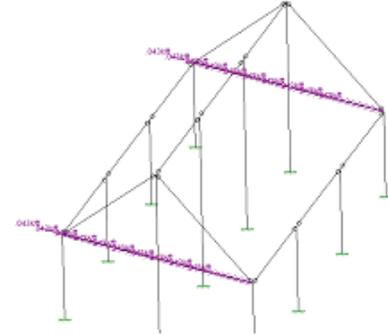
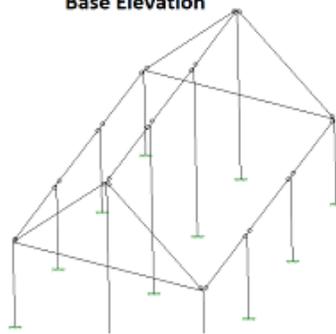
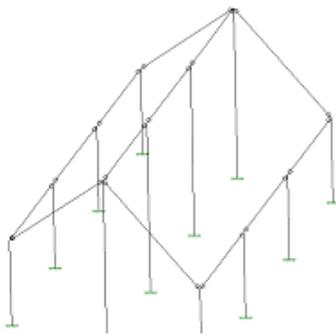
Sloped Roof Flexible Diaphragms (RISAFloor/RISA-3D Integration)

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

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



Add Horizontal Braces at Roof Base Elevation



For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Sloped Roofs**.

Flexible Diaphragms - Analysis and Results

Flexible diaphragms can only be designed in RISA-3D if they have come in from an integrated RISAFloor / RISA-3D model. This is because RISAFloor contains information about slab edges, deck direction and geometry that cannot be entered in RISA-3D. To get this design, however, you must create diaphragm regions in RISAFloor. See the **Diaphragms** topic in the [Flexible Diaphragms](#) section for more information.

Analysis / Loading For Flexible Diaphragms

Seismic Load

The program only applies seismic loads to flexible diaphragms that were generated using the [seismic load generator](#). The load is internally converted into a one way member area load where the direction of the load attribution is perpendicular to the direction of the applied load. This internal conversion is done at solution time. This area load then gets broken down into a series of "transient" distributed loads that are applied to the members which support the diaphragm. These transient loads can be viewed as Basic Load Cases. Refer to [Member Area Loads](#) for more information about how these transient loads are generated.

Wind Loads

The program only applies wind loads to flexible diaphragms that were generated using the [wind load generator](#). The load is internally converted into a distributed load at the front face of the diaphragm. Then it is broken into area loads of varying magnitudes depending on the depth of the diaphragm. In this way, an L shaped building will have less load per foot applied at the deep section of the L than the skinny section, since an equal amount of wind from each side is spread over more floor area in the deeper section.

These area loads then gets broken down into a series of "transient" distributed loads that are applied to the members which support the diaphragm. These transient load can be viewed as Basic Load Cases. Refer the section on [Member Area Loads](#) for more information about how these transient loads are generated.



Note: The shear capacities of the diaphragm panels will automatically be multiplied by 1.4 for wind forces if the [Wind ASIF](#) function is enabled.

Wood Diaphragm Design Rules

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

Floor Diaphragms								
General		Wood Diaphragms						
	Label	Database	Panel Grade	Min. Panel Thickness [in]	Max. Panel Thickness [in]	Min. Nail Spacing	Max. Nail Spacing	Nail Spacing Increment [in]
1	Floor...	IBC06-09 WS...	Rated She...	0.375	0.75	2-in.	2-in.	1
2	Roof D...	AWC 2015 OSB	Structural-I	0.375	0.469	2-in.	3-in.	1

The program uses your min/max panel thickness and nail spacing to choose an appropriate nailing and panel layout from the **Database** chosen. The database options can be viewed by clicking the  (ellipsis) button in the **Database** column.

Floor Diaphragms								
General								
Wood Diaphragms								
	Label	Database	Panel Grade	Min. Panel Thickness [in]	Max. Panel Thickness [in]	Min. Nail Spacing	Max. Nail Spacing	Nail Spacing Increment [in]
1	Floor...	IBC06-09 WSP Database	Rated She...	0.375	0.75	2-in.	2-in.	1
2	Roof D...	AWC 2015 OSB	Structural-I	0.375	0.469	2-in.	3-in.	1

Wood Diaphragm Schedule
✕

Select Manufacturer AWC 2015 OSB

Available HD Series

AWC 2015 OSB

AWC 2015 OSB Case 1_3U

AWC 2015 OSB Case 1_3B

AWC 2015 OSB Case 1_3 High Load

AWC 2015 OSB Case 2_4U

AWC 2015 OSB Case 2_4B

AWC 2015 OSB Case 2_4 High Load

AWC 2015 OSB Case 5_6U

AWC 2015 OSB Case 5_6B

AWC 2015 OSB Case 5_6 High Load

Current Selection Type Use Case

	Label	Case	Blocked	Framing Width [nominal]	Panel Grade
1	C2/4U_2_RS_5/16_6d@6/6/1	2/4	No	2	Structural-I
2	C5/6U_2_RS_5/16_6d@6/6/1	5/6	No	2	Structural-I
3	C2/4U_2_RS_3/8_6d@6/6/1	2/4	No	2	Structural-I
4	C2/4U_2_S1_5/16_6d@6/6/1	2/4	No	2	Structural-I
5	C2/4U_3_RS_5/16_6d@6/6/1	2/4	No	3	Structural-I
6	C5/6U_2_RS_3/8_6d@6/6/1	5/6	No	2	Structural-I
7	C5/6U_2_S1_5/16_6d@6/6/1	5/6	No	2	Structural-I
8	C5/6U_3_RS_5/16_6d@6/6/1	5/6	No	3	Structural-I
9	C2/4U_3_RS_3/8_6d@6/6/1	2/4	No	3	Structural-I
10	C2/4U_3_S1_5/16_6d@6/6/1	2/4	No	3	Structural-I
11	C5/6U_3_RS_3/8_6d@6/6/1	5/6	No	3	Structural-I
12	C5/6U_3_S1_5/16_6d@6/6/1	5/6	No	3	Structural-I
13	C1/3U_2_RS_5/16_6d@6/6/1	1/3	No	2	Structural-I
14	C2/4U_2_RS_3/8_8d@6/6/1	2/4	No	2	Structural-I
15	C5/6U_2_RS_3/8_8d@6/6/1	5/6	No	2	Structural-I
16	C1/3U_2_RS_3/8_6d@6/6/1	1/3	No	2	Structural-I
17	C1/3U_2_S1_5/16_6d@6/6/1	1/3	No	2	Structural-I
18	C1/3U_3_RS_5/16_6d@6/6/1	1/3	No	3	Structural-I
19	C2/4U_2_RS_7/16_8d@6/6/1	2/4	No	2	Structural-I
20	C5/6U_2_RS_7/16_8d@6/6/1	5/6	No	2	Structural-I

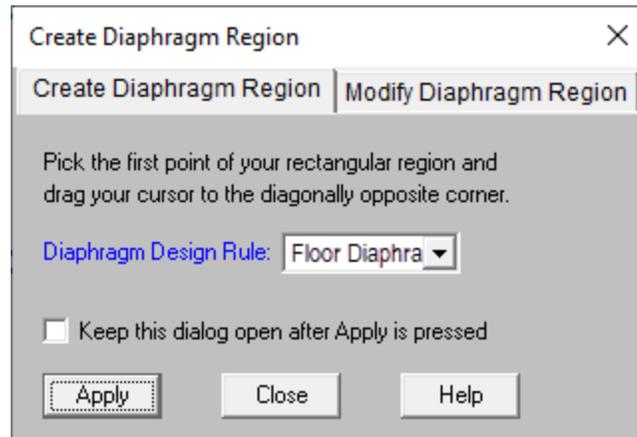
OK
Cancel

For more information on this, see the [Appendix F - Wood Shear Wall Database Files](#) topic.

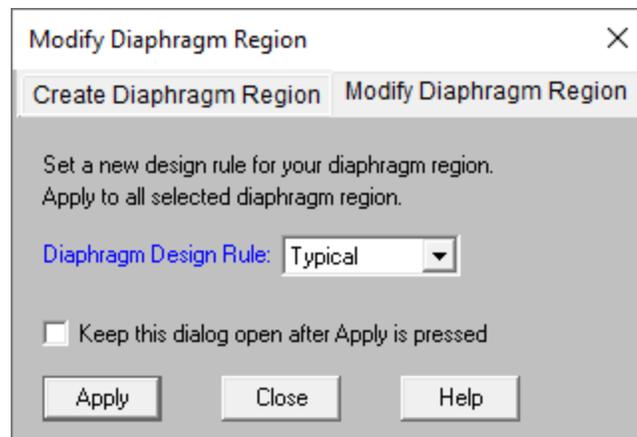
Diaphragm Regions

Diaphragm Regions are used for explicitly defining [load attribution](#), and for wood diaphragm design.

They are not *required* for flexible diaphragm analysis. To draw a diaphragm region, click on the button on the graphic editing toolbar.



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



The following are some limitations for drawing diaphragm regions:

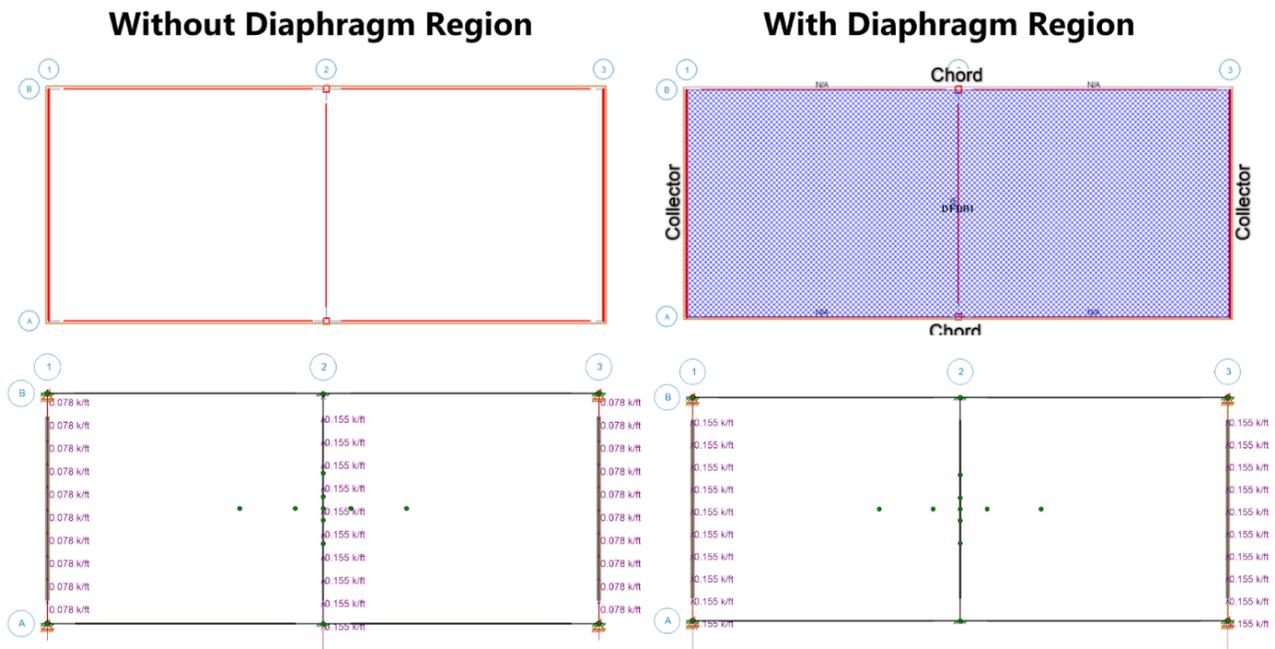
- Diaphragm Regions can only be drawn on flexible diaphragms. Therefore, a Flexible slab edge must be present around the portion of the floor where the diaphragm region is drawn.
- Diaphragm Regions must be rectangular.
- Diaphragm Regions must have a closed-circuit of **lateral** members around their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

Load Attribution

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

If Diaphragm Regions have not been drawn, loads are attributed to all horizontal members within the floor plane (except members perpendicular to load direction). Therefore, drag struts and collectors **MUST** be modeled in order to get proper load attribution into lateral frames.

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



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



Note:

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

For additional advice on this topic, please see the RISA Tips & Tricks website:
www.risa.com/post/support. Type in Search keywords: **Flexible Diaphragm Design**.

Diaphragms Spreadsheet - Wood Diaphragms Tab

Floor Diaphragms								
General		Wood Diaphragms						
	Label	Database	Panel Grade	Min. Panel Thickness [in]	Max. Panel Thickness [in]	Min. Nail Spacing	Max. Nail Spacing	Nail Spacing Increment [in]
1	R1	IBC 06-09 OS...	Structural-I	0.35	0.75	2-in.	6-in.	1



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

Database

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

Panel Grade

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

Max/Min Panel Thickness

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

Max/Min Nail Spacing

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

Nail Spacing Increment

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

Flexible Diaphragm Results - Detail Reports

Diaphragm results can be accessed by the Detail Report button  then clicking on the Flexible Diaphragm perimeter. This opens the Diaphragm Detail Report Key Plan, from which a diaphragm region can be chosen by clicking to review its results.

The Diaphragm Region detail report gives detailed information about the diaphragm design. It is split into four main sections: Input, Envelope Diagrams, Design Summary and Chord Force Summary.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Flexible Diaphragm Design**.

Input

Below is the input echo portion of the detail report:

Detail Report: D1 (Region DR1)

CRITERIA		GEOMETRY	
Code:	AWC NDS-18:ASD	Total Length (ft):	25
Design Rule:	Floor Diaphragm	Total Width (ft):	60
Panel Grade:	Other	L/W Ratio:	0.417
Panel Schedule:	IBC06-09 WSP Database	Elevation (ft):	15
SGAF:	1		

Input - Results

Result	Description
Code	Reports the code used to design the diaphragm.
Design Rule	Lists the Design Rule which has been assigned to this diaphragm region.
Panel Grade	Reports the grade of the panels used in the design (Structural-I, Rated Sheathing, or Other).
Panel Schedule	Reports the nailing schedule database used to optimize panel selection.
SGAF	The specific gravity adjustment factor, input in the Diaphragms spreadsheet, reduces the capacity of the diaphragm. Note 2 from Tables A.4.2A-A.4.2C of the NDS 2005 SDPWS gives the calculation of this value as: $SGAF = (1 - (0.5 - G))$, where G = Specific gravity of the floor framing members.

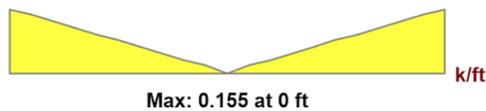
In addition the basic geometry information for the diaphragm (**Total Length, Total Width, Elevation, and L/W Ratio**) are also reported here.

Envelope Diagrams

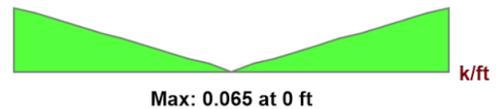
The detail report presents envelope diagrams for the shear and moment demand seen on the flexible diaphragm for both the strong and weak directions. The diagrams are color coded yellow for strong direction and green for weak direction for quick reference.

ENVELOPE DIAGRAMS

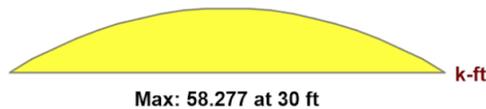
Strong Direction Shear Demand



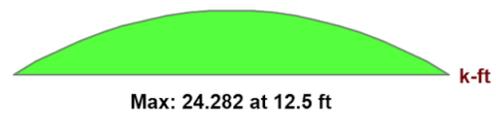
Weak Direction Shear Demand



Strong Direction Moment



Weak Direction Moment



Design Summary

The results for design summary are displayed for both strong and weak directions as shown below.

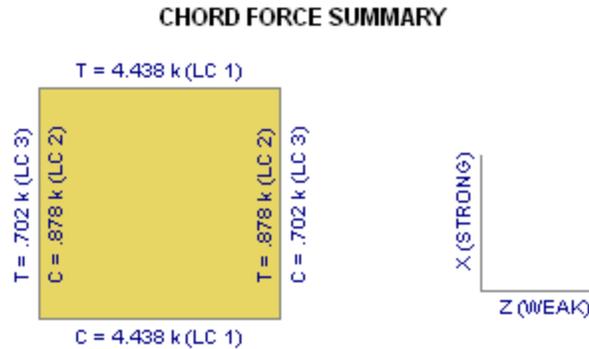
DESIGN SUMMARY			
STRONG DIRECTION		WEAK DIRECTION	
Panel Required Capacity (k/ft):	0.155	Panel Required Capacity (k/ft):	0.065
Panel Provided Capacity (k/ft):	0.16	Panel Provided Capacity (k/ft):	0.16
Ratio:	0.971	Ratio:	0.405
Governing LC:	1 (Seismic)	Governing LC:	2 (Seismic)

Design Summary - Results

Result	Description
Panel Required Capacity	The scaled maximum design shear, also reported on the envelope diagram. The maximum shear as determined from analysis is compared against F_p for seismic loads. If F_p exceeds the shear as determined by analysis then that shear is scaled up to match F_p . For more information see ASCE7-10, Section 12.10.1.1.
Panel Provided	The capacity of the designed diaphragm. For diaphragms with

Result	Description																												
<p>Capacity</p>	<p>multiple nailing zones, this will be reported as capacity at the point of maximum shear. For more information on how the program selects between the various nailing schedules provided in the database, please refer to Appendix F.</p>																												
<p>Ratio</p>	<p>Provides the code check for the diaphragm based on shear demand versus shear capacity.</p>																												
<p>Governing LC</p>	<p>Provides the load combination that controls the design. This is based on whichever load combination resulted in the highest ratio for shear capacity. In parentheses it is stated whether the governing load combination was seismic or wind based. If the panel was governed by wind, then you are allowed to use a 40% increase in panel strength. The program will automatically consider this, as long as you have checked the Increase Nailing Capacity for Wind checkbox in the Model Settings-Solution tab.</p>																												
<p>Deflection</p>	<p>The deflection of a diaphragm is comprised of two main terms, depending on the equation used.</p> <table border="1" data-bbox="553 1083 1406 1255"> <thead> <tr> <th colspan="4">UNSCALED DEFLECTIONS (Without Fp)</th> </tr> <tr> <th colspan="2">STRONG DIRECTION</th> <th colspan="2">WEAK DIRECTION</th> </tr> </thead> <tbody> <tr> <td>Maximum Shear (k/ft):</td> <td>0.155</td> <td>Maximum Shear (k/ft):</td> <td>0.065</td> </tr> <tr> <td>Flexure Component (in):</td> <td>0.029</td> <td>Flexure Component (in):</td> <td>0</td> </tr> <tr> <td>Shear Component (in):</td> <td>0.518</td> <td>Shear Component (in):</td> <td>0.09</td> </tr> <tr> <td>Total (in):</td> <td>0.547</td> <td>Total (in):</td> <td>0.09</td> </tr> <tr> <td>Governing LC:</td> <td>1</td> <td>Governing LC:</td> <td>2</td> </tr> </tbody> </table> <p>The terms are:</p> <ul style="list-style-type: none"> • The Flexure Component accounts for the deflection of the diaphragm based on bending stresses. Refer to Diaphragm Deflections for more information. • The Shear Component accounts for the deflection of the diaphragm based on shear stresses. Refer to Diaphragm Deflections for more information. <p>This deflection is based on the actual maximum shears in the diaphragm, which are given. It is not based on the scaled Fp forces, thus we report the unscaled shears.</p>	UNSCALED DEFLECTIONS (Without Fp)				STRONG DIRECTION		WEAK DIRECTION		Maximum Shear (k/ft):	0.155	Maximum Shear (k/ft):	0.065	Flexure Component (in):	0.029	Flexure Component (in):	0	Shear Component (in):	0.518	Shear Component (in):	0.09	Total (in):	0.547	Total (in):	0.09	Governing LC:	1	Governing LC:	2
UNSCALED DEFLECTIONS (Without Fp)																													
STRONG DIRECTION		WEAK DIRECTION																											
Maximum Shear (k/ft):	0.155	Maximum Shear (k/ft):	0.065																										
Flexure Component (in):	0.029	Flexure Component (in):	0																										
Shear Component (in):	0.518	Shear Component (in):	0.09																										
Total (in):	0.547	Total (in):	0.09																										
Governing LC:	1	Governing LC:	2																										

Chord Force Summary



The chord force summary gives an overall view of the chords on all sides of the diaphragm design region. The chord force calculation is simply the calculated moment divided by the length of the diaphragm (M/L). As the diagram above shows, we will give both tension and compression maximum values, if there are both tension and compression forces for the solved load combinations. The global axes are shown as a reference.

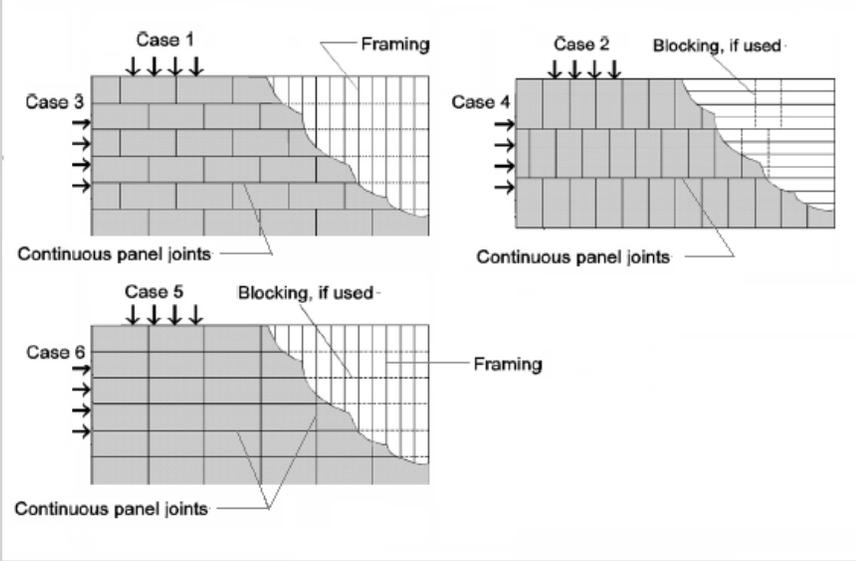
Design Details

The details of the designed panel are reported here:

DESIGN DETAIL			
Panel Label:	C2/4U_2_RS_3/8_8d@6/0/1	Panel Thickness (in):	0.375
Layout Case:	2/4	Required Nail Penetration (in):	1.375
Blocked:	No	Nail Size:	8 d

Design Detail - Results

Result	Description
Panel Label	This is the call-out that the program uses to describe the specific panel and nailing. This information is expanded upon afterward.
Layout Case	This reports the case (panel layout) used for the diaphragm design. Because each diaphragm has a strong and weak direction, the program will actually report two cases associated with each diaphragm.

Result	Description												
													
Blocked	This reports whether the diaphragm was designed as blocked or unblocked.												
Panel Thickness	<p>This reports the decimal thickness of the wood panels. For convenience the following table lists the decimals that correspond with common panel thicknesses:</p> <table border="1" data-bbox="553 1234 1409 1520"> <thead> <tr> <th>Panel Thickness</th> <th>Decimal Equivalent</th> </tr> </thead> <tbody> <tr> <td>5/16</td> <td>0.3125</td> </tr> <tr> <td>3/8</td> <td>0.375</td> </tr> <tr> <td>7/16</td> <td>0.4375</td> </tr> <tr> <td>15/32</td> <td>0.4688</td> </tr> <tr> <td>19/32</td> <td>0.5938</td> </tr> </tbody> </table>	Panel Thickness	Decimal Equivalent	5/16	0.3125	3/8	0.375	7/16	0.4375	15/32	0.4688	19/32	0.5938
Panel Thickness	Decimal Equivalent												
5/16	0.3125												
3/8	0.375												
7/16	0.4375												
15/32	0.4688												
19/32	0.5938												
Required Nail Penetration	This is the minimum nail penetration as required by the IBC/NDS. This value is taken from the diaphragm nailing schedule/database. For more information on the database refer to Appendix F .												
Nail Size	This is the size of the nail required for this call-out.												

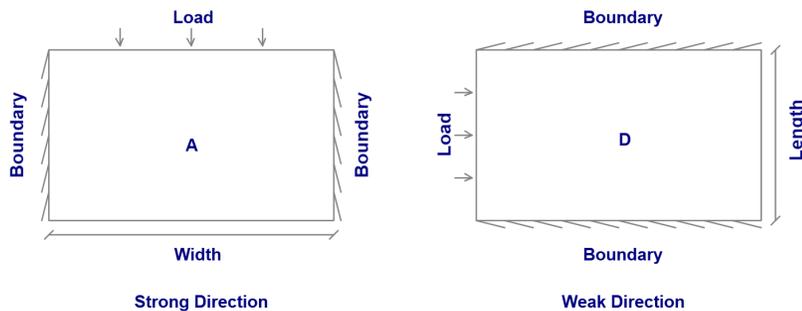
Nail Spacing Schedule

A diaphragm region is split into various nailing zones. The optimum nailing is chosen from the diaphragm nailing database for the highest point of shear of the diaphragm region in each direction. The program considers the Nail Spacing Increment as defined in the Design Rules. If a nailing arrangement is available that has spacing wider than the sum of the nailing increment and the spacing in the “higher” zone then a new nailing zone can be created. The arrangement of the nailing zones is such that they line up with the threshold shears of the capacities of the zones.

NAIL SPACING SCHEDULE

Zone	Location (ft)	Label	Lines	Framing Width (in)	Boundary (in)	Cont Edge (in)	Other Edge (in)
A	0	C2/4U_3_S1_15/32_10d@6/6/1	1	3	6	6	6
D	0	C2/4U_3_S1_15/32_10d@6/6/1	1	3	6	6	6

LAYOUTS



The nail spacing schedule reports the nail spacing for each zone. The zones are shown on a legend below.



Note: If there is only one zone defined in each direction, their default names will be A and D.

The following information is shown in the nail spacing schedule:

Nail Spacing Schedule

Column	Description
Zone	Provides the zone label so you can compare it to the zone layout below.
Location	Defines the distance to that zone from the edge of the diaphragm. Note that this is a symmetric distance from either side of the diaphragm.
Label	Provides the call-out for the diaphragm panel.
Lines	Provides the number of lines of nails on each panel edge. Normally, this

Column	Description
	value is greater than one only for High-Load diaphragms. This entry is taken directly from the diaphragm nailing schedule database and is not otherwise used in the diaphragm design.
Framing Width	Displays the nominal width required of the framing members that support the diaphragm panels. This entry is taken directly from the diaphragm nailing schedule database and is not otherwise used in the diaphragm design. <div style="border: 1px solid black; padding: 5px; background-color: #e0f0ff;">  Note: If there is only one zone, the location value will just be zero. </div>
Boundary	Provides the required boundary edge nail spacing.
Cont Edge	Provides the required nail spacing at continuous boundary edges that are parallel to the direction of load.
Other Edge	Provides the required nail spacing at all other edges. i.e. edges that are not considered to be boundaries or continuous edges parallel to load.

Collector Forces

The collector forces are not currently reported in the Detail Report. However, the applied seismic forces are considered in the normal member design of the collectors. The only issue here is that the seismic forces have not been amplified by the seismic overstrength factor (Ω).

Deflection Calculations

Flexure Component

If you think of the diaphragm as a beam with the chord members acting as the flanges, then this is the deflection due to the tension and compression forces that develop in the chords.

$$\delta_{flex} = \frac{5vL^3}{8EAW}$$

v = Maximum shear load in diaphragm region

L = Diaphragm dimension perpendicular to the direction of applied load

E = Chord modulus of elasticity

A = Chord cross-sectional area

W = Diaphragm dimension parallel to the direction of applied load

Shear Component

If you think of the diaphragm as a beam with the diaphragm sheathing acting as the web of the beam, then this component represents the deflection due to the shear forces in the sheathing. This term is based on the apparent shear stiffness (G_a) as described in the NDS *Special Design Provisions for Wind and Seismic*. As such, this term includes the effects of both elastic shear deformation of the sheathing and nail slip of the panels.

$$\delta_{shear} = \frac{0.25vL}{1000G_a}$$

v = Maximum shear load in diaphragm region

L = Diaphragm dimension perpendicular to the direction of applied load

G_a = Apparent diaphragm shear stiffness

For diaphragm regions that have only one zone in each direction the value of G_a is taken directly from the nailing database. For multi-zone diaphragm regions the program internally calculates an equivalent G_a using the formula below:

$$G_a = \frac{1.4v_s}{\frac{1.4v_s}{Gt} + 0.75\alpha e_n}$$

v_s = Diaphragm shear capacity (ASD value taken from nailing database)

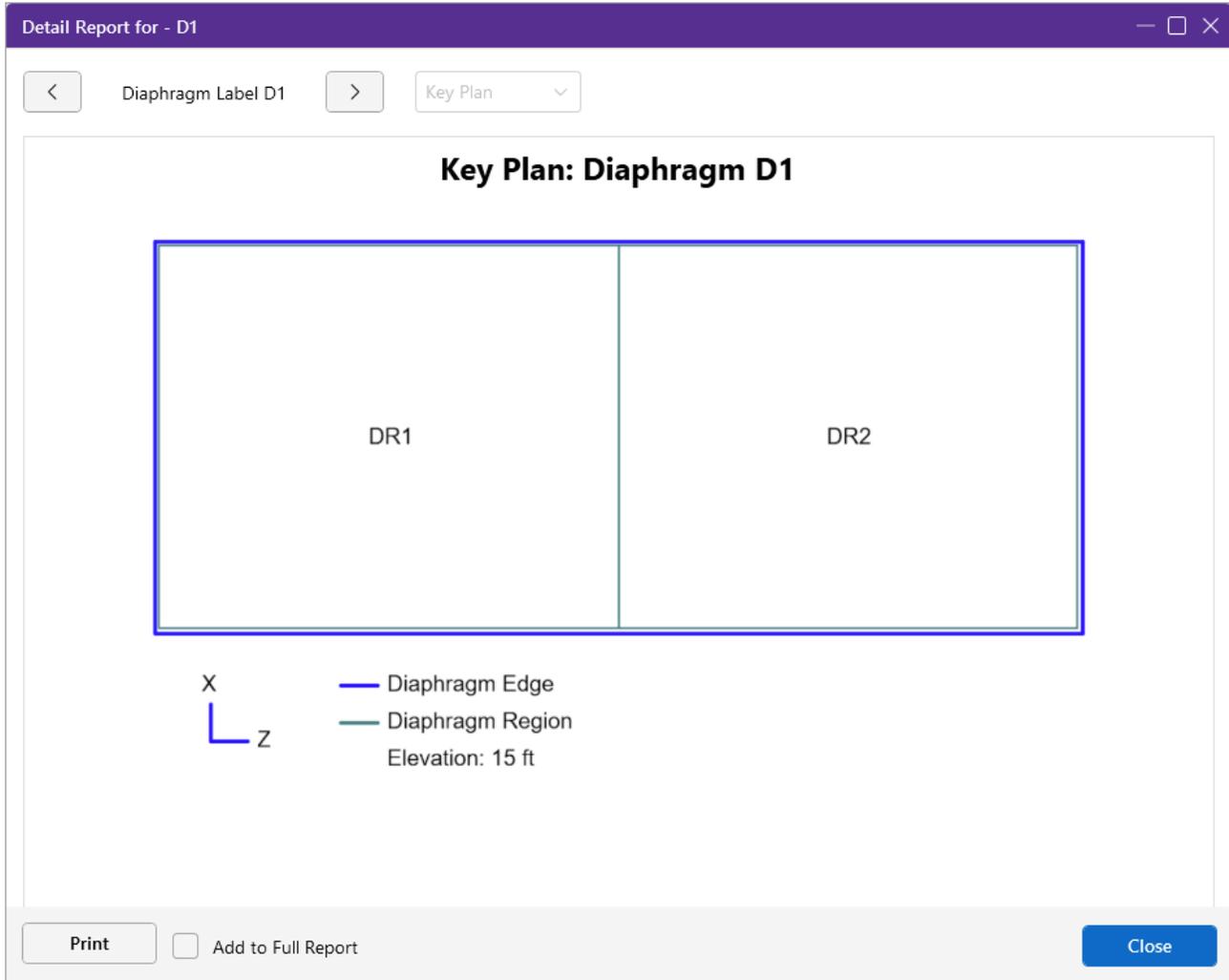
Gt = Shear stiffness of panel depth (taken from nailing database)

e_n = Nail slip (taken from nailing diaphragm nailing schedule / database)

α = Nail slip adjustment factor. This factor accounts for non-uniform nail slip across multiple zones, and is derived from the process outlined in the ATC-7 and further explained in the APA design / Construction Guide on Diaphragms and Shears Walls.

Diaphragm Key Plan

The diaphragm key plan is just a plan view of the diaphragms at a given level. The key plan can be accessed by clicking the Detail Report button  then click on the Flexible Diaphragm perimeter. This opens the Diaphragm Detail Key Plan, from which a diaphragm region can be chosen by clicking to review its results.



The key plan will list each diaphragm (D1, D2, etc.) and the diaphragm regions within (DR1, DR2, etc.). This key plan is to be used as a summary in RISA-3D for where the diaphragm regions are located within the floor plan. The elevation is also given.

In the future this key plan will give a summary of the diaphragm designs at this level.

Diaphragm Design Limitations

Diaphragm Design Limitations

Limitation	Description
Rigid Diaphragms	Currently, the program does not provide any design or code check information for Rigid diaphragms. The rigid diaphragms are solely used to attribute load to the supporting frames or walls.

Limitation	Description
Unblocked Diaphragm	The program will not optimize multiple nailing zones for unblocked diaphragms.
Chord Slip	The deformation due to chord slip is not considered in the deflections calculations.
Required Framing Size	The detail report lists a required framing nominal width, however no check is performed to ensure that the framing modeled actually meets this requirement
Diaphragm Loading	The only load seen by flexible diaphragms is the load coming directly from the automatically generated loading. Any loading added in RISAFloor or RISA-3D will not be considered in the design of the diaphragms and will not be attributed to the shear walls.
Sloped Diaphragms	Currently the program will not design sloped flexible diaphragms. At solution in RISAFloor the program will force you to change the diaphragm to rigid. This feature will be added in an upcoming release.
Moisture Content	The program is not considering a reduction in strength due to moisture content.
Orientation	No diaphragm design is done for regions placed over deck that is not parallel to the Global X and Z axes.
2021 SDPWS Provisions	2021 SDPWS provisions are not considered for flexible diaphragms.

Drift

You may calculate and report inter-story drift based on calculated node displacements. The calculations will be performed in the two horizontal translation directions, but not for the vertical direction.

Drift Definitions Spreadsheet

The Drift Definitions spreadsheet defines where the drift calculations will be performed. If the model contains diaphragms, the nodes connected to that diaphragm are automatically considered in the drift calculations. Drift calculations can be performed at specific elevations (where diaphragms don't exist). However, these elevations must be manually entered in the Drift Definitions spreadsheet. These elevations are defined with respect to the vertical axis of the model.

In addition, you can specify individual nodes to be used in drift calculation. Node definitions are included for a number of reasons. They allow models, defined in older versions of the program (which only used defined "story joints"), to produce identical drift results in the current version. They also allow you to get drift reporting at specific points of interest, even if these points don't line up well from floor to floor or don't behave the same as other nodes at that elevation. See [Drift Modeling Tips](#) for more information.



Note:

- Elevations and Diaphragms are treated interchangeably for drift calculations. Nodes must be in exactly the same location on the upper and lower floors for drift to be calculated.
- When Elevations are entered manually, there **MUST** be defined nodes at that elevation.
- When Node definitions are used, the drift calculation for the node is based only on the defined nodes. Therefore, you must also create a Node definition for the story below (unless it is at zero ft) in order to get correct behavior.
- The **Drift Definitions** spreadsheet is currently applicable to rigid diaphragms only.

	Type	Floor/Diaphragm	Node Label	Elevation [ft]
1	Diaphragm	Diaph.: 1	-	30
2	Diaphragm	Diaph.: 2	-	19
3	Diaphragm	Diaph.: 3	-	9

Inactive Diaphragms

Drift is not reported for elevations that have been flagged as "Inactive" on the Diaphragms Spreadsheet. However, the inactive diaphragm is still considered for the drift calculation for the floor above, in the determination of story height.

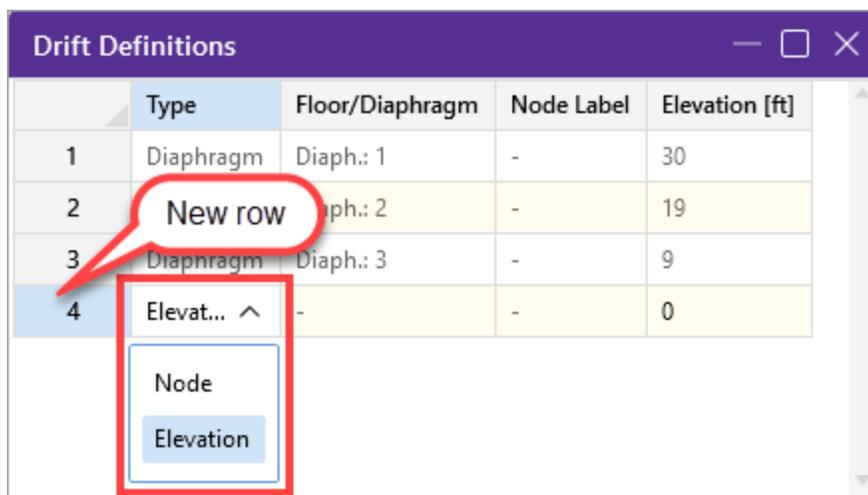
No Wind / Drift Checkbox

When the No Wind / Drift box is checked on the Diaphragms Spreadsheet, then that diaphragm will be completely ignored for drift calculations. Drift will not be reported at that level. It will also not affect the story height calculation for the floor above or below.

Define Elevation for Drift Calculation

To Define an Elevation for Drift Calculation:

1. Open the **Drift Definitions** spreadsheet from the 'Explorer' panel.
2. Add a new row to the spreadsheet:
 - a. Right-click in the spreadsheet to open an options menu.
 - b. Click on either **Add New Row to End** or **Insert Row Above Selected**, depending on where you want the new row to appear in the spreadsheet.
3. Click the **Type** column down-arrow in the new row.



	Type	Floor/Diaphragm	Node Label	Elevation [ft]
1	Diaphragm	Diaph.: 1	-	30
2	Diaphragm	Diaph.: 2	-	19
3	Diaphragm	Diaph.: 3	-	9
4	Elevat... ^	-	-	0

4. Click on **Elevation** in the list that appears.
This indicates the type of drift definition you are creating.
5. Click in the **Elevation [ft]** column of the new row and type the value for the elevation, as

shown in the following image.

Drift Definitions				
	Type	Floor/Diaphragm	Node Label	Elevation [ft]
1	Diaphragm	Diaph.: 1	-	30
2	Diaphragm	Diaph.: 2	-	19
3	Diaphragm	Diaph.: 3	-	9
4	Elevation	-	-	10



Note: The program assumes a default base elevation of 0. This will NOT be done if there are any drift definitions defined for elevations below 0.

Define Node for Drift Calculation

To Define a Node for Drift Calculation:

1. Open the **Drift Definitions** spreadsheet from the 'Explorer' panel.
2. Add a new row to the spreadsheet:
 - a. Right-click in the spreadsheet to open an options menu.
 - b. Click on either **Add New Row to End** or **Insert Row Above Selected**, depending on where you want the new row to appear in the spreadsheet.
3. Click the **Type** column down-arrow in the new row.

Drift Definitions				
	Type	Floor/Diaphragm	Node Label	Elevation [ft]
1	Diaphragm	Diaph.: 1	-	30
2	Diaphragm	Diaph.: 2	-	19
3	Diaphragm	Diaph.: 3	-	9
4	Node	-	N1	0

4. Click on **Node** in the list that appears.
This indicates the type of drift definition you are creating.

- Click in the Node Label column of the new row and type in the name, as shown in the following image.

	Type	Floor/Diaphragm	Node Label	Elevation [ft]
1	Diaphragm	Diaph.: 1	-	30
2	Diaphragm	Diaph.: 2	-	19
3	Diaphragm	Diaph.: 3	-	9
4	Node	-	N1	0

 **Note:** This node is included in the drift calculations for both horizontal deflections.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Story Drift**.

Drift Results

Open the Story Drift Spreadsheet

Once the solution is performed you may view the drift results in the ‘Story Drift’ spreadsheet.

To open the Story Drift spreadsheet:

- Ensure you have “Solved” the model so that the Results section appears in the Explorer panel.
- Click on **Story Drift** in the Explorer panel under the Results section.

This opens the Envelope Story Drift results report/spreadsheet to the first tab.

Envelope Story Drift - X-Direction, Service											
Z Direction Service		Z Direction Strength		X Direction Service			X Direction Strength				
	Story (Elevation)		Story Drift[in]	Loc (Z,X)	LC	Drift Ratio (%)	Loc (Z,X)	LC	2nd/1st Ratio	Loc (Z,X)	LC
1	Diaph.: 1 (24 ft)	max	-0.03	0, 0	19	0.354	0, 0	21	1.269	0, 0	21
2		min	-0.509	0, 0	21	0.021	0, 0	19	1.058	0, 0	19
3	Diaph.: 2 (12 ft)	max	-0.003	36, 45	19	0.004	36, 45	21	1.008	36, 45	21
4		min	-0.006	36, 45	21	0.002	36, 45	19	0.998	36, 45	20

This report lists the drift for all defined Diaphragms, Elevations and Nodes that exist in the Drift Definitions spreadsheet. The results are reported in the order in which they appear in the Drift Definitions spreadsheet.

3. Click on the **X Direction Service** tab (if not already open) to view its results.

Service level and strength level drifts are reported on different tabs of the Envelope Story Drift results spreadsheet. This is because seismic drift checks are usually checked purely for the strength level Load Combinations, while wind drift is usually checked against service level Load Combinations.

How Drift is Calculated

- **Inter-Story Drift:** To calculate inter-story drift for a particular direction, the displacement at the lower level is subtracted from the current story displacement.

For example, to calculate X direction drift for story 2, the X displacement for the node representing story 1 is subtracted from the X displacement for the node representing story 2.

- **Diaphragm and Elevation Definitions:** For Diaphragm and Elevation definitions, the drift calculations are only done at locations where columns or walls have nodes at the current level and at the level below.
- **NodeDefinitions:** For NodeDefinitions, the calculations are done for the current defined drift node compared to the nearest defined drift node at a the level below. Node definitions never look at the nodes in a Diaphragm or Elevation definition. These are user-defined drift nodes which act completely independent of the more automated Diaphragm and Elevation drift results and offer flexibility for cases where the Diaphragm or Elevation options don't give the output needed.
- **Story Height:** For story height, 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.
- **Structure Base Elevation:** The base elevation of the structure is assumed to be zero. If the 0 ft elevation should NOT be used as the base of the structure for drift calculations, then you should enter in an Elevation entry in the Drift Definitions spreadsheet to define the elevation of the base of the structure. This applies whether the base is a positive or negative value. If the value is positive, then there will be a reported drift (probably zero) at the base elevation that you can ignore.

Story Drift and Drift Ratio %

The Drift at a given level is equal to the deflection at that level minus the deflection at the level below. For the image below:

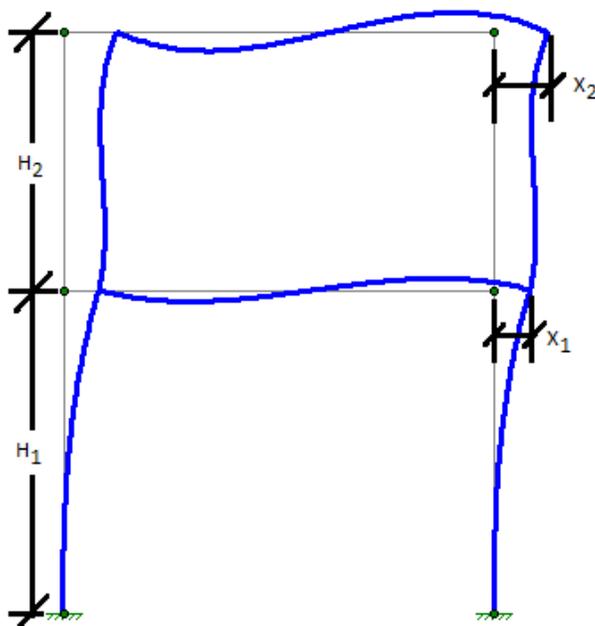
$$\text{1st Level Drift} = X_1$$

$$\text{2nd Level Drift} = X_2 - X_1$$

The Drift Ratio (%) is equal to the drift at that level divided by the height from that level to the level below. For the image below:

$$\text{1st Level drift ratio \%} = (X_1 / H_1) * 100\%$$

$$\text{2nd Level drift ratio \%} = ((X_2 - X_1) / H_2) * 100\%$$



Seismic Drift Checks

Seismic drift are reported for all codes for load combinations containing earthquake loads, but the program reports failures in red text for the ASCE 2005, 2010, and 2016 codes, based on the following table of allowable seismic drift versus Risk Category / Occupancy Category.

Drift Category	I or II	III	IV
High Drift Design	2.5%	2%	1.5%
Masonry Cantilever	1%	1%	1%
Other Masonry	0.7%	0.7%	0.7%
Other	2%	1.5%	1%

When the ASCE and IBC codes are used, the Story Drift for the strength level combinations accounts for the inelastic deflection (i.e. the C_d factor) by amplifying the node deflections at each level by C_d/I per Section 12.8.6 of ASCE-7. Where C_d is the Deflection Amplification factor intended to convert the elastic deflection levels given in the analysis to the inelastic levels used for seismic drift checks. Similarly, I is the importance factor (based on Risk / Occupancy Category) which was presumably used to amplify the seismic forces applied to the structure.

For codes other than the US codes, the drift results are not modified for C_d , I or ρ , and are never reported in red text to indicate a failure.

**Note:**

- These seismic checks are only performed for strength level load combinations.
- Stiffness reduction (per AISC Direct Analysis Method) does NOT change the drift code checks in RISA even though it may cause significant increase the reported drift.
- Load Combinations with Overstrength (Omega) seismic loading are not included in the drift reporting at all.
- To negate the effect of the redundancy factor (ρ), the node deflection results are divided by ρ before drift calculations are performed. This only occurs, of course, for load combinations where ρ was used to amplify the seismic loading. This is done because the redundancy factor is not intended to affect story drift calculations per section 12.3.4.1 item 2 of ASCE-7 2016.

2nd Order / 1st Order Deflection Ratio

The program reports the ratio of 2nd order deflection to 1st order deflection for the nodes used in the drift calculation. This is useful when determining whether it is required by the AISC code to use the Direct Analysis Method. In general, if the ratio is greater than 1.5 (or 1.7 if stiffness adjustment has been turned on) then you MUST use the Direct Analysis Method on your structure.

To facilitate this check, RISA colors the check in red whenever the ratio exceeds the 1.5 limit (or 1.7 when stiffness adjustment is used). This is done regardless of what has been chosen for the HR steel code. While the 1.5 or 1.7 may not be a code trigger in other HR steel codes, it remains because it can be an indicator of when 2nd order effects become troublesome.

There is an internal tolerance of 0.0005 inches, below which the deflection is not reported in drift results. This is because while the drift is essentially zero, testing showed cases where minor increases in these small values could result in high 2nd order / 1st order deflection ratios. Which were falsely indicating 2nd order effects as reaching high or troublesome levels.

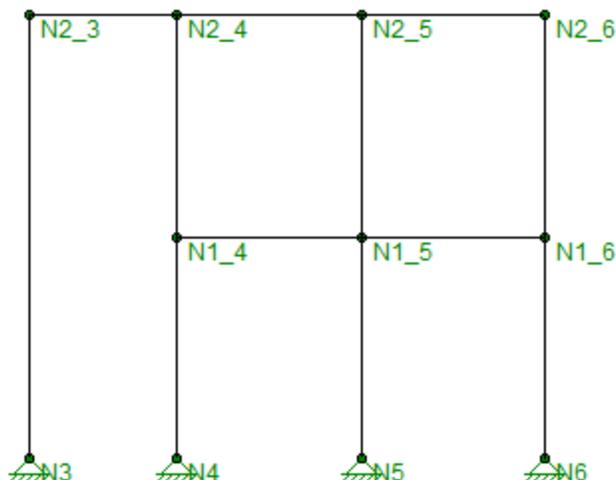
Drift Modeling Tips

Inclined Columns

When using diaphragm or elevations, the drift is reported for aligned nodes connected by columns or walls. This doesn't help for inclined columns because those nodes will very specifically NOT align from level to level. If drift reported is required for these nodes, then they should be defined using the Node option instead. When defined this way, the drift from level to level will be reported based on the nearest node at the level below.

Multi-Story Columns

In the image below, node N2_3 should have its drift calculated based on a story height that is twice what the other nodes at that level would be based on. In that case, the user should use Elevation or Diaphragm definitions for the majority of the nodes, but use Node definitions for N2_3.



Troubleshooting NC results

There are times when drift results are reported as "NC" (which stands for Not Calculate). This can occur for a number of reasons:

- The Story Drift for that floor is below the 0.005" minimum tolerance for reporting.
- If a diaphragm is flagged as inactive then all results for that diaphragm will report NC.
- When there are no aligned drift nodes between that diaphragm / elevation and the one below.
- The 2nd / 1st Ratio will report NC for any load combinations where a P-Delta analysis was not included in the solution.

DXF Files

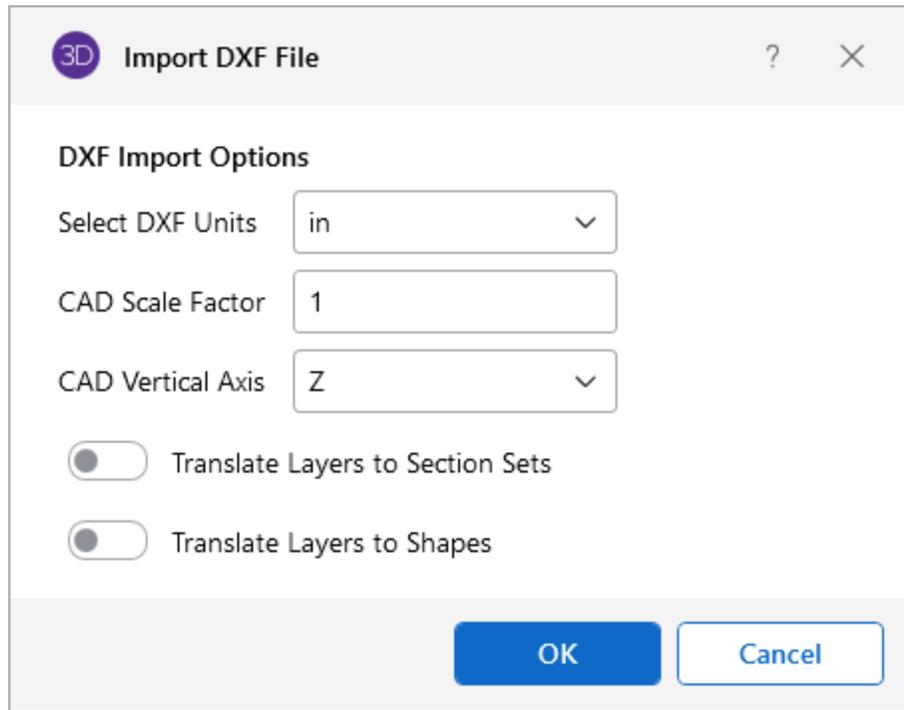
You can read and write export DXF files. Generally, you would read in a DXF file to create the geometry for a new structural model, or you could write out a DXF file from an existing model to form the basis for a model or a CAD drawing. This feature provides two-way compatibility with any other program that can read and write DXF files. This includes most major CAD programs and many analysis programs.

**Note:**

- Perform a Model Merge on any model created from a DXF file. See [Model Merge](#) for more information.
- You may want to round off the joint coordinates after importing a DXF file. You may do this from the Tools menu.
- When importing a DXF file it is essential to specify a column layer. Only beams that are fully supported will be imported.
- You can also import a DXF as a Drawing Grid. See the [Import DXF](#) section of the Graphic Editing topic for more information.

Import a DXF File

You can 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. can be present in the DXF file, but they are ignored. At this time, only the basic geometry is translated via DXF files. You have several options available for controlling how DXF files are imported.



The following table describes the options.

Import DXF File Options

Option	Usage / Description
Select DXF Units	<p>Allows you to choose the unit. Supported DXF units are: none, inches, feet, mm, cm or meters.</p> <p>You need to choose the same units as used in the CAD model from which you produced the DXF file.</p>
CAD Scale Factor	<p>Allows you to choose the scale factor that causes the DXF file to scale up or down in relation to the full scale. The default is 1.0</p> <p>For example, if you created a scaled model in AutoCAD at a scale of 1/4"=12", the appropriate scale factor to produce a full size RISA-3D model would be 48.</p>
CAD Vertical Axis	<p>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. Whereas, the default vertical axis in RISA is usually the positive Y-axis, which can be specified in ' Model Settings' under the Axis tab.</p> <p>With this option, when you import your model from a DXF file, you can tell the program to automatically rotate your geometry so that the Y axis is now the vertical axis for your RISA model.</p>

Option	Usage / Description												
Translate Layers to Section Sets	<p>This is a Yes/No slider. If you drag the slider to the right (Yes), the program translates the DXF file's layer names into RISA Section Sets Labels. The program requires that you add a prefix to each Layer Name to designate what type of material that section set is defined for. The prefixes are as follows:</p> <table border="1" data-bbox="618 554 1406 842"> <thead> <tr> <th data-bbox="618 554 1000 642">Material Type</th> <th data-bbox="1000 554 1406 642">Layer Prefix</th> </tr> </thead> <tbody> <tr> <td data-bbox="618 642 1000 684">Cold Formed Steel</td> <td data-bbox="1000 642 1406 684">CF_</td> </tr> <tr> <td data-bbox="618 684 1000 726">Concrete</td> <td data-bbox="1000 684 1406 726">CN_</td> </tr> <tr> <td data-bbox="618 726 1000 768">Hot Rolled Steel</td> <td data-bbox="1000 726 1406 768">HR_</td> </tr> <tr> <td data-bbox="618 768 1000 810">General Materials</td> <td data-bbox="1000 768 1406 810">GN_</td> </tr> <tr> <td data-bbox="618 810 1000 842">Wood</td> <td data-bbox="1000 810 1406 842">WD_</td> </tr> </tbody> </table> <p>For example, let's say you have designed a structure with Hot Rolled steel section sets that you want to call "Column" and "Girder", as well as a Wood section set called "Joist". If you do not prefix your section sets then they are all imported as General Material section sets. To have them imported into the proper Material type the column layer would have to be named "HR_Column", the girder a layer "HR_Girder" and the joists layer "WD_Joist".</p>	Material Type	Layer Prefix	Cold Formed Steel	CF_	Concrete	CN_	Hot Rolled Steel	HR_	General Materials	GN_	Wood	WD_
Material Type	Layer Prefix												
Cold Formed Steel	CF_												
Concrete	CN_												
Hot Rolled Steel	HR_												
General Materials	GN_												
Wood	WD_												
Translate Layers to Shapes	<p>This is a Yes/No slider. If you drag the slider to the right (Yes), the program takes members and assigns them to a shape based on their shape label. If no database shape corresponds to the DXF Layer Name, then these members are assigned a general RE4x4 shape or a BAR2.</p>												



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 automatically converts the "_" to a "." when the DXF file is read in.

Export a DXF File

Only the node, member, wall and element geometry are translated and used to create an ASCII DXF file. Any other information such as the boundary conditions, loads, member end releases, etc. are not translated at this time. You have several options available for controlling how DXF files are exported.

The following table describes the available options.

Export DXF File Options

Option	Usage / Description
Select DXF Units	Select the units you want the CAD model to be created in. The options for the DXF units are none, inches, feet, mm, cm and meters.
CAD Scale Factor	Enter the scale factor that causes 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

Option	Usage / Description												
	<p>is in the DXF units, which is defined below. The distance is used as entered and is not scaled by the CAD Scale factor.</p> <p>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 be separated by the standoff distance).</p>												
CAD Vertical Axis	<p>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.</p> <p>The default vertical axis in RISA is usually the positive Y-axis and may be specified on the Model Settings. When you export your model to a DXF file, you can have the program automatically rotate your geometry to match the vertical axis of your CAD program.</p>												
Joint Label	<p>Type the name of the label for the joint point entities. If you don't enter anything, the default label name is "JOINT".</p>												
Member Label	<p>Type the name of the label for the line entities. If you don't enter anything, the default label name is "MEMBER". Note that this entry is ignored if you select the 'Translate Shapes to Layers' option to translate section set database shape names to layer names.</p>												
Plate Label	<p>Type the name of the label for the plate elements, which will be represented as 3DFACE entities. If you don't enter anything, the default label name is "PLATE".</p>												
Translate Section Sets to Layer Names	<p>This is a Yes/No slider. If you drag the slider to the right (Yes), the program translates Section Set Labels to layer names. Layers are created in the DXF file corresponding to the section set labels in the RISA model. A "Yes" choice here overrides any layer name entered for the member layer.</p> <p>The program will add a prefix to each section set layer to designate what type of material that section is. The prefixes are as follows:</p> <table border="1" data-bbox="618 1591 1409 1885"> <thead> <tr> <th data-bbox="623 1591 1013 1675">Material Type</th> <th data-bbox="1013 1591 1409 1675">Layer Prefix</th> </tr> </thead> <tbody> <tr> <td data-bbox="623 1675 1013 1724">Cold Formed Steel</td> <td data-bbox="1013 1675 1409 1724">CF_</td> </tr> <tr> <td data-bbox="623 1724 1013 1772">Concrete</td> <td data-bbox="1013 1724 1409 1772">CN_</td> </tr> <tr> <td data-bbox="623 1772 1013 1820">Hot Rolled Steel</td> <td data-bbox="1013 1772 1409 1820">HR_</td> </tr> <tr> <td data-bbox="623 1820 1013 1869">General Materials</td> <td data-bbox="1013 1820 1409 1869">GN_</td> </tr> <tr> <td data-bbox="623 1869 1013 1917">Wood</td> <td data-bbox="1013 1869 1409 1917">WD_</td> </tr> </tbody> </table>	Material Type	Layer Prefix	Cold Formed Steel	CF_	Concrete	CN_	Hot Rolled Steel	HR_	General Materials	GN_	Wood	WD_
Material Type	Layer Prefix												
Cold Formed Steel	CF_												
Concrete	CN_												
Hot Rolled Steel	HR_												
General Materials	GN_												
Wood	WD_												

Option	Usage / Description
	For example, let's say you have designed a structure with Hot Rolled steel section sets called "Column" and " Girder", as well as a Wood section set called "Joist". If you type in a member layer name such as "STEEL" then all members, regardless of size, will appear on a layer named "STEEL". However, if you choose "Yes" for the <i>translate section sets to layers</i> option, then all the member that are assigned to the Column section set will appear on a layer named "HR_Column", the girders on a layer named "HR_Girder", and the joists on a layer called "WD_Joist".
Translate Shapes to Layer Names	This is a Yes/No slider. If you drag the slider to the right (Yes), the program takes members and assigns them to a layer which uses their shape label as the layer name. Layers are created in the DXF file corresponding to the shape labels in the RISA model. A "Yes" choice here overrides any layer name entered for the member layer.

**Note:**

- If BOTH the *Translate Section Sets to Layer Names* and *Translate Shapes to Layer Names* options are turned on (set to 'Yes'), only the explicitly defined shapes are placed on layers according to their shape labels. All members defined with section sets are still placed on layers according to their section set label.
- Please note that if the section set database shape designation includes one or more decimal point (".") characters, the export translates each occurrence of a decimal point character into an underscore ("_") character. For instance, a section set or shape label such as C10X15.3 will translate into a layer name of "C10X15_3" or "HR_C10X15_3".
- The DXF format does not properly recognize certain ASCII text characters for layer names (< > / \ " ; ? * | = '). Therefore, these characters should be avoided for shape or section sets when using the "translate to layer names" options.

Merge After a DXF Import

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 performing a **Model Merge**. This feature also deals 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. See [Model Merge](#) for more information.

DXF Element Numbering

Different CAD packages handle ordering of geometric data in their DXF files using two basic methods

Method 1

Entities are written out into the DXF file based on the order in which they were created in 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 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 it's own line. Each particular item is stored as two records, the first record is a group code and the second record is the data or a keyword. RISA only reads the ENTITIES section.

Group Codes

Each 2 record items start with an integer group code. RISA recognizes the following group codes:

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

First and Last Records for a DXF file

Each DXF file must start with the first record as the group code "0". The 2nd record must be the keyword "SECTION". Each DXF file must have the 2nd to last record as the group code "0". The last record must be the keyword "EOF".

Entities Section

The ENTITIES section is identified by a group code of "0", followed in the next record by the keyword "SECTION". The next record is 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 starts with a group code record of 8, followed by a record with the actual layer name.

The coordinates for the point is 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 in the POINT data but they are ignored.

The LINE format is started by a group code of “0” followed by the keyword “LINE”. The layer name for the LINE starts with a group code record of 8, followed by a record with the actual layer name.

The coordinates for the first point are started by the 10, 20 and 30 group codes respectively for the X, Y and Z coordinates. The coordinates for the second are 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 in the LINE data, but they are 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 starts 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 are started by the 10, 20 and 30 through 14, 24 and 34 group codes respectively. Other group codes and data may be present in the 3DFACE data but they are ignored.

AutoCAD Layer Names

The only valid characters in an AutoCAD layer name are letters A to Z, numbers 0 to 9, and the characters: dollar sign “\$”, underscore “_”, and dash “-”.

Dynamic Analysis

The Dynamic (Modal) Analysis uses the mass and stiffness matrices to calculate natural frequencies and natural modes of vibration for a structure, due to free or unforced vibration. The analysis calculates the modes and frequencies of vibration for the model.

The following sections provide more detailed information regarding the Dynamic Analysis and related processes.

- [Eigensolution](#) - Eigensolution refers to the process used to calculate the modes for a Dynamic Analysis.
- [Response Spectra](#) - A Response Spectra Analysis uses the Dynamic Analysis frequencies to calculate forces, stresses and deflections in the model.
- [Time History](#) - A Time History Analysis uses the mass, stiffness and damping matrices to solve for the forced vibration of the structure due to an applied load that varies with time.

Dynamic Analysis - Eigensolution

The dynamic analysis calculates the modes and frequencies of vibration for the model.

“Eigensolution” refers to the process used to calculate the modes. The frequencies and mode shapes are referred to as eigenvalues and eigenvectors. Refer to the [Program Limits](#) section for information on the maximum number of modes that can be solved for in RISA. The program can also solve for an approximate type of eigenmode called a [Ritz Vector](#).



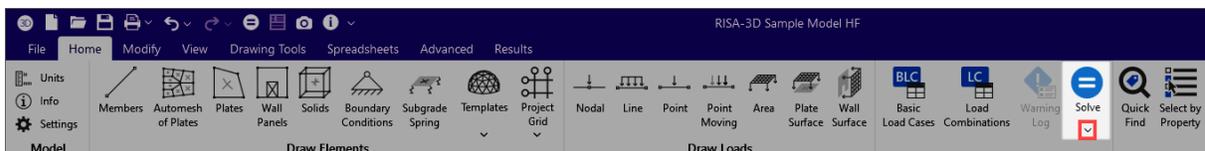
Note: The Dynamic Analysis is a prerequisite to the Response Spectra analysis, which uses the Dynamic Analysis frequencies to calculate forces, stresses and deflections in the model. For more information, see [Dynamic Analysis - Response Spectra](#).

Perform a Dynamic Analysis / Eigensolution

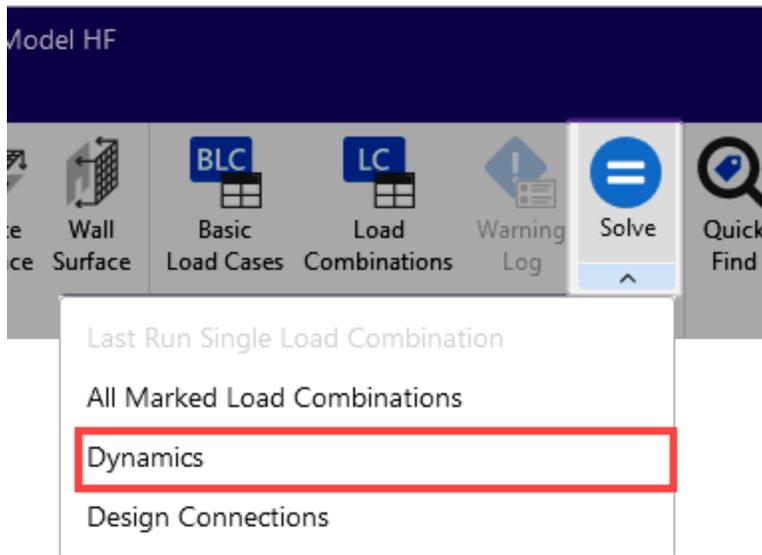
To Perform a Dynamic Analysis / Eigensolution:

Suggestion: You may wish to solve a static analysis first to verify that there are no instabilities.

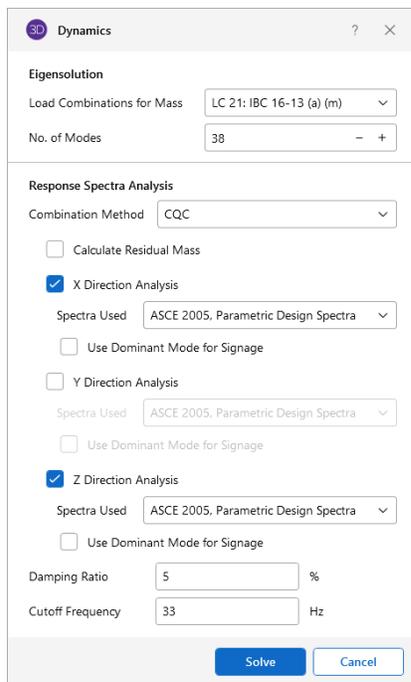
1. Go to the **Home** ribbon.



2. Click the **Solve** icon down-arrow.



3. Choose **Dynamics** from the solution options that appears, to open the Dynamics window.



4. Click the **Load Combinations for Mass** down-arrow and choose the load combination to use as the mass and the number of modes to solve.

The dynamic analysis uses a lumped mass matrix with inertial terms. Any vertical loads that exist in the **Load Combination for Mass** is automatically converted to masses based on the acceleration of gravity entry on the Solution tab of the Model Settings window. However, you must always enter the *inertial* terms as Nodal Masses.

**Note:**

- You can view the mode shapes graphically by choosing this option from the Model Display Options window.
- For information about the solver options (Standard, Accelerated and Ritz Vectors) see the Model Settings - Solution tab [Advanced](#) options. For documentation information on these solvers see the [Solution](#) section.
- For more information on that type of dynamic analysis, see the [Dynamic Analysis - Response Spectra](#) section.
- For more information about the **Calc Residual Mass?** checkbox, see the [Residual Rigid Response](#) section.
- For more information about the **Use Dominant Mode for Signage** checkbox, see the [Dynamic Analysis - Response Spectra](#) section.

Eigensolutions In a Combined RISAFloor/RISA-3D model

When a dynamic solution is run from within a combined RISAFloor / RISA-3D model, the Eigensolution portion of the dialog includes eccentric mass and Floor Diaphragm mass options as described in [RISAFloor Diaphragm Mass Options](#).

Required Number of Modes

You can 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](#) topic.

The catch is you first have to do a dynamic analysis in order to know how much mass is participating, so this becomes a trial and error process. First, pick an arbitrary number of modes (5 to 10 is usually a good starting point) and solve the RSA. If you have less than 90% mass, you'll need to increase the number of modes and try again. Keep in mind that the more modes you request, the longer the dynamic solution will take.



Note: If you are obtaining many modes with little or no mass, they are probably local modes. Rather than asking for even more modes and increasing the solution time see [Dynamics Troubleshooting – Local Modes](#) to learn how to treat the unwanted modes.

Dynamic Mass

The eigensolution is based on the stiffness characteristics of your model and also on the mass distribution in your model. There must be mass assigned to be able to perform the dynamic analysis. Mass may be calculated automatically from your loads or defined directly.

In order to calculate the amount and location of the mass contained in your model, RISA takes the vertical loads contained in the load combination you specify for mass and converts them using the acceleration of gravity defined in the [Model Settings](#). The masses are lumped at the joints and applied in all three global directions (X, Y and Z translation).

You can also specify mass directly. This option allows you to restrict the mass to a direction. In addition, you can apply a mass moment of inertia to account for the rotational inertia effects for distributed masses. See [Loads - Nodal Load / Displacement](#) to learn more about this.

**Note:**

- Only the VERTICAL loads (including vertical components of inclined loads) contained in the load combination are converted to mass.
- [The Model Settings - Axis](#) tab is where you can designate which of the three global axes is to be considered 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 for mass.
- You may want to move the mass to account for accidental torsion, which is discussed in the next section.

RISAFloor Diaphragm Mass Options

Models that get generated by RISAFloor automatically assign dynamic mass and mass moments of inertia (MMOI) to each of the RISAFloor diaphragms. This mass and MMOI is based on the self weight and loading information defined in RISAFloor and specified in RISAFloor's Model Settings.

There are 4 different sets of Mass defined for each diaphragm. The **Centered Mass** solution is when the mass is located at the mass centroid of the diaphragm. The **Plus X**, **Minus X**, **Plus Z**, and **Minus Z** solutions are when the mass has been offset to account for accidental eccentricity as specified in the [Floor Diaphragms](#) spreadsheet.

Whether this diaphragm mass / mass moment of inertia is included in the definition of dynamic mass is determined by the **Include Floor Diaphragm Mass** checkbox.

If the **Include Floor Diaphragm Mass** option is included in the analysis, then a Load Combination in most cases should not be considered for mass. The reasons you may wish to check the **Include Load Combination** box are:

- If you have RISA-3D only elements that you want to also include their mass in the eigensolution.

To do this you would need to create a load combination that either contains only this additional seismic load, or that one that includes ALL seismic weight. If you choose to include ONLY the additional seismic weight in this load combination then you would check both the **Include Floor Diaphragm Mass** and the **Include Load Combination**

checkboxes. If you include ALL seismic weight in your load combination then you would only choose the **Include Load Combination** checkbox.

- If you had a specific reason to not wish to use the RISAFloor calculated masses.

In that case you would uncheck **Include Floor Diaphragm Mass** and have only the **Include Load Combination** checkbox checked.

Eccentric Mass Solutions (for Accidental Torsion)

The eccentric mass options give an automated way to consider accidental torsion.

When more than one eigensolution option is selected (centered mass or one of the eccentric mass options), the program solves for a different set of frequencies and mode shapes for each of the mass options included in the solution. For information on how this affects response spectra analysis results, see [RISAFloor Spectra Results](#).



Note: The eccentric mass solution options have no effect on semi-rigid or flexible diaphragms.

Modeling Accidental Torsion

Most design codes require an assumed accidental torsion that is in addition to the natural torsion created by the location of mass with respect to rigidity. While RISA-3D calculates the natural torsion you may want to model additional *accidental* torsion. This can be done easily by taking advantage of the rigid diaphragm feature.



Note: Review the specific requirements of the building code to confirm this. But, most codes allow you to neglect accidental torsion for dynamic analysis and response spectra.

If you have modeled the dynamic mass at the center of mass only, then you may simply move the 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 spectra. You can then run the dynamic/rsa solution and combine the results with a static solution to check your members and plates for adequate capacity. You need to do this by running solutions for all four directions to capture the controlling effects on all frames. You will not be able to envelope your results since you are changing the dynamic results each time you move the mass. This means you'll probably want to check all your load conditions one additional time after all your member sizes work to make sure that any force redistribution in your frames hasn't caused other members to fail.

Note that when you lump all your floor mass to the center of mass, you should also enter a Mass Moment of Inertia for your diaphragm as well by applying it as a Nodal Mass to the center of mass node. The rotational inertial effects of the diaphragm mass will contribute to your torsion shears and should not be ignored in most cases. See [Loads - Nodal Load / Displacement](#) to learn more about this.

If you have not modeled the mass at discrete points that can be easily moved, then you have to apply the accidental torsion as a static load that is be part of a static analysis solution which includes the response spectra or equivalent lateral force procedure results. The magnitude of the torque is 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 have already calculated your story forces. If you are performing a response spectrum analysis, you can get the story forces exactly as the difference between the sum of the shears below and above the floor. Alternatively, you could 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 can be unconservative if your floor accelerations have large amplifications as compared to your base acceleration due to the dynamic characteristics of your building.

The torsion can be applied as a point torque that you can apply to a 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 in one solution run.



Note: For rigid diaphragms in a combined RISAFloor/RISA-3D model, the accidental eccentricity is automatically accounted for see the [Eccentric Mass Solutions](#) section for more information.

Eigensolution Convergence

The eigensolution procedure for dynamic analysis is iterative, i.e. a guess is made at the answer and then improved upon until the guess from one iteration closely matches the guess from the previous iteration. The tolerance value is specified in the [Model Settings](#) and indicates how close a guess needs to be to consider the solution to be converged. The default value of .001 means the frequencies from the previous cycle have to be within .001 Hz of the next guess frequencies for the solution to be converged. You should not have to change this value unless you require a more accurate solution (more accurate than .001?). Also, if you're doing a preliminary analysis, you may wish to relax this tolerance to speed up the eigensolution. If you get warning 2019 (missed frequencies) try using a more stringent convergence tolerance (increase the exponent value for the tolerance).

Saving Dynamic Solutions

After you've done the dynamic solution, you can save that solution to file to be recalled and used later.



Note: This solution is saved in a .__R file and will be deleted when the Save or Save As options are used to overwrite the file. You may also delete this file yourself.

Work Vectors

When you request a certain number of modes for dynamic analysis (let's call that number N), RISA tries to solve for just a few extra modes. Once the solution is complete, RISA goes back to check that the modes it solved for are indeed the N lowest modes. If they aren't, one or more modes were missed and an error is reported.

Dynamics Modeling Tips

Dynamics modeling can be quite a bit different than static modeling. A static analysis almost always gives you some sort of solution, whereas you are not guaranteed that a dynamics analysis will converge to a solution. This is due in part to the iterative nature of the dynamics solution method, as well as the fact that dynamics solutions are far less forgiving of modeling sloppiness than are static solutions. In particular, the way you model your loads for a static analysis can be very different than the way you model your mass for a dynamic analysis.

The term "dynamics solution" is used to mean the solution of the free vibration problem of a structure, where we hope to obtain frequencies and mode shapes as the results.

In general, the trick to a "good" dynamics solution is to model the structure stiffness and mass with "enough" accuracy to get good overall results, but not to include so much detail that it take hours of computer run time and pages of extra output to get those results. Frame problems are simpler to model than those that include plate elements. "Building type" problems, where the mass is considered lumped at the stories, are much easier to successfully model than say a cylindrical water tank with distributed mass. It is often helpful to define a load combination just for your dynamic mass case, separate from your "Dead Load" static case (You can call it "Seismic Mass"). Your seismic mass load combination will often be modeled very differently from your "Dead Load" static case.

Modes for discretized mass models with very few degrees of freedom may not be found by the solver, even if you know you are asking for fewer modes than actually exist. In this case it may be helpful to include the self weight of the model with a very small factor (i.e. 0.001) to help the solver identify the modes.

Self-Weight Considerations

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 between floors. This gives you at least 3 free joints between them.

Beam Vibration Considerations

When you perform a dynamic analysis of beam structures, such that you are trying to capture the flexural vibrations, (i.e., the beams are vibrating vertically or in the transverse direction), you must make sure that you have at least 3 free joints along the member between the points of support. If you use a distributed load as the mass, you must remember that some of the load will automatically go into the supports and be “lost” to the dynamic solution. In general, you get the best results by applying your mass as joint loads to the free nodes.

If you are trying to model dynamic effects on a 2D frame, you’ll want to make sure that you restrain the out-of-plane degrees of freedom. See [Boundary Conditions at ALL Nodes](#) to learn how to do this.

Calculate Residual Mass

The mass of the structure that has not been activated by the solved number of periodic modes may still be included in the response spectra analysis results. This is done by checking the **Calc. Residual Mass** box. This checkbox calculates a residual mass mode shape which helps to show where the missing mass in the structure is located. It also includes the response of this missing mass as a static correction to the response spectra results. The acceleration used to calculate this static correction vector is based on largest acceleration from the specified response spectra which occurs between the user entered Cut Off frequency (see below) and the zero period acceleration.



Note:

- The inclusion of Residual Mass does not guarantee that dynamic solution will achieve 100% mass participation, though it will normally be close.
- This feature is intended to capture missing RIGID response. When unsolved mode shapes would produce significant periodic response (at frequencies less than 33 Hz, for example), it would be more accurate to include these modes in the analysis than to assume that they are accounted for in the residual mass correction.
- The implementation of the residual mass feature is based on the requirements of ASCE 4 - 1998: Seismic Analysis of Safety-Related Nuclear Structures.

Common Application for Missing Mass Vectors

If you apply your dynamic mass with distributed loads or surface loads on members/plates that are adjacent to supports, remember that the some of the load goes directly into the support and is lost to a traditional 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 support to determine if that is the cause of the problem.

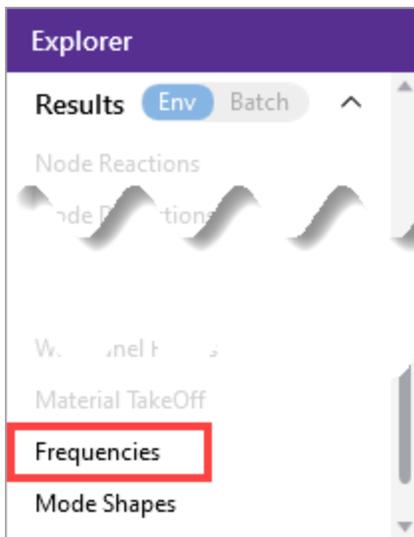
Since this mass is directly associated with boundary conditions and cannot be activated by a traditional eigensolution mode, it is referred to as the Residual Rigid Response of the structure. The common solution to account for this response, is to check the **Calc. Residual Mass?** checkbox. This calculates a "Missing Mass Vector" or "Residual Mass Vector" to account for this residual mass in the dynamic analysis.

Alternatively, you can switch to the [Ritz Vector](#) dynamic solver because Ritz modes inherently include correction for the rigid response.

Modal Frequency Results

To access the Frequencies spreadsheet:

1. Go to the Explorer panel **Results** section.



2. Click on **Frequencies**.

The Frequencies spreadsheet opens.

Frequencies						
	Mode	Frequency (Hz)	Period (Sec)	SX Participation	SY Participation	SZ Participation
1	1	0.368	2.714	0.526		
2	2	0.496	2.015	43.485		2.816
3	3	0.703	1.422	7.581		27.002
4	4	1.233	0.811	4.717		1.73
5	5	1.78	0.562	0.043		22.116
6	6	1.911	0.523	0.076		
7	7	3.801	0.263	0.115	0.022	1.512
8	8	4.121	0.243		0.059	0.026
9	9	4.495	0.222		1.776	0.06
10	10	4.525	0.221	0.025		
11	11	5.255	0.190		0.518	10.011

The Frequencies spreadsheet shows the calculated model frequencies and periods. The period is simply the reciprocal of the frequency. These values are used along with the mode shapes when a response spectra analysis is performed. The first frequency with a high participation is sometimes referred to as the model's natural or fundamental frequency. These frequency values, as well as the mode shapes, are saved and remain valid unless you change the model data, at which time they are 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 the participation factors are shown in red (as opposed to black) then the response spectra analysis (RSA) has not been performed for that direction. If the RSA has been done but a particular mode has no participation factor listed, that mode shape is not participating in that direction. This usually is because the mode shape represents movement in a direction orthogonal to the direction of application of the spectra. See [Dynamic Analysis - Response Spectra](#) for more information.



Note:

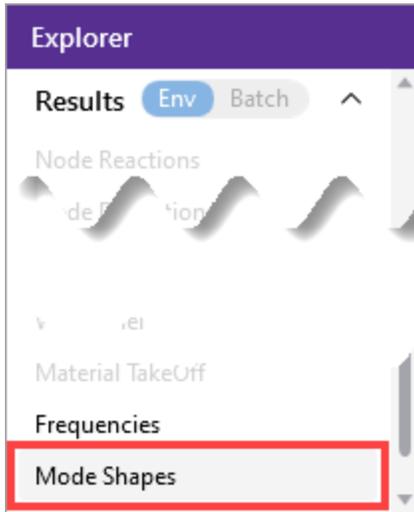
- If you have not solved an RSA for a given direction, the participation factor shows in red as an indicator. This is not an indication that there is anything wrong with the solution.
- If you are using the **Calc Residual Mass?** checkbox, you get three "residual" mode shapes that account for the missing mass vector. For more information on this see the [Residual Rigid Response](#) section.
- At this time residual / missing mass modes are not considered in Time History analysis.

Frequencies						
	Mode	Frequency (Hz)	Period (Sec)	SX Participation	SY Participation	SZ Participation
30	30	10.454	0.096	0.041	0.084	
31	31	10.95	0.091	0.018	0.4	0.092
32	32	11.355	0.088	3.651	0.035	0.074
33	33	11.46	0.087	4.603	0.034	0.064
34	34	11.513	0.087	0.053	0.281	
35	35	11.547	0.087	28.11	0.11	0.469
36	36	11.849	0.084			
37	37	11.871	0.084			
38	38	12.229	0.082	0.419		
39	Totals:			94.757	80.704	85.99

Mode Shape Results

To access the Mode Shape spreadsheet:

1. Go to the Explorer panel **Results** section.



2. Click on **Mode Shapes**.

The Mode Shape spreadsheet for the first Mode Shape Period.

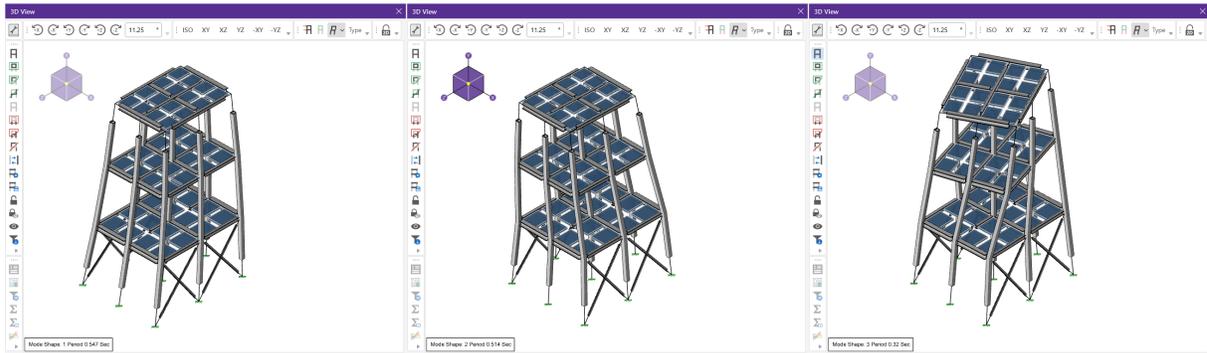
These are the model's mode shapes. Mode shapes have no units and represent only the movement of the nodes relative to each other. The mode shape values can be multiplied or divided by any value and still be valid, so long as they retain their value relative to each other. To view higher or lower modes you may select them from the drop-down list of modes on the Window Toolbar.



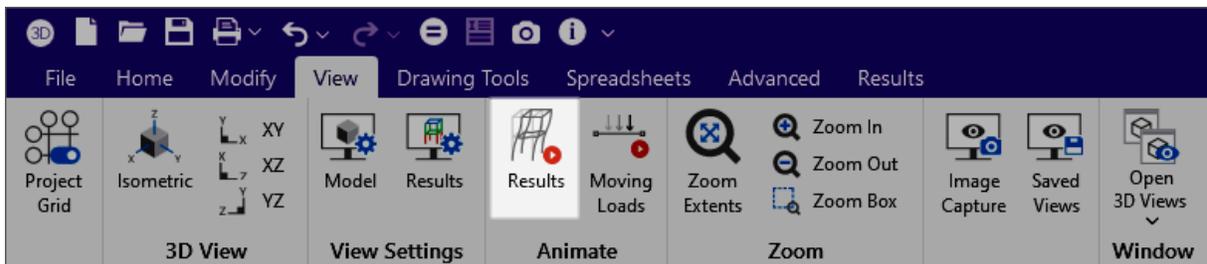
Note:

- Keep in mind that the X, Y, and Z 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.
- The rotation is in units of radians.

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.



3. (Optional) To plot and animate the mode shape of the model, got to the 'View' ribbon and click the **Results** icon in the Animate section, as shown in the following image.



This allows you to verify the mode shapes that were obtained and highlights local modes making them easy to troubleshoot. See [Results View Settings - Deflections](#) for more information.

Dynamics Troubleshooting – Local Modes

A common problem you may encounter are “localized modes”. These are modes where only a small part of the model is vibrating and the rest of the model is not. A good example of this is an X brace that is vibrating out of plane. Localized modes are not immediately obvious from looking at the frequency or numeric mode shape results, but they can be spotted pretty easily using the mode shape animation feature. Just plot the mode shape and animate it. If only a small part of the model is moving, this is probably a localized mode.

The problem with localized modes is that they can make it difficult to get enough mass participation in the response spectra analysis (RSA), since these local modes don't usually have much mass associated with them. This will show up if you do an RSA with a substantial number of modes but get very little or no mass participation. This would indicate that the modes being used in the RSA are localized modes.

Quite often, localized modes are due to modeling errors (erroneous boundary conditions, members not attached to plates correctly, etc.). If you have localized modes in your model, always try a Model Merge before you do anything else. See [Model Merge](#) for more information.

Eliminating local modes Using Boundary Conditions

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. Alternatively, in this instance you could also:

- Make the braces a weightless material. In that case no mass would be assigned to the X-brace intersection and it would not participate in the eigensolution.
- Delete the node at the crossing of X-braces. If there is no node here there is no degree of freedom. This location will not likely be crucial for a dynamic solution anyway, so eliminating the node eliminates a flexible mode with minimal mass participation.

Using Ritz Vectors to Minimize Local Modes

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 lateral seismic response of the structure. For cases like this it will often be better to switch from the standard or accelerated dynamic solvers to the Ritz Vector dynamic solver because Ritz vectors are inherently biased to avoid modes with little mass participation in the desired direction.

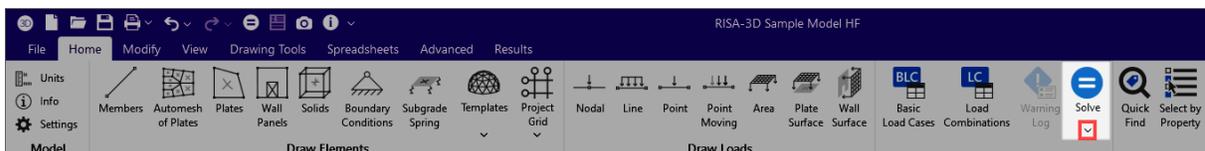
Dynamic Analysis - Response Spectra

A Response Spectra Analysis can be performed after the dynamic analysis to obtain forces, stresses and deflections. In general, the response spectra analysis procedure is based on the assumption that the dynamic response of a structural model can be approximated as a summation of the responses of the independent dynamic modes of the model.

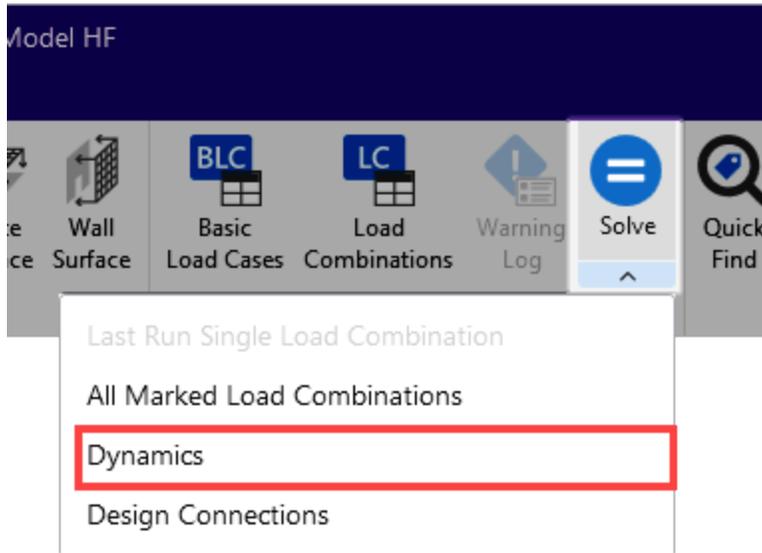
Response Spectra Analysis Procedure

To run a Response Spectra analysis:

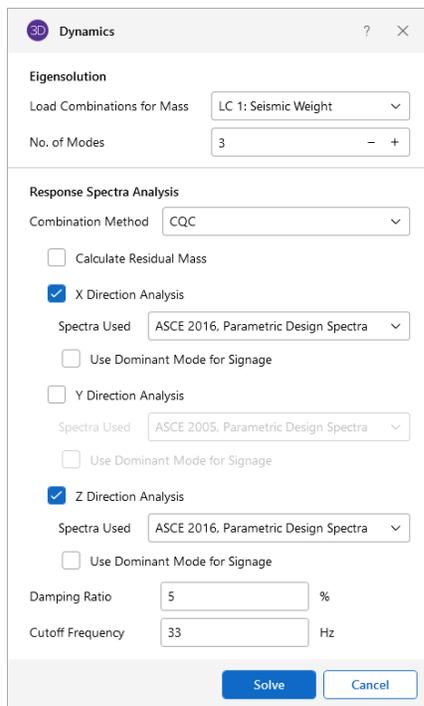
1. Go to the **Home** ribbon.



2. Click the **Solve** icon down-arrow.



3. Choose **Dynamics** from the solution options that appears, to open the Dynamics window.



4. Click the **Load Combinations for Mass** down-arrow and choose the LC you want.
5. Set the **No. of Nodes** using the “-” or “+” to increase or decrease the number.
6. Use the **Response Spectra Analysis** checkboxes to indicate which directions you want to perform the analysis.
7. Choose the spectra to be used for each direction.
8. Type the **Damping Ratio** percentage.

9. Type the **Cutoff Frequency** (in Hz).
10. Click **Solve** to run the analysis.

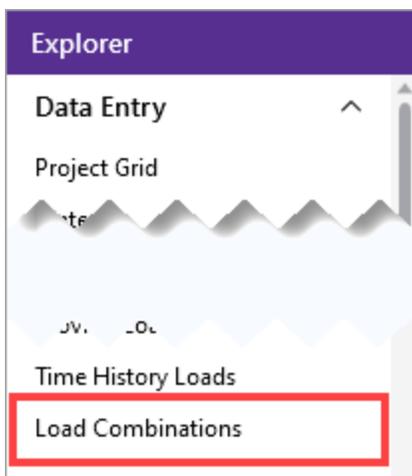
**Note:**

- For a more thorough explanation of the Eigensolution options refer to [Dynamic Analysis - Eigensolution](#).
- Upon the completion of the solution you are returned to the **Frequencies** spreadsheet and the participation yielded by the RSA is listed. To view model results such as forces/deflections/reactions you will need to create a load combination on the **Load Combination** spreadsheet that includes the spectra results. See below.
- The Dynamic Solver options are located in the Model Settings - Solution tab [Advanced](#) options.

Include Response Spectra Analysis Results in Load Combination

To include the Response Spectra analysis results in a Load Combination:

1. Go to the Explorer panel **Data Entry** section.



2. Click on **Load Combinations**.
The Load Combinations spreadsheet opens.

LC Generator	RSA Scaling Factor	Description	Solve	P-Delta	SRSS	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor
1		Seismic Weight	<input checked="" type="checkbox"/>			DL	1	SZ											
2		IBC 16-8	<input checked="" type="checkbox"/>			DL	1												
3		IBC 16-9	<input checked="" type="checkbox"/>			DL	1	LL	1	LLS	1								
4		IBC 16-10 (a)	<input checked="" type="checkbox"/>			DL	1												
5		IBC 16-12 (a) (a)	<input checked="" type="checkbox"/>			DL	1	WLX	0.6										
6		IBC 16-12 (a) (b)	<input checked="" type="checkbox"/>			DL	1	WLXP1	0.6										
7		IBC 16-12 (a) (c)	<input checked="" type="checkbox"/>			DL	1	WLXP2	0.6										
8		IBC 16-12 (a) (d)	<input checked="" type="checkbox"/>			DL	1	WLZ	0.6										
9		IBC 16-12 (a) (e)	<input checked="" type="checkbox"/>			DL	1	WLZP1	0.6										
10		IBC 16-12 (a) (f)	<input checked="" type="checkbox"/>			DL	1	WLZP2	0.6										
11		IBC 16-12 (a) (g)	<input checked="" type="checkbox"/>			DL	1	WLX	0.45	WLZ	0.45								

- Click the **BLC** ellipsis button.
A Set Basic Load Combination Entry window opens.

3D Set Basic Load Combination Entry

Load Category Code: DL - Dead Load

Seismic Options: Sds ρ Ω Capacity-Limited (CL)

Basic Load Case: 1: Dead Load

Moving Load

Time History Load

Spectral Load: SX (X direction response spectra results)

Response Spectra Analysis Options: Ω SF ρ

OK Cancel

- Choose **Spectral Load** and then click the down-arrow.
- Choose "**SX**", "**SY**", or "**SZ**" as the BLC entry.
 - SX** for the X direction RSA results
 - SY** for the Y direction RSA results
 - SZ** for the Z direction RSA results
- (Optional) To scale the spectral results, click in the **Factor** column of the row and type the spectra-scaling factor, as shown in the following image.

Load Combinations							
Combinations		Design					
LC Generator		RSA Scaling Factor					
	Description	Solve	P-Delta	SRSS	BLC	Factor	BLC
1	Dynamic Mass	<input type="checkbox"/>			DL	1	
2	Unscaled SX	<input checked="" type="checkbox"/>			SX	1	
3	Unscaled SZ	<input checked="" type="checkbox"/>			SZ	1	

Note: You can include more than one spectra solution in a single load combination (as shown in the following image). If you do this, you can also have RISA-3D combine the multiple RSA results using an SRSS summation. To use the SRSS summation, 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.

Load Combinations																	
Combinations		Design															
LC Generator		RSA Scaling Factor															
	Description	Solve	P-Delta	SRSS	BLC	Factor	BLC										
1	Dynamic Mass	<input type="checkbox"/>			DL	1	LL	2									
2	Unscaled SX	<input checked="" type="checkbox"/>			SX	1											
3	Unscaled SZ	<input checked="" type="checkbox"/>			SZ	1											
4	DL+LL+SX	<input checked="" type="checkbox"/>			DL	1	LL	1	SX	0.34							
5	DL+LL+SZ	<input checked="" type="checkbox"/>			DL	1	LL	1	SX	-0.34							
6	DL+LL+SZ	<input checked="" type="checkbox"/>			DL	1	LL	1	SX	0.37							
7	DL+LL+SZ	<input checked="" type="checkbox"/>			DL	1	LL	1	SX	-0.37							

Response Spectra

The response spectra represent the maximum response of any single degree of freedom (SDOF) system to a dynamic base excitation. The usual application of this method is in seismic (earthquake) analysis. Earthquake time history data is converted into a "response spectrum". With this response spectrum, it is possible to predict the maximum response for any SDOF system. By "any SDOF system", it is meant a SDOF system with any natural frequency. "Maximum response" means the maximum deflections, and thus, the maximum stresses for the system.

Response Spectra Analysis Procedure

In the response spectra analysis procedure, each of the model's modes is considered to be an independent SDOF system. The maximum responses for each mode are calculated independently. These modal responses are then combined to obtain the model's overall response to the applied spectra.

The response spectra method enjoys wide acceptance as an accurate method for predicting the response of any structural model to any arbitrary base excitation, particularly earthquakes. Building codes require a dynamics based procedure for some structures. The response spectra method satisfies this dynamics requirement. The response spectra method is easier, faster and more accurate than the static procedure so there really isn't any reason to use the static procedure.

If you wish to learn more about this method, an excellent reference is *Structural Dynamics, Theory and Computation* by Dr. Mario Paz (1991, Van Nostrand Reinhold).

Frequencies Outside the Spectra

If a response spectra analysis is solved using modal frequency values that fall outside the range of the selected spectra, RISA extrapolates to obtain spectral values for the out-of-bounds frequency. If the modal frequency is below the smallest defined spectral frequency, a spectral velocity is used for the modal frequency that results 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 tends to converge to the maximum ground displacement. If the modal frequency is above the largest defined spectral frequency, a spectral velocity is used for the modal frequency that results 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

	Mode	Frequency (Hz)	Period (Sec)	SX Participation	SY Participation	SZ Participation
1	1	0.575	1.739	83.526		
2	2	1.353	0.739			
3	3	1.785	0.56	10.346		
4	Totals:			93.873		

The mass participation factors reported on the **Frequencies** spreadsheet reflect how much each mode participated in the Response Spectra Analysis solution. Remember that the RSA involves calculating separately the response for each mode to the applied base excitation represented by the spectra. Here is where you can tell which modes are important in which directions. Higher participation factors indicate more important modes. The participation factor itself is the percent of

the model's total dynamic mass that is deflecting in the shape described by the particular mode. Thus, the sum of all the participation factors in a given direction can not exceed 100%.

The amount of participation for the mode may also reflect how much the mode moves in the direction of the spectra application. For example, if the 1st mode represents movement in the global Y direction it won't participate much, if at all, if the spectra is applied in the global X direction. You can isolate which modes are important in which directions by examining the mass participation.

**Note:**

- Usually for the RSA to be considered valid, the sum of the modal participation factors must equal or exceed **90%** (e.g. ASCE7-10, Section 12.9.1). If you do an RSA and the total participation is less than 90%, you need to return to the dynamic solution and redo the dynamic analysis with more modes. If you are getting a lot of modes with little or no participation see [Dynamics Troubleshooting – Local Modes](#).
- You may also want to account for accidental torsion. See [Modeling Accidental Torsion](#) to learn how to do this.
- Models with a large amount of mass lost into boundary conditions may have difficulty achieving 90% mass participation. See [Dynamics Modeling](#) for more information.

Modal Combination Option

There are three choices for combining your modal results: CQC, SRSS or Gupta. In general, you'll want to use either CQC or Gupta.

- **CQC** stands for "Complete Quadratic Combination".

Use CQC for models where you don't expect much rigid response.

A complete discussion of this method will not be offered here, but if you are interested, a good reference on this method is *Recommended Lateral Force Requirements and Commentary, 1999*, published by SEAOC (Structural Engineers Assoc. of Calif.). In general, the CQC is a superior combination method because it accounts for modal coupling quite well

- Use Gupta for models where the rigid response could be important.

An example of one type of model where rigid response would be important is the analysis of shear wall structures.

- Use the SRSS method in case you need to compare results with the results from some older program that does not offer CQC or Gupta.

CQC stands for "Complete Quadratic Combination". A complete discussion of this method will not be offered here, but if you are interested, a good reference on this method is *Recommended Lateral Force Requirements and Commentary, 1999*, published by SEAOC (Structural Engineers Assoc. of Calif.). In general, the CQC is a superior combination method because it accounts for modal coupling quite well.

The Gupta method is similar to the CQC method in that it also accounts for closely spaced modes. In addition, this method also accounts for modal response that has “rigid content”. For structures with rigid elements, the modal responses can have both rigid and periodic content. The rigid content from all modes is summed algebraically and then combined via an SRSS combination with the periodic part which is combined with the CQC method. The Gupta method is fully documented in the reference, **Response Spectrum Method**, by Ajaya Kumar Gupta (Published by CRC Press, Inc., 1992).

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

Unsigned (All Positive) Results and the Use Dominant Mode For Signage Checkbox

A response spectra analysis involves calculating forces and displacements for each mode individually and then combining these results. The problem is both combination methods offered (SRSS and CQC) use a summation of squares approach that loses the sign. This means that all the results are positive, except reactions, which are all negative as a result of the positive displacements.

Because the RSA results are unsigned you cannot directly add the results to other static loads in you model. One way around this is to treat the RSA results as both positive and negative by manually providing the sign. Using two combinations for each RSA result, one with a positive factor and the other with a negative factor you can capture the maximum deflections, stresses and forces when combining with other loads. See [Load Combinations with RSA Results](#) for an example.

The mass participation may indicate that a model is dominated by a single mode in a direction. You can base the signs for the final combined RSA results on the signs for the RSA for this single dominant mode, by checking the box that says “**Use Dominant Mode for Signage?**”. When selected, the Mode that has the highest mass participation in that direction is considered to be the dominant mode.

Limitations on Response Spectra Results

Because of the statistical method used to combine the results, response spectra results can be confusing or misleading to engineers that are not familiar with the process. The confusion is usually associated with the fact any single response spectra result is correct, but that it is not known whether the sign of that result should be positive or negative. This results in a number of limitations that should be considered when using response spectra analysis.

Limitations on Response Spectra Results

Limitation	Description
Statics	By default with an RSA solution the results will not obey statics, due to the nature of combining the modes via SRSS, CQC, etc. Thus, if you view the reactions, member forces, etc., the model will not obey statics. If the model has a "dominant" mode in the given direction then the normal path forward would be to select the Use

Limitation	Description
	<p>Dominant Mode for Signage? checkbox when solving for the dynamic solution.</p> <p>For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: Response Spectra Reactions Satisfy Statics.</p>
Deflected Shape	<p>By default with an RSA solution the results will not obey statics, due to the nature of combining the modes via SRSS, CQC, etc. Thus, if you view the deflected shape you will see that the deflected shape looks odd. Similar to the "Statics" limitation above, one possible path forward is to use the Use Dominant Mode for Signage? checkbox when solving your dynamic solution.</p> <p>For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: Response Spectra Reactions Satisfy Statics.</p>
Plate and Wall Contours	<p>Plate contours are not presented to the user for RSA load combination unless the Use Dominant Mode for Signage? option has been chosen. This is because the results can be misleading due to the "Statics" issue described above.</p>
Wall Panel Reactions	<p>Wall Panel reactions consist of a number of internal nodal reactions that are then summed together across the wall. Since the signs of those reactions are unknown, the reactions for each Wall Panel cannot be correctly calculated. There are two methods of dealing with this issue:</p> <ul style="list-style-type: none"> • Use the solution described in the "Statics" limitation. • Use the Wall Forces spreadsheet results or Wall Detail Report instead. <div data-bbox="553 1371 1409 1698" style="border: 1px solid black; padding: 10px; margin-top: 10px;"> <p> Note: The Wall Force results (in the detail reports and the Wall Force Results spreadsheet) are calculated correctly because each force is calculated and stored for each mode, which allows a final RSA result to be calculated using the combination method (SRSS, CQC etc) chosen by you. A future revision to the program may be to calculate and store wall reactions for each mode similar to what is done for the Wall Forces spreadsheet.</p> </div>
Internal Force Summation Tools	<p>Because the Internal force summation tool relies on plate corner forces, these results are not available for wall panels for any load combination which contains RSA results. Internal force summation tools may be used on other portions of the model. However, they will face the same issue that is described under the "Statics" limitation above. For that reason the results are suppressed unless</p>

Limitation	Description
	the Use Dominant Mode for Signage? option has been chosen.

Other Options

Cut Off Frequency

This is the frequency used by the Gupta method to calculate the upper bound for modes having periodic and rigid content. The “rigid frequency” is defined as “The minimum frequency at which the spectral acceleration becomes approximately equal to the zero period acceleration (ZPA), and remains equal to the ZPA”. If nothing is entered in this field, the last (highest) frequency in the selected response spectra will be used.

For seismic response, the typical cutoff frequency would be 33 Hz.

Damping Ratio

The damping ratio entered here is used in conjunction with the CQC and Gupta combination methods. This single entry is used for all the modes included in the RSA, an accepted practice. A value of 5% is generally a good number to use. Typical damping values are:

- 2% to 5% for welded steel
- 3% to 5% for concrete
- 5% to 7% for bolted steel, wood

RISAFloor Spectra Results

When a response spectra analysis is run in a combined RISAFloor/RISA-3D model with multiple eigen solutions (centered mass versus eccentric mass solutions), the results for each item is based on the worst case of the response spectra for the multiple modes. An example would be that node N1 gets its base reaction value from the centered mass response spectra solution, while another node, N2, gets its base reaction from the Plus X eccentric mass response spectra solution.

When using the **Use Dominant Mode for Signage** option with multiple eigen solutions under RISAFloor, the dominant mode is defined by the centered mass solution with highest mass participation in that direction.

RSA Scaling Factor (Manual Scaling)

The most difficult part of the entire RSA procedure is normally calculating the scaling factor to be used when including the RSA results in a load combination.

The ASCE 7 uses a particular “shape” for its spectra (See Figure 11.4-1), but the parameters S_{DS} and S_{D1} make it specific to a particular site. However, the ASCE 7 imposes several requirements regarding the minimum design values. ASCE 7-16 Section 12.9.1.4.1 specifies a modification factor, V/V_t (ASCE 7-10 Section 12.9.4.1 specifies a modification factor, $0.85 \cdot V/V_t$), that can be used to

scale the response spectra results to something less than or equal the base shear calculated using the static procedure (ASCE 7 Sect. 12.8).

The static base shear (V) is calculated using the equations in ASCE 7-16 Sect. 12.8.1

Note that there are limiting values for the static base shear in ASCE 7-16 equations 12.8-3 through 12.8-6.

Therefore, in order to calculate the proper scaling factor, we need to know what the *unscaled* RSA base shear (this is called the Elastic Response Base Shear in the IBC) is, and we also need to calculate the value of " V " (static base shear). The calculation of V isn't particularly difficult because the two values that present the biggest problem in this calculation (T and W) are provided by RISA. To calculate the value of W , simply solve a load combination comprised of the model seismic dead weight. This almost certainly will be the same load combination you used in the **Dynamics** settings for the **Load Combination for Mass**. The vertical reaction total is your " W " value.

The T value is simply the period associated with the dominant mode for the direction of interest. For example, if you're calculating the scaling factor for a Z-direction spectra, determine which mode gives you the highest participation for the Z direction RSA. The period associated with that mode is your T value. Note that there are limiting values for T , see ASCE 7-16 section 12.8.2.

Calculating the unscaled RSA base shear also is very straightforward. Just solve a load combination comprised of only that RSA, with a factor of 1.

Example:

- Assuming we're looking at a Z direction RSA, enter "SZ" in the **BLC** field of the **Load Combinations** spreadsheet and for the **Factor** enter "1". Leave all the other BLC fields blank.
- Solve the load combination and look at the Z direction reaction total. This total value is the unscaled RSA base shear.
- To get the correct scaling factor, solve this equation:

$$\text{Scale Factor} = (V / \text{Unscaled RSA base shear})$$

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



Note: The ASCE 7 has additional requirements for vertical seismic components. (See ASCE 7-16 section 12.4.2.2).

RSA Scaling Factor (Automatic Scaling)

The most difficult part of the entire RSA procedure is normally calculating the scaling factor to be used when including the RSA results in a load combination. The Scaling Factor tool can be used to automatically calculate the RSA scaling factor for each direction within your model.

The RSA scaling factors automatically default to 1.0 and can only be changed in the Scaling Factor window. You can open the window by clicking the scale factor button, **RSA Scaling Factor**, at the top of the **Load Combinations** spreadsheet.



Note: Currently these factors are only calculated when you open this window and click the **Calculate** button. Therefore, if you have previously calculated these and then made changes to your model, you need to come back into this window and re-click the **Calculate** buttons to update the scaling factors.

Base Shear

The **Base Shears** section of the Spectra Scaling Factor window box reports to you the static base shear and the un-scaled RSA base shear for each direction. If these values have not been calculated yet, then you may press the **Calculate** button. This launches the [Seismic Load Generation](#) window for the calculation of the **Static Base Shear**, or the Dynamic Solution dialog for the calculation of the **Unscaled Base Shear** for your [Response Spectra](#) results.

3D Response Spectra Scaling Factor
? X

Scaling Factor Program Calculated ▼

Apply scaling factor to current Response Spectra Analysis (RSA) load combinations

Program Calculated

	Z Direction	X Direction	
Base Shear Multiplier	1	1	
Static Base Shear	226.475	226.475	kips Calculate
Unscaled Base Shear	231.496	236.712	kips Calculate

	SF Z Direction	SF X Direction	
ELF Limit	0.978	0.957	Update
I/R Limit	0.333	0.333	

Governing Limit	0.978	0.957	
-----------------	-------	-------	--

OK
Cancel

Scaling Factor

Program Calculated

This section compares the ELF (Equivalent Lateral Force) Limit, as reduced by the Base Shear Multiplier, and the I/R Limit. These values are explained in detail below.

- The **Base Shear Multiplier** is used in the calculation of the ELF Limit. This multiplier can be used to reduce the calculated (unscaled) base shear lower (typically no more than 85%) than the Equivalent Lateral Force Method per ASCE 7-10. For the 1997 UBC, this is covered in section 1631.5.4. For the 2000 IBC, this is covered in section 1618.7. For the 2015 NBC, it is discussed in clause 4.1.8.12 (8). In the ASCE 7-16, it is discussed in section 12.9.1.4.1.

- The **ELF Limit** is the limiting scaling factor calculated per the Equivalent Lateral Force procedure:

$$ELF\ Limit = \frac{(Base\ Shear\ Multiplier * Static\ Base\ Shear)}{Unscaled\ Base\ Shear}$$

- The **I/R Limit** simply takes the elastic dynamic base shear and reduces it to an inelastic base shear. This value is calculated per the I (Importance Factor) and R (Response Modification Coefficient). You may edit these values in the Seismic tab of [Model Settings](#).
- The **Controlling Limit** is the governing (larger) value of the ELF limit and I/R limit.

User Input

You may manually override the program calculated factor by selecting the **User Input** option and entering in an alternate value.

Apply Checkbox

The **Apply SF to all RSA Load Combinations** checkbox applies to the SF scale factors for all load combinations that currently reference response spectra results. The final Load combinations will appear similar to those shown below:

Description	Solve	P-Delta	SRSS	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor
1 Dynamic Mass	<input type="checkbox"/>			DL	1	LL	0.2										
2 DL+LL+SX	<input checked="" type="checkbox"/>			DL	1	LL	1	SX*SF	0.34								
3 DL+LL+SX	<input checked="" type="checkbox"/>			DL	1	LL	1	SX*SF	-0.34								
4 DL+LL+SZ	<input checked="" type="checkbox"/>			DL	1	LL	1	SX*SF	0.37								
5 IBC 16-12 (a) (a)	<input checked="" type="checkbox"/>			DL	1	LL	1	SX*SF	-0.37								

Automatic Response Spectra Generation

1997 UBC

You can have the 1997 UBC spectra generated automatically by selecting "UBC 97, Parametric Design Spectra" for your RSA. The C_a and C_v 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 C_a and C_v values. The default values listed are for Seismic Zone 3, Soil Type "Se" (Soft Soil Profile). These values can be edited in the **Seismic** tab of [Model Settings](#).

2000 IBC

You can have the 2000 IBC spectra generated automatically by selecting "IBC 2000, Parametric Design Spectra" for your RSA. The S_{DS} and S_{D1} seismic coefficients are needed to calculate the values for the IBC 2000 spectra. See Figure 1615.1.4 in the IBC for the equations used to build the

spectra. See section 1615.1.3 to obtain the S_{DS} and S_{D1} values. These values can be edited in the **Seismic** tab of [Model Settings](#).

2005 ASCE

You may have the 2005 ASCE spectra generated automatically by selecting "ASCE 2005, Parametric Design Spectra" for your RSA. The S_{DS} , S_{D1} , and TL seismic coefficients are needed to calculate the values for the ASCE 2005 spectra. See Figure 11.4-1 in ASCE-7 2005 for the equations used to build the spectra. See section 11.4.4 to obtain the S_{DS} and S_{D1} values and Figures 22-15 thru 22-20 for the TL value. These values can be edited in the **Seismic** tab of [Model Settings](#).

2010 ASCE

You can have the 2010 ASCE spectra generated automatically by selecting "ASCE 2010, Parametric Design Spectra" for your RSA. The S_{DS} , S_{D1} , and TL seismic coefficients are needed to calculate the values for the ASCE 2010 spectra. See Figure 11.4-1 in ASCE-7 2010 for the equations used to build the spectra. See section 11.4.4 to obtain the S_{DS} and S_{D1} values and Figures 22-12 thru 22-16 for the TL value. These values can be edited in the **Seismic** tab of [Model Settings](#).

2016 ASCE

You can have the 2016 ASCE spectra generated automatically by selecting "ASCE 2016, Parametric Design Spectra" for your RSA. The S_{DS} , S_{D1} , and TL seismic coefficients are needed to calculate the values for the ASCE 2016 spectra. See Figure 11.4-1 in ASCE-7 2016 for the equations used to build the spectra. See section 11.4.5 to obtain the S_{DS} and S_{D1} values and Figures 22-14 thru 22-17 for the TL value. These values can be edited in the **Seismic** tab of [Model Settings](#).

2005 NBC

You can have the 2005 NBC spectra generated automatically by selecting "NBC 2005 Parametric Design Spectra" for your RSA. The Site Class and the S_a values are needed to calculate the values for the NBC 2005 spectra. Please see section 4.1.8.4(7) to obtain the S_a values and Table 4.1.8.4(A) for the Site Class. Please see section 4.1.8.4(7) for the equations used to build the spectra. These values can be edited in the **Seismic** tab of [Model Settings](#).

2010 NBC

You can have the 2010 NBC spectra generated automatically by selecting "NBC 2010 Parametric Design Spectra" for your RSA. The Site Class and the S_a values are needed to calculate the values for the NBC 2010 spectra. Please see section 4.1.8.4(7) to obtain the S_a values and Table 4.1.8.4(.A) for the Site Class. Please see section 4.1.8.4.(7) for the equations used to build the spectra. These values can be edited in the **Seismic** tab of [Model Settings](#).

2015 NBC

You can have the 2015 NBC spectra generated automatically by selecting "NBC 2015 Parametric Design Spectra" for your RSA. The Site Class and the S_a values are needed to calculate the values for the NBC 2015 spectra. Please see section 4.1.8.4(7) to obtain the S_a values and Table 4.1.8.4(.A) for the Site Class. Please see section 4.1.8.4.(7) for the equations used to build the spectra. These values can be edited in the **Seismic** tab of [Model Settings](#).

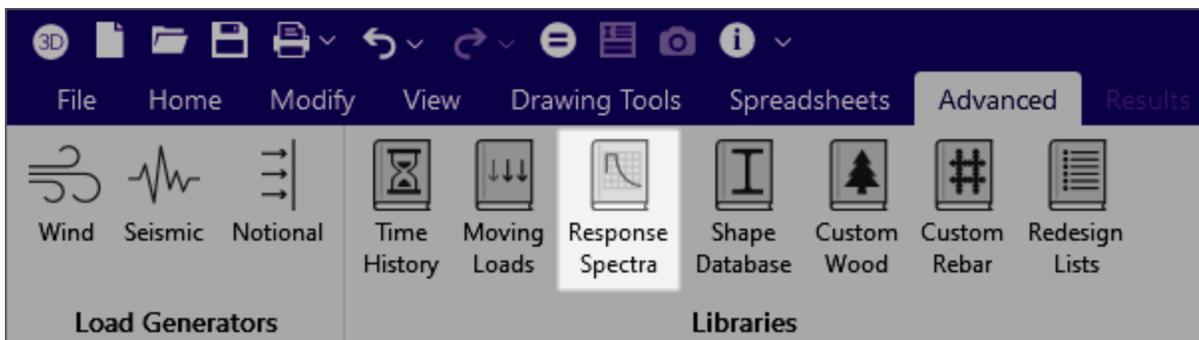
Adding and Editing Spectra

You can add your own spectra to the database and edit and delete them once they are created. You can add/edit spectra data pairs in any configuration by choosing between Frequency or Period and between the three spectral values. You can 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.

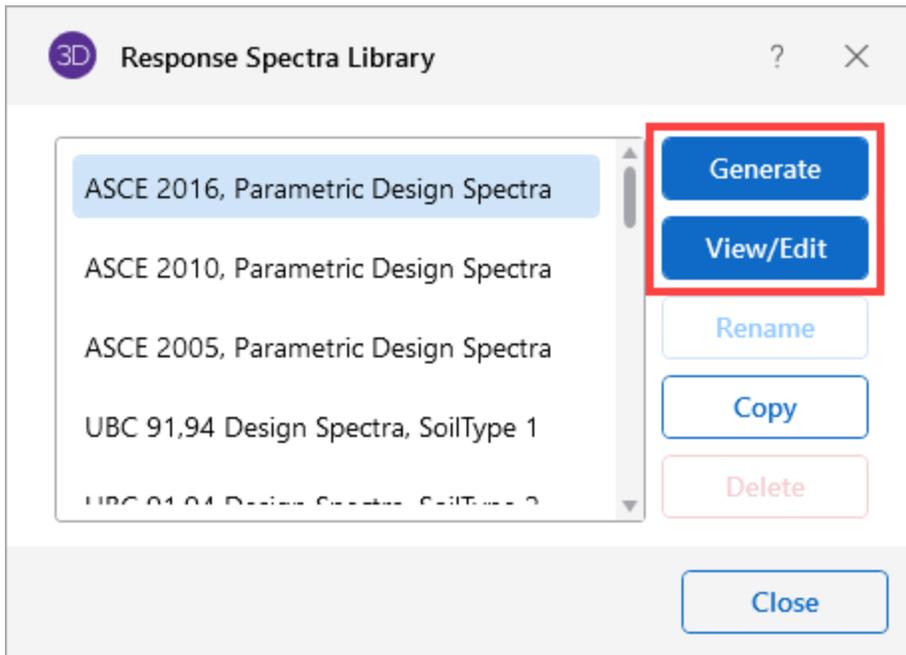
Add or Modify a Spectra

To add or edit a Spectra:

1. Open the **Advanced** ribbon.



2. Click the **Response Spectra** icon.
The Response Spectra Library window opens.



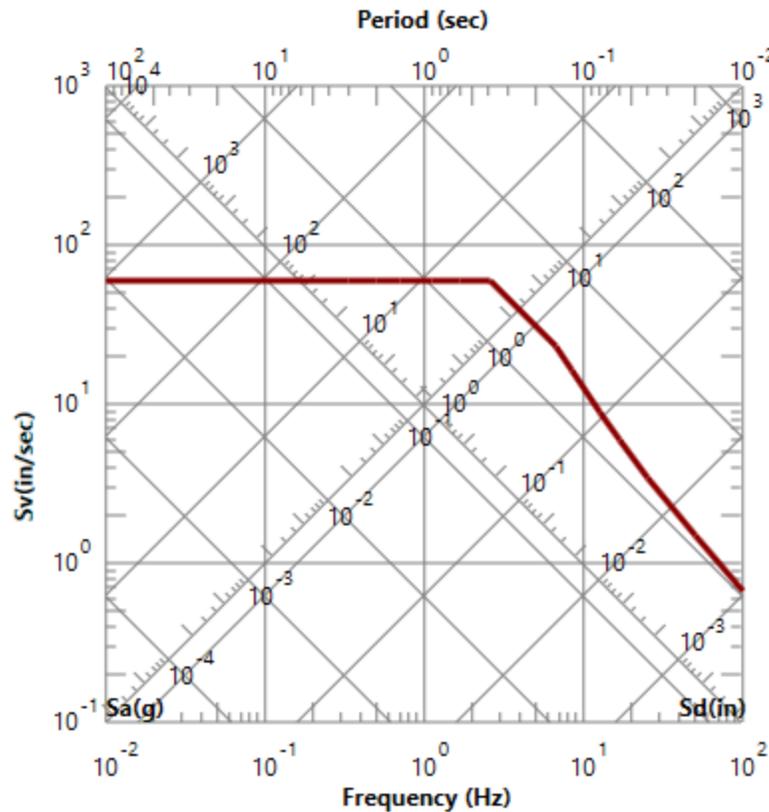
3. Click the **Generate** or **View/Edit** button.
4. Select the format to be used and specify the parameters.
5. (Optional) Click the **?** **Help** icon for help on an item.

**Note:**

- Zero values are not allowed in the data.
- The spectra data is not currently stored with the RISA model file. Instead it is stored in the RSPECT32.FIL database file located in the directory set using [3D Button - Application Settings - File Locations](#). If a file with a custom spectra needs to be transferred to another computer, then this file must also be transferred to the new computer.

Tripartite Response Spectra Plot

This plot is a convenient logarithmic representation of all the values of interest in the response spectra definition.



These values are as follows:

- Frequency (f)
- Period (T)
- Pseudo Velocity (Sv)
- Pseudo Acceleration (Sa)
- Pseudo Displacement (Sd)

The relationships between these values (for the undamped case) is as follows:

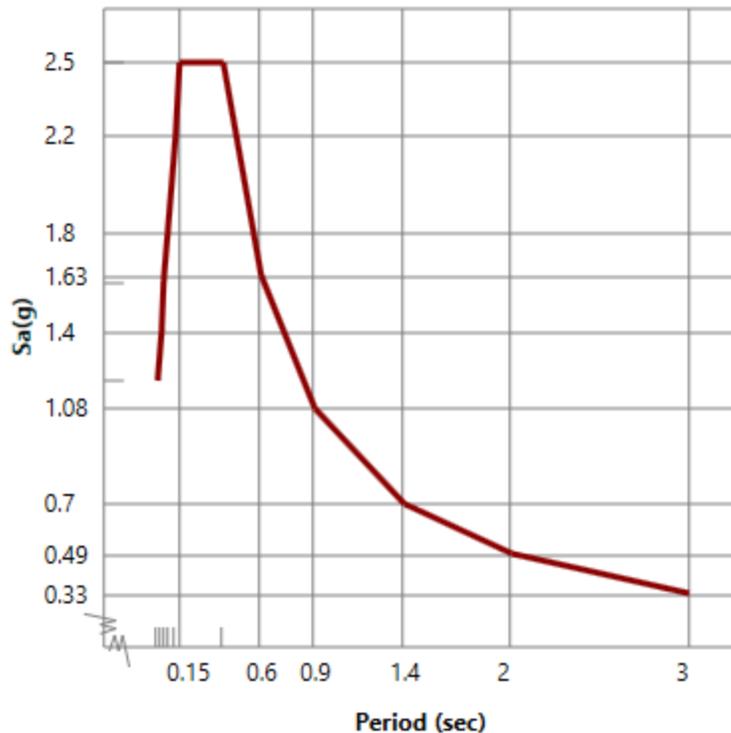
- $T = 1. / f$
- $Sv = Sd * 2\pi f = Sa / 2\pi f$

For the tripartite plot, the frequency values are plotted along the bottom with the reciprocal period values displayed along the top. The ordinate axis plots the Sv values (labeled on the left side) and the diagonal axes plot the Sa (lower left to upper right) and Sd (upper left to lower right) values.

The spectra data itself is represented with the thick red line. Therefore, to determine the Sv, Sa or Sd value for a particular frequency or period, locate the desired period or frequency value along the abscissa axis and locate the corresponding point on the spectra line. Use this point to read off the Sv, Sa and Sd values from their respective axes. Remember, all the axes are logarithmic!

Single Spectra Plot

This plot is a Response spectra representation that is given in the building codes. The vertical axis can plot the spectra using Pseudo-Acceleration, Pseudo-Velocity, or Pseudo Displacement on a vertical or logarithmic scale. The horizontal axis plots the Period or Frequency using a Linear or Logarithmic scale.



These values are as follows:

- Frequency (f)
- Period (T)
- Pseudo Velocity (Sv)
- Pseudo Acceleration (Sa)
- Pseudo Displacement (Sd)

The relationships between these values (for the undamped case) is as follows:

- $T = 1. / f$
- $Sv = Sd * 2\pi f = Sa / 2\pi f$

The spectra data itself is represented with the thick red line. Therefore, to determine the Sv, Sa or Sd value for a particular frequency or period, locate the desired period or frequency value along the

horizontal axis and locate the corresponding point on the spectra line. Use this point to read off the S_v , S_a and S_d values from their vertical axis.

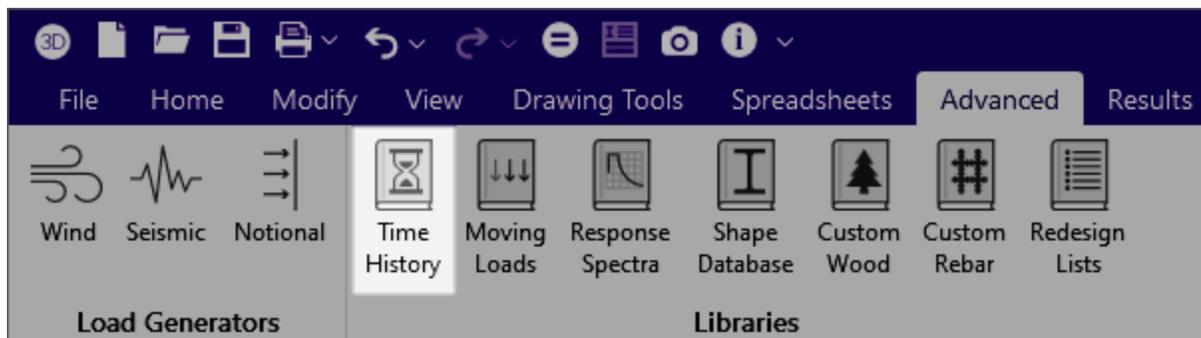
Dynamic Analysis - Time History

An Eigensolution uses the mass and stiffness matrices to calculate natural frequencies and natural modes of vibration for a structure due to free or unforced vibration. A Time History Analysis uses the mass, stiffness and damping matrices to solve for the forced vibration of the structure due to an applied load that varies with time. Therefore, the program must have information about the force (as a function of time) applied to the structure and the structure's damping in order to solve the dynamic analysis.

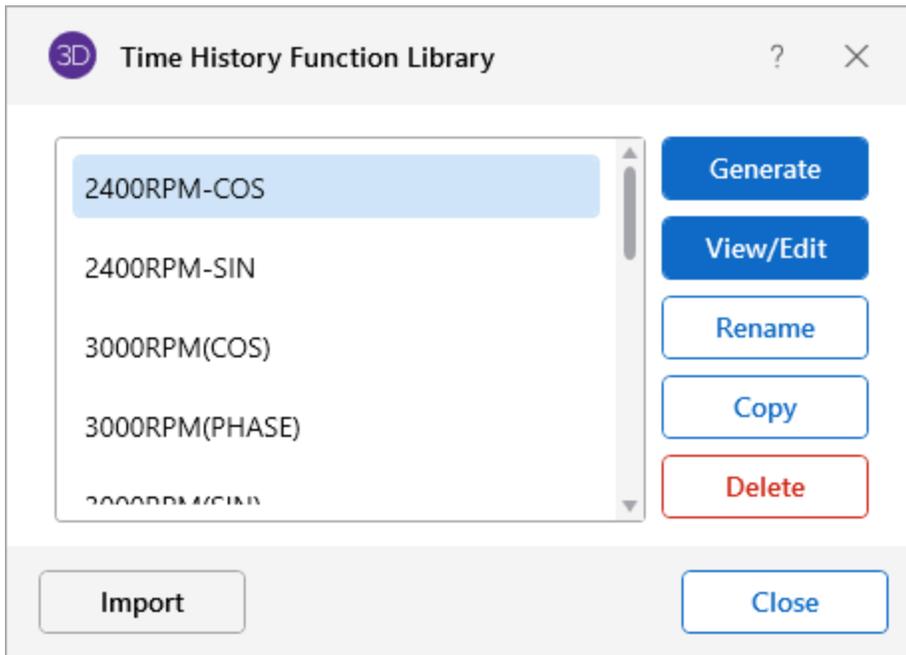
Time History Functions / Pattern

To open the Time History Function Library:

1. Go to the **Advanced** ribbon .
2. Click the **Time History** icon (as shown in the following image).



The Time History Function Library window opens.



The time history functions listed in the window are each stored in their own *.FIL file in the Time History sub-folder of the RISA working directory. The path to these files is specified in **3D Button** ▶ **Application Settings** on the [File Locations](#) tab.

Time History Load Functions / Patterns may be [Generated](#) or [Imported](#). Typically, seismic blast loading functions are imported. Loading from dynamic equipment is likely to be generated from a simple sinusoidal function.

Generated Loads

Generated Loads may be used for simple sine and cosine functions. The generate option also allows for those functions to be assigned Ramp Up functions at the beginning of the pattern, or Coast Down functions at the end of the pattern. These functions can adjust how both the frequency and magnitude change with time.

Generate Time History Function

Base Function | Ramp Up | Coast Down

Name: NEW PATTERN

Function Type: Sin

Freq/Period: 1 Hz

Duration: 5 **Complete Last Cycle**

Scale Factor: 1

Phase: 0

Apply

Result

Time (sec): 0 to 5

Y-axis: -1.5 to 1.5

Display Range

Time Min: 0 Max: 5 Steps: 1

Values

Time	Value
1	0
2	0.382
3	0.707
4	0.923
5	1
6	0.923
7	0.707
8	0.382
9	0
10	-0.382

Print **Export** **OK** **Close**

Base Function - tab

Base Function

Name: NEW PATTERN

Function Type: Sin

Freq/Period: 1 Hz

Duration: 5 **Complete Last Cycle**

Scale Factor: 1

Phase: 0

Apply

Time History Function: Base Function - tab Settings

Setting	Description
Function Type	Function Type allows you to define the type of function. Currently, a generated function may be defined as either a sine or cosine. <ul style="list-style-type: none"> • Sin - Sine • Cos - Cosine

Setting	Description
Freq / Period	Freq / Period allows you to enter the function based on whatever criteria (Hertz, Rotations per Minute, or natural period) is most convenient for them. First choose the criteria, then enter the frequency / period. Criteria options are: Hz - Hertz RPM - rotations per Minute sec - (Seconds) the natural period of time
Duration (sec)	Duration allows you to change the length (in seconds) of the Time History function cycle.
Complete Last Cycle	The Complete Last Cycle button allows you to change the duration of the function so that the last cycle is completed.
Scale Factor	Scale Factor allows you to scale a function up or down by a given amount. This is often entered in as the total force produced by the sinusoidal function.
Phase (deg)	Phase allows you to identify the sinusoidal loading angle (in degrees) that starts at values other than the maximum or zero.
Apply	The 'Apply' button forces the plotted image of the function to be updated. The text of this button turns bold when the program detects that changes have been made that are not reflected in the current function plot.

Display Range

This section controls the display of the function. This may be useful when it is necessary to zoom in on certain times within the overall pattern / function, especially ramp up or coast down regions. This allows you to verify that the function is created correctly and that no discontinuity exists at the ramp up or ramp down transition points.

Display Range

Time
 Min
 Max
 Steps

Time History Function: Display Range Settings

Setting	Description
Steps	The number of steps is not as relevant to generated functions because these are displaying general functions. The default displayed step is 0.001 seconds in the generation window and currently cannot be changed.
Apply	The 'Apply' button (found under the Base function section) forces the plotted image of the function to be updated. The text of this button turns bold when the program detects that changes have been made that are not reflected in the

Setting	Description
	current function plot.



Note: The 0.001 second display can be problematic for high frequency functions greater than about 100 Hz. But, this is merely a limitation in the display behavior and does not represent a problem with the internally created function.

Ramp Up - tab

The *Ramp Up* tab controls how the frequency and magnitude change with time at the beginning of the function. When **AutoCalc** option is selected for the magnitude that means that it will vary with the square of the frequency. This can be thought of as the behavior of the equipment when it is first turned on before it attains its normal operating speed.

Ramp Up

Duration	<input type="text"/>	sec
Initial Frequency	<input type="text"/>	Hz
Freq Function	No Freq Function ▼	
Mag Function	No Mag Function ▼	

Time History Function: Ramp Up Settings

Setting	Description
Freq Function	The Frequency Function option lets you choose which frequency function to use for Ramp Up. Choices are; no Frequency Function, Linear Frequency Function or Quadratic Frequency Function.
Mag Function	The Mag Function option lets you choose which mag function to use for Ramp Up. choices are; no Mag Function, AutoCalc Mag Function, Linear Mag Function or Quadratic mag Function.
Apply	The Appl' button forces the plotted image of the function to be updated. The text of this button turns bold when the program

Setting	Description
	detects that changes have been made that are not reflected in the current function plot.

Coast Down - tab

The *Coast Down* tab controls the frequency and magnitude change with time at the end of the function. When **AutoCalc** option is selected for the magnitude that means that it will vary with the square of the frequency. This can be thought of as the behavior of the equipment as it is turned off and slows down from its operating speed.

Coast Down

Duration	<input type="text"/>	sec
Final Frequency	<input type="text"/>	Hz
Freq Function	No Freq Function ▼	
Mag Function	No Mag Function ▼	

Time History Function: Coast Down - tab Settings

Setting	Description
Freq Function	The Frequency Function option lets you choose which frequency function to use for Coast Down. Choices are; no Frequency Function, Linear Frequency Function or Quadratic Frequency Function.
Mag Function	The Mag Function option lets you choose which mag function to use for Coast Down. choices are; no Mag Function, AutoCalc Mag Function, Linear Mag Function or Quadratic mag Function.
Apply	The Appl' button forces the plotted image of the function to be updated. The text of this button turns bold when the program detects that changes have been made that are not reflected in the current function plot.

Import Load

For more complex functions, the program allows importing from a text file. That text file must be written in a particular format in order to be read properly. See information below for a brief example:

```

[TIME_HISTORY_INPUT_DATA]
[HEADER]
Original Blast
[END_HEADER]
[FUNCTION_INFO]
[.MULTIPLIER] <1>
1 ;
[.END_MULTIPLIER]
[.TIME_STEP] <1>
0.1 ;
[.END_TIME_STEP]
[.RECORD_PER_LINE] <1>
1;
[.END_RECORD_PER_LINE]
[END_FUNCTION_INFO]
[RECORDS] <7>
0.
5.
3.
1.
-1.
-0.5
0.
[END_RECORDS]
[END_FILE]

```

The following table describes the main sections for the text file import.

Text File Import Main Sections

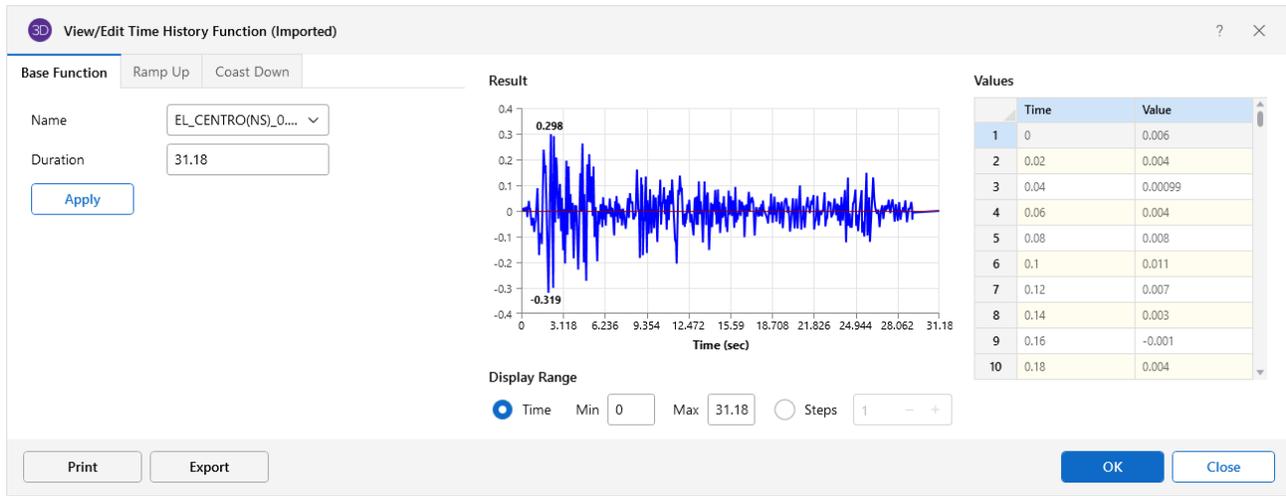
Section	Description
[HEADER]	The [HEADER] section is the name of the time history as it will appear in the library of time history patterns / functions. This section is a required

Section	Description
	field.
[FUNCTION_INFO]	The [FUNCTION_INFO] section gives information on the function itself, like time step, number of records per line, et cetera.
[.MULTIPLIER]	The [.MULTIPLIER] is used for converting the units of the record data. For example, acceleration records in RISA are assumed to be entered in terms of acceleration of gravity. If the record gives the acceleration in terms of meters per second squared, then this multiplier would be set to 0.10194 (1/9.81). This value will be assumed to be equal to 1.0 if not specifically entered.
[.TIME_STEP]	The [.TIME_STEP] is assumed to be 0.01 sec, if not entered.
[.RECORD_PER_LINE]	The [.RECORD_PER_LINE] field corresponds to the number of records per line. This will be assumed to be 1 if not entered.
	<div style="border: 1px solid black; padding: 10px; background-color: #e6f2ff;"> <p> Note:</p> <ul style="list-style-type: none"> • Acceleration based functions are assumed to already be entered in terms of the acceleration of gravity (g). Therefore, it is not necessary to multiply by 32.174 ft/s². • Text files which illustrate example functions exist in the RISA\Time History\ sub-folder of the main RISA directory. These are merely ASCII text files with an extension of *.TH to identify them as Time History functions. </div>
[RECORDS]	The [RECORDS] section gives the actual time history records. These values may be separated by spaces, semi-colons or carriage returns. The <number> which follows this entry is the number of lines of data, not the total number of records. Therefore, if the file contains 3 records per line and 500 lines of data. This entry would say <500> not <1500>.

View / Edit

The View / Edit Time History Function window looks different depending on what type of time history function is being viewed. When it is a generated sinusoidal function, there are controls that allow the user to edit the function. This editing is performed with the same controls (Freq/Period, Ramp Up, Coast Down) shown in the [Generate](#) screen.

When it is an imported time history, the only editing that can be done is to shorten the duration time of the motion. This can be done, for example, if the imported Seismic Record contains 50+ seconds of data, but the strong motion data lasts only a fraction of that time. If this is desired, then enter cutoff time in the duration and the time history function will be truncated at that time.



In this view / edit window, the **Values** table lists the time / value pairs for the function. This can be used to view the exact time at which the function reaches a maximum or the exact value of a function at a given moment in time.

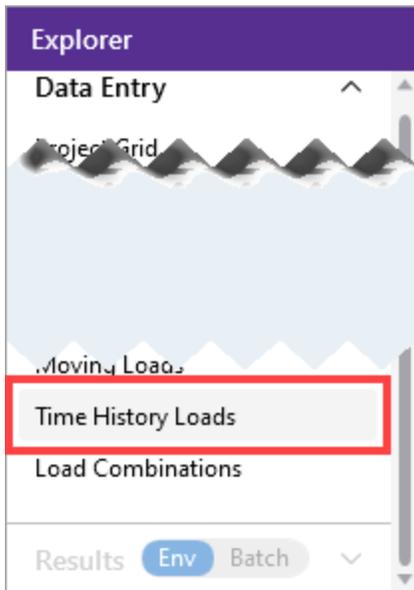
Hovering the cursor over the graph makes a vertical gray line appear that follows the cursor, so long as it is hovered over the graph area. Clicking anywhere in the graph area highlights the corresponding values in the **Values** table on the right-hand side.

Time History Loads Spreadsheet

The **Time History Loads** spreadsheet allows you to assign one of the existing time history to specific nodes or all nodes in the model.

To access the Time History Loads spreadsheet:

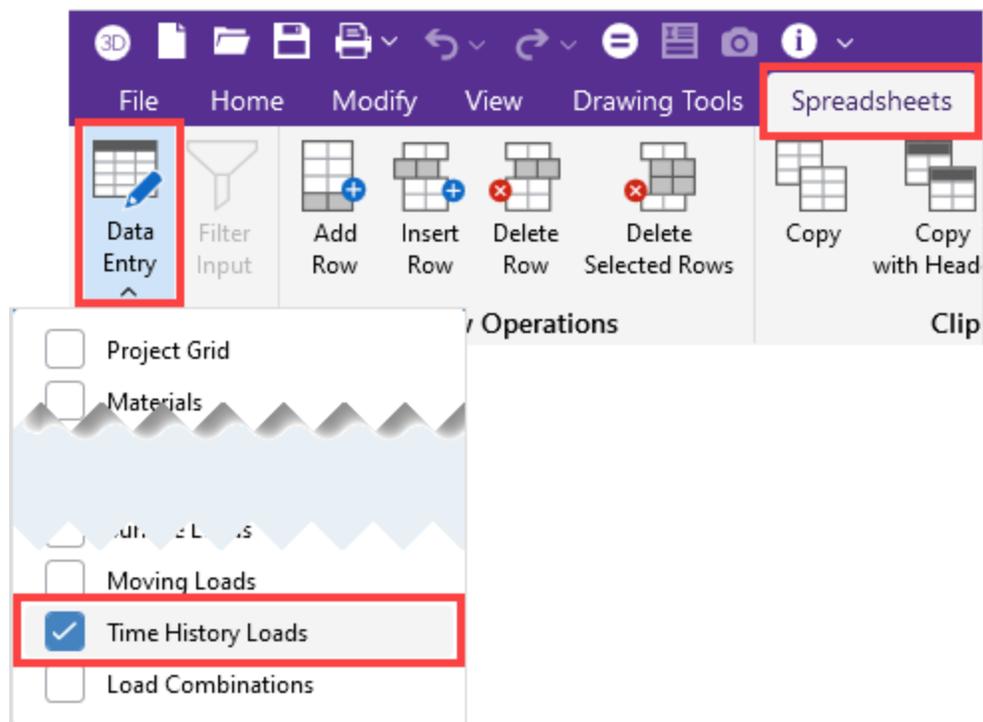
1. Click on **Time History Loads** in the Explorer panel to the right side of the model editing panel.



Alternatively, you can access the spreadsheet from the Spreadsheets ribbon.

To access from the Spreadsheets ribbon

- a. Go to the **Spreadsheets** ribbon.



- b. Click the **Data Entry** icon.
- c. Click the **Time History Loads** checkbox.

The Time History Loads spreadsheet opens.

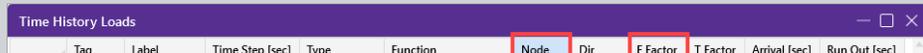
	Tag	Label	Time Step [sec]	Type	Function	Node	Dir	F Factor	T Factor	Arrival [sec]	Run Out [sec]
1	T1	TH load 1		Acceleration	2400RPM-COS	ALL	Z				
2				Acceleration	2400RPM-COS	ALL	X				

The following table provides information about the data in the Time History Loads spreadsheet.

Time History Spreadsheet Columns

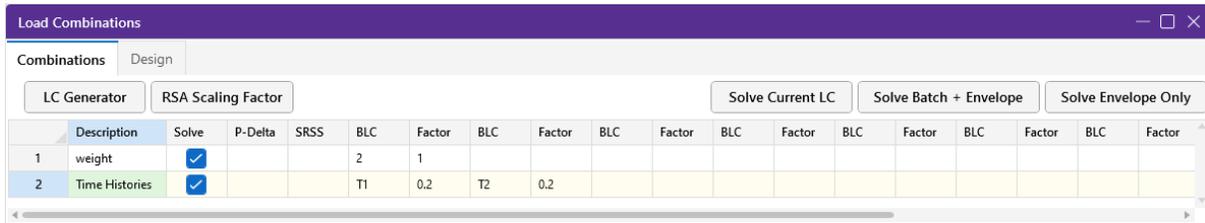
Column	Description
Tag	Each Time History Load definitions is automatically assigned a “Tag” (such as T1, T2, et cetera) as shown on the left side of the spreadsheet. These tags are assigned by the program and may not be edited. Time History Loads are included in the analysis by referencing this Tag in the Load Combinations spreadsheet. Multiple loads or lines of data may be assigned within a given time history tag.
Label	A tab’s Label is editable and is used for your reference as an identifier for that specific set of time history loads. Only one label is allowed per time history tag, even if the tag contains multiple lines of data.
Time Step (sec)	Time Step is used to specify the integration time step used in the time history analysis. Refer to the section titled What integration time step should I use? for more information on this subject. Only one time step is allowed for a given time history tag, even if the tag contains multiple lines of data.

Column	Description
	<div style="border: 1px solid black; padding: 10px; background-color: #e6f2ff;">  Note: <ul style="list-style-type: none"> You have the option of leaving the time step blank. If this is done the program automatically uses a value of 1/10th the lowest mode with at least 10% mass participation. If the time history forcing function were imported, the time step is not taken greater than the time step specified in the imported function. If your input time step is greater than either of the two above limits, a warning log message is produced. Using the HHT method for a Direct Integration solution in combination with a negative alpha integration constant, creates some energy dissipation of higher frequency response. This may allow you to set integration time steps that are greater than the values described above. If multiple Time Histories are applied at the same time for a single load combination, the lowest integration time step is used. If multiple time histories are applied to a load combinations, which do not overlap, each time history is integrated based on the smaller time step. </div>
Type	Type indicates whether this time history function applied to the model is a Force or an Acceleration. Acceleration functions are always assumed to be given as a fraction of the acceleration of gravity. Force functions have units based on the selection for "Forces" in the Units Section window.
Function	Function identifies the Time History Function / Pattern that will be applied to the model.
Node	Node indicates whether the time history load function is applied to the node specified. For seismic time history analysis, the node should be identified as ALL to indicate that the acceleration load is applied to the entire model.
Dir	Dir (Direction Setting) identifies the direction of the load. This is input with respect to the global axes, X, Y, and Z.
F Factor	<p>F Factor is a magnification factor applied to the function. This can be used for any purpose you want. A common use would be to use F Factor to account for an increase in the operating speed of dynamic equipment, or to account for tributary area for a blast analysis. A blank value defaults to 1.0.</p> <p>For example, the BLASTLOADEXAMPLE function is really a time dependent pressure resulting from some form of explosion. The load is applied to individual joints thus the F Factor is used to enter the tributary area for each node. Therefore, this effectively converts the blast load from pressure to force.</p>



Column	Description																																																																																																							
T Factor	<p>T Factor adjusts the time steps in the forcing speed of the forcing function. A factor less than 1 increases the speed of the equipment and a factor greater than 1 decreases the speed of the equipment. This can be a quick and useful way to adjust a given TH function for variations in speed instead of having to create new functions for each case. A blank value defaults to 1.0.</p> <div style="border: 1px solid #ccc; padding: 5px; margin: 10px 0;"> <table border="1" style="width: 100%; border-collapse: collapse; font-size: 8px;"> <thead> <tr> <th>Tag</th> <th>Label</th> <th>Time Step [sec]</th> <th>Type</th> <th>Function</th> <th>Node</th> <th>Dir</th> <th>F Factor</th> <th>T Factor</th> <th>Arrival [sec]</th> <th>Run Out [sec]</th> </tr> </thead> <tbody> <tr><td>1</td><td>T1</td><td>Sin/Cos</td><td>0.001</td><td>Force</td><td>3000RPM(SIN)</td><td>N1</td><td>Z</td><td>-1</td><td></td><td></td></tr> <tr><td>2</td><td></td><td></td><td></td><td>Force</td><td>3000RPM(COS)</td><td>N1</td><td>Y</td><td></td><td></td><td></td></tr> <tr><td>3</td><td>T2</td><td>Sin+Phase</td><td>0.001</td><td>Force</td><td>3000RPM(PHASE)</td><td>N1</td><td>Y</td><td></td><td></td><td></td></tr> <tr><td>4</td><td></td><td></td><td></td><td>Force</td><td>3000RPM(SIN)</td><td>N1</td><td>Z</td><td>-1</td><td></td><td></td></tr> <tr><td>5</td><td>T3</td><td>+10%</td><td>0.000909</td><td>Force</td><td>3000RPM(COS)</td><td>N1</td><td>Z</td><td>1.235</td><td>0.9</td><td></td></tr> <tr><td>6</td><td></td><td></td><td></td><td>Force</td><td>3000RPM(SIN)</td><td>N1</td><td>X</td><td>-1.235</td><td>0.9</td><td></td></tr> <tr><td>7</td><td>T4</td><td>+10%</td><td>0.001</td><td>Force</td><td>3000RPM(COS)</td><td>N1</td><td>Y</td><td>0.826</td><td>1.1</td><td></td></tr> <tr><td>8</td><td></td><td></td><td></td><td>Force</td><td>3000RPM(SIN)</td><td>N1</td><td>Z</td><td>-0.826</td><td>1.1</td><td></td></tr> </tbody> </table> </div> <p>For example, the 0.9 T Factor is used to adjust the 3000 RPM function to something that is about 11% higher ($1/0.9 = 1.11111$). This would be about 3333 RPM.</p> <p>Similarly, use a factor of 1.10 is used to adjust the 3000 RPM function to something that is about 9% lower ($1/1.10 = 0.90909$). This would be about 2727 RPM.</p> <div style="border: 1px solid #0070c0; padding: 10px; margin: 10px 0; background-color: #e6f2ff;"> <p> Note: One important consideration is that the integration time step is not automatically converted based on the T Factor. This is the reason the time step changes from 0.001 seconds to 0.000909 in the example above. This is only necessary in this example because the intent was to have exactly 20 integration steps for each full cycle of the sine wave. This means that there is an integration time step for every pi/10 radians of cyclic motion.</p> </div> <tr> <td style="background-color: #e0e0e0;">Arrival (sec)</td> <td>Arrival indicates the time at which the forcing function starts. A blank value defaults to 0.</td> </tr> <tr> <td style="background-color: #e0e0e0;">Run Out (sec)</td> <td>Run Out indicates how long to continue the time history integration after the function has ended. A blank value defaults to 0.</td> </tr>	Tag	Label	Time Step [sec]	Type	Function	Node	Dir	F Factor	T Factor	Arrival [sec]	Run Out [sec]	1	T1	Sin/Cos	0.001	Force	3000RPM(SIN)	N1	Z	-1			2				Force	3000RPM(COS)	N1	Y				3	T2	Sin+Phase	0.001	Force	3000RPM(PHASE)	N1	Y				4				Force	3000RPM(SIN)	N1	Z	-1			5	T3	+10%	0.000909	Force	3000RPM(COS)	N1	Z	1.235	0.9		6				Force	3000RPM(SIN)	N1	X	-1.235	0.9		7	T4	+10%	0.001	Force	3000RPM(COS)	N1	Y	0.826	1.1		8				Force	3000RPM(SIN)	N1	Z	-0.826	1.1		Arrival (sec)	Arrival indicates the time at which the forcing function starts. A blank value defaults to 0.	Run Out (sec)	Run Out indicates how long to continue the time history integration after the function has ended. A blank value defaults to 0.
Tag	Label	Time Step [sec]	Type	Function	Node	Dir	F Factor	T Factor	Arrival [sec]	Run Out [sec]																																																																																														
1	T1	Sin/Cos	0.001	Force	3000RPM(SIN)	N1	Z	-1																																																																																																
2				Force	3000RPM(COS)	N1	Y																																																																																																	
3	T2	Sin+Phase	0.001	Force	3000RPM(PHASE)	N1	Y																																																																																																	
4				Force	3000RPM(SIN)	N1	Z	-1																																																																																																
5	T3	+10%	0.000909	Force	3000RPM(COS)	N1	Z	1.235	0.9																																																																																															
6				Force	3000RPM(SIN)	N1	X	-1.235	0.9																																																																																															
7	T4	+10%	0.001	Force	3000RPM(COS)	N1	Y	0.826	1.1																																																																																															
8				Force	3000RPM(SIN)	N1	Z	-0.826	1.1																																																																																															
Arrival (sec)	Arrival indicates the time at which the forcing function starts. A blank value defaults to 0.																																																																																																							
Run Out (sec)	Run Out indicates how long to continue the time history integration after the function has ended. A blank value defaults to 0.																																																																																																							

- (Optional) To include Time History Loads in the analysis, reference the load's Tag in the 'Load Combinations' spreadsheet (as shown in the following example image).



What Integration Time Step Should I Use?

The answer to this question is a matter of engineering judgment, of course. And, a good reference on dynamic analysis should be consulted. However, there are a few basic factors that come into play. This section is not meant to provide definitive recommendations on what the engineer should do. Rather it is meant to identify some common practices.

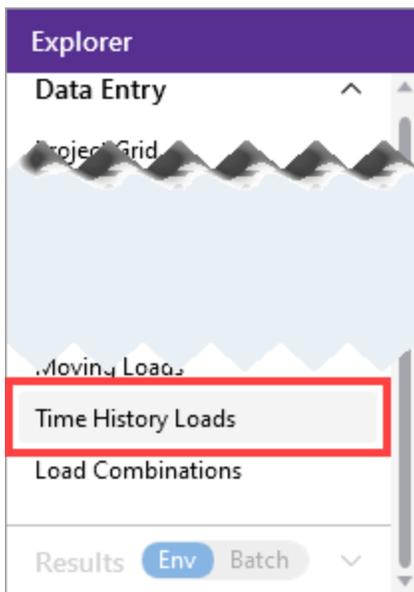
For dynamic equipment with a known excitation frequency, the answer is usually based on the operating frequency. A common choice would be to pick an integration step equal to 1/20th or 1/24th the operating period of the equipment. Though the exact fraction of the operating period can be debated, the 1/20th and 1/24th values ensure there is an integration step for every 18 or 15 degrees of rotation.

For a more random excitation (like seismic loading), the important consideration is more likely to be related to the natural frequencies of the structure which have significant mass participation. A common choice would be to make sure that the integration step is less than 1/10th the smallest significant natural period. Though again, the exact fraction of the significant period can be debated. It may also be necessary, however, to identify the frequency content of the forcing function and use a time step (similar to the 1/20th of the lowest equipment period) that can capture the frequency content of the forcing function.

Apply a Time History Load

To apply a Time History Load:

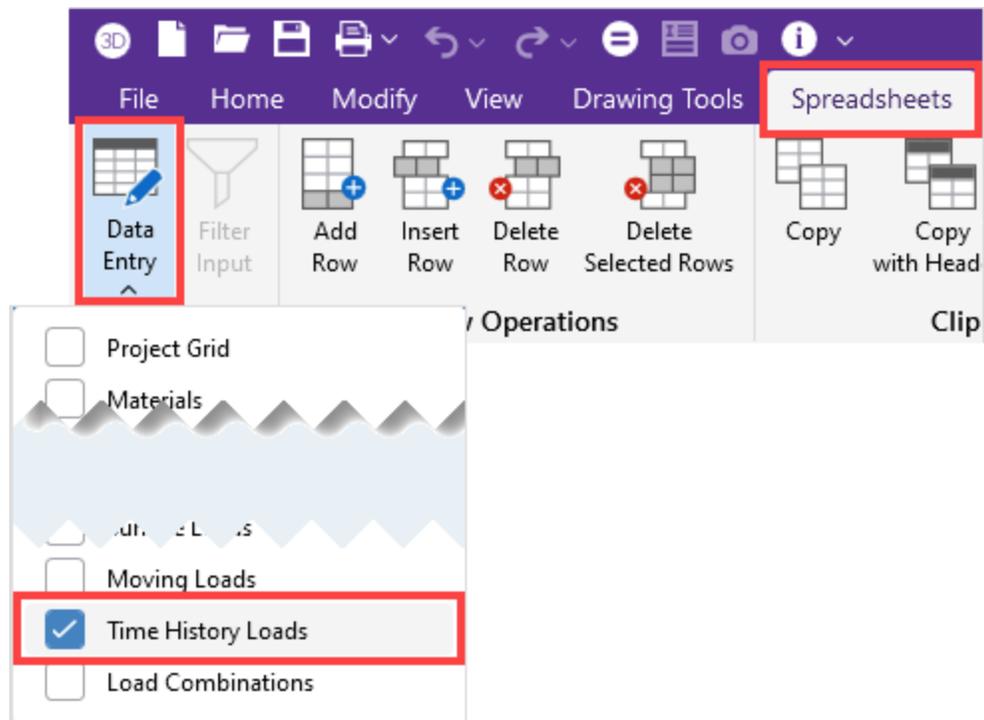
1. Click on **Time History Loads** in the Explorer panel to the right side of the model editing panel.



Alternatively, you can access the spreadsheet from the Spreadsheets ribbon.

To access from the Spreadsheets ribbon

- a. Go to the **Spreadsheets** ribbon.



- b. Click the **Data Entry** icon.
- c. Click the **Time History Loads** checkbox.

The Time History Load spreadsheet opens.

Time History Loads											
	Tag	Label	Time Step [sec]	Type	Function	Node	Dir	F Factor	T Factor	Arrival [sec]	Run Out [sec]
1	T1	Sin/Cos	0.001	Force	3000RPM(SIN)	N1	Z	-1			
2				Force	3000RPM(COS)	N1	Y				
3	T2	Sin+Phase	0.001	Force	3000RPM(PHASE)	N1	Y				
4				Force	3000RPM(SIN)	N1	Z	-1			
5	T3	+10%	0.000909	Force	3000RPM(COS)	N1	Z	1.235	0.9		
6				Force	3000RPM(SIN)	N1	X	-1.235	0.9		
7	T4	+10%	0.001	Force	3000RPM(COS)	N1	Y	0.826	1.1		
8				Force	3000RPM(SIN)	N1	Z	-0.826	1.1		

- Click in the **Function** column, then click the down-arrow and specify a function / pattern (as shown in the following image).

Time History Loads											
	Tag	Label	Time Step [sec]	Type	Function	Node	Dir	F Factor	T Factor	Arrival [sec]	Run Out [sec]
1	T1	Sin/Cos	0.001	Force	3000RPM(SIN) ▾ ...	N1	Z	-1			
2				Force	2400RPM-COS	N1	Y				
3	T2	Sin+Phase	0.001	Force	2400RPM-COS_1	N1	Y				
4				Force	2400RPM-SIN	N1	Z	-1			
5	T3	+10%	0.000909	Force	3000RPM(COS)	N1	Z	1.235	0.9		
6				Force	3000RPM(PHASE)	N1	X	-1.235	0.9		
7	T4	+10%	0.001	Force	3000RPM(SIN)	N1	Y	0.826	1.1		
8				Force	BLASTLOADEXAMPLE	N1	Z	-0.826	1.1		

- In the **Time Step (sec)** column, specify the time (in seconds) between each integration time step.
- Use the **Type** column to indicate whether the entry corresponds to a force / moment or an acceleration.
- Specify the node and direction using the **Nodes** and **Dir** fields.
- The remaining columns adjust the force factor, the time step / period of the input motion and arrival time. These may be left blank if no modification is necessary.



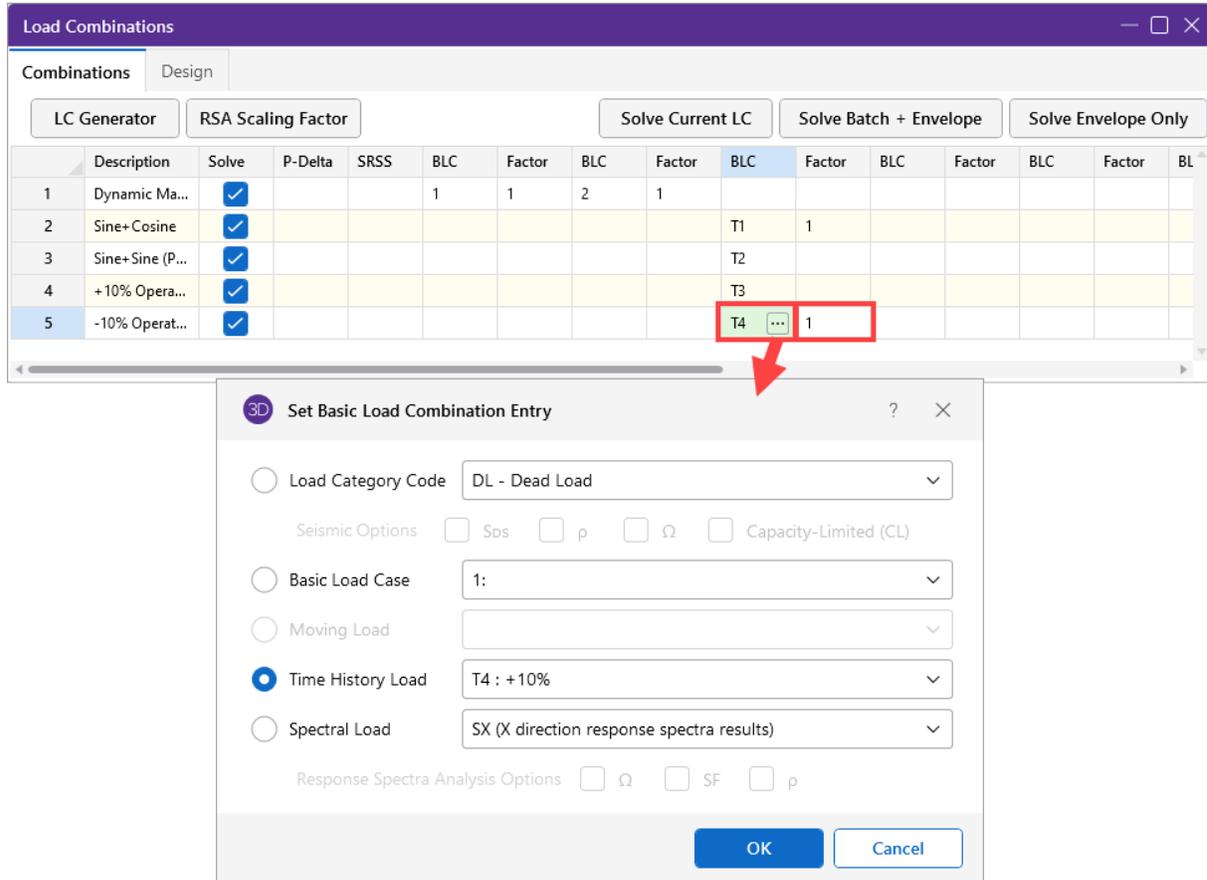
Note: If the load is to apply to all nodes in the model, enter the 'Node' label as "ALL".

Include a Time History Load in a Load Combination

To include a Time History Load in a Load Combination analysis:

- Click on the **BLC** column of the Time History Load you want to include.
- In the **BLC** column, type the Time History load Tag (**T1**, **T2**, etc.) into the cell.

Alternatively, you can click the ellipsis  button to open the **Set Basic Load Combination Entry** window where you can choose the appropriate Time History load from a drop down list, as shown in the following image.



3. In the **Factor** column, type in a corresponding BLC factor.

Required Number of Modes

The time history solution requires the use of the structure's mass matrix. In order to ensure that this matrix is present and valid, RISA requires an active Dynamic / eigen solution before a Time History Analysis can be run. The number of modes that need to be solved, however, depends on the Time History Solution method you choose.

Modal Superposition

For dynamic equipment with a known excitation frequency, the required number of modes is usually related to the operating frequency of the equipment. This means solving for all modes within certain percentage of the operating frequency of the equipment, even if those modes have little mass participation.

For more random excitation (like seismic loading), the important consideration is likely the total mass participation for the solved modes.

**Note:**

- If you are obtaining many modes with little or no mass they are probably local modes. Rather than asking for even more modes and increasing the solution time refer to [Dynamics Troubleshooting – Local Modes](#) to learn how to deal with these unwanted local modes.
- At this time residual mass / missing mass vectors are NOT considered in a Time History analysis. Therefore, models with low mass participation would be more efficiently solved using the Ritz Vectors solution option.

Direct Integration

The number of modes solved during a dynamic analysis does NOT affect the accuracy of the Time History solution when the Direct Integration solution method is chosen. The dynamic analysis is only required to ensure that RISA has a valid stiffness matrix available to be used during the time history solution.

Damping

Modal Superposition

When the Modal Superposition option is chosen, RISA requires you to enter in a single damping value which applies to all modes. While it is technically possible to assign a different damping value for specific modes, RISA does not currently support this behavior.

The modal superposition method does not support the building of a damping matrix. Therefore, your defined damping values, that are associated with translational or rotational springs, are not used in the solution.

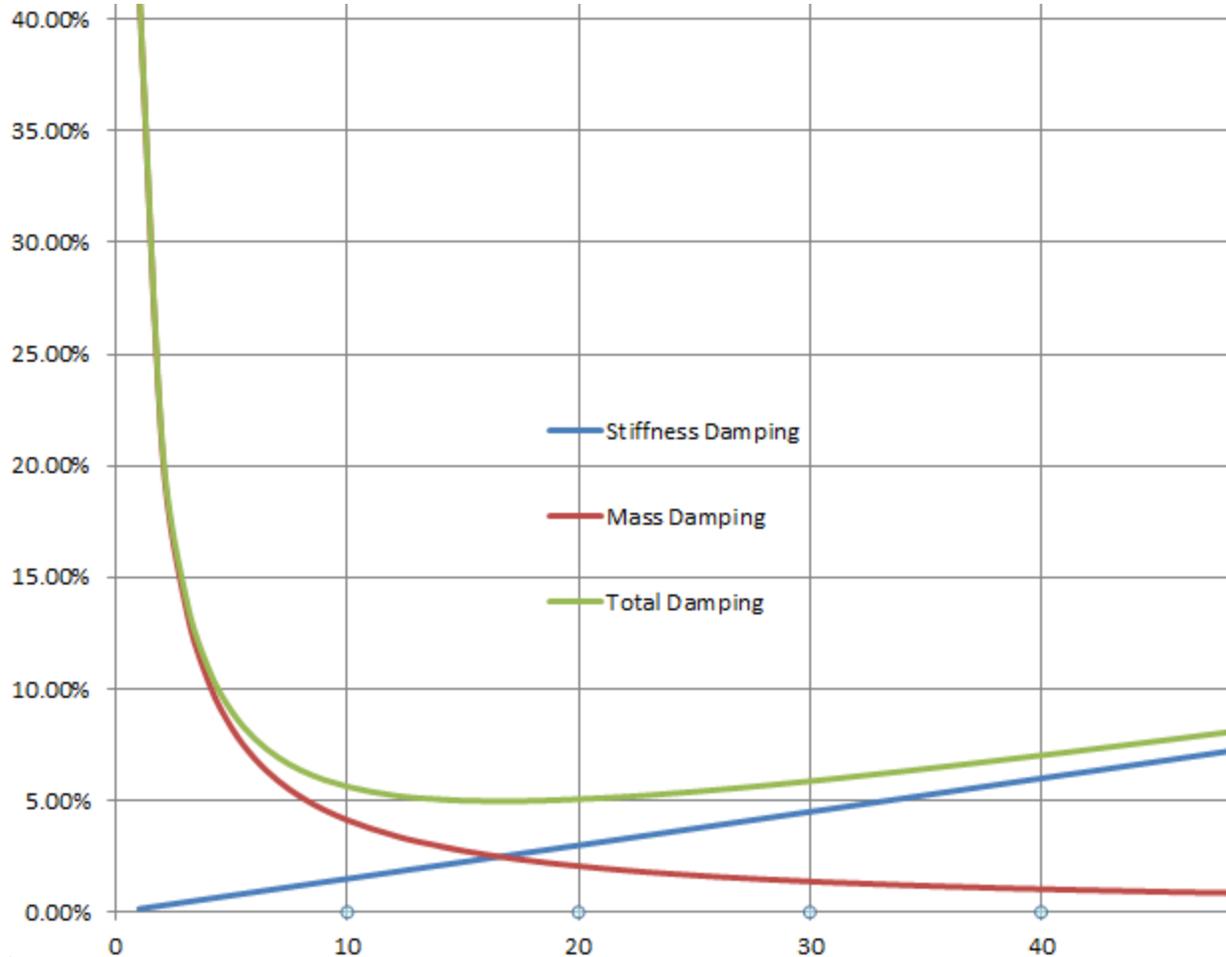
Direct Integration

When the Direct Integration solution option is chosen, the damping matrix is generally built using the classic Rayleigh Damping formulation based on the Mass proportional and Stiffness proportional damping, as shown in the following equations where mu and kappa are merely constants used to form the damping matrix out of the existing mass and stiffness matrices.

$$[C] = \mu [M] + \kappa [K]$$

The relationships between mass and stiffness and frequency means that the % damping values vary with frequency (see curve below). These values can be adjusted so that a targeted damping ratio is achieved at two specific frequencies. In the image below, the target damping ratio was 5% at 15 Hz

and 18.33 Hz. It remains relatively close to 5% between those target frequencies, but will be slightly less than 5%.



You can directly assign the mu and kappa constants, or you can choose to use the window below, which is activated by clicking on the ellipsis button in the advanced solution window of the [Model Settings](#). You need only enter in your targeted damping ratio and the two target frequencies, then the program will calculate a mu and kappa value that produces the desired damping for those frequencies.

1 **Advanced**

2 **Advanced**

3

3D Model Settings

- Solution**
- Members**
 - No. of Reported Sections
 - No. of Internal Sections
 - Member Area Load
 - Consider Shear
 - Consider Torsion
- Wall Panels**
 - Approx. Mesh Size
 - Transfer Forces
 - Increase Work
 - Include P-Delta
 - Optimize Mesh
 - Max. No. of Iterations
- Processor Core Utilization**
 - Advanced

3D Set Advanced Options

Convergence Tolerance

P-Delta: 0.5 %

Eigensolution: 1.E-

Misc

Gravity Acceleration

Merge Tolerance

Solvers

Static

Dynamic

Time History

Modal Superposition Damping: 5 %

Direct Integration (HHT) Alpha: 0

Rayleigh μ : 0.176 κ : 0.011 **Advanced**

Diaphragm

Stiffness: 1.E 7

P-Delta Analysis

Standard Nodal Shear

3D Set Rayleigh Damping Constants

For Rayleigh Damping: $[C] = \mu [M] + \kappa [K]$

- μ = Mass Damping Term
- κ = Stiffness Damping Term

It is possible to set these term to approximate constant damping within a frequency range. Use the Calculate button below to do so.

Target Damping: 5 %

Frequency Range: 15 to 18.333 Hz

Calculate **Cancel**

OK **Cancel**

Set as Default **Apply** **Cancel**



Note: It is technically possible to build the damping matrix so that different material types have different damping values. This could be accomplished by setting different μ and κ values for the different materials. The RISA interface does not currently support this type of damping, but it is being considered for a future release.

User Defined Spring Damping

When the structure is supported on spring boundary conditions, you can directly enter damping values using the boundary conditions spreadsheet or window. This directly sets the values of the damping matrix for those boundary condition degrees of freedom. The most common usage for this is when you want to use Rayleigh Damping for the super structure, but a higher damping ratio to model that represents the higher levels of damping that occur at the soil / structure interface, due to the radiation of the vibrational energy into semi-infinite material that is not part of the structural model.

Soil Properties

For industrial foundations subject to large dynamic forces, one of the variable properties to be considered are the properties (stiffness and damping) of the soil. This section is not meant to provide true recommendations on what the engineer should do. Rather it is meant to identify some common practices and point to some references which may discuss the matter in more depth.

Stiffness and Damping

One of the considerations for soil stiffness is the magnitude of the anticipated response. For most dynamic equipment, the amplitude of the vibration response may be assumed to be low. The type of elastic soil stiffness that is estimated in a soils report, may be intended to limit settlement or prevent soil failure. If so, this would be an inappropriately low estimate for a vibration response. That's because it is based on large strains at ultimate failure levels. Instead, it would be more appropriate to use a higher soil stiffness based on low magnitudes of vibration that are expected during normal operation.

The American Concrete Institute (ACI) has an excellent report titled ACI 351.3-R Foundations for Dynamic Equipment. This document references a number of other good technical sources. But, in the opinion of the Engineers at RISA, this ACI document is probably the best starting point for understanding how to set these parameters. In particular this document describes two common methods for assessing the stiffness properties of the soil.

Richart - Whitman Model: This is a frequency independent method of determining foundation stiffness (vertical, horizontal, and rocking) and damping. Equations for calculating this stiffness are found in section 4.2.1.1 of ACI 351.3-R.

Veletsos Model: This is a frequency dependent criteria for establishing soil stiffness (vertical, horizontal and rocking) and damping. The dominant operating frequency of the equipment would be the normal choice for calculating the Veletsos stiffness and damping values.

Unfortunately, it is not as simple as just calculating a value from these models and using that value. The final decision involves some engineering judgment and experience. Decisions which require engineering judgment:

- What criteria should be used for establishing the soil properties the Richard-Whitman (frequency independent) or the Veletsos (frequency dependent) model?
- How much variation should be assumed in the soil stiffness? Since the soil properties are not known precisely, do you run a separate analysis at +10% and -10% of the base values calculated?
- How do you adjust the damping values? The ACI sections referenced above describe the radiation / geometric damping that would occur. This likely accounts for the majority of the energy loss in the soil, but there is also material damping (somewhere between 2 and 5% of critical damping) that would occur as well.
- What is the maximum damping value allowed per your design criteria? There is often design criteria that will limit the maximum damping to be considered in the analysis to some percentage of critical damping.



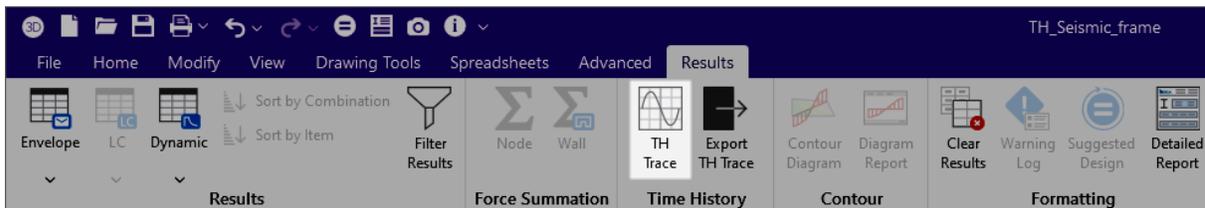
Note: User entered damping values for spring boundary conditions are only considered when using the Direct Integration method of Time History analysis and are not supported when using the Modal Superposition method.

Results - Time History Trace

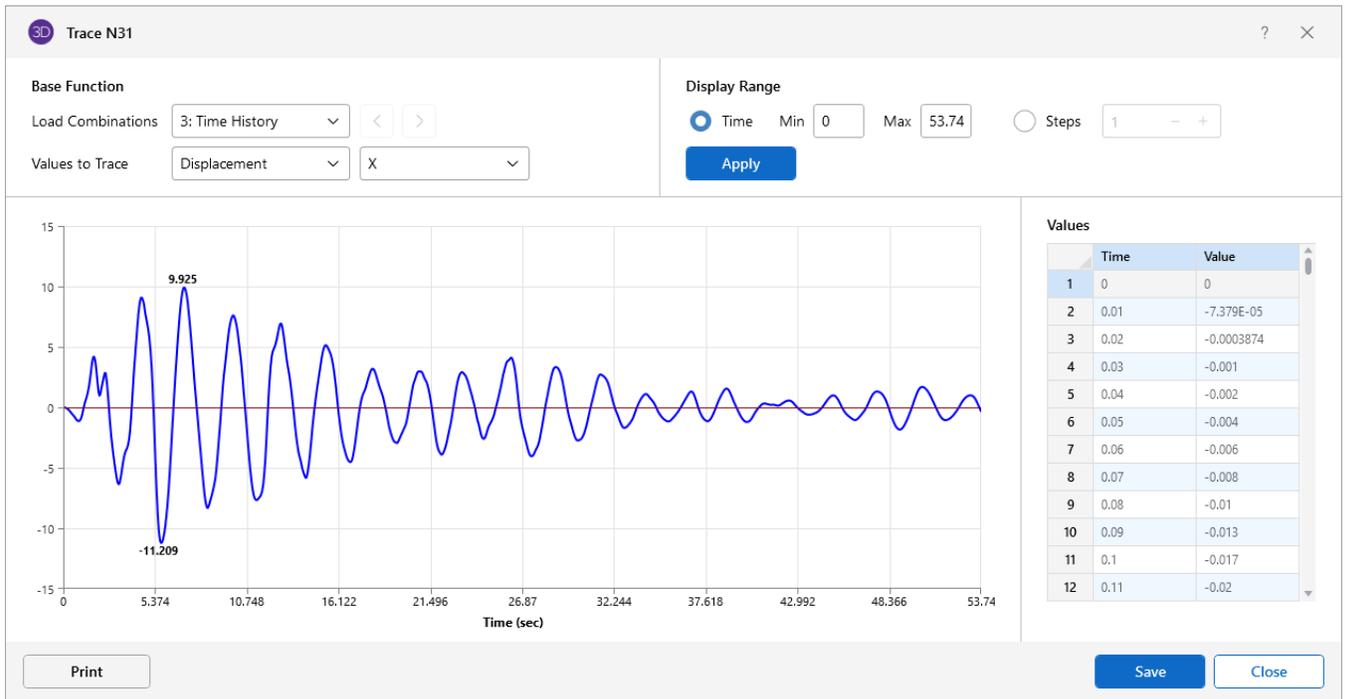
When Time History results are available, you can view a trace of particular node results as a function of time.

To view Time History Trace Results:

1. Go to the **Results** ribbon.



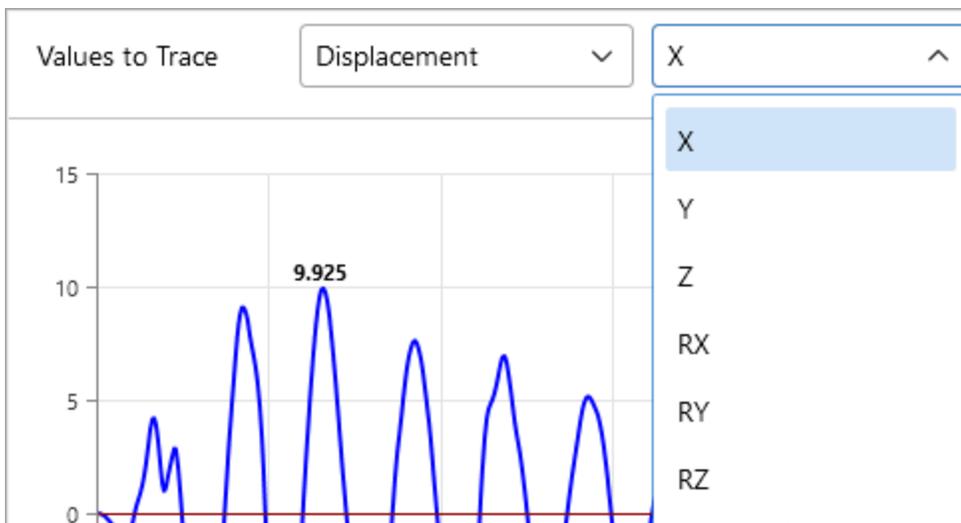
2. Click on the TH Trace icon in the Time History section.
3. In the model, click on the **Node** you want to see the Time History results for.
In the following example, we chose N31.



- (Optional) Click on the **Values to Trace** down-arrow and choose the values you want to view.

Options are Displacement, Velocity, Acceleration or Reaction.

- (Optional) Click on the second block down-arrow (as shown in the following image) to choose a different axis direction to view.



You can display as the X, Y or Z direction. Though, you can also choose to view the RX, RY or RZ rotational displacement, velocity, acceleration or moment reaction. These values are all given with respect to the global axes.

6. (Optional) Use the **Display Range** section to view only selected portions of the solved data, if extra clarity is needed for a relative tight window of time. Make sure you 'Apply' your selection for the view to change accordingly.

Display Range

Time Min Max
 Steps - +

7. (Optional) Click the **Print** button to open the Print Options window.

The Print Options window allows you to print the trace as an image. Several formats may be listed, depending on the programs you have installed on your PC.

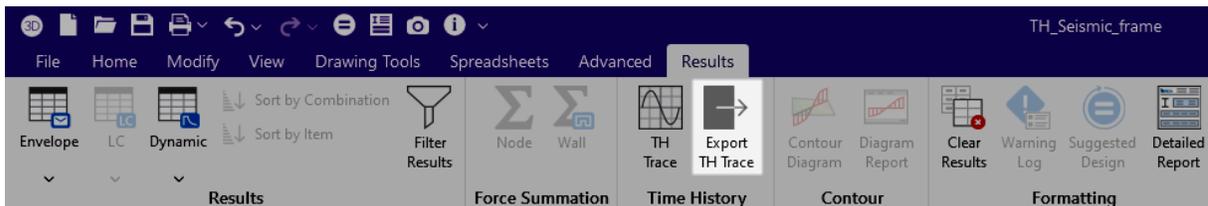
You can also export the data to an ASCII text file that can be opened in Excel or Access. See [Export Time History Traces](#) for instructions.

Results - Export Time History Traces

The Export TH Trace feature allows you to export a series of time history traces to an ASCII text file that can be opened in Excel. This can be done for a graphical selection of nodes.

To export a Time History Trace:

1. Go to the **Results** ribbon.



2. Click on the **Export TH Trace** icon in the Time History section.

This icon becomes available whenever a valid time history solution is present.

The Batch Export Time History Traces window opens.

Batch Export Time History Traces
? X

All graphically selected nodes will be exported.

Values to Export

Displacement ▼

X
 Y
 Z
 MX
 MY
 MZ

Field Delimiter

Single Space ▼

Load Combinations

Range Options All Time History LCs ▼

Range Limits None ▼

to

Report Section Options

Header Style Multi Line ▼

Time Output Steps Only ▼ Include Units

Export
Close

The following table provides descriptions for the sections in this window.

Batch Export Time History Traces

Section	Description
Values to Export	The 'Values to Export' section allows you to choose whether to export displacements, velocities, accelerations, reactions

Section	Description
	or input functions. It also lets you dictate which direction the export is being performed on.
Field Delimiter	The 'Field Delimiter' section allows you to choose what characters are used to separate individual fields.
Load Combinations	The 'Load Combinations' section allows you to export for all the solved time history load combinations or only a single one. It also allows you to export only a specific time range of the overall response. This allows you to ensure that you are getting only the steady response when performing an equipment vibration analysis and that any transient response, due to start-up effects, can be excluded.
Report Section Options	The 'Report Section Options' allows you to control the column headers in the ASCII text files, as well as whether or not you want to see the time values or time steps in the file itself.

3. Choose all the options that correspond to the data you want exported.
4. Click the **Export** button.

Results - Animate Time History Deflection

When a time history load combination has been solved, you can build a animation movie of the deflected shape of the structure as a function of time. You can speed up or slow down the animation to control the length of the movie, and you can pause it and re-start it at a particular time step or time.



Note:

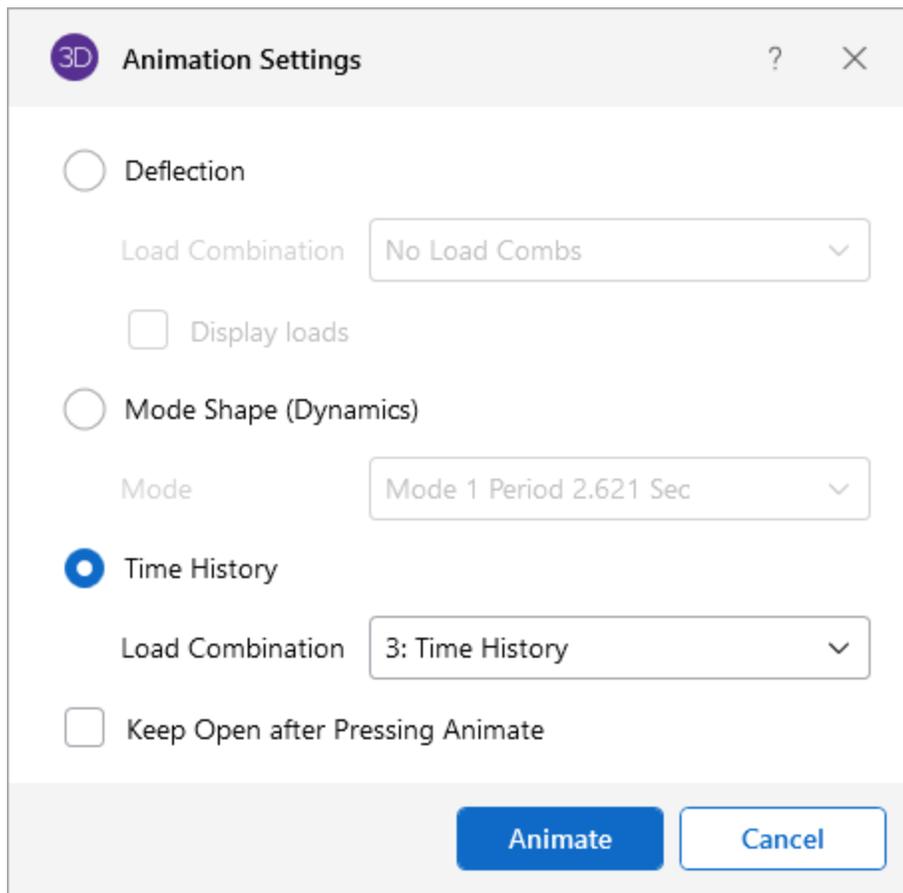
- Time History Deflection diagrams only display joint translation. Joint rotation is not currently displayed.
- The **Export Animation** feature may be used to save a video file of the animated mode shapes.

View a Time History Animation

To view a Time History animation:

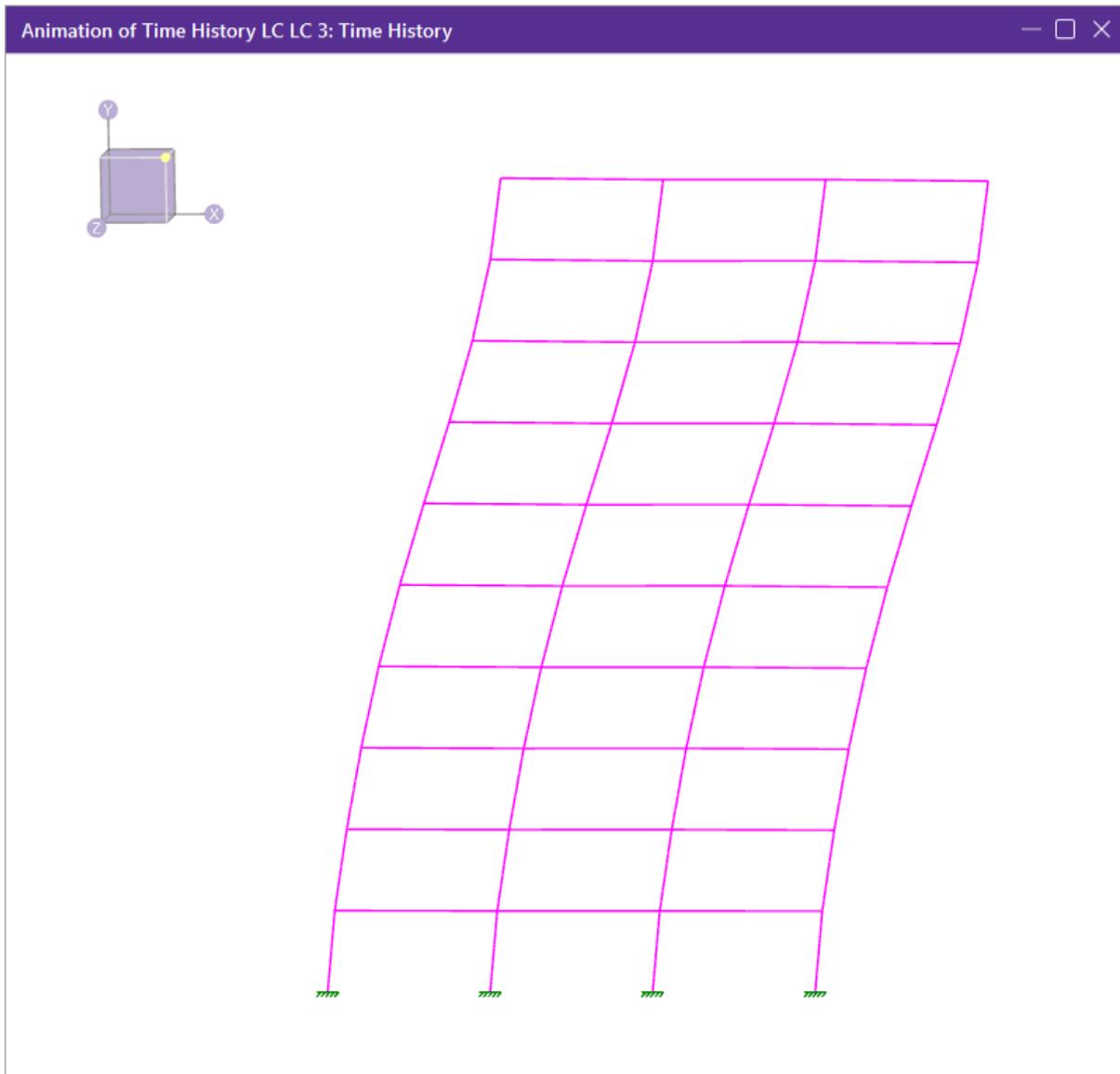
1. Go to the **View** ribbon.
2. Click the **Results** icon in the 'Animate' section.

An Animation Settings window opens.



3. Click on the **Time History** option.
4. Click the **Load Combination** down-arrow and select the load combination you would like to animate.
5. Click on  .

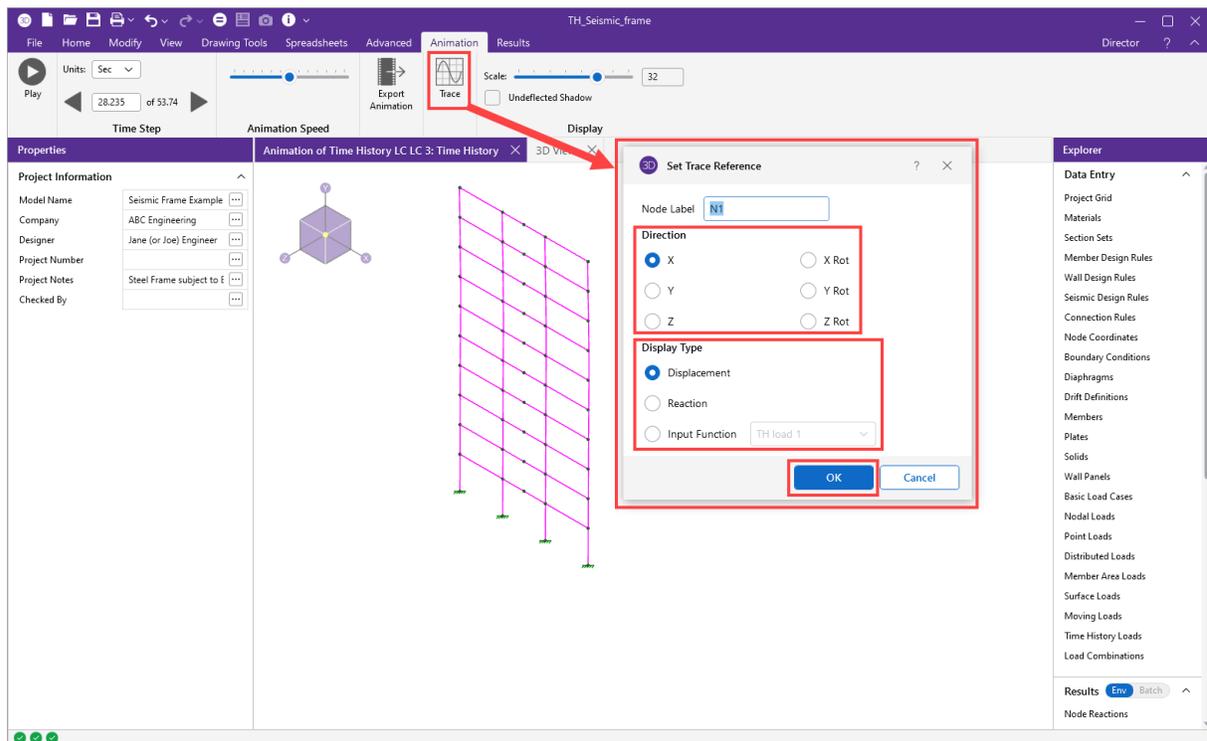
The Animation window opens displaying an animation similar to what is shown here:



Add Trace Data to the Animation

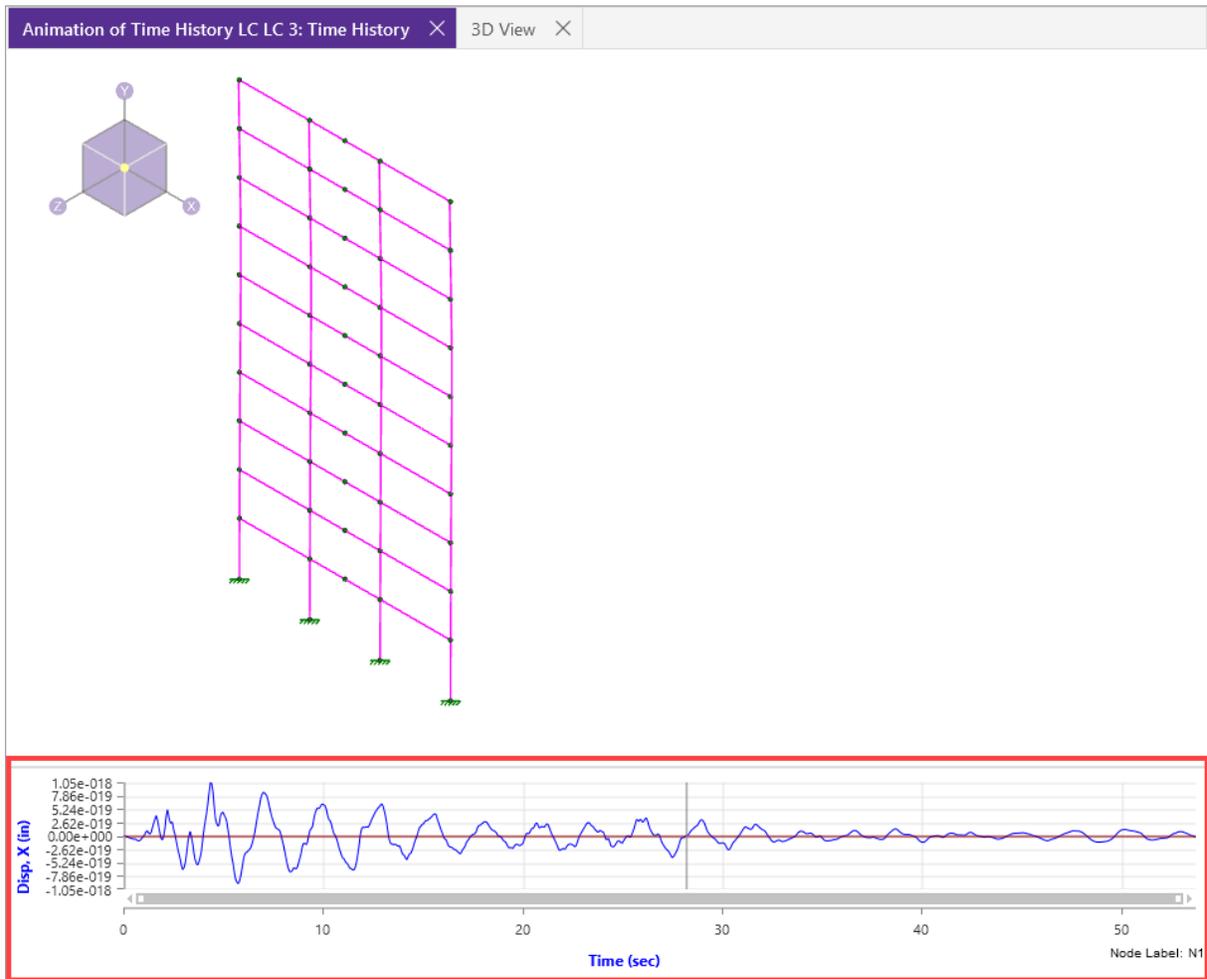
To add Trace Data to the animation:

1. Click the **Trace** icon.
A Set Trace Reference window opens.



2. Click on the node **Direction** you want to specify (X, Y or Z, or X Rot, Y Rot or Z Rot. Note that 'Rot' = rotation).
3. Click on the **Display Type** (Displacement, Reaction or Input Function).
4. Click **OK**.

A Time History Trace of the specified joint result or input function is appended to the bottom of the deflection animation, as shown in the following image.



Member Stiffness Issues

Cracking for Concrete Member

The *Design* tab of the Load Combinations spreadsheet has a 'Service' checkbox which indicates whether the loading is at service level or ultimate level.

The screenshot shows the "Load Combination Design" window with the "Design" tab selected. It features a table with columns for "Description", "ASIF", "CD", "Service", "Hot Rolled", "Cold Formed", "Wood", "Concrete", "Masonry", "Aluminum", "Stainless", and "Connection". The "Service" column contains checkboxes for each row, and this column is highlighted with a red box.

	Description	ASIF	CD	Service	Hot Rolled	Cold Formed	Wood	Concrete	Masonry	Aluminum	Stainless	Connection
1	Basic			<input type="checkbox"/>	<input checked="" type="checkbox"/>							
2				<input type="checkbox"/>	<input checked="" type="checkbox"/>							
3	Time Hist...			<input type="checkbox"/>	<input checked="" type="checkbox"/>							

Service - checked: If the Service checkbox is checked, the concrete stiffness will be the larger stiffness assumed for service level loads.

Service - unchecked: If the Service checkbox is unchecked, the concrete members will have the lower stiffness that is applicable for ultimate level loads.

**Note:**

- This applies to both Concrete members and concrete walls.
- This does NOT currently apply to Masonry walls, which always assume a level of cracking that occurs at ultimate level loads.
- Refer to relevant sections in the [Concrete Design](#), and [Wall Panel](#) topics for more information.

What Solution Method should I Use?

Two different Time History procedures are discussed in detail in the [Solution](#) topic.

The choice between the two solution methods comes down to questions about solution speed and damping. The 'Modal Superposition' method is faster for models that don't require large number of modes in the dynamic solution. But, the advantage in solution speed gets reduced as the number of modes solved gets very high. Therefore, this solution method would normally be used in cases where solution speed is the main consideration.

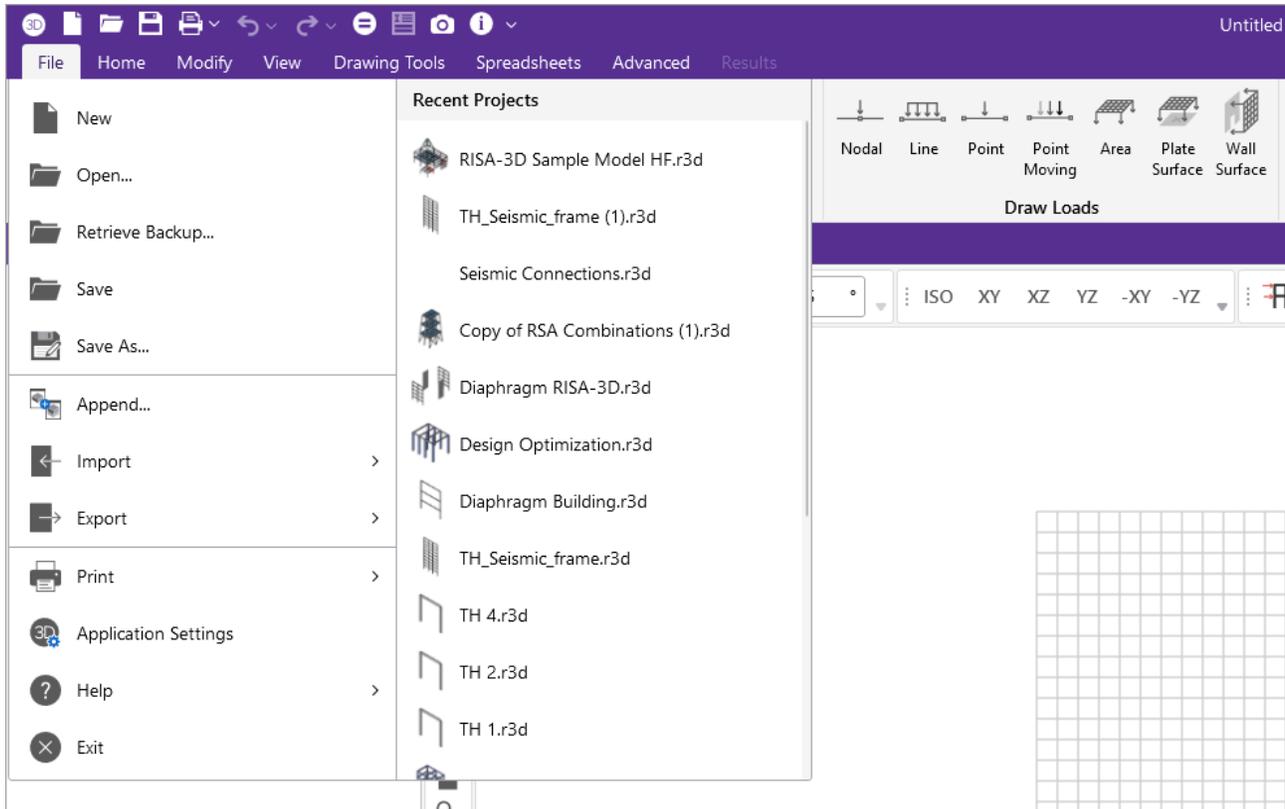
The 'Direct integration' procedure doesn't technically require any modes to be solved. However, RISA requires that one mode be solved so that the program can build and validate the mass matrix before the Time History solution. Therefore, in models where it is very difficult to capture enough modes to get a comfortable level of mass participation, the Direct Integration procedure would likely be chosen. In addition, the Direct Integration method allows you to directly set damping values at boundary condition springs. Therefore, in models where this extra sophistication in the damping terms is required, the Direct Integration method is the obvious choice.



Note: The solution time may be reduced by setting the members', solids', plates' or wall panels' activation status to "Active but Excluded". Deflections will still be calculated for all nodes, but force and stress results for the excluded elements will not be produced.

File Operations

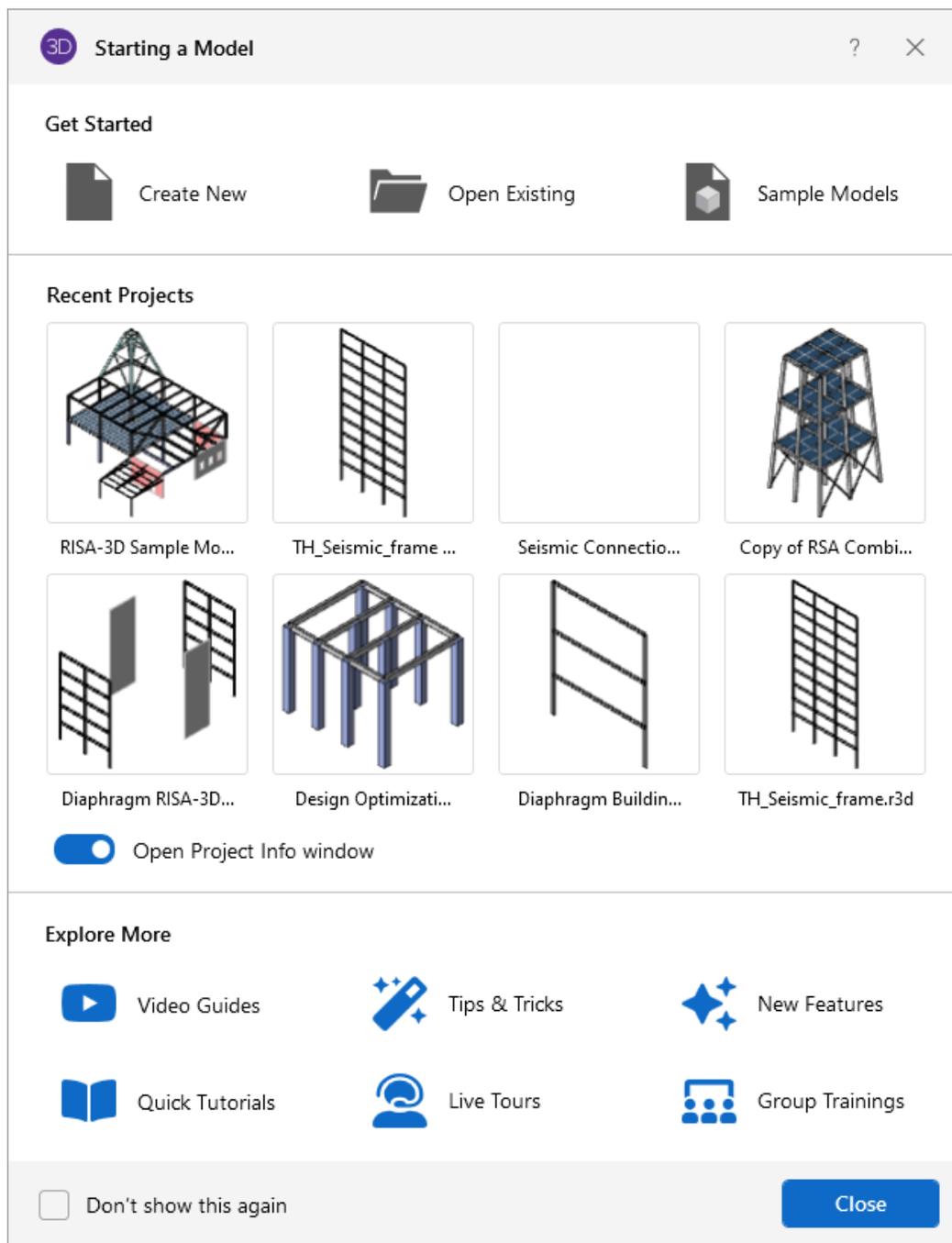
You may create, save, and recall files from the [File Menu](#) ribbon or from the [Quick Access Toolbar](#). Importing and exporting other file types may also be accomplished from the [File Menu](#), as shown in the following image.



Start Off

Starting a Model window

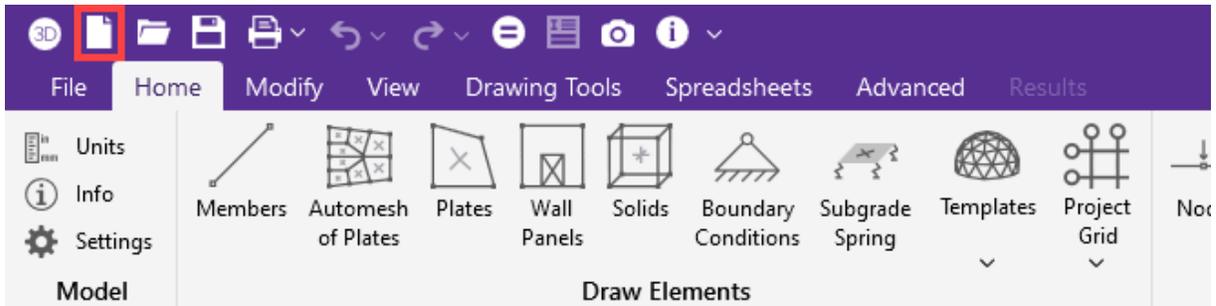
When you open the program, you are presented with a 'Starting a Model' window that includes the following options: Create New, Open Existing, Sample Models and Recent Projects. The bottom area of the window contains an Explore More section that provides links to other helpful options for learning about and working with RISA-3D.



Create a New Model

- Click the  **Create New** icon in the Starting a Model window.
OR

- Click the **New Model** icon on the Quick Access Toolbar, if you've already closed the 'Starting a Model' window, turned off the display of it (checked the **Don't show this again** checkbox) or currently have a model open in the program.

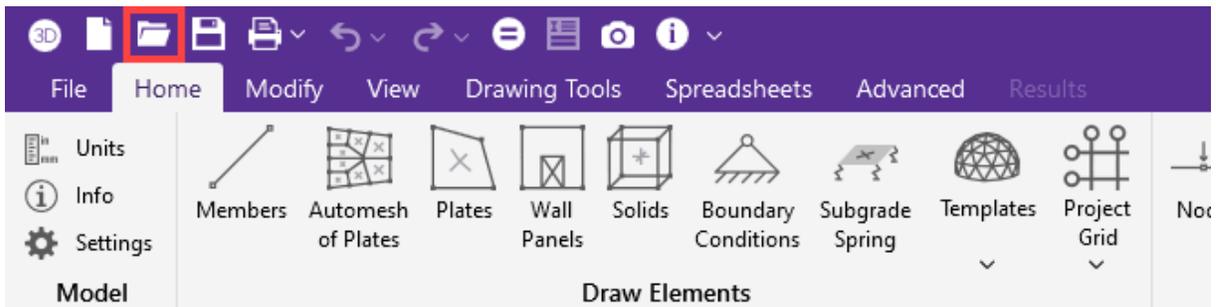


You are prompted to 'Save' if you have made any changes to the current model that has not yet been saved.

Open an Existing Model

To open an existing model:

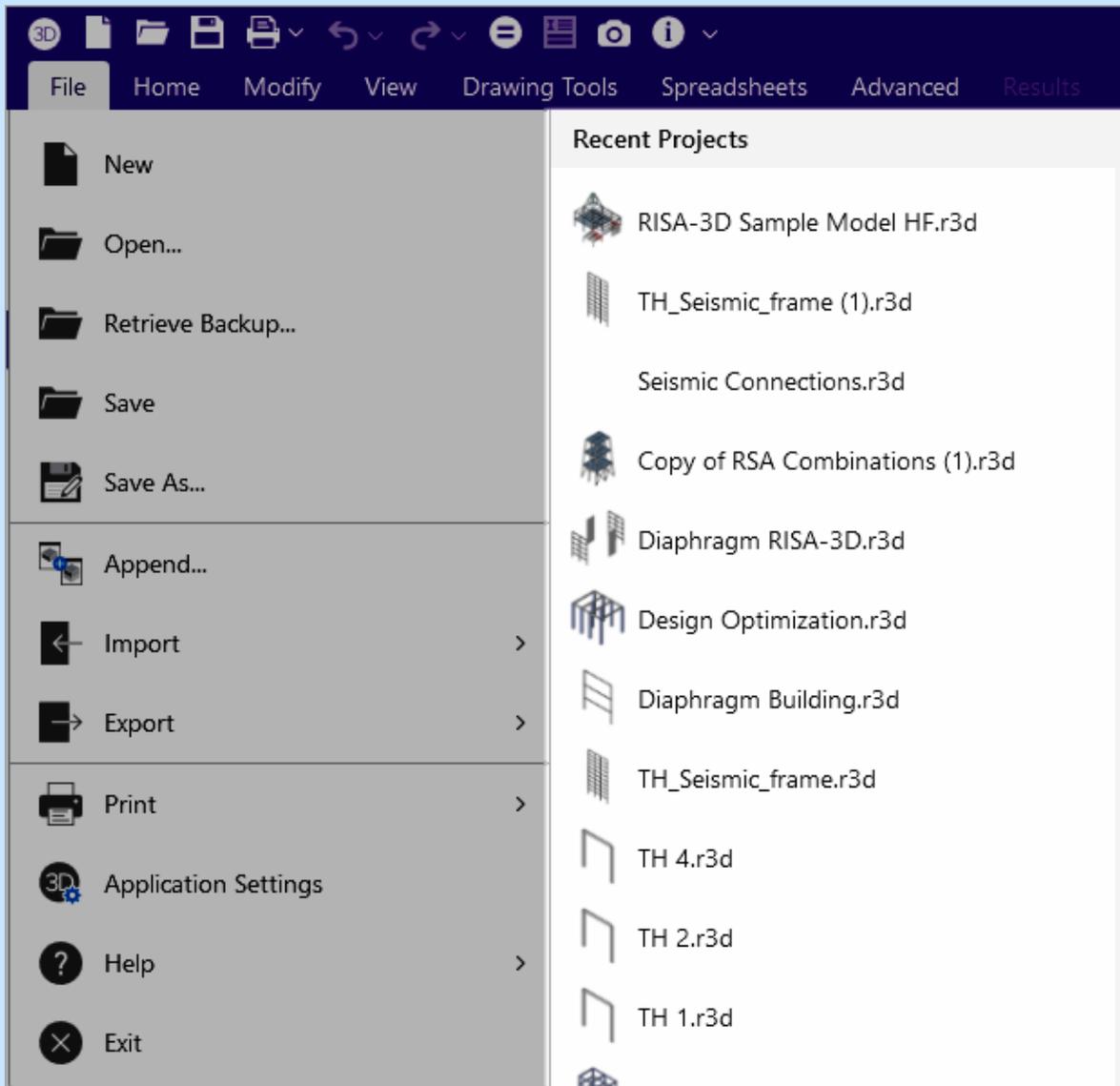
- Click the **Open Existing** icon in the Starting a Model window.
OR
- Click the **Open Model** icon on the Quick Access Toolbar, if you've already closed the 'Starting a Model' window, turned off the display of it (checked the **Don't show this again** checkbox) or currently have a model open in the program.



You are prompted to 'Save' if you have made any changes to the current model that have not yet been saved.

**Tip:**

To open a model you have used recently, click the **File Menu** and select from the list of 'Recent Projects' (shown to the right of the standard menu). Each model is listed along with a preview image of the rendered structure.



Open a Sample Model

To open a sample model:

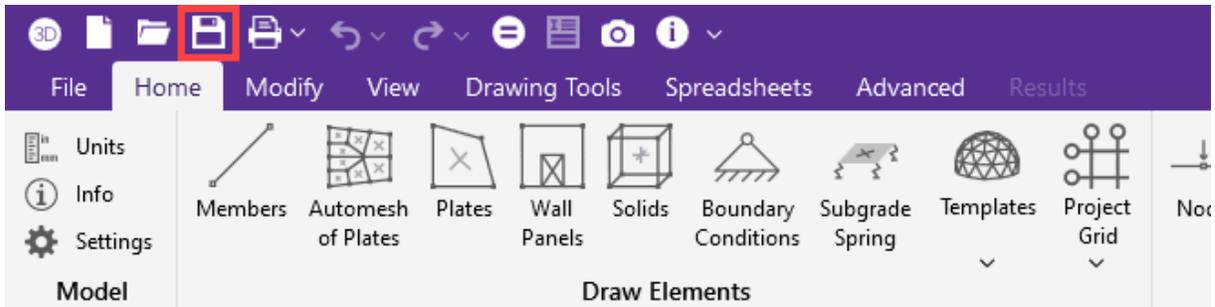
- Click the  **Sample Models** icon in the Starting a Model window.

Save a Model

New Untitled Model

To save a new, untitled model:

- Click the **Save Model** icon on the Quick Access Toolbar.

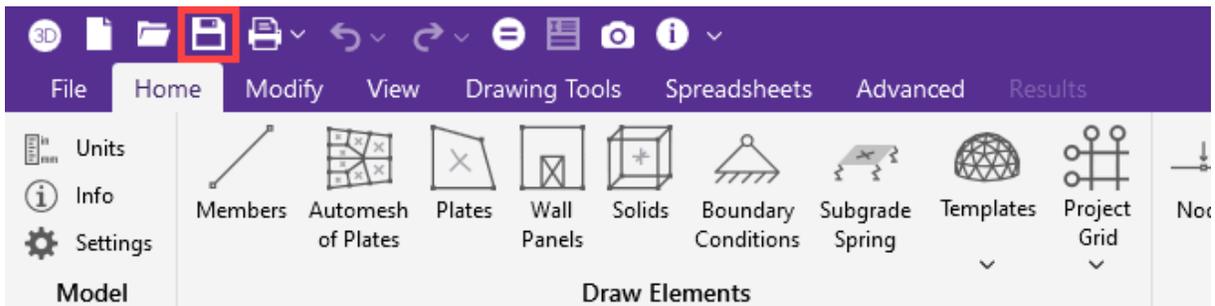


You are prompted to 'Save' the model by giving it a file name and specifying a folder to save it to.

Existing Model

To save an existing model:

- Click the **Save Model** icon on the Quick Access Toolbar.



You are not prompted to save the model. Instead, the model file is simply saved with its current file name and in the folder you've previously specified.

Append Models

You can append other models into the current model by choosing **Append** from the 'File' menu. This lets you have a library of common model parts that you can use later or to lets you divide projects into smaller parts and later combine them. This feature does not allow you to combine two unopened models. You must first open one of the models, and then append the second model into the first model. If you want to have an unmodified copy of the first model, you need to save the new combined model under a new file name.

You can only append files from version 6.0 and newer files. If you've got an older file that you want to append, you first need to open that file by itself and save it.

Your appended model will be placed in the current model using the original coordinate data that was in the appended model. The appended model is NOT moved to the origin of the current model. So, if the appended model had an X coordinate of 9.0 for all joints, then it will be placed at a X coordinate of 9.0 in the current model.

When appending, the **Drawing Toolbar** is already open in the model view, so you can easily adjust the placement of your appended model using the graphics editing tools. The entire original model is automatically unselected and your appended model is selected. If you're going to make several adjustments to the position of your appended model, you should probably click the **Lock Unselected**  icon in the '3D View' panel 'Selection' toolbar, then use the graphical editing tools to put your appended model where you want it in the current model.

You can use the labels for the Joints, Members, and Plates to help identify the parts as you append them into your current model. Use the **Relabel Joints**, **Relabel Members** and **Relabel Plates** options on the 'Tools' menu to make the label prefix unique for each of your model parts before you append them.

**Note:**

- The appended model properties will be set to the current model properties in cases where there is a conflict. For example, if both models use the same section set label, the current model properties will be used to define the section set. The same is true for material properties, moving loads, and so on.
- Loads remain in the same basic load case. For example if the appended model has loads in BLC 3 these loads will be placed in BLC 3 of the current model.
- tnxTower model files may also be appended together.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Append**.

Import and Export

You can import RISA-2D files, STAAD files, tnxTower files, DXF drawing files and BIM Exchange files. See [DXF Files](#), [Appendix C](#), and [Appendix E](#) for more on these file types.

Import Other File Types

To import other file types:

1. Click on the **File** menu.
2. Click on **Import**.
3. Choose the **File Type**.
4. Choose the file you wish to import.

5. Choose from the options associated with the type of file.
6. Perform a Model Merge. See [Model Merge](#) for more information.

Export Other File Types

To export other file types:

1. Click on the **File** menu.
2. Click on **Export**.
3. Choose the **File Type** from the list.
4. Choose the file you wish to export.
5. Choose from the options associated with the type of file.

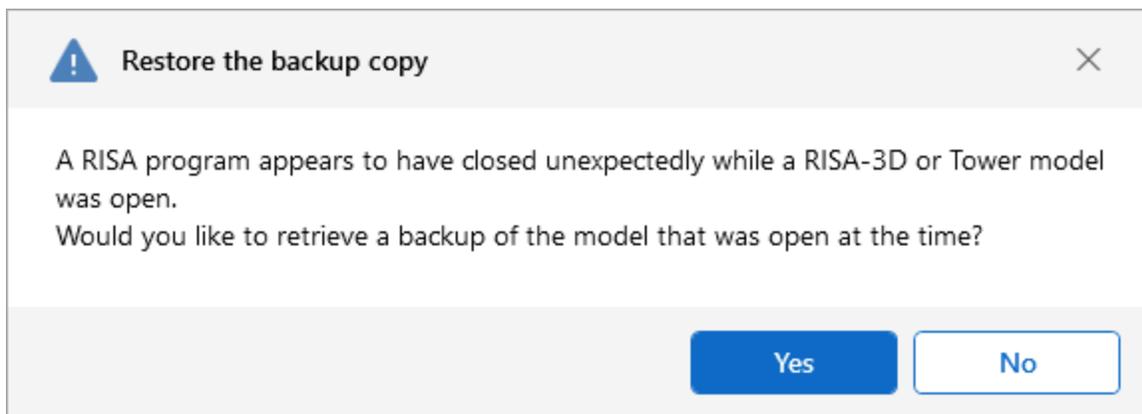
Importing RISA-2D (v4.0 and Newer)

RISA-2D models can only be imported if they are from version 4.0 or higher and are imported into the 3D XY plane. "PIN" member releases are converted to "BENPIN" in RISA-3D. If the 2D members are not assigned a database shape, the torsional stiffness, J, is set to '1'. The out of plane translational degree of freedom (Z), and the out of plane rotational degrees of freedom (2y and 2x), are fixed in the 3D model for all joints.

Automatic Backup

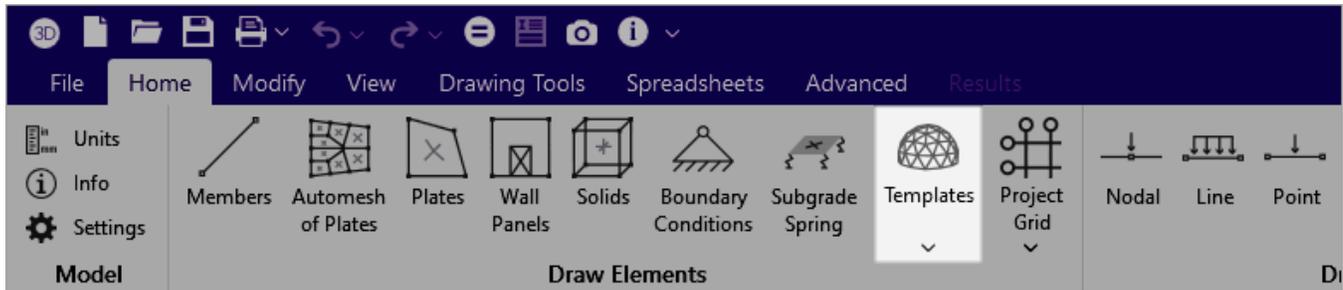
RISA-3D provides an automatic backup for systematically backing up your models. The purpose of this is to provide a means for you to recover your data in the event RISA-3D or your computer closes unexpectedly. As you work within RISA-3D your model is saved with a "_backup" suffix to a model backup folder. By default, this automatic save operation takes place every 5 minutes, but you can alter the timing and location with the [Application Settings](#) option in the **File Menu**. Backups for the last 100 models that you worked on are saved, although you can control how many models are backed up with the [Application Settings](#).

If RISA-3D does close unexpectedly, the next time you start the program the following message appears:

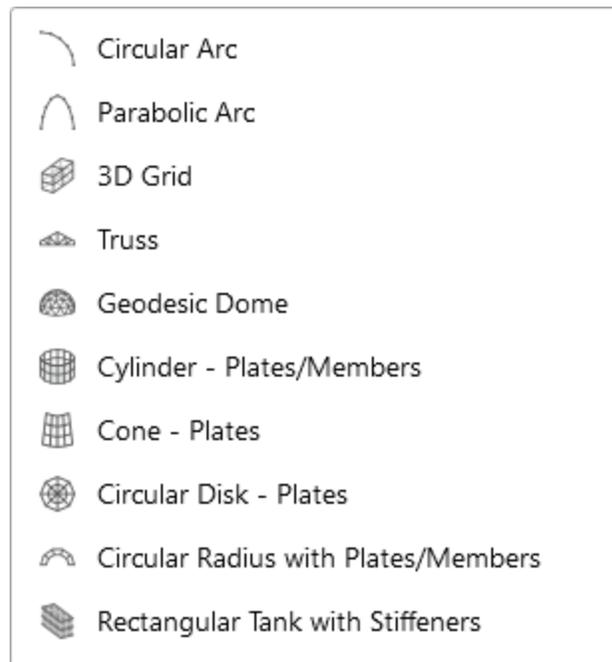


Generation

You can automatically generate regular structures or portions of structures to start a new model or add to an existing model.



Access the Generation types from the **Home** tab by clicking on the **Templates** button. 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.

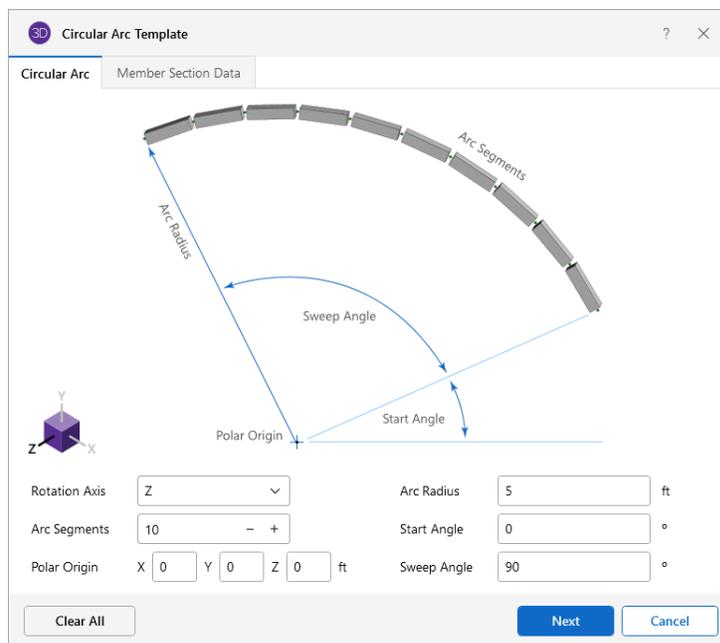


**Note:**

- 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 nodes, members and plates or just a cone of nodes.
- The generations are powerful but as a result of this some also have many options. Remember that any generation may be undone by clicking **Undo** so don't hesitate to try something just to see what it does.

Circular Arc Generation

The circular arc generation lets you generate a full circle or partial arc comprised of beam elements.



Circular Arc Generation Options

Option	Description
<i>Circular Arc - tab</i>	
Rotation Axis	Allows you to define the axis to which the direction of the arc will be oriented.
Arc Segments	Allows you to define the number of segments the arc is broken into.
Polar Origin	Allows you to define the reference point from which the arc height and width are measured, and the rotation axis is defined.
Arc Radius	Allows you to define the distance from the polar origin to the highest

Option	Description
	point on the parabolic arc.
Start Angle	Allows you to define the starting angle of the circular arc.
Sweep Angle	Allows you to define the ending angle of the circular arc.
<i>Member Section Data - tab</i>	
Material	Allows you to define the material used to assign the arc member.
Section Set	Allows you to define the section set assigned for the arc member.
x-x Rotation	Allows you to define the local x-axis rotation of the arc relative to the global X axis.
Member Prefix	Allows you to define prefixes to member labels for members created using the circular arc generator.
Node Prefix	Allows you to define prefixes to node labels for members created using the circular arc generator.

The polar origin is the center point of the arc. A global axis (X, Y or Z) is entered as the axis of rotation and the arc is in the plane normal to this axis. You can generate an arc the full 360 degrees around the axis of rotation or generate a partial arc by specifying the start and sweep angles.

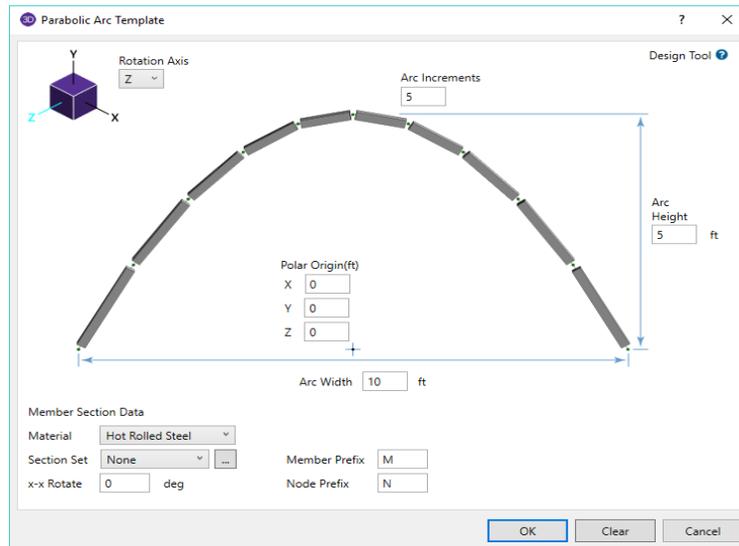
The arc radius is the length from the polar origin to the arc. The arc increment determines how many piecewise straight segments are used to model the arc.

To generate members for the arc, you must select a valid section set. This section set is used for all parts of the arc. If you don't select a section set, it generates an arc of joints without members. The "x-axis rotate" can 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 can also have unique labels assigned to the generated members, by entering a start label.

For additional advice on this topic and how to use the Circular Arc to model a Dome structure, please see the RISA Tips & Tricks website: www.risa.com/post/support. Type in Search keywords: **3D Dome**.

Parabolic Arc Generation

The Parabolic Arc generation is used to make arcs that are parabolic. The arc model is comprised of nodes and optional members.



Parabolic Arc Generation Options

Option	Description
<i>Parabolic Arc - tab</i>	
Rotation Axis	Allows you to define the axis to which the direction of the arc will be oriented.
Arc Segments	Allows you to define the number or segments the arc is broken into.
Polar Origin	Allows you to define the reference point from which the arc height and width are measured and the rotation axis is defined.
Arc Height	Allows you to define the distance from the polar origin to the highest point on the parabolic arc.
Arc Width	Allows you to distance between the two ends of the parabolic arc.
<i>Member Section Data - tab</i>	
Material	Allows you to define the material used to assign the arc member.
Section Set	Allows you to define the section set assigned for the arc member.
x-x Rotation	Allows you to define the local x-axis rotation of the arc relative to the global X axis.
Member Prefix	Allows you to define prefixes to member labels for members created using the parabolic arc generator.
Node Prefix	Allows you to define prefixes to node labels for members created using the parabolic arc generator.

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.

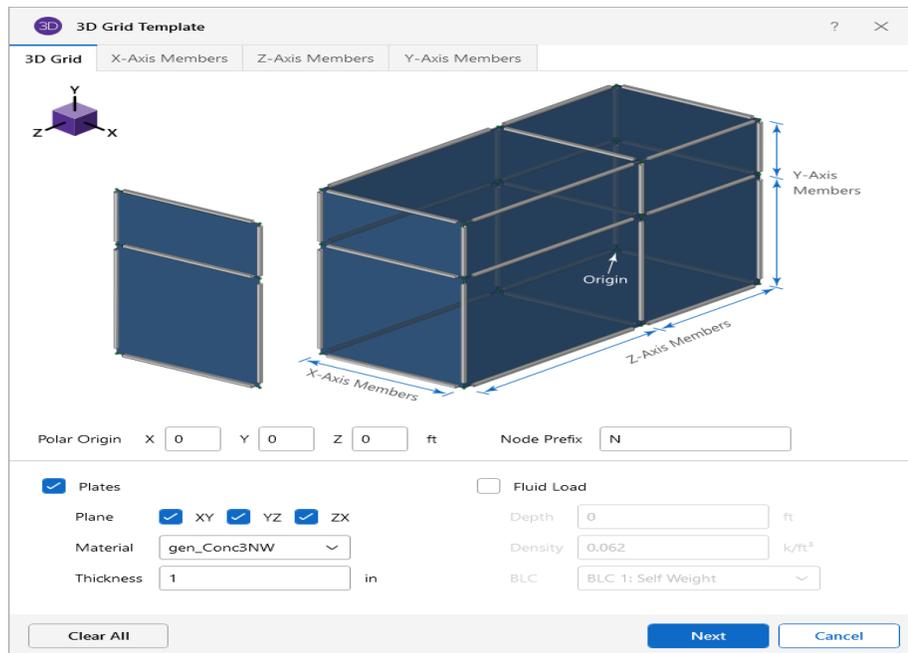
Enter the number of arc increments to be used to model the arc. The more increments used, the more closely the final geometry follows the desired parabola. The minimum increments are two, which gives you a triangular shape.

You may optionally enter a Node Label prefix that is used for all the nodes generated as part of the arc. This is useful if you want a way to track individual parts of your model by looking at node prefixes.

To generate members for the arc, you must select a valid section set. This section set is used for all parts of the arc. If you don't select a section set, it generates an arc of nodes without members. The "x-axis rotate" can be used to rotate the local axes to a desired orientation, however you may find that a K-joint better serves this purpose in some instances. You can also have unique labels assigned to the generated members, by entering a start label.

Grid Generation

This generation enables you to generate 2D or 3D grids (or even 1D (line) grids) of nodes, plates, and members with equal or unequal grid increments.



3D Grid Generation Options

Option	Description
<i>3D Grid - tab</i>	
Polar Origin	Polar Origin defines the reference point from which the arc height and width are measured and the rotation axis is defined.
Node Prefix	Node Prefix defines prefixes to node labels for members created using the parabolic arc generator.
Plates	(checkbox) When selected, this option draws plates between members generated using the Grid Generation.
Plane	Allows you to define which plane the plates are generated for.
Material	Allows you to define the material used to assign to plate elements.

Option	Description
Thickness	Allows you to define the thickness of the plate elements.
Fluid Load	(checkbox) When selected, this option automatically generates hydrostatic loads to the generated plates.
Depth	Allows you to define the height of which the hydrostatic loads should be applied onto the structure.
Density	Allows you to define the density of the hydrostatic load that is applied to the structure.
BLC	Allows you to define the Basic Load Case at which the hydrostatic loads will be generated.
<i>X-Axis Members - tab</i>	
X-Axis Members	(checkbox) When selected, this option toggles the members being drawn in the X-Axis.
Material	Allows you to define the material used to assign the member.
Section Set	Allows you to define the section set assigned for the member.
x-Axis Rotate	Allows you to define the local x-axis rotation of the member relative to the global X axis.
Member Prefix	Allows you to define prefixes to member labels for members created using the Grid generator.
Segments	(checkbox) When selected, this option allows for the drawing of members into several segments or a single member.
No. of Increments	Allows you to define how many segments of members to drawn.
Panel Length	Allows you to define the length of each panel.
<i>Z-Axis Members - tab</i>	
Z-Axis Members	(checkbox) When selected, this option toggles the members being drawn in the Z-Axis.
Material	Allows you to define the material used to assign the member.
Section Set	Allows you to define the section set assigned for the member.
x-Axis Rotate	Allows you to define the local x-axis rotation of the member relative to the global X axis.
Member Prefix	Allows you to define prefixes to member labels for members created using the Grid generator.
Segments	(checkbox) When selected, this option allows for the drawing of members into several segments or a single member.
No. of Increments	Allows you to define how many segments of members to drawn.
Panel Length	Allows you to define the length of each panel.
<i>Y-Axis Members - tab</i>	
Y-Axis Members	(checkbox) When selected, this option toggles the members being drawn in the Y-Axis.
Material	Allows you to define the material used to assign the member.
Section Set	Allows you to define the section set assigned for the member.
x-Axis Rotate	Allows you to define the local y-axis rotation of the member relative

Option	Description
	to the global X axis.
Member Prefix	Allows you to define prefixes to member labels for members created using the Grid generator.
Segments	(checkbox) When selected, this option allows for the drawing of members into several segments or a single member.
No. of Increments	Allows you to define how many segments of members to drawn.
Panel Length	Allows you to define the length of each panel.

Define the X, Y, and Z coordinates of the origin and the increments for spacing in up to three global directions. If you don't define any increments parallel to a particular axis no grids are generated in that direction. Negative increments are OK. Nodes are created at all the grid intersection points.

You can create members in the global directions between the grid points. You can 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 can have unique labels assigned to the generated members, by entering a start label. You can also use different labels for the three member directions.

You can create plates in a global plane between the grid points. Keep in mind that the grid increments you define and the global plane(s) 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 nodes. If you then enter "ZX" for the plane of the plates, no plates are 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 the first Material listed on the General tab of your Materials spreadsheet.

A hydrostatic load can also be generated along the grid of plates as you generate it. The hydrostatic load is generated using a series of uniform surface loads on your plate elements. If the fluid depth is constant along the grid, you just get uniform loads on your plates. If the fluid depth varies along the grid, you 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 is perpendicular to the plates.

The fluid depth is measured in the positive vertical direction from the starting point for the plate grid.

The default fluid density is for water, but any density can be entered. The Fluid Load BLC provides a drop down list of all the Basic Load Cases. If you've defined in any descriptions for your BLC's, these are shown in the drop down list. The surface loads generated are placed in the selected Basic Load Case.

General Truss Generation

The general TrussTemplate generation is used to make various trusses and forms of truss joists. The truss is comprised of nodes and an optional top chord, bottom chord, vertical strut member and diagonal brace member. You can make the truss out of any material type (General, Hot Rolled, Wood, etc.) by choosing your Material.

3D
Truss Template
?
×

Truss
Panels
Truss Options
Section Set & Rotation
Chord Unbraced Lengths

Plane XY ▼

Truss Type Pratt A ▼

Material Hot Rolled Steel ▼

Origin X 0 Y 0 Z 0 ft

A 0 ft

B 1 ft

C 0 ft

H1 0 ft

H2 0 ft

Clear All

Next

Cancel

Truss Template Options

Option	Description
<i>Truss - tab</i>	
Plane	Plane defines the plane in which the truss is drawn.
Truss Type	Truss Type defines the classification or categorization of truss structure.
Material	Material defines the material used to assign the truss member.
Origin	Origin defines the reference point or coordinate system used to establish a fixed position or origin point for the structural model.
<i>Panels - tab</i>	
Left Panels	Left Panels defines the number of panels left region of the truss.
Right Panels	Right Panels defines the number of panels right region of the truss.
<i>Truss Options - tab</i>	
Truss Dimensions	Truss Dimensions defines centerline to centerline or out to out dimensions
Pin-Pin Segments	Pin-Pin Segments defines if segments should be considered pinned (checked) or fixed (unchecked).
No. of Copies	No. of Copies defines the number of copies the truss is generated.
Member Prefix	Member Prefix defines the prefix used in the member label for each new member generated.
Node Prefix	Node Prefix defines the prefix used in the node label for each new node generated.
<i>Section Set & Rotation - tab</i>	
Top Chord	Top Chord defines the section set for the top chord members.
Bot Chord	Bot Chord defines the section set for the bottom chord members.
Verticals	Verticals defines the section set for the vertical members.
Diagonals	Diagonals defines the section set for the diagonal members.
<i>Chord Unbraced Lengths - tab</i>	
Top	Top defines the unbraced lengths for the top chord members.
Bottom	Bottom defines the unbraced lengths for the bottom chord members.

The **Origin** for the truss generation defines the first point of the bottom chord.

You must enter the **truss height** and the **panel lengths** to the left and right of truss "peak".

The **Truss Type** selects what configuration the truss will use. The **Truss Plane** defines the plane of the truss.

Optionally, you can enter a Node label prefix to be used for all the nodes generated as part of the truss. Similarly, you can enter a Member label prefix to be used for all the members generated as part of the truss. This is useful if you want a way to track individual parts of your model by looking at Member or Node prefixes.

You can optionally create members for the bottom chord, top chord, vertical strut and diagonal brace. For each one of these members you must assign a valid independent section set and "x-axis rotate angle". Note that you must check the checkbox for a member type before you can set any of the values to indicate that you want generation performed for that member type.

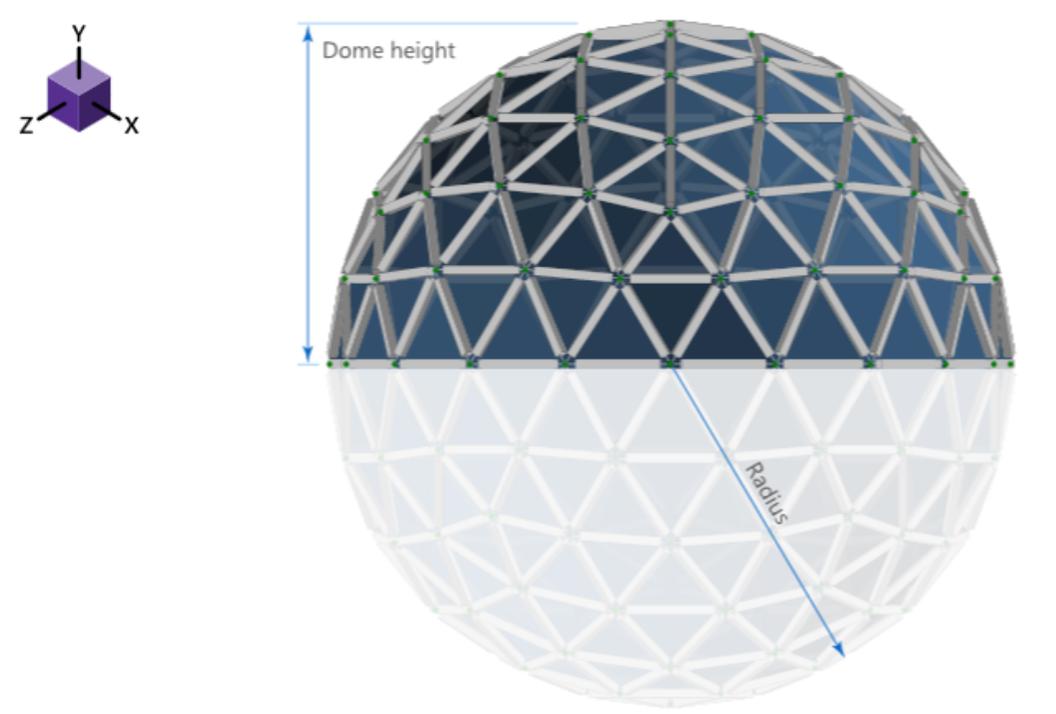
The "x-axis rotate" can be used to rotate the local axes to a desired orientation, however you may find that a K-node better serves this purpose in some instances.

Geodesic Dome Generator

The Geodesic Dome generator is used to construct a geodesic dome comprised of member and plate elements. The following window is displays and allows you to enter the necessary information for the geodesic dome properties.

3D Geodesic Dome Template
? X

Geodesic Dome
Geodesic Dome Options



Radius	<input type="text" value="5"/>	ft	Node Prefix	<input type="text" value="N"/>
Dome Height	<input type="text" value="10"/>	ft	Member Prefix	<input type="text" value="M"/>
Polar Origin	X <input type="text" value="0"/>	Y <input type="text" value="0"/>	Z <input type="text" value="0"/>	ft
			Plate Prefix	<input type="text" value="P"/>

<input checked="" type="checkbox"/> Members Material <input type="text" value="Hot Rolled Steel"/> v Section Set <input type="text" value="Roof Bracing"/> v ...	<input checked="" type="checkbox"/> Plates Material <input type="text" value="gen_Conc3NW"/> v ... Thickness <input type="text" value="1"/> in
---	---

Clear All

Next

Cancel

The following table provides descriptions for the options under both the *Geodesic Dome* and *Geodesic Dome Options* tabs in the Geodesic Dome Template window.

Geodesic Dome Template

Option	Description
<i>Geodesic Dome - tab</i>	
Radius	The Radius can be set in the graphical interface.
Dome Height	The Dome Height can be set in the graphical interface.
Polar Origin	The Polar Origin allows you to choose the dome origin coordinates.
Members	Check the Members checkbox to create strut members. Designate the member prefix, a member material, and a section set.
Plates	Check the Plates checkbox to create exterior plate members. Designate the plate prefix, a general material, and the thickness of the plate elements.
<i>Geodesic Dome Options - tab</i>	
Quantity	The Quantity section designates the number of triangular sub-meshes per side.
Platonic Solid Shape	The Platonic Solid Shape section is used to designate the number of faces of the geodesic dome.

Click **OK** on the Geodesic Dome Options tab to have the program apply this information to create the geodesic dome structure.

Cylinder Generation

The Cylinder Plates / Members generation is used to make cylinders comprised of nodes and optional members and plates.

3D Cylinder with Plates and Stiffeners Template ? X

Cylinder with Plates and Stiffeners Options

Rotation Axis: Z

Axis Increment: 10 - + Length: 1 ft

Sweep Increment: 10 - +

Sweep Angle: 90 °

Start Angle: 0 °

Radius: 5 ft

Polar Origin: X 0 Y 0 Z 0 ft

Material: Hot Rolled Steel Node Prefix: N

Clear All Next Cancel

The following table provides descriptions for the options under both the *Cylinder with Plates and Stiffeners* and *Options* tabs in the Cylinder Template window.

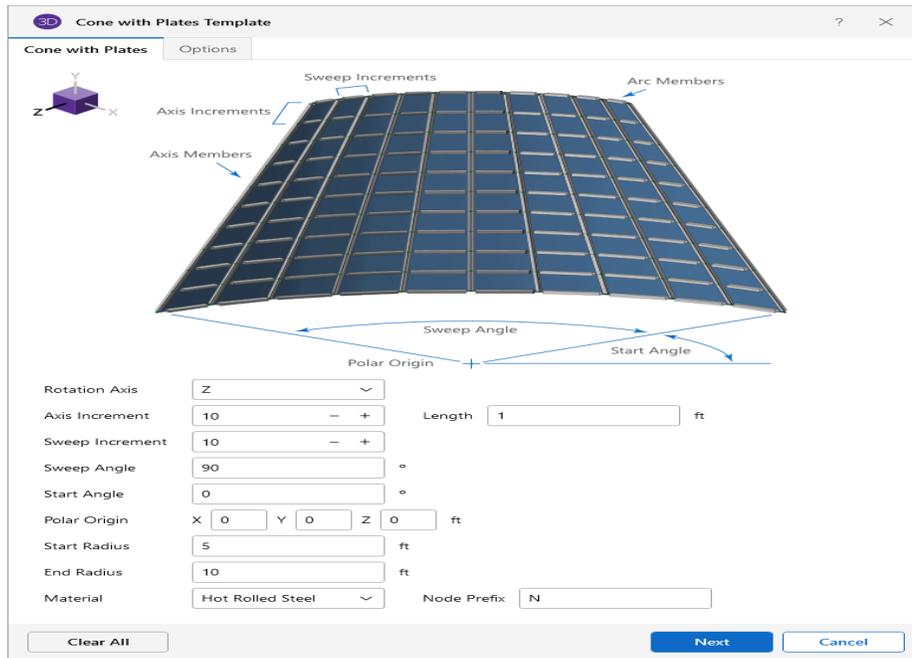
Cylinder Plates / Members Template Options

Option	Description
<i>Cylinder with Plates and Stiffeners - tab</i>	
Rotation Axis	The Rotation Axis defines the axis to which the direction of the cylinder is oriented.
Axis Increment	The Axis Increment defines the number of segments the cylinder is broken down into along the height of the cylinder.
Sweep Increment	The Sweep Increment defines the number of segments the cylinder is broken down into along the width of the cylinder. The number of Sweep Increment determines how many piecewise straight segments are used to model the arc of the cylinder. The number and length of increments along the axis of rotation is used to "extrude" the cylinder in the direction of the axis of rotation.
Sweep Angle	The Sweep Angle defines the ending angle of the cylinder arc.
Start Angle	The Start Angle defines the starting angle of the cylinder arc.
Radius	The Radius is the distance from the polar origin to the furthest point on the cylinder.
Polar Origin	The Polar Origin defines the reference point from which the arc height and width are measured and the rotation axis is defined. The Polar Origin is the point about which the arc rotates and a global axis is the axis of rotation. You can generate a cylinder the full 360 degrees around the axis of rotation or generate a partial cylinder by specifying the start and sweep angles.
Material	The Material option defines the material used to assign properties to the cylinder members.
Node Prefix	The Node Prefix Defines the prefix used in the node label for each new node generated.
<i>Options - tab</i>	
Axis Members	(checkbox) The Axis Member option allows you to toggle members being drawn in the Axis.
Arc Members	(checkbox) The Arc Member options allows you to toggle members being drawn in the arc.
Section Set	Section Set defines the section set assigned axis members or arch members. There are entries for a Section Set for both the arc members and the axis members, so these two sets of members can be different sizes. To generate these members you must select a valid section set. If you don't select a section set, you won't generate these members. Different labels may be assigned to the two member

Option	Description
	types.
X-Axis Rotate	The X-axis Rotate option defines the local x-axis rotation of the arc relative to the global X axis. It can be used to rotate the local axis in to place. However, you may find that a K-node better serves this purpose.
Member Prefix	The Member Prefix defines prefix to member labels for members created using the generator.
Plates	(checkbox) The Plates option allows you to draw plates between members generated using the Grid Generation.
Material Set	You must enter a Material Set for plates to be defined and the label prefix may be specified as well.
Thickness	You must enter Thickness for plates to be defined and the label prefix may be specified as well.
Plate Prefix	The Plate Prefix defines prefix to plate labels for plates created using the generator.
Fluid Load	(checkbox) The Fluid Load option allows you to automatically generate hydrostatic loads to the generated plates.
Depth	The Depth option defines the height of which the hydrostatic loads should be applied onto the structure.
Density	The Density option defines the density of the hydrostatic load that is applied to the structure.
BLC	The BLC option defines the Basic Load Case at which the hydrostatic loads are generated.

Cone Generation

The cone generation is used to make cones comprised of nodes and optional members and plates.



Cone with PlatesTemplate Options

Option	Description
<i>Cone with Plates - tab</i>	
Rotation Axis	The Rotation Axis defines the axis to which the direction of the cone is oriented.
Axis Increment	The Axis Increment defines the number of segments the cone is broken down into along the height of the cone.
Sweep Increment	The Sweep Increment defines the number of segments the cone is broken down into along the width of the cone.
Sweep Angle	The Sweep Angle defines the ending angle of the cone arc.
Start Angle	The Start Angle defines the starting angle of the cone arc.
Polar Origin	The Polar Origin defines the reference point from which the arc height and width are measured and the rotation axis is defined.
Start Radius	The Start Radius defines the starting location of the radius.
End Radius	The End Radius defines the ending location of the radius.
Material	The Material option defines the material used to assign properties to the cone members.
Node Prefix	The Node Prefix Defines the prefix used in the node label for each new node generated.
<i>Options - tab</i>	
Axis Members	(checkbox) The Axis Member option allows you to toggle members being drawn in the Axis.
Arc Members	(checkbox) The Arc Member options allows you to toggle members

Option	Description
	being drawn in the arc.
Section Set	<p>Section Set defines the section set assigned axis members or arch members.</p> <p>There are entries for a Section Set for both the arc members and the axis members, so these two sets of members can be different sizes. To generate these members you must select a valid section set. If you don't select a section set, you won't generate these members. Different labels may be assigned to the two member types.</p>
X-Axis Rotate	The X-axis Rotate option defines the local x-axis rotation of the arc relative to the global X axis. It can be used to rotate the local axis in to place. However, you may find that a K-node better serves this purpose.
Member Prefix	The Member Prefix defines prefix to member labels for members created using the generator.
Segments	(checkbox) When selected, this option allows for the drawing of members into several segments or a single member.
Plates	(checkbox) The Plates option allows you to draw plates between members generated using the Grid Generation.
Material	<p>The Material option defines the material used to assign properties to the cone members.</p> <p>A Material Set for plates must be defined.</p>
Thickness	<p>The Thickness option defines the thickness of the plate.</p> <p>A Thickness for plates must be defined.</p>
Plate Prefix	The Plate Prefix defines prefix to plate labels for plates created using the generator.

Specify the polar origin as the point about which the arc rotates and a global axis as the axis of rotation. You can generate a cone the full 360 degrees around the axis of rotation or generate a partial cone by specifying the start and sweep angles.

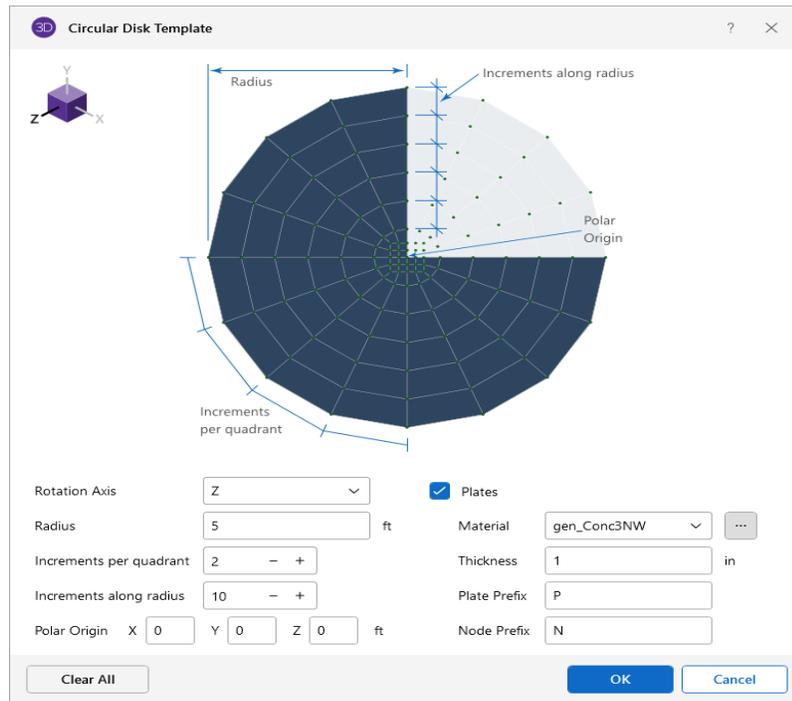
The start radius and end radius are the distances from the axis of rotation to the cone. The number of increments along the sweep determines how many piecewise straight segments are used to model the arc of the cone. The number and length of increments along the axis of rotation is used to "extrude" the cone in the direction of the axis of rotation.

There are entries for a section set for both the arc members and the axis members so these two sets of members can be different sizes. To generate these members you must select a valid section set. If you don't select a section set, you won't generate these members. Different labels may be assigned to the two member types. The "x-axis rotate " may be used to rotate the local axes in to place however, you may find that a K-node better serves this purpose.

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

Circular Disk Generation

The Circular Disk generation lets you generate a circular disk comprised of plate elements.



Circular Disk Template Options

Option	Description
Rotation Axis	The Rotation Axis defines the axis to which the direction of the circular disk is oriented.
Radius	The Radius defines the distance from the polar origin to the furthest point on the circular disk.
Increments per quadrant	The Increment per quadrant defines how many piecewise straight segments are used to model the sweep for each quadrant of the arc. <div style="border: 1px solid black; padding: 5px; background-color: #e6f2ff;"> <p> 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</p> </div>
Increments along radius	The Increments along radius defines the length of increments along the radius.
Polar Origin	The Polar Origin defines the reference point from which the circular disk height and width are measured and the rotation axis is defined.

Option	Description
Plates	(checkbox) The Plates option allows you to draw plates between members generated using the Grid Generation.
Material	The Material option defines the material used to assign properties to the circular disk members.
Thickness	The Thickness option defines the thickness of the circular disk plates. A Thickness for plates must be defined.
Plate Prefix	The Plate Prefix defines prefix to plate labels for plates created using the generator.
Node Prefix	The Node Prefix Defines the prefix used in the node label for each new node generated.

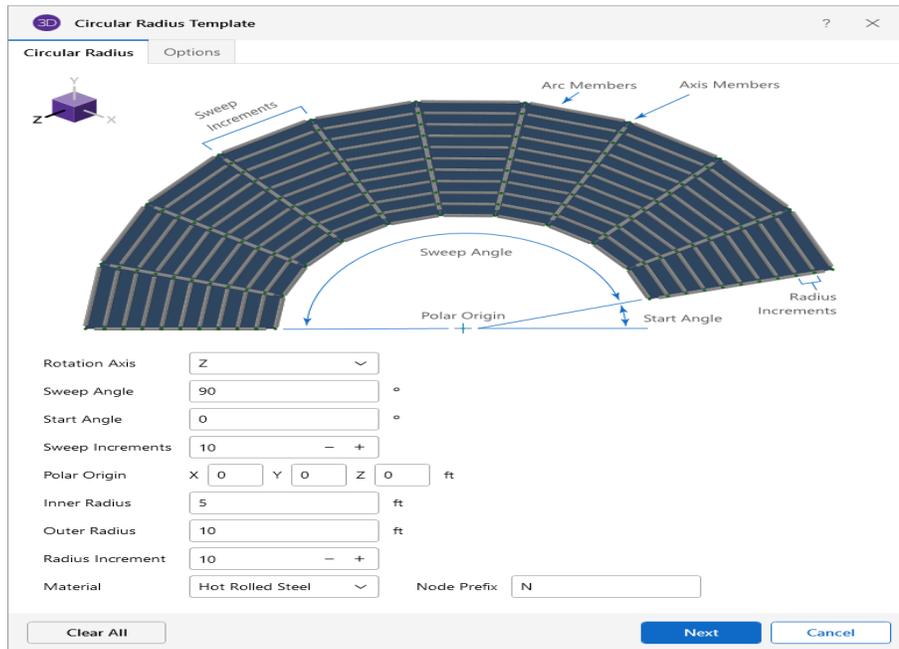
Specify the polar origin as the point about which the grid rotates and a global axis as the axis of rotation. The grid is generated the full 360 degrees around the axis of rotation.

The radius is total length from the polar origin to the outer edge of the grid. The quadrant increment determines how many piecewise straight segments are used to model the sweep for each quadrant of the arc. Note this entry must be a multiple of 2 (2,4,6, etc.) due to modeling constraints at the center of the disk. The radius increment specifies how many “rings” to use to create the grid.

You must enter a material set and thickness to have plates defined. Only quadrilateral plates are generated. You can also specify labels assigned to the plates.

Circular Radius Generation

The Circular Radius generation lets you generate a full or partial circular grid comprised of nodes and optional members and plates.



Circular Radius Template Options

Option	Description
<i>Circular Radius - tab</i>	
Rotation Axis	The Rotation Axis defines the axis to which the direction of the circular radius is oriented.
Sweep Angle	The Sweep Angle defines the ending angle of the circular radius arc.
Start Angle	The Start Angle defines the starting angle of the circular radius arc.
Sweep Increments	The Sweep Increment defines how many piecewise straight segments are used to model the arc.
Polar Origin	<p>The Polar Origin defines the center point of the disk. It is the reference point from which the arc height and width are measured and the rotation axis is defined.</p> <p>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 can generate a grid the full 360 degrees around the axis of rotation or generate a partial grid by specifying the start and sweep angles.</p>
Inner Radius	The Inner Radius defines the radius of the “hole” in the center. This distance must be greater than zero and less than the end radius entry.
Outer Radius	The Outer Radius defines the total length from the polar origin out to the edge of the grid.
Radius Increment	The Radius Increment defines the number of increments along the radius tells RISA-3D how many “rings” to use to create the grid.

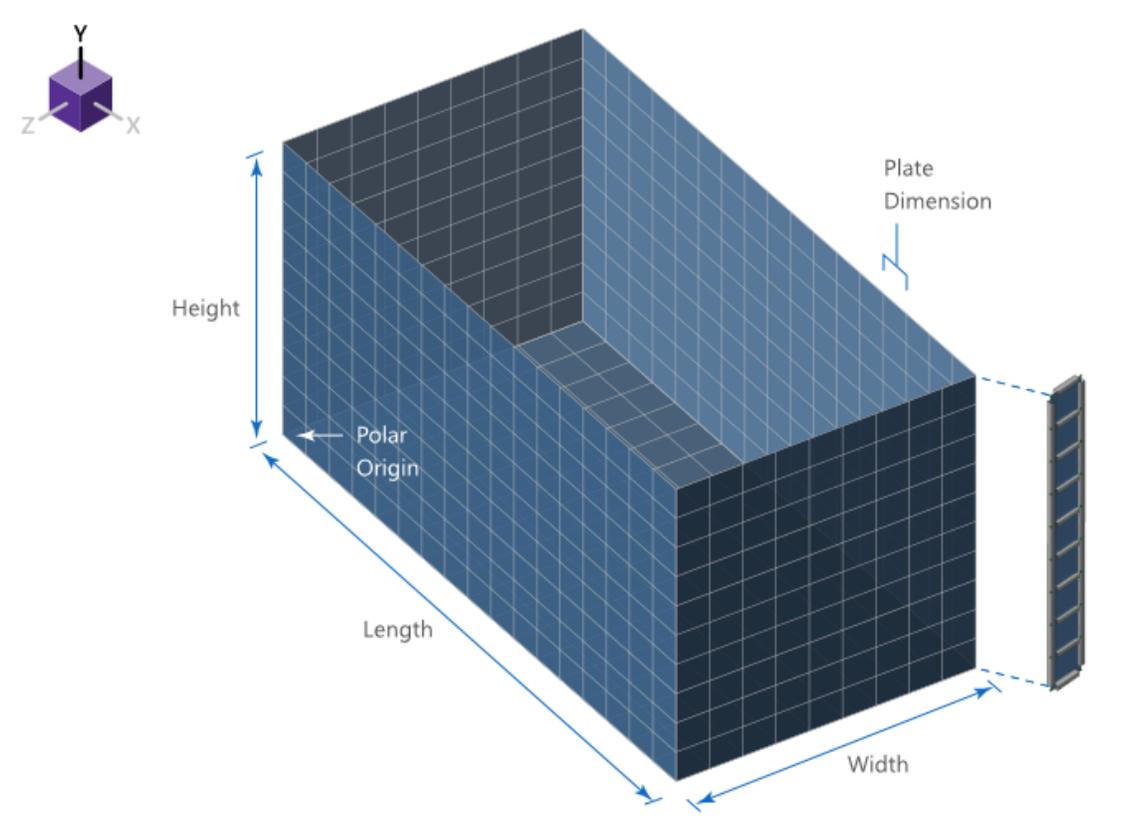
Option	Description
Material	The Material option defines the material used to assign properties to the circular radius members.
Node Prefix	The Node Prefix Defines the prefix used in the node label for each new node generated.
<i>Options - tab</i>	
Axis Members	(checkbox) The Axis Member option allows you to toggle members being drawn in the Axis.
Arc Members	(checkbox) The Arc Member options allows you to toggle members being drawn in the arc.
Section Set	<p>Section Set defines the section set assigned axis members or arch members.</p> <p>There are entries for a section set for both the arc members and the radius members so these two sets of members can be different sizes. To generate these members you must select a valid section set. If you don't select a section set, you won't generate these members. You may also have unique labels assigned to the generated members by entering a start label.</p>
X-Axis Rotate	The X-axis Rotate option defines the local x-axis rotation of the arc relative to the global X axis. It can be used to rotate the local axis in to place. However, you may find that a K-node better serves this purpose.
Member Prefix	The Member Prefix defines prefix to member labels for members created using the generator.
Segments	(checkbox) When selected, this option allows for the drawing of members into several segments or a single member.
Plates	<p>(checkbox) The Plates option allows you to draw plates between members generated using the Grid Generation.</p> <p>You enter a material set and thickness to have plates defined. Only quadrilateral plates are generated. You can also have unique labels assigned to the generated plates by entering a start label.</p>
Material	<p>The Material option defines the material used to assign properties to the cone members.</p> <p>A Material Set for plates must be defined.</p>
Thickness	<p>The Thickness option defines the thickness of the plate.</p> <p>A Thickness for plates must be defined.</p>
Plate Prefix	The Plate Prefix defines prefix to plate labels for plates created using the generator.

Rectangular Tank Generation

The Rectangular Tank generation is used to make rectangular tanks out of plate finite elements. Stiffeners and hydrostatic loads can also be modeled. The tank will be pin supported around the bottom perimeter. No supports are generated in the interior of the tank floor.

3D Rectangular Tank with Stiffeners Template ? X

Rectangular Tank with Stiffeners Options



Y
Z X

Height

Polar Origin

Length

Width

Plate Dimension

Rotation Axis: Y

Polar Origin: X 0 Y 0 Z 0 ft

Node Prefix: N

Member Prefix: M

Plate Prefix: P

Tank Height: 0 ft

Tank Width: 0 ft

Tank Length: 0 ft

Plate Max. Dimension: 1 ft

Clear All Next Cancel

Rectangular Tank Template Options

Option	Description
<i>Rectangular Tank with Stiffeners - tab</i>	
Rotation Axis	The Rotation Axis defines the axis to which the direction of the tank is oriented.
Polar Origin	The Polar Origin defines the reference point from which the tank height and width are measured and the rotation axis is defined.
Node Prefix	The Node Prefix Defines the prefix to node labels for members created using the generator.
Member Prefix	The Member Prefix defines prefix to member labels for members created using the generator.
Plate Prefix	The Plate Prefix defines prefix to plate labels for plates created using the generator.
Tank Height	The Tank Height defines the height of the tank.
Tank Width	The Tank Width defines the width of the tank.
Tank Length	The Tank Length defines the length of the tank.
Plate Max. Dimension	The Plate Max. Dimension defines the maximum plate thickness used to generate the tank.
<i>Options - tab</i>	
Plates	(checkbox) When selected, this option draws plates between members generated using the Grid Generation.
Material	The Material option defines the material used to assign properties to the plate members.
Thickness	The Thickness option defines the thickness of the plate.
Fluid Load	(checkbox) When selected, this option automatically generates hydrostatic loads to the generated plates.
Depth	Depth defines the height of which the hydrostatic loads should be applied onto the structure.
Density	Density defines the density of the hydrostatic load that is applied to the structure.
BLC	BLC defines the Basic Load Case at which the hydrostatic loads will be generated.
Top Stiffeners	(checkbox) When selected, this option will add stiffeners to the top of the tank.
Bottom Stiffeners	(checkbox) When selected, this option will add stiffeners to the bottom of the tank.
Intermediate Stiffeners	(checkbox) When selected, this option will add stiffeners to the mid-height of the tank.
Member Material	Member Material defines the material used to assign axis members or arch members.
Section Set	Section Set defines the section set assigned axis members or arch members.

Option	Description
Segments	(checkbox) When selected, this option draws members into several segments or a single member.

Specify the origin for the tank generation. This defines the bottom corner of tank floor. The tank height is always towards the positive vertical axis. The width and length is 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.

A plate element edge length can be specified. The program generates the tank out of plates which are approximately square, with an edge dimension approximately (+/- 20%) equal to this value. Regardless of this dimension there is always at least 5 plates along each side of the tank, and there is always at least three plates (vertically) between each stiffener.

Horizontal stiffeners can 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 can offset the stiffeners with rigid links in order to model the composite action of the stiffener and the tank wall due to the center of the stiffener being offset from the tank wall. When this option is not used, the stiffener and plate centerlines are at the same location and share the same joints. If this option is used, the generator automatically creates a material set for the rigid links called "TANK_RIGID_RISAMAT", and a section called "TANK_RIGID_RISASEC". The properties for these are preset and should only be modified if the rigid links are acting "flexible" relative to the rest of your model. Each time the generator is run, the properties are reset to the default values if these material/section sets have already been created.



Note: You cannot use this option for Arbitrary shapes, Tapered WF shapes, and shapes that are defined on the section screen by typing in their area, moment of inertia's and torsional stiffness. These shapes can still be used as stiffeners; you just can't have the composite offset automatically calculated and modeled.

A node label prefix can be entered, which is used for all joints generated for the tank. A valid material set must be selected in order for plates to be generated. A plate label prefix can be entered, which is used for all the plates generated for the tank.

A valid section set must be selected in order for stiffeners to be generated. A stiffener label prefix can be entered, which is used for all the members generated for the tank stiffeners. Any rigid links generated to model the offsets have a prefix of "RIGIDTANK"

A hydrostatic load can also be generated. This load is generated using a series of uniform surface loads on the plate elements. The fluid depth is measured in the positive vertical direction from the bottom of the tank and is constant along the floor giving a uniform surface load on the floor plates. The fluid depth will vary along the walls of the tank and these will increase in a "stair step" fashion, as you move down the fluid depth. The value of each uniform surface load is equal to the value of the

actual hydrostatic load at the mid-point of the plate. The load direction is perpendicular to the plates and outward.

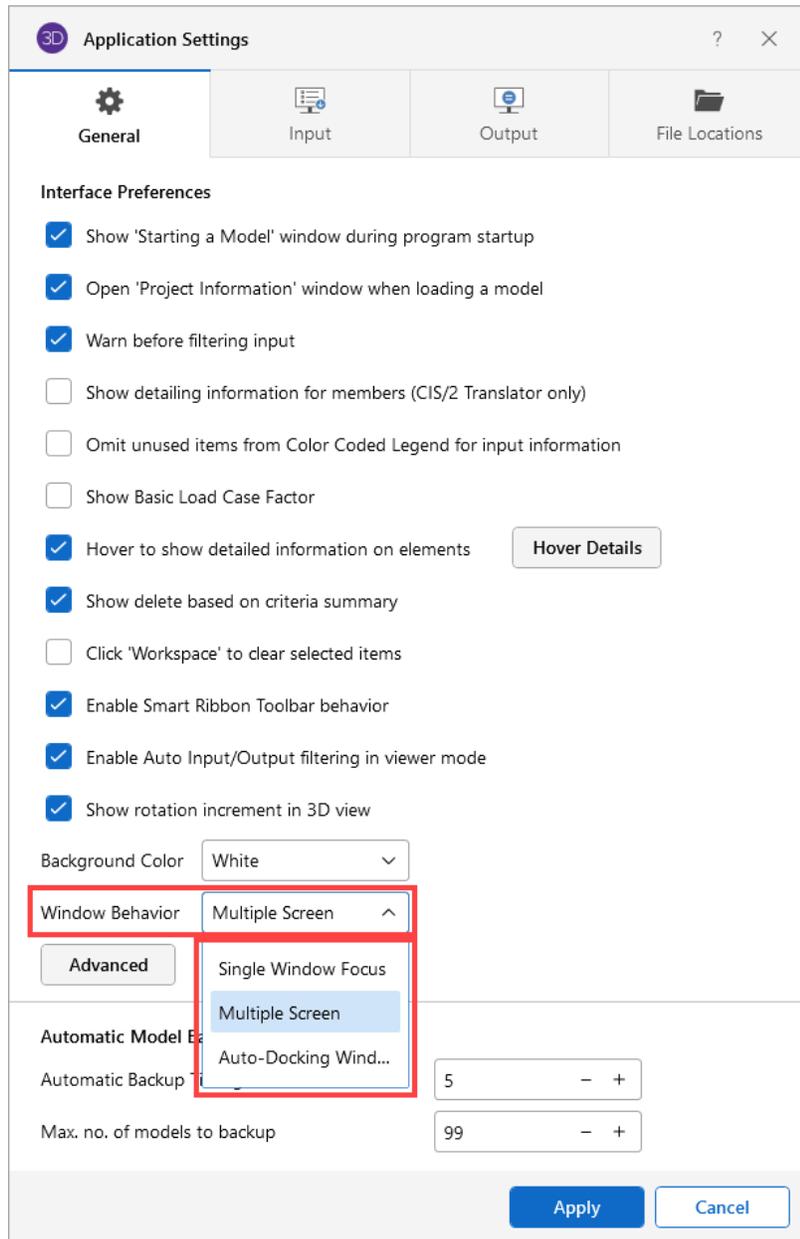
The default fluid density is for water, but any density can be entered. The Fluid Load BLC provides a drop down list of all the Basic Load Cases. If you've defined in any descriptions for your BLC's, these are shown in the drop down list. The surface loads generated are placed in the selected Basic Load Case.

Graphic Display

RISA-3D's robust graphics make building and understanding your model much easier. You can have numerous independent views open and active at the same time. Fast redraw times, responsive panning, true to scale rendering, animation capabilities, and graphic editing tools all help you build and understand your model. You can draw the model and specify loads and boundary conditions graphically. These items can be modified graphically as well. Verification of your model is made simple with true-to-scale rendering and color coded code checks. Results such as member force diagrams, color-coded stress levels, deflected shapes and animations can also be viewed for solved models.

Window Behavior

Window Behavior can be specified in the 'Application Settings' window, under the 'Window' section. The window section allows you to specify the active windows in the program.



There are three window behaviors to choose from:

- **Single Window Focus:** With this option, when you click within the 3D view, it automatically closes all open/visible spreadsheets and detail reports.
- **Auto-Docking Windows:** With this option, when you click within the 3D view, it automatically docks all open/visible spreadsheets and detail reports as new tabs.
- **Multiple Screen:** This option lets you use two monitors. It is useful when you want to keep detail reports or spreadsheets visible on a separate monitor while navigating through the

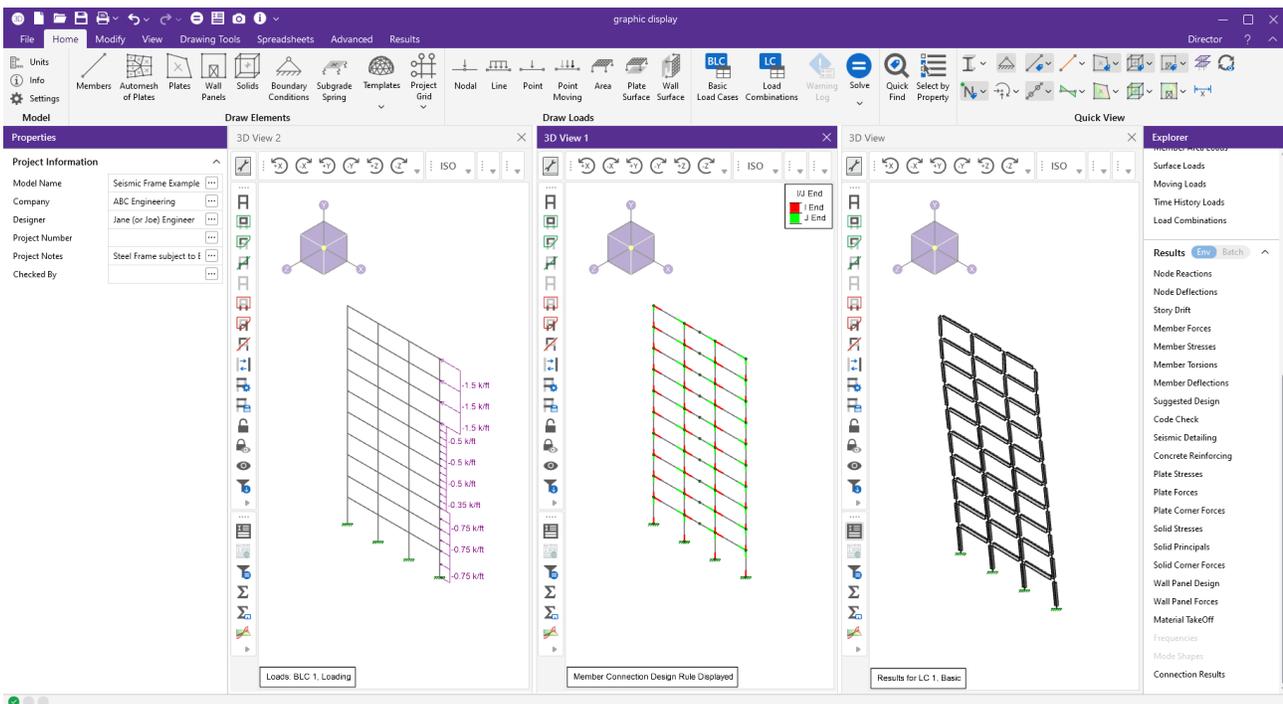
3D view. If a detail report or spreadsheet is open and is visible on the same monitor as the active 3D view, clicking within the 3D view sends the spreadsheet behind the program window.

For additional information on working with viewpoint windows, see the [Windows Behavior](#) section under Main UI.

For information about working with multiple views, see the [Multiple Windows](#) section under Graphic Display, Window Behavior.

Multiple Windows

You can have multiple views of a model open at the same time, each with its own characteristics. For example you can simultaneously view and work with members in one window and plates in another. Multiple spreadsheets can be opened as well.

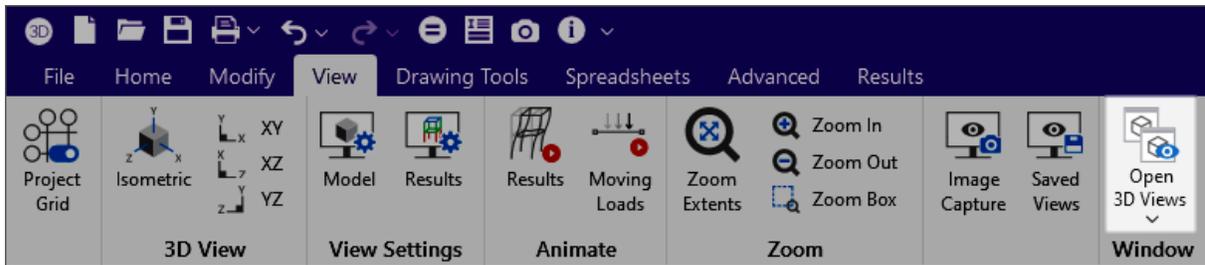


Open a New 3D View

Whether you want to open a new 3D View panel or open an additional 3D View panel, the steps are the same.

To open a new 3D View:

1. Go to the **View** ribbon.



2. Click on Open 3D Views (as shown in the previous image).

Alternatively, you can right-click in the existing 3D View and select **Add New 3D View** from the menu that appears.

A new 3D View window opens with displaying a duplicate of the model, if you already have a model open.

3. (Optional) To adjust the view of the new model window:

With the model view active, use the snap buttons (shown below) at the top of the 3D View to adjust the view of the model.

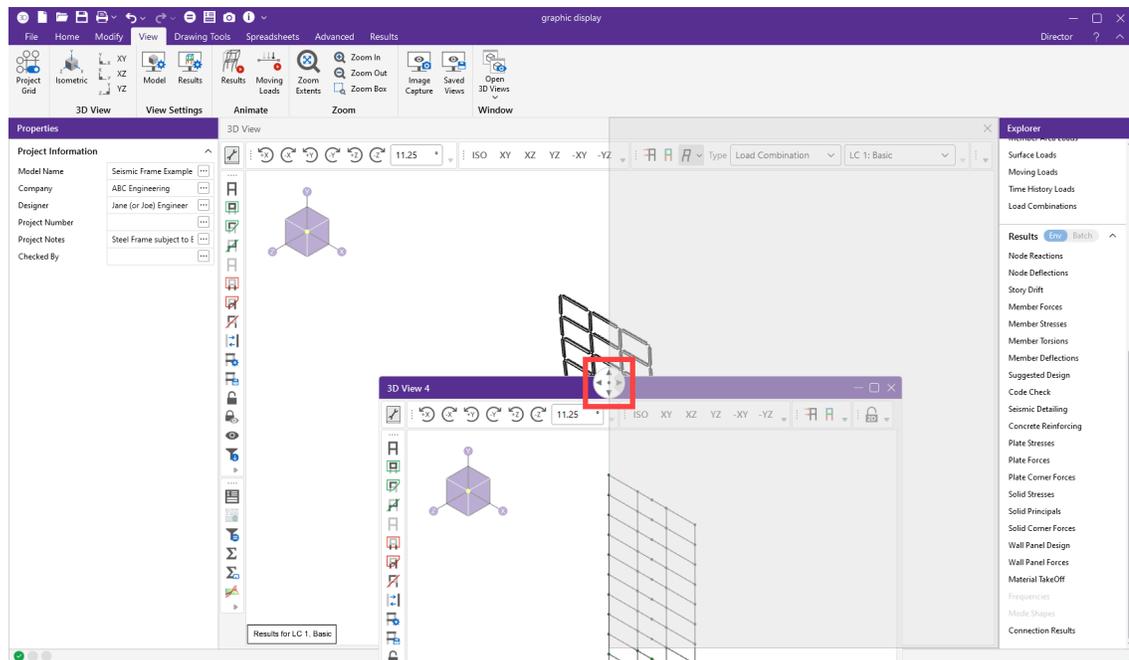


4. (Optional) To snap the 3D View panel within the model editor panel:

- a. Drag its purple title bar of any window (3D View or spreadsheet) until you see the  arrow controls appear.
- b. While still holding down the mouse button, drag over the arrow control that corresponds with where you want the view panel to be snapped.

A gray areas appears showing where the panel will be placed when you release the

mouse button.

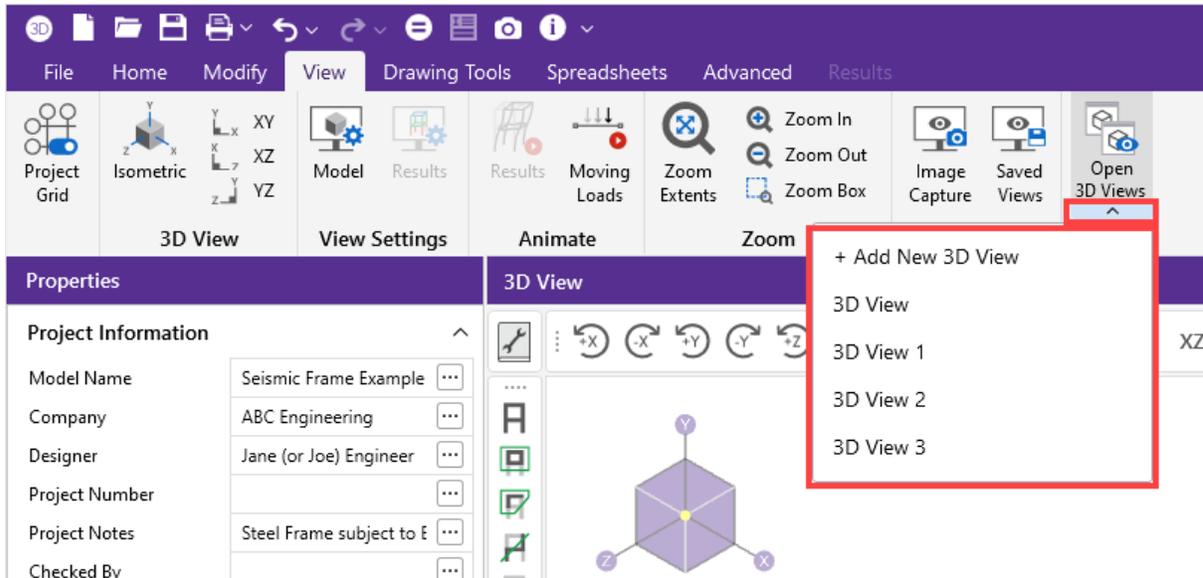


Switch Between Open 3D Views

When you have two or more 3D Views open in the view panel, you can easily switch between those views by choosing the view you want to work with from a menu.

To switch to another view to work with:

1. Open the **View** ribbon.
2. Click the **Open 3D Views** down-arrow.



A menu appears that lists each of the views that are currently open.

3. Click on the view you want to work with.

The view you chose is brought to the front of the view panel and becomes the active view to work with it.

Adjust the Viewpoint (3D View)

Adjusting the Viewpoint refers to what you can do within a 3D View panel. Following are explanations of each feature and how to use it.

Zoom Features

There are two methods for zooming; you can simply use your mouse or use the Zoom tools on the View ribbon.

Use Your Mouse to Zoom

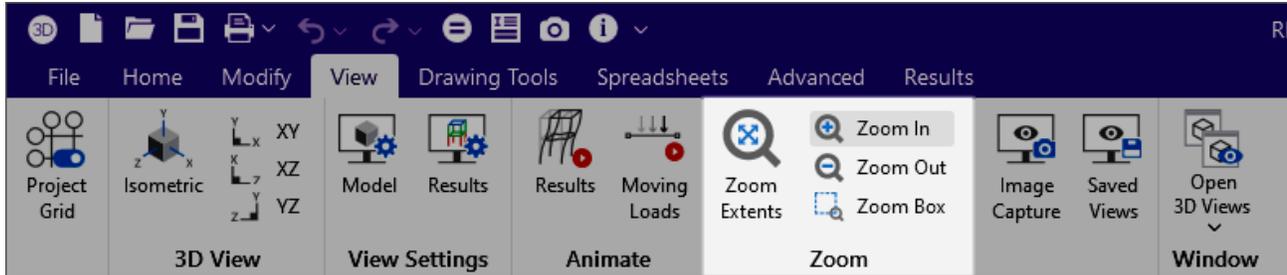
When you want to zoom in on the model to see more detail or zoom out to view the whole model at once, it is easier to use your mouse wheel.

To zoom in or out using your mouse wheel:

- To zoom **IN** rotate the wheel forward.
- To zoom **OUT** rotate the wheel backward.

Use the Zoom Tools

The Zoom tools found on the View ribbon, can also be used for zooming in or out. First, you choose the zoom state (In or Out) using the appropriate icon in the ribbon, then you use your mouse to incrementally zoom. There are two additional Zoom tools: '[Zoom Extent](#)' and '[Zoom Box](#)'. These are explained below.



Zoom Extent - tool

This tool resets the view to Full Model View. It is especially useful for resetting the model view after you've zoomed into a specific area, though it also resets from a zoomed out view.

To zoom to full model view:

- Click the **Zoom Extents** icon.

Zoom In - tool

This tool sets the Zoom state to "In" so that each time you click your mouse button, the model view is enlarged.

1. Click the  **Zoom In** icon on the View ribbon.
2. Click anywhere in the model (3D View panel) to enlarge the model view.
3. Repeat **step 2** until the model view is enlarged enough.

Zoom Out - tool

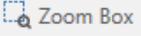
This tool sets the Zoom state to "Out" so that each time you click your mouse button, the model view is reduced.

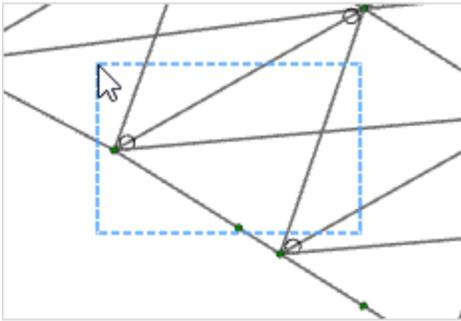
1. Click the  **Zoom Out** icon on the View ribbon.
2. Click anywhere in the model (3D View panel) to reduce the model view.
3. Repeat **step 2** until the model view is reduced enough.

Zoom Box - tool

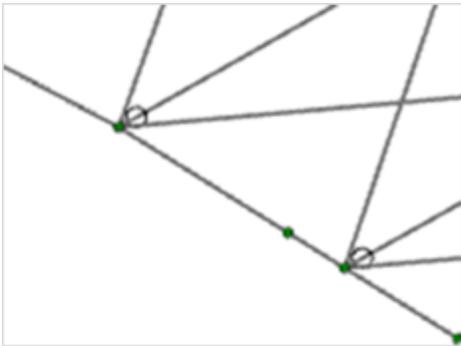
This tool lets you select a specific area to zoom into by drawing a box around the area.

To zoom into a specific area of the model:

1. Click the  **Zoom Box** icon on the View ribbon.
2. In the model view, click on one point and hold your mouse button down as you drag down to the opposite corner, creating a box around the area you want to zoom into.



3. Lift up the mouse button.
The area is selected and the view automatically zooms in on it.



Pan the View

The Pan feature lets you drag the model view to the edges of the 3D view panel. You can accomplish this by using one of two methods: your mouse scroll wheel or the right-click menu.

Pan Using Your Mouse Wheel

To pan the model view using your mouse wheel:

1. Click and hold the mouse wheel button to trigger the dynamic pan.
You'll see a panning icon  appear, indicating that you are in panning mode and can begin dragging. You can drag the model view in any direction.

2. While still holding down the mouse wheel, drag the model view in the direction where you want to move it.
3. Release the mouse wheel when the model view is placed where you want it.

Pan Using the Right-Click Menu

To pan the model view using the right-click menu:

1. Right-click anywhere in the model view panel.
2. Choose **Pan** from the menu that appears.
You'll see the panning icon  appear, indicating that you are in panning mode and can begin dragging.
3. Click and hold your left mouse button and drag the model view in the direction where you want to move it.
4. Release the mouse button when the model view is placed where you want it.
5. To turn off panning mode, click the right-mouse button.

Snap to a View

To snap to an isometric or planar view of the model:

1. Click the **Snap** button at the top of the 3D View window that corresponds to what you want.



Note that you can add your own custom snap view to the toolbar. When doing so, the current view is saved with the name you provide and added to the end of the toolbar.

Alternatively, you can right-click in the existing 3D View and select Add New 3D View.

Rotate a View

There are two methods for rotating a view; click or drag. An additional feature lets you quickly [reset](#) the model's rotation to its original, default position.

As you rotate a view, the global X-Y-Z axis orientation is shown by the orientation of the 3D box.

Click to Rotate

To use the click method to rotate a view:

1. Click the rotate icon that corresponds to the rotation degree you want to use.



You can also change the rotation angle by entering a different degree magnitude. This affects the rotation of the model as you click to rotate.

2. (Optional) To change the rotation angle:

- a. Type the degree magnitude in the box, as shown in the following image.



- b. Click the rotate icon that corresponds to the rotation angle you want the model rotated.

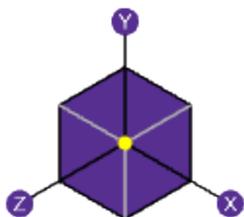
The model is set to the degree you've entered and the orientation of the 3D box changes to reflect the angle and degree.

Drag to Rotate

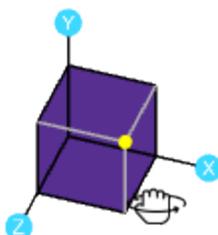
You can use one of the following drag methods to rotate the model view:

- While holding down the **SHIFT** key, press the mouse wheel to dynamically rotate the structure. See [Dynamic View Controls](#) for more information.
- Click and drag on the **3D box** to change the rotation of the model.

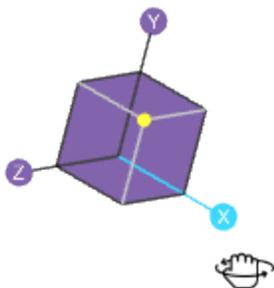
Default Rotation



Click and Drag



- Use a **global axis** to rotate the view by hovering over the axis until it turns blue, then clicking and dragging to change the rotation of the model.

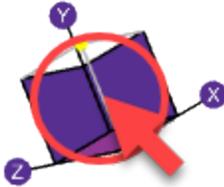


Note: If you hold down the **SHIFT** key while rotating the model using the individual axis, the model only rotates at 45 degree increments around that axis.

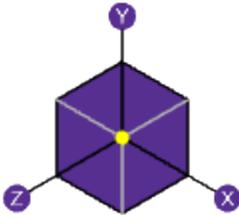
Reset Rotation to Default

To reset the rotation of the model view to its original, default position:

- Double-click in the **3D box**.

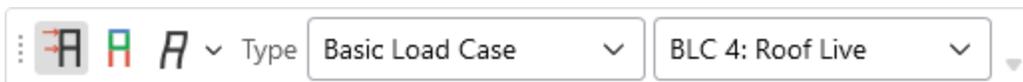


The view is reset to the original, default position and the 3D box is returned to its original default as well, shown in the following image.



3D View Display Options

Each 3D View has several display options located at the top of the view; Show Loads, Show Results and Show Deflection.



Show Loads

The 'Show Loads' icon lets you display loads in the model view.

To show loads in the model view panel:

1. Click the **Toggle Display of Loads** icon to display loads.
2. Click the **Type** down arrow and choose whether you want to display loads based on Basic Load Case, Load Category or Load Combination.
3. Click the **▼** down arrow in the second box and choose which load case, category or combination is shown in the view. Note that this box is only active when you choose a Load; it is not visible when you choose Envelope.

See [Model View Settings](#) for more control over the display of loads.

Show Results

The 'Show Results' icon lets display color coded results in your model view.

To show results:

1. Click the **Results** icon.
2. Click the **Type** down arrow and choose whether you want to display color coded results based on Basic Load Case, Load Category or Load Combination.
3. Click the \vee down arrow in the second box and choose which load case, category or combination results are to be colored coded in the model view. Note that this box is only active when you choose a Load; it is not visible when you choose Envelope.

A color legend appears and color coded results based on your choices, appear in the model.

See [Results View Settings](#) for more control over the display of results.



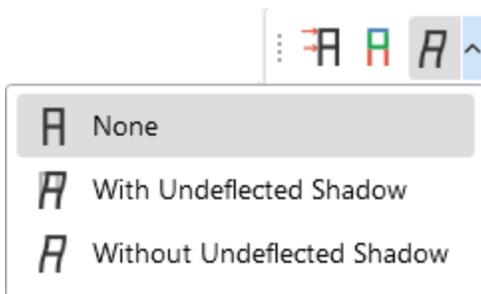
Note: Deflection and results can only be shown for Load Combination load types.

Show Deflect

The 'Show Deflect' icon lets you display the deflected shape in your model view; with or without an undeflected shadow.

To show the deflected shape:

1. Click the **Deflect** icon.
2. (Optional) Click the Deflect \vee down arrow to choose whether to view the deflected shape with or without an undeflected shadow (as shown in the following image).



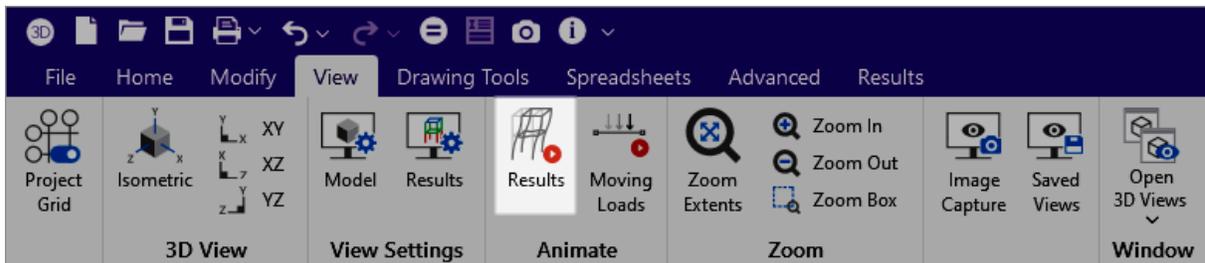
The model's deflected shape is displayed.

Animate Results

With the Animate Results tool, you can animate a model's deflected shape, mode shapes or time history loads.

To animate results:

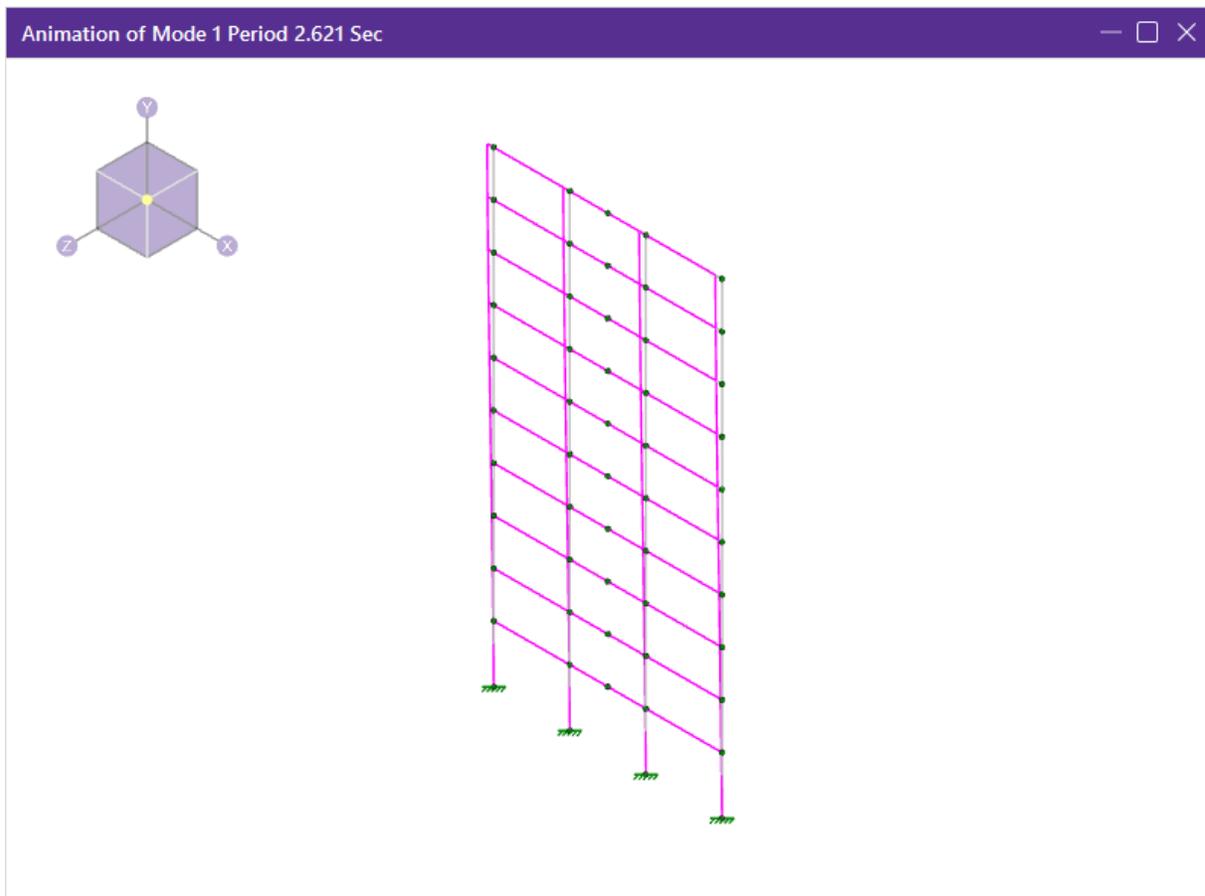
1. Open the **View** ribbon.



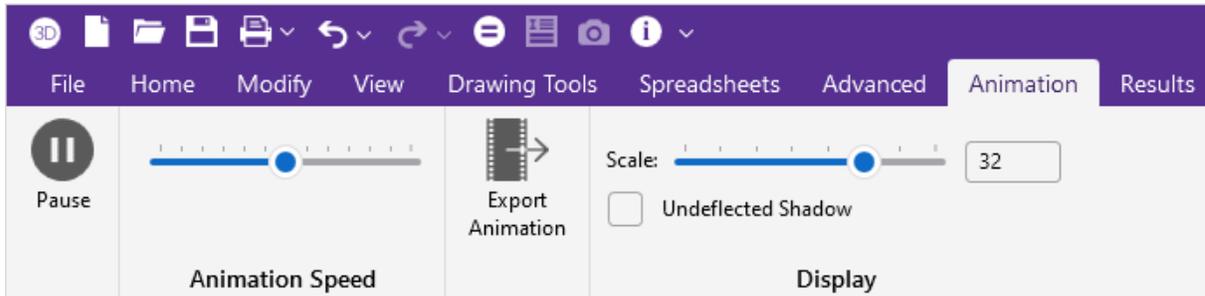
2. Click the **Results** icon in the Animate section.

This action animates deflection, mode shape (dynamic analysis) or time history.

A new model view is created with the animation.



When this model view is active, an *Animation* ribbon opens. The Animation ribbon provides additional settings depending on the type of animation selected. Additional controls are available for the animation of Time History motion, see [Animate Time History Deflection](#).



3. (Optional) Click the **Pause** icon to pause the animation.
4. (Optional) Use the **Animation Speed** slider to control the speed of the animation.
Dragging the marker to the right increases the speed, while dragging it to the left decreases it.
5. (Optional) Click the **Export Animation** icon to export the animation as a .wmv or .mp4 file.
6. (Optional) Use the **Scale** slider to control the scale of the animated deflection.
7. (Optional) Check or Uncheck the **Undeflected Shadow** check box to toggle it on or off.

Create a Custom Snapshot of a Model

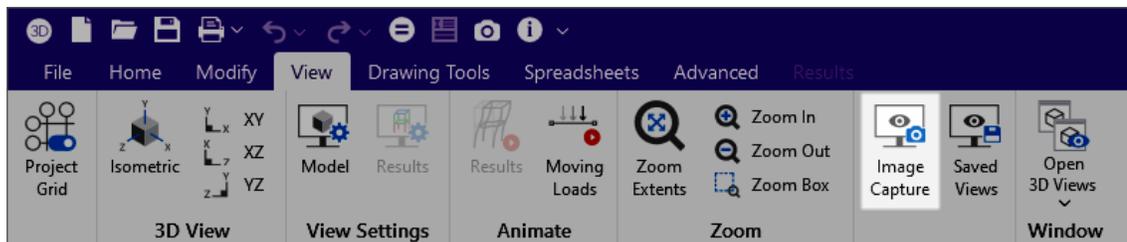
RISA-3D includes an 'Image Capture' tool on the View ribbon that allows you to create a custom snapshot by capturing a snap shot of a model and saving it. If you have a view that you like to work with or have created a view that took some time to set up, save it with this feature. All of the Model View Settings are saved with the model for later recall.

To create a custom snapshot:

1. Click the  **Snapshot** icon on the **Quick Access Toolbar**.

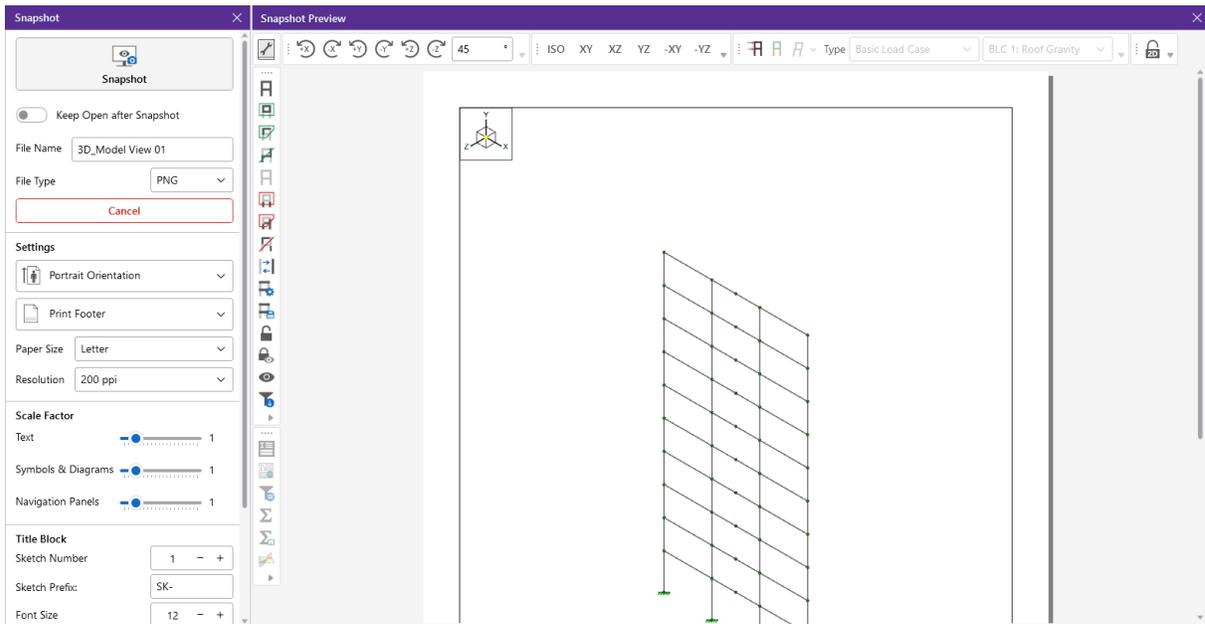
Alternatively, you can:

- a. Open the **View** ribbon.

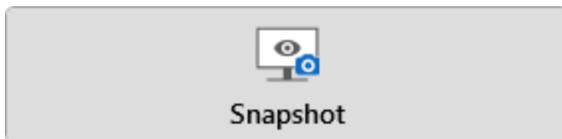


- b. Click the **Image Capture** icon.

A Snapshot Preview panel opens on the right, with a Snapshot panel to its left.



2. (Optional) To print the snapshot, please see [Printing](#) for more information.
3. To save the snapshot:
 - a. In the **File Name** box in the Snapshot panel, type a name for the snapshot you are saving.
 - b. Click the **File Type** down-arrow and choose the image format to save the snapshot in.
 - c. (Optional) Click the **Keep Open After Snapshot** slider (it turns blue when selected), to keep the Snapshot and Snapshot Preview panels open after capturing the snapshot.
4. Click the **Snapshot** block in the Snapshot panel (see following image).



A Snapshot Created message briefly appears.



Note:

- If the model has yet to be solved, the image is created in the **Model Data Files** location from **File > Application Settings > File Locations**.
- The relative path of the image file is saved with the model. If the model file is moved, remember to move the associated **Images** folder as well, so that all saved views and snapshots will still be available.

For information on viewing saved snapshots, see [Retrieve a Saved View](#) under [Save / Retrieve a Model / Snapshot View](#).

View Only Part of a Model

You may want to view only part of your model, especially if it is a big one. You may use the selection tools to give you the view that you want. You can graphically unselect parts of the model that you don't wish to see, or you can use a range of coordinates or other criteria to specify what to view.

- To select a part of the model to zoom in on, see [Zoom Box](#) under [Adjust the Viewpoint \(3D View\)](#).
- To select a part of the model to work more closely on, see [Selection Modes](#) under [Graphic Selection](#).

Save / Retrieve a Model / Snapshot View

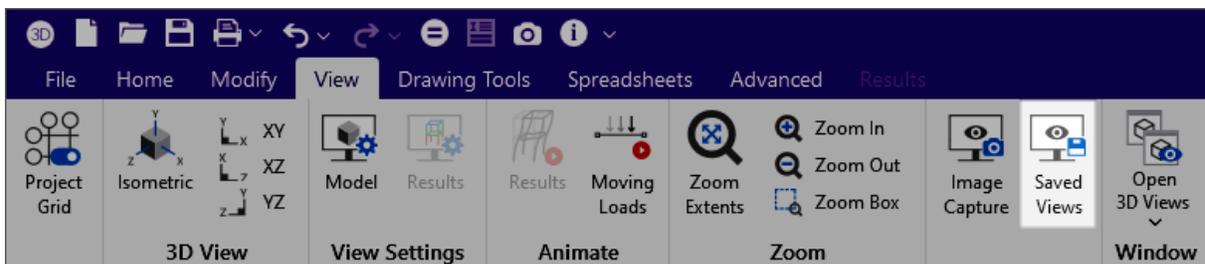
You can save and recall views for a model, and recall custom snapshot views as well. Saved views are model dependent, so any views you save stay with the model. A saved view/snapshot includes information such as the current view angle, zoom state, pan location, plot option settings, etc. So if you have a view that you like to work with or have created a view that took some time to set up, save it with this feature or the snapshot feature. All of the Model View Settings are saved with the model for later recall. Saved views do NOT include the selection state for the model. You can save selection states separately (see [Save Selections](#) for more information). See [Create a Custom Snapshot of a Model](#) for more information on creating a snapshot.

Save a Model View

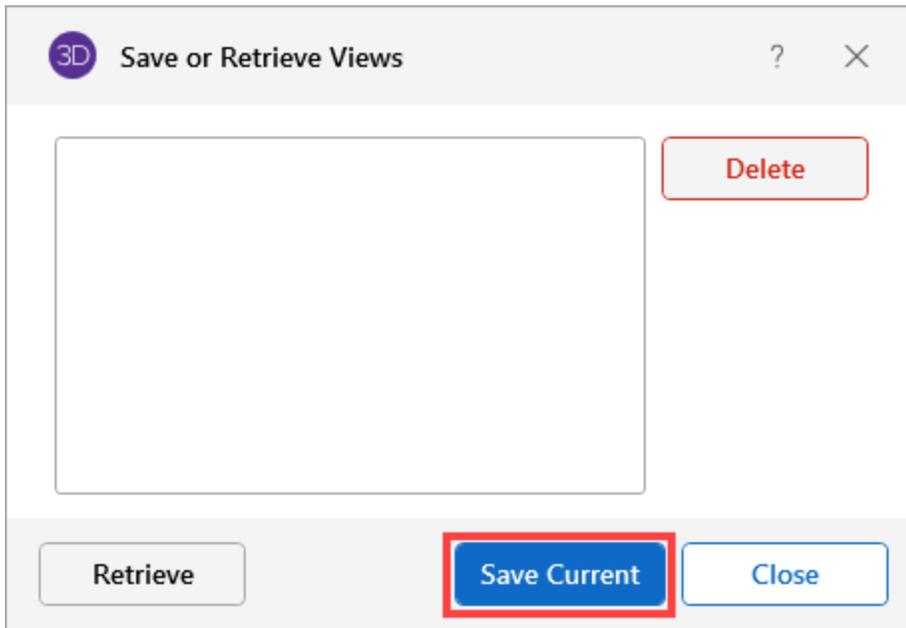
If you have a view that you like to work with or have created a view that took some time to set up, save it with this feature. All of the Model View Settings are saved with the model for later recall.

To save a model view:

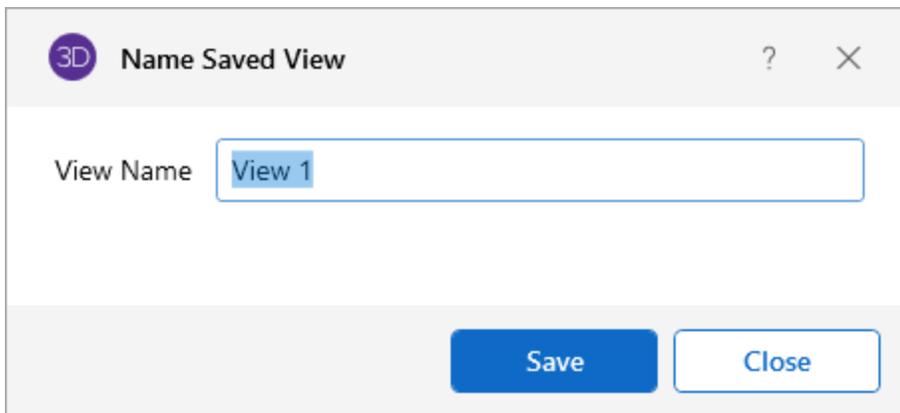
1. Open the **View** ribbon.



2. Click the **Saved Views** icon.
A Save or Retrieve Views window opens.

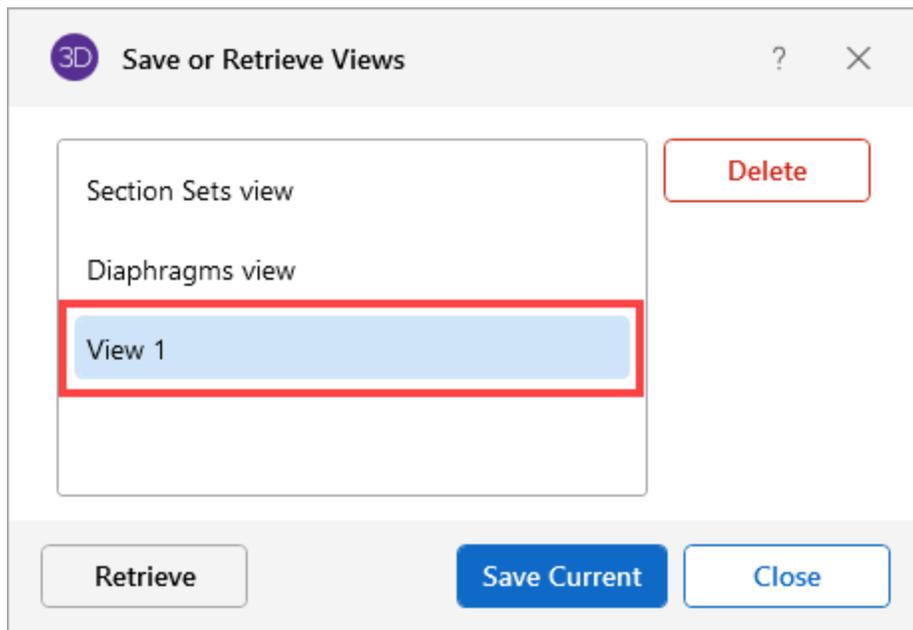


3. Click the **Save Current** button.
A Name Saved View window opens.



4. In the **View Name** box, type a name to the view.
5. Click the **Save** button.

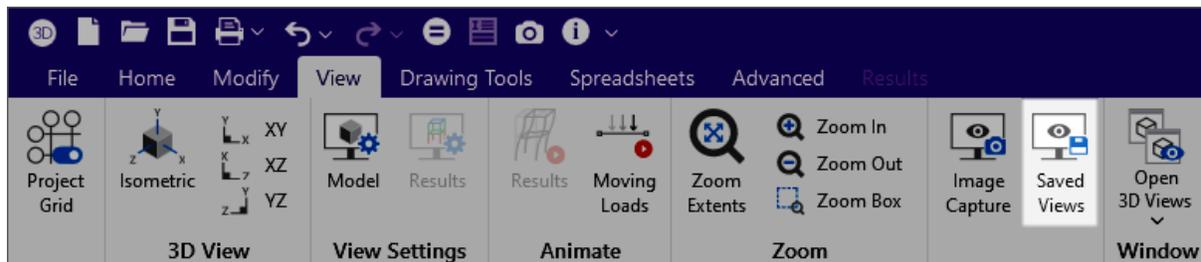
The view is saved and the Name Saved View window is closed. You are returned to the Save or Retrieve Views window where you can see the view you just saved, along with all other saved views.



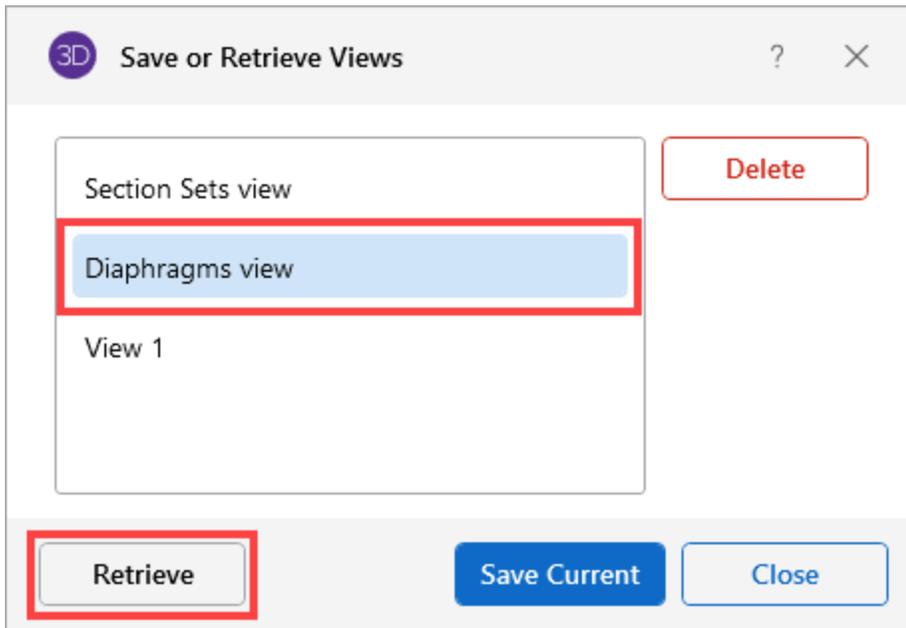
Retrieve a Saved View

To retrieve a saved model or snapshot view:

1. Open the **View** ribbon.



2. Click the **Saved Views** icon.
The Save or Retrieve Views window opens.



3. From the list, choose the view you wish to recall.
4. Click the **Retrieve** button.

The view you chose is brought forward and becomes the active view to work with.

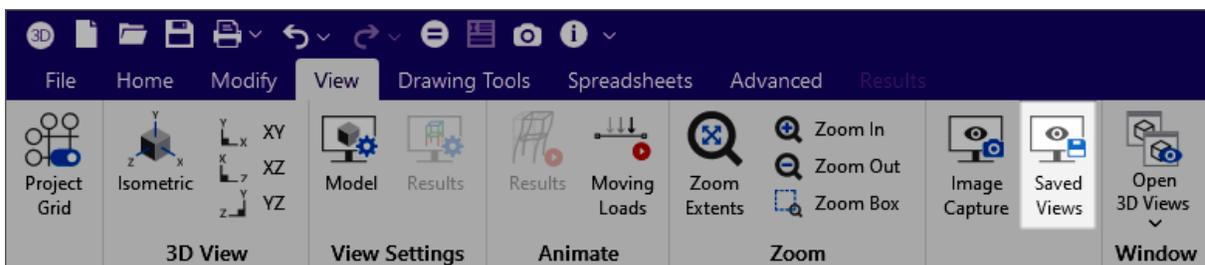
For information on deleting a view, see [Delete a Model View](#).

Delete a Model View

If you no longer need a view or snapshot view that you have saved, you can simply delete it.

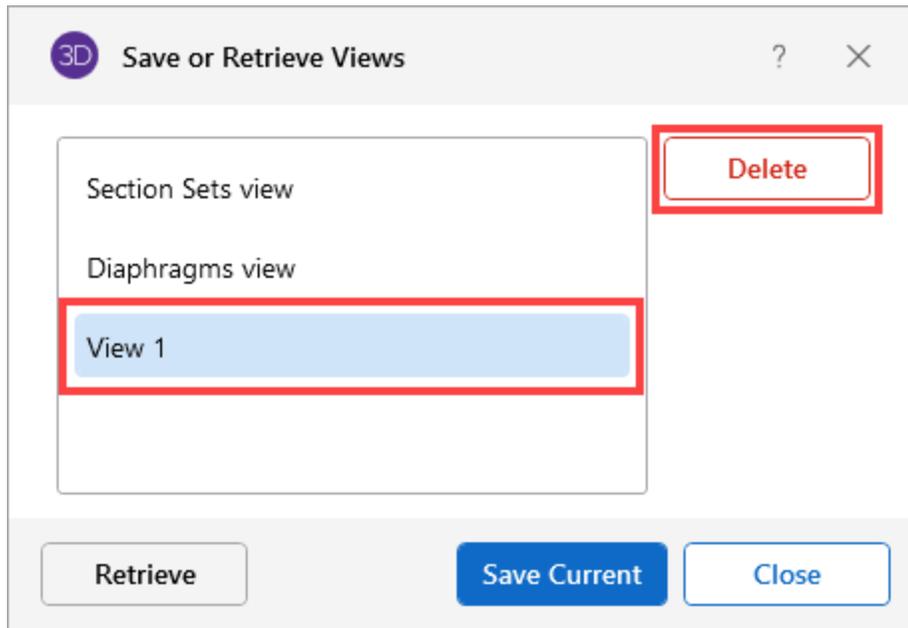
To delete a saved model view or snapshot view:

1. Open the **View** ribbon.



2. Click the **Saved Views** icon.

The Save or Retrieve Views window opens.



3. From the list, choose the view you wish to delete.
4. Click the **Delete** button.
You are asked to confirm the deletion.
5. Click **Yes**.
The view you chose is removed from the saved list.

For information on saving a view or retrieving a saved one, see [Save / Retrieve a Model / Snapshot View](#).

Graphic Editing

You may draw, edit, and load your model directly in the model views. You can draw members and plates between existing nodes or draw to grid intersections and have the nodes created automatically. You may graphically select nodes to be restrained, members to be loaded, and plates to be submeshed. All or selected parts of the model may be moved, copied, and/or modified allowing you to quickly model and edit your structure. See [Graphic Selection](#) to learn how to make selections.

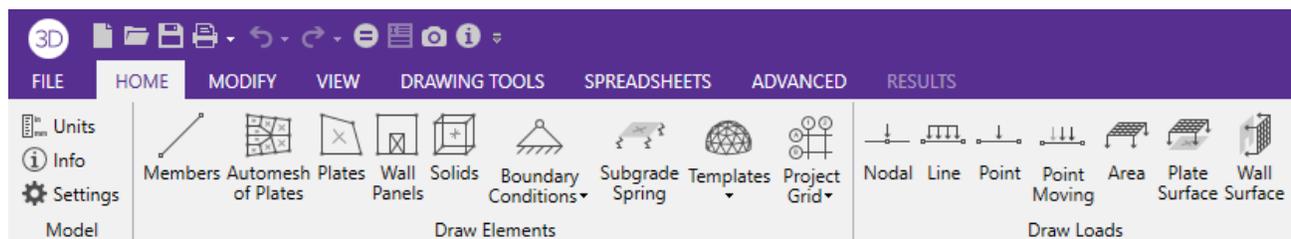
Draw / Modify Features

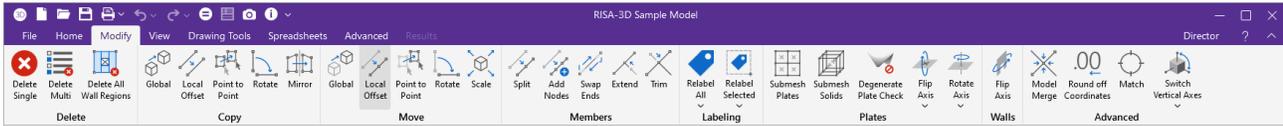
There are several graphic editing features that make the creation and modification of models quite easy. Use the **Draw Elements** tools on the **Home** ribbon and the tools on the **Modify** ribbon to utilize these features in the model view. To create new members, plates, etc., you can draw them using a drawing grid or draw "dot to dot" from existing nodes. Once you have created these items you may use other graphic features to load the model and set boundary conditions.

- See [Members](#) for more information on drawing and modifying **Members**.
- See [Plates/Shells](#) for information on drawing and modifying **Plates/Shells**.
- See [Boundary Conditions](#) for information on creating and modifying **Boundary Conditions**.
- See [Loads - Point Loads](#) for information on drawing **Point Loads**.
- See [Loads - Nodal Load / Displacement](#) for information on drawing **Nodal Loads**.
- See [Loads - Distributed Loads](#) for information on drawing **Distributed Loads**.
- See [Loads - Surface Loads](#) for information on drawing **Surface Loads**.
- See [Loads - Area Loads](#) for information on drawing **Area Loads**.

All model data is automatically recorded in spreadsheets and are always in tune with the model view. As you edit a model graphically, the spreadsheets are automatically updated and as you make changes in the spreadsheets the model views reflect these changes immediately.

All of the graphical modeling tools may be found on the **Home** and **Modify** ribbons as shown in the following images.





Where to Start

The [Project Grid](#) and [Drawing Grid](#) are often useful when you are starting a new model from scratch or adding a new section to a model. They allow you to set up grid lines which may then be used to define the members, plates, and boundaries. The differences between the two grids are simple. The Project Grid is part of the model. The Project Grid is limited to the model 'plan'. The Drawing Grid is independent of the model so you may change the grid and place it anywhere, without affecting the model and whenever it is convenient. The Drawing Grid may be placed in any global plane so you may draw in 'plan' or in 'elevation'.

There are also times when it is simpler to define nodes in the spreadsheet and then draw the members or plates between them. This might be the case if you are working with just a few nodes, or if the structure is irregular and does not lend itself to a grid.

Apply Options

The **Properties Panel** makes it very easy to apply modifications to a single member and multiple members. For example, changing the material of a beam from A36 steel to A572 steel can be accomplished through the **Members Spreadsheet**. However, if you had to apply this change to 100 beams you would not want to do that for each of them. A better way to do this would be to graphically select all of the beams and then apply the changes all at once in the **Properties Panel**.

Use the tools on the **Selection Toolbar** to choose the items you want to modify and then make the changes in the Properties panel to apply the modifications to all selected items at once. See [Graphic Selection](#) for more information on the selection tools.



Note:

- The selection and viewing tools override the graphic editing modes so that as you are editing the model you can alter the selected state. The **Properties Panel** will update based on the selection you make.
- You may also click any single node, member, or plate to view and edit it's properties in the **Properties Panel**.
- To correct any modeling errors it is a good idea to run **Model Merge** before performing a solution. Becoming familiar with this feature will also allow you to take shortcuts while modeling. See [Model Merge](#) for more information.

Undo / Redo Operations

Undo Operations

RISA-3D provides an unlimited 'Undo' capability so that you can easily correct mistakes or just back up to try different possibilities. Simply click the **Undo** icon on the **Quick Access Toolbar** as many times as you wish to undo your previous actions. The model view and the spreadsheets visually display the "undoing". Remember that spreadsheet edits are undone as well.



Note: Changes made to the selection state of the model, the zoom level or rotation of the model view, or to the Model Display Options settings are NOT undone. Only a change to the model data can be undone.

Redo Operations



RISA-3D provides an unlimited 'Redo' capability so that you can easily redo any actions that were previously undone using the 'Undo' button. Simply click the **Redo** icon on the **Quick Access Toolbar** as many times as you wish to redo actions that were previously undone. The model view and the spreadsheets visually display the "redoing". Remember that spreadsheet edits are redone as well.



Note: Changes made to the selection state of the model, the zoom level or rotation of the model view, or to the Model Display Options settings are NOT redone, since they cannot be undone to begin with. Only a change to the model data can be redone.

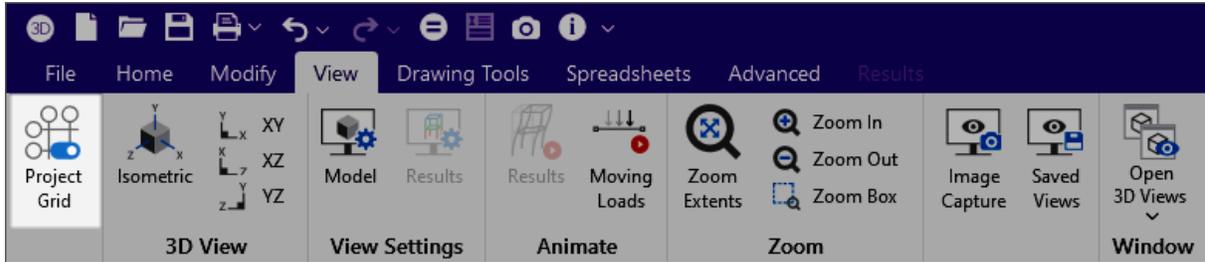
Project Grid

The Project Grid provides convenient snap points for modeling columns and walls in buildings and building-type structures. It also provides convenient terminology to refer to locations in a model, such as "Grid Intersection C-4". The Project Grid is intended to be a permanent part of the model so unlike the [Drawing Grid](#) it is saved with the model.

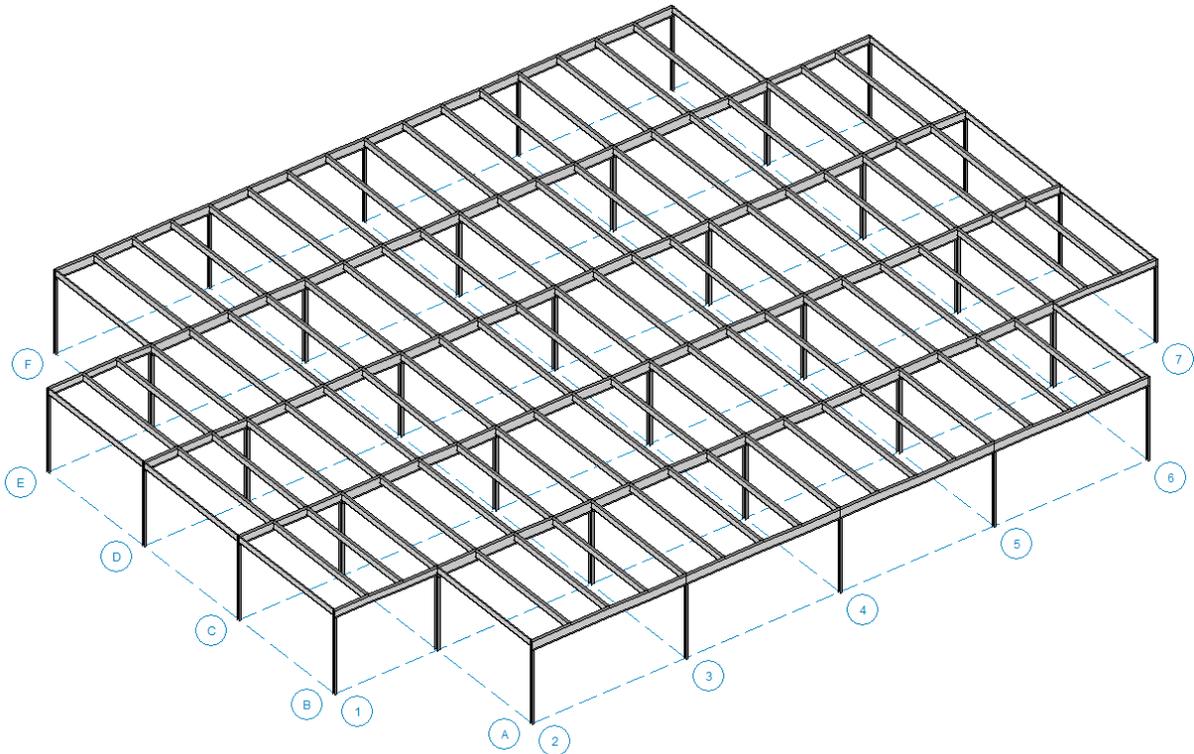
Show / Hide Project Grid Lines

To view (or hide) Project Grid lines:

1. Go to the **View** ribbon.



2. Click the **Project Grid** icon to toggle the display of the Project Grid lines “on”.
The project grid lines appear in the model viewing panel.

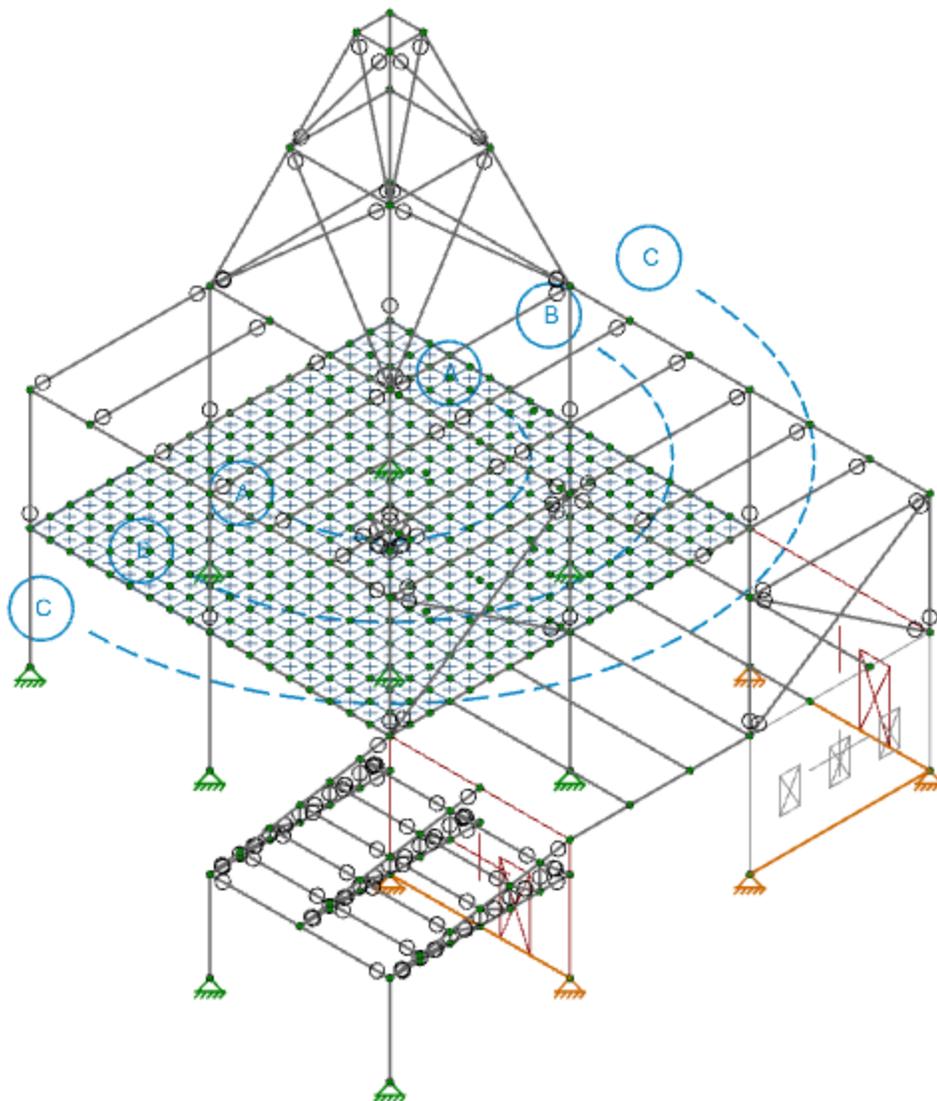


3. (Optional) Click the **Project Grid** icon again to toggle the display of the lines “off”.

Show / Hide Project Grid Arcs

To view (or hide) the Project Grid arcs:

1. Go to the **View** ribbon.
2. Click the **Project Grid** icon to toggle the display of the arcs “on”.
The project grid arcs appear in the model viewing panel.

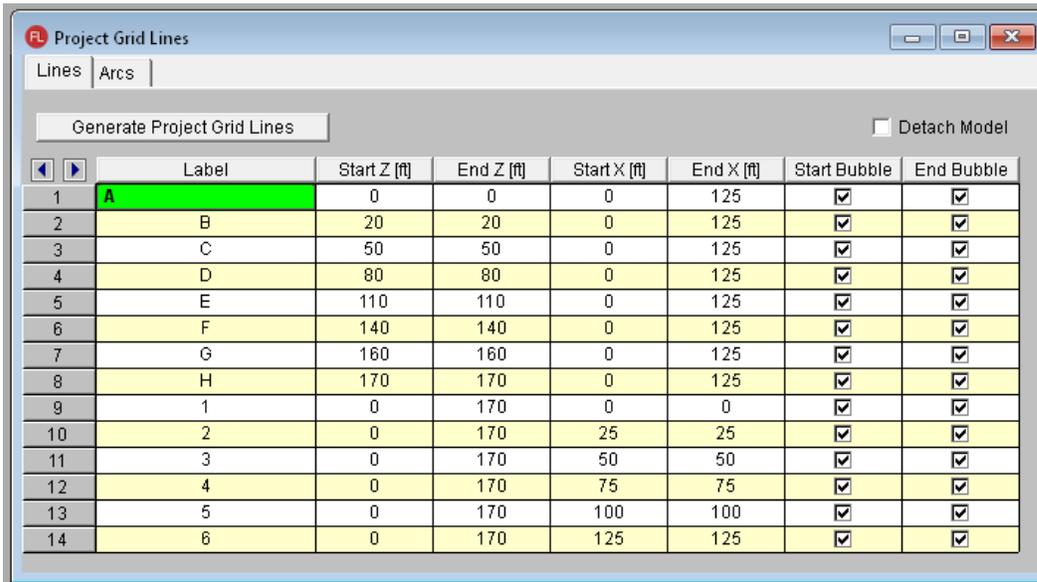


3. (Optional) Click the **Project Grid** icon again to toggle the display of the arcs “off”.

Project Grid Spreadsheet

To open the Project Grid spreadsheet:

1. Click on **Project Grid** under ‘Data Entry’ in the ‘Explorer’ panel.
The Project Grid window opens to the Lines tab.



Each Grid Line (as opposed to Grid Arcs) is defined by Start and End coordinates along the X and Z axes. This method of defining grids allows them to be oriented in any direction, including skewed grids or non-parallel grids within the same building.

- (Optional) Click the **Arcs** tab to view the generated arcs.

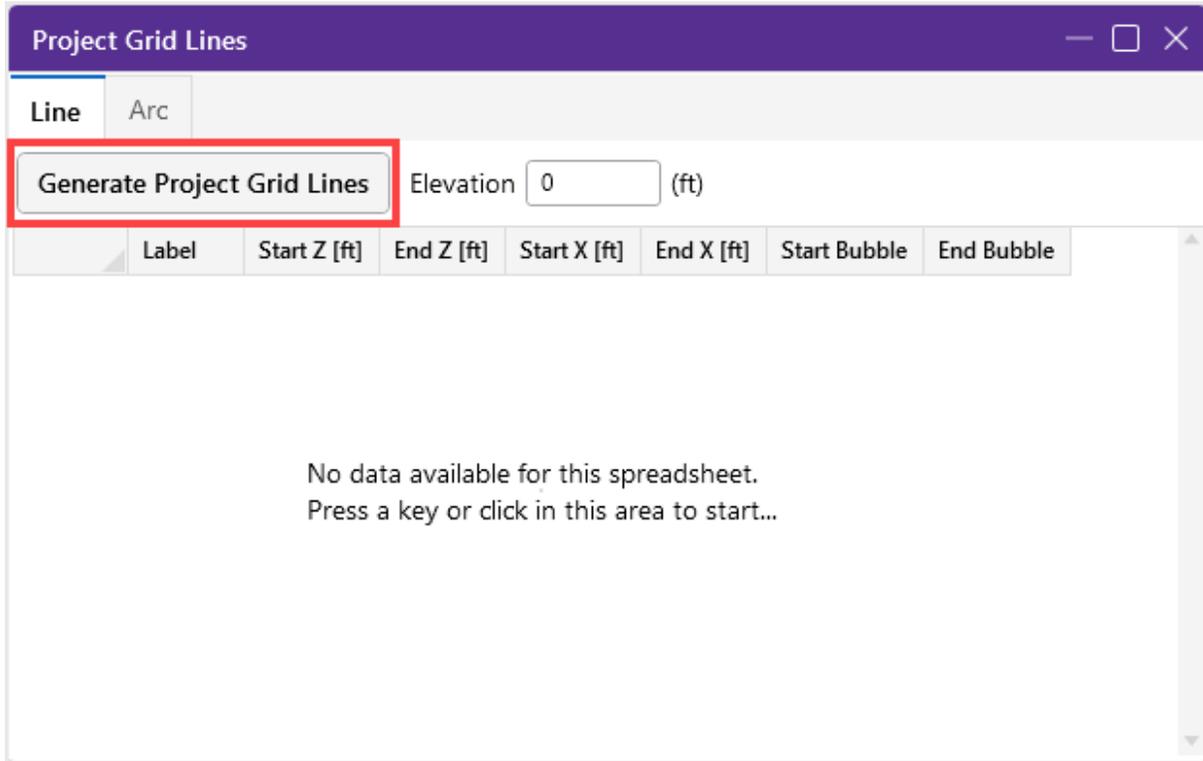
Project Grid Line Generation

Project Grid lines can be generated from the Project Grid spreadsheet or from the Home ribbon.

To generate project grid lines:

- Do one of the following.

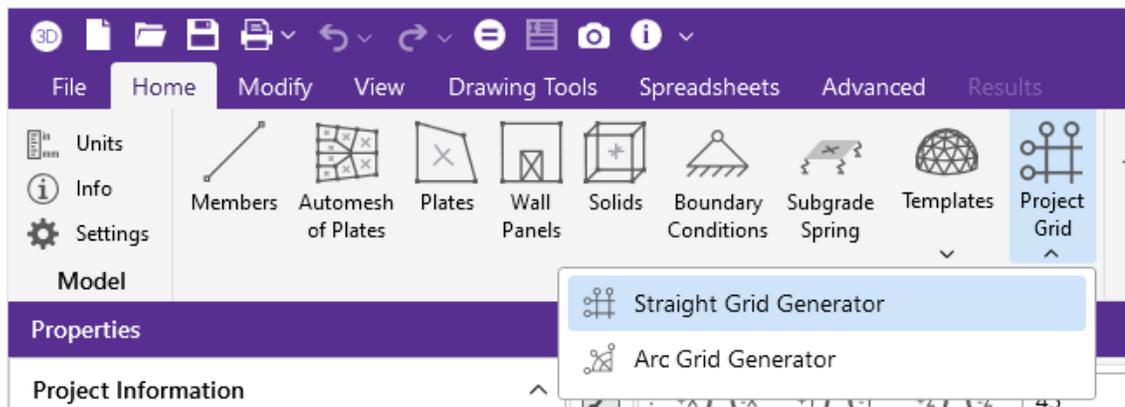
Project Grid spreadsheet - if you have the spreadsheet open:



- a. Click on the **Line** tab if not already displayed.
- b. Click the **Generate Project Grid Lines** button.

Home ribbon:

- a. Click the **Project Grid** icon.



- b. Click on **Straight Grid Generator**.

The Project Grid Lines generation window opens to the Lines tab.

Project Grid Lines

Lines | Arcs

Project Grid Origin

Z Axis ft X Axis ft

Grid Parameters

Rectangular

Z Axis Increments ft

X Axis Increments ft

Skew Angle °

Radial

Start Angle °

Angle Increments °

Radial Increments ft

Z Direction/Radial Line Labeling

Start Label (either a Letter or a Number)

Label Order A to Z Z to A

X Direction/Arc Line Labeling

Start Label (either a Letter or a Number)

Label Order A to Z Z to A

Save Current Settings as Defaults?

Undo Preview Apply Close

The generator provides the ability to generate an entire grid system at once rather than entering the grids in manually in the project grid spreadsheet.

- You can use symbols such as "@", "/" and "," when entering the increments.
- The "@" entry can be used to specify multiple, equally spaced, grid increments.
For example, if you wanted 7 increments at 10 units each, you would type "7@10" in the increment field.
- The "/" entry subdivides a larger increment into smaller equal increments.
For example, the entry "12/4" would create 4 increments of 3 units each.

- Use commas (",") to enter multiple increments in the increment field.
For example, if you wanted to define increments of 3, 4, 7 and 2 units, you could enter "3,4,7,2" in the increment field.
 - The **Preview** button lets you see the project grid lines prior to applying them.
 - The **Preview** and **Undo** buttons provide an easy way to preview a Project Grid before adding one to the model.
2. (Optional) Change the options to those you want applied.
Don't forget to "Preview" your settings before you apply them.
 3. Click **Apply** to generate and apply the project grid lines to the model.
Once the project grid is specified and displayed in the model view, it provides snap points while drawing your model. The grid exists at an elevation specified in the **Elevation** field.

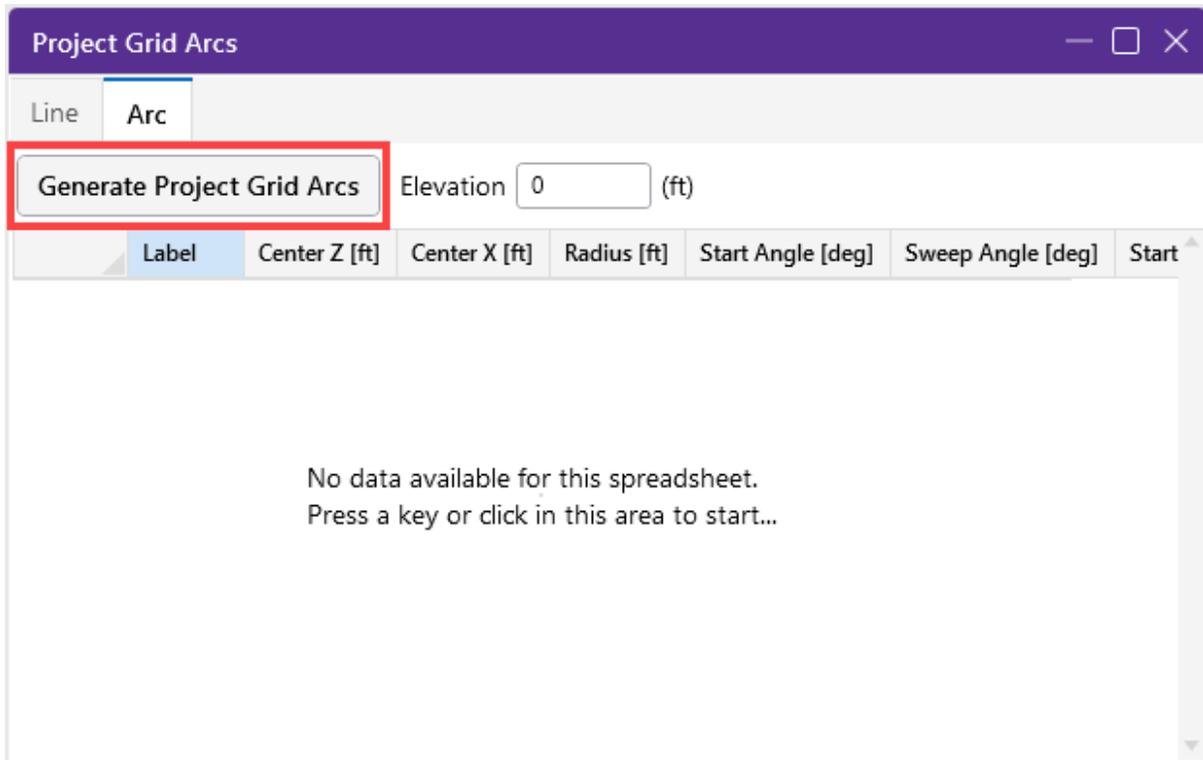
Project Grid Arc Generation

Project Grid arcs can be generated from the Project Grid spreadsheet or from the Home ribbon.

To generate project grid arcs:

1. Do one of the following.

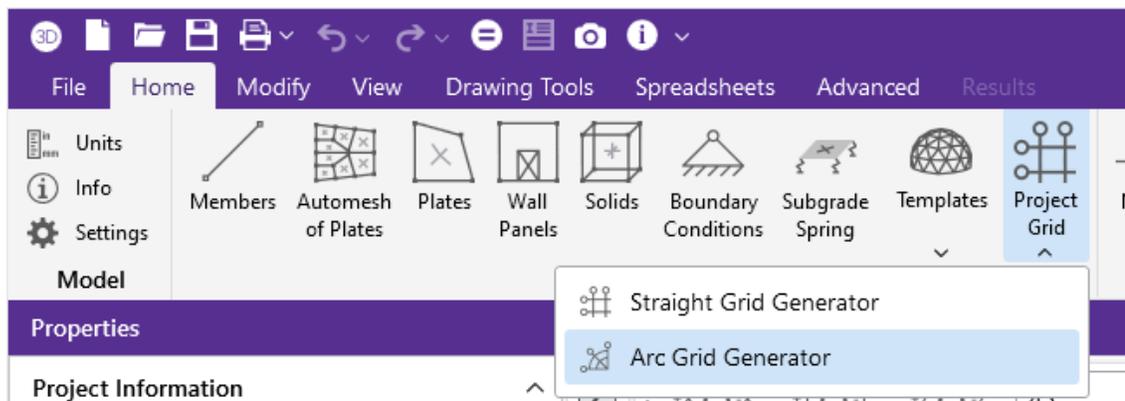
Project Grid spreadsheet - if you have the spreadsheet open:



- a. Click on the **Arc** tab if not already displayed.
- b. Click the **Generate Project Grid Arcs** button.

Home ribbon:

- a. Click the **Project Grid** icon.



- b. Click on **Arc Grid Generator**.

The Project Grid Arcs generation window opens to the Arcs tab.

Project Grid Arcs

Lines | **Arcs**

Project Grid Origin

Z Axis ft X Axis ft

Start Angle °

Sweep Angle °

Radial Increments ft

Arc Start Label (either a Letter or a Number)

Label Order: A to Z Z to A

Radial Lines

Generate Radial Lines (Spokes)

Line Start Label (either a Letter or a Number)

No. of Spokes - +

Label Order: A to Z Z to A

Save Current Settings as Defaults?

Undo Preview Apply Close

The generator provides the ability to generate an entire grid system at once rather than entering the grids in manually in the project grid spreadsheet.

- You can use symbols such as "@", "/" and "," when entering the increments.
- The "@" entry can be used to specify multiple, equally spaced, grid increments.
For example, if you wanted 7 increments at 10 units each, you would type "7@10" in the increment field.
- The "/" entry subdivides a larger increment into smaller equal increments.
For example, the entry "12/4" would create 4 increments of 3 units each.

- Use commas (",") to enter multiple increments in the increment field.
For example, if you wanted to define increments of 3, 4, 7 and 2 units, you could enter "3,4,7,2" in the increment field.
 - The **Preview** button lets you see the project grid arcs prior to applying them.
 - The **Preview** and **Undo** buttons provide an easy way to preview a Project Grid before adding one to the model.
2. (Optional) Change the options to those you want applied.
Don't forget to "Preview" your settings before you apply them.
 3. Click **Apply** to generate and apply the project grid arcs to the model.
Once the project grid is specified and displayed in the model view, it provides snap points while drawing your model. The grid exists at an elevation specified in the **Elevation** field.

Edit the Project Grid Lines (Adding or Moving Grid Lines)

Once a Project Grid has been created the grid lines can be moved, or additional grid lines can be created, by double clicking on a grid line in the model view.

3D Edit Project Grid Line ? X

Label

Start Z ft

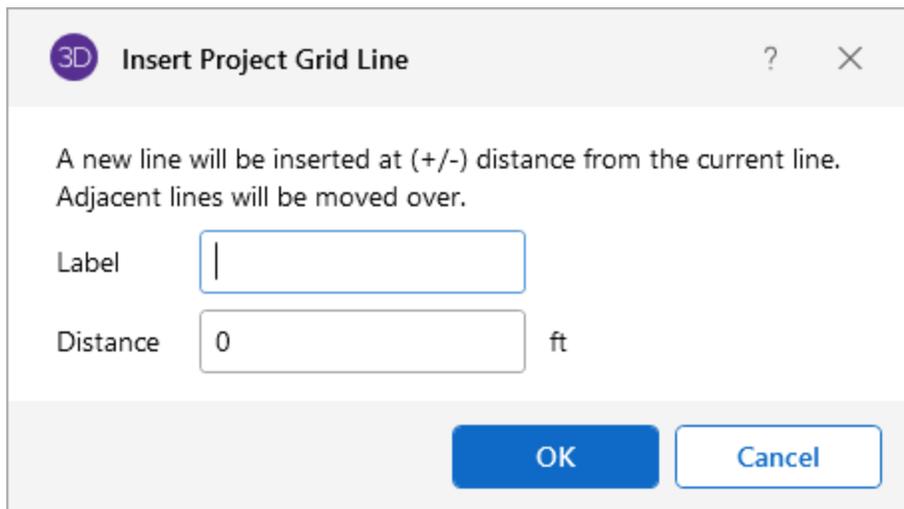
Start X ft

End Z ft

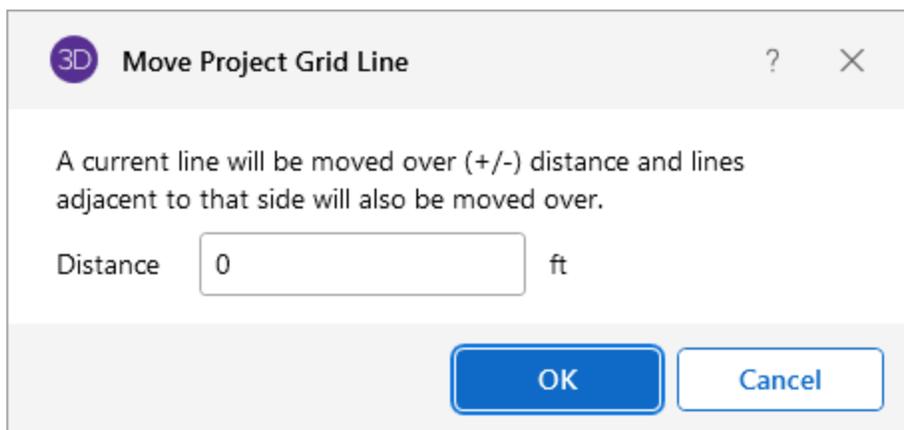
End X ft

Start Bubble End Bubble

The **Generate Parallel Line** button allows a line to be created by offsetting an existing grid line.

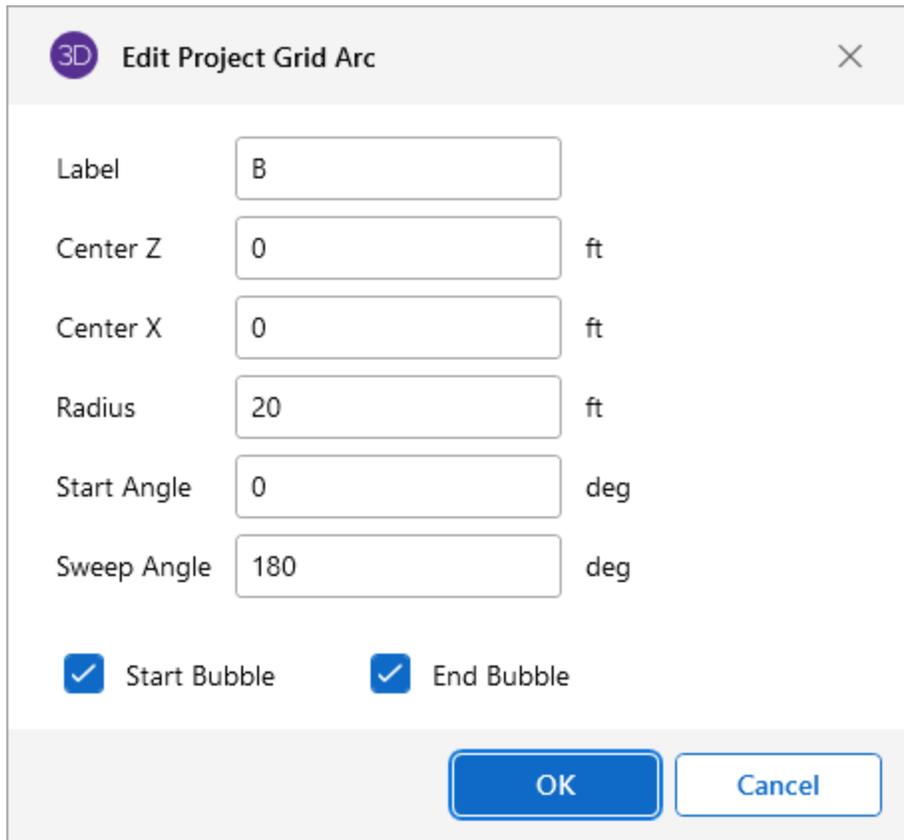


The **Move Lines** button allows a grid line to be moved by a specified distance. A positive distance moves the line in the positive global direction, while a negative distance moves the line in the opposite direction.



Edit the Project Grid Arcs (Adding or Moving Grid Arcs)

Once the Project Grid Arcs have been created, an arc's shape can be changed by double clicking on it in the model view and modifying its settings.

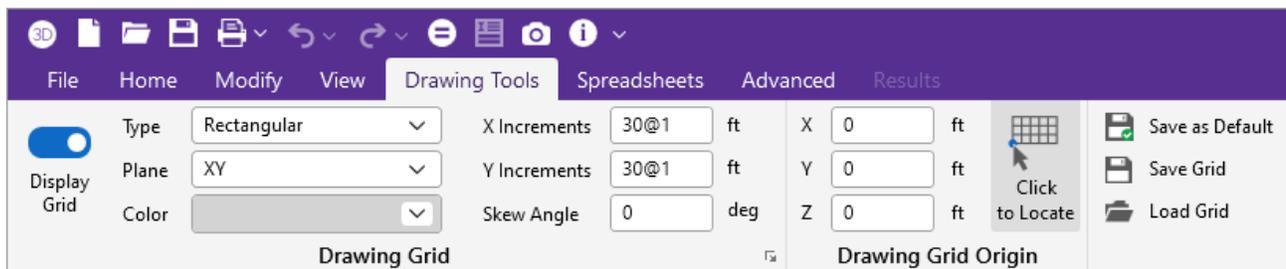


Drawing Grid

The **Drawing Grid** is a tool that lets you draw new members and plates in the model view. This grid is independent of the model, so you may change the grid as you build your model without changing any modeling that you have completed. This is because as you draw the members and plates the joints used to define them are created automatically. These joints remain in their locations if the grid is relocated.

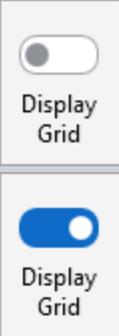
Drawing Grid Features

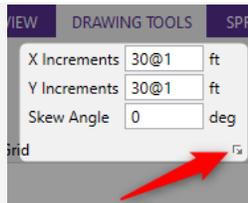
The Drawing Grid tools can be found on the Drawing Tools ribbon. These features allow you to define, save and reuse rectangular, radial and DXF underlay drawing grids. You can also set a specific grid as your default.



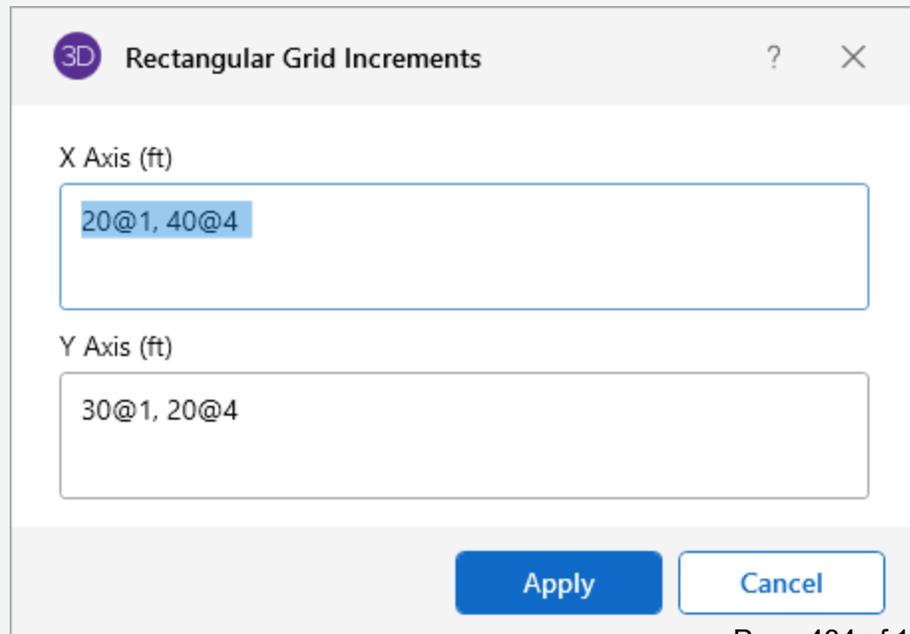
To use the Drawing Grid features refer to the following table and sections.

Drawing Grid Features

Feature	Description
Drawing Grid section	
	<p>The 'Display Grid' icon lets you turn on (show) or off (hide) the drawing grid in the current model.</p> <ul style="list-style-type: none"> • Gray slider - indicates that the drawing grid is off (hidden) • Blue slider - indicates that it is on (showing). <p>See Show / Hide the Drawing Grid for more information.</p>
Type	The 'Type' feature lets you choose between a rectangular drawing grid, radial drawing grid or DXF underlay.
Plane	The 'Plane' feature lets you place the grid in any one of the three global planes.
Color	The 'Color' feature lets you choose what color you would like the drawing grid lines displayed in.
X Increments__ft	The 'X Increments' feature lets you set the number of and width between the drawing grid lines.
Y Increments__ft	The 'Y Increments' feature lets you set the number of and height between the drawing grid lines.
Skew Angle__deg	The 'Skew Angle' lets you skew the drawing grid perpendicularly to the global axis plane.



The icon located to the bottom right of the 'Skew Angle' feature lets you open the "Rectangular Grid Increments" window.



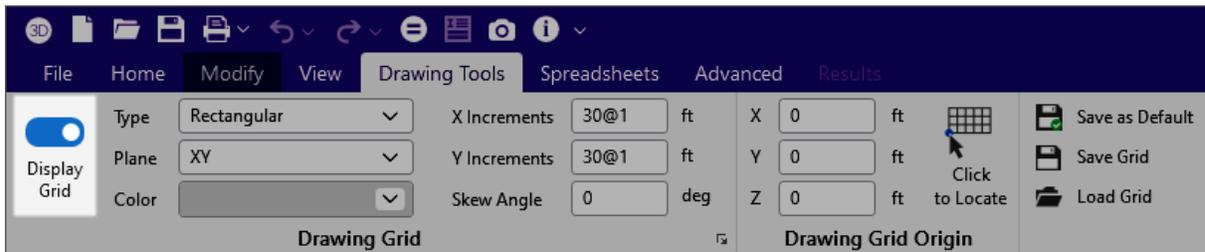
Feature	Description
	The “Rectangular Grid Increments” window lets you enter more X and Y incremental values.
Drawing Grid Origin section	
X__ft	The ‘X’ grid origin lets you set the increment, in feet, for the X direction. For more information see Rectangular Drawing Grid .
Y__ft	The ‘Y’ grid origin lets you set the increment, in feet, for the Y direction. See Rectangular Drawing Grid for more information.
Z__ft	The ‘Z’ grid origin lets you set the increment, in feet, for the Z direction. See Rectangular Drawing Grid for more information.
	The ‘Click to Locate’ icon lets you specify or relocate the origin of the drawing grid. See Relocating the Drawing Grid for more information.
Save and Recall features section	
 Save as Default	The ‘Save as Default’ feature lets you save any of the grid information as the default settings so that when you start a new model that information is already there. See Save a Drawing Grid Default for more information.
 Save Grid	The ‘Save Grid’ feature lets you save a drawing grid so you can recall it for later use. Saved drawing grids are model independent, i.e. when you save a grid, you can reuse it in any other model you are working with in the future. See Save a Drawing Grid for more information.
 Load Grid	The ‘Load Grid’ feature lets you recall a saved drawing grid. See Recall a Drawing Grid for more information.

Show / Hide the Drawing Grid

The ribbon includes a feature that works like a toggle; allowing you to show or hide the drawing grid in the current model.

To show or hide the drawing grid:

1. Open to the **Drawing Tools** ribbon.



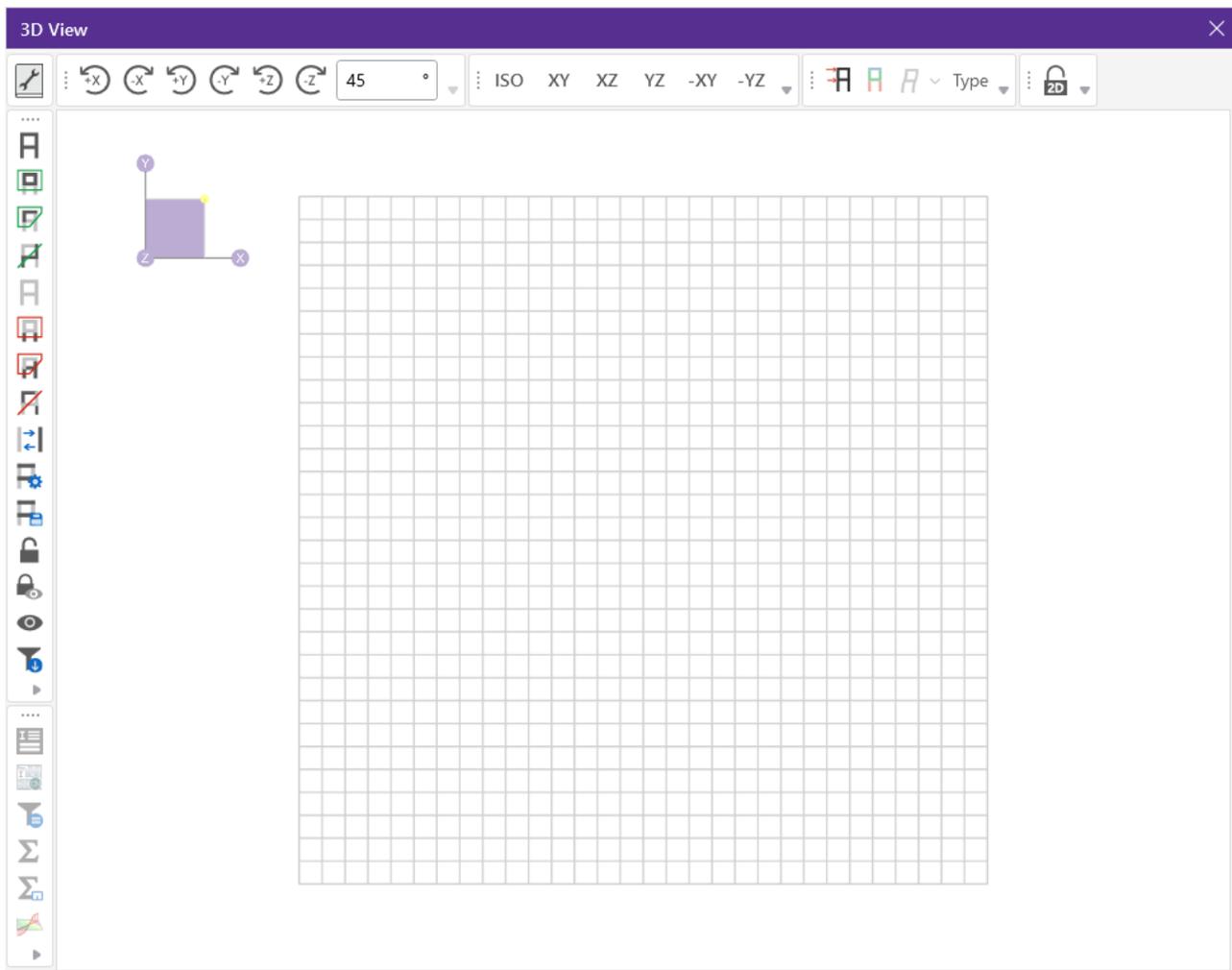
2. Click on the **Display Grid** icon.

- When the Display Grid slider is **blue**, it indicates that the drawing grid is turned 'on' (shown).
- When the Display Grid slider is **gray**, the drawing grid turned 'off' (hidden).

Drawing Grid Type

Rectangular Drawing Grid

The rectangular drawing grid is defined by increments in two directions. The **X, Y, Z Origins** are where you want the grid increments to start. The **X, Y, Z Increments** are the distances between the grid points or lines in two global directions.



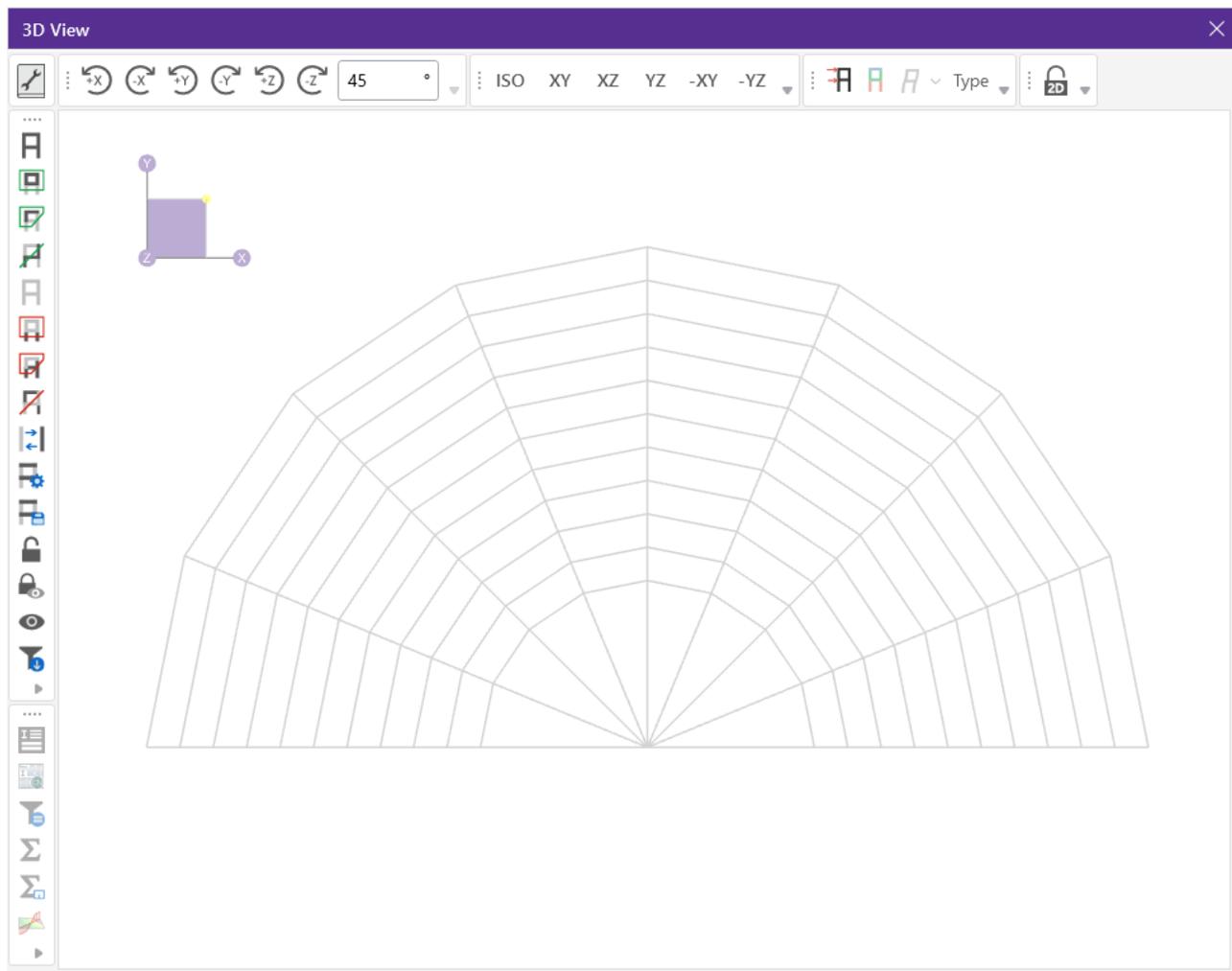
You may use symbols such as "@", "/" and "," when entering the drawing grid increments.

- The "@" entry may be used to specify multiple, equally spaced, grid increments.
For example, if you wanted 7 increments at 10 units each, you would type "7@10" in the increment field.
- The "/" entry subdivides a larger increment into smaller equal increments.
For example, the entry "12/4" would create 4 increments of 3 units each.
- Use commas (",") to enter multiple increments in the increment field.
For example, if you wanted to define increments of 3, 4, 7 and 2 units, you could enter "3,4,7,2" in the increment field.

Once the drawing grid is specified and displayed in the model view, it provides snap points while drawing your model.

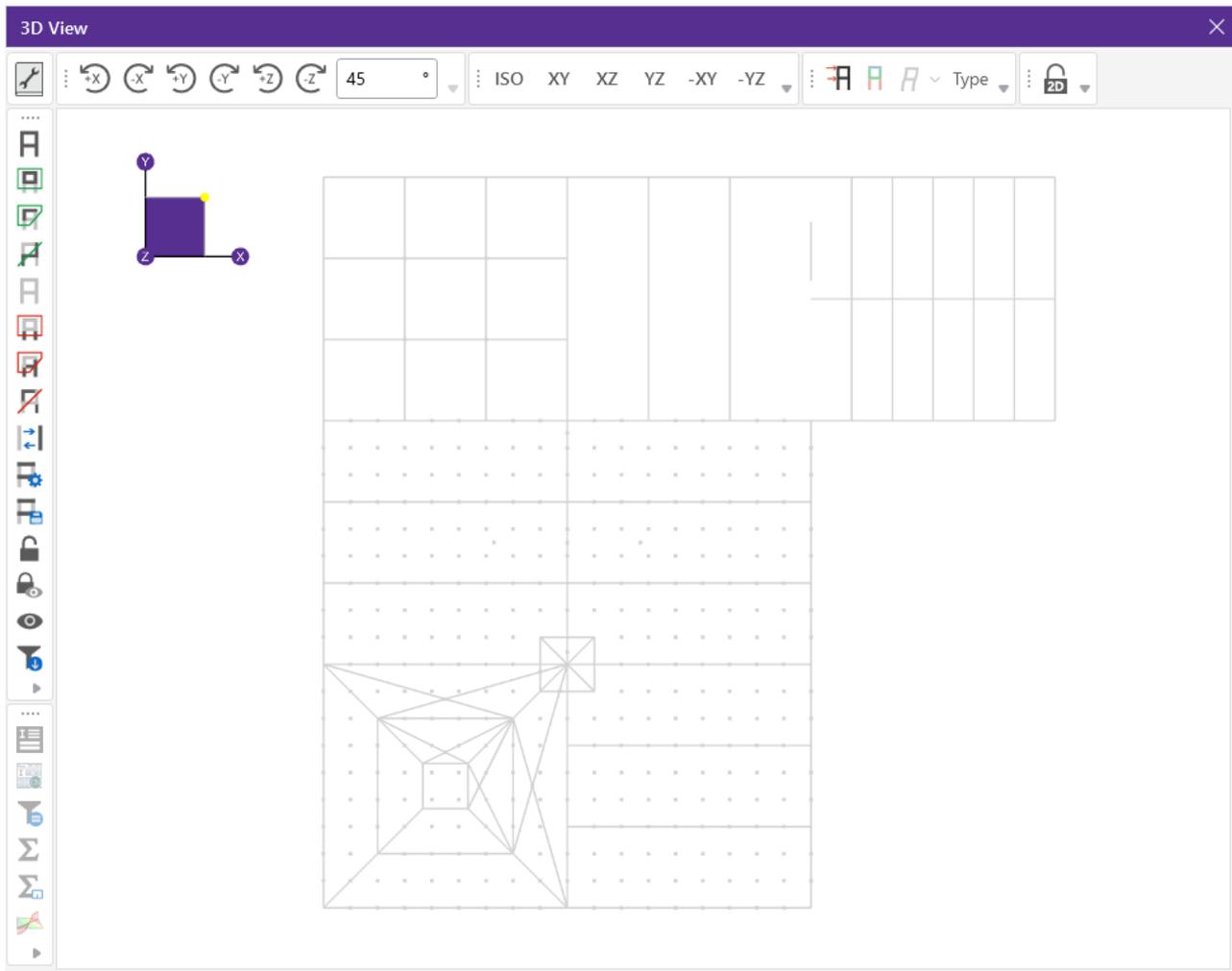
Radial Drawing Grid

Increments in two polar directions define a radial drawing grid. The **Drawing Grid Origin** is the point about which the grid will rotate. The default is at the global origin (0,0,0). The **Start Angle** defines the angle from the global axis that the first spoke will be drawn. The **Angle Increment** controls the number and angular spacing of the spokes in the grid. The **Radial Increments** controls the number and location of the rings in the grid.



DXF Underlay Grid

This type should only be chosen when you want to import a DXF Grid from a DXF file. The following is an example of an imported DXF file underlay grid.



Relocating the Drawing Grid

The origin of the drawing grid may be specified in two ways. The first is to enter the exact global coordinates for the origin. This can be done by entering the values in the **X Origin**, **Y Origin**, and **Z Origin** boxes. The second option is to specify the origin by clicking on an existing point in the model. This option is available by clicking the **Click to Locate** button, and then clicking on the specific location in the model view where you wish the drawing grid origin to be located. The drawing grid origin is then moved to this point.

Skewed Drawing Grid

The rectangular drawing grid can be skewed by specifying a skew angle. This option is available in the **Drawing Grid Section** of the **Drawing tools** ribbon. This skew angle allows for the creation of a regular rectangular drawing grid, but displayed in the model view at the specified angle, inclined from the global axis.

Save a Drawing Grid Default

You may save any of the grid information as the default setting so that when you start a new model that information is already there.

To save a drawing grid as the default:

1. In the **Drawing Grid** and **Drawing Grid Origin** sections of the 'Drawing Tools' ribbon, enter the information that you want to save.
2. Click on  **Save as Default** **Save as Default** in the 'Drawing Tools' ribbon.

Save a Drawing Grid

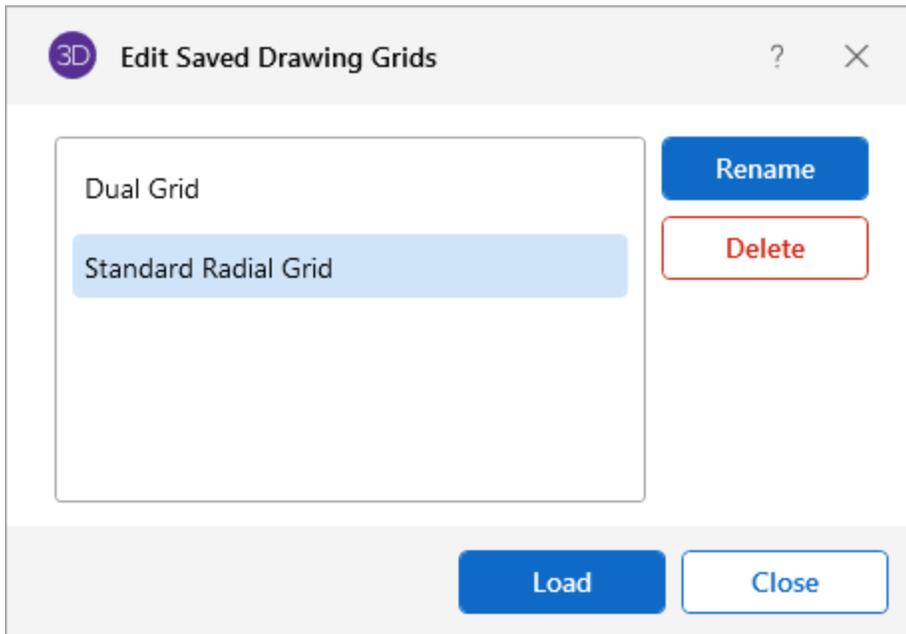
To save a drawing grid:

1. Use the table above to define the grid settings.
2. Click on  **Save Grid** **Save Grid** in the 'Drawing Tools' ribbon, after defining the grid.
You are prompted for a name for the drawing grid. Once you provide a name, the drawing grid is saved and added to the list of grids available for recall.

Recall a Drawing Grid

To recall a previously defined drawing grid:

1. Click on  **Load Grid** **Load Grid** in the 'Drawing Tools' ribbon.
The Edit Saved Drawing Grids window opens.



2. Click on the drawing grid you want.
3. Click the **Load** button.



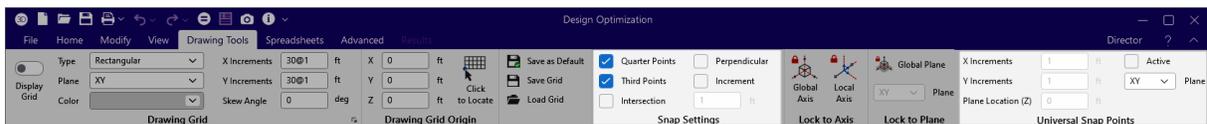
Note: You can also “Rename” drawing grids and “Delete” drawing grids from the Edit Saved Drawing Grids window. To choose either option, click the **Rename** or **Delete** button after choosing the drawing grid.

Snap Points

Snap Points let you draw in the model view without the use of grids.

To view or modify the snap point settings:

1. Go to the **Drawing Tools** ribbon.
2. Go to the **Snap Settings** section.



3. (Optional) Make whatever modifications you want.
 - You can set the program to automatically snap to the **Quarter Points** and/or **Third Points** and/or **Intersections** of a member by checking the appropriate boxes.

- The **Universal Snap Points** section is used to define a snap grid for "free" drawing to any incremental location on a plane. To activate this feature, check the **Active** box.

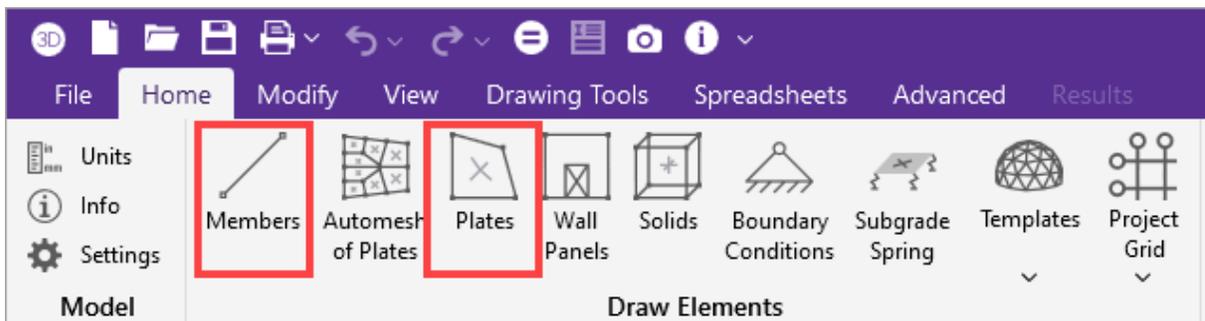
When snap points are activated, a red dot or asterisk appears on your screen as you move your drawing cursor over one of these points after you have selected to draw an element. The exact coordinates of this point, and whether it is a 1/3 or 1/4 point of a member, are reported in the status bar at the very bottom of the main application window just beneath the workspace as well as next to the cursor.

Lock to Axis

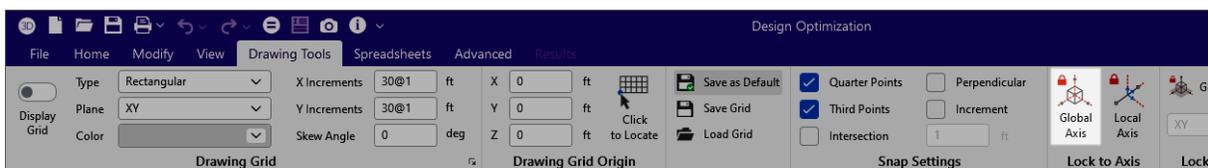
The "Lock to Axis" feature can be used to draw members and plates while locked to the global or local axis.

Lock to Global Axis

1. Go to the **Home** ribbon.



2. Click on either **Draw Members** or **Draw Plates**.
3. Go to the **Drawing Tools** ribbon.



4. Click on **Global Axis** in the 'Lock to Axis' section.
5. Click in the **3D View** to specify the origin of the lock axis, which is shown in dashed red lines.

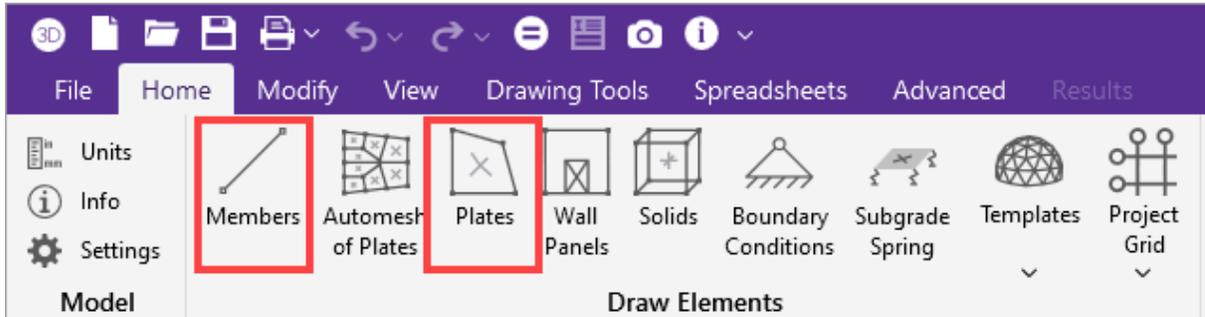
This means the lock axis will follow the global axis orientation.

As you draw, the origin of the lock axis changes with every subsequent click. The axis continues to follow the global axis orientation.

6. To **disengage the drawing tool**, right-click twice.

Lock to Local Axis

1. Go to the **Home** ribbon.



2. Click on either **Draw Members** or **Draw Plates**.
3. Go to the **Drawing Tools** ribbon.

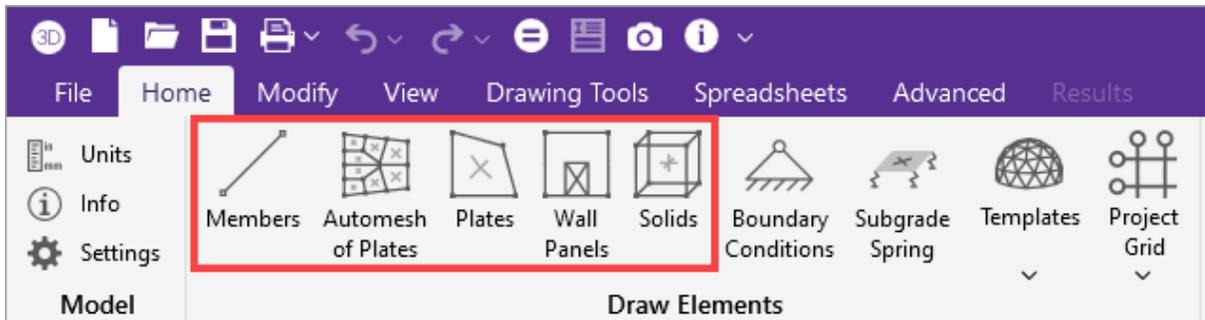


4. Click on **Local Axis** in the 'Lock to Axis' section.
5. Click on a member, plate or wall panel to specify the local axis to lock to.
6. Click in the 3D View to specify the origin of the lock axis, which is shown in dashed red lines.
7. As you draw, the origin of the lock axis changes with every subsequent click.
The axis orientation remains locked to the local axis of the originally specified element.
8. To **specify a new local axis** to lock to, right-click twice (the cursor changes to a hand) and click on a new member, plate or wall panel.
9. To **disengage the drawing tool**, right-click three times.

Lock to Plane

The "Lock to Plane" feature can be used while drawing members, wall panels, or plates, to draw while locked to a specific global plane.

1. Go to the **Home** ribbon.



2. Click on the element you want to draw.
3. Go to the **Drawing Tools** ribbon.



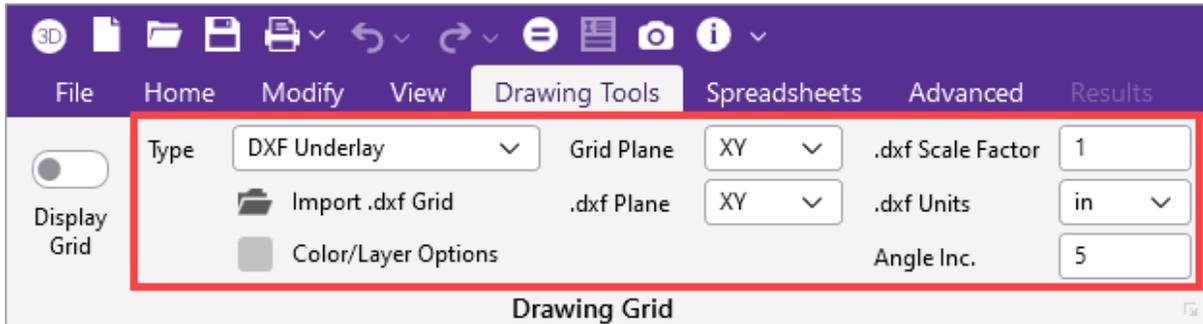
4. Click on **Global Plane** in the 'Lock to Plane' section.
5. Click the **Plane** arrow choose the plane (XY, XZ or YZ).
6. Click in the 3D View to specify the location of the plane.
7. To **reset the origin of the plane**, right-click once; while still in the drawing tool, make another click to reset the origin of the plane.
8. To **disengage the drawing tool**, right-click twice.

Import DXF

Import DXF lets you import a DXF drawing into the model view as a grid. The DXF image is imported so that you can snap to any point or intersection to aid in the drawing of your model. The DXF Import Drawing Grid supports Lines, Polylines, Circles, Arcs, Polylines with Arcs, and Points.

To Import a DXF Drawing Grid:

1. Go to the **Drawing Tools** ribbon.
2. Click on the **Type** arrow and choose **DXF Underlay**.

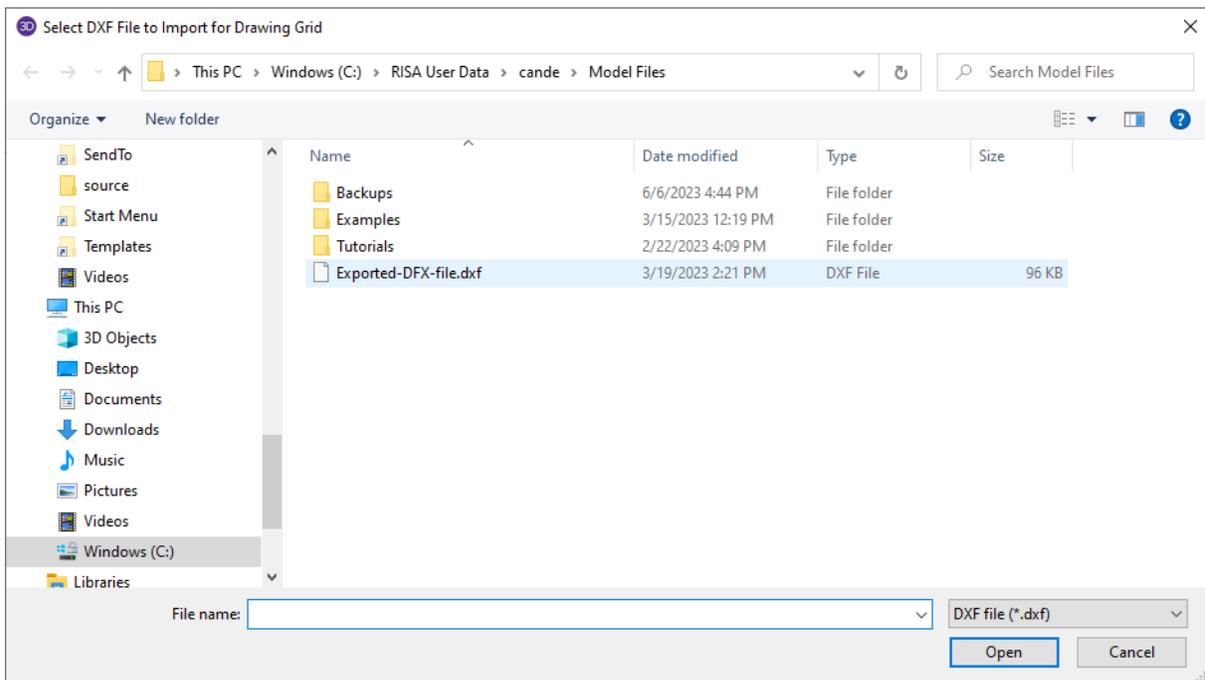


The following table provides information on the various related Drawing Grid settings.

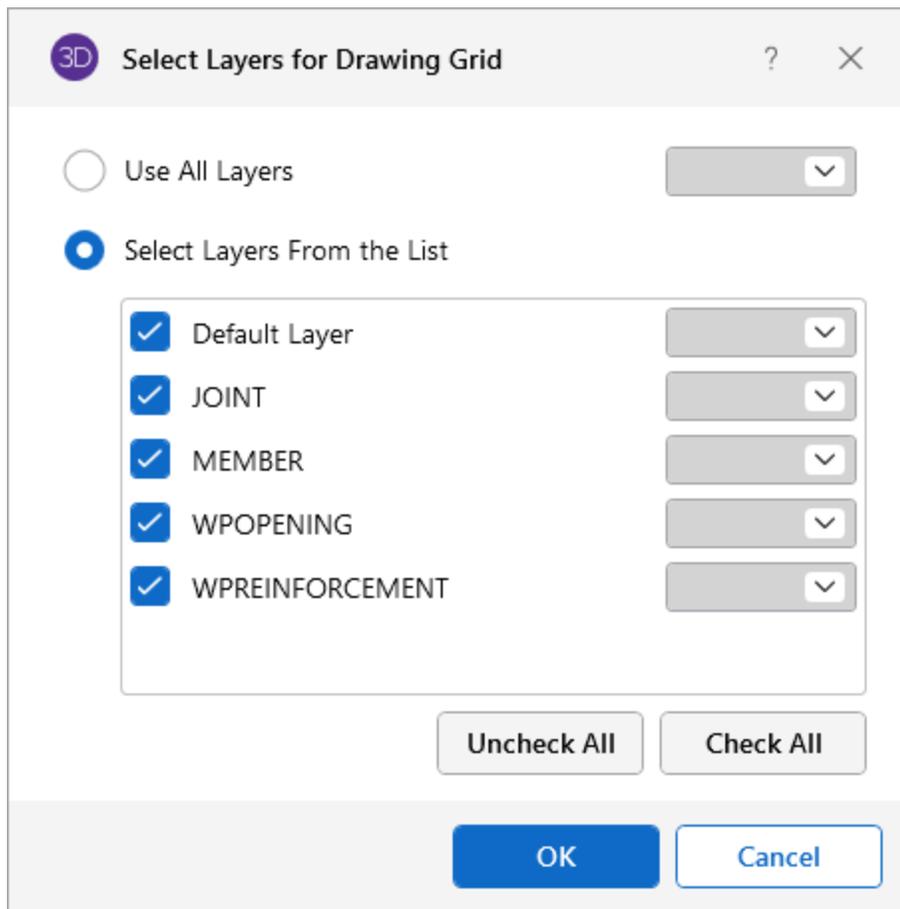
DXF Import Drawing Grid Settings

Setting	Description / Explanation
Grid Plane	This setting lets you choose the RISA drawing plane for the DXF file shall be placed.
.dxf Plane	This setting lets you choose the DXF drawing plane to import from the DXF file. The DXF file will be flattened to this plane and all lines/points visible from this plane will be imported.
.dxf Scale Factor	This setting lets you enter the scale factor that will cause the DXF file to be scaled up or down to full scale. For instance, if you have created a scaled model in AutoCAD at a scale of 1/4"=12", then the appropriate scale factor to produce a full size RISA-3D model would be 48. The default is 1.0
.dxf Units	This setting lets you choose the same 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.
Angle Increment	The program breaks an arc or circle into straight line segments to allow you to snap to them. This setting indicates the number of degrees that an imported arc or circle should be broken into.

- Assign the DXF import options like **Scale Factor**, **Units** and **Angle Inc.** as well as the **.dxf Plane**.
- Click on **Import .dxf Grid**.
A Select DXF File to Import for Drawing Grid window opens.

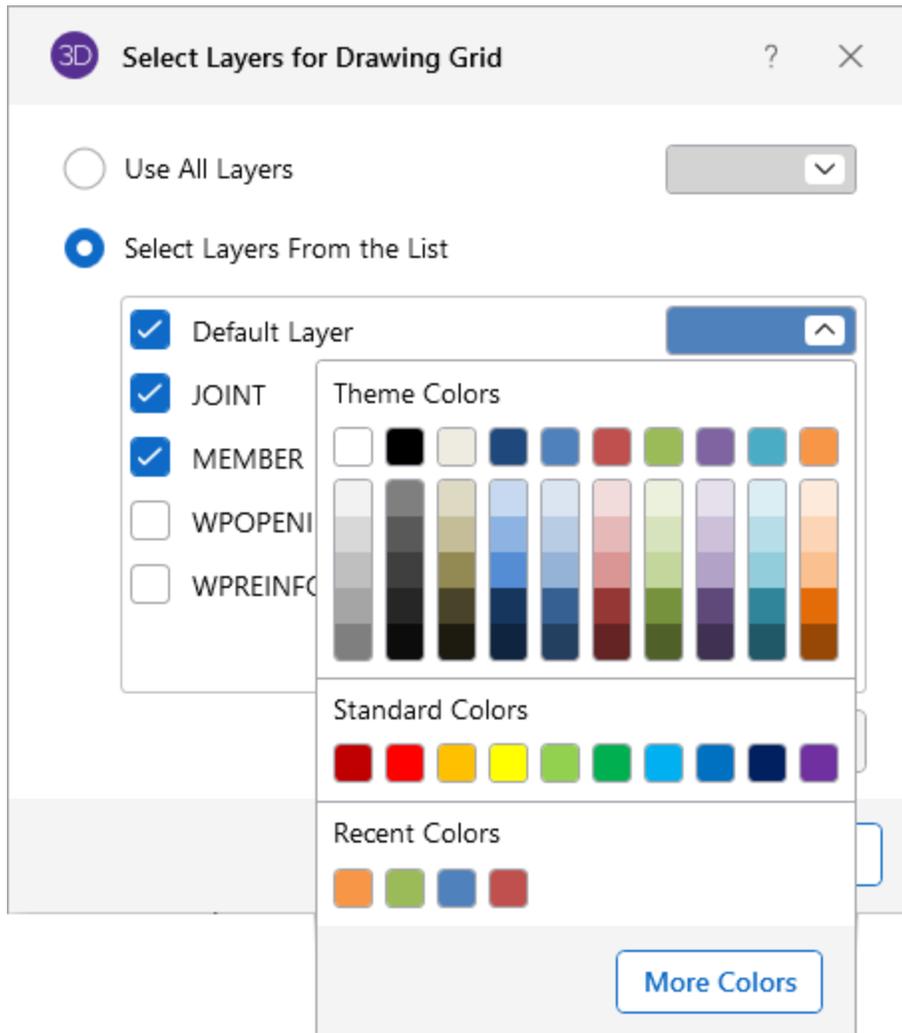


5. Choose the DXF file to import then click **Open**.
Once the file is imported, a Select Layers for Drawing Grid window opens.



The app defaults to 'Use All Layers'.

6. Choose whether to **Use All Layers** or **Select Layers from the List**.
7. If you chose 'Select Layers From the List', check the layers you want and uncheck those you don't want.
You can select the color of the grid lines/points from the color box next to each layer name.
8. (Optional) Click on a layer's color box arrow, to select a color other than the gray default.



9. Click **OK** to save your choices and close the window.
10. Set the **Drawing Grid Origin** or use the drop down menu to select **Click to Locate** origin.

Locating the DXF Drawing Grid

The origin of the drawing grid may be specified in two ways. The first is to enter the exact global coordinates for the origin. This can be done by entering the values in the X, Y, and Z fields of the **Grid Origin** section. The second option is to specify the origin by clicking on an existing point in the model. This option is available by selecting the **Click to Locate Origin** option from the drop down menu, clicking the **OK** button, and then clicking on the specific location in the model view where you wish the drawing grid origin to be located. After you have browsed for your file, the drawing grid origin will then be set to this point.

Delete Elements

You can delete parts of a model based on the current selection state, or you can click on the items you wish to delete individually.

Two noteworthy features are the ability to delete unattached nodes and zero length members. Sometimes in the process of modeling you accidentally create unwanted unattached nodes or zero length members. These two parameters give you a convenient way to remove these unwanted items.



Note: Primary nodes for rigid diaphragms are not treated as unattached nodes.



Note: If you request deletion of displayed loads, you'll get exactly that. Any load currently displayed will be deleted. By controlling what loads are displayed via the **Loads** tab in the **Model View Settings** window, you can easily delete specific types of loads for particular basic load cases, load categories or load combinations.

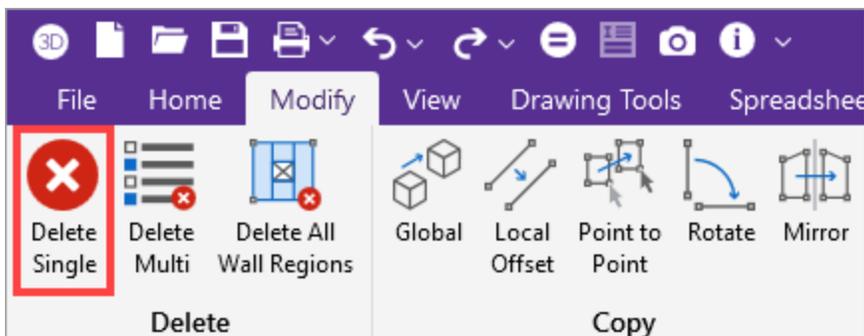


Note: You may undo any mistakes by clicking the  **Undo** button in the Quick Access toolbar.

Delete a Single Model Part

To delete a single model part, or multiple model parts, one at a time:

1. Go to the **Modify** ribbon.



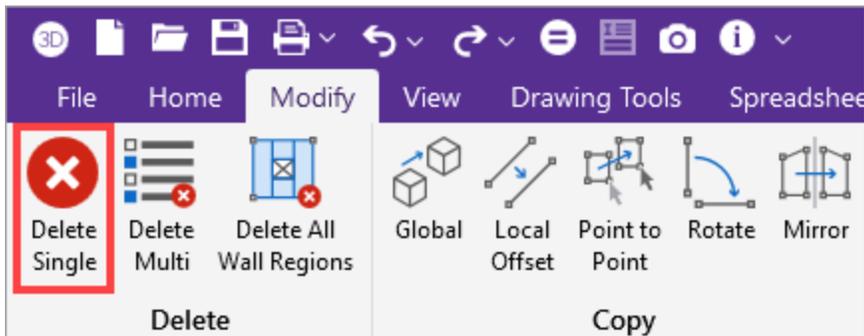
2. Click the **Delete Single** icon.
3. Click on the model part you want to remove.
4. Repeat **step 3** for each model part you want to remove.
5. When finished, click the **Delete Single** icon again, to toggle off the single delete feature.

Delete Multiple Model Parts

If you wish to delete based on the current selection state, you must use the check boxes to define the criteria the program will use to perform the deletion. Only items that are selected and that have their check boxes "checked" will be deleted. The choices let you delete nodes, members, plates, wall panels, solids and/or loads.

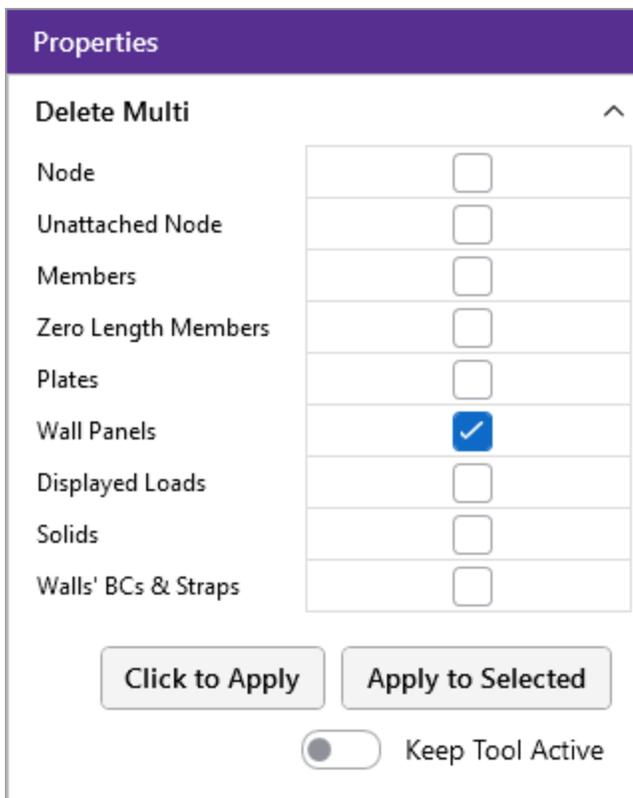
To delete multiple model parts at once:

1. Go to the **Modify** ribbon.



2. Click the **Delete Multi** icon.

The Properties panel displays a list of all the parts that can be deleted.



3. Do one of the following, depending on exactly what parts you want to delete.

To remove ALL of one or more part types

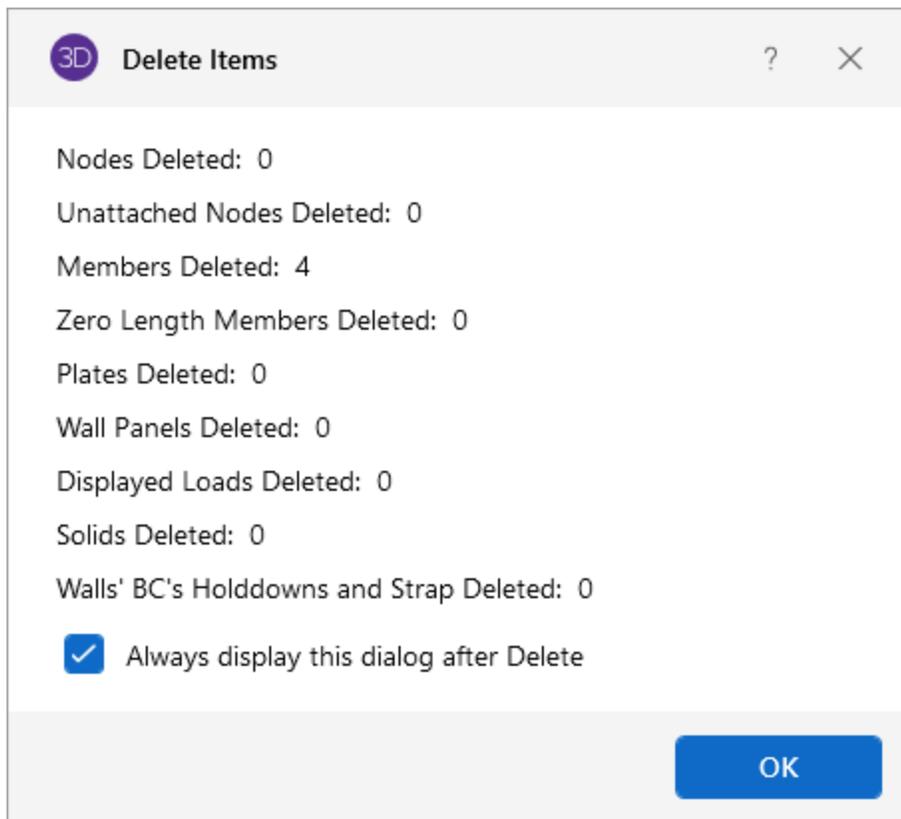
- a. Check the box of each part type you want deleted from the model.
- b. (Optional) Click on the **Keep Tool Active** slider if you want the 'Delete Multi' list to remain open in the Properties panel, so that you can delete additional part types or parts.
- c. Click the **Click to Apply** button.

This action removes all parts of the type(s) you have chosen in the Properties panel.

To remove specific parts of multiple types

- a. Check the box of each part type you want to delete a part from in the model.
- b. Hold down the **ctrl** key (on your keyboard) while clicking on each part in the model that you want to delete.
- c. Click the **Apply to Selected** button.

A delete Items window opens displaying the number of each type of model part that will be deleted.



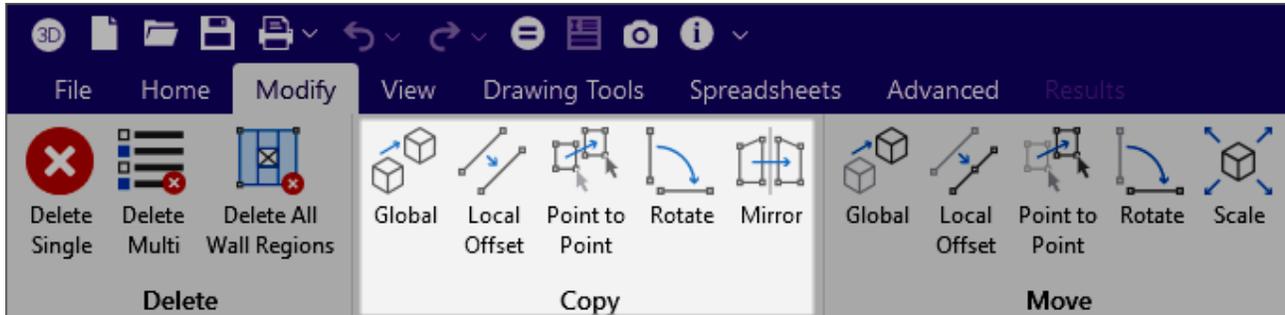
- d. Click **OK**.

The window closes and the items are deleted from the model view.

Keep in mind that if you delete nodes, any members, plates, etc. attached to the deleted nodes **MUST** also be deleted, regardless of whether those elements are selected or not.

Copy Model Elements

The model elements copy options can be found on the Modify ribbon in the Copy section.



Copy Selected Items

To copy selected items:

1. Open a model view if one is not already open.
 - a. Go to the **View** ribbon.
 - b. Click the **Open 3D Views** icon.
2. Select the items you want to copy.
3. To *linearly* copy the selection:
 - a. Go to the **Modify** ribbon.
 - b. Under the **Copy** section, choose one of the following options:
 - [Global](#)
 - [Local Offset](#)
 - [Point to Point](#)
 - [Rotate](#)

See explanations in the following sections for more information about these choices.

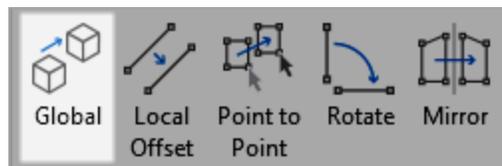
4. To *Mirror* the selection, click the **Mirror** icon and specify the mirror plane.

**Note:**

- Be sure to check your member orientations after performing the copy. RISA-3D will apply the default member orientation if you do not explicitly define it. You may use a **K joint** to help maintain orientations. See [Defining Member Orientation](#) for more information.
- Use the "@" symbol to specify multiple equal increments. For example specifying "3@10" will give you 3 copies at 10 units apart.
- You can undo any mistakes by clicking the **Undo** icon in the Quick Access toolbar.

Global Copy

The Global icon lets you copy the selected items by entering in increments in any or all of the global directions. The selected items will copy the increment distance(s) that you have entered.



Use the "@" symbol to specify multiple equal increments. For example specifying "3@10" will give you 3 copies at 10 units apart.

Properties

Copy Global ^

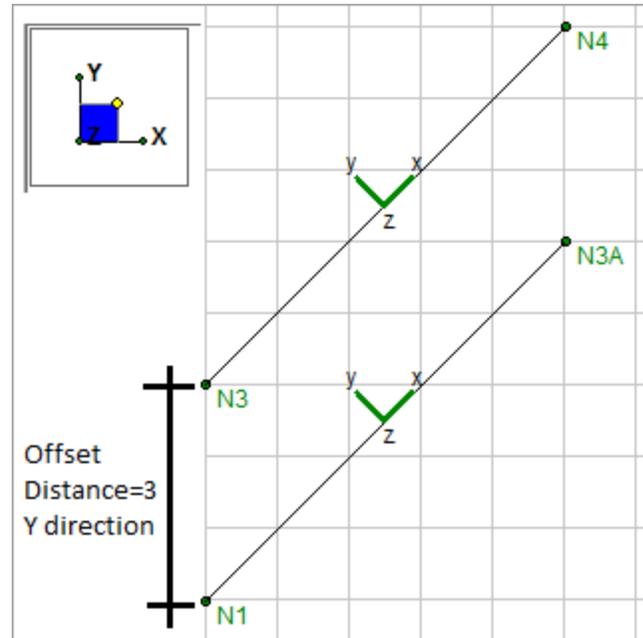
X Increment, ft	
Y Increment, ft	
Z Increment, ft	
Diagonal	<input type="checkbox"/>
Copy Loads	<input type="checkbox"/>
Connect Bays	<input type="checkbox"/>
Bay Fixity	Fixed End Struts v

Click to Apply

Apply to Selected

Keep Tool Active

For example, if you indicate 3 feet in the Y direction then click on the member, it will be copied 3 feet in the positive global Y direction.



Select the **Diagonal** checkbox to combine the orthogonal increments into a single resultant increment vector to copy the elements in a direction other than the three global orthogonal directions.

Checking the **Copy Loads?** box causes ALL loads associated with the original selected model elements to be copied to the corresponding newly created model elements.

Checking the **Connect Bays?** box causes new members to be generated that connect all member end nodes of the originally selected model elements with the corresponding member end nodes in the newly created model elements. You can also indicate whether the new member end releases are to be **Fixed** or **Pinned** by selecting the corresponding option from the **Bay Fixity** section.

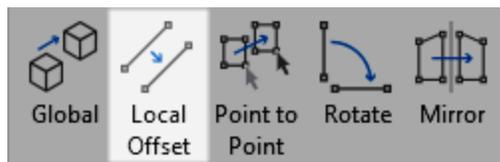


Note:

- Only member end nodes will be inter-connected. The corner nodes of Plates and Solids will NOT be inter-connected.
- Member end nodes that have a boundary condition associated with them will NOT be inter-connected.

Local Offset Copy

The Local Offset icon lets you copy selected members at an offset simply by entering the offset distance and then clicking on the member to be offset.



Properties

Copy Local Offset ^

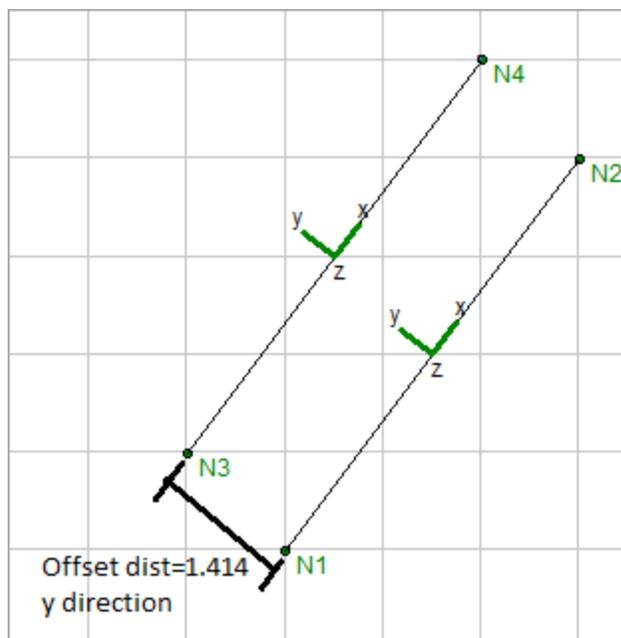
Direction x v

Increment, ft 0

Click to Apply
Apply to Selected

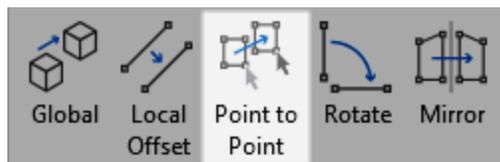
Keep Tool Active

If you indicate 3 feet in the local y direction then click the on the member, it will be copied 1.414 feet in the positive local y direction.



Point to Point Copy

The Point To Point icon lets you copy the selected items by clicking on any two points. The selected items will copy the distance between the two points in the direction of the first to second click.



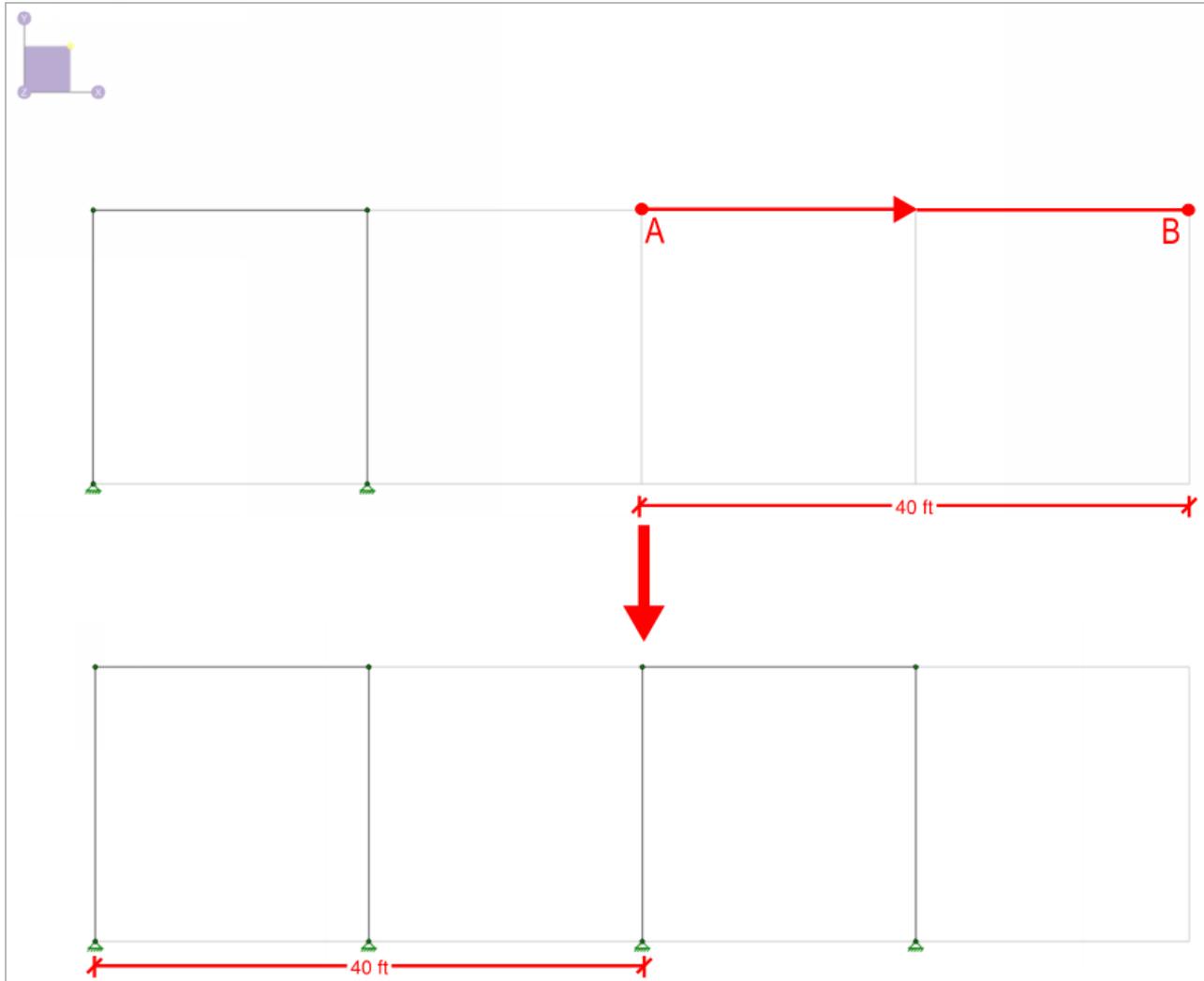
Properties

Copy Point to Point ^

Copy Loads	<input type="checkbox"/>
Connect Bays	<input type="checkbox"/>
Bay Fixity	Fixed End Struts v

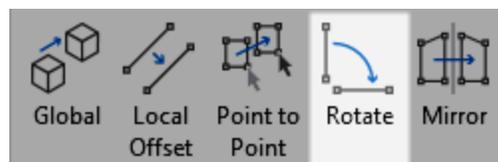
Keep Tool Active

For example, if you click two points: A then B in the model below, the frame will copy 40 feet in the positive X direction.



Rotate Copy

You can copy the selected part of the model by rotating the copies about an axis. Simply enter which axis is to be rotated about, the rotation angle, the number of increments along the rotation angle, and the location of the polar origin (the point rotated about). Only two of the three polar coordinate values are used, the value corresponding to the axis of rotation is not used.



Instead of specifying coordinates for a polar origin, it may instead be chosen by clicking in the model view. Select **Click to Locate**, specify the appropriate **Rotation Axis** and **Rotation Angle**, then click

Click to Apply or **Apply to Selected**. The cursor shows a cross-hair, with which a polar origin can be specified by left-clicking on a Joint or snap point.

For example, suppose you wish to copy the selected part of your model in six 30° increments (for a total of 180°) about the Y-axis. You would enter "Y" as the axis of rotation, "180" as the rotation angle and "6" as the number of increments. For the polar origin, enter the X and Z coordinates of where you want the Y-axis of rotation to pass through the XZ plane.

Checking the **Connect Bays?** box causes new members to be generated that connect all member end nodes of the originally selected model elements with the corresponding member end nodes in the newly created model elements. You can also indicate whether the new member end releases are to be **Fixed** or **Pinned** by selecting the corresponding option from the **Bay Fixity** drop down menu.



Note:

- Only member end nodes will be inter-connected. The corner joints of Plates and Solids will NOT be inter-connected.
- Member end nodes that have a boundary condition associated with them will NOT be inter-connected.

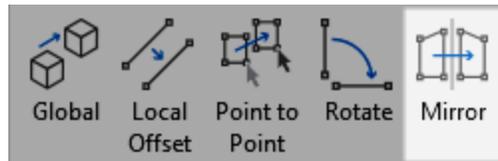
Checking the **Adjust Local Axis** box causes the local axis of each member to be rotated with the newly created model elements. Otherwise, the member local axes will retain their original orientation

with respect to the global axis. The local axes of Plates and Solids will ALWAYS be rotated with the newly created model elements.

Checking the **Copy Loads?** box causes ALL loads associated with the original selected model elements to be copied to the corresponding newly created model elements.

Mirror Copy

You can mirror selected parts of your model.



Enter the global plane that you want to mirror about and enter a location along the normal axis to move the mirror plane location. If the mirror plane location is left blank, the mirror plane is placed at the origin.

Properties	
Copy Mirror ^	
Plane	XY v
Z Location, ft	0
Copy Loads	<input type="checkbox"/>
Connect Bays	<input type="checkbox"/>
Bay Fixity	Fixed End Struts v
Copy Labels v	
<input type="button" value="Click to Apply"/> <input type="button" value="Apply to Selected"/>	
<input checked="" type="checkbox"/> Keep Tool Active	

For example, suppose you want to mirror part of your model about the XY plane at a location +3.0 feet from the origin along the Z-axis. You would enter "XY" as the Mirror Plane and enter a "3.0" for the mirror plane location. (If the mirror plane is XY, the normal axis is the Z-axis).

Checking the **Connect Bays?** box causes new members to be generated that connect all member end nodes of the originally selected model elements with the corresponding member end nodes in the newly created model elements. You can also indicate whether the new member end releases are to be **Fixed** or **Pinned** by selecting the corresponding option from the **Bay Fixity** drop down menu.

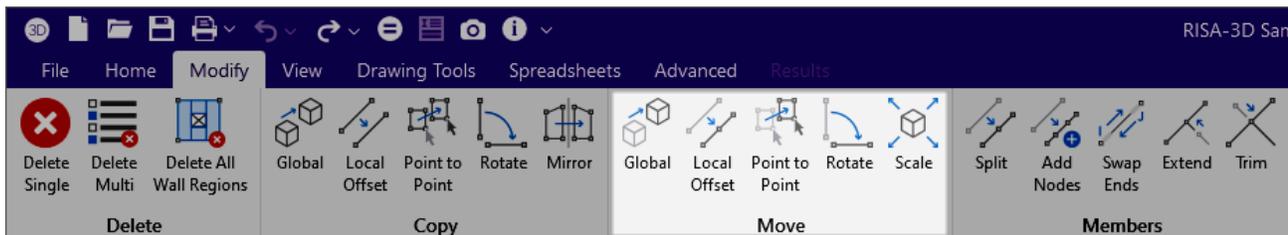
**Note:**

- Only member end nodes will be inter-connected. The corner nodes of Plates and Solids will NOT be inter-connected.
- Member end nodes that have a boundary condition associated with them will NOT be inter-connected.

Checking the **Copy Loads?** box causes ALL loads associated with the original selected model elements to be copied to the corresponding newly created model elements.

Move / Rotate Model Elements

The model elements move options can be found on the Modify ribbon in the Move section.



Move or Rotate Selected Items

To move or rotate selected items:

1. Open a model view if one is not already open.
 - a. Go to the **View** ribbon.
 - b. Click the **Open 3D Views** icon.
2. Select the items you want to move by clicking on them or clicking and dragging over them.
3. To *linearly* move the selection:
 - a. Go to the **Modify** ribbon.
 - b. Under the **Move** section, choose one of the following options:
 - **Global** (See [Global Move](#) for more information.)
 - **Local Offset** (See [Local Offset Move](#) for more information.)
4. Click the **Point to Point** icon, to move the selection based upon two selected points for direction and distance.
See [Point to Point Move](#) for additional information.
5. Click the **Rotate** icon and specify the axis and angle, to move the selection in a polar fashion.
See [Rotate Move](#) for additional information.

6. Click the **Scale** button and enter a scaling factor, to scale the selection.

See the [Scale Elements](#) section for more detailed information.

**Note:**

- Be sure to check your member orientations after performing the copy. RISA-3D will apply the default member orientation if you do not explicitly define it. You may use a **K joint** to help maintain orientations. See [Defining Member Orientation](#) for more information.
- You can undo any mistakes by clicking the **Undo** button in the Quick Access toolbar.

Global Move

You can move the selected part of the model. Just enter the desired translation distances in the global axes directions.

Properties

Move Global ^

X Increment, ft

Y Increment, ft

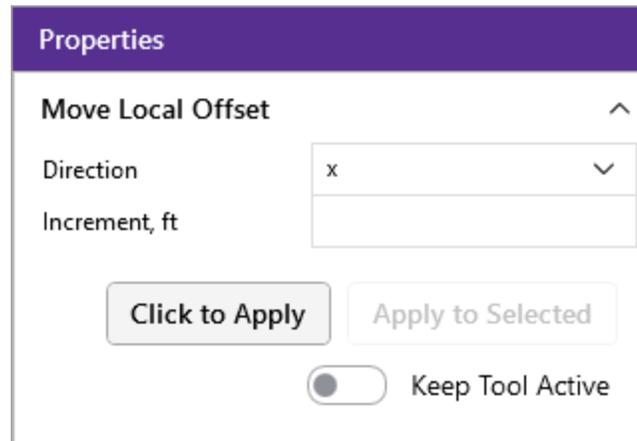
Z Increment, ft

Click to Apply Apply to Selected

Keep Tool Active

Local Offset Move

You can offset elements in the model by choosing a local axes direction, specifying an increment, then clicking the element of the mode you wish to move.

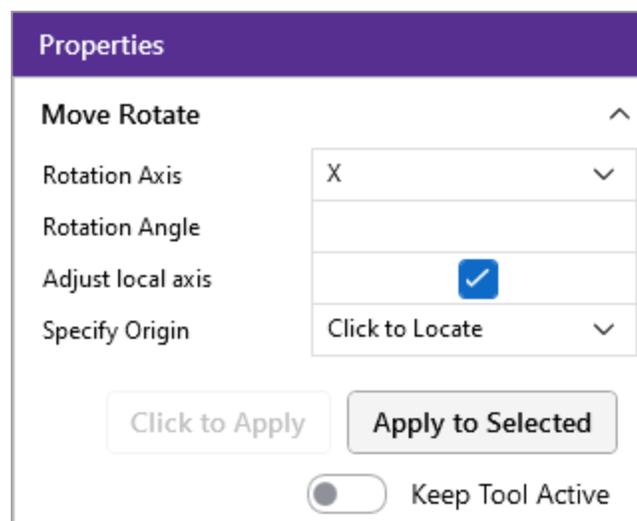


Point to Point Move

You can move elements in the model by clicking on any two points. The selected items will move the distance and direction based upon the vector drawn from your first to your second points.

Rotate Move

To rotate the selected parts of the model about an axis enter an axis of rotation, the desired rotation angle, and the coordinates of the polar origin (the point rotated about). Only two of the three polar coordinate values are used, the value corresponding to the axis of rotation is ignored. Instead of specifying coordinates for a polar origin, it may instead be chosen by clicking in the model view. Select **Click to Locate**, specify the appropriate Rotation Axis and Rotation Angle, then click **Click to Apply** or **Apply to Selected**. The cursor will show a finger pointer, with which a polar origin may be specified by left-clicking on a node or snap point.



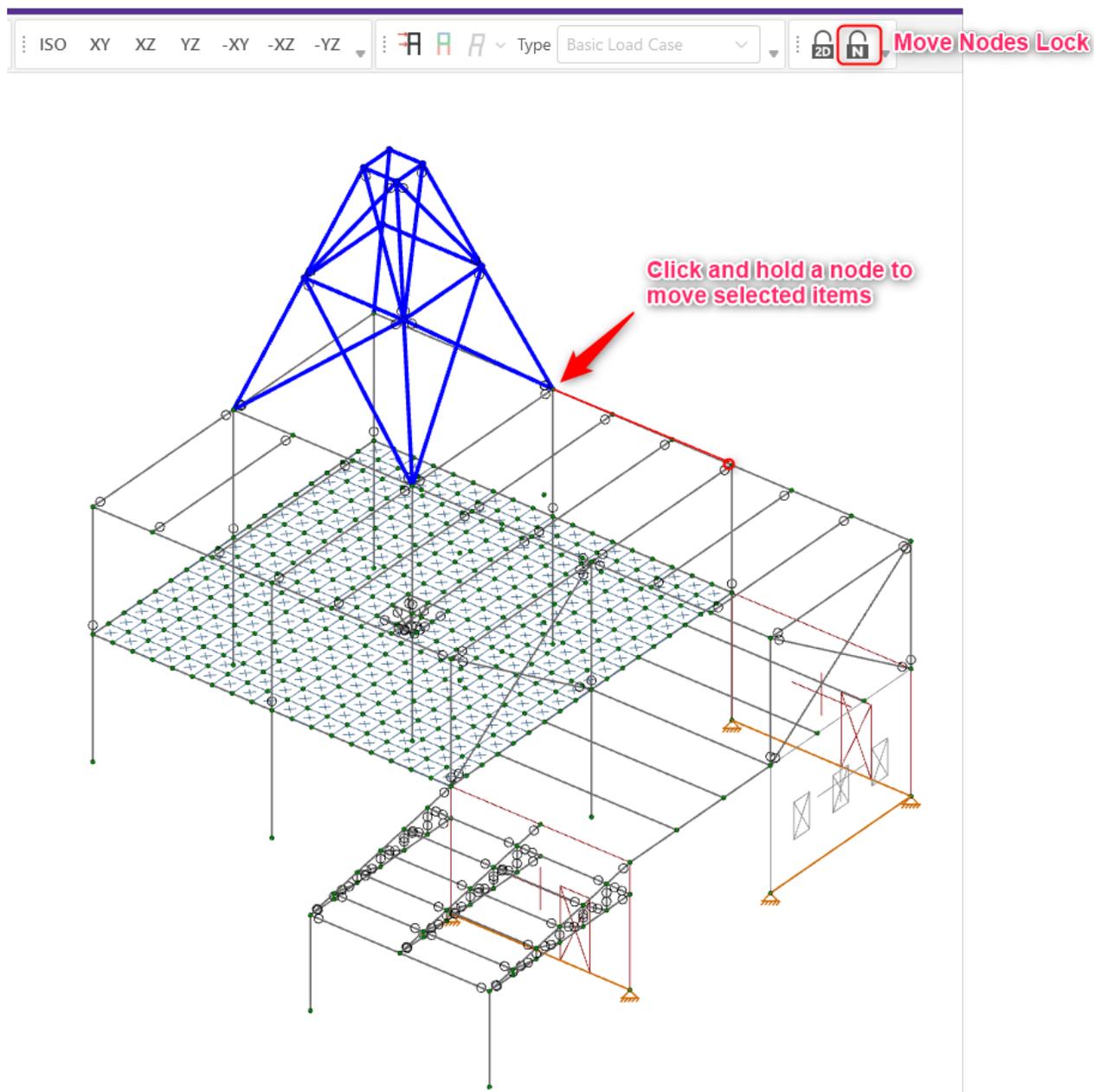
For example, say you wish to rotate the selected part of your model 90° about the Z-axis. You would enter "Z" as the axis of rotation and "90" as the rotation angle. Under specify origin, select **Coordinates** or **Click to Locate**. If **Coordinates** enter the X and Y coordinates of where you want the Z-axis of rotation to pass through the XY plane. If **Click to Locate** is selected, press **Apply to Selected** then click a location in the 3D view.

Checking the **Adjust Local Axis** box will cause the local axis of each member to be rotated along with the model elements. Otherwise, the member local axes will retain their original orientation with respect to the global axis. The local axes of Plates and Solids will ALWAYS be rotated along with the model elements.

Drag and Move in 3D View

You can quickly move selected elements using the drag-and-move feature. To turn this feature on or off, click the "Move Nodes Lock" button  on the ribbon toolbar. Unlocking this will enable the drag-and-move feature.

To use this feature, select the elements you wish to move, then click and hold on a reference node (similar to a Point-to-Point move). A red line will appear in the 3D view, indicating that you are in drag-and-move mode. Next, hover over the destination node and release; the selected elements will move to the new location. If you want to move the elements in the Global X, Y, or Z directions, you can hold the Shift key while performing the drag-and-move actions.



Scale Elements

The scale feature lets you change the size of selected items. Selected nodes, members and plates are all affected when the scaling is applied. Loads are not scaled except distributed and surface loads that are a function of the size of the element.

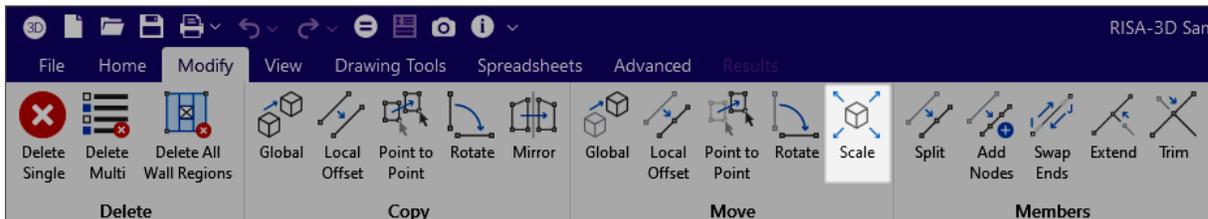


Note: If you wish to scale in a direction other than a global direction you can rotate the model to a global direction, scale it, and rotate it back.

Scale Selected Items

To scale selected items:

1. Open a model view if one is not already open.
 - a. Go to the **View** ribbon.
 - b. Click the **Open 3D Views** icon.
2. Select the items you want to scale by clicking on them or clicking and dragging over them.
3. Go to the **Modify** ribbon.



4. Click the **Scale** icon.
The Properties panel displays the Move Scale options.

Properties

Move Scale ^

X Factor	3
Y Factor	2
Z Factor	4
Specify Origin	Coordinates v
X, ft	
Y, ft	
Z, ft	

Click to Apply

Apply to Selected

Keep Tool Active

5. Change the scale factors you want.

6. Click the **Specify Origin** arrow and choose an option.
 - **Coordinates:** If you choose this option, enter an origin about which to scale and a scale factor to apply to each global direction.
 - **Click to Locate:** If you choose this option, click a node to define the origin. The origin is the point that remains stationary during the scaling.

The factors are applied in each direction to adjust the node coordinates, member lengths and plate sizing. The factors work so that a factor of one has no affect, a factor of two will double the size of the item, and so on.

When you scale items the node coordinates that define the items are moved. For this reason, the scaling is applied to all selected items plus unselected nodes of selected items.

For example, if you have selected members to scale but have unselected some of the end nodes, the nodes are scaled anyway since this is the only way to scale the member.

7. Click **Click to Apply** or **Apply to Selected** button.



Note: You may undo any mistakes by clicking the  **Undo** button in the Quick Access toolbar.

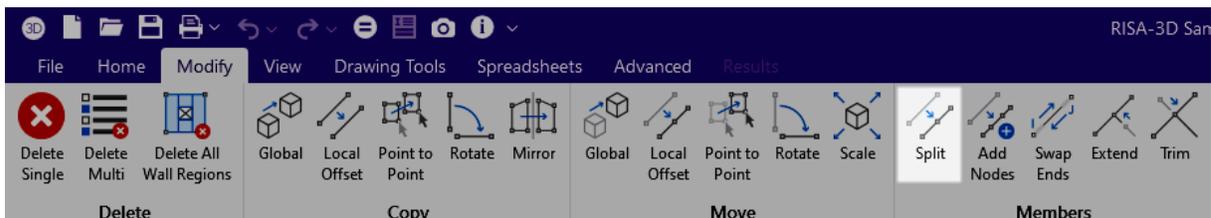
Split Members

You can split members into a number of equal length pieces or you may split into two members at a specified location from the I joint. You may specify the split location as a length or a percentage of length. For example specifying **%50** will split the members at the midpoints.

Split One or More Members

To split one or more members:

1. Open a model view if one is not already open.
 - a. Go to the **View** ribbon.
 - b. Click the **Open 3D Views** icon.
2. Go to the **Modify** ribbon.



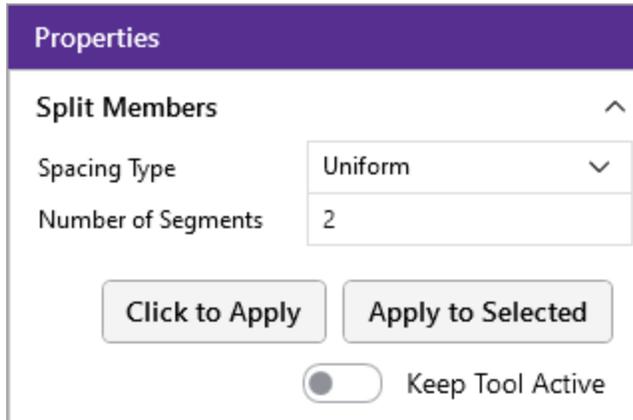
3. Modify a single member at a time or an entire selection of members.

Single member at a time

To modify a single member at a time:

- a. Click on the **Split** icon.

The Properties panel displays the Split Members options.



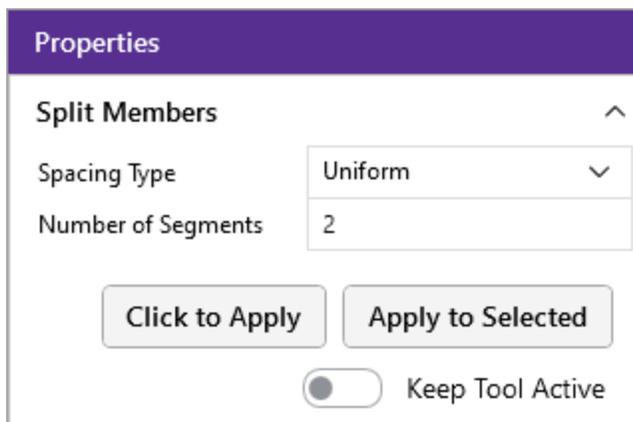
- b. Set the parameters for splitting the member(s).
- c. Click on the **Click to Apply** button.
- d. Click on the member(s) you want to split.

Modify an entire selection of members

To modify an entire selection of members at once:

- a. Click on the members you want to split.
- b. Click on the **Split** icon.

The Properties panel displays the Split Members options.



- c. Set the parameters for splitting the members.
- d. Click the **Apply to Selected** button.



Note: You may undo any mistakes by clicking the  **Undo** button in the Quick Access toolbar.

Add Nodes to Members

Adding Nodes to Members

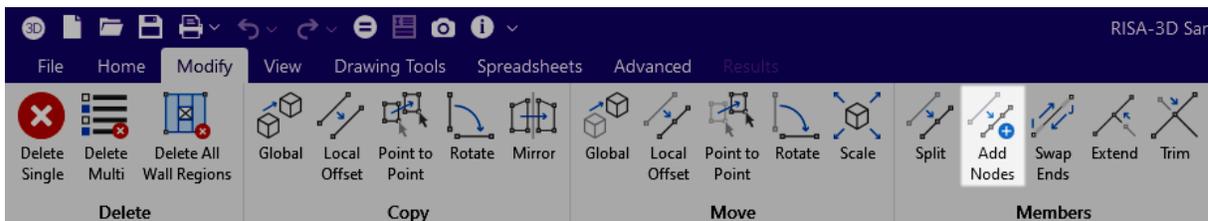
You can add nodes to members at equal length intervals or you may add nodes at specific locations from the I Joint. You can specify the location as a length or a percentage of length. For example specifying **%25** will add nodes at the quarter points of the members.

For Physical members it is likely that you will not want to actually split the member but prefer instead that joints be placed along the member so that you may specify other members, plates, loads or boundary conditions.

Add Nodes

To add nodes to a single member at a time or an entire selection of members:

1. Open a model view if one is not already open.
 - a. Go to the **View** ribbon.
 - b. Click the **Open 3D Views** icon.
2. Go to the **Modify** ribbon.



3. Click the **Add Nodes** icon in the Members section.
4. Set the parameters for adding nodes to the members.
5. Modify members.

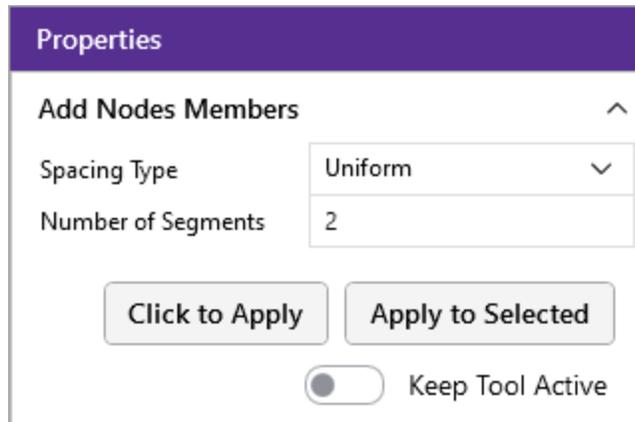
You may choose to modify a single member at a time or an entire selection of members.

Single member at a time

To modify a single member at a time:

- a. Click on the **Add Nodes** icon.

The Properties panel displays the Split Members options.



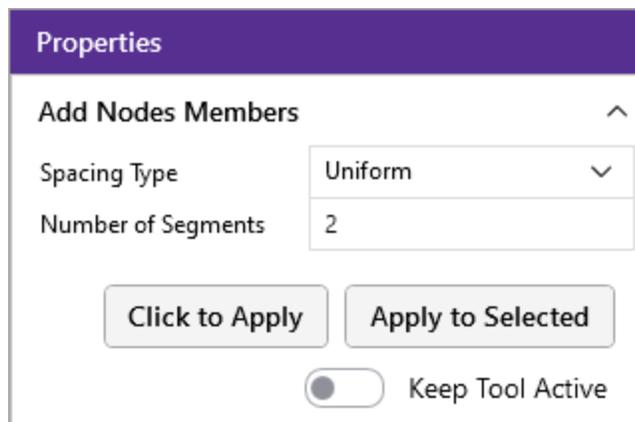
- b. Set the parameters for adding nodes to the member(s).
- c. Click on the **Click to Apply** button.
- d. Click on the member(s) you want to add nodes to.

Modify an entire selection of members

To modify an entire selection of members at once:

- a. Click on the members you want to add nodes to.
- b. Click on the **Add Nodes** icon.

The Properties panel displays the Split Members options.



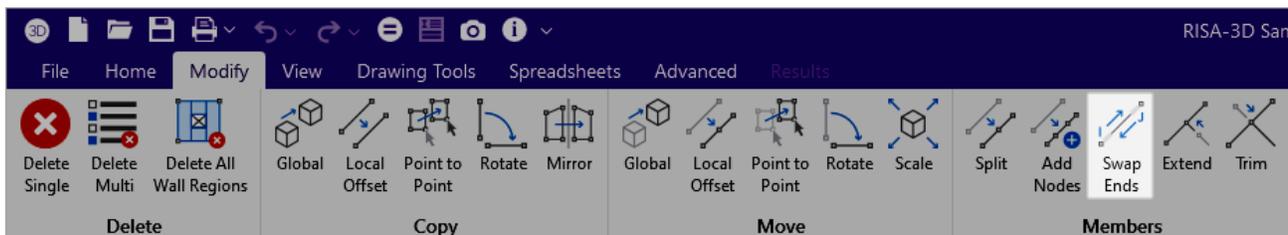
- c. Set the parameters for adding nodes to the members.
- d. Click the **Apply to Selected** button.



Note: You may undo any mistakes by clicking the  **Undo** button in the Quick Access toolbar.

Swap Member Ends

The **Swap Ends** icon lets you swap the start (I-node) and end (J-node) of a member. This can be useful if you are trying to assign connection rules to your members and need to quickly swap columns such that their I-end is at the bottom of the column.



You can also choose to swap the loads with the member ends. If you do not swap the loads, they remain graphically as they are.

Trim / Extend Members

As you modify the model, you may find that you will need to trim or extend a member. The Trim/Extend tool will allow you to modify the ends of any member in relation to another member. This extension is for members only, not walls or plates.



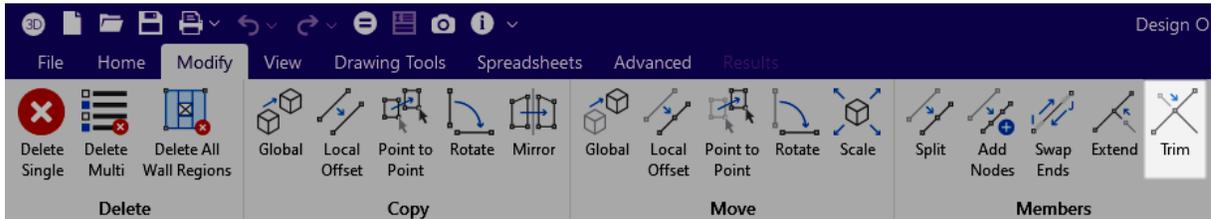
Note:

- This tool is sensitive to the end that you click. Each member has an I and J end. Make sure to click on the member side that you want to modify.
- You need to click on the member NOT the node. This tool modifies the length of the member.

Perform a Trim

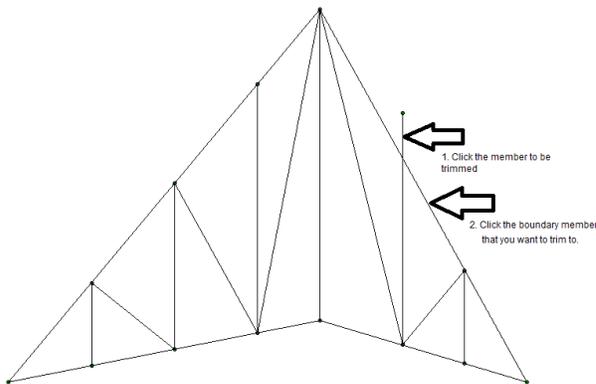
To perform a trim:

1. Open a model view if one is not already open.
 - a. Go to the **View** ribbon.
 - b. Click the **Open 3D Views** icon.
2. Go to the **Modify** ribbon.

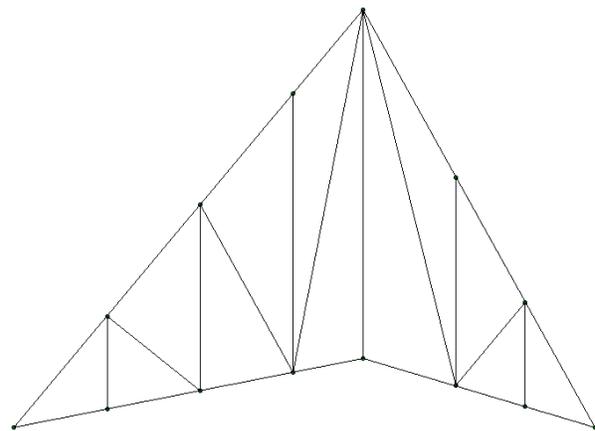


3. Click The **Trim** icon in the Members section.
4. Click on the member end to Trim.
5. Click on the boundary member.

End to Trim and then boundary member



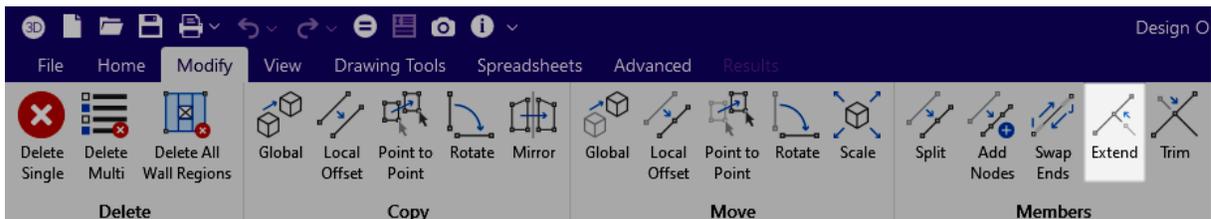
Results from the trim



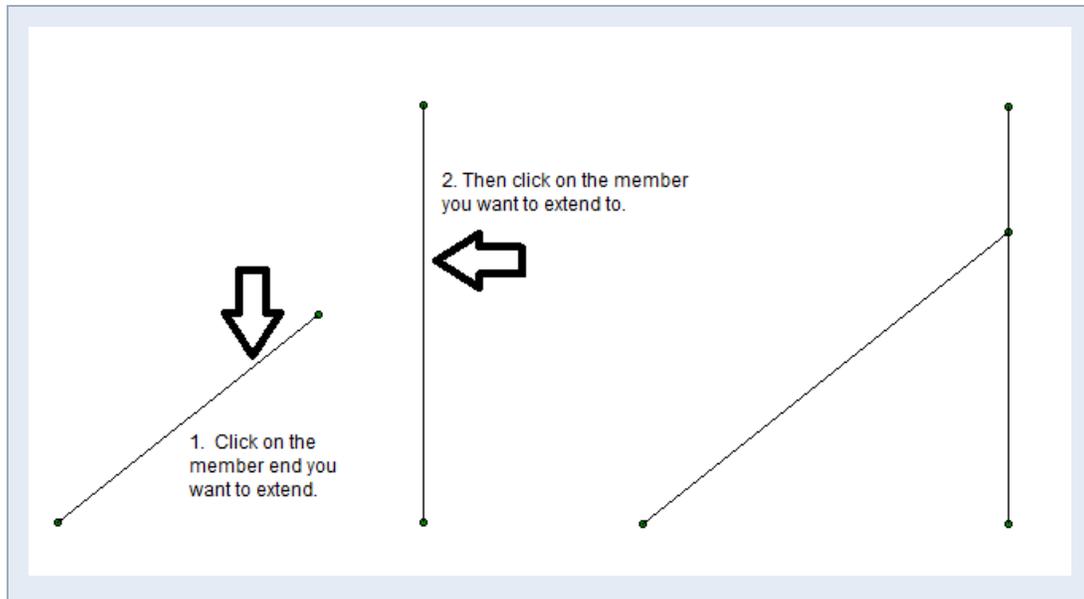
Perform an Extend

To perform an extend:

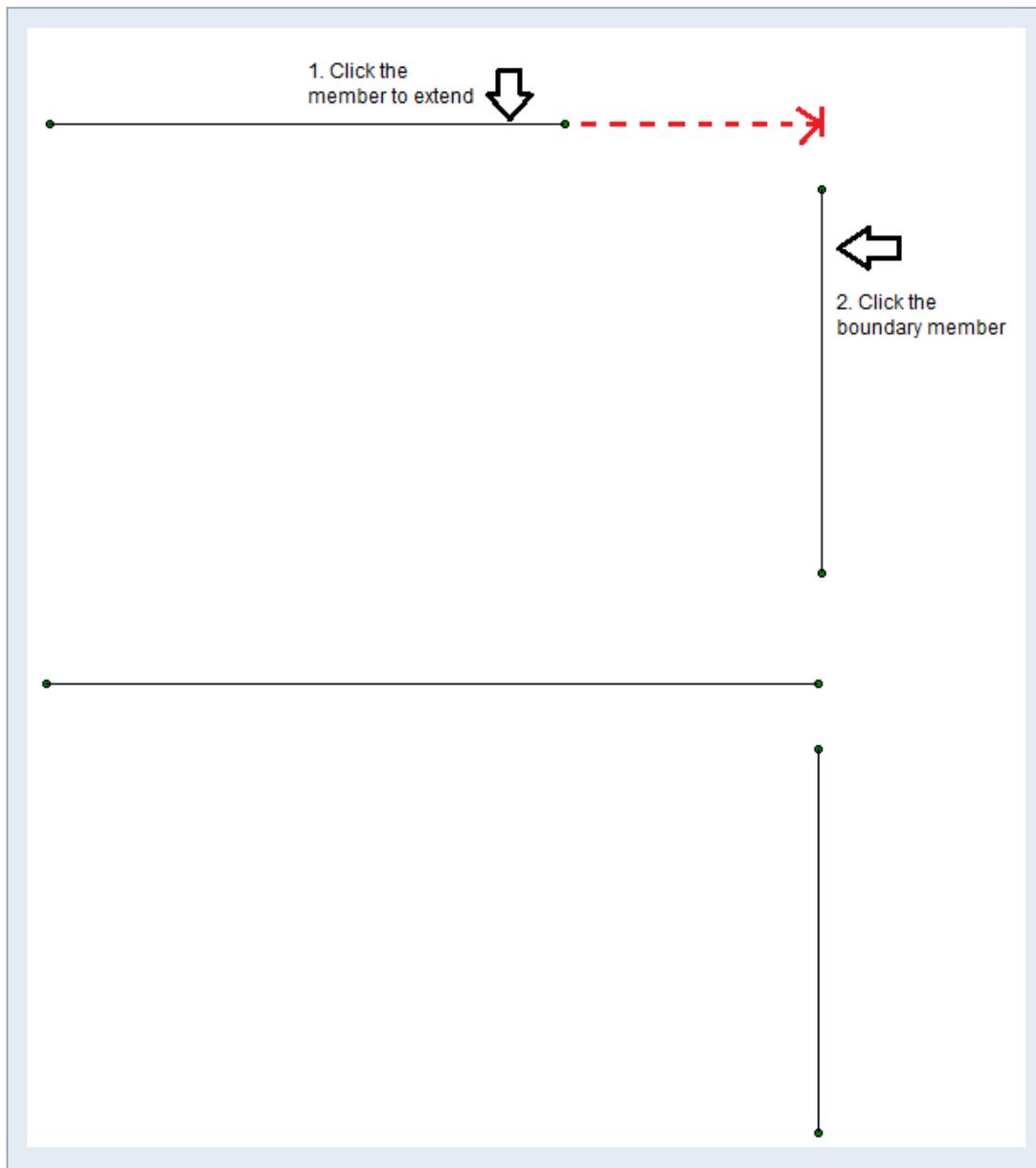
1. Open a model view if one is not already open.
 - a. Go to the **View** ribbon.
 - b. Click the **Open 3D Views** icon.
2. Go to the **Modify** ribbon.



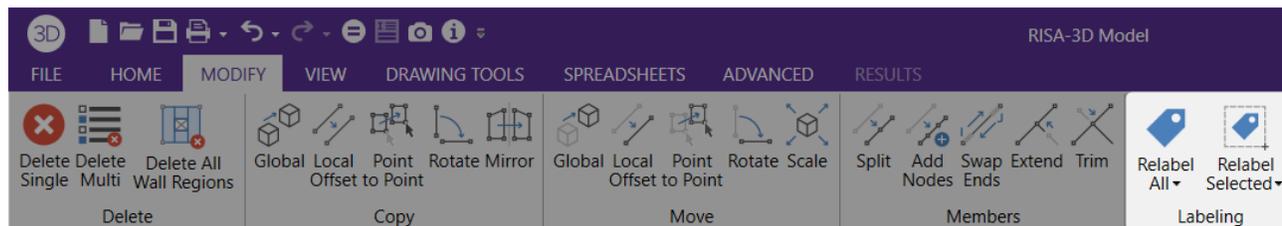
3. Click on the member end to Extend.
4. Click on the boundary member.

Example 1**Example 2**

The Extend tool can also be used to extend a member to intersection of that member's projected location.



Re-Label Selection Items



You can use the **Relabel** icons on the **Modify** ribbon to re-label only selected items or all items. This is useful when you want all members at a certain elevation to have a prefix which denotes the floor on which they are located.

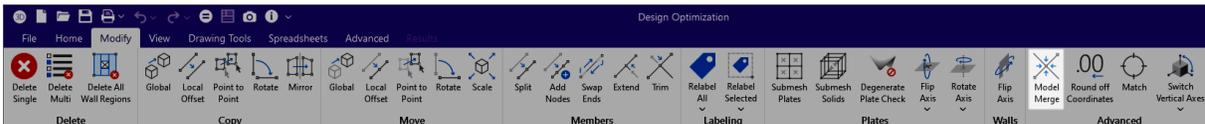
Merge Model Elements

As you build your model, you may find that you need to perform a model merge from time to time. In fact, if you count on doing this, you can generally build your models faster and let the model merge feature do a lot of the work for you. See [Merge Tolerance](#) to learn about inputting the merge tolerance.

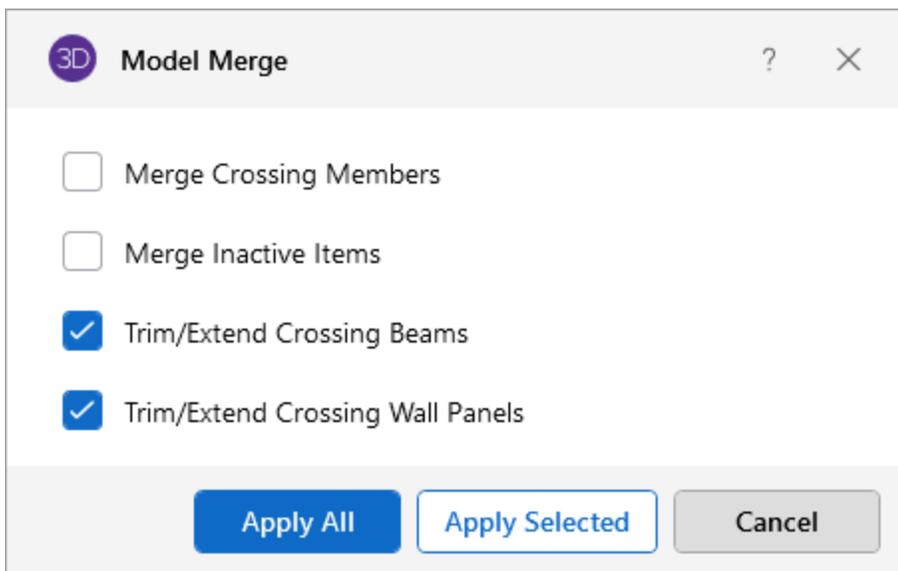
Perform a Model Merge

To perform a model merge:

1. Open a model view if one is not already open.
 - a. Go to the **View** ribbon.
 - b. Click the **Open 3D Views** icon.
2. Go to the **Modify** ribbon.



3. Select the items you wish to merge.
Typically you will want everything to be selected.
4. Click the **Model Merge** icon on the Modify ribbon in the Advanced section.
The Model Merge window opens.



5. Set the parameters for the new merge.
6. Click either **Apply All** or **Apply Selected**, depending on if you are applying the merge to all items or just selected items.

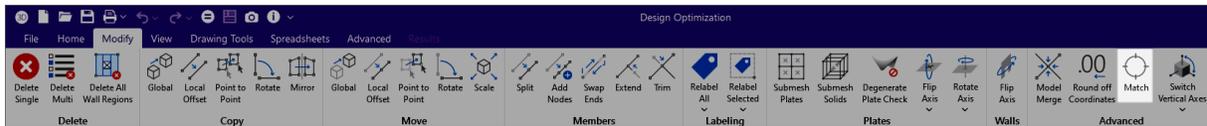
Match

Use the Match tool to match the properties of one element to another element.

Use the Match Tool

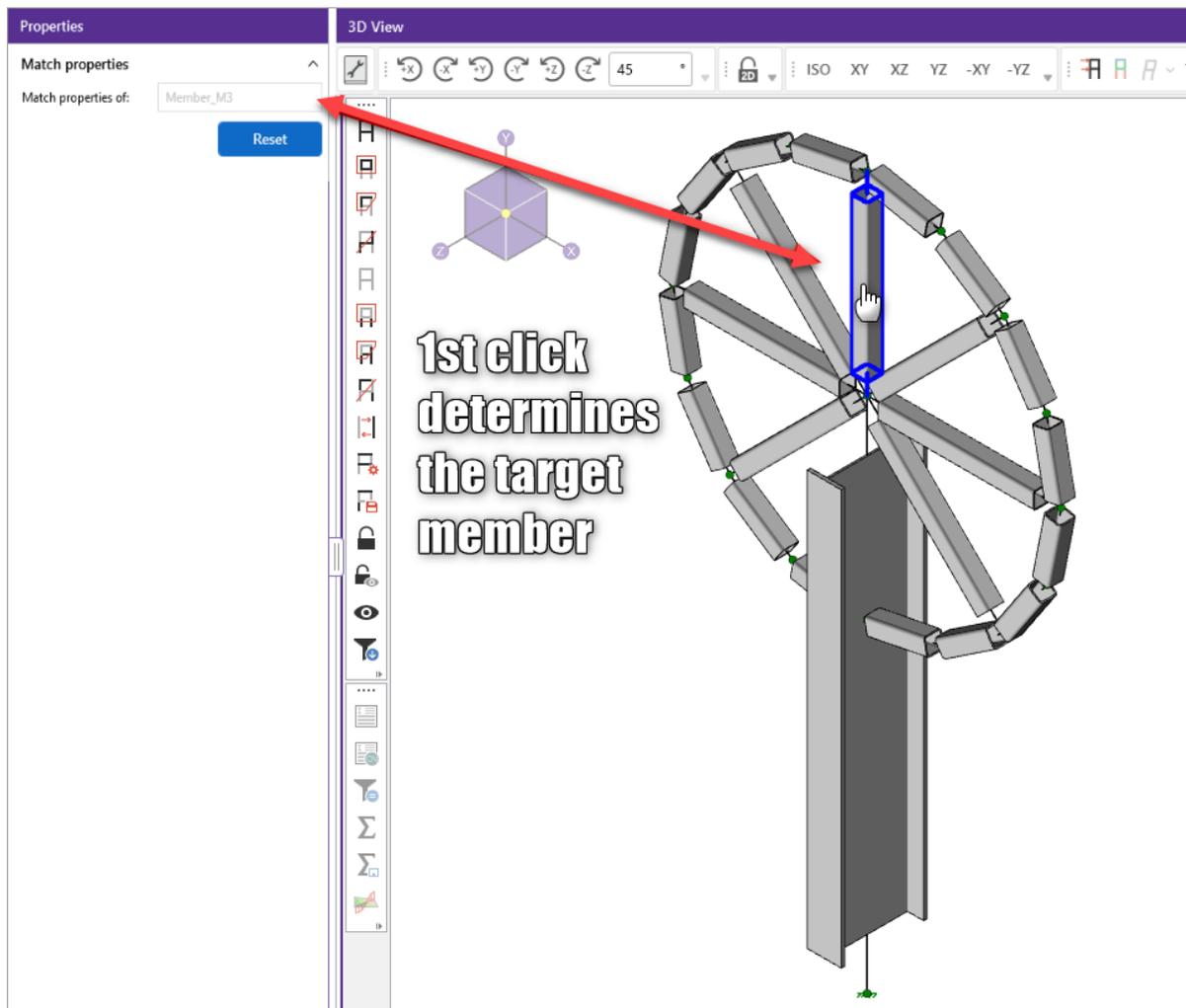
To use the Match tool:

1. Go to the **Modify** ribbon.

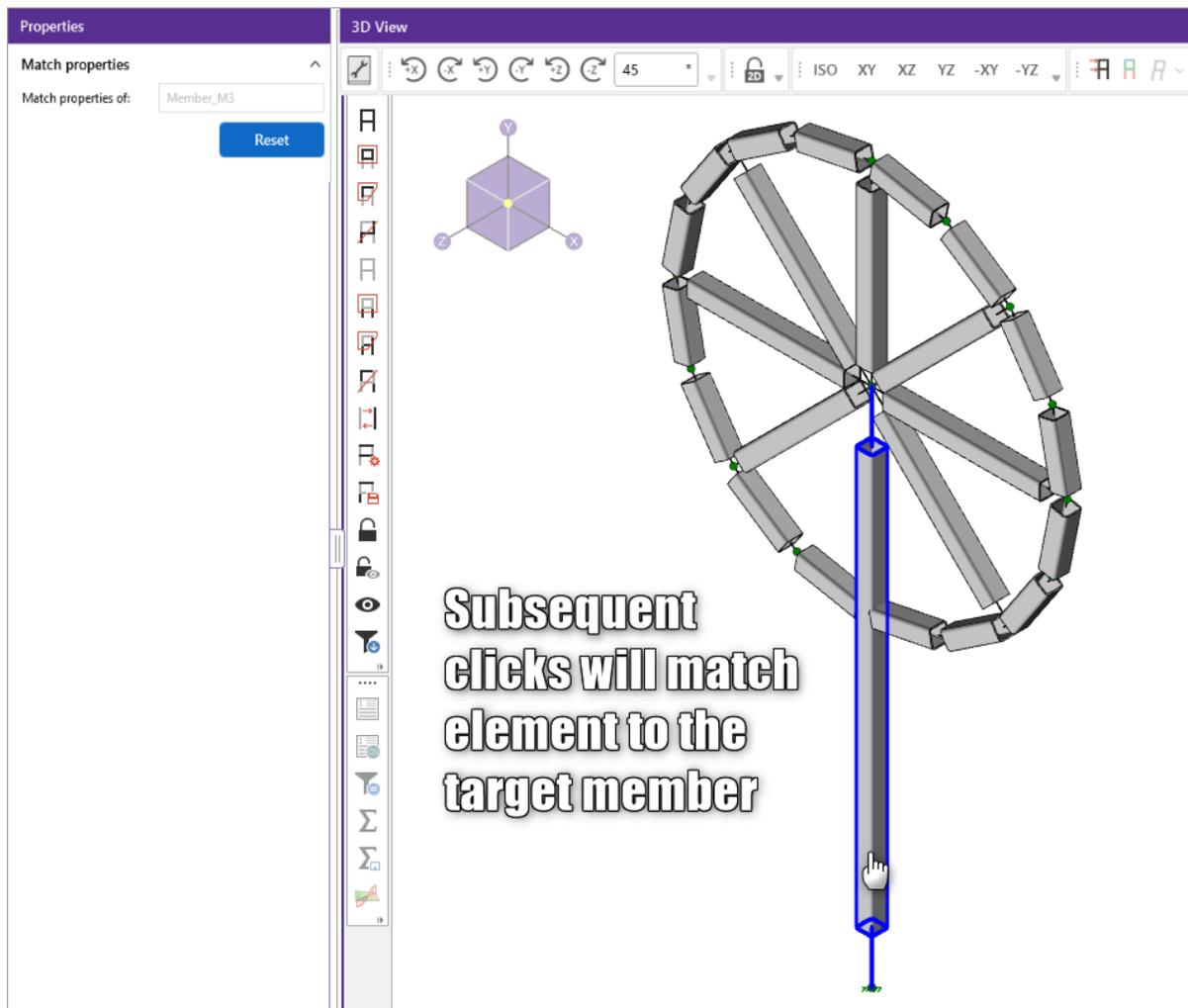


2. Click on the **Match** icon in the Advanced section.
3. Select the target element you would like to copy the properties of.

The label of this element appears in your properties panel in the **Match properties of:** section.



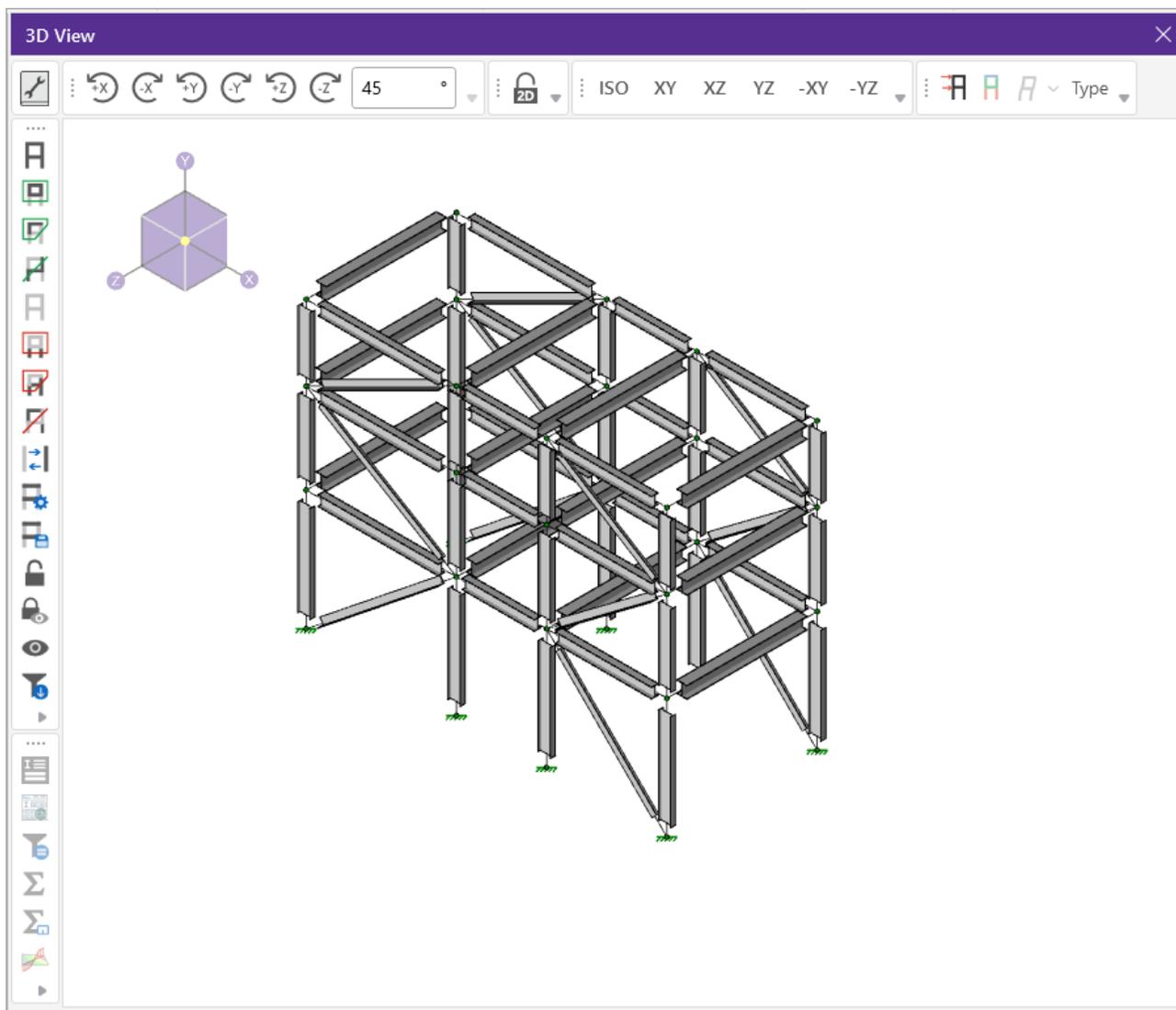
4. Choose the elements that you would like to match the properties to.
These elements will now share all the properties of the element that was selected in step 3.

**Note:**

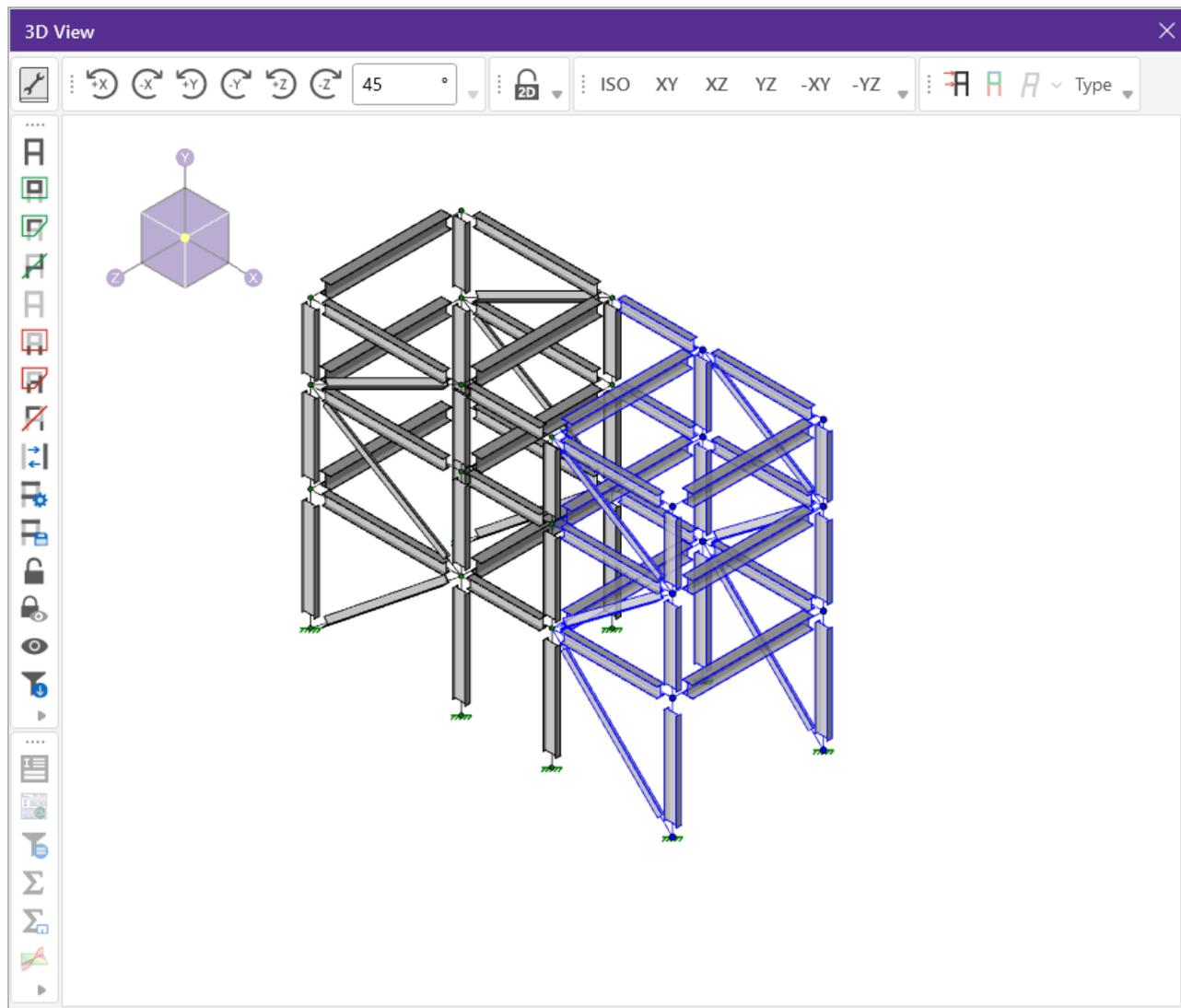
- Subsequent clicks after the first click will continue matching elements to the first member.
- If you would like to create a new target member to match the properties of, click the **Reset** button in the Properties Panel. This makes the next element you select become the new target member to match properties of.
- Currently the program matches all of the properties of the target member. There is no way to match partial properties at this time.

Graphic Selection

You can graphically select items in order to view or modify part of the model or to print part of the results. When used in conjunction with the graphic editing features, the selection feature allows you to quickly model and make modifications to the model. If used with results it allows you to view and print only the input or results that you want.



The elements that you see in the model views have two possible states; selected and unselected. By default, all items are unselected, and therefore fully displayed with no selection highlight. If you select any items, they will be highlighted in a blue shade. To select or unselect an item simply click on it with the left mouse button. You may also use one of the selection tools below to select multiple items.

**Note:**

- As an alternative to using the selection tools, some operations offer a **Click to Apply** option, which lets you modify items by clicking or boxing them with the mouse. This is useful when working with a few items.
- Inactive items can only be selected with **Criteria Selection**.
- Unselected items remain as part of the model for the solution. To remove items from the analysis you need to make them inactive.
- If you choose not to display items by turning them off in the [Model View Settings](#) window, they are removed from the 3D View and will not be able to be selected/unselected until you turn the display back on.

Selection Shortcuts

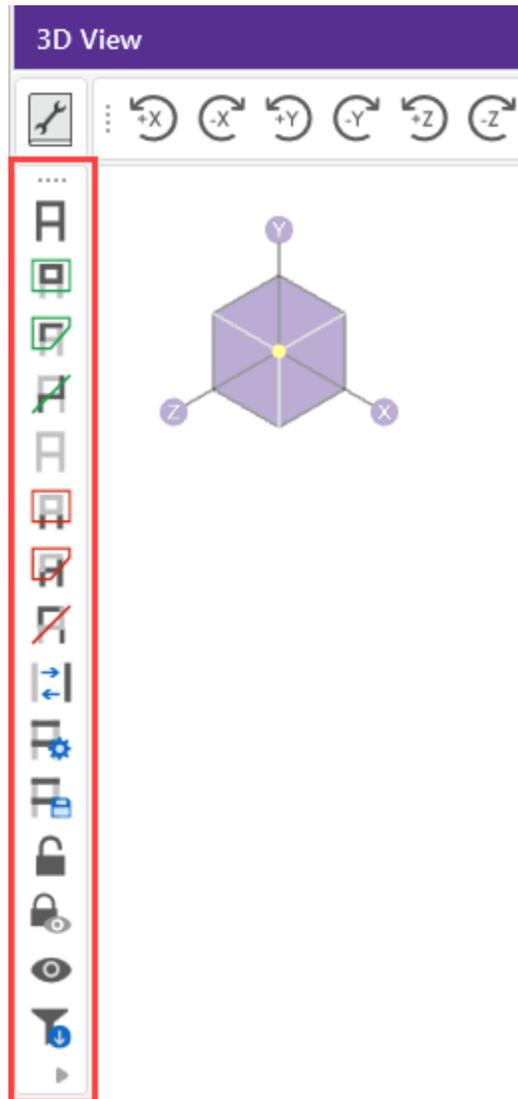
- You can *add elements to a selection state* by pressing **Ctrl** on your keyboard and clicking with the left mouse button.
- When pressing **Ctrl+Left Mouse Click**, a plus sign appears adjacent to the mouse cursor . You can now select a single element to add to the selection state, or you can select multiple elements by boxing them with the mouse.
- Similarly, you can *remove elements from a selection state* by pressing **Shift** on your keyboard and clicking with the left mouse button.
- When pressing **Shift+Left Mouse Click**, a minus sign will appear adjacent to the mouse cursor . You can now deselect a single element from the existing selection state, or you can deselect multiple elements by boxing them with the mouse.

Cycle Selection

- To cycle through elements that are near the cursor, press the **Ctrl** key while **Scrolling the mouse wheel**, either forward or backward. This can be useful if you have a model with elements blocking others that you are interested in selecting and do not want to rotate the model to a position that exposes the desired elements.

Selection Modes

The **Selection Toolbar** is the vertical toolbar located on the left side of the screen (as shown in the following image). This toolbar lets you select joints, members, and plates in model views, and is only active when a model view is active.



Some of the tools are for one-time applications such as **Select All**. Other tools, such as **Box Select**, place you in a selection mode that remains active until you cancel it. The current mode is indicated by the mouse pointer and by the state of the button. While in a selection mode the button stays depressed until you click it again, choose another button, or press the ESC key. You can have more than one model view open and be in different modes in each view.

**Note:**

- There are other types of graphic modes such as **editing** and **viewing** (zooming) modes. The viewing mode overrides the selection mode, which overrides the editing mode. This lets you alter the selected state while you are editing the model. You are returned to the current editing mode when you terminate a selection mode.
- To cancel a selection mode press the ESC key or click the right mouse button.

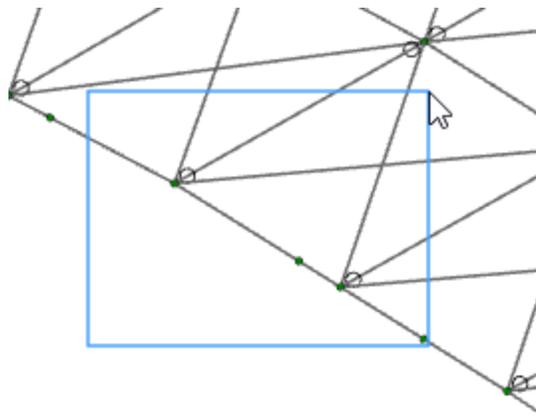
Select All and Unselect All

The **Select All**  and **Unselect All**  tools let you select or unselect all of the active joints, members, and plates in the model.

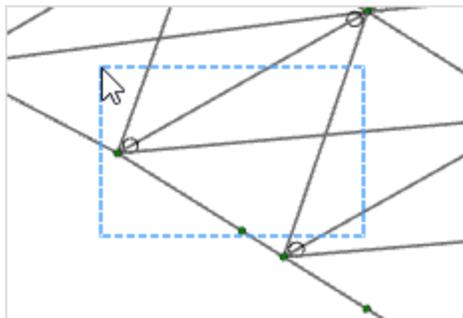
Box Select and Unselect Modes

The **Box Select**  and **Box Unselect**  tools let you draw a box around the items that you wish to select or unselect. Members and plates must be entirely within the box in order for them to be affected.

When selecting members by boxing, dragging your mouse from left to right will select members completely encompassed within the selection. This selection style is denoted by the solid blue line type of the selection box.



Alternatively, dragging your mouse from right to left, will select all elements that you box through. This selection style is denoted with a striped blue line type.



Polygon Select and Unselect

The **Polygon Select**  and **Polygon Unselect**  tools let you draw a polygon around the items that you wish to select or unselect. Members and plates must be entirely within the polygon in order for them to be affected.

Line Select and Unselect

The **Line Select**  and **Line Unselect**  tools let you draw a line through the items that you wish to select or unselect. Any element the line crosses will be affected. This is useful when choosing items between other items such as columns between floors.

Invert Selections

The **Invert Selection**  button is used to invert the selected state of the model. When clicked, all selected items are made unselected and all previously unselected items are made selected. For example, this can be very useful when many items are to be selected in a large model. Simply click on the undesired items to select them, then click the **Invert Selection** button to make them unselected and everything else selected.

Criteria Selections

The **Criteria Selection**  button lets you select items based on a wide range of criteria such as coordinates, labels and many other conditions. When the **Criteria Select** button is clicked, you will be presented with the Criteria Selection Dialog with options grouped by tabs across the top. Each tab represents groups of criteria that you may use to refine your selection.

The options are numerous, making it easy to quickly achieve complicated selections. This is a powerful tool so it is worth taking the time to experiment with the options so that you will know how to use it to your advantage. The various tabs in the window are described in the sections that follow.

Nodes

The screenshot shows the 'Select Elements by Property' dialog box with the 'Nodes' tab selected. The dialog has a title bar with a 3D icon, a question mark, and a close button. Below the title bar is a toolbar with icons for Nodes, Members, Plates, Wall Panels, Solids, Loads, and Coordinates. The 'Nodes' tab is active. Under 'Select by:', there are several options with checkboxes and input fields or dropdown menus:

- Node Label: N10 to 15
- X Translation: Free
- Y Translation: Free
- Z Translation: Free
- X Rotation: Free
- Y Rotation: Free
- Z Rotation: Free
- Diaphragm: Detached From

At the bottom of the dialog, there are three checkboxes:

- Overwrite Lock
- Overwrite previous selection
- Close dialog after Select/Unselect

At the bottom of the dialog, there are four buttons: 'Clear All', 'Select', 'Unselect', and 'Close'.

You can **Select** or **Unselect** joints as follows.

- **Node Label** – Specify a range of node labels. Specifying only one label selects just that one node.
- **Boundary Codes** – Specify boundary condition criteria in all six degrees of freedom.

- **Diaphragm (Primary Node or Detach From)** – When checked, and you’ve chosen (from the dropdown menu) for either **Primary Node** or **Detach From**, it lets you select the nodes that currently define diaphragms or nodes that currently have their “Detach From Diaphragms” flag set.
- **Overwrite Lock** - When checked, it applies the specified criteria to be selected or unselected regardless of any part of the model being locked.
- **Overwrite previous selection** - When checked, it overwrites the previous selection shown in the 3D View.



Note: If no criteria are specified, all the nodes will be selected/unselected.

Members

3D Select Elements by Property
? X


Nodes


Members


Plates


Wall Panels


Solids


Loads


Coordinates

Select by:

Member Label to

Parallel to Member

Parallel to Axis

Section Set/Shape

Design List

Materials

Member Type

Design Rule

Function

Seismic Design Rule

Tension/Comp

Connection Rule

Inactive (makes Active)

Overwrite Lock

Overwrite previous selection

Close dialog after Select/Unselect

Clear All

Select

Unselect

Close

You can **Select** or **Unselect** members based on the following criteria.

- **Member Label** – Specify a range of member labels. If only one label is entered, only that one member will be selected.
- **Parallel** – Select members that are parallel to a certain member, whose label you enter, or parallel to a global axis.

- **Member Properties** – Specify a Section Set/Shape, Design List, Materials, Member Type, Design Rule, Function, Seismic Design Rule, Tension/Compression-Only, or Connection Rule to be applied when selecting members.
- **Overwrite Lock** – When checked, it applies the specified criteria to be selected or unselected regardless of any part of the model being locked.
- **Overwrite previous selection** – When checked, it overwrites the previous selection shown in the 3D View.



Note: If no criteria are specified, all the members will be selected/unselected.

Plates

Select Elements by Property

Nodes Members **Plates** Wall Panels Solids Loads Coordinates

Select by:

Plates Label to

Parallel to Plate

Parallel to Plane XY ▾

Thickness Range in to in

Materials N/A ▾

Inactive (makes Active)

Plane Stress

Overwrite Lock

Overwrite previous selection

Close dialog after Select/Unselect

Clear All Select Unselect Close

You can **Select** or **Unselect** plates based on the following criteria.

- **Plate Label** – Specify a range of plate labels. If only one label is entered, only that one plate will be selected.
- **Parallel** – Select plates that are parallel to certain plate, whose label you enter, or parallel to a global plane.

- **Plate Properties** – Specify a minimum and maximum Thickness and Material to be applied when selecting plates. You may also select only the [plane stress](#) plates. You also may select plates that are currently inactive. Using this criterion will make these plates active. To make them inactive once again you can then use **ModifyPlates**.
- **Overwrite Lock** – When checked, it applies the specified criteria to be selected or unselected regardless of any part of the model being locked.
- **Overwrite previous selection** – When checked, it overwrites the previous selection shown in the 3D View.



Note: If no criteria are specified, all the plates will be selected/unselected.

Wall Panels

The screenshot shows the 'Select Elements by Property' dialog box with the 'Wall Panels' tab selected. The dialog has a title bar with a 3D icon, a question mark, and a close button. Below the title bar is a toolbar with icons for Nodes, Members, Plates, Wall Panels (selected), Solids, Loads, and Coordinates. The main area is divided into two sections. The top section, labeled 'Select by:', contains six options, each with a checkbox and a corresponding input field or dropdown menu: 'Wall Panel Label' (two text boxes separated by 'to'), 'Parallel to Wall' (one text box), 'Parallel to Plane' (a dropdown menu with 'XY' selected), 'Materials' (a dropdown menu with 'N/A' selected), 'Design Rule' (a dropdown menu with 'N/A' selected), and 'Inactive (makes Active)'. The bottom section contains three options, each with a checkbox: 'Overwrite Lock', 'Overwrite previous selection', and 'Close dialog after Select/Unselect'. At the bottom of the dialog are four buttons: 'Clear All', 'Select' (highlighted in blue), 'Unselect', and 'Close'.

You can **Select** or **Unselect** panels based on the following criteria.

- **Wall Panel Label** – Specify a range of panel labels. If only one label is entered, only that one panel will be selected.
- **Parallel** – Select panels that are parallel to certain panel, whose label you enter, or parallel to a global plane.

- **Wall Panel Properties** – Specify a Material Type (Wood, Masonry, etc) and/or Material Set (Larch, Clay, etc). You also may select panels that are currently inactive. Using this criterion will make these panels active. To make them inactive once again you can then use **ModifyPanels**.
- **Overwrite Lock** – When checked, it applies the specified criteria to be selected or unselected regardless of any part of the model being locked.
- **Overwrite previous selection** – When checked, it overwrites the previous selection shown in the 3D View.



Note: If no criteria are specified, all the panels will be selected/unselected.

Solids

3D Select Elements by Property ? X

Nodes Members Plates Wall Panels **Solids** Loads Coordinates

Select by:

Solid Label to

Materials N/A ▾

Inactive (makes Active)

Overwrite Lock

Overwrite previous selection

Close dialog after Select/Unselect

Clear All Select Unselect Close

You can **Select** or **Unselect** solids based on the following criteria.

Solid Label – You may specify a range of Solid labels. If only one label is entered, only that one Solid will be selected.

Material Properties – Select or unselect solid elements base on their material. You also may select solids that are currently inactive. Using this criterion will make these solids active. To make them inactive once again you can then use **ModifySolids**.

Overwrite Lock – When checked, it applies the specified criteria to be selected or unselected regardless of any part of the model being locked.

Overwrite previous selection – When checked, it overwrites the previous selection shown in the 3D View.

**Note:**

- If no criteria are specified, all the solids will be selected/unselected.
- The **ONLY** way to make an inactive solid active again is to use this criteria select dialog.

Loads

The screenshot shows the 'Select Elements by Property' dialog box with the 'Loads' tab selected. The dialog has a title bar with a 3D icon, a question mark, and a close button. Below the title bar is a toolbar with icons for Nodes, Members, Plates, Wall Panels, Solids, Loads, and Coordinates. The 'Loads' icon is highlighted. The main area contains the following options:

Select by:

Basic Load Case: N/A (dropdown)

Load Type: Point (dropdown)

Take absolute values of loads in the model

Direction: X (dropdown)

Min Magnitude: 0 (text input) kips, kip-ft

Max Magnitude: 0 (text input) kips, kip-ft

Overwrite Lock

Overwrite previous selection

Close dialog after Select/Unselect

At the bottom, there are four buttons: 'Clear All', 'Select' (highlighted in blue), 'Unselect', and 'Close'.

You can **Select** or **Unselect** loads based on the following criteria.

- **Basic Load Case** – Specify a Basic Load Case for which to select or unselect loads.
- **Load Type** – Select or unselect loads based on the load type. You also may select a direction or min/max magnitude.

- **Overwrite Lock** – When checked, it applies the specified criteria to be selected or unselected regardless of any part of the model being locked.
- **Overwrite previous selection** – When checked, it overwrites the previous selection shown in the 3D View.

 **Note:** If no criteria are specified, all the loads will be selected/unselected.

Coordinates

 Select Elements by Property
? X


Nodes


Members


Plates


Wall Panels


Solids


Loads


Coordinates

Coordinate Range:

X to ft

Y to ft

Z to ft

Coordinate Range:

Nodes ▼

Members ▼ Also consider Members Tab

Plates ▼ Also consider Plates Tab

Wall Panels ▼ Also consider Walls Tab

Solids ▼ Also consider Solids Tab

Overwrite Lock

Overwrite previous selection

Close dialog after Select/Unselect

Clear All

Select

Unselect

Close

You can select or unselect elements based on coordinate criteria and combine this with the criteria on the other tabs.

- **Coordinate Range**– Specify minimum and maximum bounds in the global directions. Items within these bounds AND meeting the criteria in the other groups will be selected/unselected.
- **Nodes** – Specify that nodes in the coordinate range be selected or not.
- **Members** – Specify that one or both member end nodes be within the bounds. You may also specify that no members be selected within the range.
- **Plates** – Specify that one or all plate corner nodes be within the bounds. You may also specify that no plates be selected within the range.
- **Wall Panels** – Specify that one or all panel corner nodes be within the bounds. You may also specify that no panel be selected within the range.
- **Solids** – Specify that one or all solid corner nodes be within the bounds. You may also specify that no solid be selected within the range.
- **Also Consider Members/Plates/Walls/Solids Tab** – Lets you include selection criteria from the other pages Members, Plates, Wall Panels, Solids and combine it with the coordinate range specified on this page.



Note: If the **Also Consider ... Tab** checkbox is checked, all criteria will be applied together so that the affected items will meet all of the criteria. For example if you specify a range of node labels and coordinates, only the nodes within the coordinate bounds and within the label range will be selected/unselected.

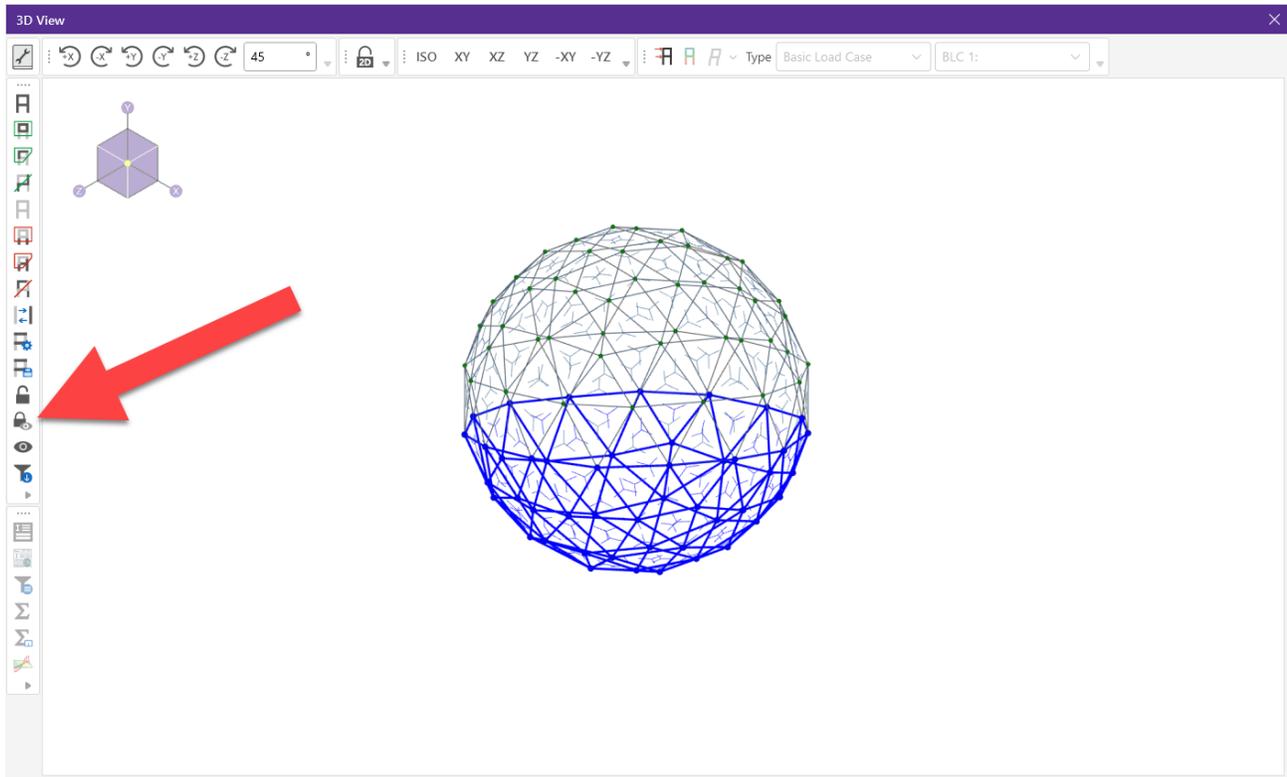
Lock & Dim Lock

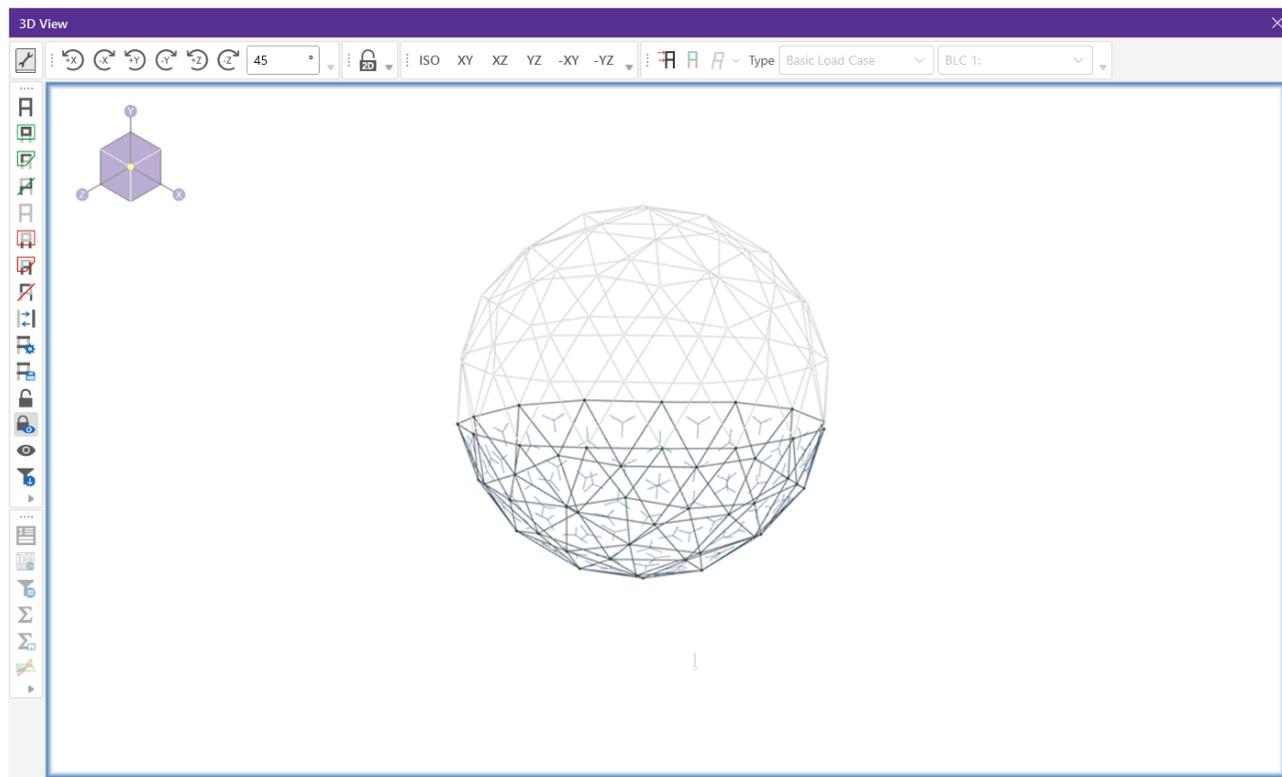
Click the **Lock Unselected**  button to cause all currently unselected items to stay unselected and be visually removed from the current model view. This is useful when you are editing or printing a portion of a model and need to clear the model view of all items not involved. For example in a multistory building when working on one floor you can lock the other floors. Just unselect the entire model, then use box select to select the floor you would like to work on, then press the Lock Unselected button and the rest of the model will stay unselected until you press the button again to turn it off. This tool is also helpful when trying to select an item that is behind other items.

To Unlock the unselected members in the model view, click the **Unlock Unselected**  button and the unselected items that were previously "removed" from the model view will be returned to the view in the unselected state.

Clicking the **Dim Lock**  button causes all currently unselected elements to become visually dim or greyed out. While in the dim lock state, you will not be able to click on these dimmed elements nor will they graphically show any labels or results. This is useful for when you want to keep a portion of

your structure for visual reference only but also have it stay out of your way when modeling or reporting graphical results.



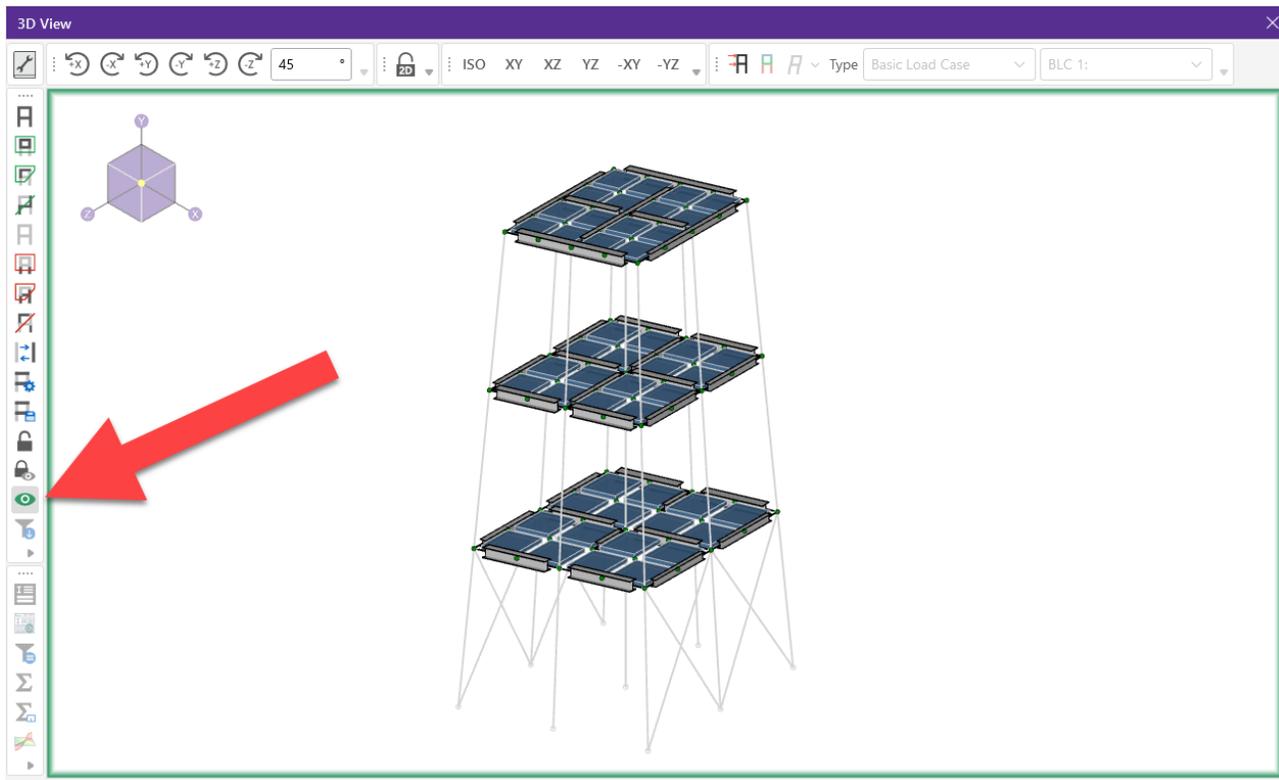


 **Note:** When you are in the dim lock state, you see a blue border around the 3D view panel.

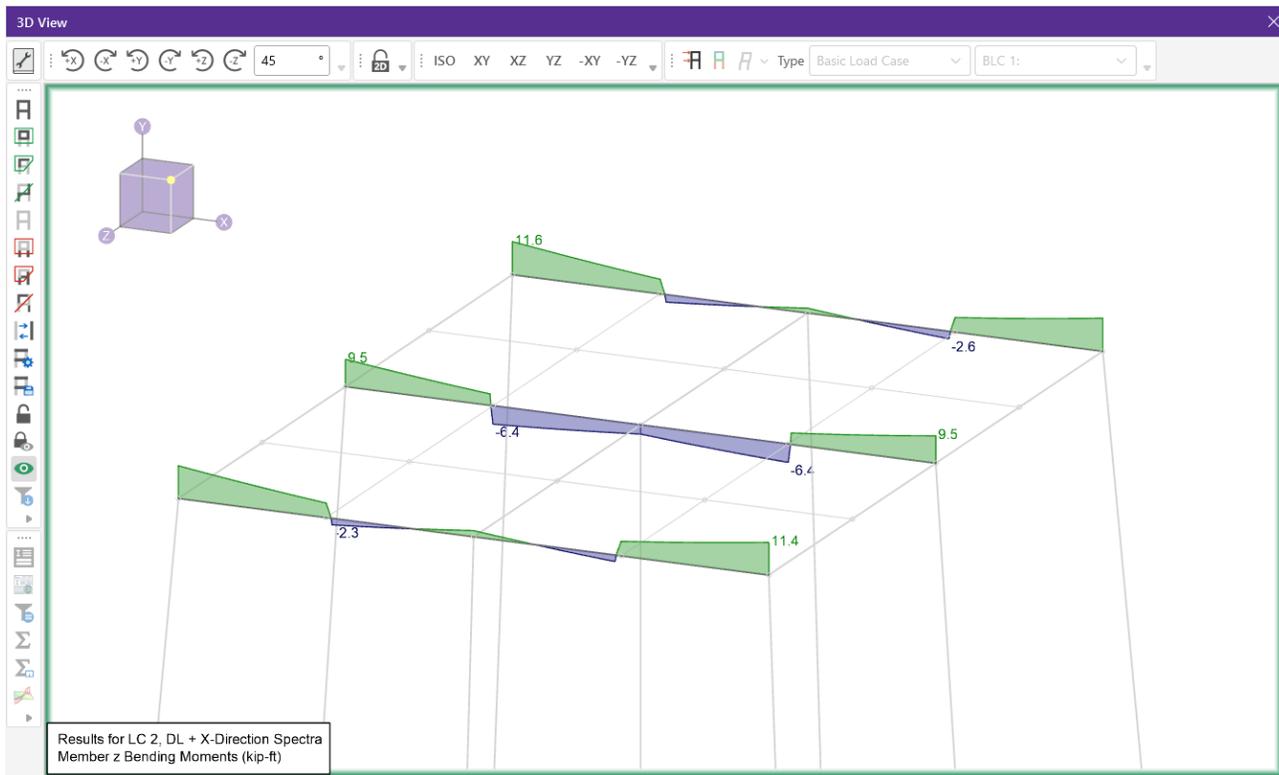
To unlock the **Dim Lock** state, simply click the  button and the dimmed elements will return back to their original state

Viewer Mode

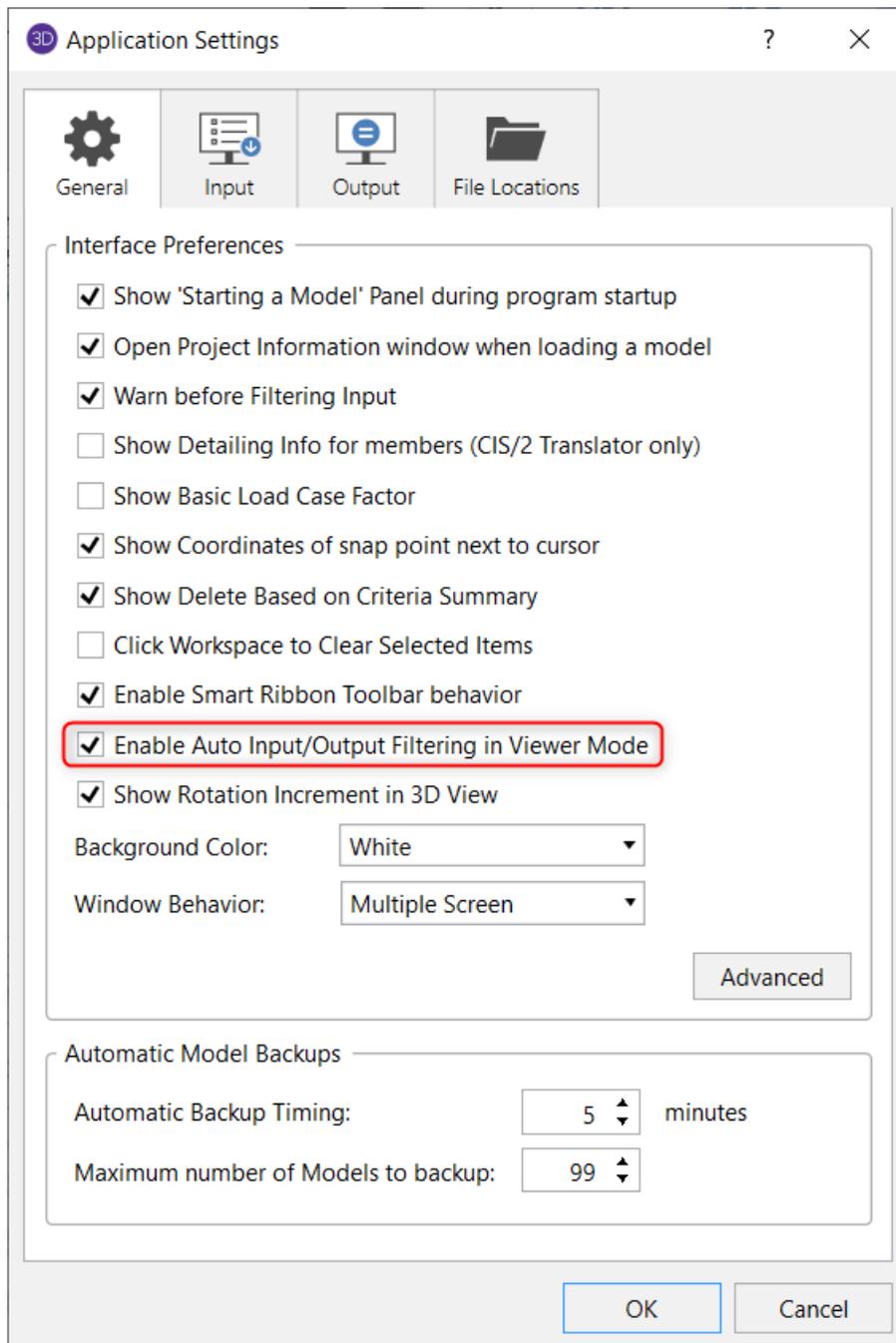
Similar to the 'Dim Lock' feature, the **Viewer Mode** icon  causes all currently unselected elements to become greyed out. However, in viewer mode, you can select or unselect elements with a single click.



While in viewer mode, labels, results and other element specific information is only be displayed for elements that are selected. You can quickly click on different elements to make them visible or grayed out.



In addition to the graphic display, both the input and result spreadsheets automatically filter its information based on the elements that are selected in the viewer mode. This automatic filtering can be turned off within your Application settings > General tab.

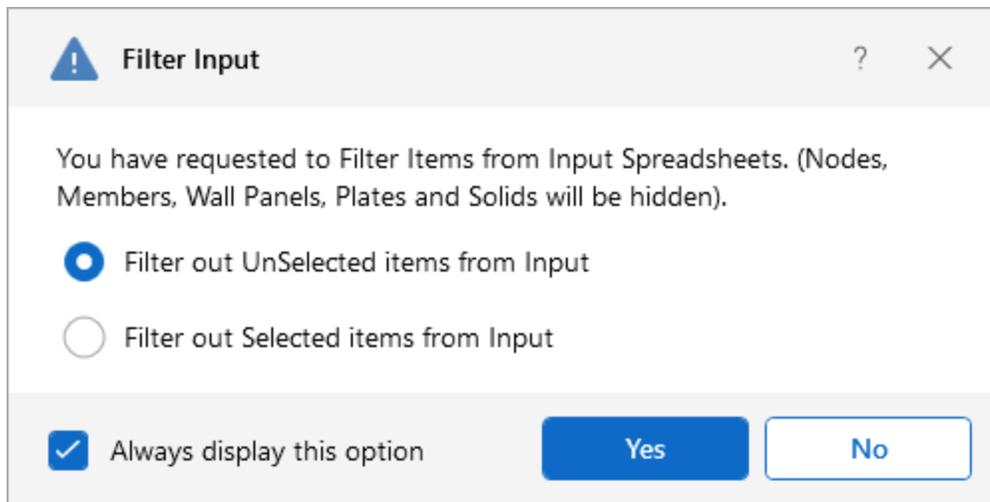


Note that while in viewer mode, a green border appears around the 3D view.

It is also important to note that while in viewer mode, it is not possible to model or make edits to your model. The intention behind this is that viewer mode is primarily focused on 'viewing' your elements and associated element information.

Input Filter

To filter the input spreadsheets based on your selection, click the **Input Filter**  option in the *Selection Toolbar*. You have the option to filter out the UnSelected items or the Selected items from the input spreadsheets.



When you open any input spreadsheet, you can tell if it has been filtered by the filter icon in the spreadsheet header.

Member Primary Data										
Primary	Advanced	Hot Rolled	Cold Formed	Wood	Concrete Beam	Concrete Column	Aluminum	Stainless	RISACconnection	
	Label	I Node	J Node	K Node	Rotate(deg)	Section/Shape	Type	Design List	Material	Design Rule
1	M12	N23	N24			W10X49	Column	Wide Flan...	Steel AS...	EXT. GIRDER
2	M25	N24	N14			INT BEAM 1	Beam	None	A992	INT BEAM 1
3	M38	N22	N24			EXT. GIRDER	Beam	None	A992	EXT. GIRDER
4	M46	N39	N50			INT BEAM 1	Beam	None	A992	INT BEAM 1
5	M47	N41	N51			INT BEAM 1	Beam	None	A992	INT BEAM 1

Graphic Selection from Spreadsheets

In many spreadsheets, both Data Entry and Results, you can select rows on the spreadsheet and use the right-click menu to select the **Select Marked Lines in Current View** or **Unselect Marked Lines in the Current View** options. This graphically selects or unselects the items corresponding to the

selected rows in the top most model view window. This is very useful for highlighting failing members in a spreadsheet and then having them graphically selected in a model view to see where the problems are.

Save Selections

You can save and recall various selection states for a model. If the model is altered after a selection state has been saved, the saved selection state will also be altered. Any new items (joints, members...) will be set to "selected" in any selection states saved prior to the creation of the new item.

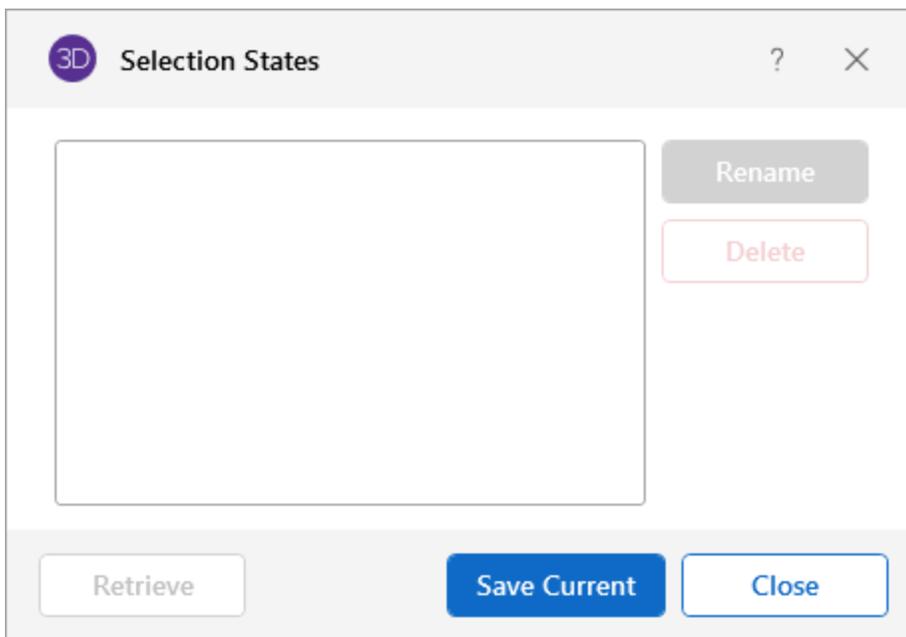
Save a Selection

You can have up to 16 different saved selections in a model.

To save a selection state:

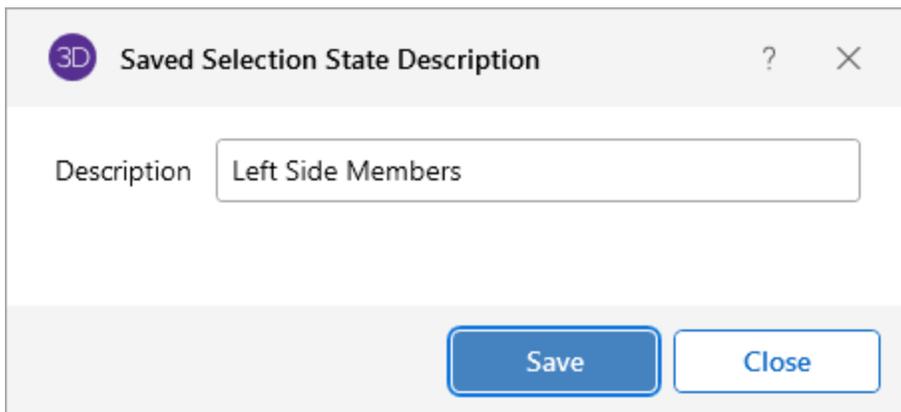
1. Click the **Saved Selections**  button on the *Selection Toolbar*.

The Selection States window opens.



2. Click **Save Current**.

The Saved Selection State Description window opens.



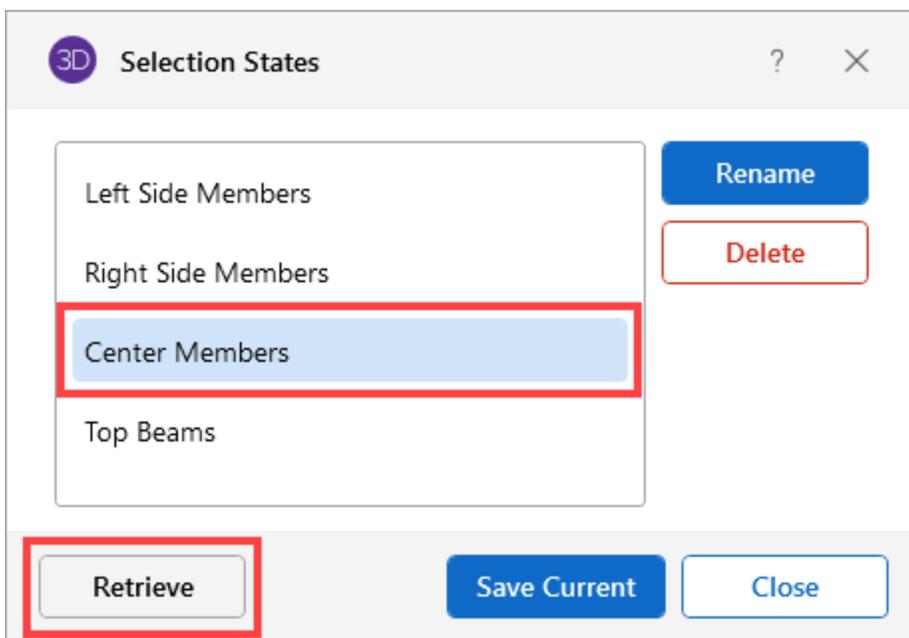
3. Type a **Description** for the selection.
4. Click **Save**.

Retrieve a Saved Selection

To retrieve a saved selection:

1. Click the **Saved Selections**  button on the *Selection Toolbar*.

The Selection States window opens.



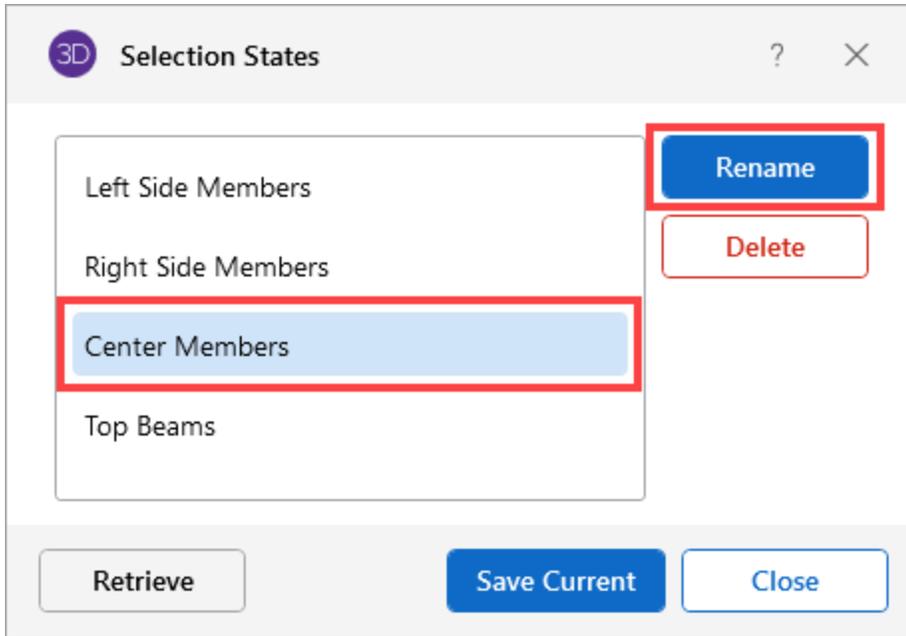
2. Choose the selection state from the list on the left.
3. Click the **Retrieve** button.

Rename a Saved Selection

To rename a saved selection:

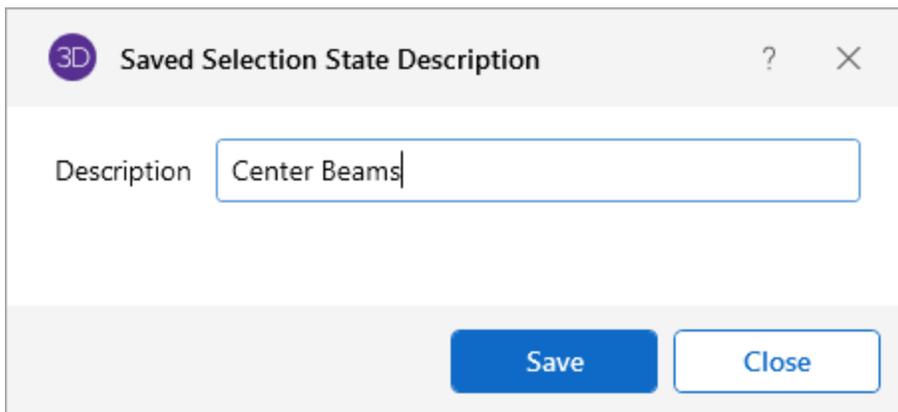
1. Click the **Saved Selections**  button on the *Selection Toolbar*.

The Selection States window opens.



2. Choose the selection state from the list on the left.
3. Click the **Rename** button.

The Saved Selection State Description window opens.



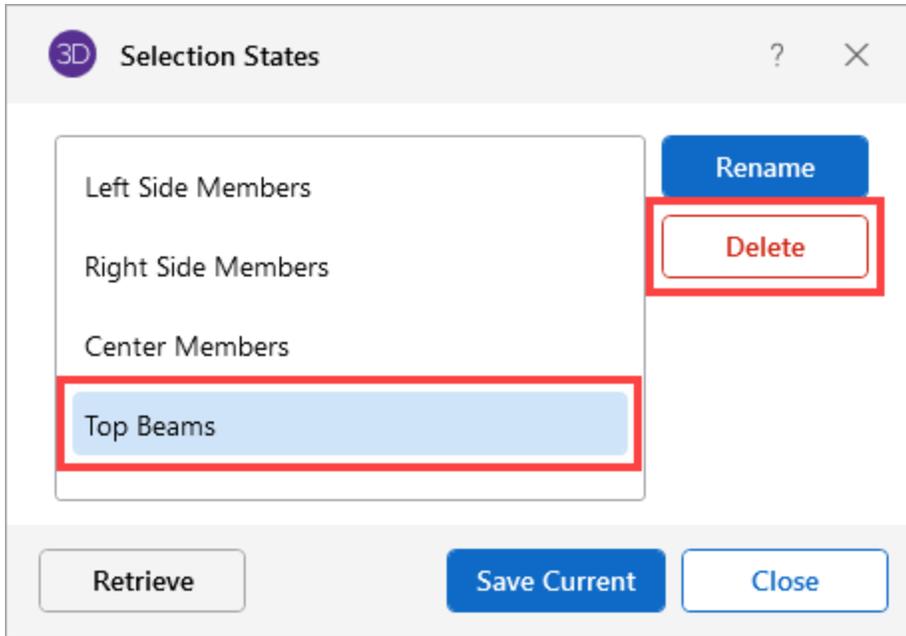
4. In the **Description** box, type a new name for the selection.
5. Click **Save**.

Delete a Saved Selection

To delete a saved selection:

1. Click the **Saved Selections**  button on the *Selection Toolbar*.

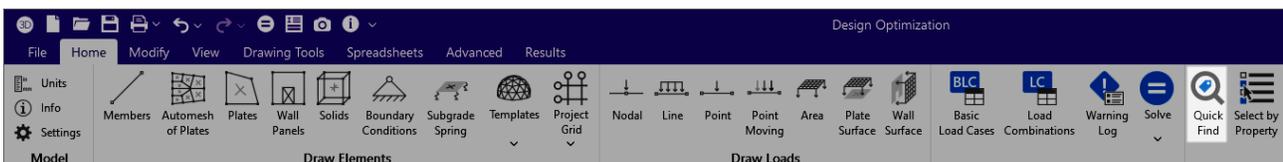
The Selection States window opens.



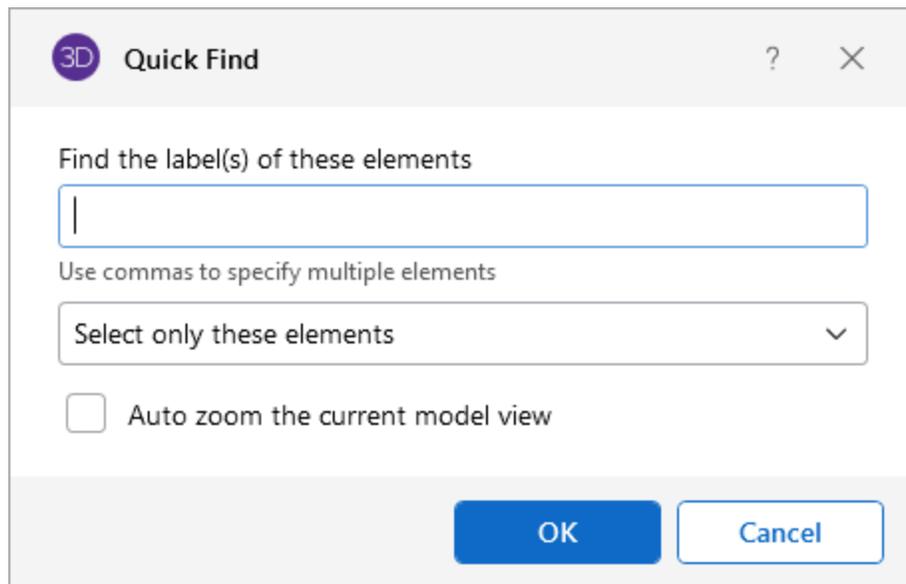
2. Choose the selection state from the list on the left.
3. Click the **Delete** button.

Quick Find

You can quickly find elements in a model using the **Quick Find** tool located on the **Home** ribbon toolbar.



Clicking the **Quick Find** icon opens the Quick Find window.



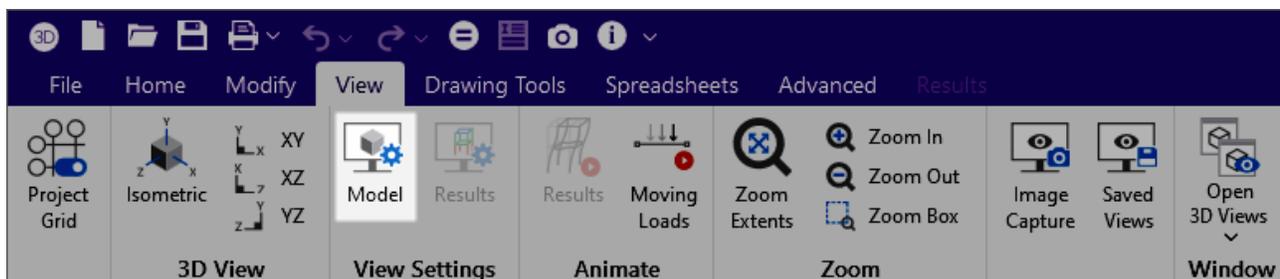
In this window, you can type in the labels of the elements you wish to find, using commas to specify multiple elements. You can also use the drop down to choose between **Select only these elements** or **Add these elements to the existing selection**. The Quick Find window also includes an **Auto zoom the Current Model View** (to these elements) checkbox that when checked, automatically zooms the current model view.

Graphic View Settings

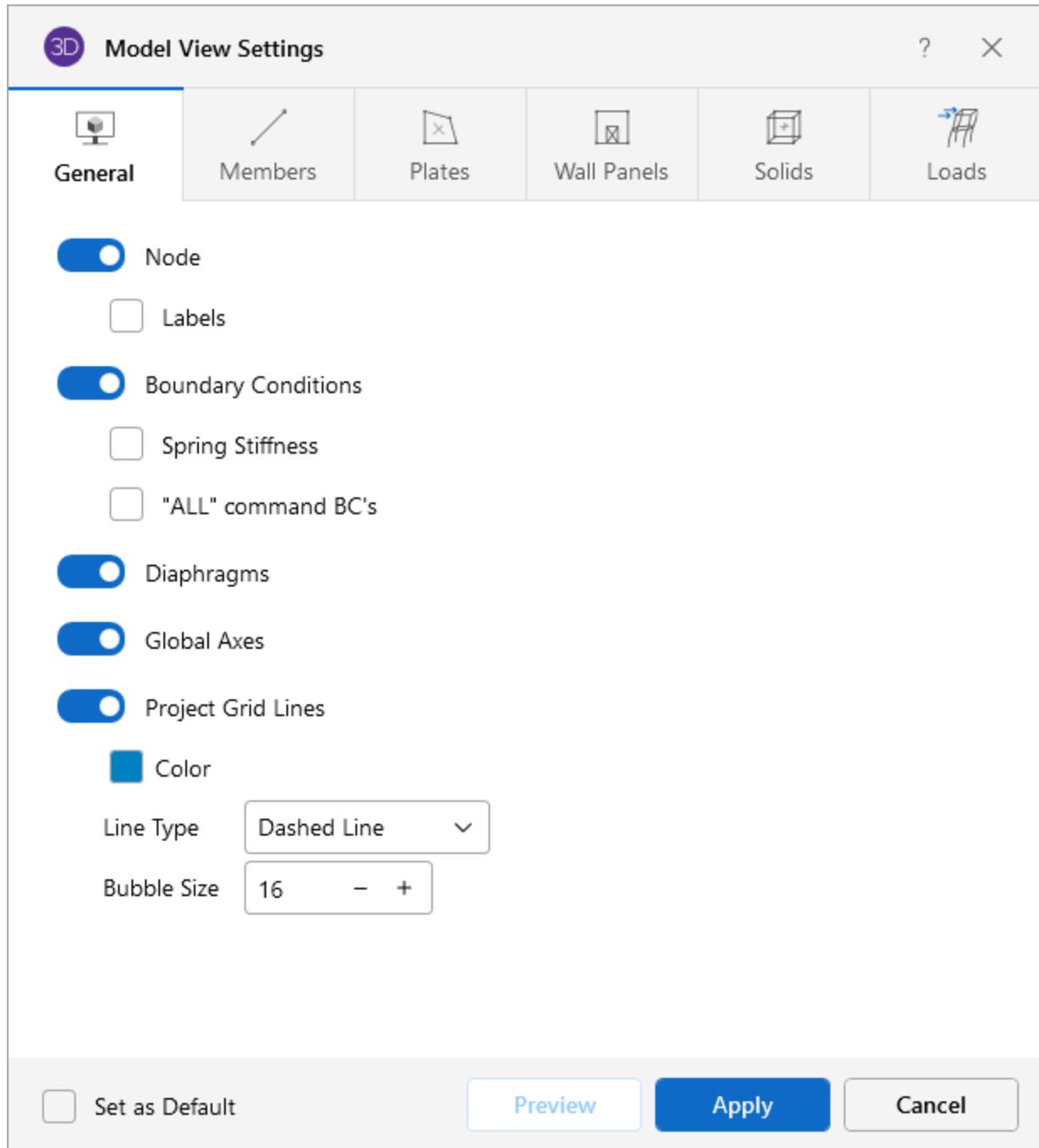
The graphical view settings for both models and results can be found on the View ribbon. These settings offers the ability to define graphical plotting choices in both model and results views.

Graphics - Model View Settings

While the 'Quick View' icons on the 'Home' ribbon (see [Graphic Display](#) for more information) offers some common graphical plotting choices, many more options are located in the **Model View Settings** window.



The Model View Settings window can be accessed by clicking the **Model** icon in the View Settings section of the **View** ribbon.



The options in this window are organized into groups of items. Each group can be accessed by clicking on the tabs along the top of the window. Some options are mutually exclusive, while others are conditional. Indented sub-options are not available unless you choose to turn on the parent group option (click on its slider).

Each of the groups presents the option to turn off (or on) its particular items in the view. This is not the same as deselecting the items. Turning items on and off is independent from selecting and deselecting. For example if you make a selection of members and then turn off members all together, the selection still stands for any modification applied to members.

The tabs in the Model View Settings window are described in the following sections:

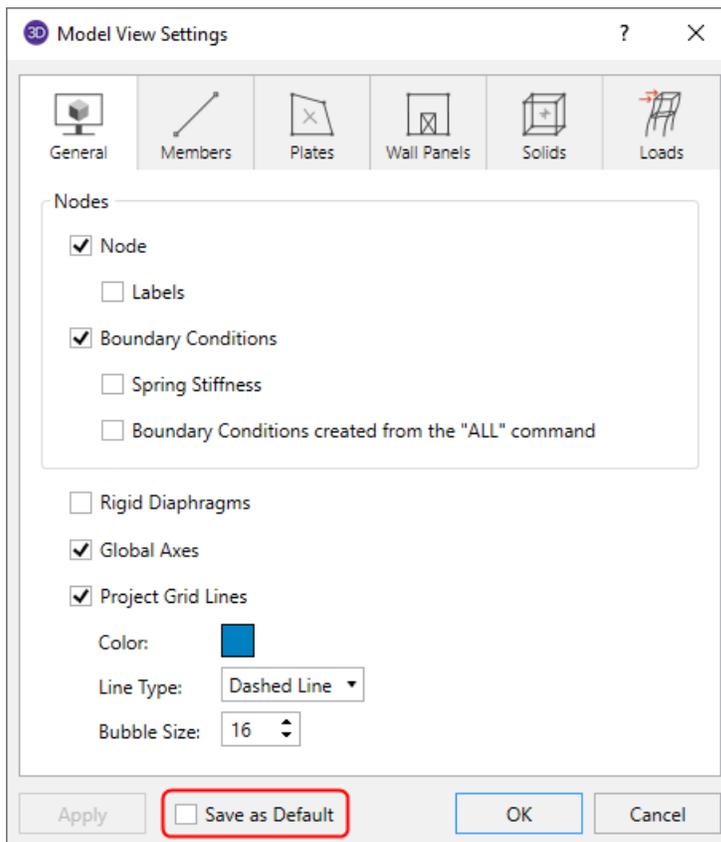
- [General](#)
- [Members](#)
- [Plates](#)
- [Wall Panels](#)
- [Solids](#)
- [Loads](#)
- [Diaphragms](#)

Set as Default - checkbox

You can make any settings in the Model View Settings window the default start-up settings. To do this, go through all the Model View Settings tabs and set all the options to what you want as the start-up defaults. Once that is done, select the **General** tab and check the option at the bottom of the window **Set as Default**.

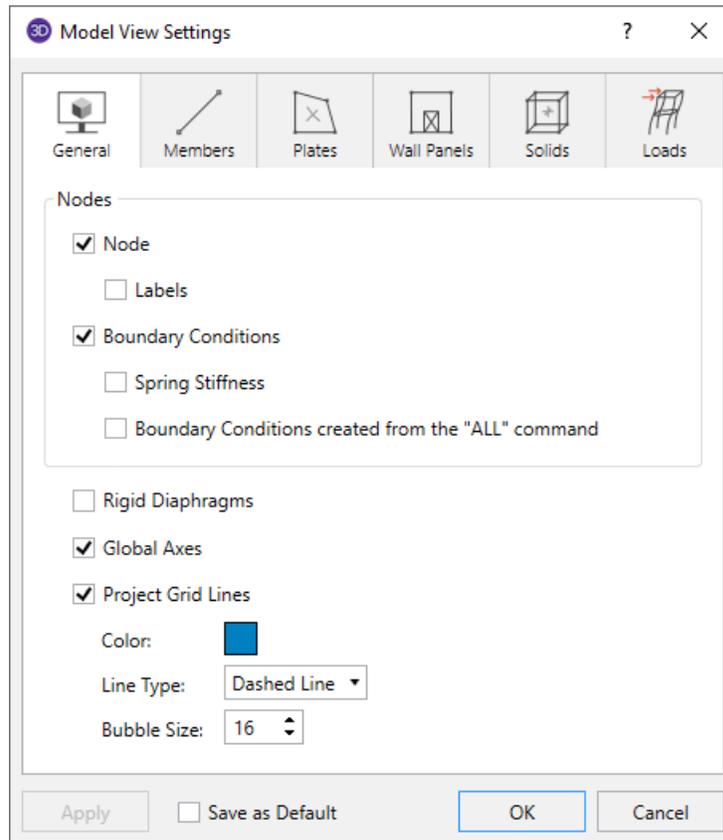


Note: Remember, the Set as Default checkbox applies to ALL tabs in the Model View Settings window.



General

Access the graphical options for nodes by clicking the **Model** icon in the View Settings section of the **View** ribbon and selecting the **General** tab shown below.



You can specify that **Nodes** are/ are not displayed, and you can include the joint label.

You can display **Boundary Conditions** and as part of this option you can display any spring stiffnesses that you have specified. The “ALL” boundary condition is not shown unless it is also specified.

Rigid Diaphragms are plotted if its box is checked.

You can indicate whether the global axes icon is to be displayed in the upper left corner of the model view by checking the **Global Axes** box.

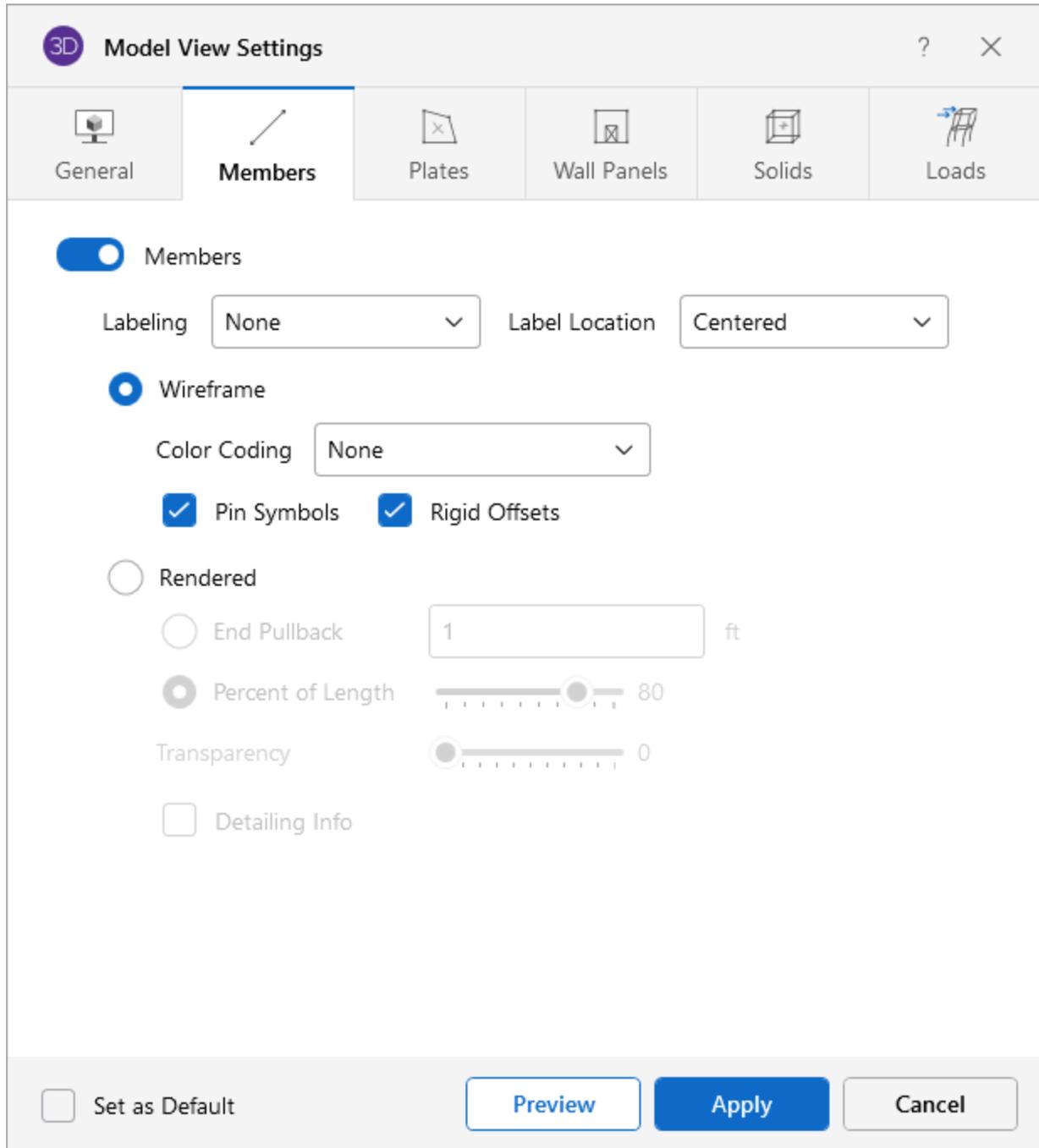
Checking the **Project Grid Lines** box turns on the display of the project grid in the current model view. The color and line type for the grid can also be customized. Click the colored rectangle to choose a custom color. Choose from the **Line Type** drop down to display the project grid with dashed or solid lines. The **Bubble Size** drop down list allows you to choose a font size for the grid labels and the bubbles that encompass them.



Note: You can make any model view settings the default start-up settings. See [Set as Default - checkbox](#) for more information on this feature.

Members

Access the graphical Model View Settings for members by clicking the **Model** icon in the View Settings section of the **View** ribbon and selecting the **Members** tab shown below.



You can specify that no members are drawn, or you can draw them as Wireframe (with a Label or Color Code) or Rendered.

Click on the **Members** slider to not display the members in the model at all.

With the **Wireframe** options you can include textual member information alongside each member. The **Labeling** drop down list provides the choices you have as to what information to be displayed alongside each member. The Wireframe option also lets you display member **Pinned End** conditions and/or either **Rigid End Offsets**.

Wireframe options also include color coding members which plot the members using various colors to represent particular information such as, the section set assigned to each member, member type, etc. Use the **Color Coding** drop down list that provides the choices you have as to how the colors are to be assigned to each member. The key that defines these colors is shown in the upper right corner of the model view.

The **Rendering** option produces true-to-scale renderings of the members. These can be rendered full length or pulled back by a specified distance or a percentage of the member length for a better understanding of complicated intersection areas and a better view of the member cross-section. This option is also very useful for verifying member orientations and adjusting the the transparency of the rendering.

The **Detailing Info** check box pertains to information that is for export through the CIS/2 Translator. See the [Cardinal Points](#) topic for more information.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Graphics**

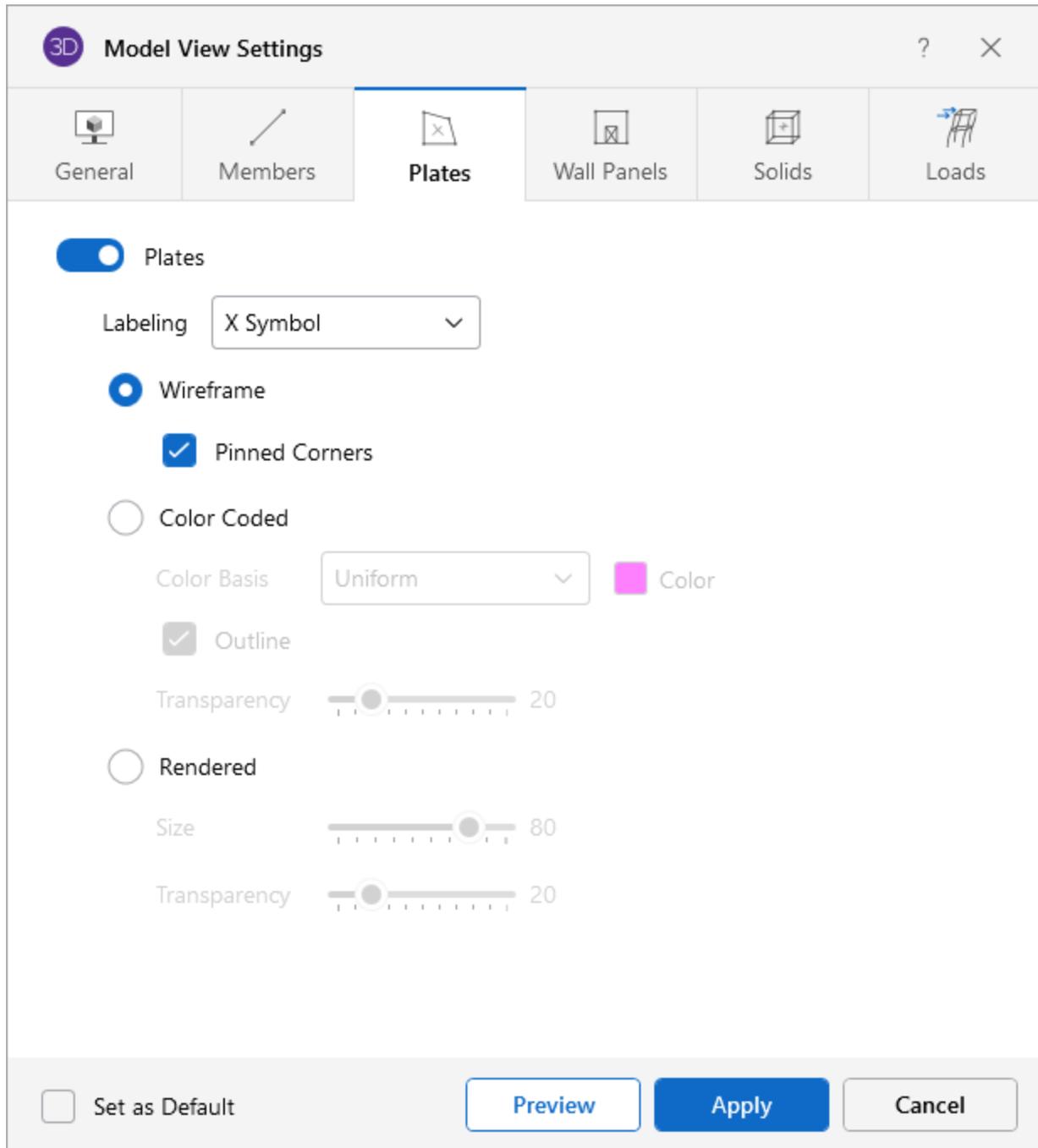
For additional advice on this topic, please see the RISA Tips & Tricks website: www.risa.com/post/support. Type in Search keyword: **Graphics**.



Note: You can make any model view settings the default start-up settings. See [Set as Default - checkbox](#) for more information on this feature.

Plates

Access the graphical Model View Settings for plates by clicking the **Model** icon in the View Settings section of the **View** ribbon and selecting the **Plates** tab shown below.



This tab lets you specify that plates are to be drawn as wireframe, color coded, or rendered elements, or that they are not to be shown at all.

Click the **Plates** slider to not display the plates in the model at all.

When the **Wireframe** option is selected, you can include information on the plates when plotted. The **Labeling** drop down list provides the choices you have as to what information will be displayed on

each plate, such as the plate label, material, thickness, or the plate local axes. The wireframe option also allows you to display plate **Pinned Corner** conditions.

When the **Color Coded** option is selected, plates are plotted with different colors that are mapped in a key shown in the upper right corner of the model view. You can color the plates by material set or specify a uniform color for all plates. You can toggle the **Outline** of the plates on and off. The color coded plates themselves may be presented with varying **Transparency** with 100% transparency meaning completely see through and 0% indicating completely solid.

When the **Rendered** option is selected, true to scale representations of the plates are drawn with thickness. You can also display the rendered plates at a percentage of their **Size**. This is useful in understanding orientation and connectivity in complex views as shown below.

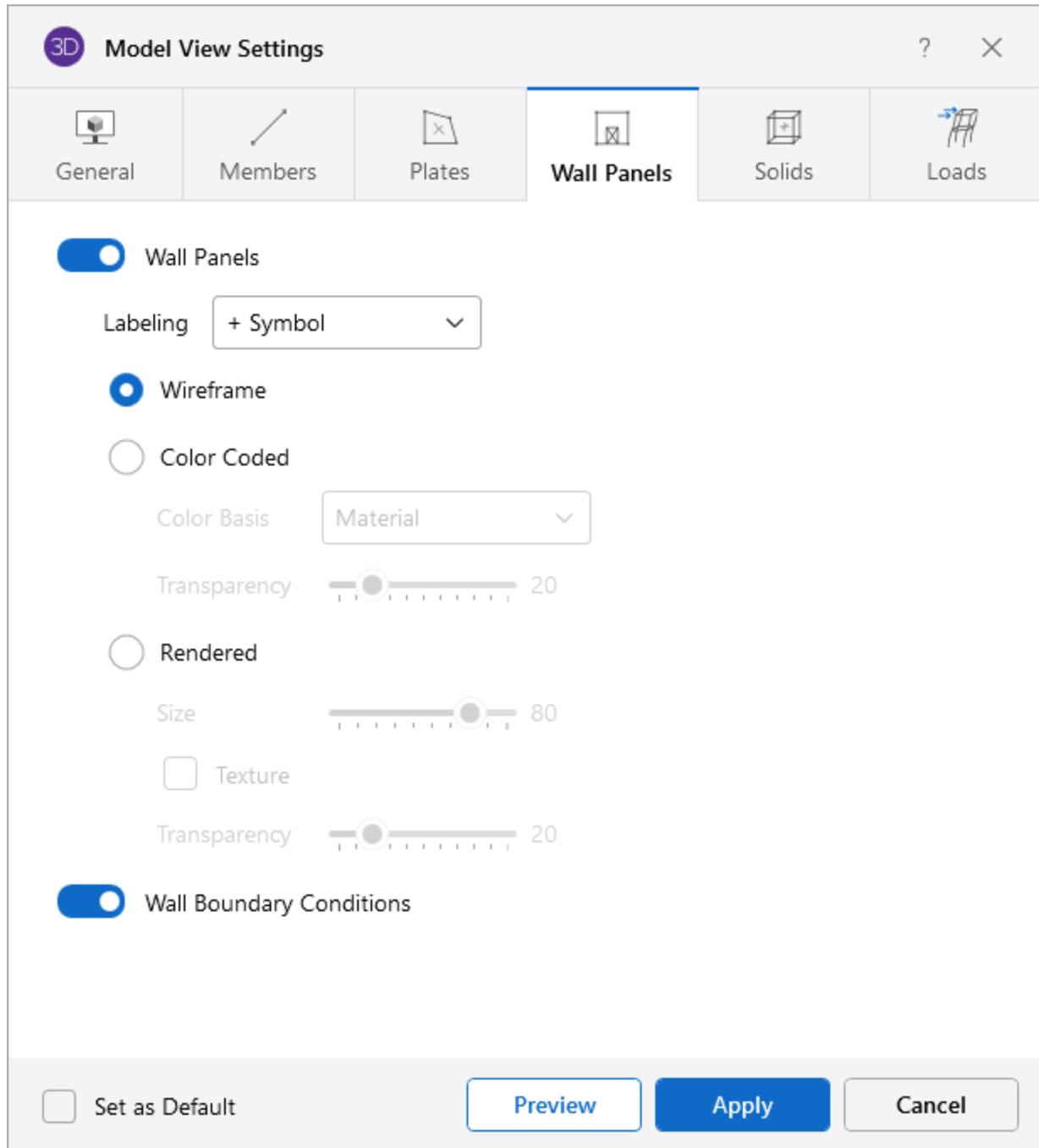
The **Transparency** setting described above for Color Coded plates also applies to Rendered plates.



Note: You can make any model view settings the default start-up settings. See [Set as Default - checkbox](#) for more information on this feature.

Wall Panels

Access the graphical Model View Settings for wall panels by clicking the **Model** icon in the View Settings section of the **View** ribbon and selecting the **Wall Panels** tab shown below.



The **Wall Panels** tab lets you specify that panels are to be drawn as wireframe, color coded, or rendered elements, or that they are not to be shown at all.

Click the **Wall Panels** slider to not display the wall panels in the model at all.

When the **Wireframe** option is selected, you can include information on the panels when plotted. The **Labeling** drop down list provides the choices you have as to what information will be displayed on

each wall panel, such as the wall panel label, material, or the wall panel number. You also have the option to see the mesh that the program automatically creates internally for all Wall Panels.

When the **Analysis Mesh** option is checked on, wall panels are plotted showing the internal plate mesh automatically generated during a solution. This is useful for verifying and understanding how the forces/stresses are distributed within a panel.

When the **Color Coded** option is selected, wall panels are plotted with different colors that are mapped in a key shown in the upper right corner of the model view. You can color the panels by material set or specify a uniform color for all panels. The color coded panels themselves may be presented with varying **Transparency** with 100% transparency meaning completely see-through and 0% indicating completely solid.

When the **Rendered** option is selected, true to scale representations of the panels are drawn with thickness. The **Texture** checkbox shows the wall panels with different texture depending on the wall panel material. You can also display the rendered panels at a percentage of their size. This is useful in understanding orientation and connectivity.



Note: You can make any model view settings the default start-up settings. See [Set as Default - checkbox](#) for more information on this feature.

Solids

Access the graphical Model View Settings for solids by clicking the **Model** icon in the View Settings section of the **View** ribbon and selecting the **Solids** tab shown below.

The screenshot shows the 'Model View Settings' dialog box with the 'Loads' tab selected. The dialog has a title bar with a 3D icon, the text 'Model View Settings', and help and close buttons. Below the title bar is a tabbed interface with six tabs: 'General', 'Members', 'Plates', 'Wall Panels', 'Solids', and 'Loads'. The 'Loads' tab is active and contains the following settings:

- A 'Loads' toggle switch is turned on.
- 'Magnitudes' is checked.
- 'Basic Loads Case' is set to '1: Self Weight' (dropdown).
- 'Load Category' is set to 'DL - Dead Load' (dropdown).
- 'Load Combination' is set to 'LC 1: IBC 16-1' (dropdown).
- Under 'Load Types', the following are checked: 'Area', 'Node', 'Point', 'Distributed', 'Surface (Plate)', and 'Surface (Wall)'. 'Transient (Wall)' is unchecked.

At the bottom of the dialog, there is a 'Set as Default' checkbox, a 'Preview' button, an 'Apply' button, and a 'Cancel' button.

The **Solids** tab lets you specify that solids are to be drawn as wireframe, color coded, or rendered elements, or that they are not to be shown at all.

Click the **Solids** slider to not display the solids in the model at all.

When the **Wireframe** option is selected, you can include information on the solid elements when plotted. The **Labeling** drop down list provides the choices you have as to what information will be displayed on each solid, such as the label, material, or number.

When the **Color Coded** option is selected, solids are plotted with different colors that are mapped in a key shown in the upper right hand corner of the model view. You can color the solids by material set or specify a uniform color for all solids. The color coded solids themselves may be presented with varying **Transparency** with 100% transparency meaning completely see-through and 0% indicating completely solid.

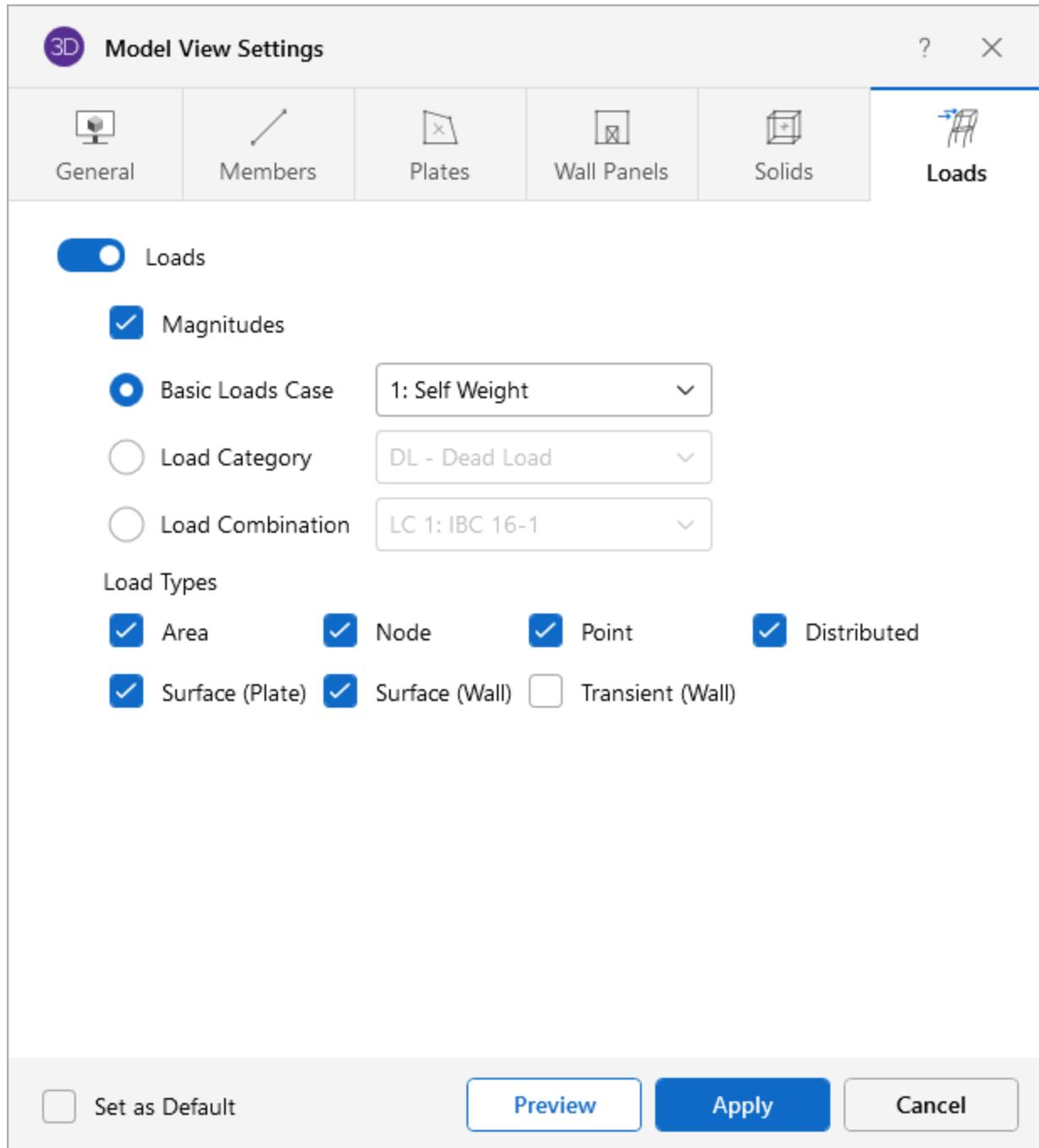
When the **Rendered** option is selected, true to scale representations of the solids are drawn. You can also display the rendered panels at a percentage of their size.



Note: You can make any model view settings the default start-up settings. See [Set as Default - checkbox](#) for more information on this feature.

Loads

Access the graphical Model View Settings for loads by clicking the **Model** icon in the View Settings section of the **View** ribbon and selecting the **Loads** tab shown below.



The **Loads** tab lets you specify that loads are to be displayed graphically, or that they are not to be shown at all.

Click the **Loads** slider to not display the loads in the model at all.

Check **Magnitudes** to display the numerical magnitude of the loads. You can specify that the loads be drawn as **Basic Load Cases**, **Load Categories** or **Load Combinations** by selecting the

corresponding radio button. The load types to be displayed can be selected by checking the corresponding boxes.



Note: The **Transient (Wall)** check box will show how loading is attributed to wall panel nodes. For a distributed load applied to the top of the wall this check box will show you how the line load is attributed as point loads to each plate corner at the top of the wall. For out of plane loading on walls with openings, this check box will show how the load in the area of the opening is attributed as point loads to the edges of the opening in a two-way distribution.

Once you set the Model View Settings for loads, you can then run through all of the load cases, load categories or load combinations by choosing them from the drop down list at the top of the **3D View**.



Basic Loads Case

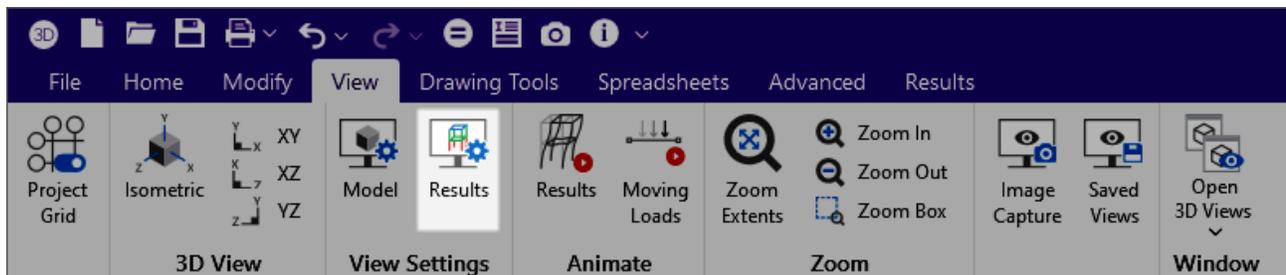
1: Self Weight



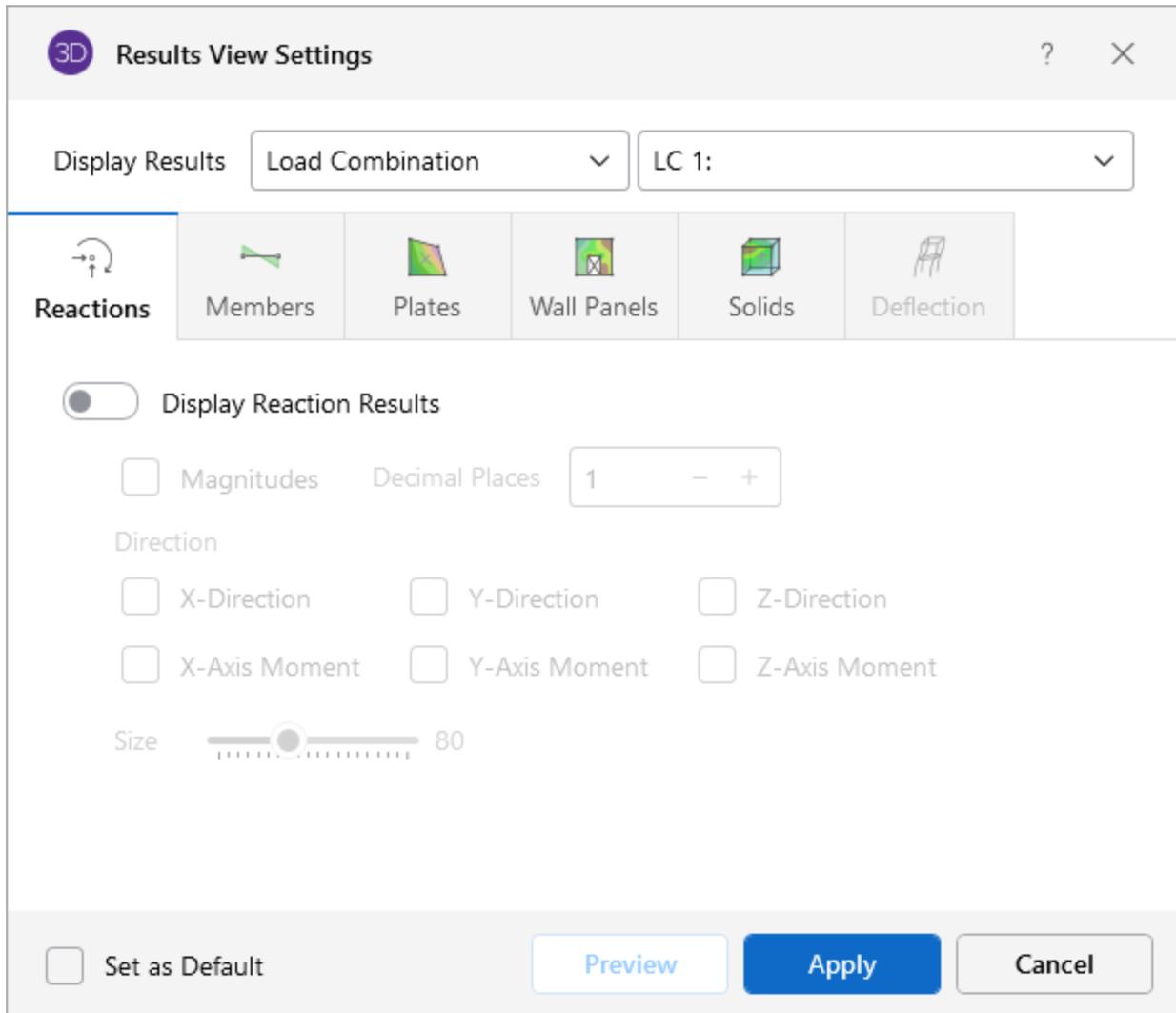
Note: You can make any model view settings the default start-up settings. See [Set as Default - checkbox](#) for more information on this feature.

Results View Settings

The **Quick View** buttons on the **Home** tab (see [Graphic Display](#) for more information) offers some common graphical plotting choices, but these are just a few of the options available. Many more options related to results are located in the **Results View Settings** window which can be accessed by clicking the **Results** icon in the 'View Settings' section of the 'View' ribbon.



The options are organized into groups of items. Access each group by clicking on the tabs along the top of the window (as shown in the following image). Some of the options are mutually exclusive and others are conditional. Typically, only one radio button option may be chosen at a time, while you may be allowed to select multiple check boxes.



Each of the groups presents the option to turn off the results for those particular items in the view by unchecking the Display Results option located at the top of each tab.

The displayed results are based on the specified Load Combination from the top of the Results View Settings window.

Please see the following sections for more information about the setting options in the Results View Settings window:

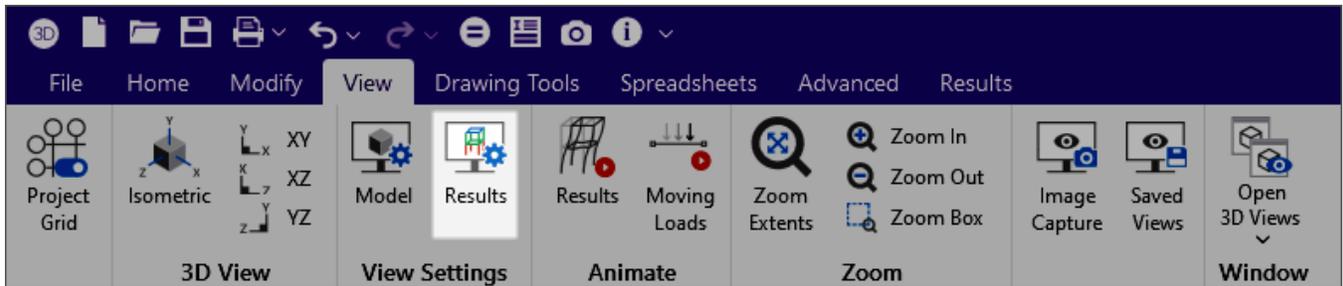
- [Reactions](#)
- [Members](#)
- [Plates](#)
- [Wall Panels](#)
- [Solids](#)

- [Diaphragms](#)
- [Deflection](#)

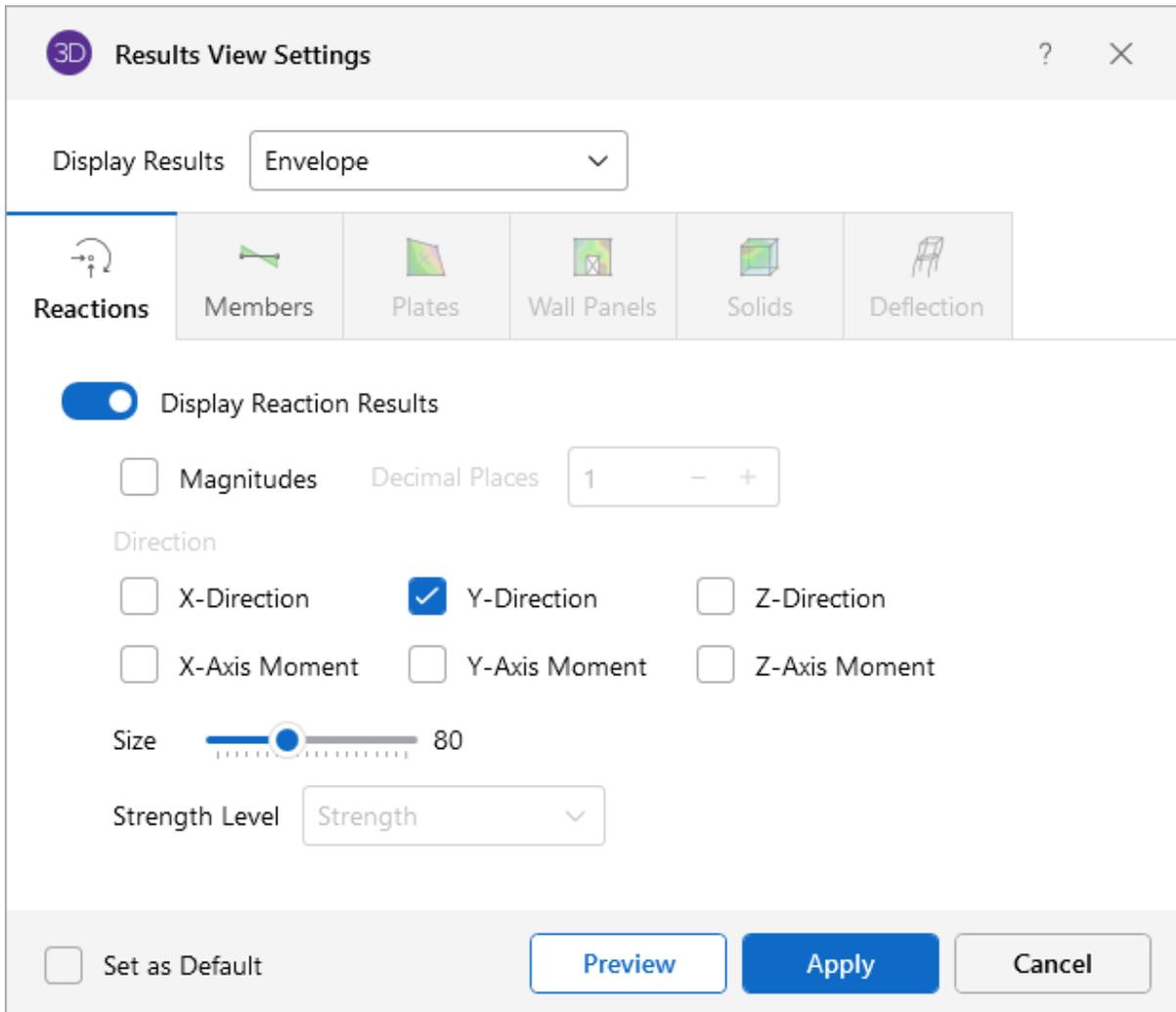
Reactions

To access the graphical options for displaying reactions:

1. Go to the **View** ribbon.



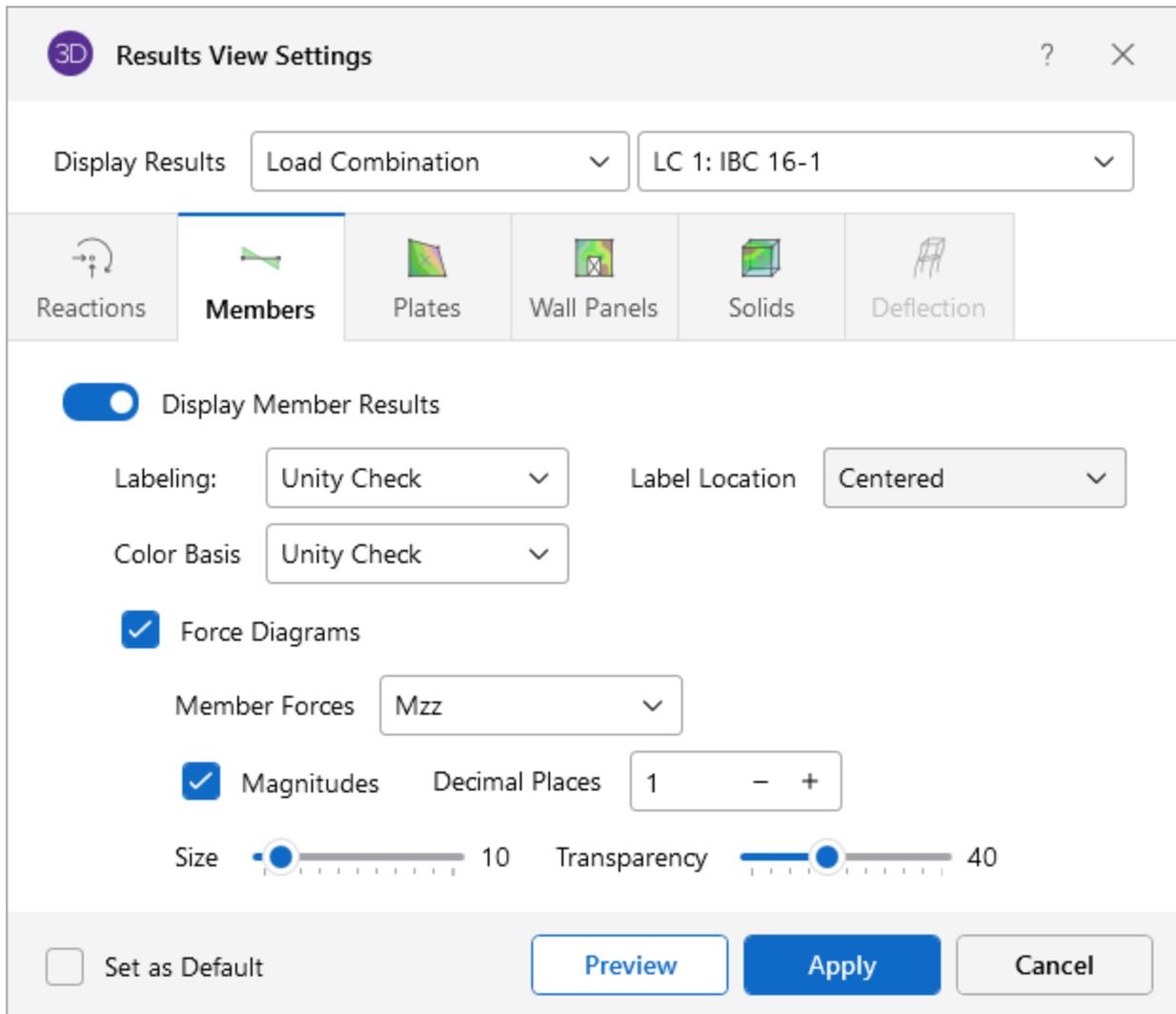
2. Click the **Results** icon in the View Settings section.
The Results View Settings window opens.



3. Click on the **Reactions** tab if not already open.
4. Modify the settings are needed.
 - Click on the **Display Reaction Results** slider to make it turn gray, if you don't want to display any reaction results.
 - You can show the **Reactions** in any of the six degrees of freedom, if a single or a batch solution has been performed. You can also include the magnitude values and specify the number of decimal places and display arrow size.
For Batch solutions you must choose which combination you want to view at the top of the Results View Settings window.
 - If running **Overstrength** load combinations, you can then show the Overstrength level or Strength level envelope reactions.
5. (Optional) Click **Preview** to see the results displayed in the model.
6. Click **Apply** to apply the setting changes to the model.

Members

Access the graphical Results View Settings for members by clicking the **Results** icon in the View Settings section of the on the View ribbon and then clicking the **Members** tab (as shown in the following image).



Click on the **Display Member Results** slider to make it turn gray, if you don't want to display any member results.

You can specify that member results are to be shown with labeling or using a color basis.

The **Labeling** drop down list provides the choices you have as to what information will be displayed alongside each member.

Choosing a **Color Basis** for member results uses various colors to represent particular results such as the unity check, stress levels, etc. The drop down list provides the choices you have as to how the

colors are to be assigned to each member. If a single or batch solution has been performed you may color the members by the code check or the stress magnitudes. For batch solutions you must choose which combination you want to view at the top of the Results View Settings dialog. The key that defines these colors is shown in the upper right corner of the model view.



Note: The **Tension** and **Compression** color coding options depend on the total axial tension or compression in the member. This includes the axial (tension or compression) stress plus the axial stress due to bending.

If the model has been solved you can also plot the member force **Diagrams** along each member and include the **Magnitudes**. You can set the number of **Decimal Places** for the magnitudes as well as control the graphic scaling of these force diagrams using the **Size** slider. The diagrams themselves can be presented with varying degrees of **Transparency** with 100% transparency meaning an outline of the force diagram and 0% indicating a solid fill. If a batch solution has been performed you can also choose which combination you want to view at the top of the Results View Settings dialog.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Graphics**.

Plates

Access the graphical Results View Settings for plates by clicking the **Results** icon in the View Settings section of the View ribbon and then clicking the **Plates** tab (as shown in the following image).

3D Results View Settings ? X

Display Results: Load Combination LC 1: IBC 16-1

Reactions Members **Plates** Wall Panels Solids Deflection

Display Plate Results

Contour Basis: Qx Contour Lines Color Gradient

Range: Custom Min: -1 Max: 1

Corner Forces Direction: X-Direction

Set as Default Preview Apply Cancel

You can specify that plate results are to be shown as contour lines, color gradient, or that they are not to be shown at all. Click on the **Display Plates Results** slider to make it turn gray, if you don't want to display any plate results.

If a single or batch solution has been performed, the contour lines represent the force or stress results for the plates. Each color line represents a specific value. You also have the option to plot the stress and force results using filled color contours where each fill color represents a range of values. For batch solutions, you must choose which load combination you want to view at the top of the Results View Settings window.

The **Contour Basis** drop down list is where you select the specific result to be contoured (Qx, Fx, Mx, Von Mises, etc.). The **Range** controls determine how the contour colors are assigned. You can either contour the full range of the results by choosing **Automatic**, or, if you are only interested in a specific range of values, you can choose **Custom** and enter your own max and min values.

The **Color Contours** are plotted with a global smoothing algorithm that allows the contour to vary across the plates. Because of this, the plotted contour results differ slightly from the tabulated results in the spreadsheets. This effect is heightened in regions of a high rate of change such as loads or boundaries. The contours can be drawn as either lines or as color filled areas.

Along with any of the above options you may also plot the plate **Corner Forces**. These are the global direction forces applied to the corners of the plates that hold them in equilibrium. The **Direction** options specify which corner force (or moment) is to be displayed. For batch solutions you must choose which combination you want to view at the top of the Results View Settings dialog.

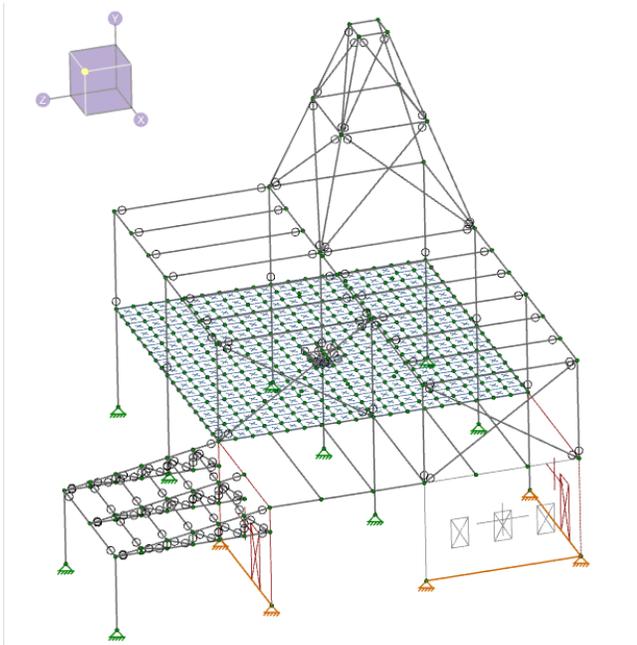
Wall Panels

Access the graphical Results View Settings for panels by clicking the **Results** icon in the View Settings section of the View ribbon and then clicking the **Wall Panels** tab (as shown in the following image).

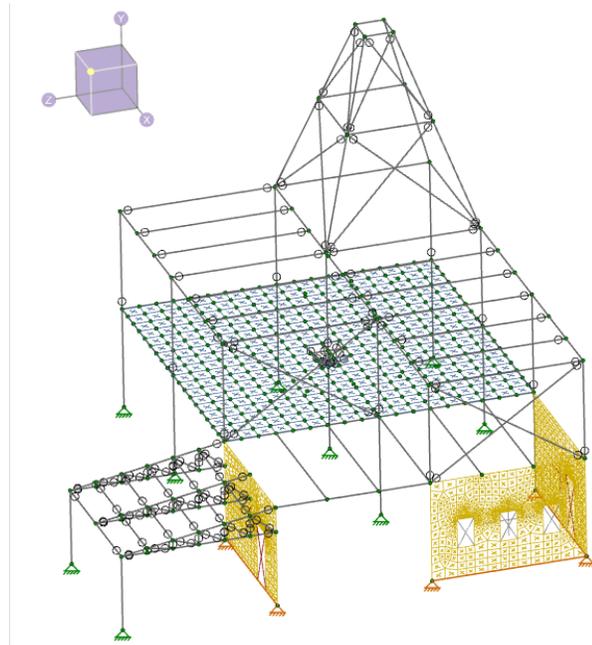
The screenshot shows the 'Results View Settings' dialog box with the 'Wall Panels' tab selected. The dialog has a title bar with a '3D' icon and window controls. Below the title bar, there are two dropdown menus: 'Display Results' set to 'Load Combination' and 'LC 1: IBC 16-1'. A horizontal ribbon contains six tabs: 'Reactions', 'Members', 'Plates', 'Wall Panels' (selected), 'Solids', and 'Deflection'. Below the ribbon, there are two toggle switches: 'Analysis Mesh' (checked) and 'Display Wall Panel Results' (checked). Under 'Display Wall Panel Results', there are two radio buttons: 'Contour Lines' (selected) and 'Color Gradient'. Below these are two dropdown menus: 'Contour Basis' set to 'Qx' and 'Range' set to 'Custom'. To the right of the 'Range' dropdown are input fields for 'Min' (-1) and 'Max' (1). At the bottom left is a checkbox for 'Set as Default'. At the bottom right are three buttons: 'Preview', 'Apply', and 'Cancel'.

Click on the **Analysis Mesh** slider to turn this option on, if you want to view the internal mesh the program creates of the wall panel in order to perform the analysis. The following images show the analysis mesh turned off and on, respectively.

Analysis Mesh turned OFF



Analysis Mesh turned ON



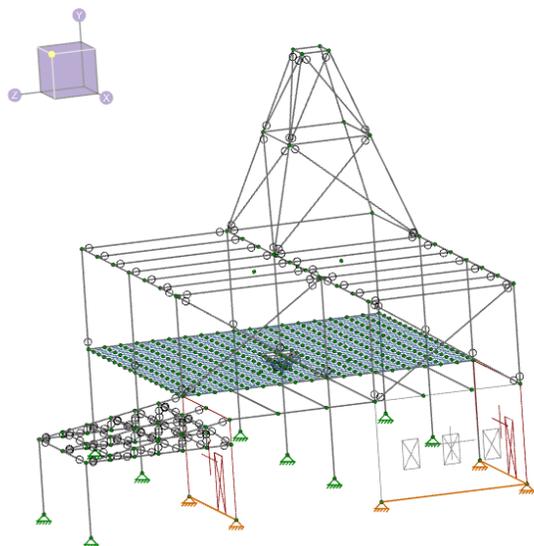
Click on the **Display Wall Panel Results** slider to turn this option on, if you want display any wall panel results.

If a single or batch solution has been performed, you can display contour lines representing the force or stress results for the panels. Each color line represents a specific value. You also have the option to plot the stress and force results using filled color contours where each fill color represents a range of values. The wall panels are plotted with different colors that are mapped in a key shown in the upper right corner of the model view. For batch solutions you must choose which load combination you want to view at the top of the Results View Settings dialog.

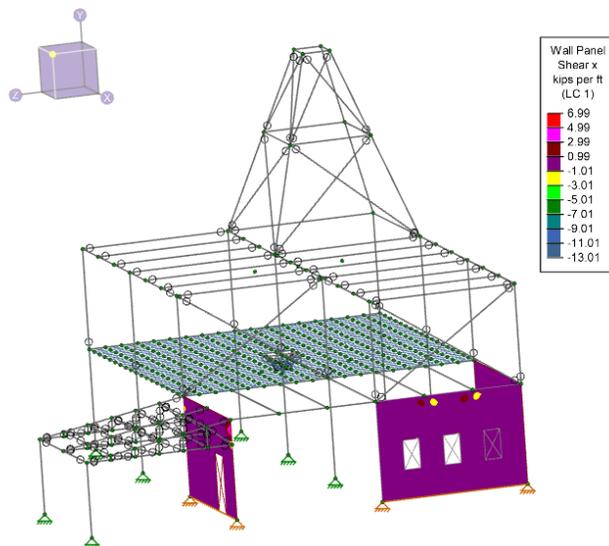
- The **Contour Basis** drop down list is where you select the specific result to be contoured (Qx, Fx, Mx, Von Mises, etc.). The **Range** controls determine how the contour colors are assigned. You can either contour the full range of the results by choosing **Automatic**, or, if you are only interested in a specific range of values, you can choose **Custom** and enter your own max and min values.
- The **Color Contours** are plotted with a global smoothing algorithm that allows the contour to vary across the panels. Because of this, the plotted contour results differ slightly from the tabulated results in the spreadsheets. This effect is heightened in regions of a high rate of change such as loads or boundaries. The contours can be drawn as either lines or as color filled areas.

The following examples show the Display Wall Panel Results option turned off and on respectively, for the QX Contour Basis with the Color Gradient option also chosen.

Display Wall Panel Results turned OFF



Display Wall Panel Results turned ON



Solids

Access the graphical Results View Settings for solids by clicking the **Results** icon in the View Settings section of the View ribbon and selecting the **Solids** tab (as shown in the following image).

Click on the **Display Solid Element Results** slider to make it turn gray, if you don't want to display any solid element results.

The **Contour Basis** drop down list is where you select the specific result to be contoured (SigmaX, Y or Z, Sigma1, 2 or 3, Von Mises, etc.). The **Range** controls determine how the contour colors are assigned. You can either contour the full range of the results by choosing **Automatic**, or, if you are only interested in a specific range of values, you can choose **Custom** and enter your own max and min values.

If a single or batch solution has been performed, you can display contour lines representing the force or stress results for the solids. Each color line represents a specific value. For batch solutions, you must choose which load combination you want to view at the top of the Results View Settings dialog.

The **Color Contours** are plotted with a global smoothing algorithm that allows the contour to vary across the solid elements. Because of this, the plotted contour results will differ slightly from the tabulated results in the spreadsheets. This effect will be heightened in regions of a high rate of

change such as applied loads or boundary conditions. The contours may be drawn as either lines or as color filled areas.

Diaphragms

Access the graphical Results View Settings for Semi-Rigid Diaphragms for models integrating from RISAFloor by clicking the **Results** icon in the View Settings section of the View ribbon and selecting the **Diaphragms** tab shown below.

The screenshot shows the 'Results View Settings' dialog box for Diaphragms. The title bar includes a '3D' icon, the text 'Results View Settings', and standard window controls. Below the title bar, there are two dropdown menus: 'Display Results' set to 'Load Combination' and 'LC 1: IBC 21/ASCE Strength 1'. A horizontal ribbon contains icons for 'Reactions', 'Members', 'Plates', 'Wall Panels', 'Solids', 'Diaphragms' (which is highlighted), and 'Deflection'. Below the ribbon, there are two toggle switches: 'Wireframe Analysis Mesh' (checked) and 'Display Diaphragm Results' (checked). Under 'Display Diaphragm Results', there are two options: 'Contour Lines' (selected with a radio button) and 'Color Gradient' (unselected). The 'Contour Lines' option has a 'Contour Basis' dropdown set to 'QZ' and a 'Range' dropdown set to 'Auto'. The 'Range' section also includes 'Min' and 'Max' input fields with values '-1' and '1' respectively. At the bottom left is a 'Set as Default' checkbox. At the bottom right are three buttons: 'Preview', 'Apply', and 'Cancel'.

The **Draw Diaphragms As...** section of this tab lets you specify that solids are to be drawn as wireframe, contour, or that they are not to be shown at all.

When the **Wireframe** option is selected, you can include the contours or the FEA plate submesh after a single or batch solution has been performed. The display of line contours represents the force or stress results for the internal plates. Each color line represents a specific value. For batch solutions,

you must choose which load combination you want to view in the **Combination To Use For Results Display** drop down list.

The **Contours** are plotted with a global smoothing algorithm that allows the contour to vary across the internal plate elements. The contours are drawn as color filled areas.

The **Contouring Control** section determines what force or stress result is to be contoured and how the contour colors are to be assigned. The **Value** drop down list is where you select the specific result to be contoured (QX, QY, MX, MY, etc.). Note that the forces are based on the global axis as indicated with a capital axis letter (MX vs Mx). The **Range** controls determine how the contour colors are assigned. You can either contour the full range of the results by choosing **Calculated Maximum to Minimum**, or, if you are only interested in a specific range of values, you can choose **User Defined** and enter your own max and min values.



Note: You can make any model view settings the default start-up settings. See [Set as Default - checkbox](#) for more information on this feature.

Deflection

Access the Results View Settings for deflections by clicking the **Results** icon in the View Settings section of the View ribbon and selecting the **Deflection** tab (as shown in the following image).

The screenshot shows the 'Results View Settings' dialog box with the 'Deflection' tab selected. At the top, 'Display Results' is set to 'Deflections' and the load combination is 'LC 21: IBC 16-13 (a) (m)'. The 'Deflection' tab is highlighted among other options like Reactions, Members, Plates, Wall Panels, and Solids. A main toggle for 'Display Deflected Model (using Model View Settings)' is currently turned on (blue). Below it, 'Static Deflection' is selected with a radio button, and 'Display loads' is checked. The 'Time Step' is set to 1. 'Mode Shape (Dynamics)' is unselected. A 'Scale' slider is set to 32, with an 'Input Value' text box also containing 32. 'Display shadow of undeflected model' is checked. At the bottom, there is a 'Set as Default' checkbox, and 'Preview', 'Apply', and 'Cancel' buttons.

Click on the **Display Deflected Model (using Model View Settings)** slider to make it turn gray, if you don't want to display any deflected shape results.

In this tab you can specify that a deflected shape be drawn in the current model view based on either a static load combination (if a single or batch static solution has been done) or a mode shape (if a dynamic solution has been done). You can also display loads for the load combination at the same time as displaying the deflected shape. You can control the magnification of the deflections with the **Scale** text box and specify whether an undeflected shadow is to be shown by checking the **Display shadow of undeflected model** box. For batch solutions you must choose which combination you want to view at the top of the Results View Settings dialog.



Note: Time History Deflection diagrams only display joint translation. Joint rotation is not displayed.

When displaying the deflection diagram for a Time History load combination, you can select the **Time Step** for which to display the deflected shape. This section is grayed out if no time history load combination has been solved. Additional controls are available for the animation of Time History motion. Please refer to the [Dynamics - Time History](#) topic for more information.

To animate a particular deflected shape, first select the deflected shape (as described above) and then click the **Animate Results** icon on the **View** ribbon. See [Animate Results](#).

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Deflected**.

Help Options

RISA Tech, Inc. has, and will, put a great deal of effort into assisting you in getting your work done as quickly as possible. This includes providing several ways in which you can get help to understand the software.

Electronic Help File

The **Help File** is designed to help you get your work done as quickly as possible and is intended to provide:

- Procedures that lead users through the steps of completing tasks
- Context-sensitive Help topics that provide users with quick descriptions of items on their screens
- Troubleshooting topics that guide users through solutions to common problems
- Extensive discussions for a thorough understanding of engineering and modeling topics
- Easy access to related topics

The electronic help file can be accessed by clicking the **Help File**  button in the top right hand corner of the **Quick Access Toolbar**. A new window containing a Table of Contents is opened. Click on any item in the Table of Contents for extensive information on the topic.

Context Sensitive Help

Context Sensitive Help is help that you can access in context while working in a program. It provides you with the information you need where and when you want it.

You can get detailed help when working in a dialog, spreadsheet or graphic view by pressing **F1**. This launches a Help File window displaying the topic that is related to the window in which you are working. The topic is explained and links to related topics may also be provided.

RISA Technology Online

Our website, www.risa.com, provides various support information and documents.

Visit RISA Tech, Inc. on the web for:

- Download program [Manuals](#) (General Reference or Tutorial)
- Check our website for the latest [updates](#) - When a bug is discovered it is posted on the web site along with possible work-around procedures and/or service releases to update your software.
- Software [Verification Problems](#)

Tool-tips

Are you uncertain what a toolbar icon or button is for? Simply hold your mouse pointer over it without clicking. A **Tool-tip** is displayed that explains what the button does, should you decide to press it.

Tutorials

The comprehensive **Tutorials** guide you through using most features. They are real-world examples of building and solving models, making changes, and optimizing. This is the best way to quickly get up and running. The Tutorials are designed to be read in two ways. If you are already familiar with structural modeling in general you can skip the supporting text and read only the underlined action items to quickly move through the tutorial. If you want more thorough explanations of the modeling process you can read all or some of the supporting text as you see fit.

Hot Rolled Steel - Databases

Shapes are organized by **Shape Type** and **Database / Manufacturer**. Common shapes are supported such as; wide flanges, tubes, pipes, channels, etc. You can type in the names directly, select shapes from these databases or add your own shapes.

3D Shape Selection
?
×


General


Hot Rolled


Cold Formed


Wood


Aluminum


Stainless

Database AISC

Shape Type

WF

Tube

Pipe

Channel

Double Channel

WT

Double Angle

Single Angle

Solid Rectangle

Solid Round Bar

Tapered WF

Shape Name

- W44X335
- W44X290
- W44X262
- W44X230
- W40X655

Preview

Add

View

Edit

Copy

Delete



Save Database as default

OK

Cancel

RISA currently supports the following common Hot Rolled steel databases: AISC (United States), Australian, British, Canadian, Chilean, Chinese, European, Indian and Mexican.



Note: Older AISC shapes, which are no longer part of the AISC database, are automatically moved / exported to a new AISC_backup database during installation or update.

Select a Database Shape

To select a Hot Rolled Steel database shape:

- From the **Hot Rolled** tab on the **Section Sets** spreadsheet, or the **Primary** tab of the **Members** spreadsheet, click on the **Shape** column and then click the  ellipsis button.

- If opened from the **Section Sets** spreadsheet:

A 'Shape Selection: Hot Rolled' window opens.

Shape Selection: Hot Rolled

Database: AISC

Shape Type:

- WF
- Tube
- Pipe
- Channel
- Double Channel
- WT
- Double Angle
- Single Angle
- Solid Rectangle
- Solid Round Bar
- Tapered WF

Shape Name: Search by name

- LL8X8X9X0
- LL8X8X9X3
- LL8X8X9X6
- LL8X8X8X0
- LL8X8X8X3**

Preview:

8 in(D)

0.4 in(Spacing)

0.5 in(t)

16.4 in(W)

Buttons: Add, View, Edit, Copy, Delete

Save Database as default

Buttons: OK, Cancel

- Choose the **Database** to select from.
- Choose the **Shape Type** you want to select.
See [Available Database Shapes](#) for more information on shape types.
- In the **Shape Name** list, find and select the shape you want.
- Click **OK** in the 'Shape Selection: Hot Rolled' window to save your selection, close the window and return to the spreadsheet.

The shape you've chosen is displayed in the Shape or Section/Shape cell you

- If opened from the **Members** spreadsheet:
A 'Set Member Section Set or Shape' window opens.

- Click on the **Hot Rolled** option, if not already selected.
- Click on the **Assign Shape Directly** option.
- Click the **Start Shape** (ellipsis) button.

A Shape Selection window opens to the Hot Rolled tab. The window has the same options as the 'Shape Selection: Hot Rolled' window.

- Choose the **Database** to select from.
- Choose the **Shape Type** you want to select.

See [Available Database Shapes](#) for more information on shape types.

- f. In the **Shape Name** list, find and select the shape you want.
- g. Click **OK**.

You are returned to the 'Set Member Section Set or Shape' window in which the Start Shape now displays the shape you chose from the list in the 'Shape Selection' window.

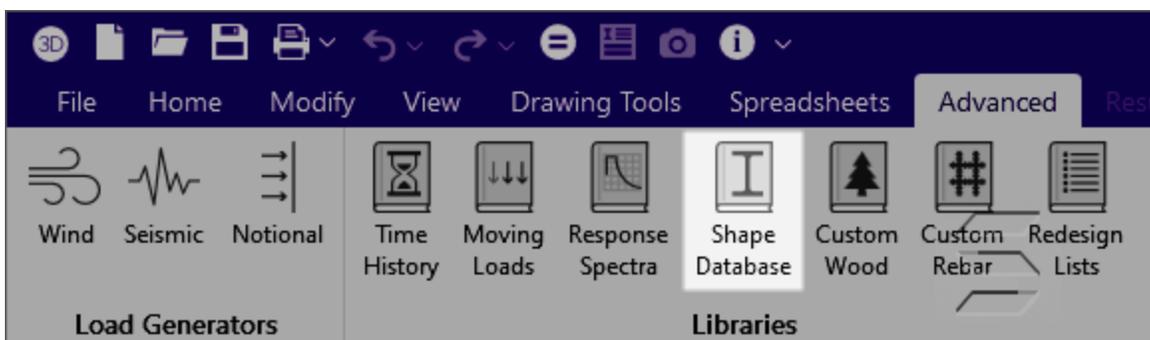
Click **OK** in the Set Member Section Set or Shape window to select the shape you chose, close the window and return to the database spreadsheet.

The shape you've chosen is displayed in the Shape or Section/Shape cell you

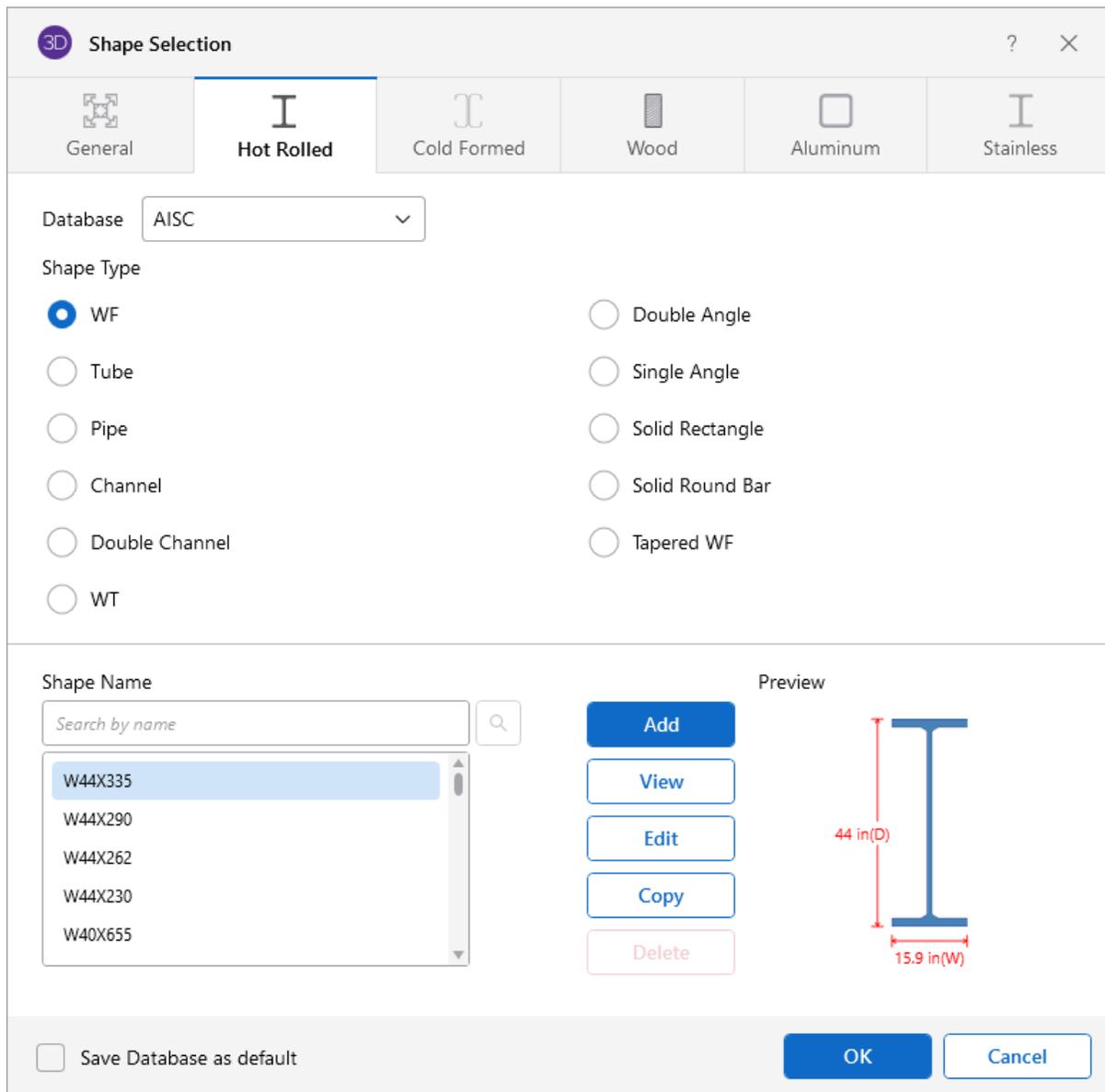
Add a Database Shape

To add a Hot Rolled Steel database shape:

1. On the **Advanced** ribbon, click the **Shape Database** icon.



The Shape Selection window opens.



2. Click on the **Hot Rolled** tab as shown in the previous image.
3. Choose the **Database** and **Shape Type** you wish to add.
See [Available Database Shapes](#) for more information on shape types.
4. Click the **Add** button.
The Add Shape window opens.

3D
Add Shape
?
✕

Basic Properties

Shape Name

d in

b_f in

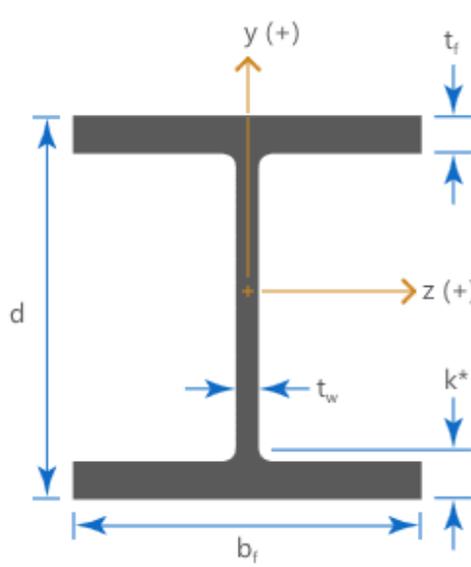
t_f in

t_w in

k_{des} in

k_{det} in

k^* is for Connection calculations only.



Calculated Properties

Area	0	in ²	r_T	0	in	<div style="background-color: #007bff; color: white; padding: 5px; border-radius: 5px; margin-bottom: 5px;">Calculate</div> <div style="background-color: #ccc; color: black; padding: 5px; border-radius: 5px; margin-bottom: 5px;">Clear</div>
I_{yy}	0	in ⁴	J	0	in ⁴	
I_{zz}	0	in ⁴	C_w	0	in ⁶	
Z_{yy}	0	in ³	W_{no}	0	in ²	
Z_{zz}	0	in ³	S_w	0	in ⁴	

Units

OK

Cancel

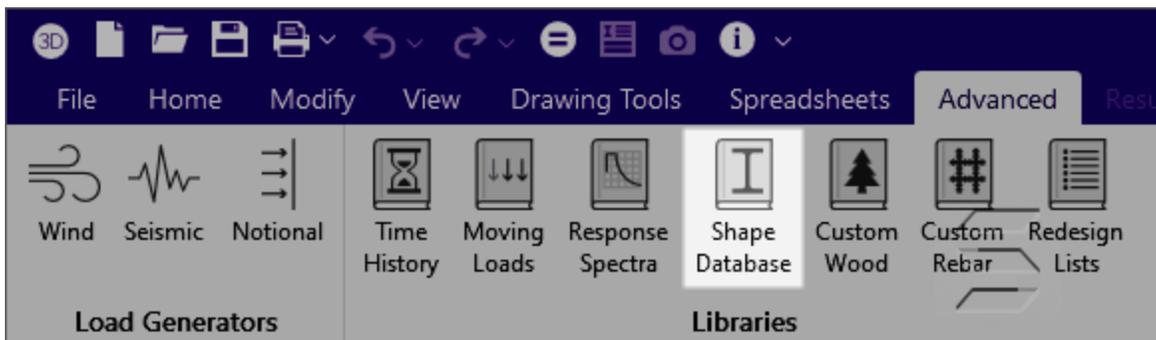
5. Under 'Basic Properties', specify a **Shape Name** for the shape and fill in the basic properties for it.
6. Click the **Calculate** button to determine the shape properties.
7. Click **OK** to accept the new shape and close the Add Shape window.

The new shape is added to the bottom of the Shape Name list in the Shape Selection window.

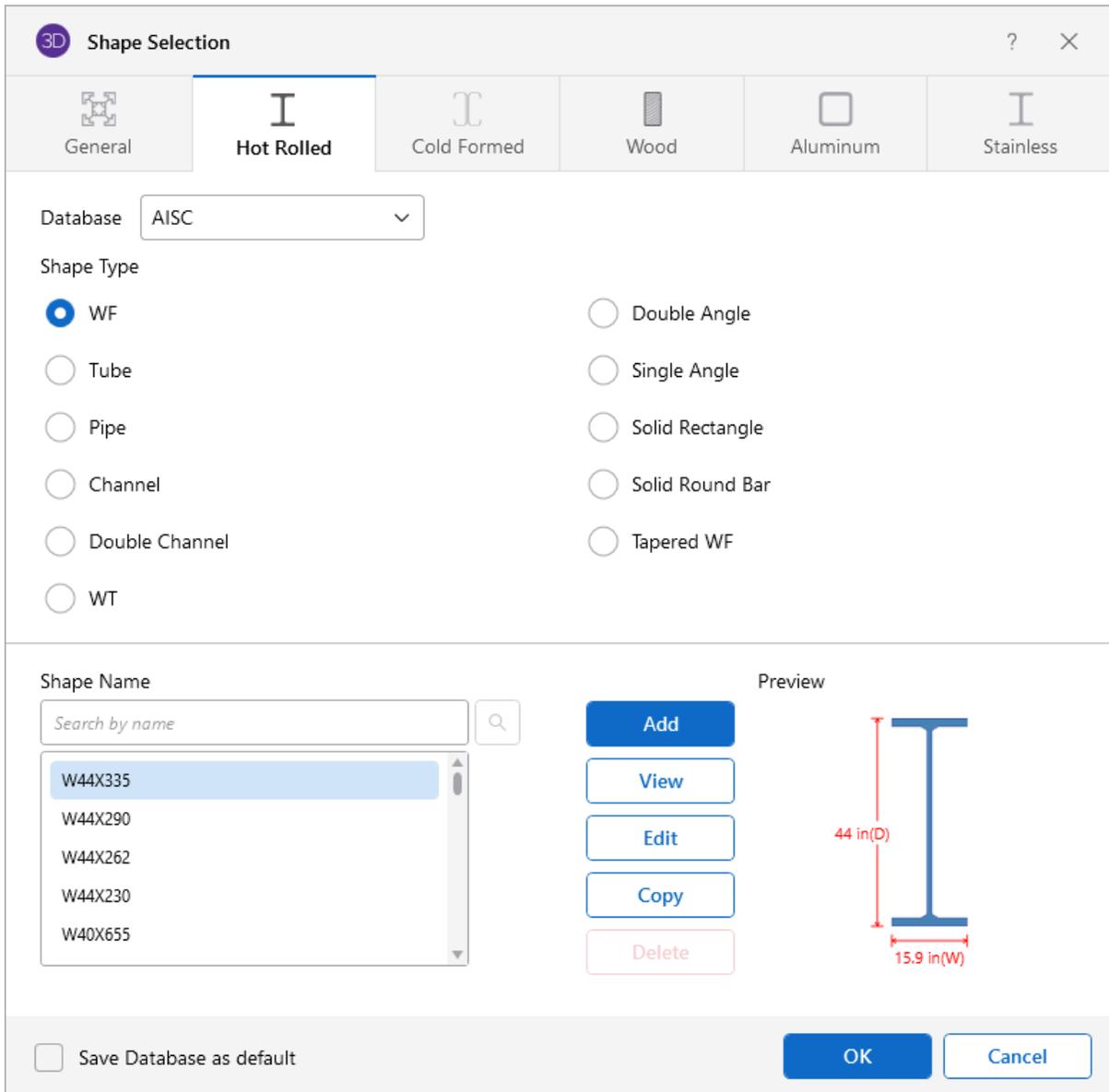
Edit a Database Shape

To edit a Hot Rolled Steel database shape:

1. Go to the **Advanced** ribbon.



2. Click on the **Shape Database** icon in the Libraries section.
The Shape Selection window opens.



3. Click on the **Hot Rolled** tab.
4. From the **Shape Name** list, choose the shape you wish to edit.
5. Click on **Copy**, if the 'Edit Shape' window opens stating that you cannot edit the default database shape.

You will be able to edit the copy, once you provide a new name for the copy and click the **Copy Shape** button and answer the question as to whether or not to allow the shape to be made available for all models.

6. Click the **Edit** button.

The shape copy opens in the Edit Shape window, as shown in the following example.

3D Edit Shape
?
×

Shape Name

Geometric Properties

d in

b_f in

t_f in

t_w in

Connection Detailing Properties

k_{des} in

k_{det} in

Section Properties

Area in²

I_{yy} in⁴

I_{zz} in⁴

Z_{yy} in³

Z_{zz} in³

C_w in⁶

W_{no} in²

S_w in⁴

r_T in

J in⁴

7. Make whatever changes you need, based on what you want to do. You have the option to either:

- Manually input the **Geometric Properties** and **Section Properties**.
- Have the program calculate the **Section Properties** based on values you input for the **Geometric Properties**.

Once an edit is made to the 'Geometric Properties' or to both the properties, you'll need to run a recalculation.

8. Click the **Re-Calc** button to have the program calculate the new **Section Properties** automatically.
9. Click **OK** to save the newly calculated, edited shape.
10. Do one of the following in the Changes to Shape Database window that appears:
 - Click **Yes**, if you want the changes to apply to all new models created on your computer.
 - Click **No**, if you want the changes to apply to ONLY the current model.

The newly calculated shape copy is saved to the Shape Name list in the Shape Selection window.

**Note:**

- New shapes are added to the bottom of the database list in a blue font, to show they were not originally from the default database.
- To delete a shape, specify the database and shape type you wish to delete and then click the **Delete** button.
- Only shapes added to the default database (shown in blue) are able to be edited or deleted.

Available Database Shapes

Wide Flange Database Shapes

For the AISC database, wide flange shapes are called out by the designation given them in the steel manuals. For example, if you wanted to use a W10x33 you would enter W10X33 as the shape name in the database shape field. M, S and HP shapes are also available. Trade Arbed shapes are called out similar to AISC shapes but with a “_ARB” suffix. I.e. to call a Trade Arbed W12X96 would enter W12X96_ARB as the shape name in the database shape field. Canadian and British shapes use the same format as the AISC shapes, but their values are metric. The depth is called out in millimeters and the mass per length is kg/meter.

Tube Database Shapes(Hollow Rectangular shapes)

The HSS tube properties are also available in the AISC database. The prefix for these tube shapes is “HSS”. The syntax is “HSSdepthXwidthXthick”, where “depth” is the tube depth, “width” is the tube width and “thick” is the tube thickness in number of 1/16ths. The nominal wall thickness is always used to call out a HSS tube, even though the design wall thickness will vary based on the manufacturing process for the tube. Tubes manufactured using the ERW process will use .93 times the nominal wall thickness as their design thickness. Tubes manufactured using the SAW process will use the full nominal thickness as their design thickness. For example, an HSS12X10X8 would be a 12” deep, 10” wide tube, and have a design wall thickness of $.465" = .93 * 1/2"$ (8/16ths). A HSS32X24X10 would be 32” deep by 24” wide, and have a design wall thickness of $5/8"$ (10/16ths)

For the Canadian database, tubes also have a “HSS” prefix and the dimensions are all called out in millimeters. British shapes use the prefix “SHS” for square tubes and “RHS” for rectangular tubes.

**Note:**

- The prefix for older AISC tube shapes is "TU". These shapes reflect the properties that were published with the older AISC 9th edition. As such, these shapes may only exist in the AISC_backup database.
- Tubes using the TU prefix will have a design wall thickness of the nominal wall thickness. The syntax is "TUdepthXwidthXthick", where "depth" is the tube depth, "width" is the tube width and "thick" is the tube thickness in number of 1/16ths. For example, TU16X12X8 would be a 16" deep, 12" wide tube with a thickness of 1/2" (8/16ths).

Pipe Database Shapes

Pipe shapes, which are hollow circular shapes, are entered as on-line shapes. The syntax for these shapes is "PIdiaXthick", where "dia" is the pipe outside diameter and "thick" is the pipe thickness (in inches or centimeters). For example (assuming US Standard units), PI10X.5 would be a 10" diameter pipe with a wall thickness of 1/2".

Channel and Double Channel Database Shapes

Channel shapes are entered with the "C" or "MC" prefix. For example C15X50 would be a valid entry. For Canadian and British shapes, the depth is called out in millimeters and the mass per length is in kg/meter. Double channels are entered with the "2C" or "2MC" prefix.

Tee Database Shapes

The Tee shapes are entered with the "WT", "MT" or "ST" prefix. For example WT15X74 would be a valid entry. For Canadian and British shapes, the depth is called out in millimeters and the mass per length is in kg/meters.

Tapered Wide Flange Shapes

Tapered Wide Flange shapes are called out by referring to the shape name that was given when it was defined in the database shape editor. Tapered WF shapes can only be defined as database shapes using the "ADD" shape function in the database editor.

Tapered WF shapes are special in that the cross sectional properties change along the length of the member. This is as opposed to prismatic members, which have the same cross sectional properties along their length. (All other shapes are prismatic members). Keep this in mind when defining the I and J joints for tapered shapes. To obtain alternate tapered shape suggestions it is best to define all Tapered WF members consistently in the shape database and handle orientation with the I and J joints.

The Tapered WF shape can also be used to define a prismatic WF with unequal flanges. Tapered wide flange shapes can taper all cross section properties independently and can also have unequal top and bottom flanges. Each basic property is assumed to taper linearly from the Start value to the End value. Shape properties like the area and the moments of inertia will be computed at any

required intermediate point from the linearly interpolated basic properties. (This means that the area and moments of inertia will probably NOT vary linearly along the member length). Intermediate shape properties are used to calculate the member stiffness and stresses. The member stiffness is computed internally from a series of piecewise prismatic sections. The error in the member stiffness computed in this manner, as opposed to the theoretically "correct" stiffness, is always less than 10%.

Note that tapered members are always treated as physical members – not finite members. All the rules and behaviors described for physical members always apply to tapered members even if the physical member flag is not set for those members. See [Physical Members](#) to learn more about this feature.

Add a Tapered Wide Flange

To add a Tapered Wide Flange to the database:

- Click the **Tapered WF** shape type, then click **Add**. Enter the basic shape properties at the **Start** and **End** locations and all the necessary parameters calculated for analysis and design at the member end points and at all the required intermediate locations.



Note: To enter a prismatic WF member with unequal top and bottom flanges, just make sure that the shape properties are the same at the Start and End points. The top and bottom flange information is entered independently.

Double Angle Database Shapes

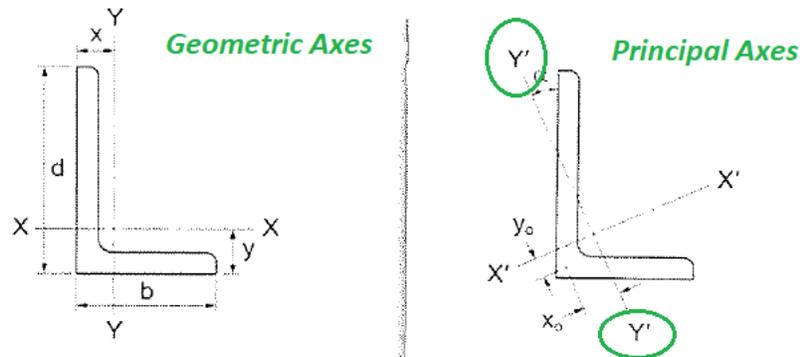
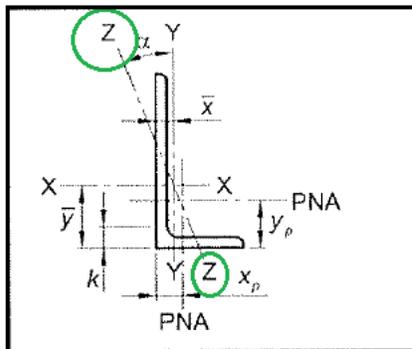
These shapes are entered with the prefix "LL". The syntax is "LLbackXflangeXthickXspace" where "back" is the back to back leg length, "flange" is the single angle flange leg length, "thick" is the angle thickness in number of 1/16ths and "space" is the space between the angles in 1/8ths. For example, LL6X3.5X5X3 would be L6X3.5 angles 5/16" thick, long legs back to back with a spacing of 3/8". For the Canadian and British shapes, all the dimensions are called out in millimeters.

Single Angle Database Shapes

Angles are entered with an "L" prefix. The syntax is "LlongXshortXthick", where "long" is the long leg length, "short" is the short leg length, and "thick" is the thickness, in number of 1/16ths. For example, L9X4X8 is a 9" by 4" angle 1/2" (8/16ths) thick. The thickness is entered as 8, because the number of 1/16ths in 1/2 is 8. For the Canadian and British shapes, all the dimensions are called out in millimeters.



Note: **rZ** is the Radius of Gyration along the rotated (Principal) vertical axis (local y). The Canadian CSA code calls this variable r_y' but RISA calls it **rZ** based on the naming convention of the AISC 360.

Per CSA S16-16 (11th edition):**Per AISC 360-16 (15th edition):****Solid Rectangle 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 Round Bar 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.

Buckling-Restrained Brace Database Shapes

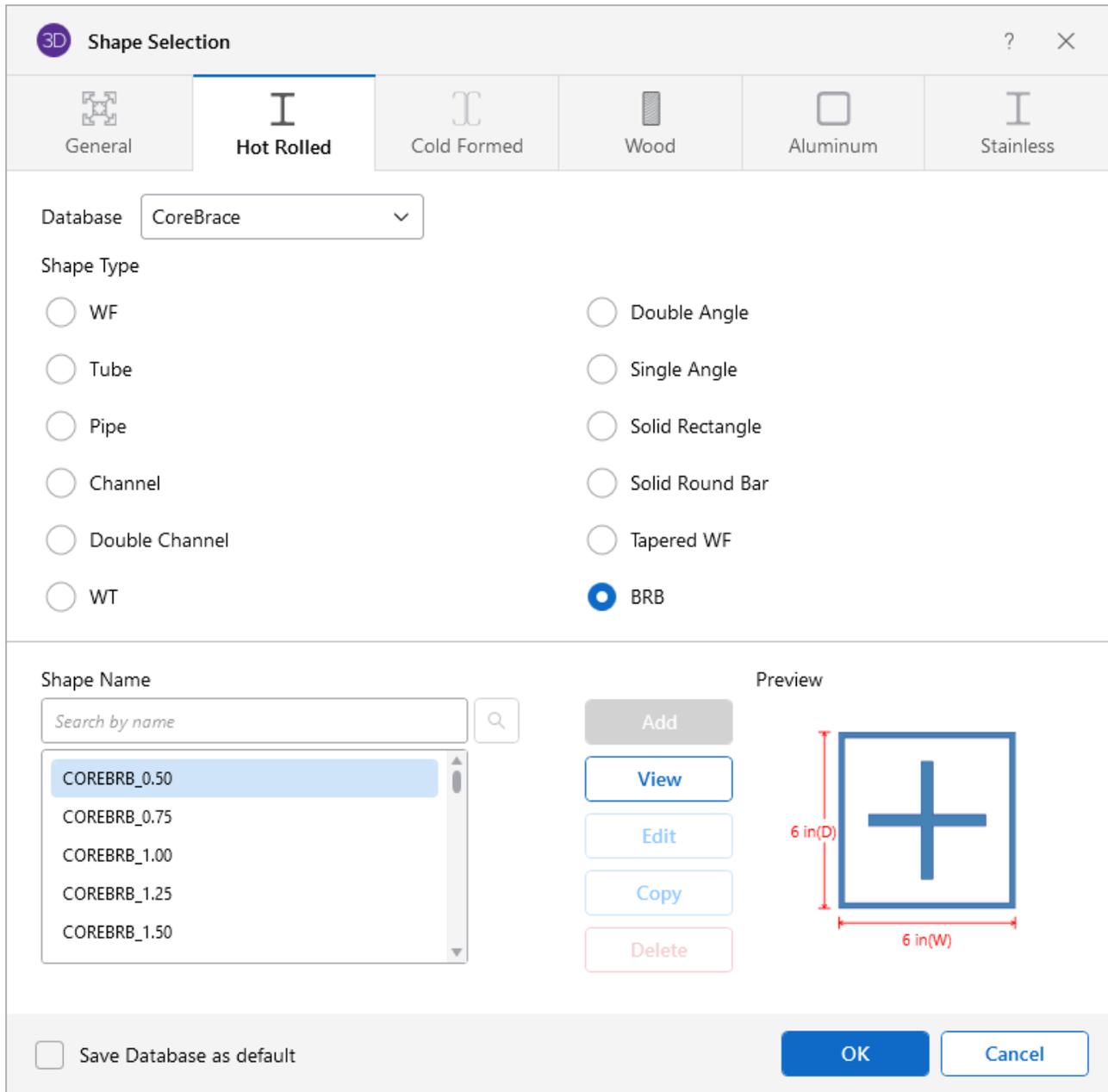


Note: The Buckling-restrained brace (BRB) shape database is only available for RISAFloor/RISA-3D integration models.

BRB shapes can be selected by choosing the BRB shape type and one of the BRB-related databases. Currently, only CoreBrace database is supported.

When CoreBrace database is chosen, the syntax for CoreBrace BRB shapes is “CoreBRB_area”, where “CoreBRB” is the prefix for the manufacturer and shape, and “area” is the steel core area. For example, CoreBRB_0.50 refers to a BRB shape manufactured by CoreBrace with a 0.50 in² steel core area. Other properties such as the casing size of the brace are based on the data provided by the manufacturers.

BRBs are not prefabricated with specific core sizes, casing sizes, or brace lengths. Instead, they are custom-fabricated for each project. Therefore, the BRB properties in RISA-3D may not be the final and shall be carefully reviewed by BRB manufacturers.



Hot Rolled Steel - Databases

Shapes are organized by **Shape Type** and **Database / Manufacturer**. Common shapes are supported such as; wide flanges, tubes, pipes, channels, etc. You can type in the names directly, select shapes from these databases or add your own shapes.

3D Shape Selection
? X


 General


Hot Rolled


 Cold Formed


 Wood


 Aluminum


 Stainless

Database AISC

Shape Type

WF

Double Angle

Tube

Single Angle

Pipe

Solid Rectangle

Channel

Solid Round Bar

Double Channel

Tapered WF

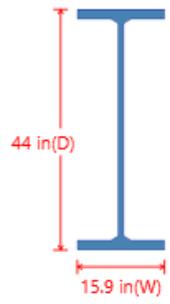
WT

Shape Name

Search by name Q

W44X335
 W44X290
 W44X262
 W44X230
 W40X655

Preview



Save Database as default

RISA currently supports the following common Hot Rolled steel databases: AISC (United States), Australian, British, Canadian, Chilean, Chinese, European, Indian and Mexican.

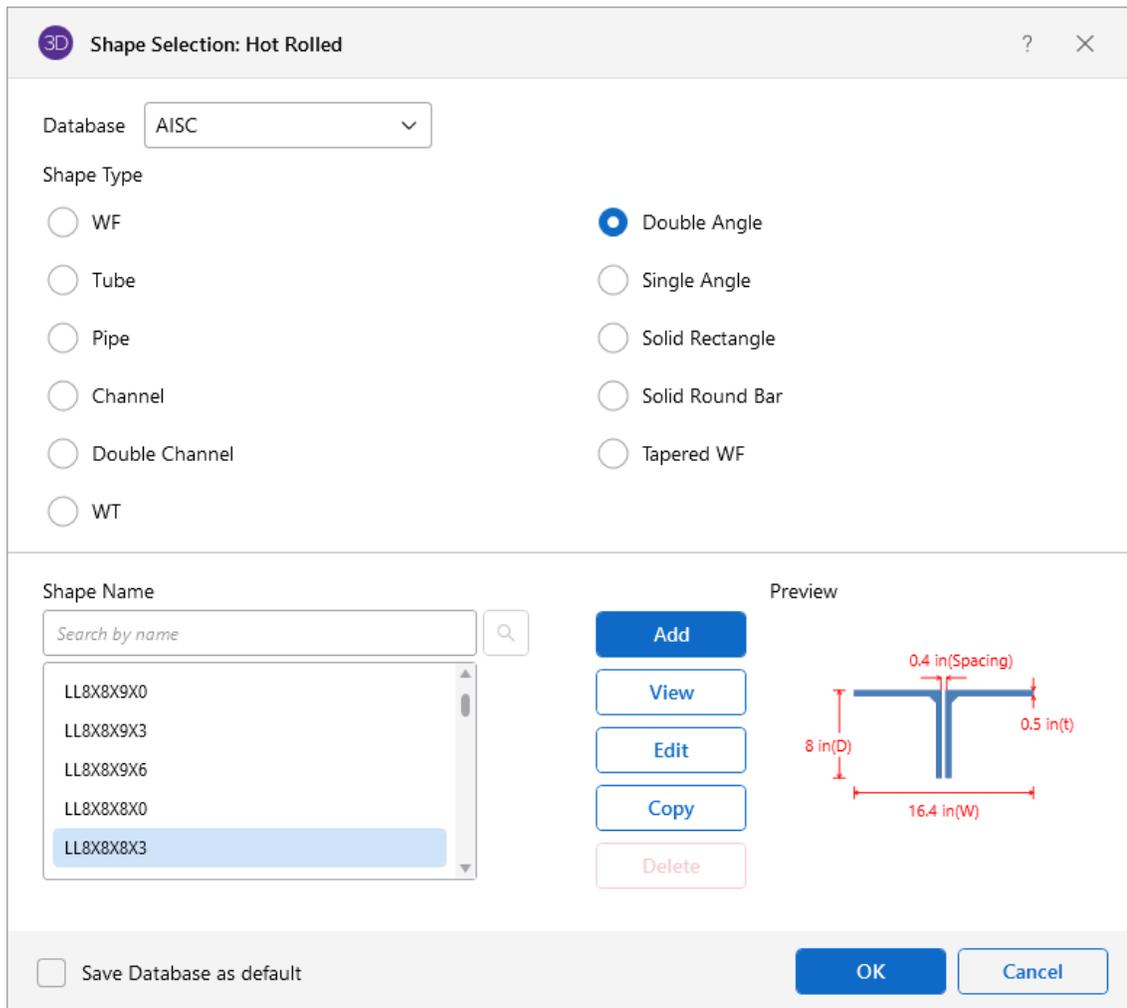


Note: Older AISC shapes, which are no longer part of the AISC database, are automatically moved / exported to a new AISC_backup database during installation or update.

Select a Database Shape

To select a Hot Rolled Steel database shape:

- From the **Hot Rolled** tab on the **Section Sets** spreadsheet, or the **Primary** tab of the **Members** spreadsheet, click on the **Shape** column and then click the  ellipsis button.
 - If opened from the **Section Sets** spreadsheet:
A 'Shape Selection: Hot Rolled' window opens.



3D Shape Selection: Hot Rolled

Database: AISC

Shape Type:

- WF
- Tube
- Pipe
- Channel
- Double Channel
- WT
- Double Angle
- Single Angle
- Solid Rectangle
- Solid Round Bar
- Tapered WF

Shape Name: Search by name

- LL8X8X9X0
- LL8X8X9X3
- LL8X8X9X6
- LL8X8X8X0
- LL8X8X8X3

Preview:

8 in(D)

16.4 in(W)

0.4 in(Spacing)

0.5 in(t)

Buttons: Add, View, Edit, Copy, Delete

Save Database as default:

Buttons: OK, Cancel

- Choose the **Database** to select from.
- Choose the **Shape Type** you want to select.
See [Available Database Shapes](#) for more information on shape types.
- In the **Shape Name** list, find and select the shape you want.

- d. Click **OK** in the 'Shape Selection: Hot Rolled' window to save your selection, close the window and return to the spreadsheet.

The shape you've chosen is displayed in the Shape or Section/Shape cell you

- If opened from the **Members** spreadsheet:

A 'Set Member Section Set or Shape' window opens.

- Click on the **Hot Rolled** option, if not already selected.
- Click on the **Assign Shape Directly** option.
- Click the **Start Shape**  (ellipsis) button.

A Shape Selection window opens to the Hot Rolled tab. The window has the same options as the 'Shape Selection: Hot Rolled' window.

- Choose the **Database** to select from.
- Choose the **Shape Type** you want to select.
See [Available Database Shapes](#) for more information on shape types.
- In the **Shape Name** list, find and select the shape you want.
- Click **OK**.

You are returned to the 'Set Member Section Set or Shape' window in which the Start Shape now displays the shape you chose from the list in the 'Shape Selection' window.

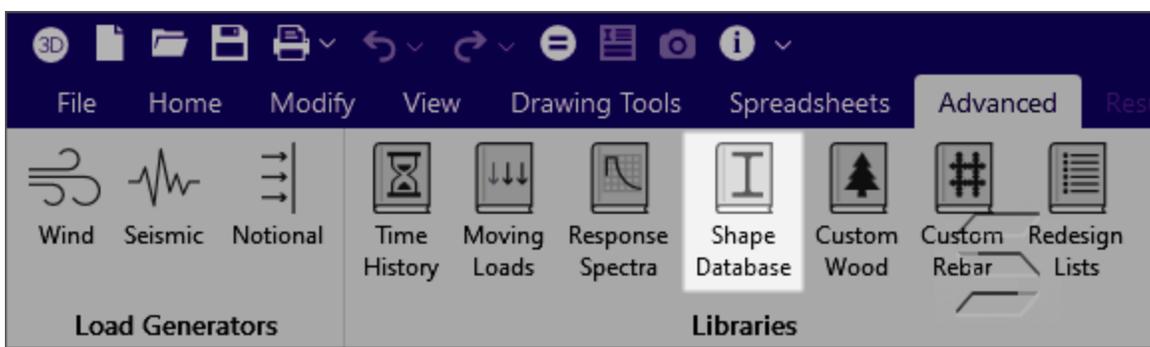
Click **OK** in the Set Member Section Set or Shape window to select the shape you chose, close the window and return to the database spreadsheet.

The shape you've chosen is displayed in the Shape or Section/Shape cell you

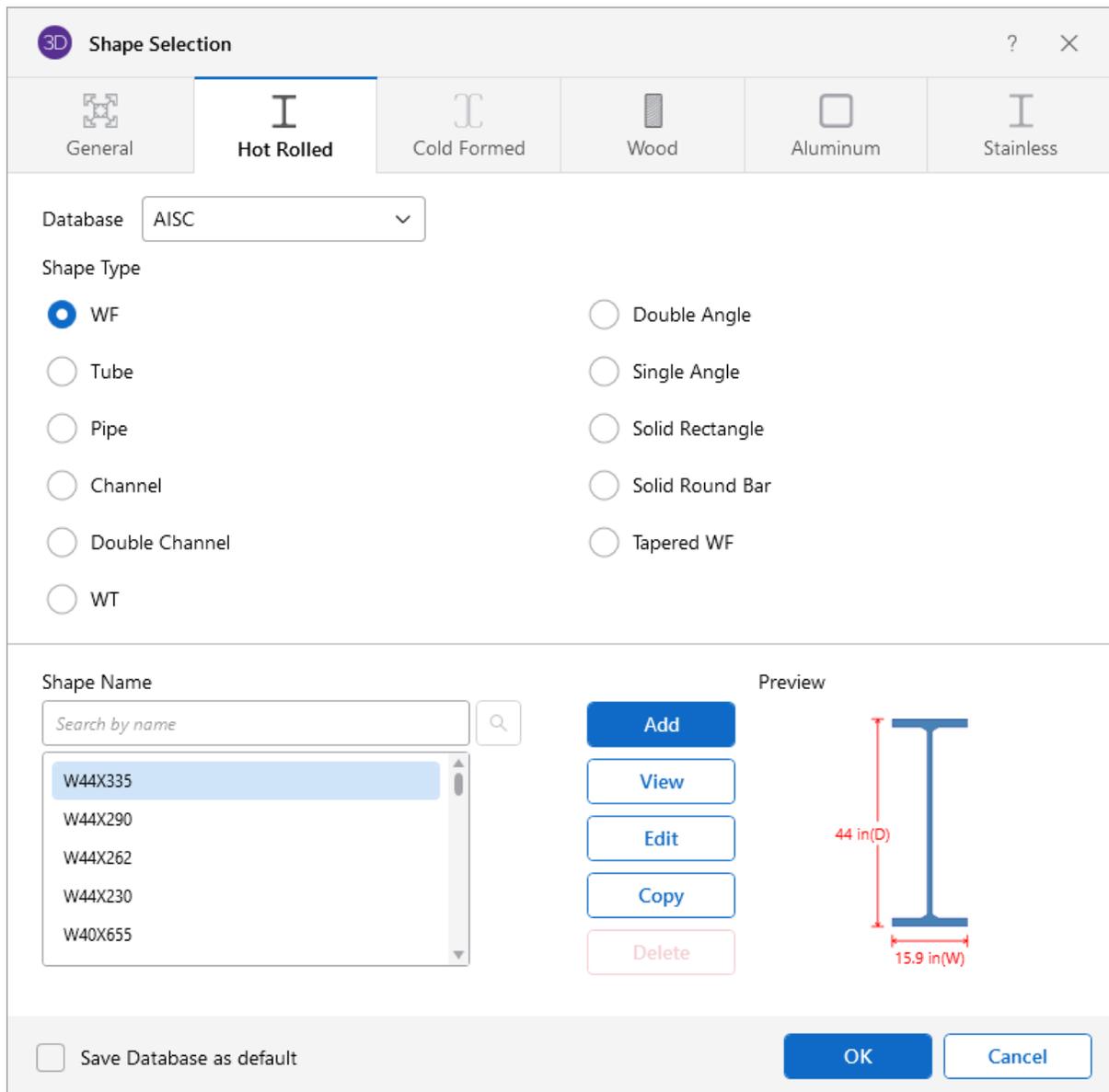
Add a Database Shape

To add a Hot Rolled Steel database shape:

1. On the **Advanced** ribbon, click the **Shape Database** icon.



The Shape Selection window opens.



2. Click on the **Hot Rolled** tab as shown in the previous image.
3. Choose the **Database** and **Shape Type** you wish to add.
See [Available Database Shapes](#) for more information on shape types.
4. Click the **Add** button.
The Add Shape window opens.

3D
Add Shape
?
✕

Basic Properties

Shape Name

d in

b_f in

t_f in

t_w in

k_{des} in

k_{det} in

k^* is for Connection calculations only.

Calculated Properties

Area	0	in ²	r_T	0	in	<div style="margin-bottom: 5px; background-color: #007bff; color: white; padding: 5px 15px; border-radius: 3px;">Calculate</div> <div style="background-color: #ccc; padding: 5px 15px; border-radius: 3px; opacity: 0.5;">Clear</div>
I_{yy}	0	in ⁴	J	0	in ⁴	
I_{zz}	0	in ⁴	C_w	0	in ⁶	
Z_{yy}	0	in ³	W_{no}	0	in ²	
Z_{zz}	0	in ³	S_w	0	in ⁴	

Units

OK

Cancel

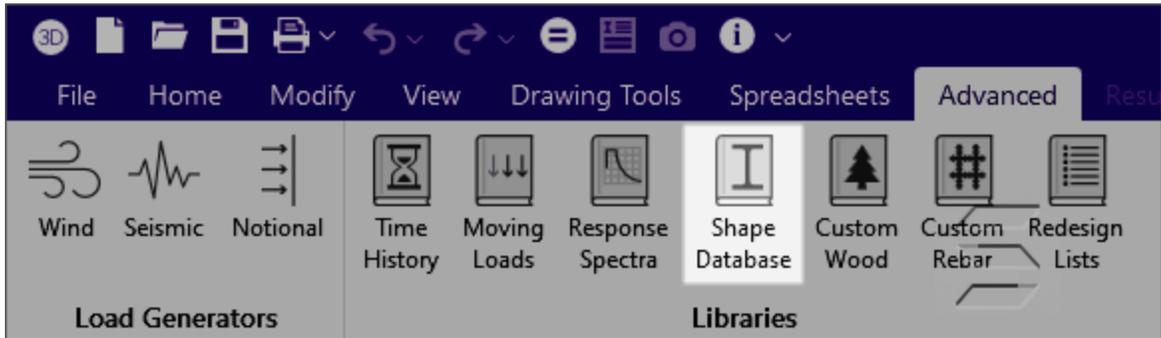
5. Under 'Basic Properties', specify a **Shape Name** for the shape and fill in the basic properties for it.
6. Click the **Calculate** button to determine the shape properties.
7. Click **OK** to accept the new shape and close the Add Shape window.

The new shape is added to the bottom of the Shape Name list in the Shape Selection window.

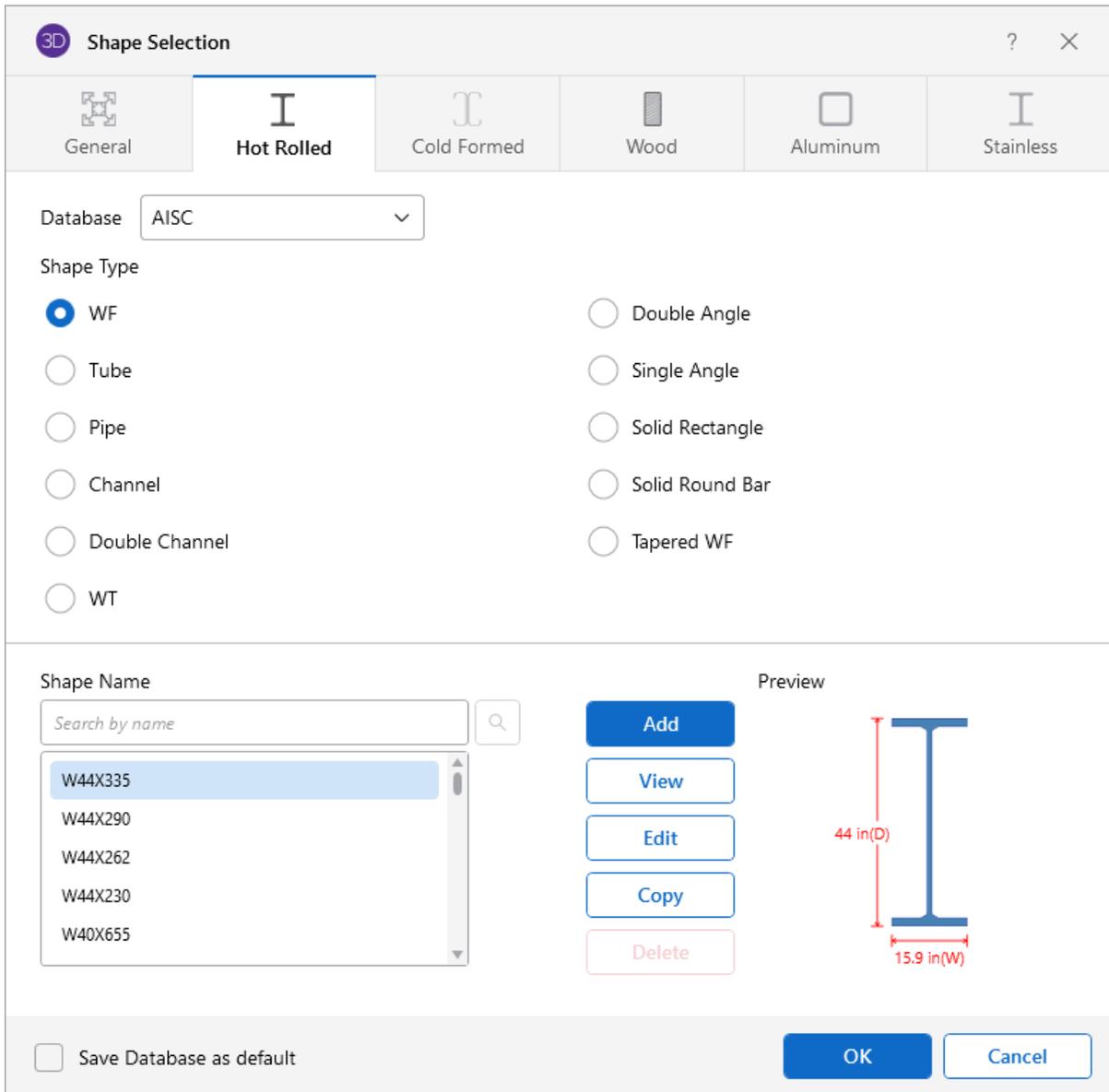
Edit a Database Shape

To edit a Hot Rolled Steel database shape:

1. Go to the **Advanced** ribbon.



2. Click on the **Shape Database** icon in the Libraries section.
The Shape Selection window opens.



3. Click on the **Hot Rolled** tab.
4. From the **Shape Name** list, choose the shape you wish to edit.
5. Click on **Copy**, if the 'Edit Shape' window opens stating that you cannot edit the default database shape.

You will be able to edit the copy, once you provide a new name for the copy and click the **Copy Shape** button and answer the question as to whether or not to allow the shape to be made available for all models.

6. Click the **Edit** button.

The shape copy opens in the Edit Shape window, as shown in the following example.

3D Edit Shape
?
×

Shape Name

Geometric Properties

d in

b_f in

t_f in

t_w in

Connection Detailing Properties

k_{des} in

k_{det} in

Section Properties

Area in²

I_{yy} in⁴

I_{zz} in⁴

Z_{yy} in³

Z_{zz} in³

C_w in⁶

W_{no} in²

S_w in⁴

r_T in

J in⁴

7. Make whatever changes you need, based on what you want to do. You have the option to either:
 - Manually input the **Geometric Properties** and **Section Properties**.
 - Have the program calculate the **Section Properties** based on values you input for the **Geometric Properties**.

Once an edit is made to the 'Geometric Properties' or to both the properties, you'll need to run a recalculation.

8. Click the **Re-Calc** button to have the program calculate the new **Section Properties** automatically.
9. Click **OK** to save the newly calculated, edited shape.
10. Do one of the following in the Changes to Shape Database window that appears:
 - Click **Yes**, if you want the changes to apply to all new models created on your computer.
 - Click **No**, if you want the changes to apply to ONLY the current model.

The newly calculated shape copy is saved to the Shape Name list in the Shape Selection window.

**Note:**

- New shapes are added to the bottom of the database list in a blue font, to show they were not originally from the default database.
- To delete a shape, specify the database and shape type you wish to delete and then click the **Delete** button.
- Only shapes added to the default database (shown in blue) are able to be edited or deleted.

Available Database Shapes

Wide Flange Database Shapes

For the AISC database, wide flange shapes are called out by the designation given them in the steel manuals. For example, if you wanted to use a W10x33 you would enter W10X33 as the shape name in the database shape field. M, S and HP shapes are also available. Trade Arbed shapes are called out similar to AISC shapes but with a “_ARB” suffix. I.e. to call a Trade Arbed W12X96 would enter W12X96_ARB as the shape name in the database shape field. Canadian and British shapes use the same format as the AISC shapes, but their values are metric. The depth is called out in millimeters and the mass per length is kg/meter.

Tube Database Shapes(Hollow Rectangular shapes)

The HSS tube properties are also available in the AISC database. The prefix for these tube shapes is "HSS". The syntax is "HSSdepthXwidthXthick", where "depth" is the tube depth, "width" is the tube width and "thick" is the tube thickness in number of 1/16ths. The nominal wall thickness is always used to call out a HSS tube, even though the design wall thickness will vary based on the manufacturing process for the tube. Tubes manufactured using the ERW process will use .93 times the nominal wall thickness as their design thickness. Tubes manufactured using the SAW process will use the full nominal thickness as their design thickness. For example, an HSS12X10X8 would be a 12" deep, 10" wide tube, and have a design wall thickness of $.465" = .93 * 1/2"$ (8/16ths) . A HSS32X24X10 would be 32" deep by 24" wide, and have a design wall thickness of $5/8"$ (10/16ths)

For the Canadian database, tubes also have a “HSS” prefix and the dimensions are all called out in millimeters. British shapes use the prefix “SHS” for square tubes and “RHS” for rectangular tubes.

**Note:**

- The prefix for older AISC tube shapes is "TU". These shapes reflect the properties that were published with the older AISC 9th edition. As such, these shapes may only exist in the AISC_backup database.
- Tubes using the TU prefix will have a design wall thickness of the nominal wall thickness. The syntax is "TUdepthXwidthXthick", where "depth" is the tube depth, "width" is the tube width and "thick" is the tube thickness in number of 1/16ths. For example, TU16X12X8 would be a 16" deep, 12" wide tube with a thickness of 1/2" (8/16ths).

Pipe Database Shapes

Pipe shapes, which are hollow circular shapes, are entered as on-line shapes. The syntax for these shapes is "PIdiaXthick", where "dia" is the pipe outside diameter and "thick" is the pipe thickness (in inches or centimeters). For example (assuming US Standard units), PI10X.5 would be a 10" diameter pipe with a wall thickness of 1/2".

Channel and Double Channel Database Shapes

Channel shapes are entered with the "C" or "MC" prefix. For example C15X50 would be a valid entry. For Canadian and British shapes, the depth is called out in millimeters and the mass per length is in kg/meter. Double channels are entered with the "2C" or "2MC" prefix.

Tee Database Shapes

The Tee shapes are entered with the "WT", "MT" or "ST" prefix. For example WT15X74 would be a valid entry. For Canadian and British shapes, the depth is called out in millimeters and the mass per length is in kg/meters.

Tapered Wide Flange Shapes

Tapered Wide Flange shapes are called out by referring to the shape name that was given when it was defined in the database shape editor. Tapered WF shapes can only be defined as database shapes using the "ADD" shape function in the database editor.

Tapered WF shapes are special in that the cross sectional properties change along the length of the member. This is as opposed to prismatic members, which have the same cross sectional properties along their length. (All other shapes are prismatic members). Keep this in mind when defining the I and J joints for tapered shapes. To obtain alternate tapered shape suggestions it is best to define all Tapered WF members consistently in the shape database and handle orientation with the I and J joints.

The Tapered WF shape can also be used to define a prismatic WF with unequal flanges. Tapered wide flange shapes can taper all cross section properties independently and can also have unequal

top and bottom flanges. Each basic property is assumed to taper linearly from the Start value to the End value. Shape properties like the area and the moments of inertia will be computed at any required intermediate point from the linearly interpolated basic properties. (This means that the area and moments of inertia will probably NOT vary linearly along the member length). Intermediate shape properties are used to calculate the member stiffness and stresses. The member stiffness is computed internally from a series of piecewise prismatic sections. The error in the member stiffness computed in this manner, as opposed to the theoretically "correct" stiffness, is always less than 10%.

Note that tapered members are always treated as physical members – not finite members. All the rules and behaviors described for physical members always apply to tapered members even if the physical member flag is not set for those members. See [Physical Members](#) to learn more about this feature.

Add a Tapered Wide Flange

To add a Tapered Wide Flange to the database:

- Click the **Tapered WF** shape type, then click **Add**. Enter the basic shape properties at the **Start** and **End** locations and all the necessary parameters calculated for analysis and design at the member end points and at all the required intermediate locations.



Note: To enter a prismatic WF member with unequal top and bottom flanges, just make sure that the shape properties are the same at the Start and End points. The top and bottom flange information is entered independently.

Double Angle Database Shapes

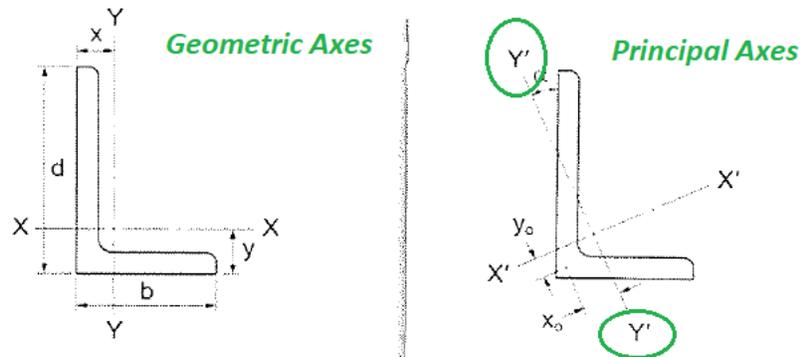
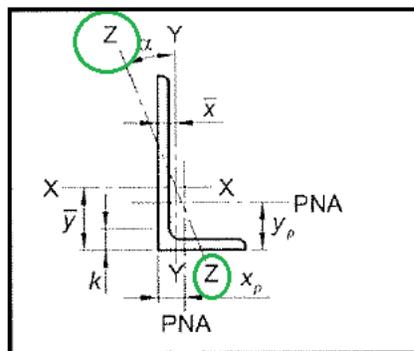
These shapes are entered with the prefix "LL". The syntax is "LLbackXflangeXthickXspace" where "back" is the back to back leg length, "flange" is the single angle flange leg length, "thick" is the angle thickness in number of 1/16ths and "space" is the space between the angles in 1/8ths. For example, LL6X3.5X5X3 would be L6X3.5 angles 5/16" thick, long legs back to back with a spacing of 3/8". For the Canadian and British shapes, all the dimensions are called out in millimeters.

Single Angle Database Shapes

Angles are entered with an "L" prefix. The syntax is "LlongXshortXthick", where "long" is the long leg length, "short" is the short leg length, and "thick" is the thickness, in number of 1/16ths. For example, L9X4X8 is a 9" by 4" angle 1/2" (8/16ths) thick. The thickness is entered as 8, because the number of 1/16ths in 1/2 is 8. For the Canadian and British shapes, all the dimensions are called out in millimeters.



Note: **rZ** is the Radius of Gyration along the rotated (Principal) vertical axis (local y). The Canadian CSA code calls this variable r_y' but RISA calls it **rZ** based on the naming convention of the AISC 360.

Per CSA S16-16 (11th edition):**Per AISC 360-16 (15th edition):****Solid Rectangle 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 Round Bar 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.

Buckling-Restrained Brace Database Shapes



Note: The Buckling-restrained brace (BRB) shape database is only available for RISAFloor/RISA-3D integration models.

BRB shapes can be selected by choosing the BRB shape type and one of the BRB-related databases. Currently, only CoreBrace database is supported.

When CoreBrace database is chosen, the syntax for CoreBrace BRB shapes is “CoreBRB_area”, where “CoreBRB” is the prefix for the manufacturer and shape, and “area” is the steel core area. For example, CoreBRB_0.50 refers to a BRB shape manufactured by CoreBrace with a 0.50 in² steel core area. Other properties such as the casing size of the brace are based on the data provided by the manufacturers.

BRBs are not prefabricated with specific core sizes, casing sizes, or brace lengths. Instead, they are custom-fabricated for each project. Therefore, the BRB properties in RISA-3D may not be the final and shall be carefully reviewed by BRB manufacturers.

3D Shape Selection
? X


General


Hot Rolled


Cold Formed


Wood


Aluminum


Stainless

Database CoreBrace

Shape Type

<input type="radio"/> WF	<input type="radio"/> Double Angle
<input type="radio"/> Tube	<input type="radio"/> Single Angle
<input type="radio"/> Pipe	<input type="radio"/> Solid Rectangle
<input type="radio"/> Channel	<input type="radio"/> Solid Round Bar
<input type="radio"/> Double Channel	<input type="radio"/> Tapered WF
<input type="radio"/> WT	<input checked="" type="radio"/> BRB

Shape Name

- COREBRB_0.50
- COREBRB_0.75
- COREBRB_1.00
- COREBRB_1.25
- COREBRB_1.50

Preview

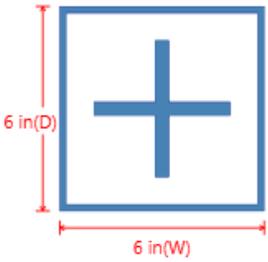
Add

View

Edit

Copy

Delete



Save Database as default

OK

Cancel

Hot Rolled Steel - Design

Design Codes

Full code checking member optimization can be applied to standard steel shapes based on the following codes:

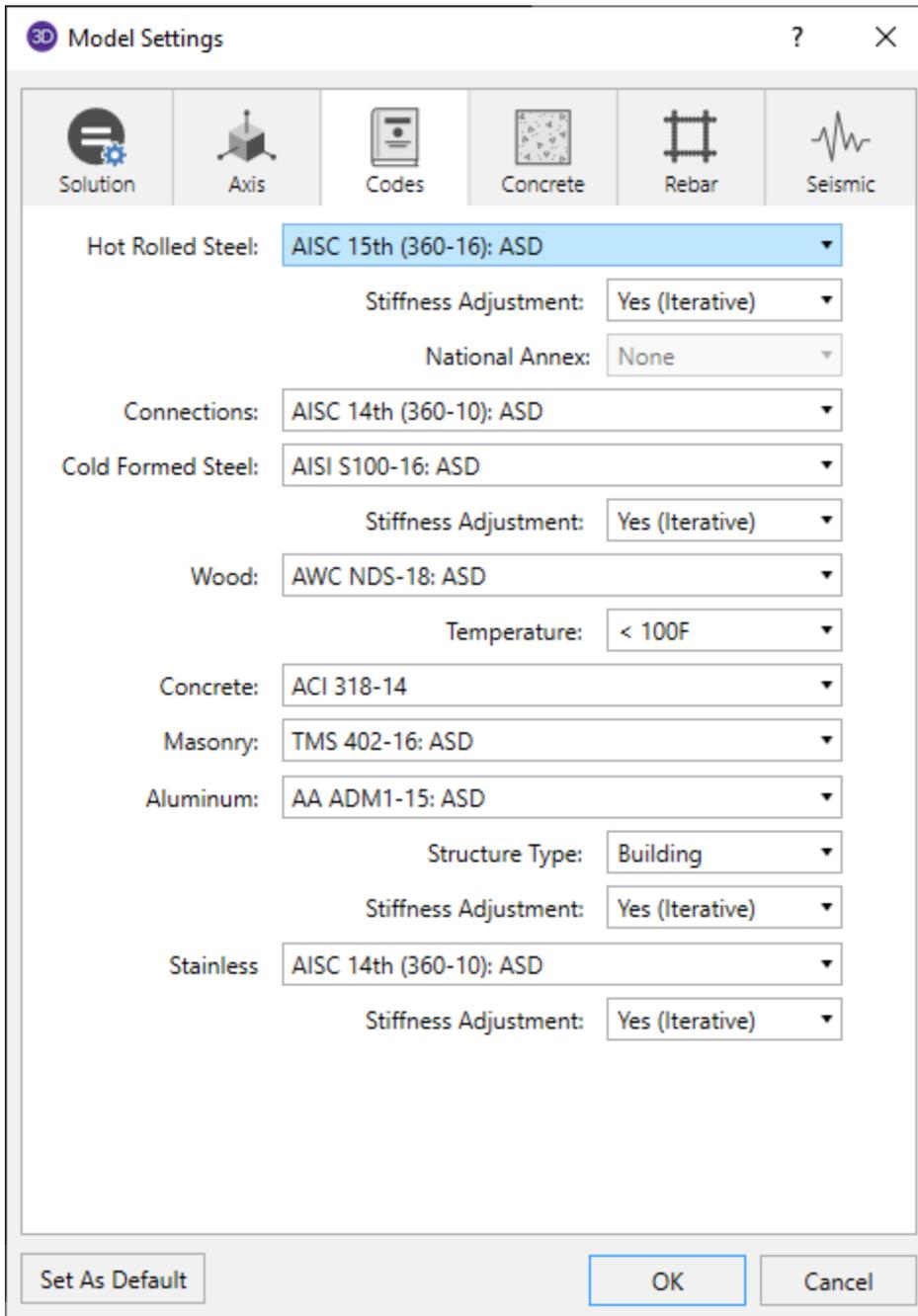
- United States:
 - AISC 360-16 (15th Edition) ASD & LRFD
 - AISC 360-10 (14th Edition) ASD & LRFD
 - AISC 360-05 (13th Edition) ASD & LRFD
 - AISC LRFD (2nd and 3rd Editions)
 - AISC ASD (9th Edition)
- Canada:
 - CSA S16-14
 - CSA S16-09
 - CSA S16-05
 - CSA S16-01
 - CSA S16.1-94
- Europe/Great Britain:
 - EN 1993-1-1: 2014 (including the U.K. National Annex provisions)
 - EN 1993-1-1: 2005 (including the U.K. National Annex provisions)
 - ENV 1993-1-1: 1992
 - BS 5950-1: 2000
- India:
 - IS 800: 2007
 - IS 800: 1998
- Australia:
 - AS 4100-1998
- New Zealand:
 - NZS 3404: 1997

The calculations performed encompass all the code requirements (including the local buckling criteria for slender compression elements in Appendix B of the 2nd, 3rd, and 9th Edition AISC codes) except those noted in the *Limitations* section of this document

Apply a Steel Design Code

To apply a Hot Rolled Steel design code:

1. Open the **Model Settings** window.
(Home ribbon > Model section > Settings link.)



2. Click on the **Codes** tab.
3. Choose the steel code from the **Hot Rolled Steel, Codes** drop down list.
4. Click **Apply** to save your choice and close the window.
5. Open the **Members** spreadsheet.
6. Click on the **Hot Rolled** tab.

7. Enter the appropriate bracing information and factors.



Note: For code checking to be performed on a member, the member must be defined with a database shape on the **Section Sets** or **Members** spreadsheet.

Design Parameters Spreadsheet

The **Hot Rolled Steel Design Parameters** spreadsheet records the design parameters for the steel code checks and may be accessed by selecting **Members** on the **Spreadsheets** menu and then selecting the **Hot Rolled** tab. These parameters are defined for each individual member and may also be assigned graphically. See [Modifying Member Design](#) to learn how to do this.

Hot Rolled Steel Design Parameters														
Primary	Advanced	Hot Rolled	Cold Formed	Wood	Concrete Beam	Concrete Column	Aluminum	Stainless	RISAConnection					
	Label	Shape	Length [ft]	Lb y-y [ft]	Lb z-z [ft]	Lcomp top [ft]	Lcomp bot [ft]	L-Torque [ft]	K y-y	K z-z	Cb	Function	Channel Conn.	a [ft]
1	M70	W10X12	18	Segment		1						Gravity	N/A	N/A
2	M71	W10X17	18	Segment		1						Lateral	N/A	N/A
3	M72	W10X17	18	Segment		1						Lateral	N/A	N/A
4	M73	W10X12	18	Segment		1						Lateral	N/A	N/A
5	M74	W10X17	18	Segment		1						Lateral	N/A	N/A
6	M75	W10X12	18	Segment		1						Lateral	N/A	N/A
7	M76	W10X17	18	Segment		1						Lateral	N/A	N/A
8	M77	W10X17	18	Segment		1						Lateral	N/A	N/A
9	M78	W10X12	18	Segment		1						Gravity	N/A	N/A
10	M79	W10X17	18	Segment		1						Lateral	N/A	N/A
11	M80	W10X12	18	Segment		1						Gravity	N/A	N/A
12	M81	W10X12	18	Segment		1						Gravity	N/A	N/A

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 separately in the [Tapered Members](#) section.

The following table provides descriptions for some of the columns in the spreadsheet.

Design Parameters Spreadsheet - Columns

Label	You can assign a unique Label to all of the members. Each label must be unique, so if you try to enter the same label more than once you will get an error message. You can relabel at any time with the Relabel options on the Tools menu.
Shape	The member Shape or Section Set is reported in the second column. This value is listed for reference only and may not be edited as it is dictated by the entry in the Section/Shape column on the Primary tab.
Length	The member Length is reported in the third column. This value may not be edited as it is dependent on the member end coordinates listed on the Primary Data tab. It is listed here as a reference for unbraced lengths which are discussed in the next section.

General Design Parameters

Unbraced Lengths

See the [Unbraced Lengths](#) topic.

K Factors (Effective Length Factors)

See the [Unbraced Lengths](#) topic.

Sway Flags

See the [Unbraced Lengths](#) topic.

AISC Design Parameters

Cm - Interactive Bending Coefficient

Cm Coefficients are described in Chapter H of the 9th Edition AISC (ASD) code. If these entries are left blank, they will be automatically calculated.

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



Note: The Cm factor is not used for LRFD code checking because the Chapter C requirement that P-Delta effects be considered is met with a direct P-Delta analysis. LRFD code checks will not be performed without P-Delta analysis.

Cb - Bending Coefficients

Cb Factors are described in Chapter F of the AISC code and are used in the calculation of the nominal flexural strength, Mn. If this entry is left blank, it will be calculated automatically for AISC code checks.

The calculation of **Cb** is based on the unbraced length of the compression flange and the moment diagram for the unbraced segment in question. If a specific unbraced length is entered by the user, the program cannot interpret the location of brace points and the Cb value will default to '1.0'. In some cases, it may be better to enter 'segment' as the unbraced length for a physical member. When 'segment' is entered, the Cb value will be calculated individually for each segment of the beam based on that segment's moment diagram.



Note:

- For members designated as "Tapered WF" shapes, Cb is always calculated per the AISC 360 section F1 equation.
- AISC 9th Ed always assumes Cb = 1.0.

Function for Stiffness Reduction

The **Function** entry may be set to either 'Lateral' or 'Gravity' using the drop down list in the spreadsheet. If the Adjust Stiffness option is set to Yes on the Codes tab of the Model Settings Dialog, then all members with a 'Lateral' Function will be considered for the stiffness reduction required per the AISC Direct Analysis Method.

The **Flexural Stiffness Reduction** of the Direct Analysis Method will be applied to all 'Lateral' members. This process involves evaluating certain conditions:

- **Primary Stiffness Adjustment Requirements**

The program first checks whether a member satisfies the three primary stiffness adjustment requirements:

- The member is defined as "Lateral".
- The stiffness adjustment option is enabled for the specific material being used.
- The chosen material code permits stiffness adjustment for the selected material.

- **Stiffness Adjustment Calculation**

Once the above conditions are met, the software proceeds to calculate the stiffness adjustments for the member. This includes reducing the area and moment of inertia based on predefined multiplication factors:

- Steel Material: Area and moment inertia are multiplied by 0.8.

- **Exclusion Criteria**

However, certain exclusion criteria are considered to determine whether to apply the stiffness adjustments or not. The following conditions must be met simultaneously for exclusion:

- The material is Hot Rolled
- The selected steel code is AISC 13th ed.
- The member type is not specified as:
 - Column or v-brace for area reduction.
 - Column or beam for moment of inertia reduction.

If all the exclusion criteria are met, the original area and moment of inertia values will be retained without applying the stiffness adjustment multiplication factors. This ensures that certain types of HR members, when using the AISC code, maintain their original stiffness characteristics.

The program can perform an iterative analysis during the solution depending on the value of τ_b . In this case, the stiffness matrix is recomputed for each iteration until the value of τ_b converges within 1 percent for all 'Lateral' members in compression. In the unlikely event that τ_b is less than zero, the value of τ_b is considered to be 1.e-5. When used in the analysis, the value for τ_b will be listed in the Detail Report for that member.

When the users sets the Adjust Stiffness flag on the [Model Settings](#) to **Yes (Tau =1.0)**, then the program will use a Tau of 1.0 in the stiffness analysis and no iteration of the stiffness matrix is necessary. This option is a good feature for models which take a long time to solve or which have not yet been proportioned to control drift. Stiffness adjustment calculation is still taken into account unless exclusion criteria is met.

The **Axial Stiffness Reduction** of the Direct Analysis Method behaves differently. For the AISC 360-05, 360-10, and 360-16, the reduction will be applied to all 'Lateral' members.

**Note:**

- The stiffness reduction required by the Direct Analysis Method will be ignored if the Adjust Stiffness option is not selected on the [Codes](#) tab of the **Model Settings**, or if the design code chosen does not have an option for stiffness reduction.
- The Direct Analysis Method requires the use of reduced flexural stiffness for all members whose flexural stiffness is considered to contribute to the lateral stability of the structure. This will only apply to members designated as type 'Column' or 'Beam.' If the user assigns type 'None' to the member, the Flexural Stiffness Reduction will not be applied.
- When the Adjust Stiffness flag is set to **Yes (Tau=1.0)**, then the code requires a higher value for the applied [Notional Loads](#).
- When the stiffness adjustment option is enabled from the model settings (either Yes (Tau=1.0) or Yes (Iterative)), the program will include the predefined multiplication factors for area and moment of inertia for lateral members in addition to the specified Tau value.

Channel Conn.

The type of connection for double channels is reported in the Channel Conn. column. This input is only for back to back channels and is used in the calculation of KL/r.

a - Connector Spacing

For double channels, the connector spacing 'a' is used in the calculation of KL/r. This input is only used for back to back channels.

Allowable Stress Increase Factor (ASIF)

AISC 9th Edition

Increasing of allowable stresses may be allowed when forces are transient. You can enter an **ASIF** factor on the **Load Combinations Spreadsheet** to allow the increase for a specific load combination. The ASIF factor is then applied to the allowable stresses in accordance with section A5. The ASIF factor also is applied to the Euler values (F'e).



Note: If the allowable stress increase is being used, the final code check value should still be compared to '1.0'.

All Other US Codes

Setting the **ASIF** factor to a value greater than '1.0' will not cause the capacities to be increased by that factor. However, setting the **ASIF** factor to a value greater than '1.0' is used as a flag to use of the seismic compactness criteria of Table I-8-1 of AISC 341-05 Seismic Provisions of Steel Buildings. Specifically we will use the limiting width-thickness ratios from this table for capacity calculations (for compression flange local buckling for example). In these cases, we use Table I-8-1 of AISC 341-05 as opposed to Table B4.1 of AISC 360-05.

AISC Limitations

AISC 360-16 (15th Ed.)

Wide Flanges:

- Flexural torsional buckling per Chapter E is included when L_{torque} is greater than either L_{by} or L_{bz} .
- Flange local buckling per Chapter F is only included when the web is compact and the flange is either non-compact or slender.

Rectangular Tubes - Lateral torsional buckling is checked per section F7.4, however this section only applies to strong axis bending.

Code Checks - For combined bending and tension (Section H1.2) the code-allowed modification of C_b is not applied. For combined bending and compression, Section H1.3 is not considered, meaning that the program can be over conservative in some situations.

Single Angles - Single angles in compression are not checked for Section E4 because no standard single angle shapes have a slenderness (b/t) > 20 . They are also not checked for Section E5 as there is insufficient information regarding the connections and usage of the member.

WT and LL Shapes - This code does not address the rare case where Lateral Torsional (or Flexural Torsional) Buckling occurs for WT's and double angles bent about their weak axis. Therefore, only yielding is checked for weak axis bending.

Double Channels - Double channel connector spacing requirements are not checked. Additionally, double channel design is only available in AISC 360-16 and AISC 360-10 codes.

AISC 360-05 (13th Ed.) and AISC 360-10 (14th Ed.)

Wide Flanges:

- For AISC 360-10, flexural torsional buckling per Chapter E is included when L_{torque} is greater than either L_{byy} or L_{bzz} .
- For AISC 360-05, flexural torsional buckling per Chapter E is only included when $L_{torque} > L_{byy}$ and $L_{torque} > L_{bzz}$.
- Flange local buckling per Chapter F is only included when the web is compact and the flange is either non-compact or slender.

Code Checks - For combined bending and tension (Section H1.2) the code-allowed modification of C_b is not applied. For combined bending and compression, Section H1.3 is not considered, meaning that the program can be over conservative in some situations.

Single Angles - Single angles in compression are not checked for Section E4 because no standard single angle shapes have a slenderness (b/t) > 20 . They are also not checked for Section E5 as there is insufficient information regarding the connections and usage of the member.

WT and LL Shapes:

- This code does not address the rare case where Lateral Torsional (or Flexural Torsional) Buckling occurs for WT's and double angles bent about their weak axis. Therefore, only yielding is checked for weak axis bending.
- For slender WT sections, the compressive capacity is calculated with Q_a always assumed equal to 1.0.

ASD 9th Edition Limitations

Wide Flange Shapes - Code checks for shapes that qualify as plate girders, as defined by Chapter G, are not performed. Plate girders that can be checked by the provisions of Chapter F will have code checks calculated.

Channels - The AISC 9th Edition (ASD) specification does not specifically address the allowable stress for weak axis bending of channels. Therefore, the program uses the most similar formula for the weak axis bending of wide flanges ($0.75 \cdot F_y$). For a complete and thorough treatment of channel code checks, refer to the LRFD specification.

WT and LL Shapes - ASD allowable bending stresses calculated for WT and LL shapes use Chapter F for cases when the stem is in compression. This is not technically correct, but the ASD code does not provide direction regarding other means of calculating the allowable bending stress in this situation. In the interim, the LRFD code directly addresses this situation, so it is recommended that you use the LRFD code to check WT and LL shapes that have their stems in compression.

Neither the ASD or LRFD codes address the rare case where Lateral Torsional (or Flexural Torsional) Buckling occurs for WT's and double angles bent about their weak axis.

RE Shapes - Rectangular bar members (on-line shapes) are assigned allowable bending stresses for the yielding limit state only. Lateral torsional buckling is not considered because the ASD code doesn't directly address this for rectangular shapes. The strong axis bending stress is assigned as $0.66 \cdot F_y$ and the weak axis bending stress is assigned as $0.75 \cdot F_y$. If you have a case where lateral

torsional buckling may govern, you should use the LRFD code, since the LRFD code does address this limit state.

LRFD 2nd and 3rd Edition Limitations

Wide Flange Shapes - LRFD code checks for shapes that qualify as plate girders as defined by Chapter G are not performed.

Single Angles - Single angles are only checked for Euler buckling. They are not checked for Flexural-Torsional buckling.

WT and LL Shapes - Neither the ASD or LRFD codes address the rare case where Lateral Torsional (or Flexural Torsional) Buckling occurs for WT's and double angles bent about their weak axis.

Tapered Wide Flanges - ASD 9th edition code checks can be performed on tapered members with equal or unequal top/bottom flanges, with the restriction that the compression flange area must be equal to or larger than the tension flange area. LRFD 2nd edition code checks are limited to tapered members with equal area flanges. Code checks are performed using Appendix F, Chapter F, and Chapter D as applicable. Note that the rate of taper is limited by Appendix F, and the program enforces this. The interaction equations in Appendix F are used to compute the final code check value. These equations also include the effects of weak axis bending, if present. Torsional warping effects on Tapered WF members are NOT considered.

Prismatic Wide Flanges with Unequal Flanges - ASD code checks for prismatic WF members with unequal flanges are also limited to shapes that have the compression flange area equal to or larger than the tension flange area. LRFD code checks currently cannot be performed for prismatic WF members with unequal flanges.

Pipes and Bars - For pipes and round bars, the code check is calculated based on an SRSS summation of the y and z-axis stresses calculated for the pipe or bar. This is done because these circular shapes bend in a strictly uniaxial fashion and calculating the code check based on a biaxial procedure (as is done for all the other shapes) is overly conservative.

Single Angles - Code checking (LRFD or ASD) on single angle shapes is based on P/A (axial load/axial strength or axial stress/allowable axial stress) only. This is because the load eccentricity information needed for a meaningful bending calculation is not available. Only Euler buckling is considered for single angles, flexural-torsional buckling is NOT considered. Single angles will have the following message displayed on the Code Check Spreadsheet to remind the user of the axial only code check: "Code check based on z-z Axial ONLY"

Please see [Single Angle Stresses](#) for more information on the calculation of single angle stresses.

AISC 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 can click the cell that contains the message and look to the status bar to view the full message. The following are the messages that may be listed:

"AISC Code Check Not Calculated"

This message is displayed when the member is not defined with a database shape, a steel code is not specified on the **Model Settings** or no units were specified. For LRFD this message is displayed if the steel yield stress is less than 10ksi.

"Web is slender per Table B5.1, handle as a plate girder"

The ratio h/t_w exceeds the limiting criteria listed in Table B5.1. This means Chapter G (plate girders) governs.

"Compressive stress f_a 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 $> 6 * \text{width}$ (Sec F3-1) where $\text{width} = b_f - 3 * t_w$, 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 $13,000/F_y$ (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 $13000/F_y$ specified in Section B5-b for ASD, Section B5-3b for LRFD.

" $KL/r > 200$ for compression member"

Section B7 recommends that KL/r for compression members not exceed 200. For the ASD 9th edition code, a procedure is presented to handle when KL/r exceeds 200. Thus, for ASD 9th edition, $KL/r > 200$ is permitted. For all other AISC codes no guidance is provided as to what to do if $KL/r > 200$, so exceeding this limit is not permitted. You can override this in the **Application Settings** which can be accessed by clicking on the **File** menu.

"Taper Flange area is not constant per App. A-F7-1 (b)"

The limitations of Appendix F for the design of web tapered members include the restriction that the flange area shall be constant along the length of the member. This member's flange area changes along its length. See Appendix Section F7.1 (b).

"Taper rate exceeds gamma limit per App. A-F7-1 (c)"

The limitations of Appendix F for the design of web tapered members include a limit on how steep the rate of taper can be along the member length. This member's taper rate exceeds the limit given by equation A-F7-1. See Appendix Section F7.1 (c).

"Flanges not equal, currently don't do LRFD App. F1 calcs"

The requirements for Wide Flange members with unequal flanges in the LRFD, Appendix F1, are not addressed.

"Taper Comp. flange < Tension flange, per App A-F7-1 (b)"

The limitations of Appendix F for the design of web-tapered members include the restriction that the flange areas of the top and bottom flange must be equal. The compression flange may be larger than the tension flange. However equation A-F7-4 is unconservative for cases where the compression flange is smaller than the tension flange. See Appendix Sections F7.1 (b) and F7.4.

Canadian 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.

w1 - Interactive Bending Coefficient

w1 is the coefficient to determine equivalent uniform bending effect in beam-column, as described in Section 13.8.5 of CSA S16-09. If left blank, will calculate it. The w1 value is dependent on the member's end moments, which may change from one Load Combination to the next. Therefore it is a good idea to leave this entry blank and let calculate it.

w2 - Bending Coefficient

w2 is the coefficient to account for increased moment resistance of a laterally unsupported doubly symmetric beam segment when subject to a moment gradient, as described in Section 13.6a of CSA S16-09. If left blank, will calculate it. The w2 value is dependent on the moment in the member, which

may change from one Load Combination to the next. Therefore it is a good idea to leave this entry blank and let calculate it.

Canadian Limitations

It is assumed the transverse load on the member is occurring through the member's shear center. This means secondary torsional moments that may occur if the load is not applied through the shear center are not considered.

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.

S16-14, S16-09

Tapered Wide Flanges - No code checking is done for "Tapered WF" members which have different "Start" and "End" Depths, or which have top and bottom flanges of different widths. In other words, the "Tapered WF" may only be used for prismatic, doubly symmetric sections.

Single Angles - Single angles in compression are not checked for Clause 13.3.3.2 or 13.3.3.3 because there is insufficient information regarding the connections and usage of the member. They are not checked for Clause 13.3.2 (Flexural-Torsional Buckling) either. Instead they are checked for Euler Buckling about their geometric or principal axes per Clause 13.3.1. The slenderness classification for single angles is based on the longer leg.

S16-09 recommends using a "rational analysis" to account for lateral-torsional buckling and shear checks on single angles, so RISA uses the AISC 360-10 (14th Edition) provisions for these checks.

Class 4 Members - Members with *both* Class 4 (slender) webs *and* Class 4 (slender) flanges are not designed. Per S16-09, Clause 13.5.c.i these members must be designed per S136-07. Where only the web *or* the flange is Class 4 (slender) the capacity is determined per the provisions of S16-09, Clause 13.5c.

The code is not clear on whether this applies to WT, Double Angle, and Single Angle members, since they do not have a "web". Therefore, for these member types, the program calculates an effective (reduced) yield stress for each leg/flange/stem and uses the smallest value for the entire section.

Pipes and Round HSS - The code does not address how to determine the shear capacity of Class 3 or Class 4 (Noncompact, Slender) pipes. Therefore no design is done for those members. Class 1 and 2 member capacities are determined per the provisions of S16-09.

Shear Capacity - The code does not address how to determine the shear capacity of WTs, Double Angles, or Single Angles where shear buckling is a consideration (S16-09 Clause 13.4.3). Therefore no design is done for members where d/w or b_{ei}/t exceeds $1014/(F_y)^{1/2}$.

WTs - The lateral-torsional buckling moment capacity calculation (Clause 13.6e.i) in the code does not address how to calculate L_u when the stem of the WT is in compression. Therefore L_u is always taken as zero, which is conservative. The code also does not address how to calculate β_x when the

stem is in compression. Therefore, I_{yc} is taken as zero, which results in a negative value of β_x , which is conservative.

Double Angles - The code does not address how to calculate lateral-torsional buckling moment capacity. RISA therefore uses the AISC 360-10 (14th Edition) provisions for these checks, as they are a widely accepted rational method. When this method is used, it is assumed that $C_b = \omega_2$. The slenderness classification for double angles is based on the longer leg.

Welded Reduced Flange (WRF) Members- Since these members are classified within RISA as "Tapered", there is currently no design done for them. This limitation will be removed in a future release.

Torsional Buckling and Flexural Torsional Buckling (S16-09 and earlier) - The limit states of torsional buckling and flexural torsional buckling are not considered for wide flange and channel members. This means that the value F_{ez} is not calculated for these members per Clause 13.3.2. The program takes the lesser of F_{ex} and F_{exy} for axial buckling capacity for the S16-09 and earlier codes. These limit states are fully considered for S16-14 because an L_{torque} input was added for S16-14 that was not present in earlier code implementations.

Compressive Strength (S16-09 and earlier) - For the equations in section 13.3.1, the parameter "n" is assigned a value of 1.34 for all shapes for S16-09 and earlier codes. This is conservative for WWF shapes and HSS shapes that are stress-relieved. For S16-14, members designated as type "Tapered WF" use $n = 2.24$ while members designated as type "WF" use $n = 1.34$.

Double Channels - Double channel design for Canadian code is only available in S16-14. Connector spacing requirements are not checked for double channels.

S16.1-94, S16-01, S16-05

WT and LL Shapes - The criteria in the AISC LRFD 2nd Edition code is used to perform code checks on WT and LL shapes since the Canadian code does not explicitly specify how to calculate the flexural strength of WT and LL shapes.

The Canadian code does not address the rare case where Lateral Torsional (or Flexural Torsional) Buckling occurs for WT's and double angles bent about their weak axis.

Tapered Wide Flanges - The AISC LRFD 2nd code is used to perform code checks on Tapered WF shapes when the Canadian code is specified. The Canadian code CAN/CSA S16.1-94 does not address web-tapered members.

Single Angles - Code checking on single angle shapes is performed for tension only. Single angles will have the following message displayed on the Steel Code Check Spreadsheet to remind the user of the tension only code check: "Single Angle code check based on Axial Tension ONLY"

Please see [Single Angle Stresses](#) for more information on the calculation of single angle stresses.

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.

Canadian 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 can click the cell that contains the message and look to the status bar to view the full message. Following are the messages that may be listed specifically for the Canadian Code:

"Tension member with L/r ratio > 300 (CAN Sect. 10)"

The maximum L/r ratio for this tension member exceeds the limit shown in Section 10.2.2 of the CAN/CSA S16.1-94 code.

"KL/r > 200 for compression member (CAN Sect. 10)"

The maximum L/r ratio for this compression member exceeds the limit shown in Section 10.2.1 of the CAN/CSA S16.1-94 code. You can override this check in the **Application Settings** which can be accessed by clicking on the **File** button.

"Can't do unity check for slender compression member"

Compression strengths are not calculated for shapes that contain elements where the width to thickness ratios are classified as "slender".

"Can't do unity check for flexural member with slender web"

Flexural strengths are not calculated for shapes which have a web depth to thickness ratio classified as "slender".

"Currently can't do unity check for member with slender flanges"

Flexural strengths are not calculated for shapes which have a flange width to thickness ratio classified as "slender".

"Can't do unity check for single angles in compression"

Compression strengths are not calculated for single angle members.

British 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.

m- Interactive Bending Coefficients

Section 4.8.3.3.4 of the British code describes the m coefficient. If these entries are left blank, RISA will calculate them. The m value is influenced by the sway condition of the member and is dependent on the member's end moments, which will change from one Load Comb to the next. It's a good idea to leave these entries blank and let RISA calculate them.

m-LT - Bending Coefficients

This coefficient is discussed in Section 4.3.6.6 of the BS5950-1:2000 code and is used in the calculation of the flexural strength. If this entry is left blank it will be calculated automatically. This value also is impacted by the member's sway condition and is dependent on the member's end moments so it may be a good idea to let it be calculated internally. An exception to this would be for cantilever members in sway frames, this value should be 1.75, and it will be automatically calculated as 1.0. This will be addressed in a future program version.

British Limitations

It is assumed the transverse load on the member is occurring through the member's shear center. This means secondary torsional moments that may occur if the load is not applied through the shear center are not considered.

Tapered Members - Tapered WF shapes are done per the AISC LRFD 2nd specification at this time. The appropriate sections of the BS5950-1 specification will be used in a later release.

Torsional Warping Effects - Combined bending and warping torsional stresses in WF and Channel shapes are handled per the AISC publication "Torsional Analysis of Steel Members". A later release will use the SCI publication "Design of Members Subject to Combined Bending and Torsion" for combined bending and warping stresses when the British Hot Rolled Steel code is selected.

Single Angles - Single angles are checked for axial and shear forces only. No bending is considered at this time. A later release will consider the requirements in Annex I4 for single angles.

Secondary Moments per Annex I - The program does not yet consider the internal secondary moments described in Annex I.

EuroCode 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.

C_m Factors

(Equivalent Uniform Moment Factors per the 2005/14 EuroCode)

If these entries are left blank, RISA will calculate them per Annex B of the 2005/ 2014 EuroCode otherwise the user can choose to override the calculation by manually entering a value. The C_m

value is influenced by the sway condition of the member and is dependent on the member's end moments, which will change from one Load Combination to the next. It's a good idea to leave these entries blank and let RISA calculate them.

For HSS Tubes, the program always uses Table B.1. For other members not subject to torsion deformations (wide flange members with an unbraced length equal to zero) the program will use Table B.1. for the 2014 code. All other members, the program will use table B.2.

C_m -LT Factor

(LTB Equivalent Uniform Moment Factors per the 2005 EuroCode)

If this entry is left blank, RISA will calculate it per Table B.3 in Annex B of the 2005 EuroCode, otherwise the user can choose to override the calculation by manually entering a value. The C_m LT value is influenced by the sway condition of the member and is dependent on the member's end moments, which will change from one Load Combination to the next. It's a good idea to leave these entries blank and let RISA calculate them.

B_m - Interactive Bending Coefficients

(per the 1992 EuroCode)

Section 5.5.4 (7) of the EuroCode describes the B_m coefficients. If these entries are left blank, RISA will calculate them. The B_m value is influenced by the sway condition of the member and is dependent on the member's end moments, which will change from one Load Comb to the next. It's a good idea to leave these entries blank and let RISA calculate them.

B_m -LT Bending Coefficient

(LTB Bending Coefficient per the 1992 EuroCode)

This coefficient is discussed in Section 5.5.4 (7) of the 1992 EuroCode code and is used in the calculation of the flexural strength. If this entry is left blank it will be calculated automatically. This value also is impacted by the member's sway condition and is dependent on the member's end moments so it may be a good idea to let it be calculated internally. An exception to this would be for cantilever members in sway frames, this value should be 1.75, and it will be automatically calculated as 1.0. This will be addressed in a future program version.

C1 Factor

(Moment Distribution Modification Factor)

If this entry is left blank, RISA will calculate it per the explicit equation presented in the widely accepted article ["Lateral-Torsional Buckling of Steel Beams: A General Expression for the Moment Gradient Factor" by López, Yong, and Serna \(2006\)](#), as there is no suitable generic formula presented in the EuroCode.

$$C_1 := \frac{\sqrt{\sqrt{k_1} \cdot A_1 + \left(\frac{(1-\sqrt{k})}{2} \cdot A_2\right)^2} + \frac{(1-\sqrt{k})}{2} \cdot A_2}{A_1} \quad \text{López, Yong, and Serna Eqn. (8)}$$

$$C_2 := 0$$

$$C_3 := 0$$

Where:

$$A_1 := \frac{M_{max}^2 + \alpha_1 \cdot M_1^2 + \alpha_2 \cdot M_2^2 + \alpha_3 \cdot M_3^2 + \alpha_4 \cdot M_4^2 + \alpha_5 \cdot M_5^2}{(1 + \alpha_1 + \alpha_2 + \alpha_3 + \alpha_4 + \alpha_5) \cdot M_{max}^2} \quad \text{López, Yong, and Serna Eqn. (10)}$$

$$A_2 := \left| \frac{M_1 + 2 \cdot M_2 + 3 \cdot M_3 + 2 \cdot M_4 + M_5}{9 \cdot M_{max}} \right| \quad \text{López, Yong, and Serna Eqn. (11)}$$

Otherwise the user can override this calculation by manually entering a value.



Note: C2 and C3 are taken as zero because RISA relies on the "general case" lateral torsional buckling equations (see section 6.3.2.2) rather than the more specialized section for rolled and welded I shaped sections (6.3.2.3).

EuroCode Buckling Curve Factors

This factor is discussed in Section 6.3.1.2 of the 2014 EuroCode and is used in the calculation of the axial compression. Table 6.1 is used except for when Lateral Torsional Buckling is considered then Table 6.4 is used instead. The following illustrates what RISA uses for the buckling curve from Table 6.2:

Table 6.2 (as implemented by RISA)

Cross Section	Limits	Material Type	Imperfection Factor (Weak Axis)	Imperfection Factor (Strong Axis)
Wide Flange: $h/b > 1.2$	$t_f \leq 40$ mm	$f_y = 460$ MPa Other	0.13 0.34	0.13 0.21
	$40 \text{ mm} < t_f \leq 100$ mm	$f_y = 460$ MPa Other	0.21 0.49	0.21 0.34
	$t_f > 100$ mm	$f_y = 460$ MPa Other	0.49* 0.76*	0.49* 0.76*
Wide Flange: $h/b \leq 1.2$	$t_f \leq 100$ mm	$f_y = 460$ MPa Other	0.21 0.49	0.21 0.34
	$t_f > 100$ mm	$f_y = 460$ MPa Other	0.49 0.76	0.49 0.76
Tubes & Pipes	2014 code (Cold Formed)	All	0.34	0.34
	prior codes (Hot Finished)	$f_y = 460$ MPa Other	0.13 0.21	0.13 0.21
Channels			0.49	0.49
WT and LL			0.49	0.49
Single Angles	2014 code		0.34	0.34
	prior		0.49	0.49
Solid Bar or Rect			0.49	0.49
Tapered WF	$t_f \leq 40$ mm (Hot Finished)		0.49	0.34
	$t_f > 40$ mm (Hot Finished)		0.76	0.49

* The EuroCode does not give guidance on which curve to use in these situations.

Buckling Curve	a ₀	a	b	c	d
Imperfection Factor	0.13	0.21	0.34	0.49	0.76

EuroCode Limitations

Tapered Members - Tapered member design per the Eurocode is not supported at this time.

Torsional Warping Effects - Combined bending and warping torsional stresses in WF and Channel shapes are handled per the AISC publication "*Design Guide #9- Torsional Analysis of Steel Members*".

WT and Double Angle Limitations - The EuroCode does not address the rare case where Lateral Torsional (or Flexural Torsional) Buckling occurs for WT's and double angles bent about their weak axis. Therefore, the calculation of M_{cr} is based on AISC LRFD equation and used in the code checks for Lateral Torsional (or Flexural Torsional) Buckling.

Lateral Torsional Buckling - The value M_{cr} used in the lateral-torsional buckling capacity of beams relies on a factor C1. When the C1 field is left blank it is automatically calculated per the explicit equation presented in the widely accepted article "[Lateral-Torsional Buckling of Steel Beams: A General Expression for the Moment Gradient Factor](#)" by Lopéz, Yong, and Serna (2006), as there is no suitable generic formula presented in the EuroCode. C2 and C3 are taken as zero because RISA relies on the "general case" lateral torsional buckling equations (see section 6.3.2.2) rather than the more specialized section for rolled and welded I shaped sections (6.3.2.3).

Single Angles - Single angles are checked for axial and shear forces only for EuroCodes prior to 2014 Edition. No bending is considered for Eurocodes prior to 2014 Edition.

EuroCode Special Messages

"Rho >= 1.0, No code check calculated"

In beams with high shear the code implements a moment strength reduction factor (ρ), but does not place an upper limit on it. If this value exceeds 1.0 the beam has a negative moment capacity, which is irrational. The program therefore prevents a code check for these circumstances.

Indian 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.

Cm Interactive Bending Coefficients

If these entries are left blank, RISA will calculate them. The m value is influenced by the sway condition of the member and is dependent on the member's end moments, which will change from one Load Comb to the next. It's a good idea to leave these entries blank and let RISA calculate them.

Indian Buckling Curve Factors

This factor is discussed in Section 7.1.2.2 of the 2007 Indian Design code and is used in the calculation of the axial compression. The following illustrates what RISA uses for the buckling curve from Table 7:

Table 10 Buckling Class of Cross-Sections (in RISA)

Cross Section	Limits	Imperfection Factor (Weak Axis)	Imperfection Factor (Strong Axis)
Wide Flange $h/b > 1.2$	$tf \leq 40\text{mm}$	0.34	0.21
	$40\text{mm} < tf \leq 100\text{mm}$	0.49	0.34
	$tf > 100\text{mm}$	0.76*	0.76*
Wide Flange $h/b \leq 1.2$	$tf \leq 100\text{mm}$	0.49	0.34
	$100\text{mm} < tf$	0.76	0.76
Tubes & Pipes		0.34	0.34
Channels		0.49	0.49
WT and LL		0.49	0.49
Single Angles	2014 code	0.49	0.49
Solid Bar or Rect		0.49	0.49
Tapered WF	$tf \leq 40\text{mm}$ (Hot Finished)	0.49	0.34
	$40\text{mm} < tf$ (Hot Finished)	0.76	0.49

* The Indian Code does not give guidance on which curve to use in these situations.

Buckling Curve	a	b	c	d
Imperfection Factor	0.21	0.34	0.49	0.76

Indian Limitations

Tapered Members - Tapered WF shapes are done per the AISC ASD 9th (for the 1998 code) and LRFD 2nd (for the 2007 code) specifications at this time. The appropriate sections of the IS:800 specification will be used in a later release.

Torsional Warping Effects- Combined bending and warping torsional stresses in WF and Channel shapes are handled per the AISC publication "Torsional Analysis of Steel Members".

WT and Double Angle Limitations - The Indian code does not address the case where Lateral Torsional (or Flexural Torsional) Buckling occurs for WT's and double angles bent about their weak axis. For the 2007 code, RISA will use the AISC 360-05 (aka AISC 13th edition) formulas for calculating member capacity for these failure states.

Shear Stress - For all shapes, the average shear stress is not checked per section 6.4.2 at this time.

Class 4 (Slender) Sections - No code checking is performed for class 4 sections for the IS 800: 2007 code.

Single Angles - Single angles are checked for axial and shear forces only. No bending is considered at this time.

Australian & New Zealand 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.

Alpha_m Factor

(Moment Modification Factor)

- If this entry is left blank, RISA calculates it based on the full length of the member.
- When unbraced lengths (Lb_comp_top and Lb_comp_bot) are explicitly defined (enter a number), the program does not have the segmental information of the member for alpha_m calculation.

It assumes the alpha_m used for this particular member to be 1.0.

- When unbraced lengths (Lb_comp_top and Lb_comp_bot) are defined by physical nodes along the member with the entry 'Segment' typed in, alpha_m is calculated automatically using equation 5.6.1.1(2) based on segments defined by nodes along the member.

The alpha_m value is influenced by the sway condition of the member and is dependent on the member's end moments, which changes from one Load Combination to the next.

Australian & New Zealand Limitations

Tapered Members - Tapered WF shapes are done per the AISC LRFD 3rd specification at this time. The appropriate sections of the ENV 1993-1-1 specification are used in a later release.

Torsional Warping Effects- Combined bending and warping torsional stresses in WF and Channel shapes are handled per the AISC publication "Torsional Analysis of Steel Members".

WT and Double Angle Limitations - The NZ / AS codes do not address the rare case where Lateral Torsional (or Flexural Torsional) Buckling occurs for WT's and double angles bent about their weak axis.

Torsional Unbraced Length - At present the program assumes 'Lcomp' as 'Le' in case of lateral torsional buckling. Section 5.6.3 provides a procedure for calculating 'Le' which is not being addressed at this time.

Special Provisions for Cantilevers - At present the program does not address section 5.6.2 for cantilever elements.

General Member Design Limitations

Composite Steel Design - This is not done in RISA-3D. RISAFloor has this capability.

Welded Sections - There is a basic assumption in the program that the steel sections are hot rolled not welded. Therefore, any code provisions that assume additional stresses due to welding a built-up cross section are not specifically accounted for. For the AISC-LRFD codes this generally means that the flange residual stress (F_r) is always taken as 10ksi, as for a rolled shape. The only exception is for Tapered WF shapes where it is always taken as 16.5 ksi, as for a welded shape.

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

Single Angles - In all codes other than AISC 360-05 and 360-10, single angles are only checked for axial loading. Flexural effects are not considered in the code check calculation. Under AISC 360-05 and 360-10 single angles are checked for combined bending and axial about either their principal or geometric axes depending on how their [unbraced length](#) has been defined.

- Lateral-Torsional buckling does not apply to Geometric bending angles, and is therefore not checked for them.
- Lateral-Torsional buckling does not apply to minor-axis bending on Principal bending angles, and is therefore not checked for them.
- The provisions of Section E5 of the specification are not considered. The angle is considered to be loaded concentrically, and on both legs.
- Interaction equation H2-1 is used for all the code check on all angles (equal and unequal leg).

Double Angles - For y axis buckling (where stitch connections would be experiencing shear), the program only considers KL/r of the overall built up shape and does NOT attempt to reduce the KL/r value based on the spacing between connectors. Therefore the program assumes that there are pre-tensioned bolts or welds spaced closely enough to allow the double-angles to act as one unit per AISC 360-10 Eqn E6-2a.

Net Section for Tension Capacity - The tension capacity is calculated using the Gross area. A later release will include a "net area factor" that the user can enter to indicate what the effective area should be for the tension capacity calculation.

Solid Rectangles - Solid rectangular members will not be checked for lateral torsional buckling. This limitation is only important for tall, slender shapes as LTB should not be a realistic limit state for short and thick or square shapes.

Torsional and Flexural Torsional Buckling of Doubly Symmetric Shapes - Flexural Torsional Buckling and Torsional Buckling are checked for the following shape types for AISC 360-10 (14th Edition) and AISC 360-16 (15th Edition) steel codes:

- Non Slender WF
- Non Slender Channels

- Non Slender WTs
- Non Slender Double Angles
- Non Slender Single Angles

Flexural Torsional Buckling and Torsional Buckling are also checked for Non Slender WTs and Double Angles for the AISC 360-05 (13th Edition Steel Code).

For WTs $(KL/r)_m$ per section E4 (a) is calculated per equation E6-1 with 'a' always assumed to be 0. This results in $(KL/r)_m = (KL/r)_o$.

P-Little Delta Analysis - An incremental P-Delta re-iterative approach is used in RISA 3D. This method does NOT automatically account for P-Little Delta effects. If these effects are expected to be significant please refer to the [P-Little Delta](#) section.

Notional Loads - Currently Notional Loads are not automatically included in the analysis and it is expected that the user will create their own set of Notional Loads when required. Please see the [Load Generation - Notional Loads](#) topic for more information on using the program to generate and apply notional loads.

Tapered Members

The analysis for tapered wide flange members is handled internally by breaking the member into a series of 14 piecewise prismatic members. Then the program will condense out the extra degrees of freedom when assembling the stiffness matrix.

This does a very good job for member stiffness and basic analysis. However, because the extra degrees of freedom are eliminated before the analysis, it does not currently account for the P-Little Delta effect. If this effect is desired, please refer to the [P-Little Delta](#) topic for techniques to add this effect into your frame analysis.

AISC 13th, 14th, & 15th Editions

(Design Guide 25)

These design specifications do not actually include any design provisions for tapered members. However, the AISC and MBMA jointly released *Design Guide 25 - Frame Design Using Web-Tapered Members*, which provides a design procedure consistent with the AISC design specifications.

RISA follows the recommendations of the *Design Guide* for tapered member codes checks whenever the AISC 13th, 14th, or 15th edition steel codes are selected.

Design Checks

The member detail reports will display not only the governing capacity values, but also the governing equations/limit state that provided the reported capacity. These limit states (and any design assumptions) are listed below:

In-Plane Flexural Buckling (z-z): The RISA implementation uses the Equivalent Moment of Inertia method described in Appendix A of Design Guide 25.

Out-of-Plane Flexural Buckling (y-y): This calculation uses the weak axis properties from the mid-section of the unbraced length.

TB (Torsional Buckling): Columns with equal flanges will only be checked for Torsional Buckling when the torsional unbraced length ($K*L_{torque}$) is greater than the weak axis buckling unbraced length ($K*L_{yy}$).

FTB (Torsional Buckling): RISA currently only checks flexural torsional buckling when the tapered member has unequal flange widths and (for cases when flange widths are equal) when the ratio of the flange thickness is greater than 1.5. This calculation uses the weak axis properties from the mid-section of the unbraced length.

LTB (Lateral Torsional Buckling): The RISA implementation uses the procedure for Single Linear Tapered Members for most members. In cases where both the thickness and depth of the way change, however, the program will use the more generalized procedure.

CAT (Constrained Axis Torsional Buckling): This is only checked when $L_{comp-bottom}$ is greater than $L_{comp-top}$. The assumption is that the top flange is the outside flange and may be more restrained due to the presence of purlins or girts. The calculation of a_s and a_c assume a girt or purlin depth of zero.

Combined Stress Equations: The RISA implementation uses the force based combined strength equations from chapter H1 of the AISC specification. The program does not consider the alternate stress based combined stress equations based on chapter H2 of the AISC specification.

Shear Strength: The shear check equations are based on section G2 of the AISC specification and do not consider the effects of stiffeners or tension field action.

Please see the Tapered Member [results](#) topic for more information about the results reporting.

Unbraced Length and Section Properties: When the user manually enters an unbraced length the program doesn't explicitly know where the brace points are and must make an assumption. This is done in order to determine the section properties at the middle of the unbraced length which are used in the calculation of member capacities. RISA assumes the brace points are equally spaced along the length of the member at the distance given by the user.

For example if the user entered in an unbraced length of 4 ft for a tapered member that is 10 ft long, the program will assume that there is bracing at 0ft, 4ft, 8ft and 10ft.

Limitations

In general, this *Design Guide* allows for design of a wider range of members than permitted over the older AISC Appendix F provisions. However, there are still a number of limitations that will prevent RISA from reporting a code check. These are listed below:

Cb Calculation: Currently, RISA does not calculate C_b per Design Guide 25. Instead it is always calculated per the AISC 360 section F1 equation.

Tension Rupture: The tension rupture limit state is not considered.

Flange Q_s Calculations: RISA will calculate a Q_s value for both flanges and conservatively use the lower value in design calculations.

Direct Analysis Method: Currently, the Tau_b stiffness adjustments for the Direct Analysis Method is based on the worst case section in the tapered member. This worst case stiffness adjustment is applied to the whole member.

55 ksi Limit: The *Design Guide* includes a 55 ksi limit on maximum yield strength. If this is violated a code check will still be calculated, but a warning log message will be given.

ASD 9th, LRFD 2nd, LRFD 3rd

For these design specifications, AISC included an Appendix F covering the design of tapered wide flange members. When calculating code checks per these provisions, the design parameters used in RISA are subject to the following additions / restrictions:

AutoCalc of K values: The **Kyy** factor for Tapered WF shapes can be approximated by RISA-3D, and is the same as for a regular prismatic member. The **Kzz** value cannot be approximated by 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.

Cb value: For Tapered WF members, the Cb field is actually used to enter the "B" value. B is described in Appendix F7.4 of the ASD (9th Edition) and LRFD (2nd & 3rd Edition) codes. This value is not calculated automatically and if it is left blank, a value of '1.0' will be used per the commentary for Appendix F7. The Cb term used in the Chapter F equations is calculated internally for Tapered WF members and will be shown on the Member Detail report and in the Code Check Spreadsheet.

Cm values: 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.

Foreign Codes

Canadian - w1: 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 will calculate them.

Canadian - w2: 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.)

British - m: For Tapered WF members, the LRFD code will be used and the Cmyy(m_yy) value will be used for C'myy, and the Cmzz(m_zz) value will be used for C'mzz. The C'm values are described in Appendix F7.6 of the LRFD codes. These terms are used in the interaction equations in Appendix F for the LRFD code. If these entries are left blank RISA will calculate them.

British - m_LT - For Tapered WF members the LRFD code will be used and the m-LT field is used for the “B” value. B is described in Appendix F7.4 of the LRFD codes. The C_b term used in the Chapter F equations in the LRFD code is always calculated internally for Tapered WF members and will be shown on the Member Detail report and in the Unity check spreadsheet. If the m-LT entry is left blank a value of 1.0 will be used for B, per the commentary for Appendix F7. (The value of “B” is not calculated at this time.)

Euro Code - Cm/Bm: For Tapered WF members, the AISC LRFD 2nd Edition code is used and the C_{myy}(m_yy) value will be used for C'myy, and the C_{mzz}(m_zz) value will be used for C'mzz. The C'm values are described in Appendix F7.6 of the LRFD codes. These terms are used in the interaction equations in Appendix F for the LRFD code. If these entries are left blank RISA will calculate them.

Euro Codes - Cm_LT / Bm_LT: For Tapered WF members the AISC LRFD 2nd Edition code is used and the m-LT field is used for the “B” value. B is described in Appendix F7.4 of the LRFD codes. The C_b term used in the Chapter F equations in the LRFD code is always calculated internally for Tapered WF members and will be shown on the Member Detail report and in the Unity check spreadsheet. If the m-LT entry is left blank a value of 1.0 will be used for B, per the commentary for Appendix F7. (The value of “B” is not calculated at this time.)

Indian Codes - Cm: For Tapered WF members, the LRFD code will be used and the C_{myy}(m_yy) value will be used for C'myy, and the C_{mzz}(m_zz) value will be used for C'mzz. The C'm values are described in Appendix F7.6 of the LRFD codes. These terms are used in the interaction equations in Appendix F for the LRFD code. If these entries are left blank RISA will calculate them.

Buckling-Restrained Brace (BRB) Members

RISA follows Section F4 of the AISC 341-16 code for the design of BRB braces when AISC 15th edition steel code is selected.

BRB Design Parameters

The BRB Design Parameters Spreadsheet records the design parameters for the BRB code checks and may be accessed by selecting Members on the Spreadsheets menu and then selecting the BRB tab. These parameters are defined for each individual member and may also be assigned graphically through the Properties panel.

BRB Design Parameters										
Primary	Advanced	Hot Rolled	Cold Formed	Wood	Concrete Beam	Concrete Column	Aluminum	Stainless	RISAConnection	BRB
	Label	Shape	Length [ft]	Fy Max [ksi]	Fy Min [ksi]	Density [k/ft]	Beta	Omega	Kf	Connection
1	M41	COREBRB_2.75	18.028	46	38	Auto-Calc	Auto-Calc	Auto-Calc	Auto-Calc	Bolted
2	M42	COREBRB_2.75	18.028	46	38	Auto-Calc	Auto-Calc	Auto-Calc	Auto-Calc	Bolted
3	M43	COREBRB_2.75	18.028	46	38	Auto-Calc	Auto-Calc	Auto-Calc	Auto-Calc	Bolted
4	M44	COREBRB_2.75	18.028	46	38	Auto-Calc	Auto-Calc	Auto-Calc	Auto-Calc	Bolted
5	M45	COREBRB_3.00	16.008	46	38	Auto-Calc	Auto-Calc	Auto-Calc	Auto-Calc	Bolted
6	M46	COREBRB_3.00	16.008	46	38	Auto-Calc	Auto-Calc	Auto-Calc	Auto-Calc	Bolted
7	M47	COREBRB_3.00	16.008	46	38	Auto-Calc	Auto-Calc	Auto-Calc	Auto-Calc	Bolted
8	M48	COREBRB_3.00	16.008	46	38	Auto-Calc	Auto-Calc	Auto-Calc	Auto-Calc	Bolted

FyMax, FyMin

These are the upper and lower bounds of the yield strength of the BRB steel core. These values are based on BRB manufacturer's data on the products. FyMax and FyMin are used in different design calculations, see Design Checks and BRB parameter Calculations below for more information.

Density

The density of BRB is reported in this column. If you choose the default "Auto-Calc", the weight density (weight per linear length) is calculated automatically based on BRB manufacturer's data. Note that this weight density is an approximate value and shall be carefully reviewed by BRB manufacturer to ensure its accuracy. User can also input this value manually.

Beta, Omega

The compression strength adjustment factor (β) and strain hardening adjustment factor (ω) can be defined. These parameters are calculated automatically based on BRB manufacturer's data if Auto-Calc is selected. User can also input this value manually.

Kf

The stiffness modification factor Kf can be defined in this column. The BRB member is more rigid at the end zone (connection to beams/columns) comparing to the steel core at the middle portion. Initially, the program calculates the BRB axial stiffness based on steel core only, therefore a stiffness modification factor is needed to capture the actual stiffness of whole member. This parameter (Kf ≥ 1.0) is calculated automatically based on BRB manufacturer's data if Auto-Calc is selected. User can also input this value manually.

Connection

The connection type is defined in this column. Different connection types yield different Kf values due to the connection geometry. Currently Bolted, Welded, and Pinned connection types are supported.

BRB Analysis Consideration

RISA-3D calculates the BRB force demand neglecting the effect of gravity loads per AISC 341-16 Section F4.3. It only considers lateral load categories such as ELs, WLs, NLs, and RSA forces (e.g. SX, SZ) in BRB design. For instance, DL will be neglected in a load combination $0.9DL+1.0EL$ to obtain the BRB force demand. It is assumed that all gravity members have been designed in RISAFloor for gravity loads when there is no braces yet.

Since the brace end zone (brace connections to beam/column) has a higher stiffness than the steel core, a stiffness modification factor Kf is needed to accurately capture the BRB stiffness. The Kf can be input manually by users or automatically calculated by the program. See BRB Parameter Calculation section below for additional information for this factor. The BRB's stiffness used in analysis will be obtained by multiplying the steel core stiffness by Kf factor.

BRBs shall be used to resist axial load only, so only axial load code check is performed for BRBs. Pin-Pin end releases are set as default for BRB shapes. Currently the program does not support fixed end release for BRBs.

Design Codes (AISC 15th) and Required Design Checks

RISA follows Equation (F4-1) in Section F4.5.2 of AISC 341-16 code to calculate the BRB design axial strength in tension and compression. Specified minimum yield stress of the steel core (F_{ysc}) is taken as the F_{yMin} in the BRB properties.

$$P_{ysc} = F_{ysc} A_{sc} \quad (F4-1)$$

$$\phi = 0.90 \text{ (LRFD)} \quad \Omega = 1.67 \text{ (ASD)}$$

BRB Parameter Calculations

The compression strength adjustment factor (β) and strain hardening adjustment factor (ω) can be user-input or automatically calculated by the program. If Auto-Calc was used, RISA follows the procedure in Section F4.2b to calculate these factors. The stress-strain curves for the steel core material are provided by BRB manufacturers. The larger value of the expected deformation corresponding to a) 2% of the story height, or b) two times the design story drift was used in the calculation of these factors. The BRB strain at two times design story drift was estimated using $2 \cdot \delta_{BRB} \cdot C_d \cdot (F_{yMin}/E) / (\rho \cdot l_e)$, this equation conservatively assumes the yield strength of steel core is fully utilized and shall yield an upper bound of beta, omega factors. K_f was calculated based on the geometries of BRB, connection, and beam/column etc.

β , ω will be used to determine the adjusted brace strength in compression (P_{cmax}) and in tension (P_{tmax}) per Section F4.2:

Ry=1.0 for CoreBrace BRB		
Tension	$P_{tmax} := \omega \cdot R_y \cdot F_{ymax} \cdot A_{sc} = 57.801 \text{ kips}$	341-16 F4.2a
Compression	$P_{cmax} := \beta \cdot \omega \cdot R_y \cdot F_{ymax} \cdot A_{sc} = 66.828 \text{ kips}$	341-16 F4.2a



Note: The calculation of the factors β , ω , and K_f follows the recommendations and database provided by manufacturers, these factors shall be carefully reviewed by users as well as manufacturers when finalizing the BRB design.

BRB Limitations

BRB Shapes: Currently, only CoreBrace products are available for BRB shapes.

Integration: Buckling-restrained brace design is available for models that are transferred from RISAFloor.

Hot Rolled Steel - Code Check

AISC Code Check Results

Access the **Steel Code Check** spreadsheet by selecting the **Results** menu from the 'Explorer' panel, then clicking on **Code Checks**.

Envelope AISC 14th (360-10): ASD Steel Code Checks															
Hot Rolled Steel	Cold Formed Steel			Wood	Concrete Beams			Concrete Columns			Aluminum	Stainless			
Member	Shape	Code Check	Loc[ft]	LC	Shear Check	Loc[ft]	Dir	LC	Pnc/om...	Pnt/om...	Mnyy/om [k-ft]	Mnzz/om [k-ft]	Cb	Eqn	
1	F2_M35	W27X84	0.817	15.861	6	0.224	0	y	6	272.593	739.521	82.834	608.782	1.025	H1-1b
2	F2_M36	W30X90	0.914	14.421	6	0.321	28.842	y	6	319.391	787.425	86.577	706.088	1	H1-1b
3	F2_M37	W24X62	1.23	17.978	6	0.254	33.842	y	6	161.913	544.91	39.126	381.737	1.002	H1-1b
4	F2_M38	W24X76	1.165	14.421	6	0.347	28.842	y	6	294.012	670.659	71.357	499.002	1.001	H1-1b
5	F1_M35	W24X62	1.216	15.861	6	0.251	0	y	6	161.927	544.91	39.126	381.737	1.003	H1-1b
6	F1_M36	W27X84	1.056	14.421	6	0.324	28.842	y	6	313.957	739.521	82.834	608.782	1	H1-1b
7	F1_M37	W24X62	1.23	17.978	6	0.254	33.842	y	6	161.913	544.91	39.126	381.737	1.002	H1-1b
8	F1_M38	W27X84	1.043	14.421	6	0.32	28.842	y	6	313.957	739.521	82.834	608.782	1.001	H1-1b
9	CS20 (...)	W14X68	0.982	15	6	0.036	15.625	y	6	404.787	598.802	92.066	286.926	3	H1-1a
10	CS21 (...)	W14X61	0.74	15	6	0.036	15.625	y	6	360.572	535.928	81.836	254.491	3	H1-1a

**Note:**

- The [Member Detail Report](#) gives more values used to perform the code check.
- See [Spreadsheet Operations](#) to learn how to use **Find**, **Sort** and other options.
- See [Results View Settings - Members](#) to learn how to plot member results.
- See the code specific topics on the [Hot Rolled Steel - Design](#) page for additional restrictions / limitations.

Unity Check (UC) Results

The **UC Max** value represents the maximum Unity Check value produced by the interaction of the axial and bending stresses. This would be a factored ratio of actual to allowable stress, or demand vs capacity. The governing code equation that resulted in this value is displayed in the far right column for reference as well.

The **Shear UC** represents a similar ratio based on the shear provisions of the design code. The location for the shear check is followed by "y" or "z" to indicate the direction of the shear.

Generally, if UC Checks are less than 1.0, the member is considered passing. If either of them is greater than 1.0, the member is considered failing. If the value is greater than 9.999 it will be listed as "9.999".

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

Location of the Governing Unity Check

The **Loc** field tells at what location the maximum code check occurs measured from the I-joint location (starting end) of the member. See [Model Display Options – Members](#) to learn how to view the code check results graphically.

The moving load results are enveloped and the governing load combination and step location is shown for each result value under the "LC" column. The first number is the load combination, the second is the step number: (load combination - step number). See [Moving Loads](#) to learn more.

Design Capacities

For ASD 13th, 14th, and 15th Edition code checking, **Pnc/Om**, **Pnt/Om**, **Mnyy/Om** and **Mnzz/Om** are the design capacities divided by the Omega safety factors. Whereas the LRFD code checks will display the capacities multiplied by the Phi factor: **Phi*Pnc**, **Phi*Pnt**, **Phi*Mnyy**, and **Phi*Mnzz**.

For **Pnc** is calculated according to the provisions of Chapter E in the AISC 13th, 14th, or 15th Editions. **Pnt** is based on Chapter D. The **Mn** values are calculated based on Chapter F. Note that for RISA-, "zz" corresponds to "xx" in the AISC code, i.e. RISA- substitutes Mnzz for Mn_x, to maintain consistency with the member local axis system.



Note: AISC code checking requires a P-Delta analysis to satisfy the requirements of Chapter C, so if P-Delta analysis is NOT turned on (via the P-Delta flag) code checks won't be performed.

Cb Coefficient

The Cb coefficient is calculated based on the description presented in Chapter F, section F1. Refer to the section on [Cb input](#) for more information about this calculation.

Tapered Wide Flanges (ASD) - For Tapered WF shapes, the Cb value shown will be the Cb value that RISA- calculated internally, NOT the value that was entered and used for the "B" value. The values shown in the Cm fields will be the C'm values that were used for the Appendix F calculations. The controlling equation from Appendix F will be listed.

Tapered Wide Flanges (LRFD) - For Tapered WF shapes, the Cb value shown will be the Cb value that RISA- calculated, NOT the Cb value that was entered and used for the "B" value. The controlling equation from Chapter H will be listed.

Governing Equation

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. If there is no moment demand in the member being checked, then the program will calculate the unity ratio using Chapter D where it compares the axial demand to the axial capacity of the member using $P_u / (\Phi * P_c)$.



Note:

- The requirements of Section H3 of the LRFD code are satisfied since RISA-calculates and includes torsional warping effects.
- When the governing equation is flagged with an asterisk (*) that means the equation was modified slightly. Example AISC equation H1-1b was used, but the code check was not allowed to be less than $P_u / (\Phi * P_c)$.

Load Combination

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

Canadian Code Check Results

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

Envelope CAN/CSA-S16-14 Steel Code Checks																	
Hot Rolled Steel		Cold Formed Steel		Wood	Concrete Beams			Concrete Columns		Aluminum	Stainless						
	Member	Shape	Code Check	Loc[ft]	LC	Shear Check	Loc[ft]	Dir	LC	Cr [k]	Tr [k]	Mr y-y [k-ft]	Mr z-z [k-ft]	w2	w1 y-y	w1 z-z	CSA Eqn
1	F2_M35	W24X68	0.74	15.865	11	0.186	0	y	11	223.772	904.5	91.875	663.75	1.03	0.6	1	13_8_2
2	F2_M36	W27X84	0.716	14.425	11	0.225	28.85	y	11	305.009	1111.5	124.5	915	1	0.6	1	13_8_2
3	F2_M37	W24X68	0.702	17.983	11	0.176	33.85	y	11	223.759	904.5	91.875	663.75	1.002	0.6	1	13_8_2
4	F2_M38	W24X76	0.785	14.425	11	0.238	28.85	y	11	304.227	1008	107.25	750	1.001	0.6	1	13_8_2
5	F1_M35	W24X68	0.694	15.865	11	0.174	0	y	11	223.772	904.5	91.875	663.75	1.003	0.6	1	13_8_2
6	F1_M36	W24X62	1.123	14.425	11	0.264	28.85	y	11	205.67	819	58.875	573.75	1.001	0.6	1	13_8_2
7	F1_M37	W24X68	0.702	17.983	11	0.176	33.85	y	11	223.759	904.5	91.875	663.75	1.002	0.6	1	13_8_2
8	F1_M38	W24X55	1.265	14.425	11	0.298	28.85	y	11	155.228	729	49.875	502.5	1.001	0.6	1	13_8_2
9	CS20 (...)	W14X61	0.804	15	11	0.026	15.938	y	11	493.238	805.5	123	382.5	2.5	0.601	0.55	13_8_2b
10	CS21 (...)	W14X48	0.888	15	11	0.026	15.938	y	11	288.183	634.5	73.5	294	2.5	0.599	0.551	13_8_2b
11	CS22 (...)	W14X48	1.083	15	11	0.039	15.938	y	11	288.183	634.5	73.5	294	2.5	0.599	0.55	13_8_2b



Note:

- The [Member Detail Report](#) gives more values used to perform the code check.
- See [Spreadsheet Operations](#) to learn how to use **Find**, **Sort** and other options.
- See [Results View Settings - Members](#) to learn how to plot member results.
- See the code specific topics on the [Hot Rolled Steel - Design](#) page for additional restrictions / limitations.

Unity Check (UC) Results

The **UC Max** value represents the maximum Unity Check value produced by the interaction of the axial and bending stresses. This would be a factored ratio of actual to allowable stress, or demand vs capacity. The governing code equation that resulted in this value is displayed in the far right column for reference as well.

The **Shear UC** represents a similar ratio based on the shear provisions of the design code. The location for the shear check is followed by "y" or "z" to indicate the direction of the shear.

Generally, if UC Checks are less than 1.0, the member is considered passing. If either of them is greater than 1.0, the member is considered failing. If the value is greater than 9.999 it will be listed as "9.999".

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

Location of the Governing Unity Check

The **Loc** field tells at what location the maximum code check occurs measured from the I-joint location (starting end) of the member. See [Model Display Options – Members](#) to learn how to view the code check results graphically.

Design Capacities

Cr, **Mr_{yy}** and **Mr_{zz}** are the factored resistances calculated for the member. **Cr** is the compressive resistance, calculated according to the provisions of the design code. The **Mr** values are the moment resistances, calculated based on provisions of the design code. Note that for RISA-, "zz" corresponds to "xx" in the Canadian code, i.e. RISA- substitutes Mr_{zz} for Mr_x, to maintain consistency with the member local axis system.



Note: 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.

W2 Coefficient

The w₂ coefficient is calculated based on the description presented in Section 13.6. Whether or not the member is subject to sidesway is determined from the setting for the strong axis sway flag on the **Member Design** spreadsheet. The w₁ coefficients, described in Section 13.8.4 are also listed. These are also influenced by the sway flag settings.

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/C_b value that RISA- 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.)

Governing Equation

The final field lists the controlling equation for the code check. This will be one of the equations from Section 13.8.1, 13.8.2, or 13.9.



Note: The requirements of Section 15.11 are satisfied since RISA- calculates and includes torsional warping effects.

Load Combination

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

British Code Check Results

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

Envelope BS 5950-1:2000 Steel Code Checks																	
Hot Rolled Steel		Cold Formed Steel			Wood		Concrete Beams			Concrete Columns		Aluminum		Stainless			
	Member	Shape	Code Check	Loc[ft]	LC	Shear Check	Loc[ft]	Dir	LC	Pc [k]	Pt [k]	Mc y-y [k-ft]	Mc z-z [k-ft]	m LT	m y-y	m z-z	BS Eqn
1	M70	W10X12	0.345	9	9	0.062	18	y	4	177	177	6.829	45.424	0.925	1	0.95	4.8.3.2
2	M71	W10X17	0.858	18	10	0.084	18	y	4	137.419	249.5	7.398	15.037	0.44	1	0.4	4.8.3.3...
3	M72	W10X17	0.785	0	14	0.08	0	y	6	137.419	249.5	7.398	15.037	0.44	1	0.4	4.8.3.3...
4	M73	W10X12	0.504	9	14	0.062	18	y	7	12.452	177	4.588	45.424	0.925	0.95	0.95	4.8.3.3...
5	M74	W10X17	0.987	0	8	0.087	0	y	4	137.419	249.5	11.097	15.037	0.803	1	0.838	4.3.6
6	M75	W10X12	0.314	8.504	5	0.06	17.008	y	5	177	177	6.829	45.424	0.925	1	0.95	4.8.3.2
7	M76	W10X17	0.96	18	4	0.088	18	y	6	137.419	249.5	7.398	15.037	0.703	1	0.734	4.8.3.3...
8	M77	W10X17	0.856	18	10	0.084	18	y	4	137.419	249.5	11.097	15.037	0.44	1	0.4	4.3.6
9	M78	W10X12	0.248	9	9	0.045	18	y	6	177	177	6.829	45.424	0.925	1	0.95	4.8.3.2
10	M79	W10X17	1.174	0	4	0.093	0	y	6	219.687	249.5	11.097	15.037	0.743	1	0.776	4.3.6
11	M80	W10X12	0.34	8.752	10	0.045	17.504	y	4	13.14	177	4.588	45.424	0.925	0.95	0.95	4.8.3.3...



Note:

- The [Member Detail Report](#) gives more values used to perform the code check.
- See [Spreadsheet Operations](#) to learn how to use **Find**, **Sort** and other options.
- See [Results View Settings - Members](#) to learn how to plot member results.
- See the code specific topics on the [Hot Rolled Steel - Design](#) page for additional restrictions / limitations.

Unity Check (UC) Results

The **UC Max** value represents the maximum Unity Check value produced by the interaction of the axial and bending stresses. This would be a factored ratio of actual to allowable stress, or demand vs capacity. The governing code equation that resulted in this value is displayed in the far right column for reference as well.

The **Shear UC** represents a similar ratio based on the shear provisions of the design code. The location for the shear check is followed by "y" or "z" to indicate the direction of the shear.

Generally, if UC Checks are less than 1.0, the member is considered passing. If either of them is greater than 1.0, the member is considered failing. If the value is greater than 9.999 it will be listed as "9.999".

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

Location of the Governing Unity Check

The **Loc** field tells at what location the maximum code check occurs measured from the I-joint location (starting end) of the member. See [Model Display Options – Members](#) to learn how to view the code check results graphically.

Shear Capacity

The shear capacity, **Pv**, is calculated per section 4.2.3 using the listed equations for the shear area of each section type. Shear buckling per section 4.4.5 is also considered.

The reduction in design strength, "py", per table 9 in section 3.1.1 is automatically considered by the program. The reduction is based on the yield stress of the steel material used for the member and the maximum thickness of the cross section.

Compression Capacity

The compression capacity, **Pc** is calculated based on section 4.7. The effective lengths shown in table 22 are shown as "K" factors in the program (K*length is the effective length) and these can be approximated by the program based on the sway flags and the end release conditions of the member (the connection type is NOT considered.) The user can also specify the effective length by entering the appropriate [K factor](#).

For the case of a sway member having pinned ends, a value of 2.1 for K is used as table 22 does not address this condition. The program considers all H sections as I sections for purposes of picking the proper strut curve in table 23 in section 4.7.5. For the same table and section, the program does not do interpolation for shapes that have maximum thicknesses between 40 and 50 mm, per footnote 1.

Tension Capacity

The tension capacity, **Pt**, is calculated based on section 4.6 and currently done using the gross section.

Slenderness checks

Width to thickness checks are done for all shapes based on Tables 11 and 12 in section 3.5. Complete Effective section properties for the Area, Plastic Modulus, and Section Modulus, are done per section 3.5.6 and 3.6.

Moment Capacity

The moment capacity, **Mc**, is calculated per section 4.2.5, including consideration for the amount of shear at the section. The limit states of lateral torsional buckling, per section 4.3.6 and web buckling per section 4.4.4.2 are also considered.

Equivalent Uniform Moment Factors

The equivalent uniform moment factors (m_y , m_z , m_{LT}) used in section 4.8.3 and shown in table 26 of section 4 are calculated automatically by the program based on the sway flag and member moment diagram. The user can also enter their own values.

Governing Equation

The final field lists the controlling equation for the code check. Combined axial and bending stresses are computed based on the equation shown in section 4.8, with the governing equation reported. Combined bending and warping is calculated per the AISC specification and design guide.

Biaxial bending and biaxial shears on Pipes and solid circular bars are done using the square root sum of the squares of the forces applied in or about the local y and local z directions.

Load Combination

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

EuroCode Code Check Results

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

Envelope EC3 1993-1-1:2014 Steel Code Checks																		
Hot Rolled Steel	Cold Formed Steel		Wood	Concrete Beams			Concrete Columns		Aluminum		Stainless							
Member	Shape	Code Check	Loc[ft]	LC	Shear Check	Loc[ft]	Dir	LC	Nc.Rd [k]	Nt.Rd [k]	Mcy.Rd [k-ft]	Mcz.Rd [k-ft]	Cm.LT	Cmyy	Cmzz	C1	EC3 Eqn	
1	M70	W10X12	0.345	9	9	0.057	18	y	4	177	177	4,588	45,424	0.95	0.6	0.95	1.124	EN6_42
2	M71	W10X17	0.824	18	10	0.078	18	y	4	129,608	249.5	11,667	35.56	0.4	0.6	0.4	2.731	EN6_62
3	M72	W10X17	0.751	0	14	0.074	0	y	6	129,608	249.5	11,667	35,734	0.4	0.6	0.4	2.751	EN6_62
4	M73	W10X12	0.531	9	14	0.057	18	y	7	177	177	4,588	45,424	0.95	0.95	0.95	1.124	EN6_42
5	M74	W10X17	0.898	0	5	0.081	0	y	4	20,532	249.5	11,667	24,592	0.627	0.6	0.627	1.693	EN6_54
6	M75	W10X12	0.314	8,504	5	0.054	17,008	y	5	177	177	4,588	45,424	0.95	0.6	0.95	1.124	EN6_42
7	M76	W10X17	0.913	18	4	0.082	18	y	6	20,532	249.5	11,667	22,497	0.696	0.6	0.696	1.522	EN6_54
8	M77	W10X17	0.681	18	13	0.078	18	y	4	129,608	249.5	11,667	36,866	0.4	0.6	0.4	2.877	EN6_62
9	M78	W10X12	0.248	9	9	0.041	18	y	6	177	177	4,588	45,424	0.95	0.6	0.95	1.124	EN6_42
10	M79	W10X17	0.836	0	9	0.086	0	y	6	249.5	249.5	11,667	24,312	0.636	0.6	0.636	1.67	EN6_62
11	M80	W10X12	0.358	8,752	10	0.041	17,504	y	4	177	177	4,588	45,424	0.95	0.95	0.95	1.124	EN6_42

**Note:**

- The [Member Detail Report](#) gives more values used to perform the code check.
- See [Spreadsheet Operations](#) to learn how to use **Find**, **Sort** and other options.
- See [Results View Settings - Members](#) to learn how to plot member results.
- See the code specific topics on the [Hot Rolled Steel - Design](#) page for additional restrictions / limitations.

Unity Check (UC) Results

The **UC Max** value represents the maximum Unity Check value produced by the interaction of the axial and bending stresses. This would be a factored ratio of actual to allowable stress, or demand vs capacity. The governing code equation that resulted in this value is displayed in the far right column for reference as well.

The **Shear UC** represents a similar ratio based on the shear provisions of the design code. The location for the shear check is followed by "y" or "z" to indicate the direction of the shear.

Generally, if UC Checks are less than 1.0, the member is considered passing. If either of them is greater than 1.0, the member is considered failing. If the value is greater than 9.999 it will be listed as "9.999".

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

**Note:
Important !**

- The bending axes in the member local coordinate system are reversed between RISA and the Eurocode. All input and results are shown in the member local coordinate system used by RISA. For a wide flange shape, the “z-z” axis is parallel to the flanges and the “y-y” axis is parallel to the web.
- The reduction in yield strength, “fy”, per table 3.1 in section 3.2.2.1 is automatically considered by the program. The reduction is based only on the maximum thickness of the cross section. Yield strengths for sections with a maximum thickness of 40mm or less are not reduced. Yield strengths for sections with a maximum thickness greater than 40mm are reduced by 20 N/mm².

Location of the Governing Unity Check

The **Loc** field tells at what location the maximum code check occurs measured from the I-joint location (starting end) of the member. See [Model Display Options – Members](#) to learn how to view the code check results graphically.

Compressive Capacity

The compression capacity, **Nc.Rd** is calculated based on section 6.2.4 and 6.3.1. The effective lengths shown in tables E.2.1 and E.2.2 of the 1992 Eurocode are shown as “K” factors in the program (K*length is the effective length.) These can be approximated by the program based on the sway flags and the end release conditions of the member (the connection type is NOT considered.) The approximate K factor values for reasonable “pinned” and “fixed” conditions are taken from the British BS 5950-1:2000 code. The user can also specify the effective length by entering the appropriate [K factor](#). For the case of a sway member having pinned ends, a value of 2.1 for K is used as a reasonable limit.

Tension Capacity

The tension capacity, **Nt.Rd**, is calculated based on section 6.2.3 and currently done using the gross section.

Slenderness checks

Width to thickness checks are done for all shapes based on table 5.2 from the 2005 EuroCode.

Shear Capacity

The shear capacity, **Vpl.Rd**, is calculated per section 6.2.6 using the listed equations for the shear area of each section type. Shear buckling per section 5.6.3 of the 1992 Eurocode is also considered.

Moment Capacities

The moment capacity, **Mc.Rd**, is calculated per section 6.2.5, including consideration for the lateral-torsional buckling.

Equivalent Uniform Moment Factors

The equivalent uniform moment factors (C_m & C_{mLT} per the 2005 code) are calculated per Table B.3 of Annex B unless they are manually input by the user in the [Member Design Properties](#) spreadsheet.

The equivalent uniform moment factors (β_m & β_{mLT} per the 1992 code) used in section 5.5.4 and shown in figure 5.5.3 of section 5.5.4 are calculated automatically by the program based on the sway flag and member moment diagram unless they are manually input by the user in the [Member Design Properties](#) spreadsheet.

Governing Equation

The final field lists the controlling equation for the code check. Combined axial and moment is checked for the limit state of stress and high shear. The limit states of flexural buckling and lateral torsional buckling as well as web buckling are also considered. The governing section is also reported. Combined bending and warping is calculated per the AISC specification and increases the moment about the y-y axis.

Biaxial bending and biaxial shears on Pipes and solid circular bars are done using the square root sum of the squares of the forces applied in or about the local axes.

Load Combination

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

Indian Code Check Results

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

Envelope IS 800: 2007 Steel Code Checks																
Hot Rolled Steel		Cold Formed Steel		Wood	Concrete Beams			Concrete Columns		Aluminum		Stainless				
	Member	Shape	Code Check	Loc[ft]	LC	Shear Check	Loc[ft]	Dir	LC	Fa [ksi]	Ft [ksi]	Fb y-y [ksi]	Fb z-z [ksi]	Cmy	Cmz	IN Eqn
1	M70	W10X12	0.379	9	9	0.071	18	y	4	160.909	160.909	4.17	41.294	0.6	0.95	IN9.3.1.3
2	M71	W10X17	No Val													
3	M72	W10X17	No Val													
4	M73	W10X12	0.584	9	14	0.071	18	y	7	160.909	160.909	4.17	41.294	0.95	0.95	IN9.3.1.3
5	M74	W10X17	No Val													
6	M75	W10X12	0.345	8.504	5	0.068	17.008	y	5	160.909	160.909	4.17	41.294	0.6	0.95	IN9.3.1.3
7	M76	W10X17	No Val													
8	M77	W10X17	No Val													
9	M78	W10X12	0.273	9	9	0.052	18	y	6	160.909	160.909	4.17	41.294	0.6	0.95	IN9.3.1.3
10	M79	W10X17	No Val													
11	M80	W10X12	0.394	8.752	10	0.051	17.504	y	4	160.909	160.909	4.17	41.294	0.95	0.95	IN9.3.1.3

**Note:**

- The [Member Detail Report](#) gives more values used to perform the code check.
- See [Spreadsheet Operations](#) to learn how to use **Find**, **Sort** and other options.
- See [Results View Settings - Members](#) to learn how to plot member results.
- See the code specific topics on the [Hot Rolled Steel - Design](#) page for additional restrictions / limitations.

Unity Check (UC) Results

The **UC Max** value represents the maximum Unity Check value produced by the interaction of the axial and bending stresses. This would be a factored ratio of actual to allowable stress, or demand vs capacity. The governing code equation that resulted in this value is displayed in the far right column for reference as well.

The **Shear UC** represents a similar ratio based on the shear provisions of the design code. The location for the shear check is followed by "y" or "z" to indicate the direction of the shear.

Generally, if UC Checks are less than 1.0, the member is considered passing. If either of them is greater than 1.0, the member is considered failing. If the value is greater than 9.999 it will be listed as "9.999".

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

Location of the Governing Unity Check

The **Loc** field tells at what location the maximum code check occurs measured from the I-joint location (starting end) of the member. See [Model Display Options – Members](#) to learn how to view the code check results graphically.

Compressive Capacity

The allowable compressive stress, **F_a**, is calculated based on section 5.1. The effective lengths shown in table 5.2 are shown as “K” factors in the program (K*length is the effective length) and these can be approximated by the program based on the sway flags and the end release conditions of the member (the connection type is NOT considered.) The user can also specify the effective length by entering the appropriate [K factor](#). For the case of a sway member having pinned ends, a value of 2.1 for K is used as table 5.2 does not address this condition.

Tension Capacity

The allowable tensile stress, **F_t**, is calculated based on section 4.1 and is currently done using the gross section.

Slenderness Checks

The maximum slenderness ratios shown in table 3.1 are reported in the warning log when they are exceeded. The program still calculates capacities for the members and reports a code check.

Width to thickness checks are done for all shapes based on section 3.5.2.1. Complete effective section properties for the Area and Section Modulus are calculated.

Shear Capacity

The allowable shear stress, **F_v**, is used per section 6.4.1. This value is checked per the maximum shear stress in the section based on elastic theory.

Moment Capacity

The strong axis allowable bending stress, **F_b**, is calculated per section 6.2. The limit state of lateral torsional buckling, per sections 6.2.4 and 6.2.6, is considered. The allowable bending stress is increased per section 6.2.4.1 as appropriate. The weak axis bending stress is assigned per section 6.2.5.

Equivalent Uniform Moment Factor, C_m

The equivalent uniform moment factor, **C_m**, used in section 7.1 are calculated automatically by the program based on the sway flag per section 7.1.3. The user can also enter their own values in the [Member Design Parameters](#) spreadsheet.

Governing Equation

The final field lists the controlling equation for the code check. Combined axial and bending stresses are computed based on the equations shown in section 7.1, with the governing equation reported. Combined bending and warping is calculated per the AISC ASD specification.

Biaxial bending and biaxial shears on Pipes and solid circular bars are done using the square root sum of the squares of the forces applied in or about the local y and local z directions.

Load Combination

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

Australian Code Check Results

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

Envelope AS 4100-1998 Steel Code Checks																
Hot Rolled Steel	Cold Formed Steel			Wood	Concrete Beams			Concrete Columns		Aluminum	Stainless					
Member	Shape	Code Check	Loc[ft]	LC	Shear Check	Loc[ft]	Dir	LC	Phi*N [k]	Phi*Nt [k]	Phi*Mny [k-ft]	Phi*Mnz [k-ft]	alpha_m	AS Eqn		
1	M70	W10X12	0.341	9	9	0.072	18	y	4	140.754	159.3	5.739	45.848	1	AS8_3_...	
2	M71	W10X17	0.748	18	10	0.1	18	y	4	120.76	224.55	9.988	36.04	2.5	AS8_4_...	
3	M72	W10X17	0.662	0	14	0.095	0	y	6	120.76	224.55	9.988	36.04	2.5	AS8_4_...	
4	M73	W10X12	0.456	9	14	0.072	18	y	7	140.754	159.3	6.193	47.25	1	AS8_3_...	
5	M74	W10X17	0.889	0	8	0.104	0	y	4	224.55	224.55	9.988	20.112	1.395	AS4_5_2	
6	M75	W10X12	0.311	8.504	5	0.069	17.008	y	5	140.754	159.3	5.739	45.848	1	AS8_3_...	
7	M76	W10X17	0.812	18	4	0.105	18	y	6	120.76	224.55	9.988	23.829	1.653	AS8_4_...	
8	M77	W10X17	0.746	18	10	0.1	18	y	4	224.55	224.55	9.988	36.04	2.5	AS4_5_2	
9	M78	W10X12	0.246	9	9	0.052	18	y	6	140.754	159.3	5.739	45.848	1	AS8_3_...	
10	M79	W10X17	1.1	0	4	0.11	0	y	6	224.55	224.55	9.988	22.188	1.539	AS4_5_2	
11	M80	W10X12	0.318	8.752	4	0.052	17.504	y	4	140.754	159.3	6.193	47.25	1	AS8_3_...	



Note:

- The [Member Detail Report](#) gives more values used to perform the code check.
- See [Spreadsheet Operations](#) to learn how to use **Find**, **Sort** and other options.
- See [Results View Settings - Members](#) to learn how to plot member results.
- See the code specific topics on the [Hot Rolled Steel - Design](#) page for additional restrictions / limitations.

Unity Check (UC) Results

The **UC Max** value represents the maximum Unity Check value produced by the interaction of the axial and bending stresses. This would be a factored ratio of actual to allowable stress, or demand vs capacity. The governing code equation that resulted in this value is displayed in the far right column for reference as well.

The **Shear UC** represents a similar ratio based on the shear provisions of the design code. The location for the shear check is followed by "y" or "z" to indicate the direction of the shear.

Generally, if UC Checks are less than 1.0, the member is considered passing. If either of them is greater than 1.0, the member is considered failing. If the value is greater than 9.999 it will be listed as "9.999".

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

Location of the Governing Unity Check

The **Loc** field tells at what location the maximum code check occurs measured from the I-joint location (starting end) of the member. See [Model Display Options – Members](#) to learn how to view the code check results graphically.

Compressive Capacity

The allowable compressive stress, **Phi*N**, is calculated based on clause 6.3. The effective lengths shown in Figure 4.6.3.2 are shown as “K” factors in the program (K*length is the effective length) and these can be approximated by the program based on the sway flags and the end release conditions of the member (the connection type is NOT considered.) The user can also specify the effective length by entering the appropriate [K factor](#).

Tension Capacity

The allowable tensile stress, **Phi*Nt**, is calculated based on clause 7.2 and is currently done using the gross section.

Slenderness Checks

Width to thickness checks are done for all shapes based on Table 5.2. Complete effective section properties for the Area and Section Modulus are calculated.

Shear Capacity

The allowable shear stress, **Phi*Vv**, is used per clause 5.11. This value is checked per the maximum shear stress in the section based on elastic theory.

Moment Capacity

The strong axis allowable bending stress, **Phi*Mn**, is calculated per section 5. The limit state of lateral torsional buckling, per clause 5.6 is considered.

Moment Modification Factor, α_m

The moment modification factor, **Alpha_m**, is calculated automatically by the program based on clause 5.6.1.1. When explicit unbraced lengths are used, the program does not have the member

segment information to calculate α_m . It will conservatively assume α_m to be 1.0. The user can also enter their own values in the [Member Design Parameters](#) spreadsheet.

Governing Equation

The final field lists the controlling equation for the code check. Combined axial and bending stresses are computed based on the equations shown in clause 8.4, with the governing equation reported. Combined bending and warping is calculated per the AISC ASD specification.

Biaxial bending and biaxial shears on Pipes and solid circular bars are done using the square root sum of the squares of the forces applied in or about the local y and local z directions.

Load Combination

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

New Zealand Code Check Results

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

Envelope NZS 3404: 1997 Steel Code Checks															
Hot Rolled Steel		Cold Formed Steel		Wood	Concrete Beams		Concrete Columns		Aluminum	Stainless					
Member	Shape	Code Check	Loc[ft]	LC	Shear Check	Loc[ft]	Dir	LC	Phi*N [k]	Phi*Nt [k]	Phi*Mny [k-ft]	Phi*Mnz [k-ft]	alpha_m	NZS Eqn	
1	M70	W10X12	0.341	9	9	0.072	18	y	4	140.754	159.3	5.739	45.848	1	NZ8_3_...
2	M71	W10X17	0.748	18	10	0.1	18	y	4	120.76	224.55	9.988	36.04	2.5	NZ8_4_...
3	M72	W10X17	0.662	0	14	0.095	0	y	6	120.76	224.55	9.988	36.04	2.5	NZ8_4_...
4	M73	W10X12	0.466	9	14	0.072	18	y	7	140.754	159.3	5.994	46.637	1	NZ8_3_...
5	M74	W10X17	0.889	0	8	0.104	0	y	4	224.55	224.55	9.988	20.112	1.395	NZ8_4_...
6	M75	W10X12	0.311	8.504	5	0.069	17.008	y	5	140.754	159.3	5.739	45.848	1	NZ8_3_...
7	M76	W10X17	0.812	18	4	0.105	18	y	6	120.76	224.55	9.988	23.829	1.653	NZ8_4_...
8	M77	W10X17	0.746	18	10	0.1	18	y	4	224.55	224.55	9.988	36.04	2.5	NZ8_4_...
9	M78	W10X12	0.246	9	9	0.052	18	y	6	140.754	159.3	5.739	45.848	1	NZ8_3_...
10	M79	W10X17	1.1	0	4	0.11	0	y	6	224.55	224.55	9.988	22.188	1.539	NZ8_4_...
11	M80	W10X12	0.323	8.752	4	0.052	17.504	y	4	140.754	159.3	5.994	46.637	1	NZ8_3_...



Note:

- The [Member Detail Report](#) gives more values used to perform the code check.
- See [Spreadsheet Operations](#) to learn how to use **Find**, **Sort** and other options.
- See [Results View Settings - Members](#) to learn how to plot member results.
- See the code specific topics on the [Hot Rolled Steel - Design](#) page for additional restrictions / limitations.

Unity Check (UC) Results

The **UC Max** value represents the maximum Unity Check value produced by the interaction of the axial and bending stresses. This would be a factored ratio of actual to allowable stress, or demand vs capacity. The governing code equation that resulted in this value is displayed in the far right column for reference as well.

The **Shear UC** represents a similar ratio based on the shear provisions of the design code. The location for the shear check is followed by "y" or "z" to indicate the direction of the shear.

Generally, if UC Checks are less than 1.0, the member is considered passing. If either of them is greater than 1.0, the member is considered failing. If the value is greater than 9.999 it will be listed as "9.999".

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

Location of the Governing Unity Check

The **Loc** field tells at what location the maximum code check occurs measured from the I-joint location (starting end) of the member. See [Model Display Options – Members](#) to learn how to view the code check results graphically.

Compressive Capacity

The allowable compressive stress, **Phi*N**, is calculated based on clause 6.3. The effective lengths shown in Figure 4.6.3.2 are shown as "K" factors in the program (K*length is the effective length) and these can be approximated by the program based on the sway flags and the end release conditions of the member (the connection type is NOT considered.) The user can also specify the effective length by entering the appropriate [K factor](#).

Tension Capacity

The allowable tensile stress, **Phi*Nt**, is calculated based on clause 7.2 and is currently done using the gross section.

Slenderness Checks

Width to thickness checks are done for all shapes based on Table 5.2. Complete effective section properties for the Area and Section Modulus are calculated.

Shear Capacity

The allowable shear stress, **Phi*Vv**, is used per clause 5.11. This value is checked per the maximum shear stress in the section based on elastic theory.

Moment Capacity

The strong axis allowable bending stress, $\Phi \cdot M_n$, is calculated per section 5. The limit state of lateral torsional buckling, per clause 5.6 is considered.

Moment Modification Factor, α_m

The moment modification factor, α_m , is calculated automatically by the program based on clause 5.6.1.1. The user can also enter their own values in the [Member Design Parameters](#) spreadsheet.

Governing Equation

The final field lists the controlling equation for the code check. Combined axial and bending stresses are computed based on the equations shown in clause 8.4, with the governing equation reported. Combined bending and warping is calculated per the AISC ASD specification.

Biaxial bending and biaxial shears on Pipes and solid circular bars are done using the square root sum of the squares of the forces applied in or about the local y and local z directions.

Load Combination

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

Tapered Member Results

The spreadsheet results for tapered members gives the same information as other [AISC Code Checks](#). However, the Detail Reports for tapered members designed to *AISC Design Guide 25* give some additional information.

As shown in the image below, the code checks list the member capacity calculated for axial, flexure, and shear. In addition to the capacity values, the table reports the equation number used to arrive at that capacity and the failure mode which caused it (please see the example below). The table also gives the slenderness parameters that apply for that failure state.

AISC 15th (360-16): ASD Code Check -- (Design Guide 25: Tapered Members)				
Limit State	Required	Available	Unity Check	Result
Applied Loading - Bending/Axial	-	-	-	-
Applied Loading - Shear + Torsion	-	-	-	-
Axial Tension Analysis	0 k	297.53 k	-	-
Axial Compression Analysis	0 k	90.866 k	-	-
Flexural Analysis (Strong Axis)	13.365 k-ft	45.138 k-ft	-	-
Flexural Analysis (Weak Axis)	0 k-ft	23.236 k-ft	-	-
Shear Analysis (Major Axis y)	6.233 k	54.571 k	0.114	PASS
Shear Analysis (Minor Axis z)	0 k	101.048 k	0	PASS
Bending & Axial Interaction Check (UC Bending Max)	-	-	0.296	PASS



Note: We recommend clicking on each “Limit State” to access the expanded report for detailed calculations.

The following failure modes are listed for axial and strong axis bending:

- **(z-z)** : In-Plane or Strong Axis flexural buckling of the member due to axial force.
- **(z-z)** : Out-of-Plane or Weak Axis flexural buckling of the member due to axial force.
- **TB** : Torsional buckling of the member due to axial force.
- **FTB** : Flexural Torsional Buckling of the member due to axial force.
- **CAT** : Constrained Axis Torsional Buckling of the member due to axial force.
- **FLB** : Flange Local Buckling due to flexure.
- **WLB** : Web Local Buckling.
- **TFY** : Tension Flange Yielding due to flexure.
- **CFY** : Compression Flange Yielding due to flexure.
- **LTB** : Lateral Torsional Buckling of the member due to flexure.

Please see the Tapered Member [design](#) topic for more information about these limit states.

BRB Member Results

The spreadsheet results for BRB members are summarized under the BRB tab in the Code Checks spreadsheet. It covers the brace UC check values under axial load and the adjustment factors for brace stiffness (K_f) and strength (ω , β). It also includes the tension and compression capacities of the braces. See Buckling-Restrained Brace (BRB) Members for additional information on the parameters.

Envelope AISC 15th (360-16): ASD BRB Code Checks											
Hot Rolled Steel	Cold Formed Steel	Wood	Concrete Beams	Concrete Columns	Aluminum	Stainless	BRB				
Member	Shape	Code C...	Loc[ft]	LC	Kf	omega	beta	Pnc/om...	Pnt/om...	Eqn	
1	M41	CoreBRB_2.75	0.952	0	4	1.431	1.257	1.156	62.575	62.575	F4-1
2	M42	CoreBRB_2.75	0.952	0	2	1.431	1.257	1.156	62.575	62.575	F4-1
3	M43	CoreBRB_2.75	0.952	0	4	1.431	1.257	1.156	62.575	62.575	F4-1
4	M44	CoreBRB_2.75	0.952	0	2	1.431	1.257	1.156	62.575	62.575	F4-1
5	M45	CoreBRB_3.00	0.899	0	1	1.457	1.267	1.161	68.263	68.263	F4-1
6	M46	CoreBRB_3.00	0.899	0	3	1.457	1.267	1.161	68.263	68.263	F4-1
7	M47	CoreBRB_3.00	0.899	0	1	1.457	1.267	1.161	68.263	68.263	F4-1
8	M48	CoreBRB_3.00	0.899	0	3	1.457	1.267	1.161	68.263	68.263	F4-1

The detail report is also customized for BRB members. For force diagrams, only axial load related diagrams are reported. No shear/bending/torsion diagrams are reported since BRBs are supposed to resist axial load only. In the BRB Check tab, the Kf, omega, and beta values are reported and the axial compression/tension code checks are also presented:

BRB Check	-61.372 k	68.263 k	0.899	Pass
$K_f = 1.457$	Brace stiffness modifier			
Adjustment Factors at 2% Story Drift				
$\Delta_{b,x} = 0.157 \text{ ft}$	Brace Deformation at 2% story drift			
$CB_{2\%drift} = 0.017$	Brace Strain at 2% story drift			
$\omega_1 = 1.267$	Strain hardening adjustment factor			
$\beta_1 = 1.161$	Compression strength adjustment factor			
Adjustment Factors at 2 x Design Drift				
$CB_{2xDD} = 0.009$	Brace Strain at 2 x Design Drift			
$\omega_2 = 1.132$	Strain hardening adjustment factor			
$\beta_2 = 1.092$	Compression strength adjustment factor			
$\omega = \max(\omega_1, \omega_2) = 1.267$	Governing strain hardening adjustment factor			
$\beta = \max(\beta_1, \beta_2) = 1.161$	Governing compression strength adjustment factor			
Axial Tension Analysis				
$A_{sc} = 3 \text{ in}^2$	Area of steel core			
$F_{ymin} = 38 \text{ ksi}$	Lower bound yield strength			
$\omega_t = 1.67$				
$P_{nt} = \frac{F_{ymin} A_{sc}}{\omega_t} = -68.263 \text{ k}$	Allowable tensile strength			
$P_{r(ten)} = -61.372 \text{ k}$				
$UC_t = \frac{P_{r(ten)}}{P_{nt}} = 0.899$				
Axial Compression Analysis				
$\omega_c = 1.67$				
$P_{nc} = \frac{F_{ymin} A_{sc}}{\omega_c} = 68.263 \text{ k}$	Allowable compressive strength			
$P_{r(comp)} = 0 \text{ k}$				
$UC_c = \frac{P_{r(comp)}}{P_{nc}} = 0$				



Note: BRB axial load demand is obtained by analysis neglecting the gravity loads effects. However, the axial load diagrams are displaying the analysis result based on the original load combination including the gravity loads effects. So the reported axial load demand might be different from the envelop results in the force diagrams.

Stainless Steel

Stainless steel shape properties and design are based on the recommendations in the AISC Design Guide 27 Structural Stainless Steel. The procedure outlined in the design guide follows the 2010 AISC Specification and 14th Edition Steel Manual.

Database Shapes

Shapes are organized by **Shape Type** and **Database / Manufacturer**. Common shapes are supported such as wide flanges, tubes, pipes, channels, etc. You can type in the names directly, select shapes from these databases, or add your own shapes.

RISA currently supports only the AISC steel database.

3D Shape Selection
? X


General


Hot Rolled


Cold Formed


Wood


Aluminum


Stainless

Database AISC SS ▼

Shape Type

WF

Single Angle

Tube

Solid Rectangle

Pipe

Solid Round Bar

Channel

WT

Double Angle

Shape Name

Search by name

🔍

- W44X335_SS
- W44X290_SS
- W44X262_SS
- W44X230_SS
- W40X655_SS

Preview

Add

View

Edit

Copy

Delete



Save Database as default

OK

Cancel

Select a Stainless Steel Database Shape

To select a Stainless Steel database shape:

- From the **Stainless** tab on the **Section Sets** spreadsheet, or the **Primary** tab of the **Members** spreadsheet, move the cursor to the **Shape** field and click .

- Specify the database and shape type you wish to use and then select from the list of available shapes.

Shape Designations: Stainless steel shapes are similar to the AISC database for all shapes. See [Hot Rolled - Databases](#) for information on the shape designations.

Shape Designations

Stainless steel shapes are similar to the AISC database for all shapes. See [Hot Rolled - Databases](#) for information on the shape designations.

Design

Full code checking member optimization can be applied to standard stainless steel shapes based on the AISC Design Guide 27 Structural Stainless Steel which refers to the 2010 AISC Specification and 14th Edition Steel Manual.

How To Apply a Steel Design Code

- On the **Model Settings - Codes** tab, select the steel code from the drop down list.

Materials	Codes	Stiffness Adjustment
Hot Rolled Steel	AISC 15th (360-16): ASD	Yes (Iterative)
	Seismic Detailing: AISC 341-16 and AISC 358-16	
Connections	AISC 15th (360-16): ASD	
Cold Formed Steel	AISI S100-20: ASD	Yes (Iterative)
	CFS Walls: AISI S400-20 & AISI S240-20	
Wood	AWC NDS-18 / SDPWS-21 ASD	
	Temperature: < 100F	
Concrete	ACI 318-19	
Masonry	TMS 402-16: ASD	
Aluminum	AA ADM1-20: ASD	Yes (Iterative)
	Structure Type: Building	
Stainless	AISC 14th (360-10): ASD	Yes (Iterative)

Set as Default Apply Cancel

- On the **Stainless** tab of the **Members** spreadsheet, enter the appropriate bracing information and factors.



Note: For code checking to be performed on a member, the member must be defined with a database shape on the **Section Sets** or **Members** spreadsheet.

Design Parameters Spreadsheet

The **Stainless Steel Design Parameters** spreadsheet records the design parameters for the stainless steel code checks. It can be accessed by selecting **Members** from the 'Data Entry' section of the 'Explorer' panel and then selecting the **Stainless** tab. These parameters are defined for each individual member and can also be assigned graphically. See [Modifying Member Design](#) to learn how to do this.

Stainless Steel Design Parameters												
Primary	Advanced	Hot Rolled	Cold Formed	Wood	Concrete Beam	Concrete Column	Aluminum	Stainless	RISACONNECTION			
	Label	Shape	Length [ft]	Lb y-y [ft]	Lb z-z [ft]	Lcomp top [ft]	Lcomp bot [ft]	L-Torque [ft]	K y-y	K z-z	Cb	Function
1	M311	W8X10...	10									Lateral
2	M312	W8X10...	10						1	1		Lateral
3	M313	W8X10...	10									Lateral
4	M314	W8X10...	10									Lateral
5	M337	W8X10...	3.334									Lateral
6	M338	W8X10...	3.334									Lateral
7	M345	W8X10...	3.334									Lateral
8	M347	W8X10...	3.334									Lateral
9	M211	W8X10...	24.322									Lateral
10	M215	W8X10...	24.322									Lateral
11	M218	W8X10...	24.322									Lateral

The design for stainless steel follows the design for [Hot-Rolled Steel Design](#) with a few exceptions outlined below.

Implementation Notes

b/t ratio

The stainless steel **b/t ratios** differ from Tables B4.1a and B4.1b in Chapter B of the 14th Edition AISC code. The **b/t ratios** used are listed in the AISC Design Guide 27 Stainless Structural Steel in Tables 3-1 and 3-2.

Axial Compression

The guidelines given in AISC Specification Chapter E apply except with some modifications to the equations and some additional limitations for slender members. See AISC Design Guide 27 for the specific changes.

Bending Capacity

The guidelines given in AISC Specification Chapter F apply except with some modifications to the equations and some additional limitations for slender members. See AISC Design Guide 27 for the specific changes.

Flexure Code Checks

The analysis will be provided for flexure but there will not be any flexure code checks for single angles, tees, sections with slender webs, unequal angles, equal legs angles with slender sections, and round HSS.

Round HSS Shear Buckling

The limit state of shear buckling for round HSS (AISC Specification Equations G6-2a and G6-2b) are beyond the scope of the AISC Design Guide 27 and thus are not included in RISA. The analysis will be provided but there will not be any shear code checks for round HSS.

Combined Torsion

The design of members subject to combined torsion, flexure, shear, and/or axial force is beyond the scope of the AISC Design Guide 27.

Slender WT's and Angle Sections

The Design Guide does not directly address the case of WT's with slender webs or angles with slender webs. Therefore, the program uses modified versions of the similar equations from the AISC specification. This should be considered only an approximation of the capacity of these slender members and should be reviewed and accepted (or rejected) based on the engineering judgment of the user. The equations used are the following:

For slender single angles (or double angles when the legs spacing is nonzero), the equations used in RISA are:

$$Q_s := 1.407 - 1.071 \cdot \frac{b}{t} \cdot \sqrt{\frac{F_y}{E}} \quad \text{When} \quad 0.38 \cdot \sqrt{\frac{E}{F_y}} < \frac{b}{t} < 0.8 \cdot \sqrt{\frac{E}{F_y}}$$

$$Q_s := 0.338 \cdot \frac{E}{F_y \cdot \left(\frac{b}{t}\right)^2} \quad \text{When} \quad \frac{b}{t} > 0.8 \cdot \sqrt{\frac{E}{F_y}}$$

For WT's with slender stems / webs the equations used in RISA are:

$$Q_s := 1.329 - 0.865 \cdot \frac{b}{t} \cdot \sqrt{\frac{F_y}{E}} \quad \text{When} \quad 0.38 \cdot \sqrt{\frac{E}{F_y}} < \frac{b}{t} < 0.9 \cdot \sqrt{\frac{E}{F_y}}$$

$$Q_s := 0.44 \cdot \frac{E}{F_y \cdot \left(\frac{b}{t}\right)^2} \quad \text{When} \quad \frac{b}{t} > 0.9 \cdot \sqrt{\frac{E}{F_y}}$$

Limitations

- The code checks for Stainless in RISA are based on the AISC 360-10 Specification and 14th Ed Manual only.
- Stainless code checks only apply to Hot-Rolled Stainless; ASCE 8 is for cold formed stainless and is not considered in RISA.
- RISA does not consider the Precipitation Hardening Stainless Steel tension rods which have different safety and resistance factors.
- Serviceability checks for deflection based on the secant modulus Es are not implemented at this time. Deflections checks use the modulus of elasticity specified in the Materials spreadsheet.

Stainless Code Check Results

Access the **Stainless Steel Code Check** spreadsheet by selecting the **Results** tab ▶ **Envelope** (or **LC**) button **Code Check**. Alternatively, on the **Results** toolbar you can select **Code Check**.

Envelope AISC 14th (360-10): ASD Stainless Steel Code Checks															
Hot Rolled Steel		Cold Formed Steel		Wood	Concrete Beams			Concrete Columns			Aluminum	Stainless			
	Member	Shape	Code Check	Loc[ft]	LC	Shear Check	Loc[ft]	Dir	LC	Pnc/om...	Pnt/om [k]	Mnyy/om [k-ft]	Mnzz/om [k-ft]	Cb	Eqn
1	M311	W8X...	0.076	1.771	14	0.014	10	y	14	87.924	366.619	17.134	56.042	1.24	H1...
2	M312	W8X...	0.068	5.209	13	0.012	0	y	1	87.924	366.619	17.134	50	1.1...	H1...
3	M313	W8X...	0.085	1.042	13	0.014	10	y	13	87.924	366.619	17.134	57.005	1.2...	H1...
4	M314	W8X...	0.066	8.959	14	0.017	0	y	14	87.924	366.619	17.134	71.345	1.5...	H1...
5	M337	W8X...	0.139	3.334	1	0.01	3.334	y	13	303.283	366.619	17.134	91.552	1.1...	H1...
6	M338	W8X...	0.156	3.334	1	0.014	0.313	y	14	303.283	366.619	17.134	91.552	1.7...	H1...
7	M345	W8X...	0.156	3.334	1	0.015	3.334	y	13	303.283	366.619	17.134	91.552	1.76	H1...
8	M347	W8X...	0.139	3.334	1	0.01	0	y	17	303.283	366.619	17.134	91.552	1.1...	H1...

**Note:**

- The [Member Detail Report](#) gives more values used to perform the code check.
- See [Spreadsheet Operations](#) to learn how to use **Find**, **Sort** and other options.
- See [Results View Settings - Members](#) to learn how to plot member results.

Unity Check (UC) Results

The results for stainless steel follows the same results reporting as that for [Hot-Rolled Steel Results](#).

Loads

As you input loads they are grouped into separate sets of [Basic Load Cases](#). These are the basic building blocks of the loads applied to the structure, consisting of discrete loads and self-weight of the structure. The basic load cases can also be grouped into load categories such as dead load and live load. In the end, the model will be solved against load combinations that are built with the basic load cases and categories.

You can view the loads on your model. This is an excellent way to verify the loads that are on the model. You can plot them by load case, load category or load combination. See [Model View Settings - Loads](#) to learn to do this.

Each of the discrete load types is described in it's own section:

- [Area Loads](#)
- [Distributed Loads](#)
- [Joint Loads/Forced Displacements](#)
- [Moving Loads](#)
- [Point Loads](#)
- [Surface Loads](#)
- [Thermal Loads](#)

Self Weight (Gravity Load)

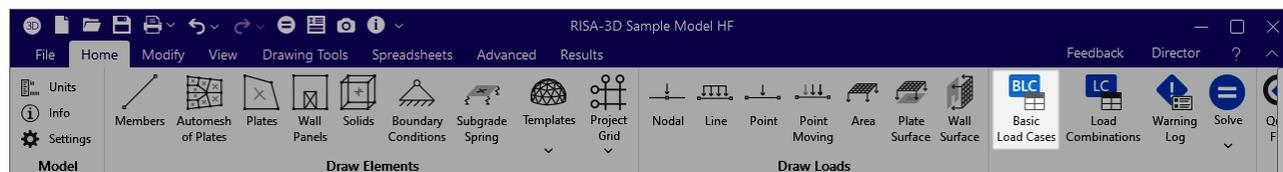
The structure's self-weight can be automatically calculated and included in a solution. The inclusion of self-weight can be specified within a Basic Load Case or as part of a Load Combination.

The self-weight is calculated as a full-length uniform load across each member of the model and as a surface load on the plates. The magnitude of the load is the area times the material weight density.



Note: If a member has offsets defined, the offset distances are not included in the self-weight calculation.

Add Self Weight to a Basic Load Case



- Select the **Basic Load Case** spreadsheet by clicking the **Basic Load Cases** icon in the **Home** ribbon. On the line of the load case you wish to contain self-weight, enter a load factor in the appropriate direction column.

For example, if you wish to apply 90% of the self-weight in the negative Y direction (down), you would put -0.9 as the factor in the **Y Gravity** column.

Add Gravity Load to a Load Combination

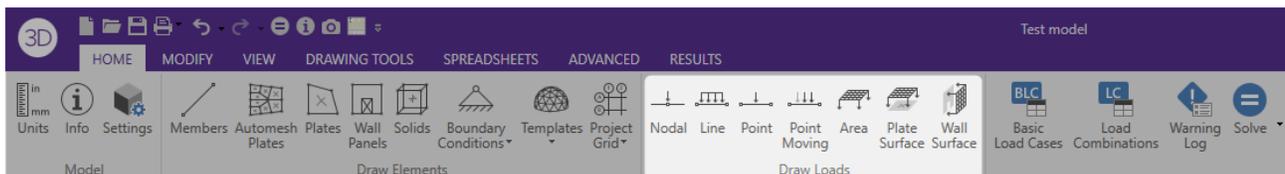
- To include the self-weight in a Load Combination, put "X", "Y" or "Z" in the BLC field. The letter used indicates which global direction the self-weight is applied in. Also put a factor in the Factor field. For example, if you wish to apply 90% of the self-weight in the negative Y direction (down), you would put "Y" as the BLC and "-0.9" as the BLC factor.



Note:

- You can have multiple self-weight definitions, i.e. you could have self-weight applied in the "Y" and "X" directions in the same or different load cases if you wished.
- Don't inadvertently include the self-weight of the structure twice by specifying it within a combination that includes a Basic Load Case that also contains the self-weight.

Draw Loads



You can graphically apply loads to the model. You can apply one load at a time by clicking the mouse, or apply loads to entire selections at once. All of the graphical loading tools can be found in the **Draw Loads** section of the **Home** ribbon.

Modify Loads

Modifying loads can be done in the spreadsheets. In the 'Explorer' panel, click on the spreadsheet that you wish to modify. You can also go to the 'Spreadsheets' ribbon, click the **Data Entry** icon and then choose the spreadsheet that you wish to modify. You can then move through different Basic Load Cases by using the drop-down list on the **Window Toolbar**.

See [Spreadsheet Operations](#) to learn how you can modify the loads. You can also copy or delete entire Basic Load Cases from the **Basic Load Case** spreadsheet. See [Copying Basic Load Cases](#) and [Deleting Basic Load Cases](#) for more information.

Alternatively you can directly click on the loads in the 3D View and edit the selected load within the Properties panel.

From the Properties panel, you can change the configuration of the load as well as edit or create a basic load case by clicking the triple dot in the BLC parameter.

Properties	
Area Load ^	
Node A	N419
Node B	N420
Node C	N421
Node D	N415
Direction	Y v
BLC	2: Superimposed... v ...
Load Direction	B-C v
Magnitude, ksf	-0.01
Inactive	Active v

Delete Loads

Deleting loads can be done in the spreadsheets. In the 'Explorer' panel, click on the spreadsheet that you wish to modify. You can also go to the 'Spreadsheets' ribbon, click the **Data Entry** icon and then choose the spreadsheet that you wish to modify. You can then move through different Basic Load Cases by using the drop-down list on the **Window Toolbar**.

You can use spreadsheet operations such to help modify the loads. You can also delete entire Basic Load Cases from the **Basic Load Case** spreadsheet. See [Deleting Basic Load Cases](#).

Alternatively you can directly click on the loads in the 3D View and press the **Delete** key.

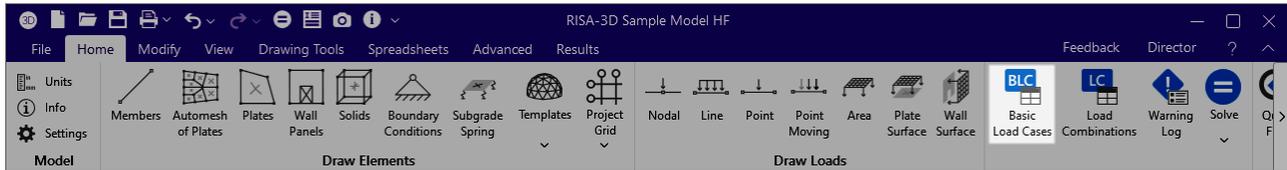
Basic Load Cases

When loads are defined they are grouped into basic load cases (BLC's). You are allowed up to 1000 separate Basic Load Cases. These are the basic building blocks of the final load combinations applied to the structure. The basic load cases may be assigned to load categories such as dead load and live load. These basic cases and categories are then combined to define load combinations used in analysis. A BLC can be comprised of any type of load, such as joint loads, distributed loads, member point loads, etc.

Any time you are applying or viewing loads, graphically or in the spreadsheets, they are assigned to the current basic load case. The current basic load case is displayed in the **Model View Toolbar** in a

drop down list. To change to another BLC simply choose it from this list. If you wish to change the name the BLC's You can do this in the **Basic Load Case** spreadsheet.

Access the **Basic Load Cases** spreadsheet from the 'Home' ribbon by clicking on the **Basic Load Cases** icon, or from the 'Explorer' panel under the 'Data Entry' section.



When Adding Loads to a Basic Load Case

- When adding loads graphically, click on the toolbar button for the type of load you wish to add and specify the BLC along with the load definition.
- When adding loads with the spreadsheets, open the spreadsheet for the load type you want to add and select the BLC from the list on the Window Toolbar.

Basic Load Case Spreadsheet

The **Basic Load Cases** spreadsheet groups and summarizes the loads applied to the model. Access the **Basic Load Cases** spreadsheet from the 'Home' ribbon by clicking on the **Basic Load Cases** icon. Note that you can also access it from the 'Explorer' panel under the 'Data Entry' section.

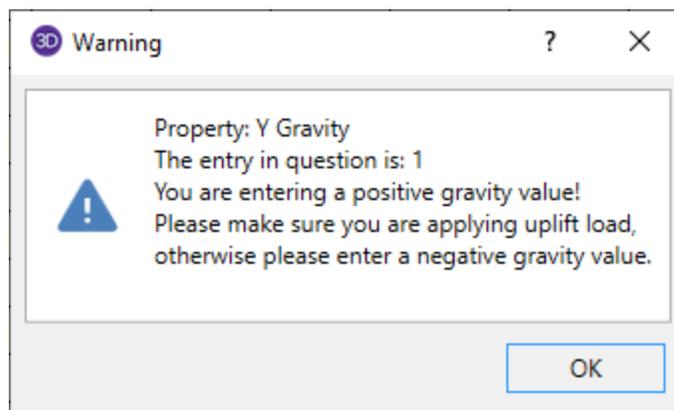
	BLC Description	Category	X Gravity	Y Gravity	Z Gravity	Nodal	Point	Distributed	Area(Member)	Surface(Plate/Wall)
1	Self Weight	DL		-1						
2	Superimposed Dead	DL						15	2	320
3	Occupancy Live	LL							2	320
4	Roof Live	RLL							3	
5	Equipment Load	DL				4				
6		None								
7		None								
8		None								
9		None								

Information about Basic Load Cases (BLC's) is recorded on the **Basic Load Case Spreadsheet** and the loads themselves are recorded in the load spreadsheets for each load type.

You can enter descriptions for each BLC in the first column. These descriptions are primarily for your own use. The descriptions will print as part of the input printout and you can also display the description when plotting the loads for the BLC or choosing them from the list.

The second column is used to assign the BLC to a load category such as Dead Load or Live Load. Simply choose the category from the drop down list. You can then build load combinations for analysis by referring to the categories rather than list each basic load case.

The next three columns may be used to specify that the self-weight of the structure be included in a load case. Simply enter a factor in the column that represents the direction of the self-weight. Typically you will enter a value of “-1” in the **Y Gravity** column assuming that Y is the vertical axis. If you enter a positive value, you will see a warning message like below which is simply to alert you of a positive gravity value.

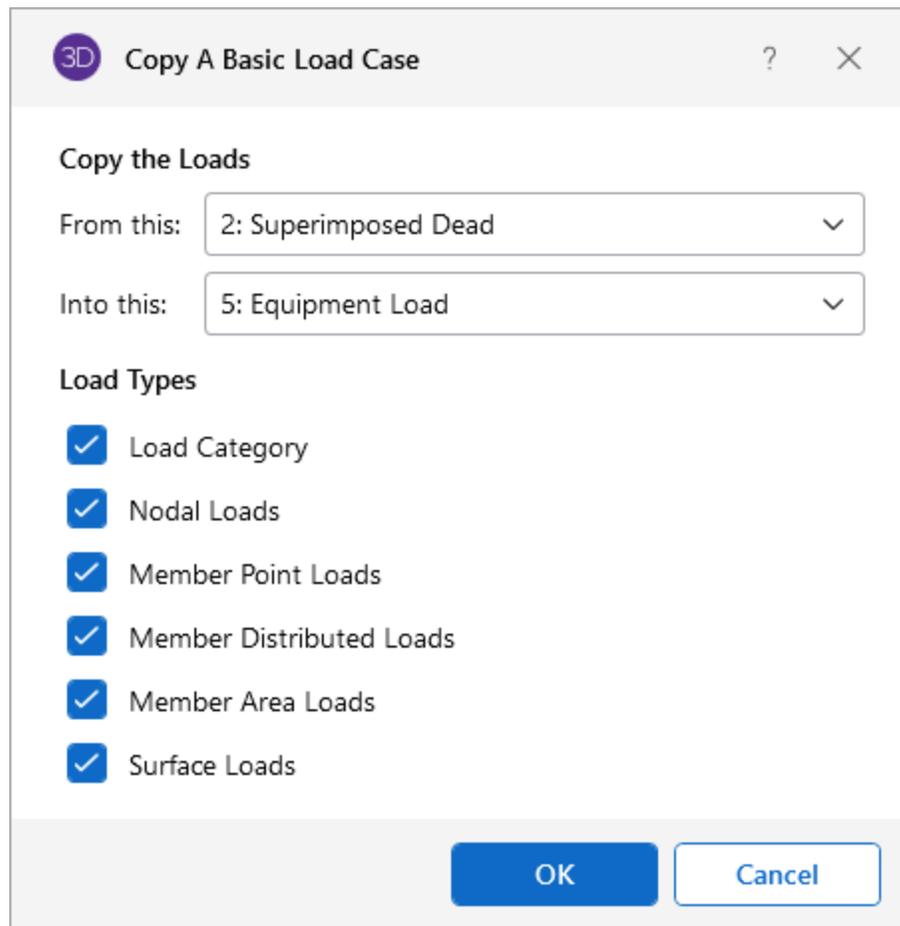


The remaining columns display the quantity of each type of load that is contained in the BLC. You can not edit these values **but You can click on the quantities to open the spreadsheet and view the loads that it represents**. For example, the previous figure has 15 distributed loads as part of BLC 2. Clicking on the number 15 with the mouse will open the Distributed Loads spreadsheet and display these loads.

Copy Basic Load Cases

You can 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.

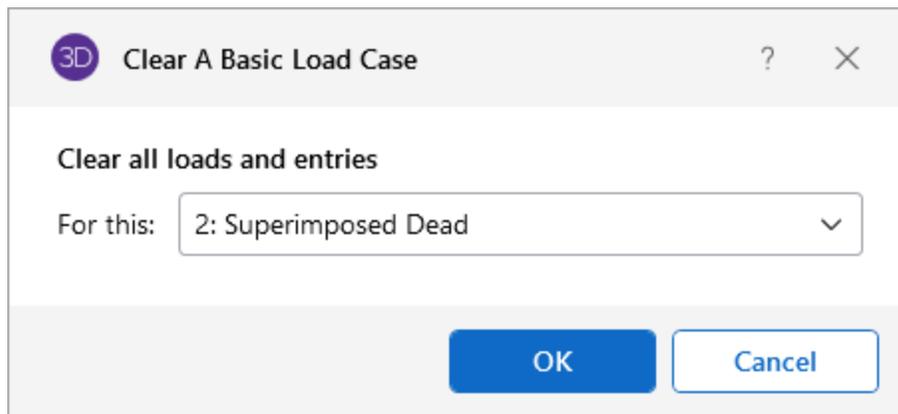
To do this, open the **Basic Load Case** spreadsheet and right click on the load case that you wish to copy and select **Copy Basic Load Case** from the drop-down list. The Copy A Basic Load Case dialogue will open and allows you to specify the load case that you want to copy the loads into, as well as the types of loads that you want to copy. For example, if you check **Nodal Loads** and uncheck all the other load types, only the nodal loads will be copied.



Any loads copied into a BLC will be added to any loads that may already exist in that BLC.

Delete Basic Load Cases

You can automatically clear all the loads in a BLC. To do this, open the **Basic Load Case** spreadsheet and then right click on the load case that you wish to delete and select **Clear Basic Load Case** from the drop-down list.

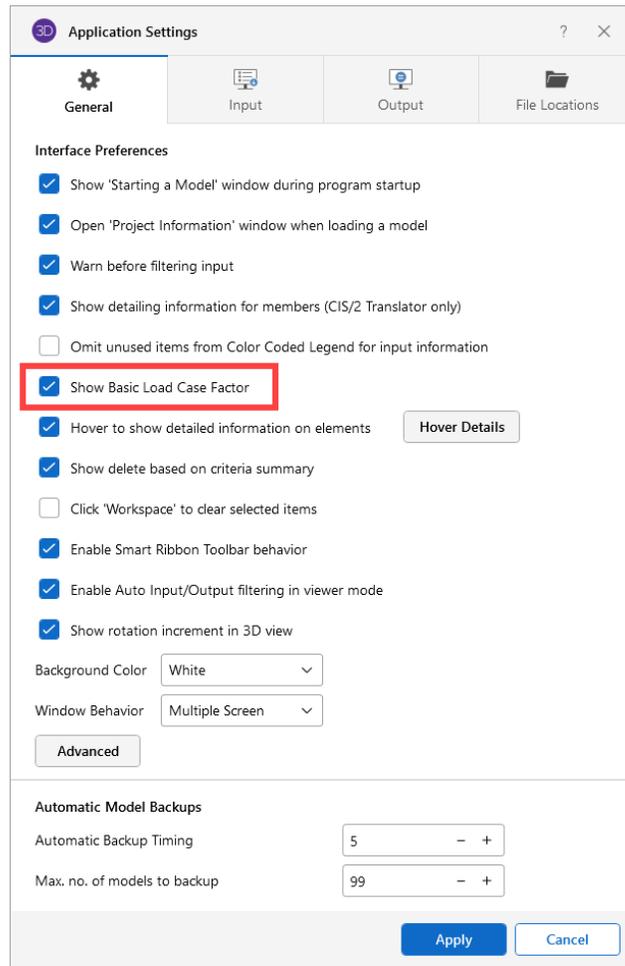


All of the loads in this BLC will be deleted including the description, category information and any self-weight information.

Scale Basic Load Cases

You can apply a scaling factor to an entire **Basic Load Case** in the **Basic Load Case** spreadsheet. These scaling factors give the user the flexibility and freedom to modify and manipulate the loading conditions in the model.

To use this feature, you will need to enable it in the **Application Settings**. By default, the program will have this feature inactivated.



Once enabled, a new column appears in the **Basic Load Case** spreadsheet. **Basic Load Cases** may be scaled by both negative and positive values.

	BLC Description	BLC Factor	Category	X Gravity	Y Gravity	Z Gravity	Nodal	Point	Distributed	Area(Member)	Surface(Plate/Wall)
1	Self Weight		DL		-1						
2	Superimposed Dead		DL						15	2	320
3	Occupancy Live		LL							2	320
4	Roof Live		RLL							3	
5	Equipment Load		DL				4				
6			None								
7			None								
8			None								



Note: When this feature is disabled in the **Application Settings**, the column will be hidden but the values in the column will still be applied to the **Basic Load Cases**. Therefore, it is imperative that values in this column be cleared out before hiding the column if the intent is to not include the factors in the design.

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 can do this on the **Basic Load Cases** spreadsheet. These categories may then be combined to define load combinations used in analysis.

Categories are easy to use and are very helpful. They allow you to place your loads into groups that are commonly used in load combinations, while keeping them in separate load cases. This is especially helpful in large models where the loads might occupy many different load cases. They also allow you to define load combinations that are easily understood because they resemble the combinations as they appear in the building code.

Load Category	Description
DL	Dead Load
LL	Live Load
EL	Earthquake Load
WL	Wind Load
SL	Snow Load
RLL	Roof Live Load
LLS	Live Load Special (public assembly, garage, storage, etc.)
TL	Long Term Load (creep, shrinkage, settlement, thermal, etc.)
SLN	Snow Load Non-shedding

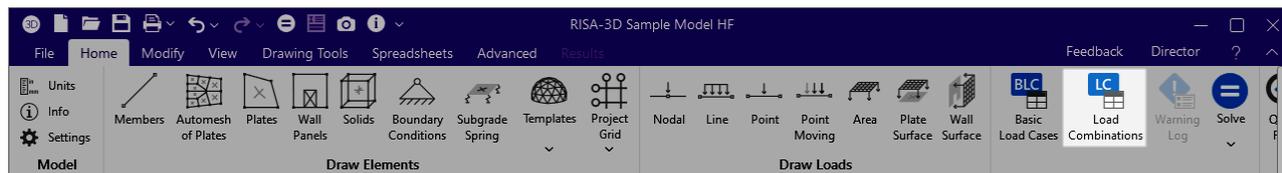
Load Category	Description
HL	Hydrostatic Load
FL	Fluid Pressure Load
RL	Rain Load
PL	Ponding Load
EPL	Earth Pressure Load
IL	Impact Load
OL1 - OL100	Other Load 1 - 100 (generic)
ELX, ELY, ELZ	Earthquake Load along global X-axis, Y-axis, Z-axis
WLX, WLY, WLZ	Wind Load along global X-axis, Y-axis, Z-axis
WL+X, WL+Y, WL+Z	Wind Load along positive global X-axis, Y-axis, Z-axis
WL-X, WL-Y, WL-Z	Wind Load along negative global X-axis, Y-axis, Z-axis
WLXP1, WLYP1, WLZP1	Partial Wind Load 1 along global X-axis, Y-axis, Z-axis
WLXP2, WLYP2, WLZP2	Partial Wind Load 2 along global X-axis, Y-axis, Z-axis
ELX+Z, ELX+Y	Eccentric Earthquake Load along global X-axis shifted along positive global Z-axis, Y-axis

Load Category	Description
ELX-Z, ELX-Y	Eccentric Earthquake Load along global X-axis shifted along negative global Z-axis, Y-axis
ELZ+X, ELZ+Y	Eccentric Earthquake Load along global Z-axis shifted along positive global X-axis, Y-axis
ELZ-X, ELZ-Y	Eccentric Earthquake Load along global Z-axis shifted along negative global X-axis, Y-axis
ELY+X, ELY+Z	Eccentric Earthquake Load along global Y-axis shifted along positive global X-axis, Z-axis
ELY-X, ELY-Z	Eccentric Earthquake Load along global Y-axis shifted along negative global X-axis, Z-axis
NL, NLX, NLY, NLZ	General notional load and along global X-axis, Y-axis, Z-axis
WLX+R, WLY+R, WLZ+R, WLX- R, WLY-R, WLZ-R,	Roof wind loads in the positive and negative X-axis, Y-axis, Z-axis

Using categories is optional. Remember though that if you define combinations of categories you must define these categories in the **Basic Load Case** spreadsheet. If you don't, the combinations will have no loads.

Load Combinations

During solution the model is loaded with a combination of factored Load Categories and/or Basic Load Cases, both of which are defined on the **Basic Load Cases Spreadsheet**. These combinations, load factors, and other parameters are defined on the **Load Combinations Spreadsheet**. Most standard load combinations are included in the program. See [Solution](#) to learn how to solve load combinations.



Add Load Combinations Manually

To add Load Combinations manually:

1. From the 'Home' ribbon, click on the **Load Combinations** icon.
2. Enter load combinations by pairing loads in the **BLC** fields with factors in the **Factor** fields.

Add Auto-Generated Building Code Combinations

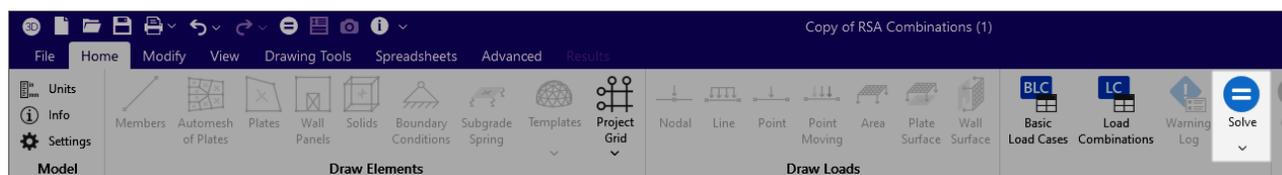
To add auto-generated building code combinations:

1. From the 'Home' ribbon click on the **Load Combinations** icon.
2. Click on the button at the top of the spreadsheet.
3. On the 'Gravity' tab, choose the **Load Combination Region** and **Load Combination Code** from the drop down lists.
4. Choose the desired **Notional Load** options.
5. Choose the desired **Roof Live Load** options.
6. Click the **Generate** button.
7. Click on the **Wind** tab and choose the desired **Wind Load** options.
8. Click the **Generate** button.
9. Click on the **Seismic** tab and choose the desired **Seismic Load** options.
10. Click the **Generate** button.
11. Modify the generated combinations and options in the spreadsheet as necessary.

**Note:**

- The generated building code combinations are made up of Load Categories and Factors. Loads that are not assigned to these categories will not be included in the combinations upon solution.
- All generated combinations are added and marked for the envelope solution. You can remove combinations from the envelope after adding them.
- You can specify P-Delta options and SRSS combinations for each combination after you have added them.
- Verify the Wind/Seismic ASIF (allowable stress increase factor), and the Timber CD settings on the **Design** tab for combinations after you add them.
- You can save any preferred load combinations as the default by clicking the **Save As Defaults**  button on the **Window Toolbar**.

Solve Load Combinations



Solve Single Load Combinations

To solve a single load combination:

1. On the 'Home' ribbon, click the **Solve** icon.
2. Choose the **Single Load Combination** from the drop down list.
3. Click the **Solve** button.

Solve Multiple Load Combinations

To solve multiple load combinations:

1. On the 'Load Combinations' spreadsheet, check the **Solve** checkbox for each combination to be solved.
2. On the 'Home' ribbon, click the **Solve** icon.
3. Choose the **Envelope** or **Batch** option.
4. Click the **Solve** button.



Note:

- The **Envelope** solution is where all combinations with a checkmark in the **Solve** field are solved simultaneously. The **Maximum and Minimum** results of these solutions are listed, along with the number of the controlling load combination for each solution result.
- The **Batch** solution is where all combinations with a checkmark in the **Solve** field are solved simultaneously. **All** results of these solutions are listed.

Load Combinations Spreadsheet

The **Load Combinations Spreadsheet** records the combinations of loads for solution. It can be accessed by choosing **Load Combination** on the from the 'Explorer' panel under 'Data Entry'.

	Description	Solve	P-Delta	SRSS	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	
1	IBC 16-1	<input checked="" type="checkbox"/>			DL	1.4																	
2	IBC 16-2 (a)	<input checked="" type="checkbox"/>			DL	1.2	LL	1.6	RLL	0.5													
3	IBC 16-3 (a)	<input checked="" type="checkbox"/>			DL	1.2	LL	0.5	RLL	1.6													
4	IBC 16-3 (b) (a)	<input checked="" type="checkbox"/>			DL	1.2	RLL	1.6	WLX	0.8													
5	IBC 16-3 (b) (d)	<input checked="" type="checkbox"/>			DL	1.2	RLL	1.6	WLZ	0.8													
6	IBC 16-3 (b) (m)	<input checked="" type="checkbox"/>			DL	1.2	RLL	1.6	WLX	-0.8													
7	IBC 16-3 (b) (e)	<input checked="" type="checkbox"/>			DL	1.2	RLL	1.6	WLX	-0.6	WLZ	-0.6											
8	IBC 16-3 (b) (t)	<input checked="" type="checkbox"/>			DL	1.2	RLL	1.6	WLX	-0.6	WLZ	0.6											
9	IBC 16-3 (b) (v)	<input checked="" type="checkbox"/>			DL	1.2	RLL	1.6	WLXP1	-0.601	WLZP2	-0.601											
10	IBC 16-4 (a) (a)	<input checked="" type="checkbox"/>			DL	1.2	LL	0.5	RLL	0.5	WLX	1.6											
11	IBC 16-4 (a) (d)	<input checked="" type="checkbox"/>			DL	1.2	LL	0.5	RLL	0.5	WLZ	1.6											
12	IBC 16-4 (a) (q)	<input checked="" type="checkbox"/>			DL	1.2	LL	0.5	RLL	0.5	WLX	1.2	WLZ	1.2									

Combinations Tab

The first column, **Description**, is strictly for your reference. Enter any descriptive label you wish and it will be displayed with the results when the combination is solved.

The next three columns are for options that apply to each load combination.

The **P-Delta** entry is used to enable an analysis of member secondary effects. See [P-Delta Analysis](#) for more information.

The **Solve** checkbox is used to indicate which combinations should be included in batch or envelope solutions. See [Solution](#) for more information.



Note: Per Section C1 of Chapter C (Design for Stability) of the AISC 15th Edition steel code, a factor of 1.6 will be applied to all load combinations for which a P-Delta analysis is to be performed (and for which the Hot Rolled box is checked on the Design tab) whenever 'AISC 15th: ASD' is selected on the Codes tab of the Model Settings Dialog. This factor is applied prior to conducting the P-Delta analysis and the results are subsequently divided by a factor of 1.6.

The **SRSS** entry is used to combine response spectra results by the Square Root of Sum of Squares. See [SRSS Combination of RSA Results](#) for more information.

The next eight pairs of columns (**BLC, Factor**) are for defining what loads are to be part of the combination, along with factors for each. Select load categories from the drop down lists in the **Category** columns of the spreadsheet. The following are the entries you can use in the **BLC** field:

Entry	Description
Number	Entering a number includes that particular Basic Load Case, i.e. enter "3" for BLC 3
Category	Enter a category code such as DL, LL, etc to include all loads in that category
Lnn	Enter "Lnn" to nest the loads from another combination, where "nn" is the number of the other combination, i.e. "L3" means include all loads from load combination 3.
Mnn	Use this entry to include a moving load where "Mnn" is the moving load tag from the moving load spreadsheet
Sn	Includes response spectra results for the global direction "n", i.e. enter "SX" to include response spectra results calculated in the global X direction, "SY" for global Y, etc.
Tn	Use this entry to include a time history load where "Tn" is the time history load tag from the Time History Loads spreadsheet

There are also a number of BLC Prefixes that can be included in your load combinations. You can have these generated by the [LC Generator](#) or you can click on the triple dot button in the BLC cell to select them manually. Rho, SDS, and Omega (Ω) are defined in the [Seismic tab](#) of the Model Settings, and SF is accessed from the RSA Scaling Factor button within the Load Combination spreadsheet. They are explained below:

Prefix	Description
SF	Scaling Factor - applied to RSA results
ρ (Rho)	Redundancy factor - applied to EL loads for seismic load combinations (or RSA results) Note: If ELX and ELZ used, the ρ from the X and Z direction will be used directionally. If EL is used, the ρ from the X direction will be used and the Z direction will be disregarded. If ELY is used, the ρ from the Z direction will be used and the X direction will be disregarded.
SDS	Spectral Response Acceleration Parameter for Short Periods - applied to DL for seismic load combinations
Ω (Om)	Overstrength Factor - applied to EL loads for seismic load combinations (or RSA results)

Prefix	Description
	Note: If ELX and ELZ are used, the Ω from the X and Z direction will be used directionally. If EL is used, the Ω from the X direction will be used and the Z direction will be disregarded. If ELY is used, the Ω from the Z direction will be used and the X direction will be disregarded.
CL	Capacity-Limited Forces - applied to EL loads for seismic load combinations for vertical braced frame seismic design (e.g. SCBF, BRBF as VBrace). The program will conduct capacity-limited design for beams and columns using braces' capacity-limited forces in lieu of the applied seismic loads.

In the **Factor** column, enter a multiplier to be applied to the loads being included.

You can also use the  button in the BLC cell to help you specify the loads. Choose from load categories, basic load cases, or spectral results or moving loads from the drop-down lists.

Notice the **Seismic Options** and **RSA Options** that are described in the Prefix table above. For additional advice on this topic, please see the RISA Tips & Tricks website: www.risa.com/post/support. Type in Search keywords: **Seismic Factors**.

Design Tab

	Description	ASIF	CD	Service	Hot Rolled	Cold Formed	Wood	Concrete	Masonry	Aluminum	Stainless	Connection
1	IBC 16-1			<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
2	IBC 16-2 (a)			<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
3	IBC 16-3 (a)			<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
4	IBC 16-3 (...)			<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
5	IBC 16-3 (...)			<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
6	IBC 16-3 (...)			<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
7	IBC 16-3 (...)			<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
8	IBC 16-3 (...)			<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
9	IBC 16-3 (...)			<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
10	IBC 16-4 (...)			<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
11	IBC 16-4 (...)			<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
12	IBC 16-4 (...)			<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>

The first field, the **Description**, is strictly for the user's reference. Enter any descriptive label you wish and it will be displayed with the results when the combination is solved.

The **ASIF** column is used to set the allowable stress increase factor used for both the AISC: ASD code checking and ACI 530 ASD (masonry allowable stress design). See [Allowable Stress Increase](#) regarding its usage for Hot Rolled Steel.

The **CD** factor is the load duration factor and is only necessary for NDS timber design using ASD (see [Load Duration Factor](#)). **Note:** this is **KD** for Canadian (CSA O86) wood design.

The **lambda** factor is the time effect factor and is only necessary for NDS timber design using LRFD (see [Time Effect Factor](#)).

The **Service** check box is used in the following applications:

- **Cracked Concrete Section Properties:** Per the provisions of ACI 318-14 Section 6.6.3.2.2 (ACI 318-11 Section R10.10.4.1) the cracked section properties of beams, columns and walls can be multiplied by a factor of 1.43 for service level loads ($\leq 1.0 \cdot I_g$). All concrete members and walls apply this factor when the Service box is checked. Note that this 1.43 factor does not apply to masonry walls. Masonry wall stiffness is NOT dependent on this check box.
- **Story Drift:** The story drift results are broken up by each lateral direction into Service and Strength load combinations. These tabs can be seen in the Story Drift results spreadsheet.
- **Member Deflections:** The first two tabs of the Member Deflections spreadsheet provide deflection and deflection ratio results for Service and Strength load combinations.

The next seven check boxes designate which load combinations should be used for the code checking of each material type. The example shown uses one set of Load Combinations for the NDS wood design, and another for the ACI concrete design. Member results (Forces, Stresses, Torsion) will only be reported for a members if it's material type has been checked for that load combination. Member results for general material will be shown for all load combinations. For example when using

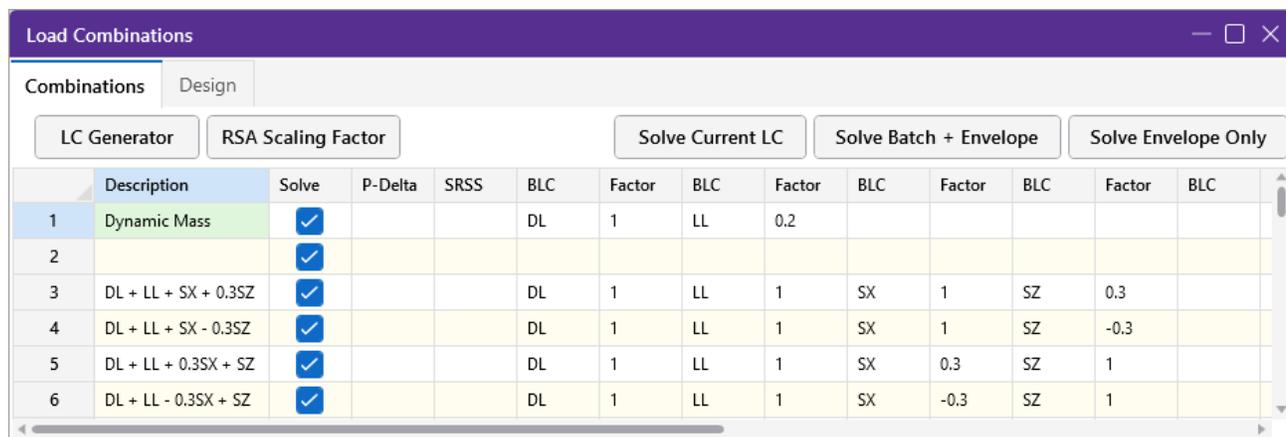
steel and wood in your model, you can design your steel for one set of load combinations and your wood for another.

The last check box for **Connection** defines:

- If this LC should be used in designing hot-rolled steel connections with RISACONNECTION.

Load Combinations with RSA Results

The results from response spectra analyses in the X, Y, and Z direction can also be included in the load combinations. Remember, when you perform a response spectra analysis (RSA), you specify in which global direction the spectrum is applied. RISA-3D will retain the three RSA solutions (one for each direction) simultaneously.



Combinations		Design											
LC Generator		RSA Scaling Factor		Solve Current LC			Solve Batch + Envelope			Solve Envelope Only			
	Description	Solve	P-Delta	SRSS	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC
1	Dynamic Mass	<input checked="" type="checkbox"/>			DL	1	LL	0.2					
2		<input checked="" type="checkbox"/>											
3	DL + LL + SX + 0.3SZ	<input checked="" type="checkbox"/>			DL	1	LL	1	SX	1	SZ	0.3	
4	DL + LL + SX - 0.3SZ	<input checked="" type="checkbox"/>			DL	1	LL	1	SX	1	SZ	-0.3	
5	DL + LL + 0.3SX + SZ	<input checked="" type="checkbox"/>			DL	1	LL	1	SX	0.3	SZ	1	
6	DL + LL - 0.3SX + SZ	<input checked="" type="checkbox"/>			DL	1	LL	1	SX	-0.3	SZ	1	

To include the RSA results for a particular direction in a load combination, enter "**Sn**" in the BLC field, where n is the global direction. Suppose you wanted to include X direction RSA results, you would enter **SX** in the BLC field. You would enter **SY** for Y direction and **SZ** for Z direction RSA results. Also be sure to put the RSA Scaling Factor for the RSA results in the Factor field. You can have more than one RSA entry in a load combination.



Note:

- Remember that RSA results are typically unsigned (all positive) and you should provide some means of accounting for this. The figure above uses two combinations for each RSA result, one with a positive factor and the other with a negative factor to capture the maximum deflections, stresses and forces. See [Unsigned \(All Positive\) Results](#) for more information.
- If you have to combine 2 or 3 different spectra results with many static load combinations, it is convenient to put all the spectra results (SX, SY, and SZ) and factors in one load combination and nest that combination in other combinations. You must set the RSA SRSS flag on each load combination for it to be performed.

SRSS Combination of RSA Results

This is used to cause orthogonal RSA results in the combination to be summed together using an SRSS (Square Root of Sum of Squares) summation. This gives a good approximation of MAXIMUM responses but it also causes all the RSA results to be positive.

You can choose from the options by clicking on the  button. The entry is "+" or "-" to indicate whether the combined RSA results are to be added (+) or subtracted (-) from the other loads in the combination.



Note:

- This flag is used to combine different spectra that are acting in different directions. This is different than the modal combination method that combines results for one direction and is specified in the **Response Spectra** settings.
- This method was only approved for use in the 1997 UBC and is no longer noted as an appropriate method in the current codes.

Load Combinations with Capacity-Limited (CL) Design

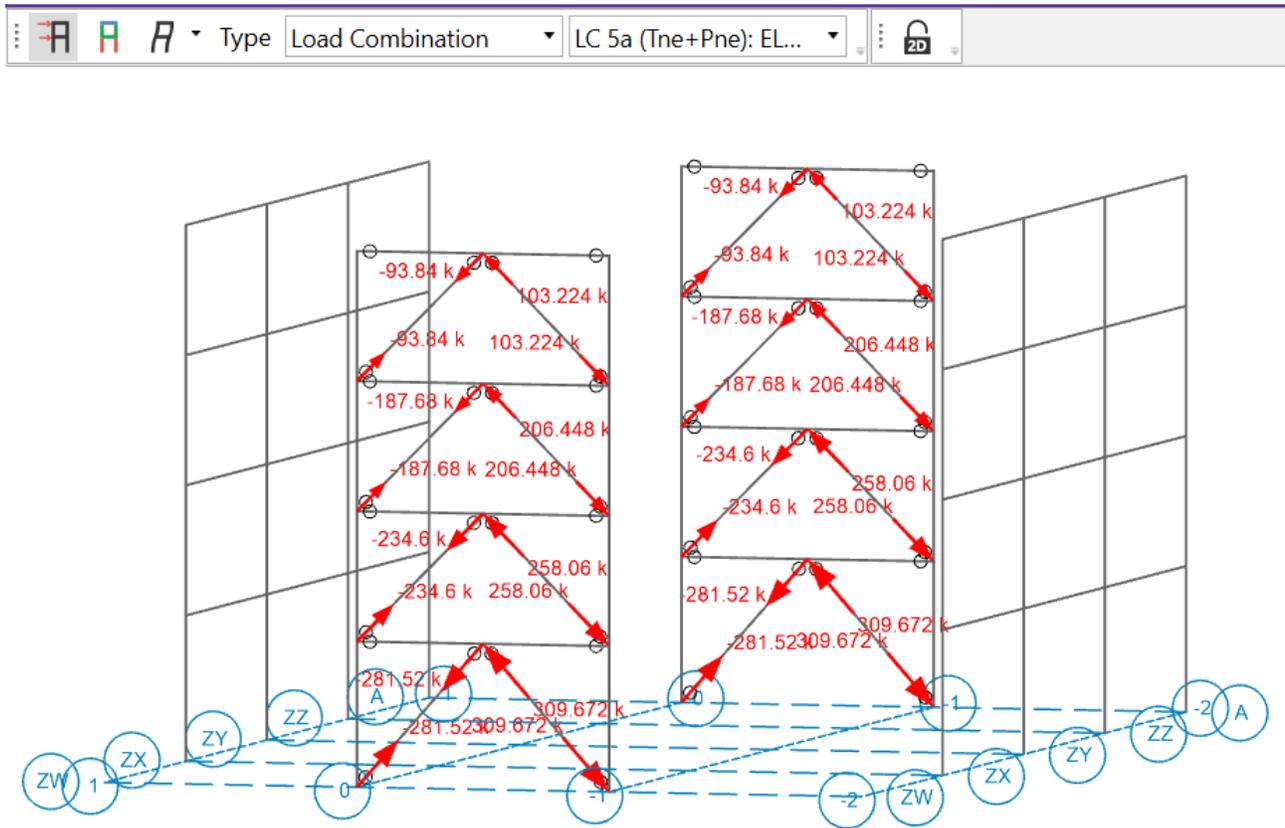
The capacity-limited design forces for braced frames (e.g. SCBF, BRBF as VBrace) can also be included in the load combinations. For capacity-limited design, the expected strengths of braces will be used in lieu of the applied earthquake loads for columns and beams design in vertical braced frames. The columns and beams will be designed to resist the expected strength of the braces to ensure braces yield first to dissipate energy during an earthquake event (e.g. Tne + Pne for BRBF, Tne + Pne or Tne + 0.3Pne for SCBF).

Load Combinations														
Combinations														
LC Generator		RSA Scaling Factor		Solve Current LC			Solve Batch + Envelope			Solve Envelope Only				
	Description	Solve	P-Delta	SRSS	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor
1	IBC 16-5 (os-a)	<input checked="" type="checkbox"/>	Y		DL	1.2	Sps*DL	0.2	ELX-CL	1	LL	0.5	LLS	1
2	IBC 16-5 (os-b)	<input checked="" type="checkbox"/>	Y		DL	1.2	Sps*DL	0.2	ELZ-CL	1	LL	0.5	LLS	1
3	IBC 16-7 (os-a)	<input checked="" type="checkbox"/>	Y		DL	1.2	Sps*DL	-0.2	ELX-CL	1				
4	IBC 16-7 (os-b)	<input checked="" type="checkbox"/>	Y		DL	1.2	Sps*DL	-0.2	ELZ-CL	1				

To include the capacity-limited (CL) design considerations for a particular direction in a load combination, enter "**ELn-CL**" in the BLC field, where n is the global direction. Suppose you wanted to consider brace capacity-limited forces for braced frames in the X direction, you would enter **ELX-CL** in the BLC field.

**Note:**

- Proper seismic design rules (e.g. SCBF, BRBF) are needed for frame members so the program can include capacity-limited (CL) design considerations. No CL considerations will be done for members with incorrect seismic design rules. The SCBF or BRBF braces must be assigned as vertical brace (VBrace) in order to run the CL Load Combinations.
- If both standard lateral loads and capacity-limited (CL) forces are specified in the same load combination, the non-CL lateral loads will be ignored. This is based on our understanding of the code requirement for CL design that the applied lateral loads in CL design shall be replaced by the brace CL forces. The program will not consider both standard lateral loads and CL forces during analysis for the same load combination.
- The capacity-limited (CL) design procedure applies to only column and beam design in braced frames. When an CL LC is solved, the program replaces braces with brace capacity-limited forces in frames that are eligible for CL design (e.g. frames specified with proper seismic design rules). Since the braces are removed during the CL design process, hidden boundary conditions are added in the lateral directions at beam ends in the braced frames to avoid frame instability. The lock of beam ends in lateral directions will not be an issue for CL design because this design process only applies to columns and beams. The brace CL forces will be transferred to adjacent columns and beams properly for their CL design. However, there are limitations due to the added hidden boundary conditions. Currently no beam axial load redistribution is considered among braced beams in the same frame line due to the added boundary conditions. Also no unbalanced beam axial load is considered for frames with braces connected to beam ends directly (e.g. diagonal braced frames). The unbalanced beam axial load is considered for V and Inverted-V braces. The display of the hidden boundary conditions can be turned on by going to the File -> Application Settings -> Output, and selecting the checkbox "Display seismic capacity-limited boundary conditions".
- Users can display capacity-limited (CL) forces after solutions. The program requires solutions to determine and display the CL forces of braces. If users display the forces under CL load combinations before solutions, the CL forces will be displayed as the associated standard seismic forces. For example, ELX-CL will be displayed as ELX loads before solution. After solution it will be displayed as the brace expected strength forces for braced frames in the X direction.



Limitations

- The analysis consideration per AISC 341-2016 F2.3c of progressive yielding and buckling of the braces for multi-tiered braced frames is not included in the current implementation of capacity-limited (CL) design.
- The exceptions under AISC 341-2010 F2.3 are not fully considered in program. This is due to the program not having enough information to determine if these exceptions apply. Users need to use their judgment to check if these exceptions apply to their structure. The only exception the program partially considers is the AISC 341-2010 F2.3 Exception 2a. Per this exception the program will perform SCBF column design using the lesser of the forces determined from load combinations using the amplified seismic loads or the CL loads. However, the program cannot automatically remove compression braces in the model. The users need to modify the models accordingly if they need to consider this exception.
- Unbalanced forces for beams are only considered for V and Inverted-V braced frames. This is due to the added hidden lateral boundary conditions for frame instability during capacity-limited (CL) design. No unbalanced beam axial load can be considered for frames with braces connected to beam ends directly (e.g. diagonal braced frames) because the

unbalanced axial loads will go into the boundary conditions directly. Also, for the same reason the program currently does not consider any redistribution of beam unbalanced axial loads among beams in the same frame line.

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.

	Description	Solve	P-Delta	SRSS	BLC	Factor	BLC								
1	DL	<input checked="" type="checkbox"/>			DL	1									
2	DL + LL	<input checked="" type="checkbox"/>			DL	1									
3		<input checked="" type="checkbox"/>													
4	DL + LL + M1	<input checked="" type="checkbox"/>			DL	1	LL	1	M1	1					
5	DL + LL + M2	<input checked="" type="checkbox"/>			DL	1.4	LL	1	M2	1					
6	DL + LL + M3	<input checked="" type="checkbox"/>			DL	1.4	LL	1	M3	1					

Nesting Load Combinations

You are allowed only 10 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.

	Description	Solve	P-Delta	SRSS	BLC	Factor	BLC								
1	Spectral Results	<input checked="" type="checkbox"/>			SX	0.13	SZ	0.25							
2	DL	<input checked="" type="checkbox"/>			DL	1									
3	DL + LL	<input checked="" type="checkbox"/>			DL	1	LL	1							
4	DL + LL + EL	<input checked="" type="checkbox"/>			DL	1	LL	1	L1	1					

Entering "Lnn" in the BLC field means include all the BLC entries (with their factors) from Load Combination "nn". For example, say Load Combination 4 has "L1" entered for one of its BLC's. This specifies to include all the BLC's (with their factors) entered in Load Combination 1 as part of Load Combination 4 (this includes self-weight and RSA entries as well). The flags for Load Combination 1 (i.e. Solve, PDelta, and SRSS entries) apply only to Load Combination 1 and will not be used when Load Combination 4 is solved.

Also, the factor we enter with the "Lnn" entry will be applied to the BLC factors entered for LC nn. Thus, if we enter "L1" with a factor of "0.9", we're including 90% of the BLC entries of Load Combination 1.



Note: These “combinations of load combinations” can only be nested to one level; i.e. the load combs referenced with the Lnn entries may not themselves have Lnn entries.

ASIF Factors for Wind / Seismic Load Combinations

Entering a value/factor in the field labeled **ASIF** (Allowable Stress Increase Factor) indicates that the load combination should be treated as a wind/seismic load combination.

	Description	ASIF	CD	Service	Hot Rolled	Cold Formed	Wood	Concrete	Masonry	Aluminum	Stainless	Connection
1	ASCE ASD 1		0.9	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
2	ASCE ASD 2			<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
3	ASCE ASD 3 (a)		1.25	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
4	ASCE ASD 3 (b)		1.15	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
5	ASCE ASD 3 (c)		1.15	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
6	ASCE ASD 4 (a)		1.25	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
7	ASCE ASD 4 (b)		1.15	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
8	ASCE ASD 4 (c)		1.15	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
9	ASCE ASD 5 (a)		1.6	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
10	ASCE ASD 6 (a)		1.6	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
11	ASCE ASD 6 (b)		1.6	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
12	ASCE ASD 6 (c)		1.6	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
13	ASCE ASD 7		1.6	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
14	ASCE ASD 8	1.333	1.6	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>

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

If LRFD code checking is selected, then any value of ASIF greater than 1.0 will indicate that the seismic provisions for the WF compactness checks are to be used (Table 8-1, p. 6-317 of the 2nd ed. LRFD).

P-Delta Load Combinations

The **P Delta** field is used to perform a P-Delta calculation for that load combination. You can 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.

Load Combinations															
Combinations Design															
LC Generator		RSA Scaling Factor		Solve Current LC				Solve Batch + Envelope				Solve Envelope Only			
	Description	Solve	P-Delta	SRSS	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC
1	ASCE ASD 1	<input checked="" type="checkbox"/>	Y		DL	1	LL								
2	ASCE ASD 2	<input checked="" type="checkbox"/>	Y		DL	1	LL	1	LLS	1					
3	ASCE ASD 4 (a)	<input checked="" type="checkbox"/>	Y		DL	1	LL	0.75	LLS	0.75	RLL	0.75			
4	ASCE ASD 4 (b)	<input checked="" type="checkbox"/>	Y		DL	1	LL	0.75	LLS	0.75	SL	0.75	SLN	0.75	
5	ASCE ASD 5 (a)	<input checked="" type="checkbox"/>	Y		DL	1	WL	0.6							
6	ASCE ASD 7	<input checked="" type="checkbox"/>	Y		DL	0.6	WL	0.6							
7	ASCE ASD 8	<input checked="" type="checkbox"/>	Y		DL	1	EL	0.7							
8	ASCE ASD 10	<input checked="" type="checkbox"/>	Y		DL	0.6	EL	0.7							

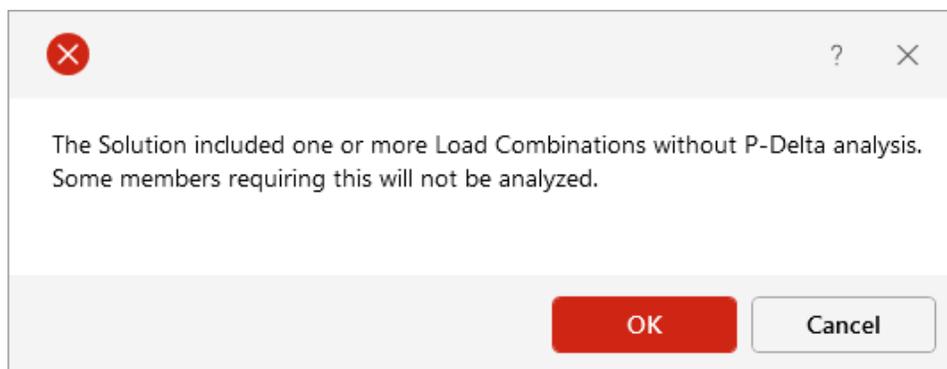
You can also perform a compression only P-Delta analysis. Invoke this option by putting a **C** in the **P Delta** field. See [P-Delta Analysis](#) for more information.



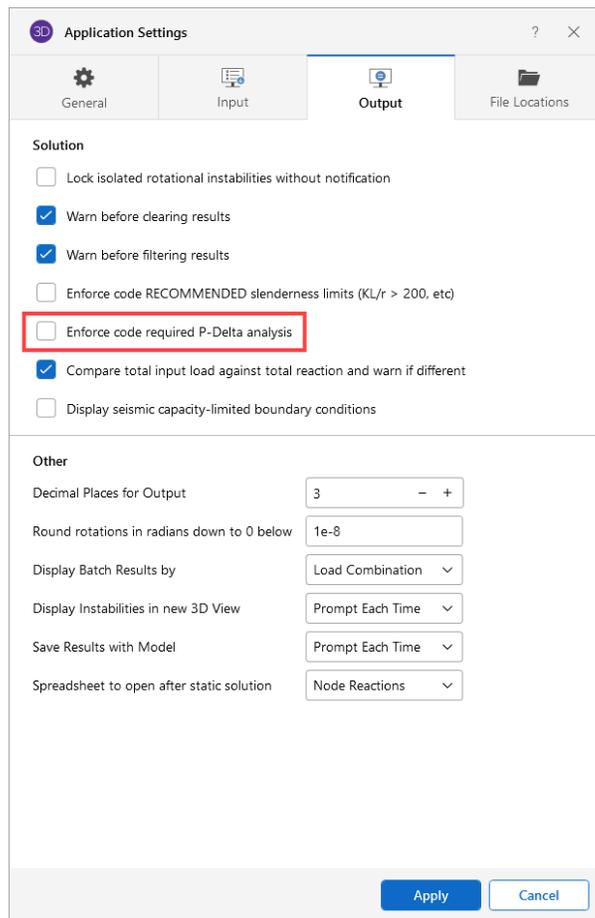
Note:

- P-Delta analysis is normally required for LRFD based code checks. If this is not desired, you can change the setting by going to the File menu and choosing Application Settings and clicking on the Output tab.
- P-Delta analysis is not performed on plates.

If P-Delta was not included in the load combination spreadsheet (PDelta column is blank) but is required by code, the following message will display upon solution.



This warning means that the solution was run but the members requiring P-Delta will not provide any design results. In order to bypass this warning to see results without P-Delta, you can turn off the "Enforce code required P-Delta analysis" option in the [Application Settings](#). An example of when this would be a recommended work flow is when investigating model instabilities. Once you are ready to view full design results for your model, it is recommended to turn this requirement back on.



Timber Design Load Duration Factor

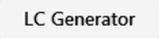
For Wood design, the load duration factor (CD) is entered in the **CD** field on the row that the particular CD factor applies to. Different load combinations would have different CD factors. For example, per the NDS code, a load combination that had only dead load, would have a CD factor of "0.9", while another combination that was comprised of dead load plus wind load would have a CD factor of "1.6".

The CD factor will only be applied to wood code checks on wood members. "Wood" members are those members whose material properties are defined on the **Wood** tab of the **Material Properties Spreadsheet**.

**Note:**

- See Table 2.3.2 in the NDS 2018 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.
- When the Canadian CSA O86 Wood design code is selected, these entries will all change to "KD". See Table 5.3.2.2 of the CSA O86-14 code for direction on what factors to enter.

Generate Building Code Combinations

Major portions of the load combinations that are specified by building codes are included and can be applied to the model for solution. These combinations can be inserted by selecting the  button from the top of the **Load Combinations** spreadsheet. This opens the 'Load Combination Generator' window, shown in the following image.

Gravity Options

The first tab in the Load Combination Generator window is **Gravity**. This tab contains **Roof Live Load** and **Notional Load** Options.

The **LC Region** refers to the various regions supported by the program (U.S., Canada, India, British, et cetera).

The **LC Code** refers to the actual code used to build the load combinations. For the United States, there are a number of different codes that could be used to build load combinations. If the only option is *Sample*, that means that no load combinations have been input for that region. See [Customizing the Load Combination Generator](#) for more information on how to add or edit combinations for that region.

These settings, changed on any of the three tabs, automatically update on the remaining tabs. Thereby making it easy to simply move through the tabs, selecting the options you require, without having to re-select the region and code data on each.

3D Load Combination Generator ? X

Gravity Wind Seismic

Load Combination Region v

Load Combination Code v

Generate Beam Deflection Load Combinations

Notional Load Options v

Reversible

Include Roof Live Load Options

Roof Live Load Snow Load Rain Load

Set as Default

Beam Deflection Options

The Generate Deflection LCs will automatically place 3 load combinations first that match DL, LL and DL + LL.

Load Combinations														
Combinations		Design												
LC Generator		RSA Scaling Factor				Solve Current LC			Solve Batch + Envelope			Solve Envelope Only		
	Description	Solve	P-Delta	SRSS	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor
1	Deflection 1	<input checked="" type="checkbox"/>	Y		DL	1								
2	Deflection 2	<input checked="" type="checkbox"/>	Y		LL	1								
3	Deflection 3	<input checked="" type="checkbox"/>	Y		DL	1	LL	1						
4	IBC 16-1	<input checked="" type="checkbox"/>	Y		DL	1.4								
5	IBC 16-2 (a)	<input checked="" type="checkbox"/>	Y		DL	1.2	LL	1.6	RLL	0.5				
6	IBC 16-3 (a)	<input checked="" type="checkbox"/>	Y		DL	1.2	LL	0.5	RLL	1.6				
7	IBC 16-3 (b) (a)	<input checked="" type="checkbox"/>	Y		DL	1.2	RLL	1.6	WLX	0.8				
8	IBC 16-3 (b) (d)	<input checked="" type="checkbox"/>	Y		DL	1.2	RLL	1.6	WLZ	0.8				
9	IBC 16-3 (b) (m)	<input checked="" type="checkbox"/>	Y		DL	1.2	RLL	1.6	WLX	-0.8				

This will match the Typical design rule from the default [Member Design Rules - Deflection](#) tab inputs.

Deflection Design							
Size/UC	Concrete Rebar	Deflection (Beams)					
	Label	LC	Ratio	LC	Ratio	LC	Ratio
1	Typical <input type="button" value="..."/>	None	N/A	None	N/A	None	N/A

Notional Load Options

The **Notional Load Options** allow the user to specify how complex the load combinations including notional loads should be.

When **None** is selected, the program will not generate any Load Combinations that include the NL load category.

The **2D Only** option is used to indicate that only the most basic notional load category (NL) will be used.

The **X and Z** option is used to indicate that the program should generate separate notional load combinations for each horizontal direction (NLX and NLZ).

When the **Reversible** box is checked, the program will generate every notional load combination twice. Once with a positive load factor, once with a negative load factor.

For additional advice on this topic, please see the RISA Tips & Tricks website: www.risa.com/post/support. Type in Search keywords: **Notional Loads**.

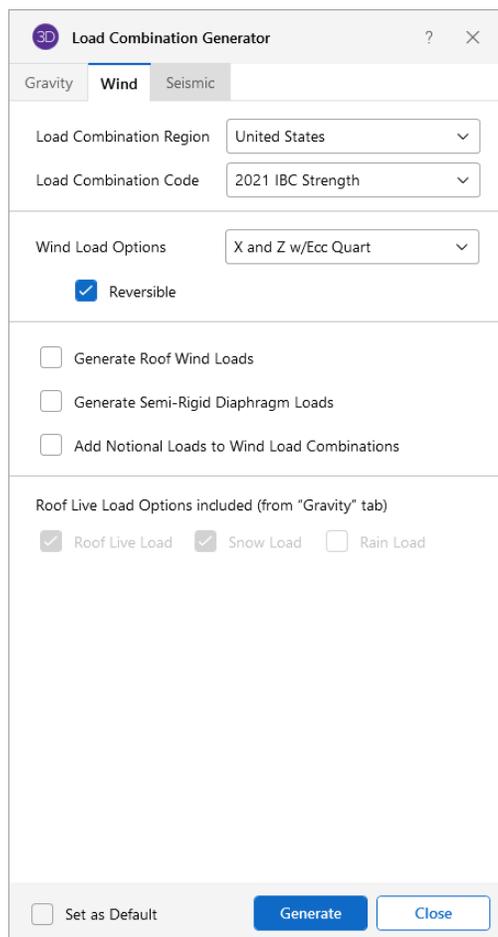
Roof Live Load Options

The **Roof Live Load Options** lets you specify how complex the load combinations including roof live loads should be.

Check the checkbox next to **RLL (Roof Live Load)**, **SL (Snow Load)**, and/or **RL (Rain Load)** to indicate that the selected load categories are to be included in the generated load combinations.

Wind Options

The second tab of the Load Combination Generator is the **Wind** tab. This tab contains **Wind Load Options**.



The screenshot shows the 'Load Combination Generator' dialog box with the 'Wind' tab selected. The dialog has three tabs: 'Gravity', 'Wind', and 'Seismic'. The 'Wind' tab is active. The settings are as follows:

- Load Combination Region: United States (dropdown)
- Load Combination Code: 2021 IBC Strength (dropdown)
- Wind Load Options: X and Z w/Ecc Quart (dropdown)
- Reversible:
- Generate Roof Wind Loads:
- Generate Semi-Rigid Diaphragm Loads:
- Add Notional Loads to Wind Load Combinations:
- Roof Live Load Options included (from "Gravity" tab):
 - Roof Live Load:
 - Snow Load:
 - Rain Load:
- Set as Default:
- Buttons: Generate, Close

Wind Load Options

The **Wind Load Options** specify how detailed the generated wind load combinations should be.

When **None** is selected as the wind load option, the program will not generate any Load Combinations that include wind load categories.

The **2D Only** option generates only the most basic wind load category (WL).

The **X and Z** option generates separate wind load combinations for each horizontal direction (WLX and WLZ when Y is set at the vertical axis).

The **X and Z w/ Ecc** option generates all possible wind load combinations that include partial / eccentric wind loading (WLX, WLXP1, WLXP2, et cetera) per Case 2 from Figure 27.3-8 in the ASCE 7-16.

The **X and Z w/ Ecc, Quart** option generates all possible wind load combinations per Cases 2, 3 and 4 from Figure 27.3-8 in the ASCE 7-16. For additional advice on this topic, please see the RISA Tips & Tricks website: www.risa.com/post/support. Type in Search keyword: **Quartering**.

The **Reversible** option generates two combinations for each wind load, one with positive load factors and one with negative load factors.

The **Generate Semi-Rigid Diaphragm Loads** option generates wind loads for BLC's using windward and leeward direction loads prescribed by the ASCE 7-16 (WLX+Z, WLX-Z, WLZ+X, WLZ-X).

Seismic Options

The third tab of the Load Combination Generator is the **Seismic** tab. This tab contains **Seismic Load Options**.

3D
Load Combination Generator
?
×

Gravity

Wind

Seismic

Load Combination Region ▼
United States

Load Combination Code ▼
2021 IBC Strength

Seismic Load Options ▼
X and Z w/Ecc

Reversible Include p Include Ev (vertical)

Include Non Ortho (100%+30%)

Add Notional Loads to Seismic Load Combinations

Overstrength LC Options ▼
X and Z

Reversible Include Ev (vertical)

Include Non Ortho (100%+30%)

Capacity-Limited Options for Braced Frames ▼
X and Z

Reversible Include Ev (vertical)

Include Non Ortho (100%+30%)

Include Non Ortho (100%+100%)

Roof Live Load Options included (from "Gravity" tab)

Roof Live Load Snow Load Rain Load

Set as Default

Generate

Close

Seismic Load Options

The **SeismicLoad Options** specify how complex the generated seismic load combinations should be.

When **None** is selected as the seismic load option, the program will not generate any Load Combinations that include the EL load category.

The **2D Only** option is used to indicate that only the most basic seismic load category (EL) will be used.

The **X and Z** option is used to indicate that the program should generate separate seismic load combinations for each horizontal direction (ELX and ELZ).

The **X and Z w/ Eccentric** option is used to indicate that the program should generate all possible seismic load combinations that include eccentricities (ELX, ELX+Z, ELX-Z, et cetera).

The **X and Z RSA** option will use the spectral seismic loads (SX, SY, SZ) instead of the regular earthquake loading (ELX, ELY, ELZ).

If you have selected the X and Z, X and Z w/ Eccentric, or X and Z RSA option, you will also have the ability to include non-orthogonal seismic loads. Simply check the **Include Non Ortho (100% + 30%)** checkbox to include these loads per the Orthogonal Combination Procedure of section 12.5.3.1(a) of the ASCE 7-16.

When the **Reversible** box is checked, the program will generate every seismic load combination twice. Once with a positive load factor, once with a negative load factor.

Check the **Include p** checkbox if you want to include the redundancy factor in your seismic load combinations. This factor is set in the Seismic tab of [Model Settings](#).

Check the **Include Ev (vertical)** checkbox if you want to include the vertical seismic load effect in your seismic load combinations. This value is calculated per equation 12.14-6 of the ASCE 7-16 using the S_{DS} value set on the Seismic tab of [Model Settings](#).

The **Capacity-Limited (CL) LC Options for Braced Frames** will generate capacity-limited design forces based on brace expected strength. The generator will use the overstrength load combinations per LC code selected, and replace the overstrength seismic forces with CL forces. The **Include Non Ortho (100%+100%) checkbox** is provided as an additional option for Capacity-Limited Braced Frames in order to meet the requirements for columns that are common to intersecting frames per AISC 341-16 D1.4a.

Check the **Add Notional Loads to Seismic Load Combinations** checkbox if you want to include notional loads applied as lateral seismic loads per the ASCE 7 and AISC codes.

The **Save as Defaults** button can be used to establish the current load combination generator settings as the defaults for future use. Clicking the **Generate** button will generate load combinations in the **Load Combinations Spreadsheet** based on the selected options.

Load Combination Generator - General Notes

The following loads are not generally included in the standard combinations but can be added by editing the combinations in the spreadsheet or by modifying the source document itself. For more information, see [Customizing the Load Combination Generator](#) below:

Category Code	Description
SX, SY, SZ	Response Spectra Results
TL	Long Term Load
HL	Hydrostatic Load
FL	Fluid Pressure Load
PL	Ponding Load Category
EPL	Earth Pressure Load
OL#	Other Load Categories



Note:

- The standard combinations are made up of Load Categories and Factors. Loads that are not assigned to these categories will not be included in the combinations upon solution.
- Some load categories do not occur in all of the design codes. Loads placed in categories that are not part of the standard combinations will not be included in the solution of these combinations.
- All combinations added from the drop down list are added to the envelope solution. You can remove combinations from the envelope after adding them.
- Verify the Wind/Seismic ASIF settings for combinations after you add them.
- After you add the combinations, verify the settings for the wood design Load Duration factor, CD/KD.
- You can specify P-Delta options and SRSS combinations for each combination after you have added them.
- See Editing Spreadsheets - [Inserting, Deleting and Clearing Cells](#) for quick ways to edit or add combinations.
- When a new installation is performed the program will back-up any existing XML files that contain load combination data rather than overwriting them. The back up copies will be given the extension BAK.

Customize the Load Combination Generator

The Load Combinations for each region are contained in an XML file which can be opened and edited using a standard spreadsheet program. An example of one of these XML files is shown below:

Label	Solve	pDelta	SRSS	ASIF	CD	ABIF	Service	Hot Rolled	Cold Formed	Wood	Concrete	Masonry	Footings	Aluminum	Stainless	BLC	Factor	BLC	Factor	BLC	Factor
IBC 16-1	1	Y						1	1		1	1	1	1	1	DL	1.4				
IBC 16-2 (a)	1	Y						1	1		1	1	1	1	1	DL	1.2	LL	1.6	LLS	1.6
IBC 16-2 (b)	1	Y						1	1		1	1	1	1	1	DL	1.2	LL	1.6	LLS	1.6
IBC 16-2 (c)	1	Y						1	1		1	1	1	1	1	DL	1.2	LL	1.6	LLS	1.6
IBC 16-3 (a)	1	Y						1	1		1	1	1	1	1	DL	1.2	RLL	1.6	LL	0.5
IBC 16-3 (b)	1	Y						1	1		1	1	1	1	1	DL	1.2	RLL	1.6	WL	0.5
IBC 16-3 (c)	1	Y						1	1		1	1	1	1	1	DL	1.2	SL	1.6	SLN	1.6
IBC 16-3 (d)	1	Y						1	1		1	1	1	1	1	DL	1.2	SL	1.6	SLN	1.6
IBC 16-3 (e)	1	Y						1	1		1	1	1	1	1	DL	1.2	RL	1.6	LL	0.5

The name of the file itself becomes the name of the Load Combination Region that appears in the Load Combination Generator. Each XML file has a series of "worksheets" and the name of each worksheet will be the name of the Load Combination Code that appears in the Load Combination Generator. You can add, edit or modify these documents to completely customize the available load combinations.



Note: When the program is updated for new versions, the existing XML files will be "backed up" and saved with a '.bak' extension while the new XML files in the update will replace them. If any customizations were made to the XML files, they can be retrieved from the back ups.

The first row of each sheet is reserved for the column headers. The recognized column headers are as follows: Label, Solve, pDelta, SRSS, ASIF, CD, ABIF, Service, Hot Rolled, Cold Formed, Wood, Concrete, Masonry, Footings, Aluminum, and Stainless,. In addition, there will be multiple pairs of BLC & Factor headers.

Other than Label and the BLC / Factor pairs, you can omit columns. If a column is omitted, the default values will be used for those entries. The order of the column labels is optional except for pairs of BLC and Factor. For the program to correspond the Factor with correct BLC, the user should always provide BLC label PRIOR to the corresponding Factor. You can insert blank columns or columns with other labels than described above. In those cases, the program omits those undefined columns.



Note:

- As shown in the above example, the wind and seismic loads should be entered as WL and EL in order for them to be "expanded" using the Wind Load Options and Seismic Load Options.
- The program will read these files from the directory specified from clicking the 3D Button ► Application Settings ► File Locations.
- SX, SY, and SZ are not valid BLC that can be used within the Load Combination Generator spreadsheet. A response spectra analysis must be done first within RISA-3D.

Loads - Nodal Load / Displacement

You can specify nodal loads, and enforced nodal displacements and nodal mass in any of the global degrees of freedom. Loads and displacements can be applied in any non-global direction by defining components of the load in the global directions. This can be accomplished graphically or in the spreadsheets. See Drawing Nodal Loads below to learn how to apply nodal loads/displacements/masses graphically.

Draw Nodal Loads

You can apply nodal loads to nodes. You must enter the load direction, magnitude and type. Make sure that you are careful to enter the correct BLC number that you want the loads assigned to. See Nodal Load/Displacement above for more information on nodal loads.

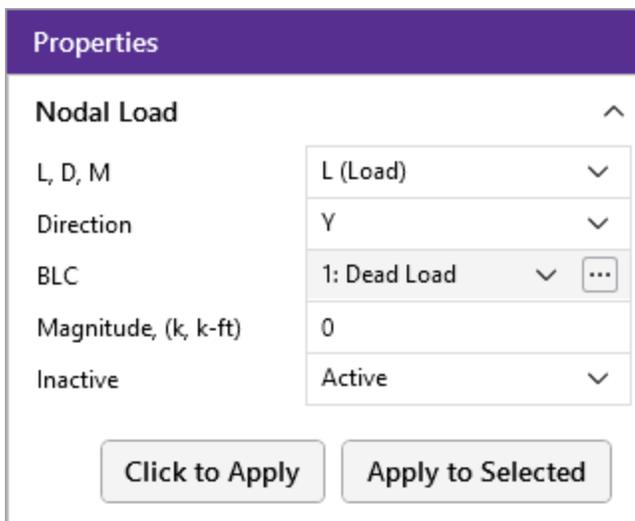
Apply Nodal Loads, Mass and Enforced Displacements

To apply nodal loads, mass and enforced displacements:

1. Go to the **Home** ribbon.



2. Click on the **Nodal** icon to display the nodal load information in the 'Properties Panel'.



3. Define the Nodal Load information:
 - a. Click the **L, D, M** arrow and choose the load type from the menu.
 - b. Click the **Direction** arrow and choose the direction.

- c. Click the **BLC** arrow and choose the Basic Load Case.
- d. Type in a **Magnitude, (k, k-ft)**, if applicable.
- e. Click the **Inactive** arrow and choose whether the Nodal Load should be 'Active' or 'Inactive'.

For help on an item, click the **Help** icon in the upper right corner of the program (as shown in the following image).

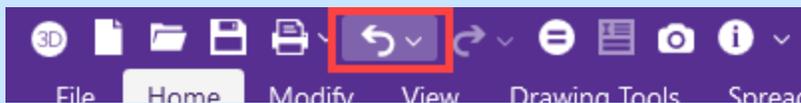


4. You can apply the load by choosing nodes on the fly or apply it to a selection of nodes.
 - To choose nodes on the fly, click the **Click to Apply** button, and click on or box the nodes with the left mouse button.
 - To apply the load to a selection, click the **Apply to Selected** button.
 - If no nodes are selected, **Apply to Selected** will assume the full model is selected and apply changes to all nodes. If any node is selected, **Apply to Selected** will only apply to the selected nodes.



Note:

- You can also specify or edit nodal loads/displacements in the **Nodal Loads Spreadsheet**.
- You can undo any mistakes by clicking the **Undo** icon in the Quick Access toolbar.



Nodal Load Spreadsheet

The **Nodal Load** spreadsheet records the loads for the nodes, It can be accessed by choosing **Nodal Loads** from the 'Data Entry' section of the 'Explorer' panel.

Nodal Loads and Enforced Displacements					
BLC 5: Equipment Load		Prev BLC		Next BLC	
	Node Label	L, D, M	Direction	Magnitude [(k, k-ft), (in, rad), (k*s ² /ft, k*s ² *ft)]	Inactive
1	N588	L (Load)	Y	-100	
2	N589	L (Load)	Y	-100	
3	N596	L (Load)	Y	-100	
4	N597	L (Load)	Y	-100	

When you open this spreadsheet You can view only one basic load case at a time. Use the drop down list on the toolbar to specify a different load case.

The Node Label specifies the node that receives the load or displacement. The same node can be listed any number of times.

The next column indicates the value is a load or an enforced displacement. Enter "L" if it's a load, "D" if it's a displacement and "M" if it is a mass.

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

The Magnitude column holds the value of the load, displacement or mass. The appropriate units for the magnitudes are displayed at the top of the column. Which units apply depends upon whether the value is a load, displacement or mass, and whether the direction is translational or rotational.

The last column allows you to set the specified load as an "Active" load that will be considered for the analysis, or an "Inactive" load that will not be included in the analysis.



Note: If you have a "Reaction" or a "Spring" boundary condition for the same degree of freedom that you have an enforced displacement assigned, NO reaction will be calculated. See [Reactions at Nodes with Enforced Displacements](#) to learn how work around this limitation.

Nodal Mass

For more sophisticated dynamics modeling, you can enter your mass directly as a mass rather than have the program convert it from a load. Using nodal masses offers several advantages such as being able to define directional mass and also the ability to specify mass moment of inertia's to account for rotational inertial effects.

The units used for Nodal Mass are derived from the current **Force** and **Length** units as specified on the **Units** settings. For example, if the current force units are Kips and the current length units are

Feet, you will need to specify your mass as kips / g and mass moments of inertia as kip-ft² / g where g is the acceleration of gravity given in those units (feet per seconds squared).

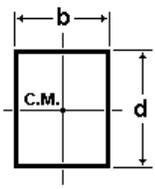
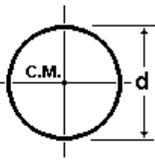
When specifying a nodal mass on the **Nodal Loads** spreadsheet, enter an “M” for the load type. The directions are defined relative to the global axes. Enter translational mass using the global **X**, **Y**, or **Z** codes and mass moments of inertia by specifying the global **MX**, **MY** or **MZ**.

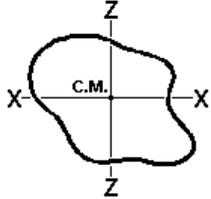
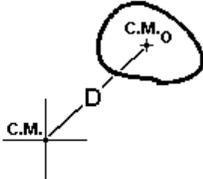
Nodal masses only allow dynamic response in the direction that they’ve been applied. This can be a very effective way to prevent local modes. A good example is a floor diaphragm modeled with plate/shell finite elements. If the mass is only specified for the two lateral directions, you will prevent any unwanted vertical modes. Care must be taken in limiting dynamic response using directional mass for complicated structures. A structure that has “coupled modes” will not give the “real” dynamic response when mass is only specified in one or two directions. A coupled mode is a mode that has mass participate in two or three directions at one time.

Nodal masses also allow you to account for rotational inertia effects by specifying a mass moment of inertia. These are particularly important when you’re using a rigid diaphragm and you’ve also lumped all your mass at one point (typically the center of mass). The rotational inertia effects contribute to the torsion on the diaphragm and should not be neglected. The following table shows some typical diaphragm shapes and the formulas to calculate their mass moment of inertias. Note that you can use the axis transformation equation to calculate the mass moment of inertia for diaphragms that are combinations of these basic shapes. For very irregular diaphragms, a more general equation is given based on the in-plane moment of inertia and the area of the diaphragm.

Mass Moment of Inertia About an Axis Through the Center of Mass

In the table below C.M. is the center of mass point. M is the total Mass of the area (typically including self weight, dead load, and a percentage of the live load) and is assumed to be uniformly distributed throughout. I_{xx} is the moment of inertia about the X-X axis. I_{zz} is the moment of inertia about the Z-Z axis. A is the area. MM_{lo} is the mass moment of inertia about some other point.

Area Plan View	Formula
	$M (b^2 + d^2) / 12$
	$M d^2 / 8$

Area Plan View	Formula
	$M (I_{xx} + I_{zz}) / A$
	$MMI_o + M D^2$

Loads - Area Loads

Area loads are loads that are applied to a planar area and automatically attributed to the members in that plane. This gives you the ability to model the loading effects on a membrane, such as live load on a deck system or wind on a curtain wall, without adding unwanted stiffness to the model. (To model load **and** stiffness You can use plate shell elements loaded with surface loads.)

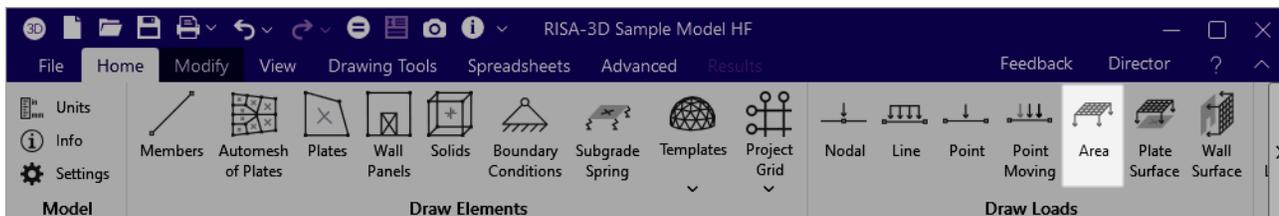
The area loads are defined by polygons of three or four sides. The area loads can be uniform or tapered. They may be assigned a direction for one-way span situations, considered as two-way membranes that transfer load in all directions, or be assigned with an "Open Structure" load distribution which will apply loads to members based solely on their projected surface area. The loads from the area are assigned to the nearest members, not to plates.



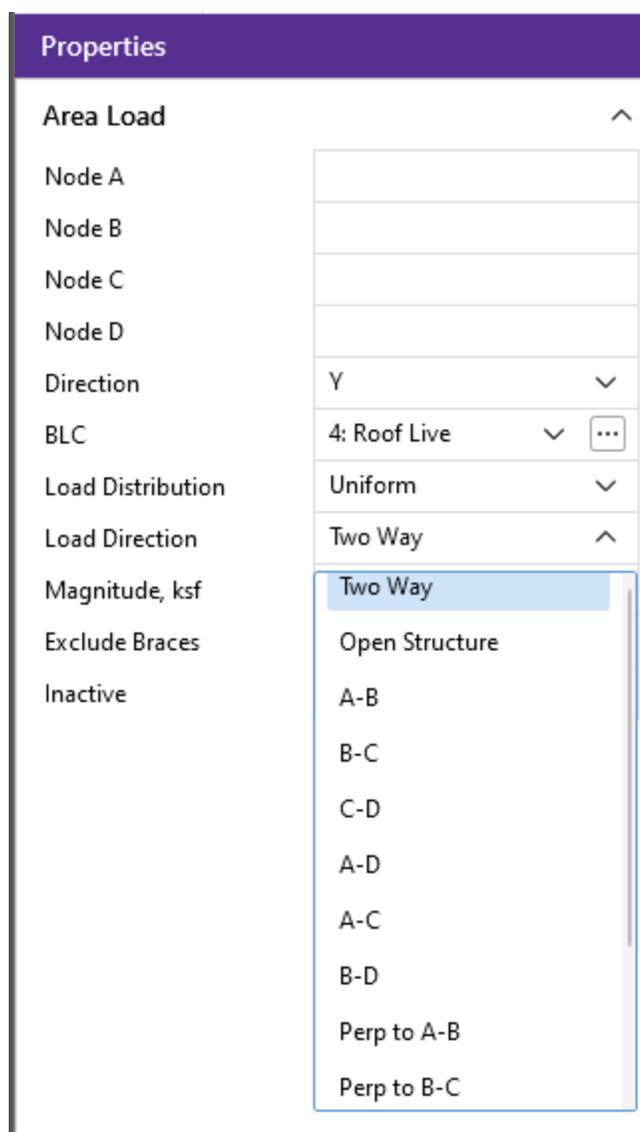
Note:

- By default, area loads are not assigned to Vbraces, Hbraces, Tension-only, Compression-only, and Euler Buckling members. If you want to include them, you can uncheck the "**Exclude Braces**" checkbox for that area load, which will be discussed in the later sections.
- The nodes that define an area load must be coplanar.
- The magnitudes of the tapered area load must be coplanar. Program marks the invalid magnitudes in red in Property Panel and Member Area Loads spreadsheet, which will be discussed in later sections.

Draw Area Loads



To apply area loads to members enter the load direction and magnitude. Make sure that you are careful to enter the correct BLC number that you want the loads assigned to.



Apply Member Area Loads

To o apply member area loads:

1. If there is not a model view already open then go to the **View** tab and select the **Open 3D Views** button to open a new view.
2. Go to the **Home** ribbon.
3. In the 'Draw Loads' section click the **Area** icon.
4. Click on four nodes to define the area. For three sided areas click on the last node twice.



Note:

- You can also specify or edit area loads in the **Member Area Loads Spreadsheet**.
- You can undo any mistakes by clicking the  **Undo** button in the Quick Access toolbar.

Member Area Loads Spreadsheet

The **Member Area Load** Spreadsheet records the area loads to be attributed to the members and may be accessed by going to the **Spreadsheets** tab **Data Entry** button ► **Member Area Loads** or by clicking the **Member Area Loads** button in the **Data Entry** Panel.

Member Area Loads													
BLC 1:		▼		Prev BLC		Next BLC							
	Node A	Node B	Node C	Node D	Direction	Load Direction	A Magnitude [ksf]	B Magnitude [ksf]	C Magnitude [ksf]	D Magnitude [ksf]	Exclude Braces	Inactive	
1	N1	N2	N4	N3	Y	Two Way	-0.01	-0.01	-0.01	-0.01	<input checked="" type="checkbox"/>		
2	N9	N10	N12	N11	Y	Two Way	-0.01	-0.01	-0.02	-0.02	<input checked="" type="checkbox"/>		
3	N17	N18	N20	N19	Y	Two Way	-0.01	-0.01	-0.03	-0.03	<input checked="" type="checkbox"/>		

When you open this spreadsheet, you can view only one basic load case at a time. Use the drop down list or the   buttons to specify a different load case. The current load case is also displayed in the title bar at the top of the spreadsheet.

The first four columns contain the nodes that define the area to be loaded. The fifth and sixth columns indicate the direction and distribution of the load, both are discussed below.

The next four columns hold the load magnitude in four different nodes. Notice area load supports both uniform and tapered loads.

The Exclude Braces column allows you to include or exclude certain members in the area load attribution, including Vbraces, Hbraces, Tension-only, Compression-only, and Euler Buckling members.

The last column allows you to set the specified load as an "Active" load that will be considered for the analysis, or an "Inactive" load that will not be included in the analysis.

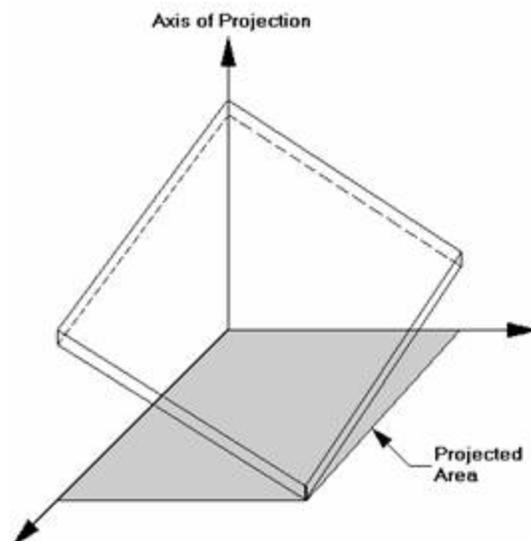
Member Area Load Direction

The direction code indicates the direction of application for the area load. The following directions are available:

Global loads are applied without being modified for projection. For example, a global Y-direction load of 1 ksf applied to an inclined plane with an area of 10 sq.ft. generates a total force of 10 kips, no matter what the incline is.

Projected loads, on the other hand, are applied in the global directions, but their actual magnitude is influenced by the planar orientation. The load is applied to the projected area of the element that is perpendicular to the load.

For example, a "PY" direction load is a projected load applied in the global Y direction. The actual magnitude of the load is the entered magnitude reduced by the ratio A/A_{xz} . "A" is the actual area of the element and A_{xz} is the element's projected area on the X-Z plane, which is always less than or equal to the actual area. See the following figure:



If the "Axis of Projection" in the figure is the Y-axis, then the shaded area is the total element area "projected" onto the plane perpendicular to the Y-axis (which happens to be the X-Z plane). The total load generated is equal to the input magnitude applied to the projected area. The generated load is then applied to the whole area, so the generated load magnitude is reduced accordingly.

For additional advice on this topic, please see the RISA Tips & Tricks website:

www.risa.com/post/support. Type in Search keywords: **Projected Loads**.



Note: For the loading perpendicular to the plane of the area polygon, the order in which you click on the joints will determine whether the load direction is into or away from the plane of the area. Clicking on the ABCD joints in a clockwise order will create a loading heading down towards the plane.

Area Load Distribution

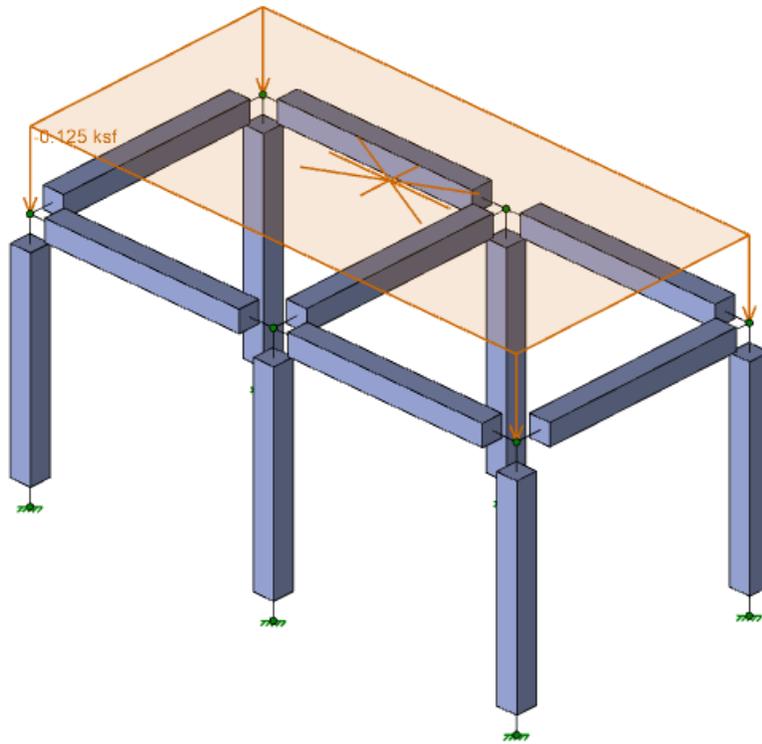
Properties	
Area Load ^	
Node A	
Node B	
Node C	
Node D	
Direction	Y ▾
BLC	1: ▾ ...
Load Distribution	Uniform ▾
Load Direction	Two Way ▾
Magnitude, ksf	0
Exclude Braces	<input checked="" type="checkbox"/>
Inactive	Active ▾

The Load Direction of the area load determines which members "support" the area load you define. You may choose between three main load direction options: two-way, one-way, or open structure. Below we will explain each of the main options. Next, you can choose to use uniform or tapered area load for Load Distribution option. The general concept of calculating loads attributed to supporting members is the same for uniform or tapered load. One thing to note is that if you use tapered load, you need to make sure the load magnitudes at those four nodes will be coplanar. If your input magnitudes failed in the coplanar check, program will use red font to remind users:

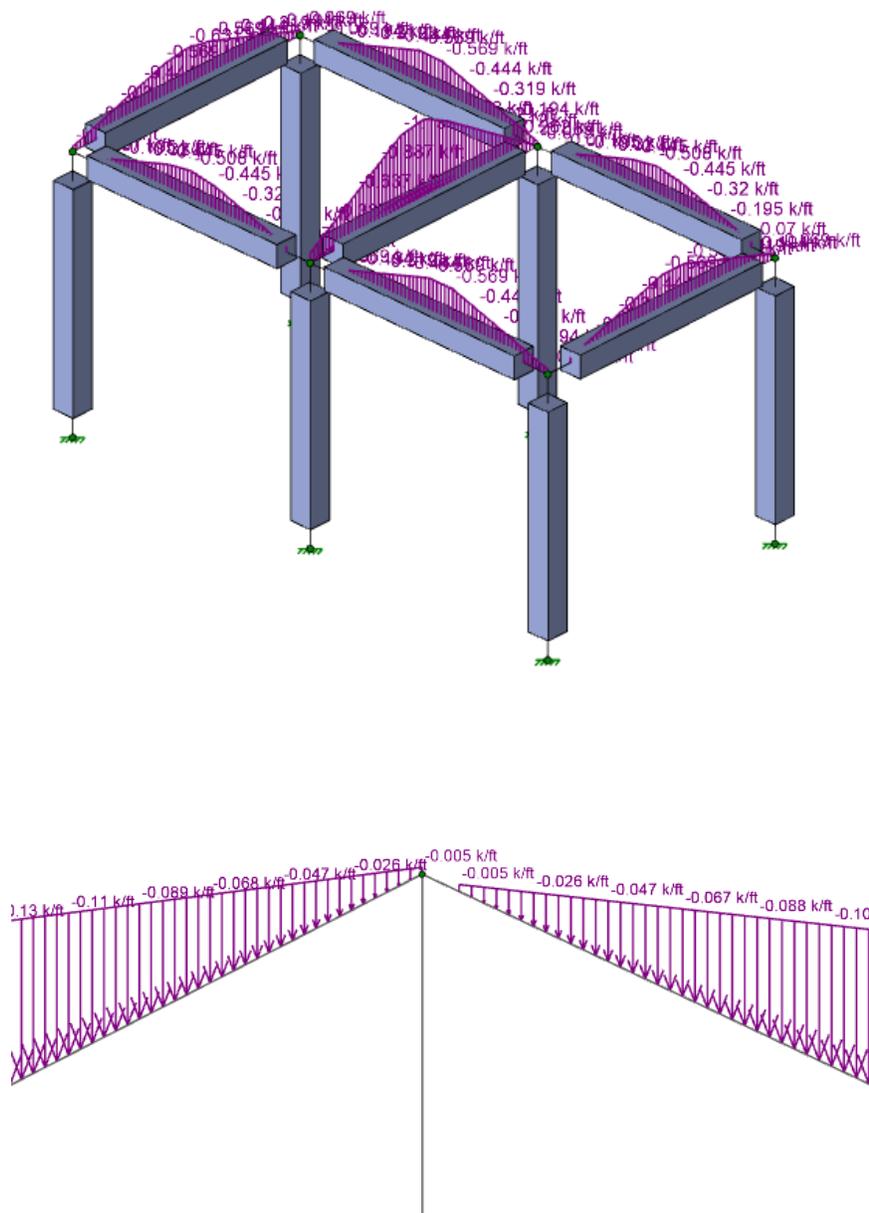
Properties	
Area Load ^	
Node A	N52
Node B	N58
Node C	N33
Node D	N15
Direction	Y
BLC	4: Roof Live
Load Distribution	Tapered
Load Direction	A-B
A Magnitude, ksf	0
B Magnitude, ksf	-0.2
C Magnitude, ksf	-0.2
D Magnitude, ksf	-0.2
Exclude Braces	<input checked="" type="checkbox"/>
Inactive	Active

Two Way

Two way area load distribution is analogous to how load travels through a two-way slab. Load will be distributed to the nearest member, regardless of direction. When a two-way area load is drawn there will be a star pattern shown to verify that load will go out in all directions. Here is an example model of a two-way slab with an area load:



At solution, we can view how this load was applied by viewing the appropriate transient Basic Load Case:

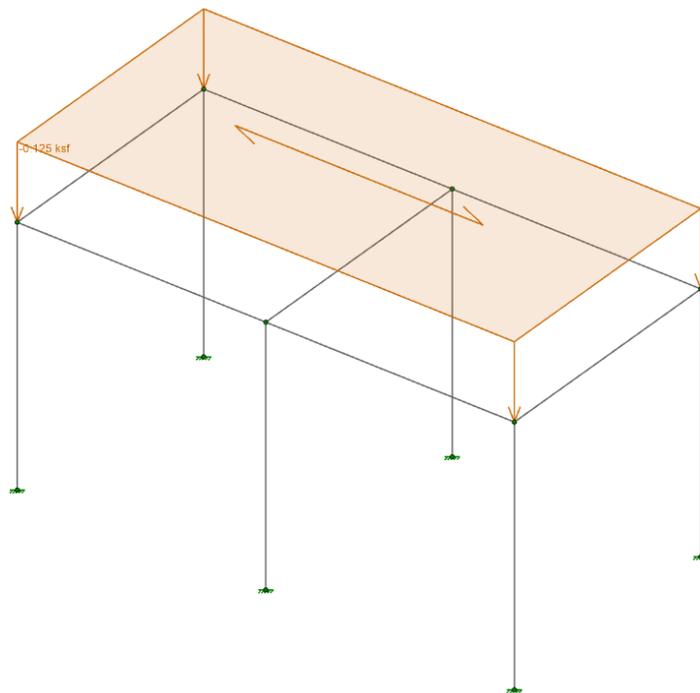
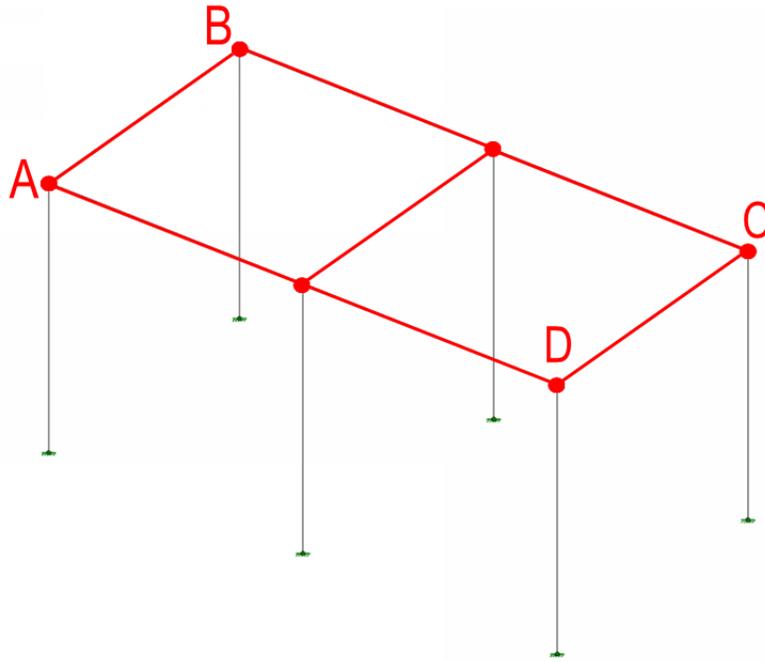


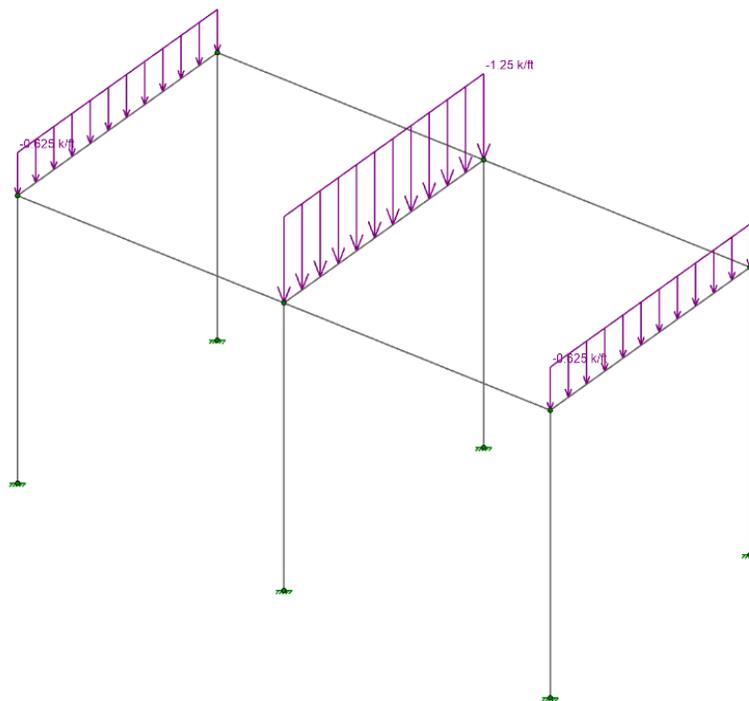
Here we can directly see how the load was applied. See the [Area Load Attribution](#) section for more information.

One Way

For a one-way distribution, you are asking the program to distribute loads in a single direction. There are multiple one-way options that all depend on the order that you click to create your area load. All options from **A-B** to **Perpendicular to B-D** are one way options. The A-B option means that the load

will span one-way in a parallel direction to the first two corners of the area load (the A and B corners). Below is an image of what a **B-C** one-way area load distribution would look like:

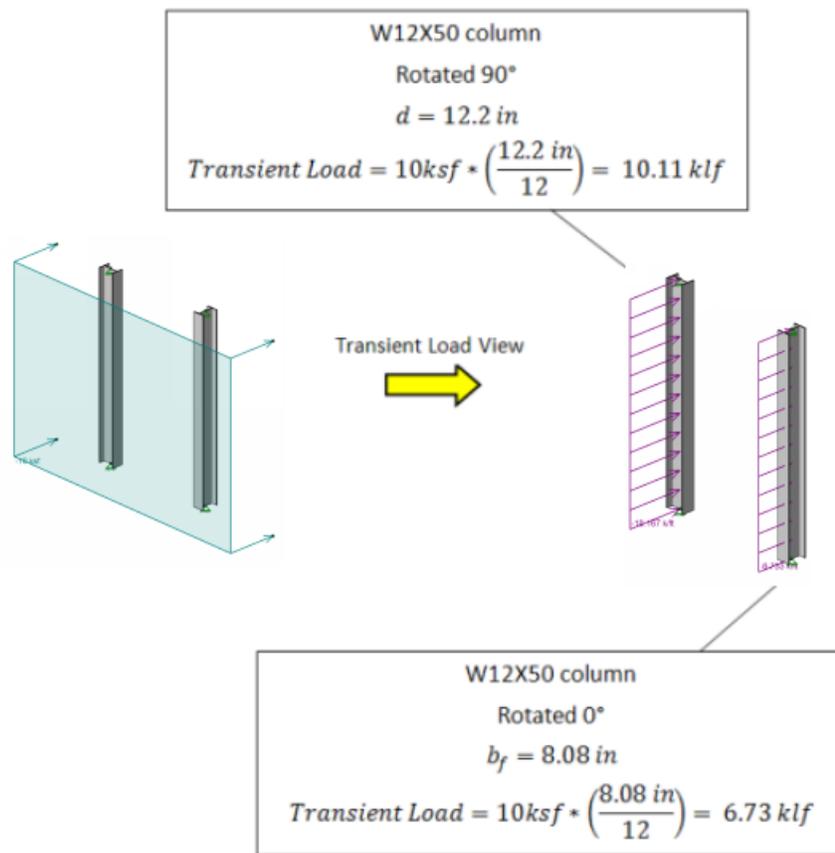




For odd geometries the "Perpendicular..." options may be necessary to get the load distribution you are looking for.

Open Structure

As a third option, You can elect for the load to be distributed as an "Open Structure." This option applies the area load to each member based solely on the projected surface area (in the direction of the loading) of that member. The "Open Structure" distribution option is generally intended for open/lattice type structures for which a wind pressure is acting uniformly on all the exposed structural surfaces of the structure. The program will distribute the area load to the members as member distributed loads calculated as the area load magnitude multiplied by the face of the member perpendicular to the direction of the "Open Structure" load. Below is an example of the calculation for the "Open Structure" attribution:

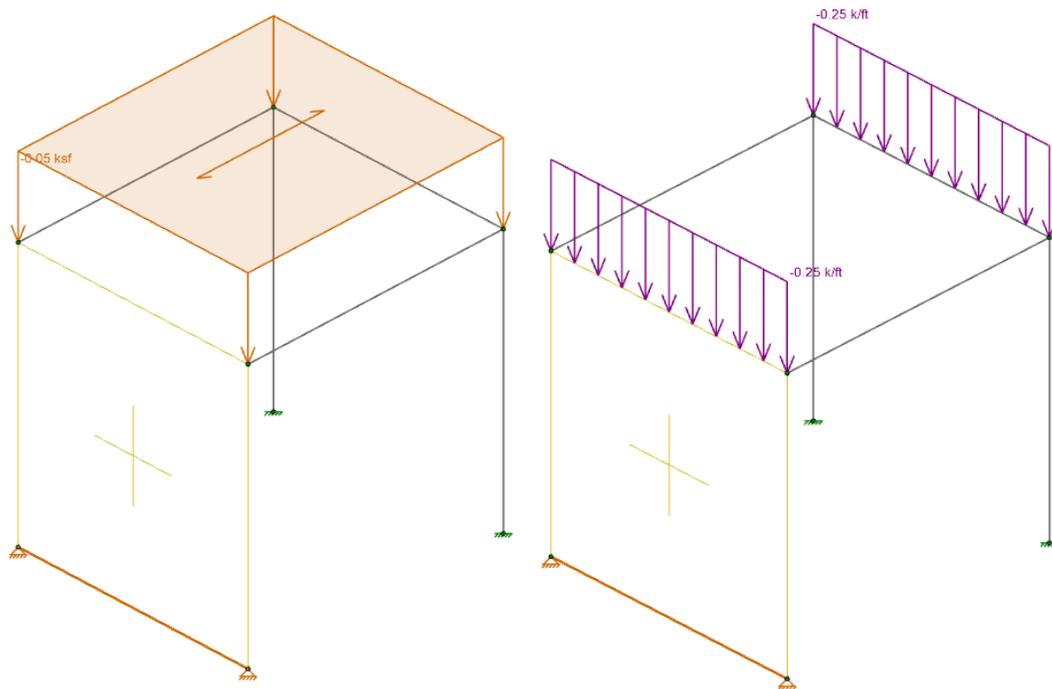
**Note:**

- The "Open Structure" load distribution option applies load to the members in the plane of the applied load as well as the members in the structure in-front of or behind that plane.
- Currently the "Open Structure" option does not take shielding into account for members behind other members.
- The example above shows a uniform open structure area load example, if you use a tapered load, the transient load on the member could be tapered line load.

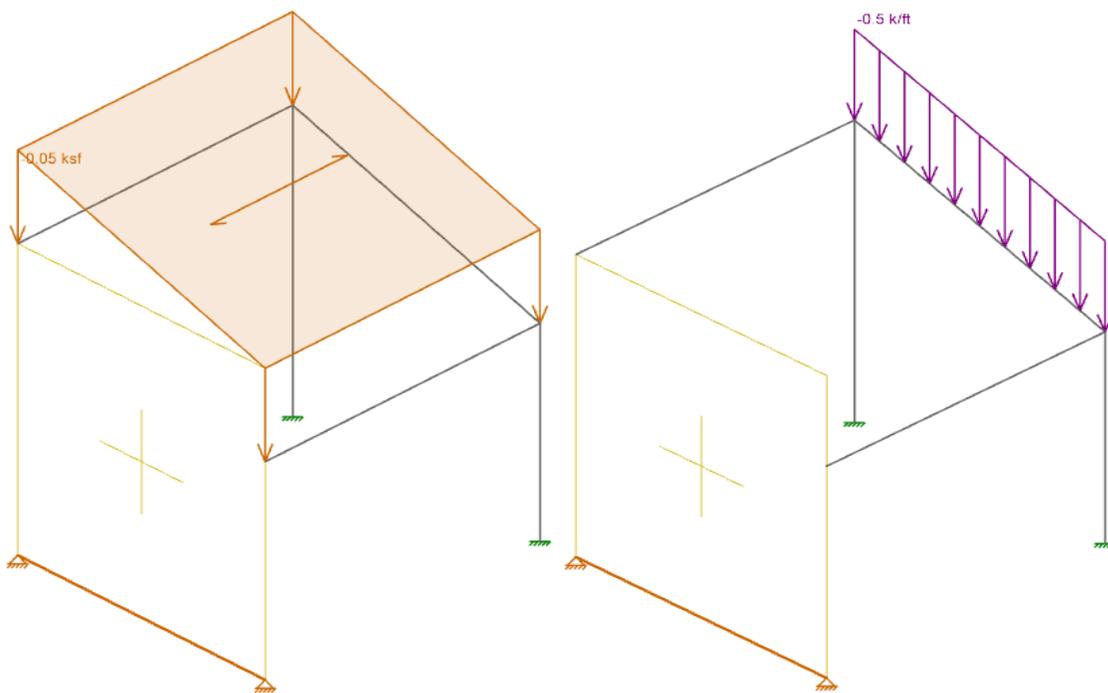
For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Member Area Load**.

Area Loads and Wall Panels

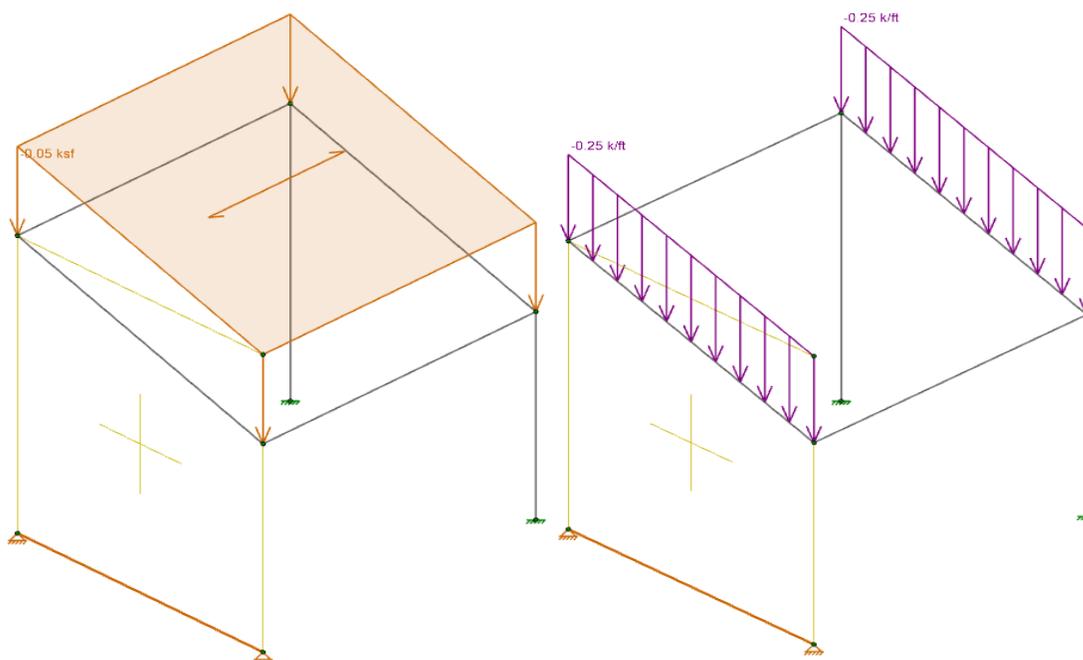
Wall panels can now accept area loads as well. This, however, will only occur at the tops of walls and at diaphragm locations. If there is an area load applied at the top of a wall or where there is a diaphragm applied, then the wall panel will receive area load in the same way as a member.



This, however, will not work if the area load is not applied at the top of the wall, or there is no diaphragm defined where the area load intersects the wall panel. In this case no load will be applied to the wall. All of the load will go in the opposite direction.



You can force area loads to attribute to walls even if there is not a diaphragm at that elevation. This can be done by drawing in a "dummy" member or ledger beam at the same location as the wall. This member will receive area load like any other member, but should then transfer its load directly into the wall.



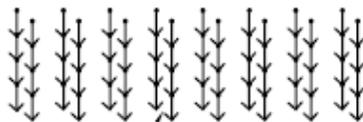
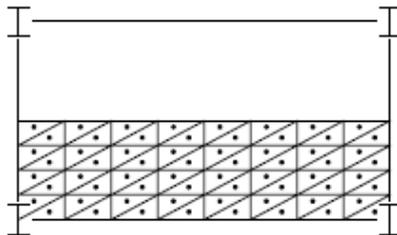
Area Load Attribution

Meshing

The program takes the area loads defined by you and breaks the loads down into finite "pieces". These pieces are broken down until the side dimension of the "piece" is less than that defined in **Member Area Load Mesh Size** in the [Model Settings - Solution](#) tab. These pieces of load are then attributed to the members in the plane of the load according to the distribution defined.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Verify Area Loads**.

ONE WAY DISTRIBUTION

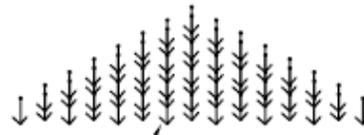
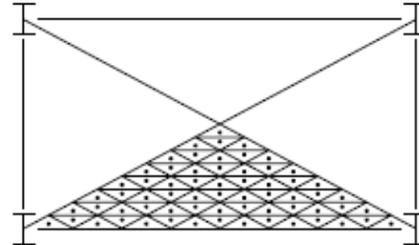


Point Loads=
Area of Load*
Load Magnitude
(Applied at
Centroid of Area)

Transient Load
Diagram (As
Seen in RISA
After Smoothing
Algorithm)



TWO WAY DISTRIBUTION



Point Loads=
Area of Load*
Load Magnitude
(Applied at
Centroid of Area)

Transient Load
Diagram (As
Seen in RISA
After Smoothing
Algorithm)

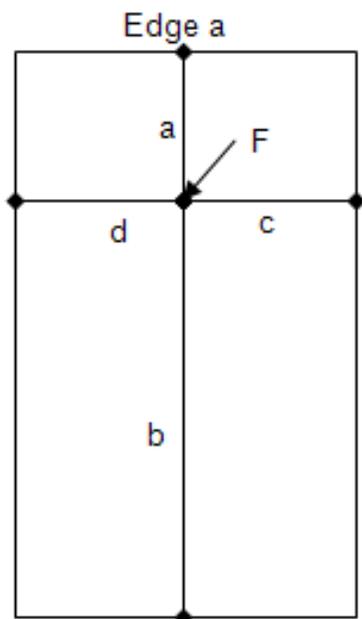


One way Attribution

One-way areas are attributed in the span direction based on the assumption of a simply supported deck. Each point load "piece" is attributed to supporting members based on the ratio of the distance of that member compared to the total deck span at that location. If there is no member in that direction then the load is attributed as a two-way load.

Two Way Attribution

Two-way areas are attributed to the members in all 4 directions based on an extension of a simply supported deck. Each point load "piece" is attributed to the members based on the ratio of the distance of that member compared to the distance of that point from the other members as shown in the image below:



$$\begin{aligned} \text{Load attributed to Edge a} &= (1/a) / (1/a+1/b+1/c+1/d) * F \\ \text{Load attributed to Edge b} &= (1/b) / (1/a+1/b+1/c+1/d) * F \\ \text{Load attributed to Edge c} &= (1/c) / (1/a+1/b+1/c+1/d) * F \\ \text{Load attributed to Edge d} &= (1/d) / (1/a+1/b+1/c+1/d) * F \end{aligned}$$

This results in a load distribution that agrees very closely with traditional hand calc methods.

Exclude Braces Option

By default, area load attribution does not include "brace-type" members, e.g., Vbraces, Hbraces, Tension-only, Compression-only, and Euler Buckling members. However, if users want to include the "brace-type" members in the load attribution, you can uncheck the "Exclude Braces" checkbox for that area load:

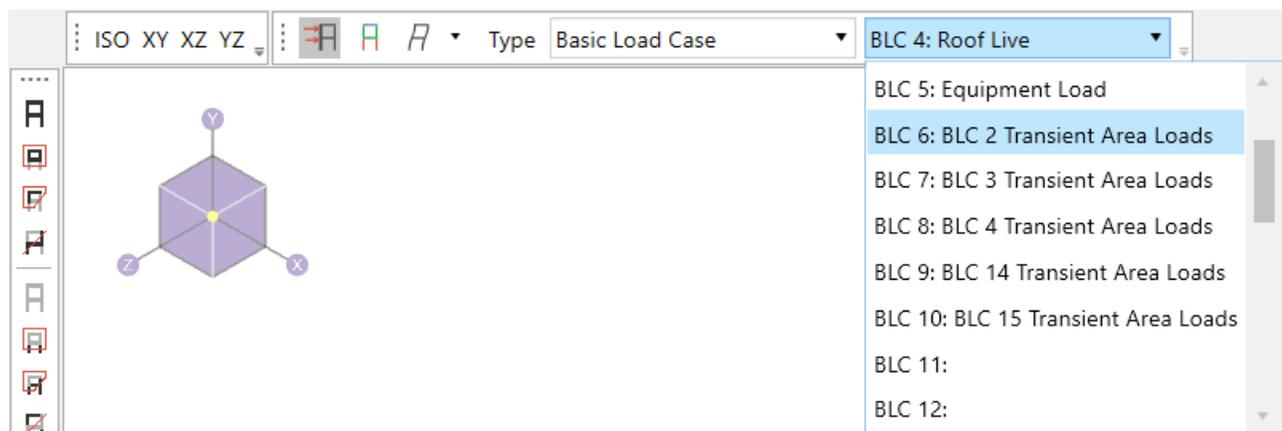
Properties

Area Load ^

Node A	N52
Node B	N58
Node C	N33
Node D	N15
Direction	Y ▾
BLC	4: Roof Live ▾ ...
Load Distribution	Tapered ▾
Load Direction	A-B ▾
A Magnitude, ksf	-0.1
B Magnitude, ksf	-0.1
C Magnitude, ksf	-0.2
D Magnitude, ksf	-0.2
Exclude Braces	<input checked="" type="checkbox"/>
Inactive	Active ▾

Transient Area Loads

After solution, the resulting distribution of the area load is stored in the first available basic load case with "BLC # Transient Area Loads" as the description. They can be viewed graphically or in the spreadsheets as a basic load case. These loads are transient which means they are only a result of the area load and will be deleted when the results are cleared and re-determined each time you solve the model. The only purpose of these transient BLC's is to let you view the exact load attribution created by the program.

**Note:**

- Members that are designated as Tension Only, Compression Only, or Euler Buckling will not receive any load during the Area Load Attribution.
- Members that are designated with a member type of VBRACE or HBRACE will not receive any load during the Area Load Attribution.

You can disconnect the attributed loads and take control of them by assigning a category to the automatically created basic load case. The original area loads will be left as they were so a subsequent solution will produce another distribution of loads that are a result of the defined area loads.

Solution Speed

Note that the area load algorithm used by RISA-3D is very versatile in that it can work in any of the global directions in a three dimensional model, so that it can be used for vertical dead or live load modeling as well as wind loads. But it can add a fair amount to the analysis time when small mesh sizes are used with large models. Sometimes a small mesh is necessary to get the desired accuracy for the area load attribution.

Loads - Point Loads

Point loads are concentrated loads applied along the span of a member or the edge of a wall panel. Defining point loads may be accomplished graphically or in the spreadsheets. See [Drawing Point Loads](#) below to learn how to draw joint loads graphically.

Draw Point Loads

To apply point loads, enter the load direction, magnitude, and location. Make sure that you are careful to enter the correct BLC number that you want the loads assigned to.



Note: In the event that a member end offset exists and the load lies on the offset the load will be applied at the end of the offset.

Properties

Point Load ^

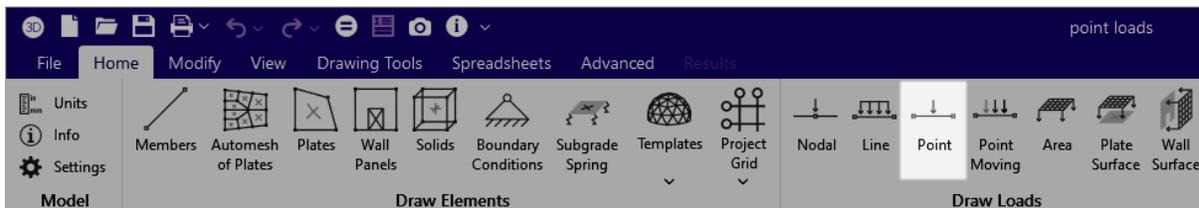
Direction	y v
BLC	1: Self Weight v ...
Magnitude, (k)	0
Location, (ft, %)	0
Inactive	Active v

Click to Apply
Apply to Selected

Apply Point Loads

To apply Point Loads:

1. Go to the **Home** ribbon.



2. In the 'Draw Loads' section, click the **Point** icon to apply a Point Load.
3. You can choose to apply the load to a single member/wall panel at a time or to an entire selection of members/wall panels.the load.
 - To apply the load to just a few elements: Click the **Click to Apply** button, then click on the members or wall panels with the left mouse button.
 - To apply the load to a selection, click the **Apply to Selected** button.

**Note:**

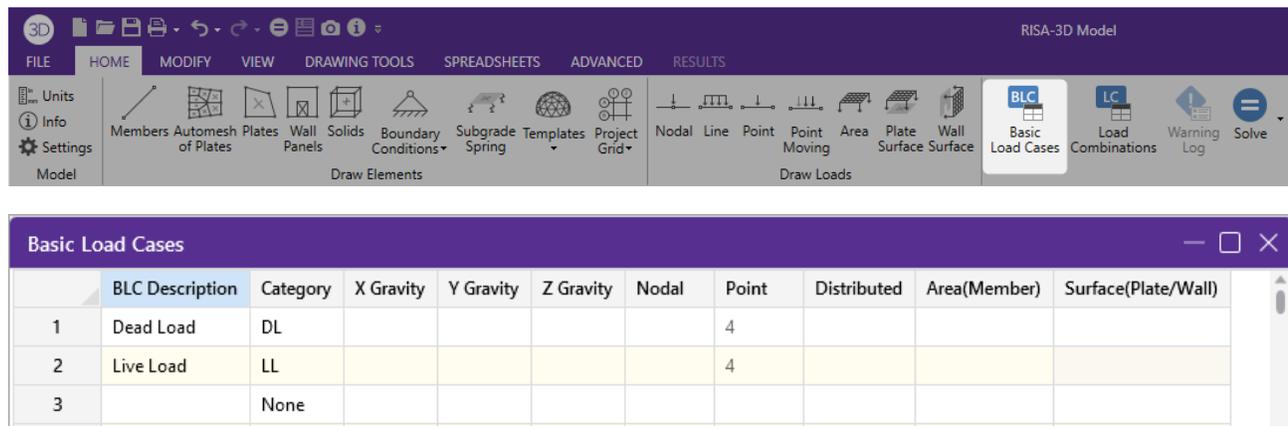
- To apply more loads with different parameters, change the parameters in the **Properties** panel and choose **Click to Apply** or **Apply to Selected** to apply the new loads.
- You can also specify or edit point loads in the **Point Load Spreadsheet**.
- You can undo any mistakes by clicking the **Undo**  button.

Point Loads from a Moving Load

Please see the [Moving Loads](#) topic for more information on how to create static point loads from a specific moving load step location.

Point Load Spreadsheet

You can access the **Point Loads** spreadsheet from the **Basic Load Cases** spreadsheet which is located on both the **Data Entry** and the **Home** tab. In the **Basic Load Cases** spreadsheet, click on a cell in the **Point** column.



The screenshot shows the RISA-3D software interface with the **Basic Load Cases** spreadsheet open. The spreadsheet has the following columns: BLC Description, Category, X Gravity, Y Gravity, Z Gravity, Nodal, Point, Distributed, Area(Member), and Surface(Plate/Wall). The data is as follows:

	BLC Description	Category	X Gravity	Y Gravity	Z Gravity	Nodal	Point	Distributed	Area(Member)	Surface(Plate/Wall)
1	Dead Load	DL					4			
2	Live Load	LL					4			
3		None								

The **Point Load Spreadsheet** records the point loads for the member elements and may be directly accessed by selecting the **Point Loads** spreadsheet from the **Data Entry** menu.

	Member Label	Direction	Magnitude [k, k-ft]	Location [(ft, %)]	Inactive
1	M1	Y	-0.906	0	
2	M1	Y	-0.906	2	
3	M1	Y	-0.906	4	
4	M1	Y	-0.906	6	

The first column contains the label of the member or wall panel to receive the load.

The direction in the second column represents the direction of the load as one of the options mentioned above.

The load magnitude is recorded in the third column. The units for the magnitude are listed at the top of the column, depending upon whether the load is a force or a moment.

The fourth column contains the location of the load. The location is the distance from the I-joint of the member and is unaffected by any member offsets. The location of the load may be defined as a percentage of member length. To define the distance from the I-joint as a percentage of member length, enter the percentage value (0 to 100), preceded by the symbol "%". For example, a load in the center of the member would be defined with a location of "%50". Using a percentage value is handy if the member's length will be changing due to editing of the model coordinates and you wish to have the load some proportional distance from the I end.

The last column allows you to set the specified load as an "Active" load that will be considered for the analysis, or an "Inactive" load that will not be included in the analysis.

The **Wall Panel Point Loads Spreadsheet** records the point loads on your wall panels. Note that the loads are specific to the BLC's and that you can use the drop-down list to scroll between your various load cases. The columns in the spreadsheet are the same as the member point loads spreadsheet.

For models integrated from RISAFloor with semi-rigid diaphragms, the **Diaphragm Point Loads Spreadsheet** records the point loads on your semi-rigid diaphragm. These loads will be grayed out since they cannot be edited.

**Note:**

- If you are using RISAFloor to bring your model into RISA-3D with a rigid or flexible diaphragm, this spreadsheet will be automatically populated with the point loads from RISAFloor.
- Loads from RISAFloor will be grayed out because they cannot be edited in RISA-3D since they are tied to the RISAFloor analysis.

Wall Panel Point Loads					
Member	Wall Panel	Diaphragm			
BLC 1: Dead Load		▼	Prev BLC	Next BLC	
	Wall Label	Direction	Magnitude [k, k-ft]	Location [(ft, %)]	Inactive
1	WP1(15ft)	Y	-8.854	64	
2	WP1(15ft)	Y	-9.815	54	
3	WP1(15ft)	Y	-9.596	35.2	
4	WP1(15ft)	Y	-9.606	26.4	
5	WP1(15ft)	Y	-9.614	17.6	
6	WP1(15ft)	Y	-9.517	0.0	

Point Load Directions

x, y, z - Load applied in local x,y or z direction

X, Y, Z - Load applied in global X,Y or Z direction

My, Mz - Moment about member local y or z axis

Mx - Torsional Moment about member local x-axis

Loads - Line Loads

Line loads are loads that are spread across all or part of a member or wall panel and can be of uniform, stepped, or varying magnitude such as triangular or trapezoidal. You may define distributed loads graphically or by using the spreadsheets. See Drawing Line Loads below to learn how to draw line loads graphically.

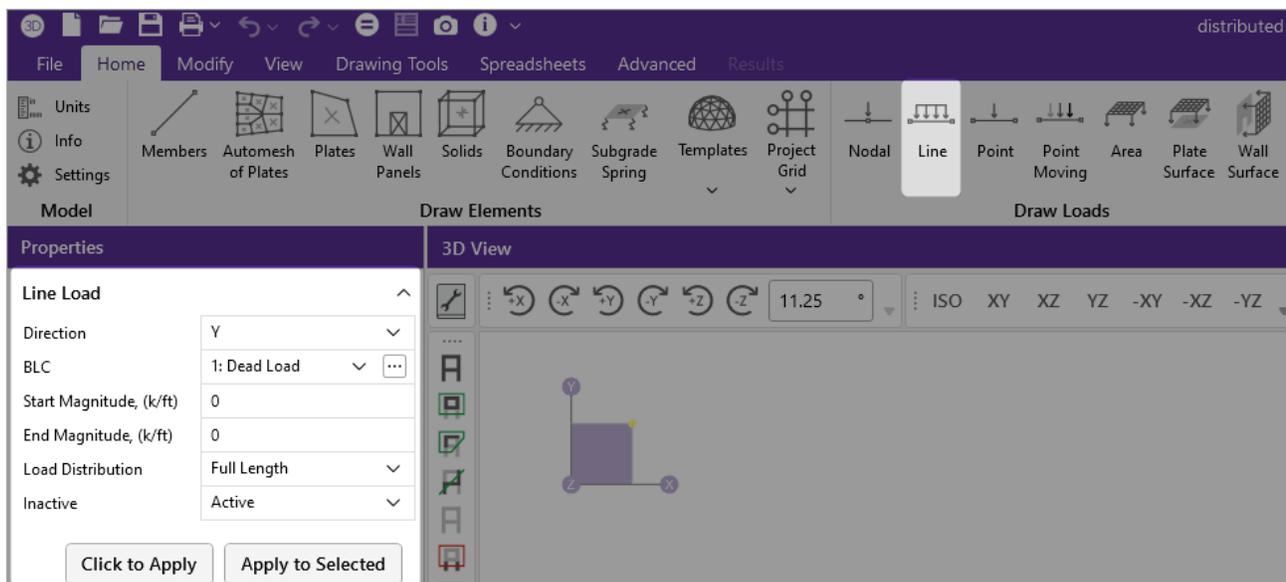


Note: If a member has offsets defined, the offset distances will NOT be loaded. The locations are still relative to the I Node, but if the start location is less than the I End Offset, the part of the load applied along the offset distance will be ignored. The same is true for the end location and the 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.

Draw Line Loads

You can draw line loads to members or wall panels. The direction of the load may either be defined in the global axes or the local axes of the member and may also be projected. For full length loads leave the Start and End Locations as zero.

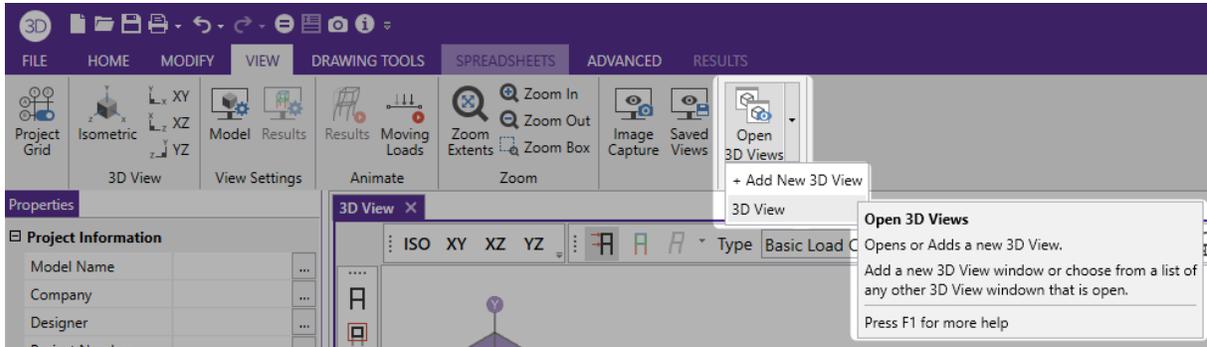
Make sure that you are careful to enter the correct Basic Load Case number that you want the loads assigned to.



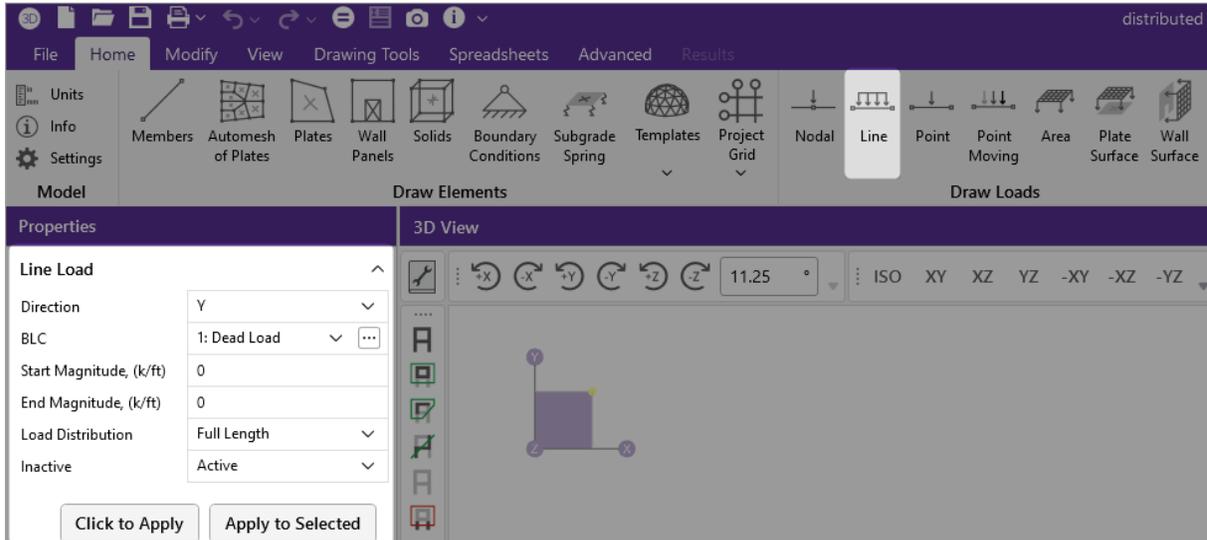
Draw Line Loads

To draw Line Loads:

1. If there is not a model view already open, choose the **Add 3D View** option from the **Open 3D Views** drop-down on the **View** ribbon



2. Click the **Draw Line Load** icon and define the load in the Property Panel.



3. You may choose to apply the load to a single element at a time or to an entire selection of members and/or wall panels.
 - To draw the load to only a few members/wall panels, choose **Click to Apply** Click on the members/wall panels with the left mouse button to apply the load.
 - To apply the load to a selection of members/wall panels, choose **Apply to Selected**. The load will then be applied to all the selected elements in the model.

**Note:**

- To apply more loads with different parameters, press **CTRL + D** to recall the **Draw Line Load** tool.
- You may also specify or edit line loads in the **Distributed Load Spreadsheet**.

Distributed (Line) Loads Spreadsheet

The **Member Distributed Load Spreadsheet** records the distributed loads for the member elements and may be accessed by selecting **Distributed Loads** from the **Data Entry** drop-down on the **Spreadsheets** tab of the ribbon, or by clicking **Distributed Loads** on the **Data Entry** toolbar.

	Member Label	Direction	Start Magnitude [k/ft, F, ksf, k-ft/ft]	End Magnitude [k/ft, F, ksf, k-ft/ft]	Start Location [(ft, %)]	End Location [(ft, %)]	Inactive
1	M4	Y	0.1	0.1	0	%100	
2	M1	Y	0.1	0.1	0	%100	
3	M2	Y	0.1	0.1	0	%100	
4	M3	Y	0.1	0.1	0	%100	
5	M5	Y	0.1	0.1	0	%100	
6	M6	Y	0.1	0.1	0	%100	

The first column contains the Label of the element being loaded.

The Direction specified in the second column indicates which axes are to be used to define the load directions and whether or not the load is to be projected. Directions are discussed in the next section.

Start and End Magnitudes of the load must be specified. Start and End locations for the load need only be specified if the load is not across the full member length. If both locations are left as zero then the load will be applied across the full member length.

The Location columns contains the location of the load. The location is the distance from the I Node of the member and is unaffected by any member offsets. The location of the load may be defined as a percentage of member length. To define the distance from the I Node as a percentage of member length, enter the percentage value (0 to 100), preceded by the symbol "%". For example, a load starting or ending in the center of the member would be defined with a start or end location of "%50". Using a percentage value is handy if the member's length will be changing due to editing of the model coordinates and you wish to have the load some proportional distance from the I end.

The last column allows you to set the specified load as an "Active" load that will be considered for the analysis, or an "Inactive" load that will not be included in the analysis.

The **Wall Panel Distributed Loads Spreadsheet** records the distributed loads on your wall panels. Note that the loads are specific to the BLCs and that you can use the drop-down list to choose a different one. The columns in the spreadsheet are the same as the member distributed load spreadsheet.

For models integrated from RISAFloor containing semi-rigid diaphragms, the **Diaphragm Distributed Loads Spreadsheet** records the distributed loads generated from the wind and seismic load generation. These lines will be grayed out because they cannot be edited from RISA-3D since they are tied to the RISAFloor analysis.



Note:

- If you are using RISAFloor to bring your model into RISA-3D with a rigid or flexible diaphragm, this spreadsheet will be automatically populated with the distributed loads from RISAFloor applied to members and wall panels.
- Loads from RISAFloor will be grayed out because they cannot be edited in RISA-3D since they are tied to the RISAFloor analysis.

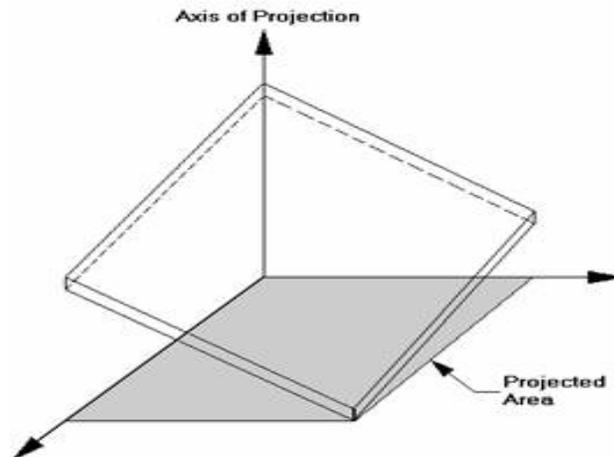
Wall Panel Distributed Loads							
Member Wall Panel							
BLC 1: Dead Load Prev BLC Next BLC							
	Wall Label	Direction	Start Magnitude [k/ft, F]	End Magnitude [k/ft, F]	Start Location [(ft, %)]	End Location [(ft, %)]	Inactive
1	WP1(10ft)	Y	0.368	0.368	0	%100	
2	WP1(0ft)	Y	0.368	0.368	0	%100	
3	WP1(0ft)	Y	0.368	0.368	0	%100	
4	WP1(0ft)	Y	0.368	0.368	0	%100	

Distributed Load Directions

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

Entry	Load Direction
Mx	Torque applied about the member's local x-axis
T (or t)	Thermal (temperature differential) load
PX	Projected load in the global X-axis direction
PY	Projected load in the global Y-axis direction

This diagram illustrates the difference between local (x, y, z) and global (X, Y, Z) direction loads:

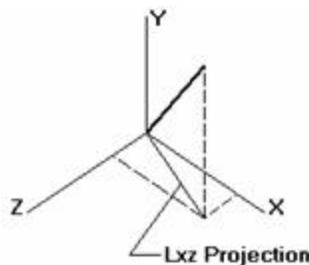


In this diagram, the local y and global Y loads shown are both negative, while the local x and global X loads are positive. As can be seen, local direction loads line up with the element's local axis directions, so their direction relative to the rest of the model changes if the member/wall panel orientation changes. Global loads have the same direction regardless of the element's orientation.

A distributed load in the M_x direction will be applied to the member/wall panel according to the right hand rule. Remember that the positive local x -axis extends from the I node of a member towards the J node.

Projected Loads

Keep in mind that global loads are applied without being modified for projection. For example, a full length Y direction load of 1 kip/foot applied to a 10 foot member inclined at 45 degrees generates a total force of 10 kips. Projected loads, on the other hand, are applied in the global directions but their actual magnitude is influenced by the member's orientation. The load is applied to the "projection" of the member perpendicular to the direction of the load. For example, a "PY" direction load is a projected load applied in the global Y direction. The actual magnitude of the load is the entered magnitudes reduced by the ratio L/L_{xz} , where L is the member's full length and L_{xz} is the member's projected length on the global X - Z plane. See the following figure:

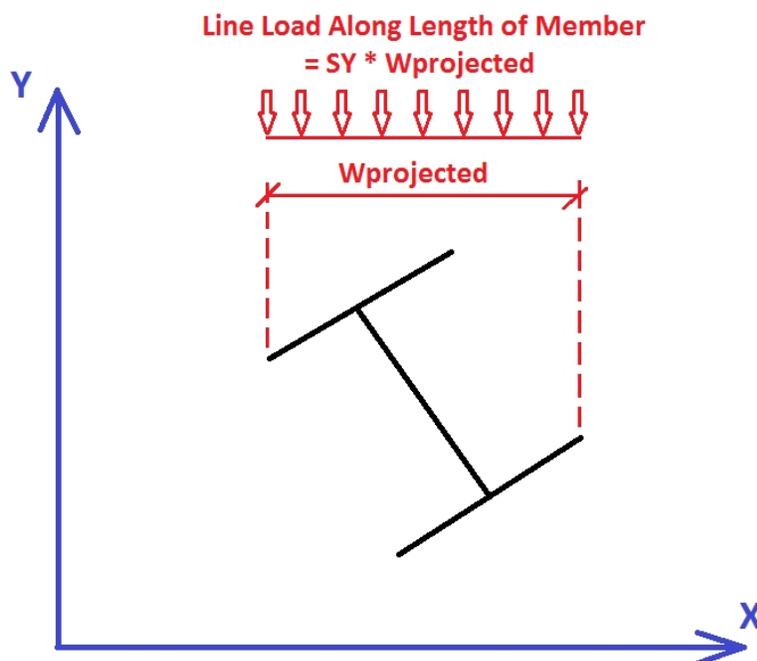


So the total load generated is equal to the input magnitudes applied along the projected length. This generated force is distributed along the full member length, so the applied magnitudes are reduced accordingly.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Projected Loads**.

Surface Loads

This is a load type that applies a pressure load to the face of the member in the direction specified. If a local axis direction is specified then the load will be applied to the strong/weak direction of the member regardless of the member orientation. If using the global coordinates then the program will take the projected face of the element in the direction in question and calculate the pressure area accordingly.



The program then takes this pressure multiplied by the width of the pressure area and creates a line load that is then applied to the member.



Note: Distributed pressure loads can not be applied to tapered members or wall panels.

Loads - Moving Loads

The standard AASHTO loads are built into the moving loads database, however you can add and save custom moving loads as well. The moving loads can be applied in any direction, so they can be used to model crane loadings (which are typically applied in 2 or 3 directions at the same location). You can have up to 5,000 moving loads in each model.

Apply Moving Loads

To apply a Moving Load:

1. Select the **Moving Loads** spreadsheet from the 'Explorer' panel 'Data Entry' section.
2. Specify a pattern in the **Pattern** column by selecting it from the drop down list.
3. In the **Increment** column specify the distance for the load pattern to be stepped through the path.
4. Specify the path by defining the **Nodes** in the remaining columns and indicate if you wish the pattern to be moved **Both Ways** through the path.

Moving Loads										
	Tag	Pattern	Increment [ft]	Both Ways	1st Node	2nd Node	3rd Node	4th Node	5th Node	6th Node
1	M1	HS20-26	1	<input checked="" type="checkbox"/>	N2	N6				
2	M2	HS20-30	1	<input checked="" type="checkbox"/>	N1	N2	N4	N3		
3	M3	HS20-30	2	<input type="checkbox"/>	N1	N2	N4	N3		

**Note:**

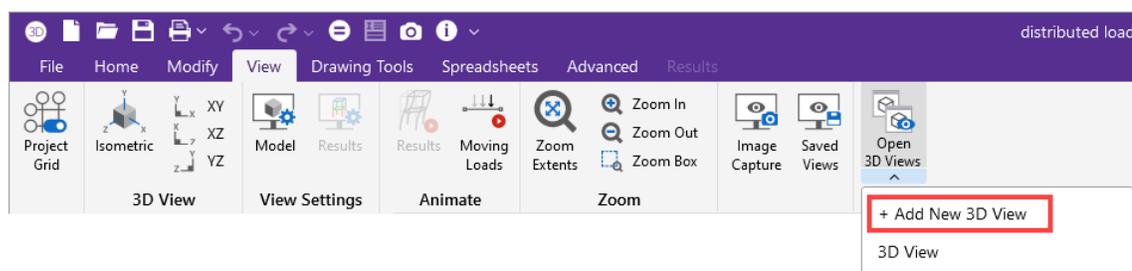
- You can skip nodes when specifying the path. The moving load feature is “smart” in the sense that it will try to find a way to get from one node to the next node in the load path sequence. The load path taken will usually be the most direct route between the nodes and may be verified by animating the moving load.
- For example, in the image below, if you want the moving load to move left to right along the beam member (M4), there is no need to specify the intermediate node (N4) in the moving load path.



Animate Moving Loads

To animate a Moving Load:

1. If there is not a model view already open:
 - a. Go to the **View** ribbon.

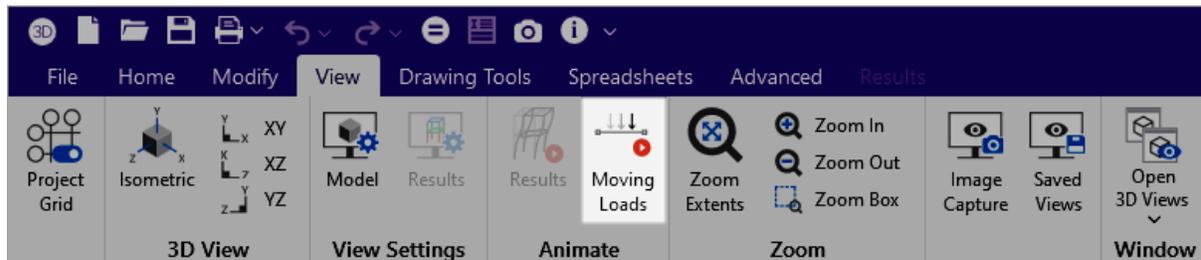


- b. In the Window section, click the **Open 3D Views** icon's arrow.
 - c. Choose **+ Add New 3D View** from the menu.

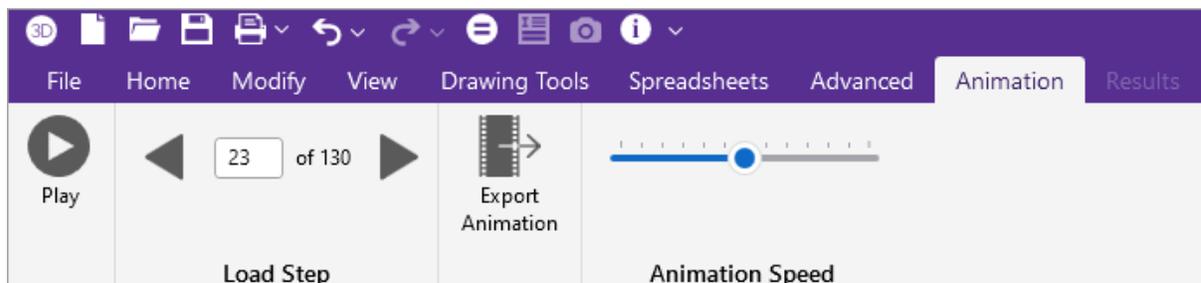
This opens a new view in which you can make any adjustments you wish to appear in the animation.

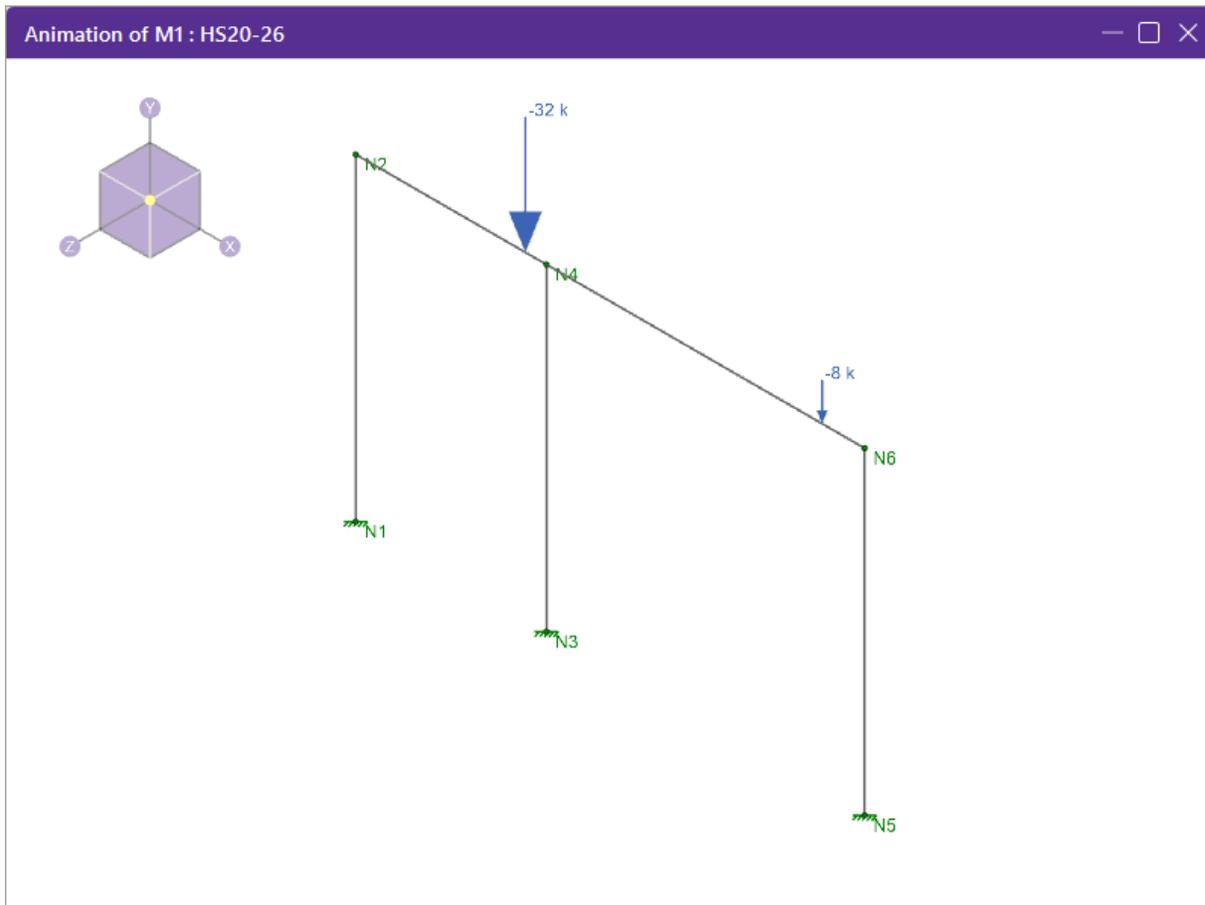
You can also make further view adjustments in the animation view.

2. Click the **Moving Loads** icon on the 'View' ribbon.



3. Select the moving load or load combination from the drop-down list at the bottom of the options and then click on the **Animate** button.

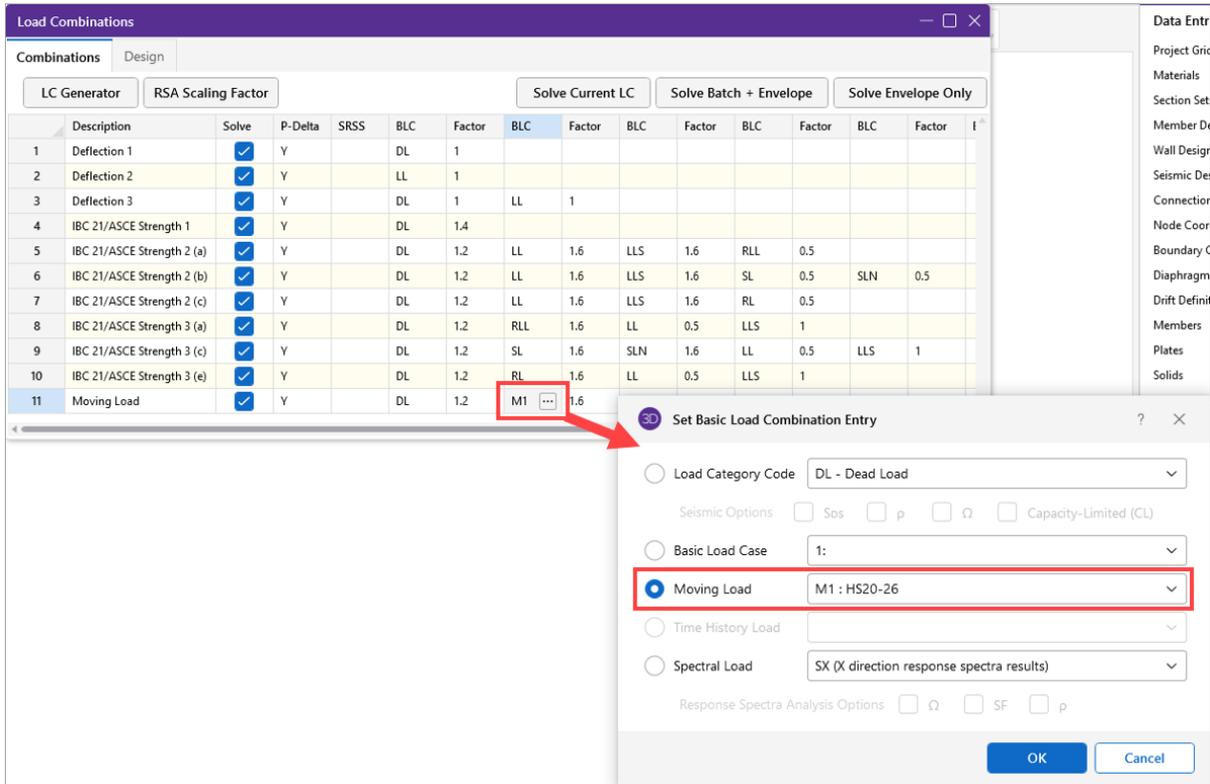




Include a Moving Load in a Load Combination

To include a Moving Load in a Load Combination analysis:

1. Specify the moving load in the **Load Combinations** spreadsheet in one of the **BLC** columns, and enter a corresponding **BLC Factor**.
2. You can either type in the moving load tag (i.e. **M1**, **M2**, etc.) directly or you can select it by clicking the ellipsis to open the **Set BLC Entry** window.



3. Choose the appropriate moving load from the drop down list.

Moving Loads Spreadsheet

The **Moving Loads Spreadsheet** records the moving loads for the member elements and may be accessed by choosing **Moving Loads** in the **Data Entry** 'Explorer' panel 'Data Entry' section..

	Tag	Pattern	Increment [ft]	Both Ways	1st Node	2nd Node	3rd Node	4th Node	5th Node	6th Node
1	M1	HS20-26	1	<input checked="" type="checkbox"/>	N2	N6				
2	M2	HS20-30	1	<input checked="" type="checkbox"/>	N'	N2	N4	N3		
3	M3	HS20-30	2	<input type="checkbox"/>	N'	N2	N4	N3		

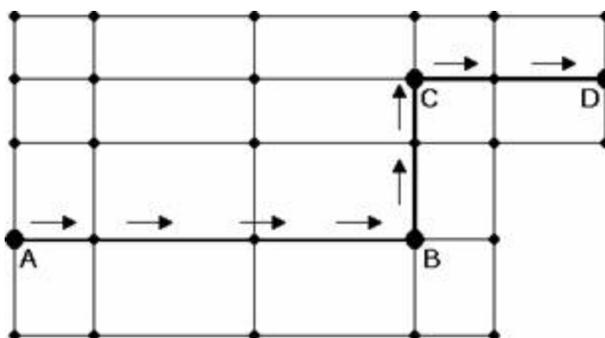
Each moving load definition is automatically assigned a **Tag** on the left side of the spreadsheet. These labels may not be edited. Moving loads are included in your analysis by referencing this label on the **Load Combinations** spreadsheet.

The **Pattern** column 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 can access the [Moving Load Patterns](#) and add or edit your own patterns by clicking on the ellipsis button.

The **Increment** column is the distance that the moving load will be moved for each step in the moving load analysis.

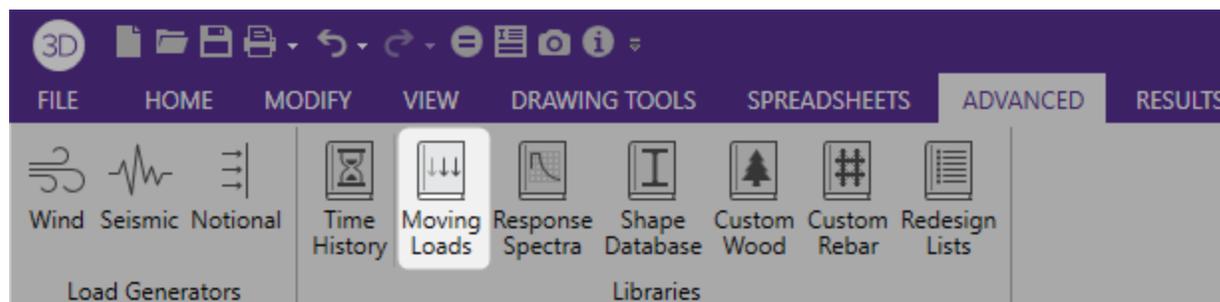
The **Both Ways** column is a check box that indicates whether the moving load pattern is to be applied in both directions of the load path or just one way along the load path. If the box is checked the load is first run from the start node all the way to the last node of the load path. The load is then 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 node 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 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:

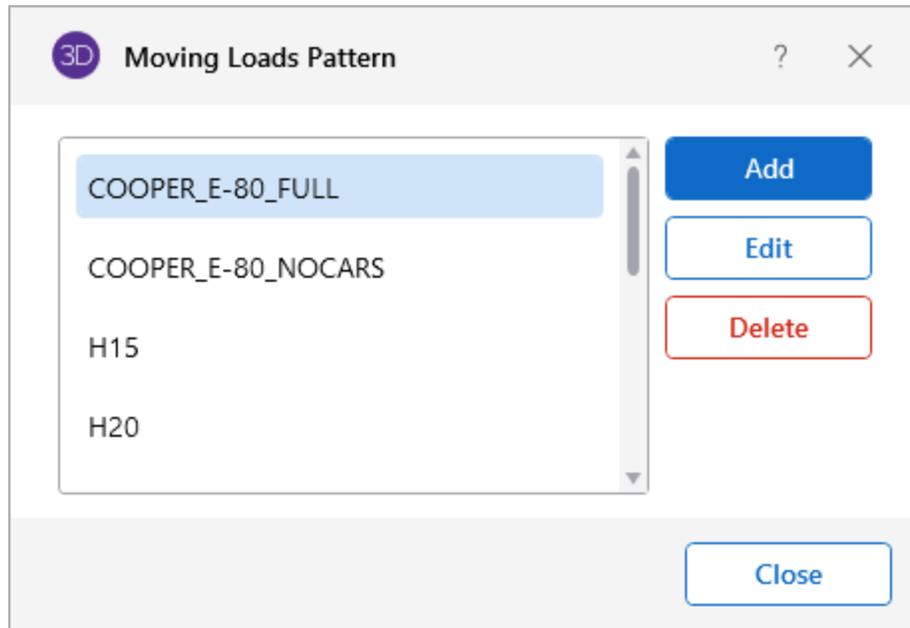


In the example moving load path shown, you would need to specify nodes A, B, C, and D as the load path nodes. You would not have to specify the joints that were in between the points where the load path changed direction; i.e., the moving load would automatically go in a straight line from node A to B, etc.

Moving Load Patterns

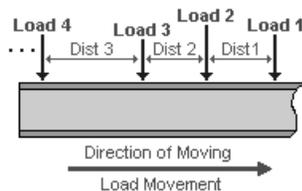


You can access the Moving Load Patterns and add or edit your own patterns by clicking on the **Moving Loads** icon on the 'Advanced' ribbon and then clicking on **Add** or **Edit**.



The file that the moving load pattern database is stored in is **ML_LIB32.FIL**. The path to this file is specified in the **Application Settings** in the **File Locations** tab. You can add up to 500 different moving load patterns in the pattern database.

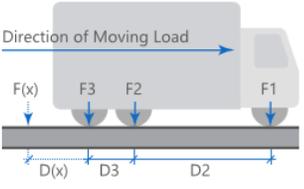
When you add a new pattern, the new pattern must have a unique name and can consist of up to 50 different loads. The sign of the load **Magnitude** will control which way the load is pointing in the direction specified in the **Direction** field. The direction can be any of the 3 global directions or the 3 local directions for the members that the load will travel over. Note that if your load travels over multiple members, a local direction load will be applied based on the local axes of each member it crosses over. There is also a special code, "V", which causes the load to be applied in the direction of the current vertical axis, whatever it is (X, Y, or Z). The **Distance** is the distance between the loads.



Note: If you have two loads at the same location but in different directions, you can input them one after the other with a Distance between them of 0 ft/m, as shown in the image below:

3D Edit Moving Load Pattern Definition
? X

Pattern Label



The direction value may be:

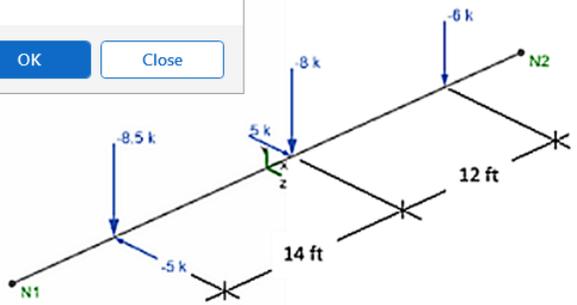
- X,Y,Z - Global Direction
- x,y,z - Local (Member) Direction
- V - Current Vertical Direction

Entries in the Distance column are distances between the loads. For example, distance D2 is the distance between load F1 and F2.

Value

	Magnitude [k]	Direction	Distance [ft]
1	-40	V	
2	-80	V	8
3	-80	V	5
4	-80	V	5
5	-80	V	5
6	-52	V	9
7	-52	V	5
8	-52	V	6
9	-52	V	5
10	-40	V	8

Clear Data
Print
OK
Close



Moving Loads Procedures

Moving loads are handled internally by applying the loads at discrete locations that are then moved through the model. A static solution is performed for the model at each load location. Typically, once the first solution is solved, the remaining loads are just solved against the existing stiffness matrix, so the stiffness matrix would not be rebuilt for each load position.



Note:

- Models that contain tension/compression only items will have their stiffness matrix rebuilt at least once at each load position. This can make the model solution take much longer than usual.
- Multiple moving loads may be assigned to a single load combination. However, they cannot be assigned different "start times". If a delay between moving loads is required, it must be accomplished by adjusting the moving load pattern.
- The load increment may be adjusted to "slow down" a moving load when multiple moving loads are applied.

Moving Loads Results

Load combinations that contain a moving load, will step the moving load through the load path and perform a solution for each position. The results are enveloped, giving maximum and minimum results of these solutions.

For these result spreadsheets, the maximum and minimum values are shown for each section location, for each active member. The governing load combination and step location is also shown for each result value under the "LC" column. The first number is the load combination, the second is the step number: (load combination - step number).

Envelope Node Reactions														
	Node Label		X [k]	LC	Y [k]	LC	Z [k]	LC	MX [k-ft]	LC	MY [k-ft]	LC	MZ [k-ft]	LC
1	N1	max	2.692	11-19	50.982	11-41	0	11-1	0	11-1	0	11-1	5.093	11-27
2		min	-0.942	11-118	-6.546	11-116	0	1	0	1	0	1	-14.115	11-84
3	N3	max	4.73	11-32	51.923	11-25	0	11-1	0	11-1	0	11-1	15.416	11-45
4		min	-2.669	11-125	0	1	0	1	0	1	0	1	-29.515	11-112
5	N5	max	0.504	11-48	51.583	11-80	0	11-1	0	11-1	0	11-1	20.011	11-87
6		min	-5.308	11-112	-1.641	11-124	0	1	0	1	0	1	-2.763	11-123
7	Totals:	max	0	11-84	64	11-23	0	11-1						
8		min	0	11-127	0	1	0	1						

Step Location

The step location tells you where the moving load was located along the moving load path when the maximum or minimum result value was obtained. This can help you recreate the static model to verify results.

The total number of steps is calculated as follows:

$$\text{Total Number of Steps} = \frac{\text{Total Path Length} + \text{Distance Between Loads} + \text{Increment}}{\text{Increment}}$$

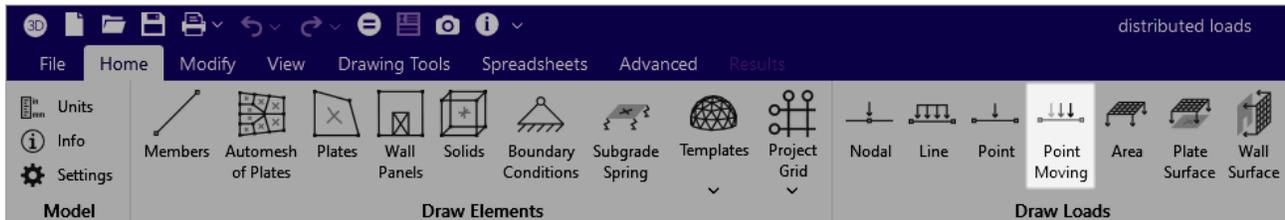


Note: The results for every load position are not stored; just the maximums and minimums. Therefore the program is unable at this time to give a detail report solution for the moving load results. However this is an enhancement that we do plan on adding in the future.

Moving Load Step Point Load Generation

Because solving moving loads produces so much output (full model design at each moving load step increment), the output is limited to Enveloped results. This means that the moving load results only show the maximum and minimum values for each output result value.

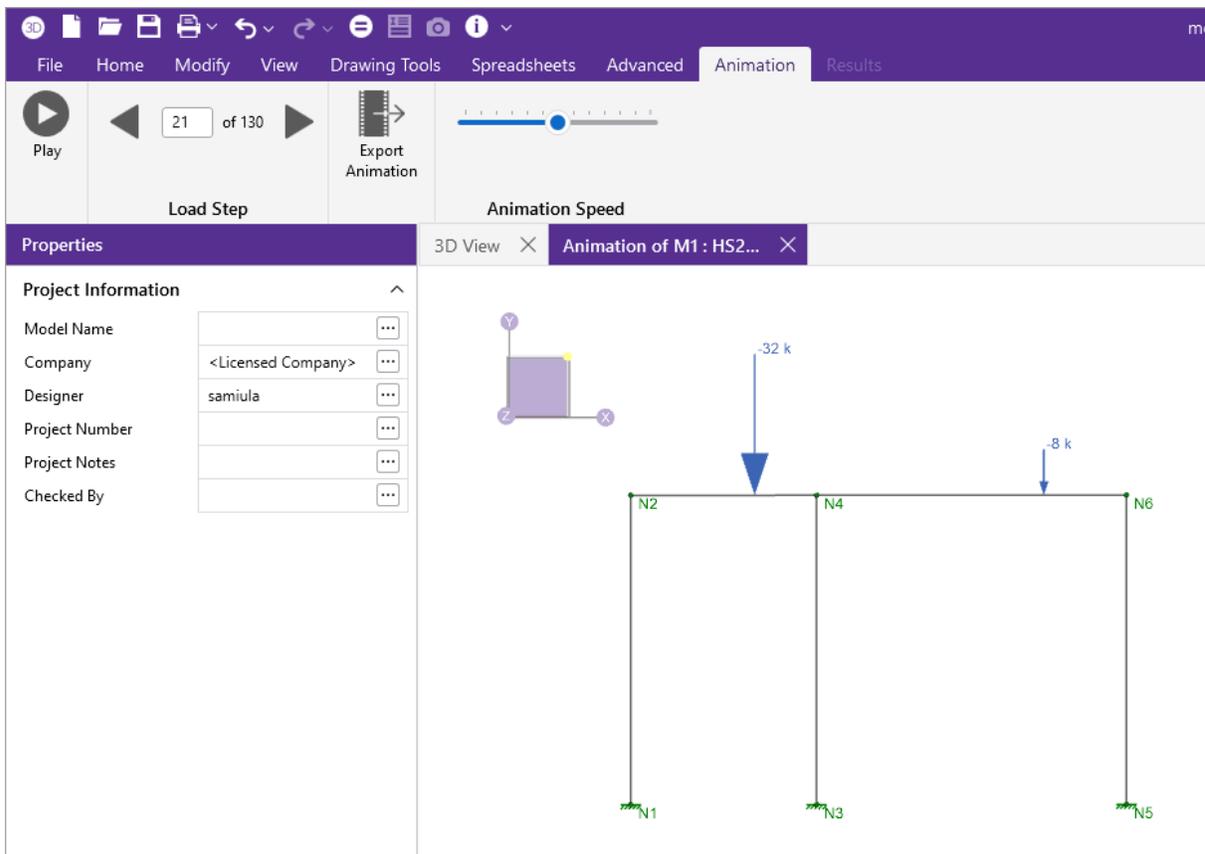
However there are times when you can want to investigate the full model design (all member results, detail reports, deflected shape, etc.) at a specific step. You can easily create a static load case of the moving loads at a specific load step using the **Point Loads from a Moving Load** feature. You can access this feature by clicking the **Point Moving** icon in the 'Home' ribbon.



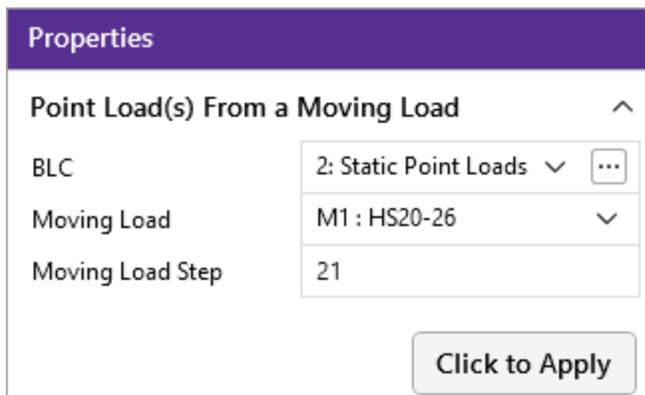
Instructions for Moving Load Step Point Load Generation

To move Load Step Point Load Generation:

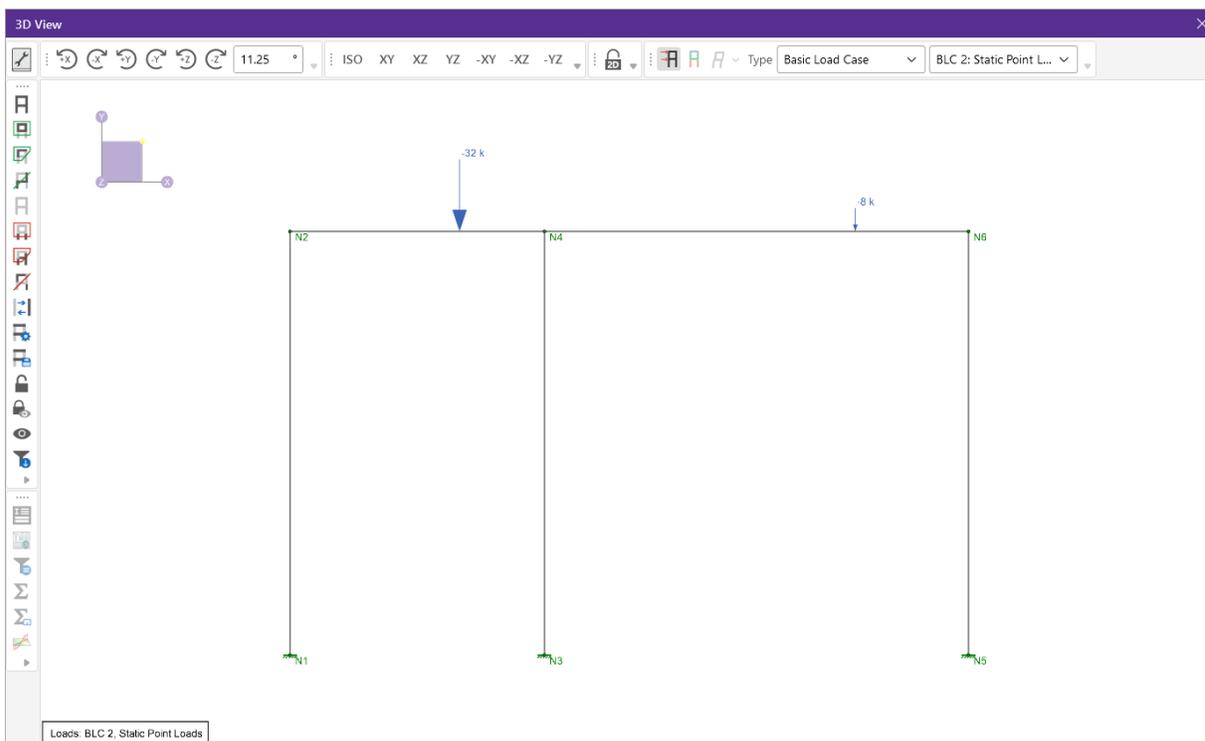
1. Determine which moving load step you want to investigate.



2. Access this feature by clicking the **Point Moving** icon in the 'Home' ribbon. The Point Load(s) From a Moving Load data appears in the Properties panel.



- a. Select which **Basic Load Case** you want the static point loads to be added to.
 - b. Select which **Moving Load** or Load Combination with a moving load you want the point loads generated from.
 - c. Select the **Moving Load Step** you want the static point loads generated from.
3. Click the **Click to Apply** button.



4. Create a Load Combination that includes the Basic Load Case selected in **step 2 a** above and solve to see the full results.

Loads - Thermal Loads

You can model the effects of temperature differentials in members and plates. For members, these loads cause the axial expansion or contraction of the member along its length, i.e. axial stress only. The temperature is assumed constant across the member's depth. For plates, these loads cause an in-plane expansion or contraction of the plate. The temperature is assumed constant through the thickness of the plate.

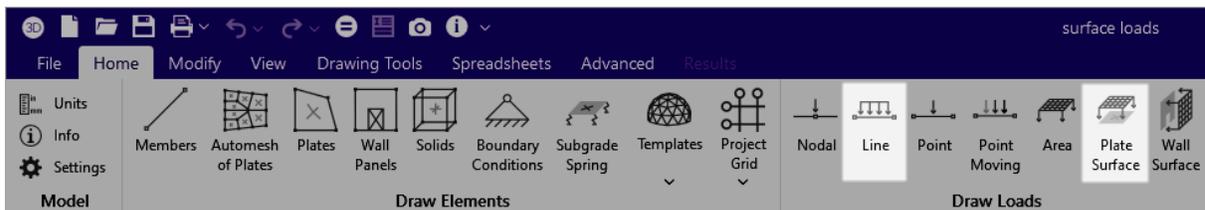


Note: The internal axial deflections for beam members are the average of the end deflections for thermal loads.

Apply a Thermal Load

To apply a Thermal Load:

1. Select the members you wish to assign a thermal load to.
2. Go to the **Home** ribbon.



3. Click the **Line** icon or the **Plate Surface Load** icon in the 'Draw Loads' section.
4. Define a distributed load with a direction of T and a magnitude, in temperature.



Note:

- You may also specify or edit thermal loads in the **Distributed Loads Spreadsheet** or the **Plate Surface Loads Spreadsheet**.
- An easy way to do thermal loadings is to define the joint temperatures (on the Joint Coordinates Spreadsheet) as all zero, so any defined thermal loads are the full stress inducing temperatures.

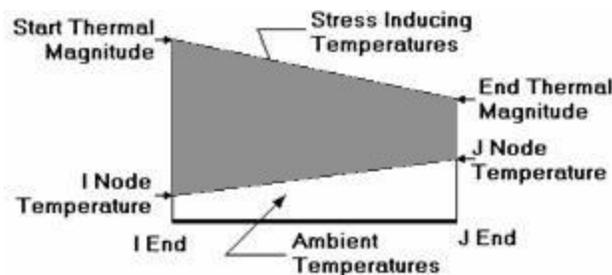
Record Thermal Loads for Members

The Coefficient of Thermal Expansion (α) is entered on the **Materials** spreadsheet. Note that this value is entered per 100,000 degrees (it is sometimes listed per 1,000 degrees).

The joint temperatures recorded on the **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.

Member Distributed Loads							
Member Wall Panel							
BLC 3: Pretension							
Prev BLC Next BLC							
	Member Label	Direction	Start Magnitude [k/ft, F, ksf, k-ft/ft]	End Magnitude [k/ft, F, ksf, k-ft/ft]	Start Location [(ft, %)]	End Location [(ft, %)]	Inactive
1	M2	T	90	90	0	%100	
2	M1	T	90	90	0	%100	
3	M3	T	90	90	0	%100	
4	M4	T	90	90	0	%100	
5	M5	T	90	90	0	%100	
6	M6	T	90	90	0	%100	

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.

Plate Surface Loads				
Plate Wall Panel				
BLC 3: Pretension				
Prev BLC Next BLC				
	Plate Label	Direction	Magnitude [ksf, F]	Inactive
1	P9	T	200	
2	P5	T	200	
3	P20	T	200	
4	P2	T	200	

Thermal Force Calculation

The joint temperatures recorded on the **Node Coordinates** Spreadsheet define the ambient thermal state of the structure. The joint temperature at the I-end of the member is interpolated across to the J end temperature to define the ambient state of the member. Thermal loads, entered as distributed loads on the **Distributed Loads** spreadsheet, induce axial stress in the member. The difference between the applied thermal load and the ambient temperature is the stress inducing temperature.

Thermal forces are calculated as follows:

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

Where,

F_t = Calculated Thermal Force

A = Member Cross Sectional Area

E = Elastic Modulus

α = Coefficient of Thermal Expansion

ΔT = Stress Inducing Temperature

Prestressing with Thermal Loads

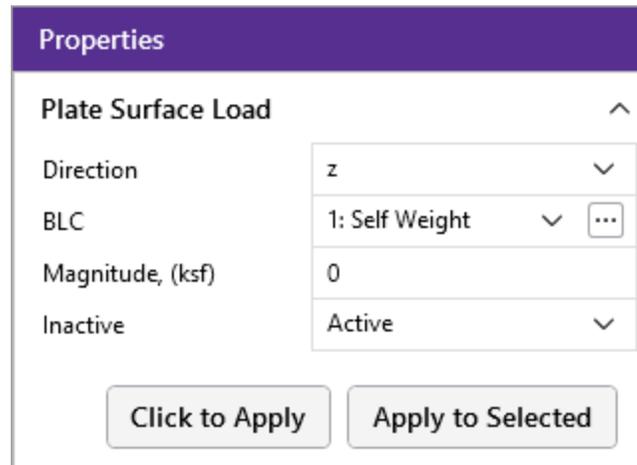
Thermal loads provide a way to introduce pre-stressing in a model. Given a desired prestress force, just back-solve the thermal force equation for the needed ΔT . Remember, as the model expands (or contracts), the prestress force may be altered.

Loads - Surface Loads

Surface loads are loads that are spread out over the surface of a plate element or wall panel. RISA-3D allows surface loads directed along the global axes, the local axes, or projected in the direction of the global axes. Loads may be input manually or assigned graphically. See Drawing Surface Loads below to learn how to draw surface loads graphically.

Draw Plate Surface Loads

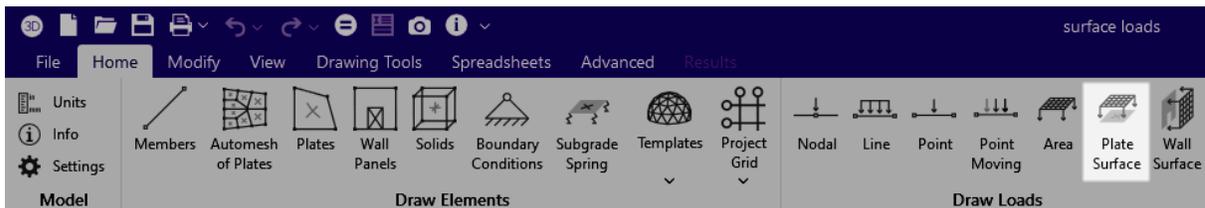
To apply surface loads to plates enter the load direction and magnitude. Make sure that you are careful to enter the correct BLC number that you want the loads assigned to. See Surface Loads above for more information on surface loads.



Apply Plate Surface Loads

To apply Plate Surface Loads:

1. Go to the **Home** ribbon.



2. Click on the **Plate Surface** icon to bring up the plate surface load information in the 'Properties Panel'.
3. Define the load in the Plate Surface Properties Panel. Use the dropdown menu to select load Direction, and BLC.

For help on an item, click the **?** Help icon and then click the item.

4. You can apply the load by choosing plates on the fly or apply it to a selection of plates.
 - To apply the load to a few plates choose **Click to Apply** and click on the plates with the left mouse button.
 - To apply the load to a selection, choose **Apply to Selected**.

If nothing is selected, **Apply to Selected** will assume the full model is selected and apply changes to all plates.

**Note:**

- You can also specify or edit surface loads in the **Surface Loads** Spreadsheet.
- Applied surface loads are converted to equivalent corner joint loads based on the plate area tributary to each corner joint.
- You can undo any mistakes by clicking the  **Undo** button on the Quick Access toolbar.

Draw Wall Panel Surface Loads

To apply surface loads to wall panels enter the load direction, start location (from bottom), height, and magnitudes (top and bottom). Make sure that you are careful to enter the correct BLC number that you want the loads assigned to. See Surface Loads above for more information on surface loads.

Properties

Wall Panel Load ^

Direction	z	v
BLC	1: Self Weight	v ...
Load Distribution	Uniform	v
Magnitude, (ksf)	0	
Load Height	Entire Wall	v
Inactive	Active	v

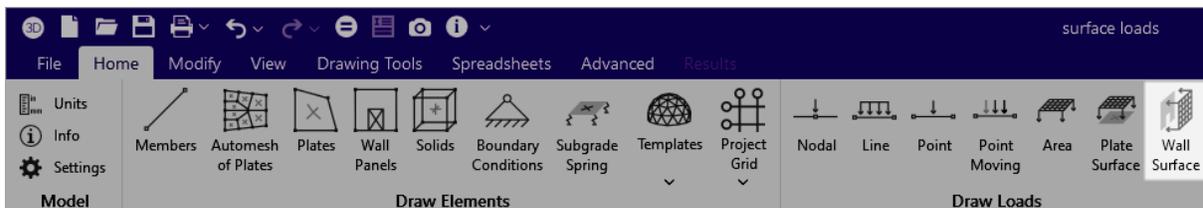
Click to Apply

Apply to Selected

Apply Wall Panel Surface Loads

To apply Wall Panel Surface Loads:

- Go to the **Home** ribbon.



2. Click on the **Wall Surface** icon to bring up the wall surface load information in the 'Properties Panel'.
3. Define the load in the Wall Surface Properties Panel. Use the dropdown menu to select load Direction, BLC, load distribution (uniform or tapered), and load height (entire wall or partial).

For help on an item, click the  **Help** icon and then click the item.

4. You can apply the load by choosing wall panels on the fly or apply it to a selection of wall panels.
 - To apply the load to a few wall panels choose **Click to Apply** and click on the wall panels with the left mouse button.
 - To apply the load to a selection, choose **Apply to Selected**. If nothing is selected, **Apply to Selected** will assume the full model is selected and apply changes to all wall panels.
 - To apply tapered wall load, choose **Tapered** from **Load Distribution** dropdown menu, and specify Top Magnitude and Bottom Magnitude.
 - To apply partial height wall load, choose **Partial** from **Load Height** dropdown menu, and specify load Start Location and Height.

**Note:**

- You can also specify or edit surface loads in the **Surface Loads** Spreadsheet.
- You can undo any mistakes by clicking the  **Undo** button on the Quick Access toolbar.

Surface Loads Spreadsheet

The **Surface Loads Spreadsheet** records the surface loads for the plate/shell elements and wall panels, and may be accessed by selecting select **Surface Loads** from the **Data Entry** menu in the **Explorer panel**. Alternatively, it can be accessed by going to the **Spreadsheets Tab**  **Data Entry** .

Plate Surface Loads				
Plate Wall Panel				
BLC 2: Superimposed De... ▾				
		Prev BLC	Next BLC	
	Plate Label	Direction	Magnitude [ksf, F]	Inactive
1	P2	z	-0.01	
2	P3	z	-0.01	
3	P4	z	-0.01	
4	P5	z	-0.01	
5	P6	z	-0.01	
6	P7	z	-0.01	

Wall Panel Surface Loads							
Plate Wall Panel							
BLC 2: Superimposed De... ▾							
		Prev BLC	Next BLC				
	Wall Panel Label	Direction	Top Magnitude [ksf, F]	Bottom Magnitude [ksf, F]	Start Location [ft]	Height [ft]	Inactive
1	WP3	z	0.05	0.05	0	0	
2	WP2	z	0.05	0.05	0	0	
3	WP1	z	0.05	0.05	0	0	

When you open this spreadsheet you can view only one basic load case at a time. Use the drop down list on the toolbar to specify a different load case. The current load case is also displayed in the title bar at the top of the spreadsheet.

Under the **Plate** tab the first column contains the label of the plate to be loaded. The second column defines the direction of the load. Direction options are discussed in the next section. The third column holds the magnitude of the load.

Under the **Wall Panel** tab the first column contains the label of the wall panel to be loaded. The second column defines the direction of the load. Direction options are discussed in the next section. The third and fourth columns hold the magnitude of the load. The fifth and sixth columns define the Start Location (bottom) and Height of the load.

For loads that apply to the full height of the wall, leave the start location and height as zero.

The last column allows you to set the specified load as an "Active" load that will be considered for the analysis, or an "Inactive" load that will not be included in the analysis.

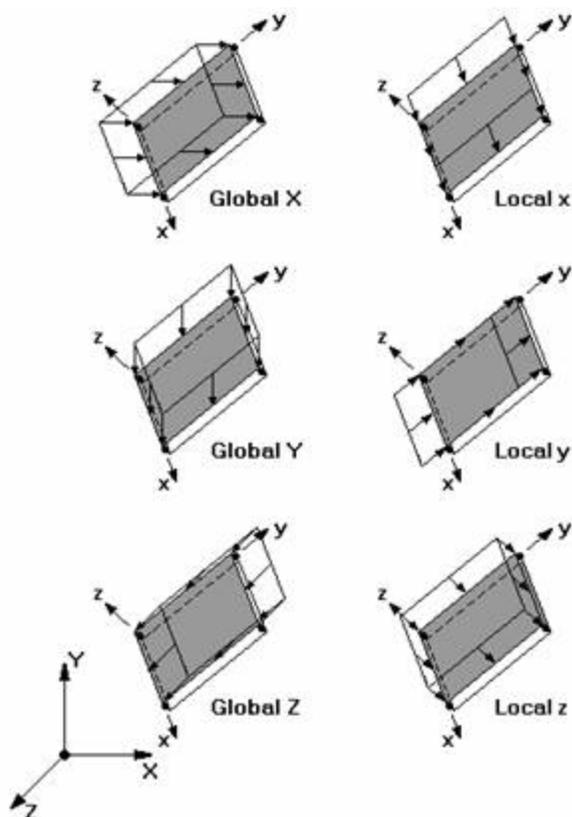
For models integrated from RISAFloor, the **Diaphragm Tab** records the surface loads applied on the semi-rigid diaphragm. These loads are automatically generated during the integration from RISAFloor to RISA-3D. These lines will be grayed out because they cannot be edited from RISA-3D since they are tied to the RISAFloor analysis. If you detach the model from RISAFloor in RISA-3D, these loads will not carry over since semi-rigid diaphragms are a feature only available for models integrated with RISAFloor.

Surface Load Directions

The direction code indicates the direction of application for the surface load. The following directions are available:

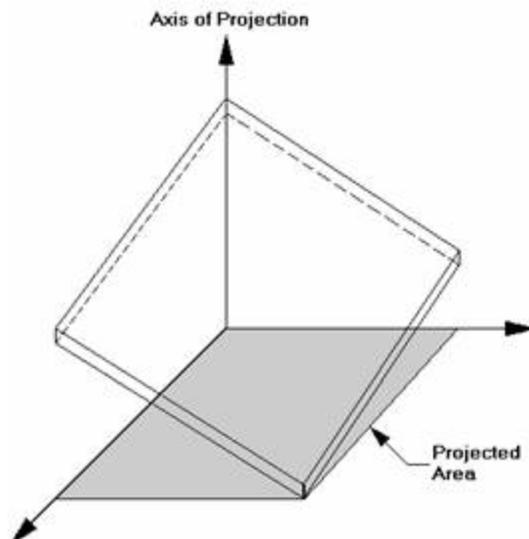
Direction Code	Applied Direction
x, y, or z	Element's local x, y, or z direction.
X, Y, or Z	Global X, Y, or Z direction
PY	Projected in the direction of the global Y axis
PX	Projected in the direction of the global X axis
PZ	Projected in the direction of the global Z axis
T	Thermal (temperature differential) load

The following diagram illustrates the difference between local (x, y, z) and global (X, Y, Z) direction loads. The local direction loads line up with the element's local axis directions, so their direction relative to the rest of the model changes if the element orientation changes. Global loads have the same direction regardless of the member's orientation. Keep in mind that global loads are applied without being modified for projection. For example, a global Y direction load of 1 kip/sq.ft. applied to an element with an area of 10 sq.ft., which is inclined at 45 degrees, generates a total force of 10 kips.



Projected loads, on the other hand, are applied in the global directions, but their actual magnitude is influenced by the element orientation. The load is applied to the projected area of the element that is perpendicular to the load.

For example, a "V" direction load is a projected load applied in the global Y direction. The actual magnitude of the load is the entered magnitude reduced by the ratio A/A_{xz} . "A" is the actual area of the element and A_{xz} is the element's projected area on the X-Z plane, which is always less than or equal to the actual area. See the following figure:

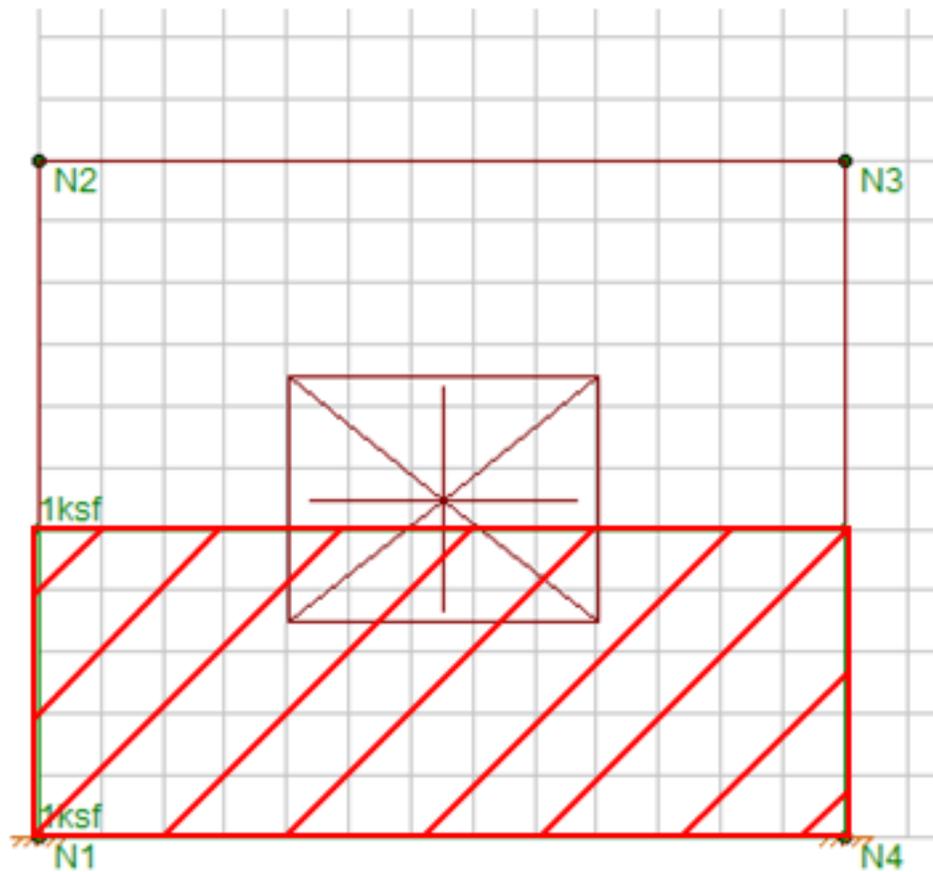


If the "Axis of Projection" in the figure is the Y-axis, then the shaded area is the total element area "projected" onto the plane perpendicular to the Y-axis (which happens to be the X-Z plane). The total load generated is equal to the input magnitude applied to the projected area. The generated load is then applied to the whole area, so the generated load magnitude is reduced accordingly.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Area Surface Loads**.

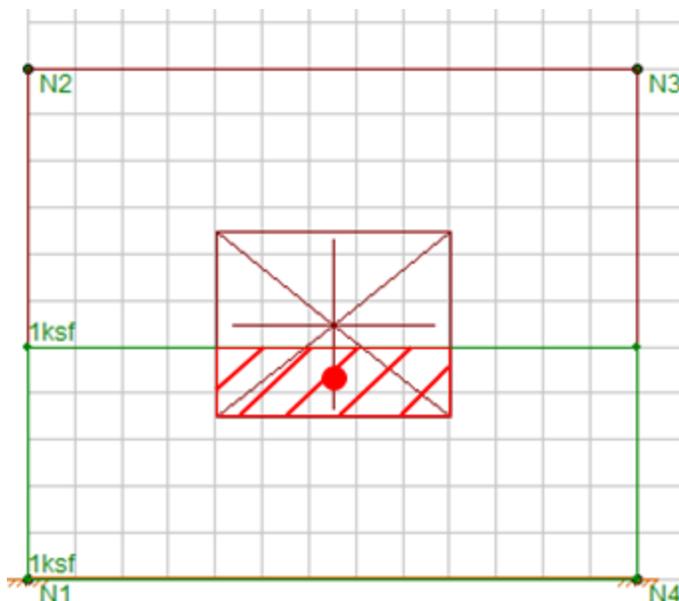
Surface Loads at Openings (Wall Panels)

When a surface load is applied to a wall panel with openings it is converted into equivalent nodal loads around the openings. The example below is a wall panel with an opening, and a surface load applied to the lower portion of the wall.

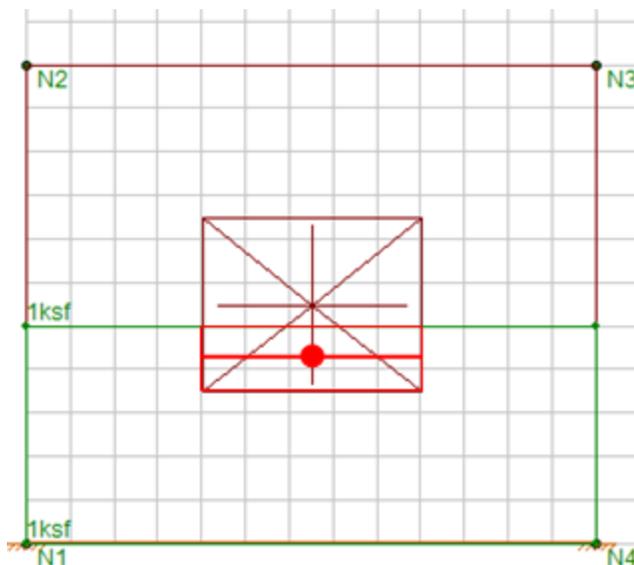


The program calculates transient nodal loads according to the following approximation procedure:

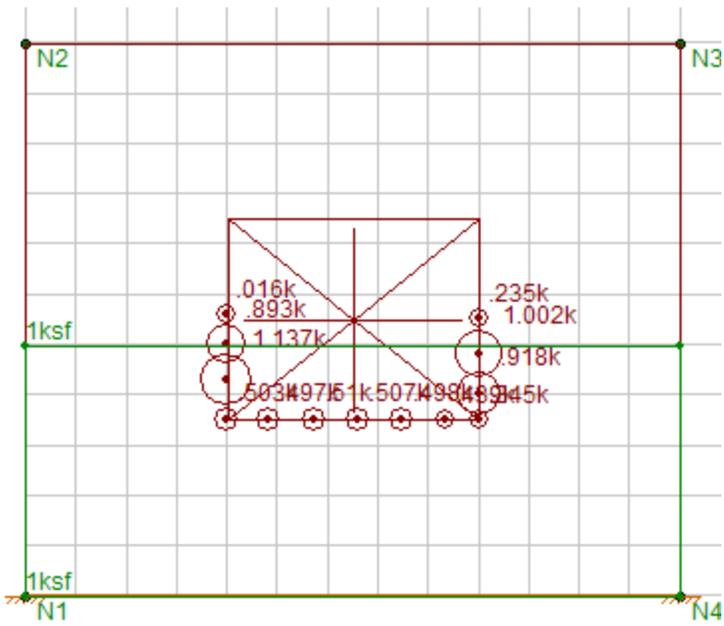
1. The centroid of the union of the surface load and the opening is determined.



2. Using the centroid location, the surface load within the opening region is split into two parts.



3. Each of the two parts the surface load are summed up and smeared as point loads along the edge nodes of the opening. The forces and moments are conserved with this approximation method. These generated nodal loads are transient, and will be deleted when the solution result is deleted. These loads are not visible by default, however if you would like to view them you can select **Include Transient WP Loads** under [Model Display Options](#). Below is an example:



Load Generation - Notional Loads

Notional Loads are used by some building codes for the stability design of a structure. They serve as a minimum lateral load, or as an alternative to modeling the actual out-of-plumbness or out-of-straightness of the structure. Instead of changing the geometry of the structure, an equivalent destabilizing load is added to the structure. There are numerical benefits to handling this out-of-plumbness issue with loads rather than geometry. Essentially, it is quicker and easier to adjust the loading on a structure than it is to modify the stiffness matrix of the structure.

The implementation of these notional loads is not based on a single code, but on the concept of using lateral forces equal to a percentage of the applied vertical load at each floor level. Codes that may require the use of notional loads include the following:

- ASCE 7: A minimum lateral load of 1% of the Dead Load of the structure should be applied at each floor as a notional load.
- AISC 360: A notional load to account for out-of-plumbness of the structure of 0.2% to 0.3% of the total gravity load (DL + LL) should be applied at each floor as a notional load.
- AS 4100: Has a default of notional load of 0.2%
- NZS 3404: Has a default notional load of 0.2%
- BS 5950: Has a default notional load equal to 0.5%
- EC 1993-1-1: Has a notional load that can vary, but which will not normally exceed 0.5% of the applied vertical load

These notional loads are normally only assumed to act for load cases which do not include other lateral forces. However, the specific requirements of the individual code may require the use of these loads for other load cases depending on the sensitivity of the structure to stability effects.

Notional loads can only be automatically generated for diaphragm/floor levels. The program will automatically calculate the center of mass and use that point as the location to apply the Notional Loads.



Note:

- The Notional Loads generated by the program are calculated for Building Structures ONLY and may not apply to non-building structures, horizontal trusses, towers, or other specialty structures.
- Notional Loads are currently only supported for rigid or flexible diaphragms. Semi-rigid diaphragms are not currently supported.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Notional Loads**.

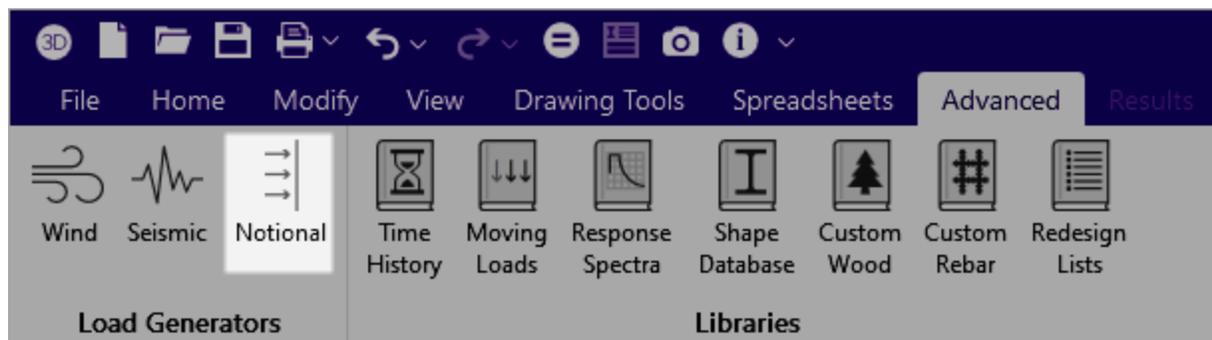
Vertical Load Used for Notional Force Calculations

The **Weight** of each diaphragm is the total tributary weight associated with it depending on the Load Combination chosen for calculating these forces in the Notional Loads window.

While computing the tributary weight at a particular diaphragm, the self weight of the members/columns and plates between any two diaphragms is equally distributed amongst these diaphragms. Any weight, or load included in the specified load combination, supported between diaphragms is distributed to the diaphragm above and below in inverse proportion to its distance from each diaphragm.

Notional Load Generation Window

The parameters used in the seismic calculations may be viewed or changed by selecting **Notional** from the Load Generators section of the **Advanced** ribbon.



The notional load window only asks for a load combination and the % of gravity loads that should be applied in the lateral direction. Notional loads are only generated for a single load combination. Therefore, you are encouraged to choose a load combination which includes the most severe loading.

3D Notional Load Generator
? X

Seismic Weight LC: 1: IBC 16-1 v

Notional Load Parameters

Percentage (%) g Z Direction: 1

Percentage (%) g X Direction: 1

Calculate Loads

Generate Loads starting at BLC: BLC: 1: Self Weight v

Notional Load Generation Input

Percentage of gravity in Z Direction: 1 %

Percentage of gravity in X Direction: 1 %

Load combination used for Story Weights: 1: IBC 16-1

Notional Load Generation Force Results

Floor Level	Height (ft)	Weight (kips)	Force Z (kips)	Force X (kips)	CG Z (ft)	CG X (ft)
Diaphragm 1	24	638.219	6.382	6.382	10.054	11.308
Diaphragm 2	12	318.256	3.183	3.183	17.928	23.774

Print
 Add to Full Report

Generate
Close

Notional Load Results

The program will only calculate the notional loads when you select **Notional** from the **Advanced** tab and clicks **Generate** to create the **X & Z Notional Basic Load Cases**.

Notional Load Generation Input

This section displays all the relevant design data entered so that it can be included on a print out with the Notional Load results.

Notional Generation Detail Results

This section displays all important calculated information such as the **Weight** associated for each floor level, the Center of Gravity location for this weight and the notional load applied to each diaphragm or floor level. To add the detailed results to the final print out, click the check box for **Add to Full Report** found in the **Notional Load Generation Input**.



Note: The Joints / Nodes used to apply seismic load may appear as a "floating joint" at that diaphragm level. While these nodes may not be attached to any beams or framing, they are attached to the diaphragm at that floor level.

Load Generation - Notional Loads

Notional Loads are used by some building codes for the stability design of a structure. They serve as a minimum lateral load, or as an alternative to modeling the actual out-of-plumbness or out-of-straightness of the structure. Instead of changing the geometry of the structure, an equivalent destabilizing load is added to the structure. There are numerical benefits to handling this out-of-plumbness issue with loads rather than geometry. Essentially, it is quicker and easier to adjust the loading on a structure than it is to modify the stiffness matrix of the structure.

The implementation of these notional loads is not based on a single code, but on the concept of using lateral forces equal to a percentage of the applied vertical load at each floor level. Codes that may require the use of notional loads include the following:

- ASCE 7: A minimum lateral load of 1% of the Dead Load of the structure should be applied at each floor as a notional load.
- AISC 360: A notional load to account for out-of-plumbness of the structure of 0.2% to 0.3% of the total gravity load (DL + LL) should be applied at each floor as a notional load.
- AS 4100: Has a default of notional load of 0.2%
- NZS 3404: Has a default notional load of 0.2%
- BS 5950: Has a default notional load equal to 0.5%
- EC 1993-1-1: Has a notional load that can vary, but which will not normally exceed 0.5% of the applied vertical load

These notional loads are normally only assumed to act for load cases which do not include other lateral forces. However, the specific requirements of the individual code may require the use of these loads for other load cases depending on the sensitivity of the structure to stability effects.

Notional loads can only be automatically generated for diaphragm/floor levels. The program will automatically calculate the center of mass and use that point as the location to apply the Notional Loads.

**Note:**

- The Notional Loads generated by the program are calculated for Building Structures ONLY and may not apply to non-building structures, horizontal trusses, towers, or other specialty structures.
- Notional Loads are currently only supported for rigid or flexible diaphragms. Semi-rigid diaphragms are not currently supported.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Notional Loads**.

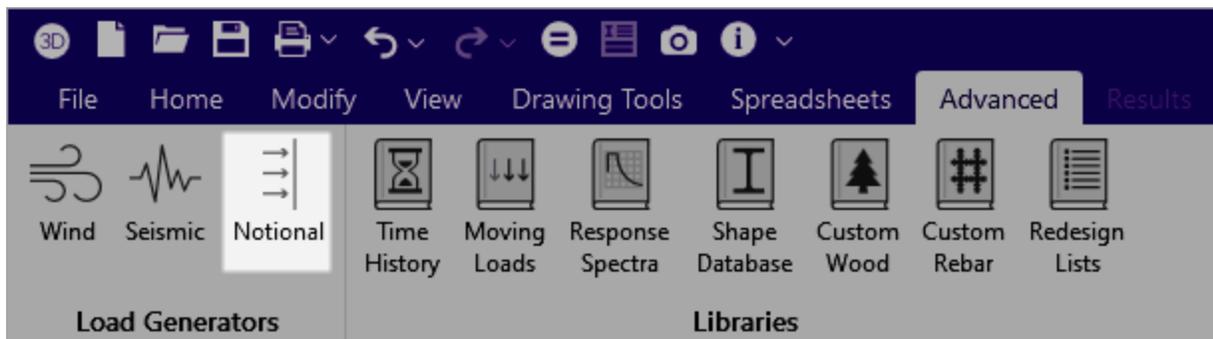
Vertical Load Used for Notional Force Calculations

The **Weight** of each diaphragm is the total tributary weight associated with it depending on the Load Combination chosen for calculating these forces in the Notional Loads window.

While computing the tributary weight at a particular diaphragm, the self weight of the members/columns and plates between any two diaphragms is equally distributed amongst these diaphragms. Any weight, or load included in the specified load combination, supported between diaphragms is distributed to the diaphragm above and below in inverse proportion to its distance from each diaphragm.

Notional Load Generation Window

The parameters used in the seismic calculations may be viewed or changed by selecting **Notional** from the Load Generators section of the **Advanced** ribbon.



The notional load window only asks for a load combination and the % of gravity loads that should be applied in the lateral direction. Notional loads are only generated for a single load combination. Therefore, you are encouraged to choose a load combination which includes the most severe loading.

Notional Load Generator
? X

Seismic Weight LC: 1: IBC 16-1 v

Notional Load Parameters

Percentage (%) g Z Direction: 1

Percentage (%) g X Direction: 1

Calculate Loads

Generate Loads starting at BLC: BLC 1: Self Weight v

Notional Load Generation Input

Percentage of gravity in Z Direction: 1 %

Percentage of gravity in X Direction: 1 %

Load combination used for Story Weights: 1: IBC 16-1

Notional Load Generation Force Results

Floor Level	Height (ft)	Weight (kips)	Force Z (kips)	Force X (kips)	CG Z (ft)	CG X (ft)
Diaphragm 1	24	638.219	6.382	6.382	10.054	11.308
Diaphragm 2	12	318.256	3.183	3.183	17.928	23.774

Print
 Add to Full Report

Generate
Close

Notional Load Results

The program will only calculate the notional loads when you select **Notional** from the **Advanced** tab and clicks **Generate** to create the **X & Z Notional Basic Load Cases**.

Notional Load Generation Input

This section displays all the relevant design data entered so that it can be included on a print out with the Notional Load results.

Notional Generation Detail Results

This section displays all important calculated information such as the **Weight** associated for each floor level, the Center of Gravity location for this weight and the notional load applied to each diaphragm or floor level. To add the detailed results to the final print out, click the check box for **Add to Full Report** found in the **Notional Load Generation Input**.



Note: The Joints / Nodes used to apply seismic load may appear as a "floating joint" at that diaphragm level. While these nodes may not be attached to any beams or framing, they are attached to the diaphragm at that floor level.

Load Generation - Seismic Loads

Building Seismic Loads can be automatically generated according to the equivalent static methods of the following codes:

- ASCE 7-2016 / IBC 2015 / IBC 2018
- ASCE 7-2010 / IBC 2012
- ASCE 7-2005 / IBC 2006 / IBC 2009
- ASCE 7-2002 / IBC 2003
- IBC 2000
- 1997 UBC
- 2001 CBC (California Amended UBC)
- Mexican NTC-04
- Indian IS 1893:02
- Canadian NBC-2015
- Canadian NBC-2010
- Canadian NBC-2005

Seismic load can only be applied at diaphragm/floor levels. The program automatically calculates the center of mass and the 5% accidental eccentricity for the various seismic load cases.

**Note:**

- The Seismic Loads generated by the program are calculated for Building Structures ONLY and may not apply to non-building structures.
- The Seismic Loads generated by the program consider accidental eccentric loading as well. If you plan on using the [Load Combination Generator](#) in the [Load Combinations](#) spreadsheet, you must use the "X and Z w/Eccentric" Seismic Load option. Otherwise the eccentric BLCs that have been generated will never actually be applied.

Seismic Weight

The **Seismic Weight** of each diaphragm is the total tributary weight associated with it depending on the Load Combination chosen for calculating these forces on the Seismic Loads dialog box.

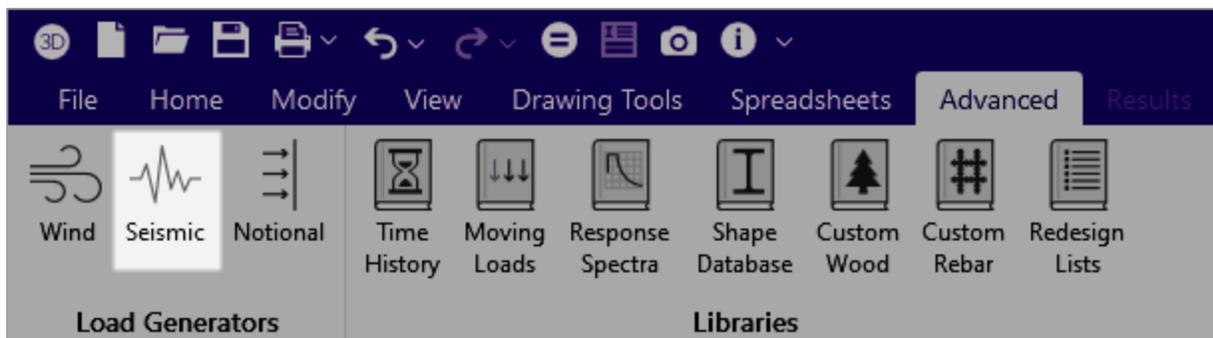
While computing the seismic weight at a particular diaphragm, the self weight of the members/columns and plates between any two diaphragms is equally distributed amongst these diaphragms. Any weight, or load included in the specified load combination, supported between diaphragms is distributed to the diaphragm above and below in inverse proportion to its distance from each diaphragm.

The total seismic weight of the whole structure is the sum of the seismic weights associated with all diaphragms and the weight associated with the base level. The base shear is always computed using the total seismic weight. The total seismic weight can be viewed using the Scaling Factor Dialog. To get here click the [RSA Scaling Factor](#) button in the [Load Combinations](#) spreadsheet.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Diaphragms**.

Seismic Design Parameters

The parameters used in the seismic calculations may be viewed or changed by selecting **Seismic** from the Load Generators section of the **Advanced** ribbon.



In RISA-3D, the weight used for the calculation of seismic loads is based solely upon the Load Combination specified as the Seismic Weight LC entered in the Seismic Loads Dialog shown below:

Seismic Load Generator
?
✕

Seismic Weight LC LC 1: IBC 16-1

RISA-3D Seismic Load Options
 Code ASCE 7-16
 Risk Category I or II
 Base Elevation 0 ft
 Include the weight of the structure in base shear calculations

Site Parameters
 S_1 0.566 g SD_s 1 g
 SD_1 0.566 g T_L 8 sec

Structure Characteristics

	Z Direction	X Direction
T	35.5	46.4
C_1	0.028	0.028
C_1 Exp.	0	0
R	3	3
Ω_0	1	1
C_d	4	4
ρ	1	1

Calculate Loads

Generate Loads starting at BLC BLC 6:

Print

Add to Full Report

Generate

Close

Seismic Generation Input
 Seismic Code ASCE 7-16 S_{D1} 0.566 g
 Risk Category I or II S_{D5} 1 g
 Seismic Weight LC LC 1: IBC 16-1 S_1 0.566 g
 Base Elevation 0 ft T_L 8 sec
X-Direction Parameters
 C_{Tx} 0.028 R_x 3 $C_{Tx,Exp}$ 0 T_x 46.4 sec
Z-Direction Parameters
 C_{Tz} 0.028 R_z 3 $C_{Tz,Exp}$ 0 T_z 35.5 sec

Seismic Generation Detail Results
 Importance Factor 1 Design Category D
Period Determination
 X - Direction T_{aX} 0.028 sec T_{XLIMIT} 0.039 sec T_x 46.4 sec
 Z - Direction T_{aZ} 0.028 sec T_{ZLIMIT} 0.039 sec T_z 35.5 sec
Base Shear Determination
 X - Direction C_{SX} 0.044 sec V_x 42.085 kips ASCE Eqn 12.8-5
 Z - Direction C_{SZ} 0.044 sec V_z 42.085 kips ASCE Eqn 12.8-5

Seismic Generation Force Results

Floor Level	Height (ft)	Weight (kips)	Force Z (kips)	Force X (kips)	CG Z (ft)	CG X (ft)
Diaphragm 1	24	638.219	37.42	37.42	10.054	11.308
Diaphragm 2	12	318.256	4.665	4.665	17.928	23.774

Seismic Generation Diaphragm Results

Floor Level	Width (Z) (ft)	Length (X) (ft)	Z Plus (ft)	Z Minus (ft)	X Plus (ft)	X Minus (ft)
Diaphragm 1	36	54	1.8	1.8	2.7	2.7
Diaphragm 2	36	54	1.8	1.8	2.7	2.7

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Generate Seismic**.

Seismic Load Parameters

Code currently allows you to choose the code to be used for seismic load generation. For reference, sections of the 2016 edition of ASCE 7 are cited to explain the various entries.

T represents the input natural periods in each lateral direction. These would typically be determined from an eigensolution analysis. If these values are not entered, then the program calculates this using the Approximate Fundamental Period as defined in section 12.8.2.1 of ASCE 7-16. This value is entered for each of the Global horizontal directions.

Page 721 of 1518

C_t is the building period coefficient as defined in 12.8.2.1 of ASCE 7-16. It is used in conjunction with the C_t exponent "x" to determine the Approximate Fundamental Period. These are defined for each of the Global horizontal directions.



Note: You can either input the period manually in the Global horizontal directions, or you can input C_t and the C_t exponent "x" and the program uses Eq. 12.8-7 of the ASCE 7-16 for calculation of the period.

R is the Response Modification Factor as defined in table 12.14-1 of ASCE 7-16. It provides a reduction for the design seismic force based on the ductility of the system. This is defined for each of the Global horizontal directions.



Note: The program offers a single R value input in each direction. There are situations where the lower portion of the structure may have a different R value than the upper portion. This can be typical with a concrete pedestal supporting a wood structure. In this case, two sets of load combinations would be created, one set in which only the Wood design check-box would be checked in the Load Combinations spreadsheet and one set in which only the Concrete design check-box is checked. In this case, the R value for the concrete pedestal would be input in the Seismic Load generator. Then, in the Load Combinations spreadsheet, the wood load combinations would have their seismic load factors factored by the ratio of the wood R value over the concrete R value. In this way, the two R values can be taken into account in the same direction.

Base Elevation determines the height at which the structure is assumed to be connected to the ground. This is important for hillside structures or structures with sub-grade floor levels. A certain amount of structure self weight may be associated with base level (or sub-grade levels) of the structure. The **Add Base Weight** check-box determines if that self weight will be added into the base shear to be distributed as lateral force through the height of the structure per section 12.8.3 of ASCE 7-16. If no elevation is chosen for base elevation, then the lowest joint in the structure is assumed to be the base elevation.

Risk Category is used to determine the importance factor assigned to the structure per table 1.5-2 of ASCE 7-16.

SD_1 represents the 5% damped spectral response Design acceleration for a 1.0 second period.

SD_S represents the 5% damped spectral response Design acceleration for short period response.

S_1 represents the 5% damped spectral response Mapped acceleration for a 1.0 second period.

T_L represents the point at which the structural response is assumed to transition from a velocity controlled response to a displacement controlled response. These values are shown on Figures 22-14 through 22-17 in the ASCE 7-16.

Seismic Weight LC is used to dictate which load combination should be used to define the weight of the structure when the seismic event is assumed to occur. In ASCE 7-16 this would be based on the criteria in section 12.7.2.

Seismic Load Results

When you activate RISA-3D via the Director menu, the program calculates the appropriate seismic loads and present the calculations in a printable report. You can open the seismic load generator at any time to view, print, or recalculate the seismic loads.

Seismic Generation Input

This section displays all the relevant design data entered so that it can be included on print outs with the Seismic Load results.

Seismic Generation Detail Results

This section reports the values used to obtain the Base Shear in each of the two global directions.

T_x and T_z are the periods which was actually used to determine an upper limit for C_s in equations 12.8-3 and 12.8-4 of ASCE 7-16 (if applicable). A user defined period can exceed the upper limit period.

T_a is the approximate period calculated per equation 12.8-7 of ASCE 7-16.

T_{Limit} is the maximum allowable period to be used in equations 12.8-3 and 12.8-4, calculated per Table 12.8-1 of ASCE 7-16.

Importance Factor is determined from Table 1.5-2 of ASCE 7-16, based on the specified [Risk Category](#).

Design Category is determined in Section 11.6 of the ASCE 7-16 and reported here.

V (Base Shear) is calculated using the Governing Equation listed next to it.

Governing Equation is the equation which was used to calculate the base shear. This is typically from 12.8 of ASCE 7-16.

C_s is the seismic response coefficient used to calculate the seismic base shear, V .

Seismic Generation Force Results

This section displays information used in distributing the seismic force to each diaphragm or story level. This includes the calculated **Height** and **Weight** of each diaphragm, the calculated **Force** in each horizontal direction and the calculated location of the Center of Gravity of the diaphragm (**CG**).



Note: In ASCE 7-16 there is no required seismic loading required for structures which fall under Seismic Design Category A. Instead, [notional loads](#) should be applied.

Seismic Generation Diaphragm Results

This section displays information used in calculating the accidental torsion values. This includes the **Width** and **Length** of each diaphragm and the distance used for the accidental eccentricity.



Note:

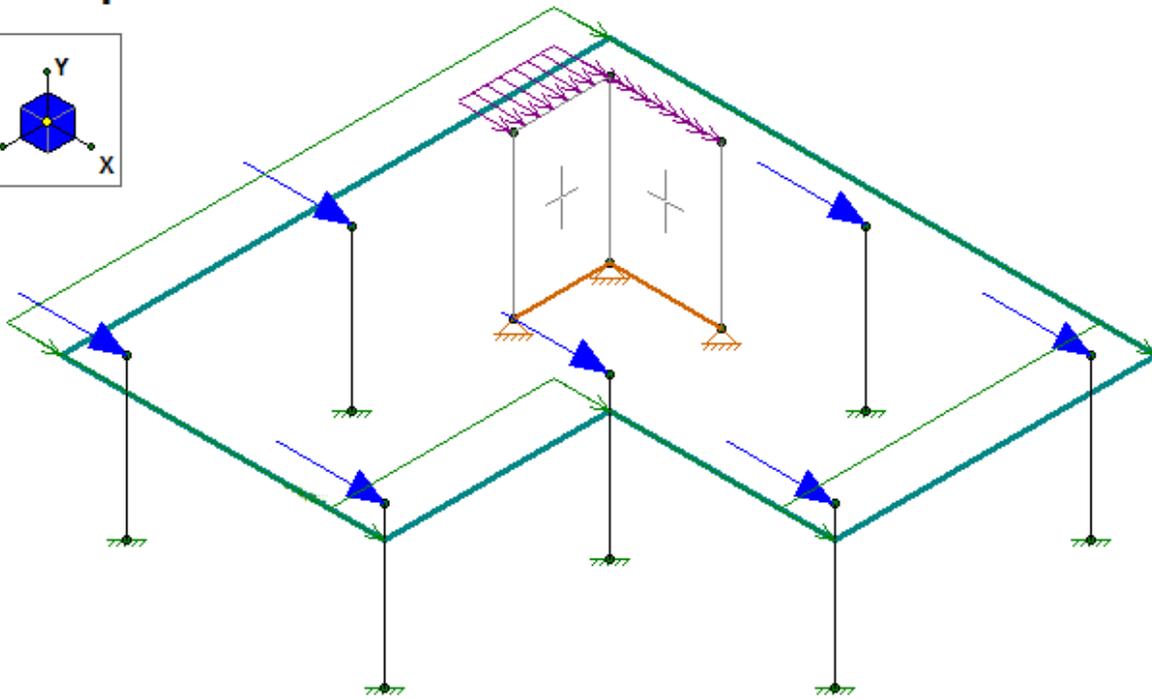
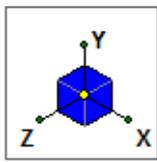
- The magnitudes of the seismic loads are based on the settings under the Seismic tab of the Model Settings dialog.
- The Joints / Nodes used to apply seismic load may appear as a "diamond" shape pattern of unattached nodes at that diaphragm level. While these nodes may not be attached to any beams or framing, they are attached to the diaphragm at that floor level. To get rid of these nodes, it is necessary to re-run the Seismic force generation with the design code set to NONE.
- Any equations based on NON-building structures are not currently taken into account for the seismic load calculations.

Semi-Rigid Seismic Loads

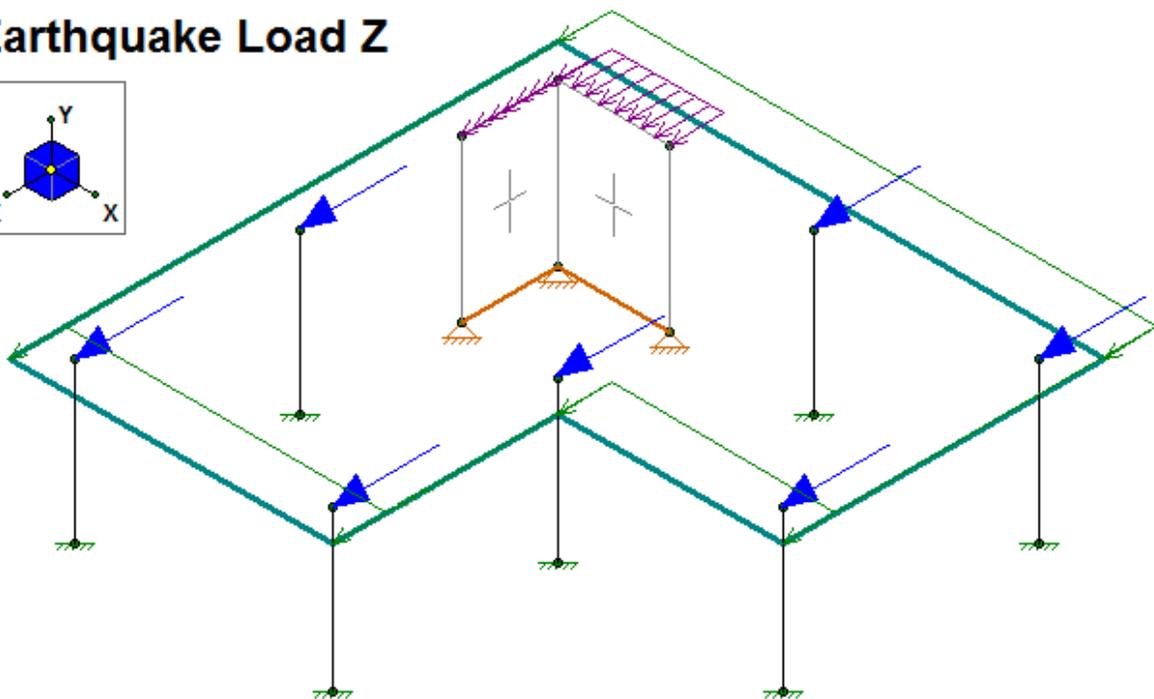
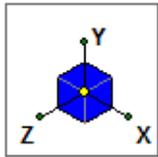
When running a combined RISAFloor/RISA-3D model the program has the ability to create Semi-Rigid Seismic loads and apply them to the diaphragm. The seismic load is calculated by taking the Total Seismic Weight and converting it into a horizontal direction by multiplying by the seismic response coefficient **Cs**.

The program applies a Diaphragm Surface load which represents the seismic contribution of the Slab weight and any additional Dyn Load. There are horizontal point loads and line loads at the top of the columns and walls which represent their respective contribution of the seismic weight. Any point, line or area loads that are "Dyn Mass" is also converted as horizontal seismic load applied directly to the diaphragm. Below shows an example of the Earthquake loads applied into a simple L-shaped building.

Earthquake Load X



Earthquake Load Z



**Note:**

- For a RISA-3D only model you need to manually apply your loads and build your diaphragm with plates. See the [Semi-Rigid Diaphragms](#) section for more information on this.
- Any Openings in the slab (Concrete Floor Slabs) will have the self weight correctly accounted for.

Load Generation - Wind Loads

"Building" Wind Loads can be automatically generated per the following codes:

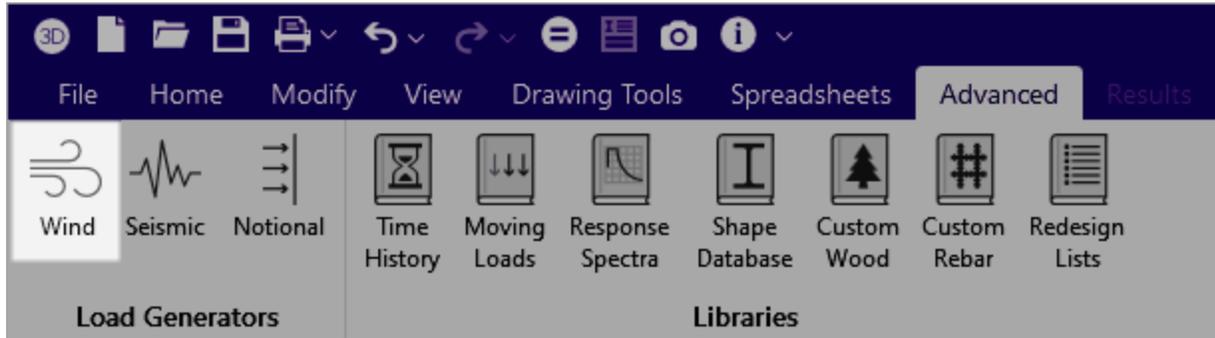
- ASCE 7-16
- ASCE 7-10
- ASCE 7-05
- ASCE 7-02
- ASCE 7-98
- ASCE 7-95
- NBC 2015 (Canadian)
- NBC 2010 (Canadian)
- NBC 2005 (Canadian)
- NTC 2004 (Mexican)
- IS 875: 1987 (Indian)



Note: The wind load generator creates [Basic Load Cases](#). You must generate wind [load combinations](#) to have the wind loads actually applied to the structure.

Wind Design Parameters

The parameters used for automated wind load generation may be viewed or changed by clicking the **Wind** icon in the **Advanced** ribbon. These settings may also be changed when the wind load window pops up when transferring between RISAFloor and RISA-3D.



The parameters and results shown below are specific to ASCE 7-16, however the concepts apply to all wind codes. For questions specific to other wind codes contact RISA [Technical Support](#).

Wind Load Generator
?
×

RISA-3D Wind Load Options

Code:

Wind Speed: mph

Exposure Cat:

Base Elevation: ft

Gust Stiffness:

Fundamental Freq.: Hz

Gust Factor (G):

Site Parameters

Topographic Factor, K_1 :

Topographic Factor, K_2 :

Topographic Factor, K_3 :

Directionality Factor, K_d :

Calculate Loads

Generate Loads starting at BLC:

Wind Generation Input

Wind Code: ASCE 7-10 Directionality Factor, K_d : 0.85

Wind Speed, V: 115 mph Gust Stiffness: Rigid

Exposure Category: C

Base Elevation: 0 ft

Topographic Effects

Factor K_1 : 0 Factor K_2 : 0 Factor K_3 : 0

Wind Generation Detail Results

Exposure Constant Alpha: 9.5 K_{zt} : 1 g

Exposure Constant, z_g : 900 h: 24 ft

Gust Effect Factor, G: 0.85 K_h : 0.937 g

Windward C_p : 0.8 q_h : 0.027 ksf

Wind Generation Floor Geometry Results

Floor Level	Height (ft)	K_z	Width (Z) (ft)	Length (X) (ft)	Leeward C_p (Z)	Leeward C_p (X)
Diaphragm 1	24	0.937	36	54	0.5	0.4
Diaphragm 2	12	0.849	36	54	0.5	0.4

Wind Generation Floor Force Results

Floor Level	q_z (ksf)	Windward Pressure (ksf)	Leeward Pressure (Z) (ksf)	Leeward Pressure (X) (ksf)	Force (Z) (kips)	Force (X) (kips)
Diaphragm 1	0.027	0.018	0.011	0.009	9.656	5.942
Diaphragm 2	0.024	0.017	0.011	0.009	18.192	11.138

Add to Full Report

RISA-3D Wind Load Options

Code specifies which code is used to generate the loads.

Wind Speed (V) is used to calculate wind pressures. See ASCE 7-16, Section 26.5.1

Exposure Category is used for multiple wind load calculations, and is defined in ASCE 7-16, Section 26.7.3

Base Elevation defines the elevation that the program considers as the "ground elevation". This is typically used for structures which have basements, or base coordinates other than 0.

Gust Stiffness is used in the calculation of the Gust Effect Factor. See ASCE 7-16, Section 26.11. (NOTE: This is only available for ASCE 7-16 & ASCE 7-10).

Fundamental Frequency is used in the calculation of the Gust Effect Factor and can be modified when the Gust Stiffness is set to Flexible.

Damping Ratio is also used in the calculation of the Gust Effect Factor. This option only appears and can only be modified when the Gust Stiffness is set to Flexible.

Gust Factor (G) is explained in ASCE 7-16, Section 26.11 and depends on the Gust Stiffness setting. See [Gust Effect Factor](#).

Site Parameters

Topographic Factors (K_1 , K_2 , K_3) are used to calculate the Topographic Factor (K_{zt}) per ASCE 7-16, Section 26.8.2.

Directionality Factor (K_d) is used to calculate wind pressures. See ASCE 7-16, Section 26.6.

Ground Elevation Factor (K_e) is calculated per ASCE 7-16, Section 26.9 using Table 26.9-1 (note: Per ASCE 7-16, Section 26.9, K_e is permitted to equal 1 for all elevations). This option only appears and can only be modified when ASCE 7-16 is selected.

Generate Roof Wind Loads is only present in RISA-3D models which are integrated with RISAFloor. This toggles the automatic generation of [sloped roof wind loads](#).

Wind Load Results

The program calculates the appropriate wind loads and presents the calculations in a printable report. You can open the wind load generator at any time to view, print or recalculate the wind loads.

Changes made to RISAFloor models which are integrated with RISA-3D models will be reflected in automatically updated wind loads when transferring between the programs.

Wind Generation Input

This section reports values which were directly or indirectly input from the [Wind Load Parameters](#).

Wind Generation Detail Results

This section reports important values that were calculated using the Wind Load Parameters, as well as the height and base elevation of the structure.

Importance Factor is calculated using the input Occupancy Category.

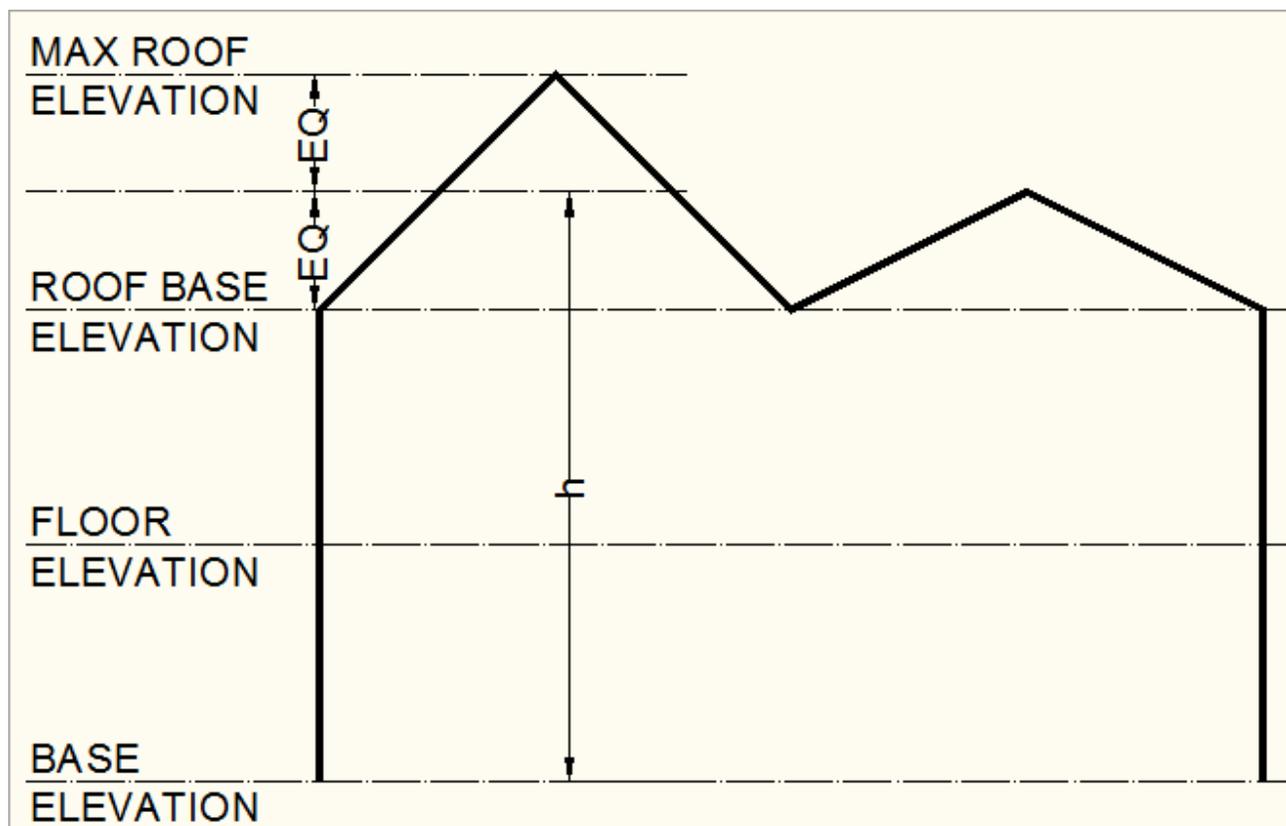
Exposure Constant Alpha (α) is determined from ASCE 7-16, Table 26.11-1 based on the input Exposure Category.

Exposure Constant z_g is determined from ASCE 7-16, Table 26.11-1 based on the input Exposure Category.

Topographic Factor (K_{zt}) is calculated per ASCE 7-16, Section 26.8.2 using the input Topographic Factors (K_1, K_2, K_3)

Ground Elevation Factor (K_e) is calculated per ASCE 7-16, Section 26.9 using Table 26.9-1 (note: per ASCE 7-16, Section 26.9 K_e is permitted to equal 1 for all elevations)

Mean Roof Height (h) is determined as illustrated below:



Exposure Coefficient (K_h) is calculated per ASCE 7-16, Table 26.10-1 at the mean roof height as illustrated above. The Exposure Constants (z_g, α) are used in conjunction with the formula in Table 26.11-1, Note 2 to calculate the Exposure Coefficient.

Windward Pressure Coefficient (Cp) is calculated per ASCE 7-16, Figure 27.3-1. The wind load generator always sets this value at 0.8 per ASCE 7-16, Figure 27.3-1.

Mean Roof Velocity Pressure (qh) is calculated per ASCE 7-16, Section 26.10.2. The input Directionality Factor and Wind Speeds are used.

Parapet Wind Pressure Coefficient (GCpn) is calculated per ASCE 7-16 Section. 27.3.4 for both windward and leeward directions.

Gust Effect

When the Gust Stiffness is set to Flexible, this section reports the values used in the calculation of the gust effect factor.

Gust Effect Factor(G) is explained in ASCE 7-16, Section 26.11. Gust Effect Factor depends on the **Gust Stiffness** input setting (NOTE: This is only available for ASCE 7-16 & ASCE 7-10).

Wind Load Generator
?
×

RISA-3D Wind Load Options

Code:

Wind Speed: mph

Exposure Cat:

Base Elevation: ft

Gust Stiffness:

Fundamental Freq.: Hz

Damping Ratio:

Site Parameters

Topographic Factor, K₁:

Topographic Factor, K₂:

Topographic Factor, K₃:

Directionality Factor, K_d:

Ground Elev. Factor, K_e:

[Calculate Loads](#)

Generate Loads starting at BLC: ▼

Add to Full Report

Wind Generation Input

Wind Code	ASCE 7-16	Directionality Factor, K _d	0.85
Wind Speed, V	115 mph	Gust Stiffness	Flexible
Exposure Category	B	Damping Ratio	0.02
Base Elevation	0 ft	Fundamental Frequency	0.9 Hz
		Ground Elevation Factor, K _e	1

Topographic Effects

Factor K₁: 0 Factor K₂: 0 Factor K₃: 0

Wind Generation Detail Results

Exposure Constant Alpha	7	K _{zt}	1	g
Exposure Constant, z _g	1200	h	24	ft
		K _h	0.657	g
Windward C _p	0.8	q _h	0.019	ksf

Gust Effect

I z-bar	0.305	z-bar	30	ft
epsilon-bar	0.333	b-bar	0.45	
L z-bar	309.993	c	0.3	
alpha-bar	0.25	l	320	ft
V-bar z-bar	74.113	ft/s	g _R	4.164

Floor Level	BX	BZ	LX	LZ	QX	QZ	RBX	RBZ	RLX	RLZ	RX	RZ	GfX	GfZ
	(ft)	(ft)	(ft)	(ft)	(ft)	(ft)								
Diaphragm 1	54	36	36	54	0.889	0.904	0.277	0.376	0.138	0.094	0.493	0.565	0.969	1.006
Diaphragm 2	54	36	36	54	0.889	0.904	0.277	0.376	0.138	0.094	0.493	0.565	0.969	1.006

Wind Generation Floor Geometry Results

Floor Level	Height (ft)	K _z	Width (Z) (ft)	Length (X) (ft)	Leeward Cp (Z)	Leeward Cp (X)
Diaphragm 1	24	0.657	36	54	0.5	0.4
Diaphragm 2	12	0.575	36	54	0.5	0.4

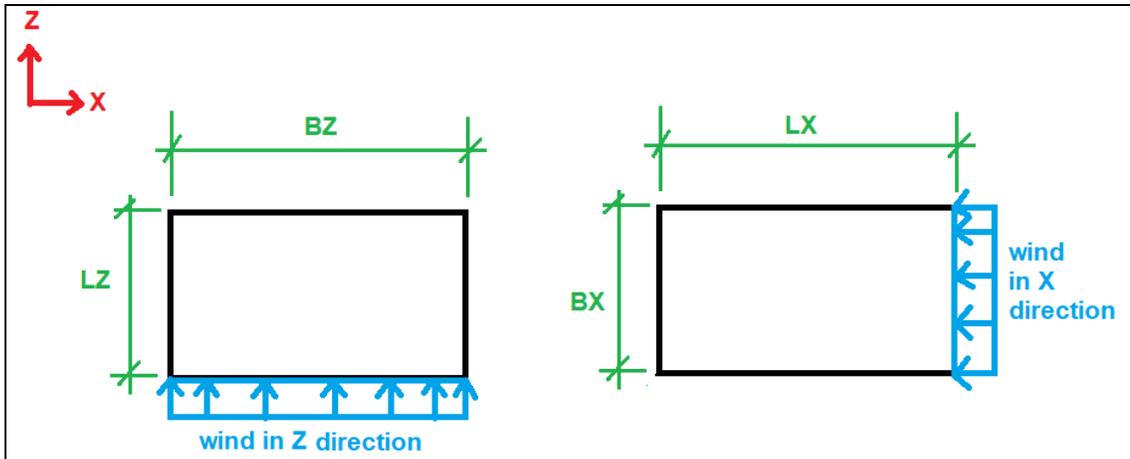
Wind Generation Floor Force Results

Floor Level	q _z (ksf)	Windward Pressure (ksf)	Leeward Pressure (Z) (ksf)	Leeward Pressure (X) (ksf)	Force (Z) (kips)	Force (X) (kips)
Diaphragm 1	0.019	0.015	0.009	0.008	7.722	4.935
Diaphragm 2	0.017	0.013	0.009	0.008	14.25	9.043

Add to Full Report

- When **Gust Stiffness** is set to **Rigid**, the Gust Factor is automatically set to 0.85.
- When the **Gust Stiffness** is set to **Flexible**, the Gust Factor is calculated per *ASCE 7-16 Eqn (26.11-10)*.
 - You must input the damping ratio and fundamental frequency for the flexible calculation.
 - Your fundamental frequency must be less than 1 Hz per *ASCE 7-16 section 26.2*.
 - Variables \bar{z} , c , I , $\epsilon_{\bar{z}}$, $b_{\bar{z}}$, $\alpha_{\bar{z}}$ are determined from *ASCE 7-16 Table 26.11-1*.
 - $I_{\bar{z}}$ is calculated per *ASCE 7-16 Eqn (26.11-7)*.
 - $L_{\bar{z}}$ is calculated per *ASCE 7-16 Eqn (26.11-9)*.
 - $V_{\bar{z}}$ is calculated per *ASCE 7-16 Eqn (26.11-16)*.
 - Variables g_Q and g_V are taken as 3.4 per *ASCE 7-16 section 26.11.5*.
 - g_R is calculated per *ASCE 7-16 Eqn (26.11-11)*.
 - N_1 is calculated per *ASCE 7-16 Eqn (26.11-14)*.
 - R_n is calculated per *ASCE 7-16 Eqn (26.11-13)*.
 - R_h is calculated per *ASCE 7-16 Eqn (26.11-15)*.
 - **QX** and **QZ** are calculated per *ASCE 7-16 Eqn (26.11-8)* for each horizontal direction.
 - **RBX** and **RBZ** are calculated per *ASCE 7-16 Eqn (26.11-15)* for each horizontal direction.
 - **RLX** and **RLZ** are calculated per *ASCE 7-16 Eqn (26.11-15)* for each horizontal direction.
 - **RX** and **RZ** are calculated per *ASCE 7-16 Eqn (26.11-12)* for each horizontal direction.
 - **BX** and **BZ**, **LX** and **LZ** are the width and lengths of the floor diaphragms for each direction. B is the dimension normal to the direction of the wind, L is the dimension

parallel with the direction of the wind.



Wind Generation Floor Geometry Results

The wall geometry used to calculate the wind loads is reported here.

Floor Level is the name of the Floor from RISAFloor. Multiple diaphragms on the same floor will each be reported separately. For sloped wall calculations the level will always be reported as "Sloped Roof".

Height is the height of each diaphragm above the input Base Elevation. For "Sloped Roof" floors from RISAFloor the height reported is the height of the highest point in each diaphragm.

Exposure Coefficient (Kz) is the calculated exposure coefficient at the diaphragm elevation. For "Sloped Roof" floors from RISAFloor the coefficient is calculated for the highest point in each diaphragm.

Width is calculated as the difference between the highest and lowest magnitude X-Coordinates on the diaphragm.

Length is calculated as the difference between the highest and lowest magnitude Z-Coordinates on the diaphragm.

Pressure Coefficient (Cp) is calculated per ASCE 7-16, Figure 27.3-1. The Height/Width and Height/Length ratios (see above) are used to calculate leeward pressure coefficients.

If the level considered is not sloping and does not have a parapet then the results here will be a single line.

Floor Plan 1	12	.575	49	53	.5	.484
--------------	----	------	----	----	----	------

If a parapet exists at this level, then information regarding the parapet will be given as well.

Base Parapet	53	.824	47	19.021
--------------	----	------	----	--------

If this level is sloping then there may also be a section giving the wind face information due to this sloping portion.

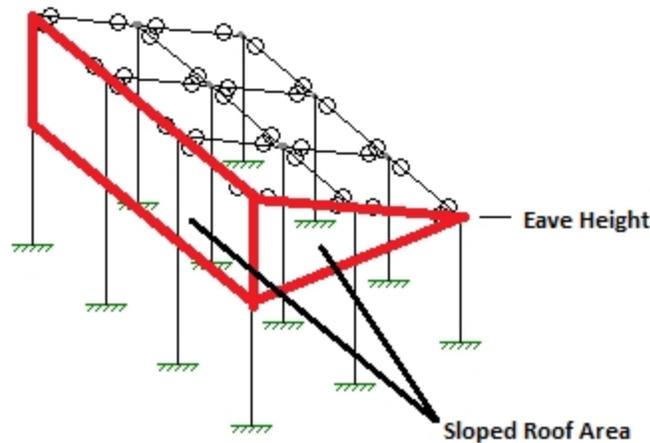
Sloped Roof 57.5 .844 64.542 (area) 168.75 (area) .276 .5

Sloped Roof Area is used to account for the wind load on walls which project above the base roof elevation due to a sloping roof. This only occurs in models which are integrated with RISAFloor. The program determines area by going from support-to-support finding vertical polygons along the slab edge, and calculating their projected area in both the X and Z directions.

The area reported is the windward followed by leeward.

Wind Generation Floor Geometry Results

Floor Level	Height (ft)	Kz	Width (X) (ft)	Length (Z) (ft)	Leeward Cp(X)	Leeward Cp(Z)
Floor Plan 1	12	.575	22	32	.5	.409
Base Parapet	16	.585	22	32	.5	.409
Sloped Roof	22	.641	300/0 (ft^2)	50/50 (ft^2)	.5	.409



Wind Generation Floor Force Results

The wind forces applied to each diaphragm are reported in this section.

Floor Level is the name of the Floor from RISAFloor. Multiple diaphragms on the same floor will each be reported separately. For sloped wall calculations the level will always be reported as "Sloped Roof".

Velocity Pressure (q_z) is calculated per ASCE 7-16, Section 26.10.2. The input Directionality Factor and Wind Speeds are used.

Windward Pressure is calculated per ASCE 7-16, Section 27.3.1. The calculated Velocity Pressure and Pressure Coefficient are used, as well as the standard Gust Effect Factor (0.85).

Leeward Pressure is calculated per ASCE 7-16, Section 27.3.1. The calculated Velocity Pressure and Pressure Coefficient are used, as well as the standard Gust Effect Factor (0.85).

Force is calculated as the Length/Width (see above) of the diaphragm, multiplied by the tributary height of the diaphragm, multiplied by the sum of the windward and leeward pressures. The

additional "Sloped Roof" wall wind force in RISAFloor integrated models is also added to this value for the Roof.

The tributary height of the diaphragm is defined as half the distance the diaphragm immediately above, and half the distance to the diaphragm immediately below. At the roof the tributary height is just half the distance to the diaphragm immediately below. At the lowest level of the structure the tributary height is half the distance to the Base Elevation, as it is assumed that all wind load below that is tributary to the ground.

The Forces calculated for each diaphragm are applied as horizontal joint loads at the center of exposure, and four "eccentric" points (per ASCE 7-16, Section 27.3.5). These joints form a diamond pattern, which can be viewed in the model.

If the level considered is not sloping and does not have a parapet then the results here will be a single line.

Floor Level	qz	Windward Pres.	Leeward Pres. X	Leeward Pres. Z	Force X	Force Z
Floor Plan 1	.017	.011	.01	.01	14.723	13.401

If a parapet exists at this level, then information regarding the parapet (and possibly the additional height of multiple height parapets) will be given as well and the total force due to both the main wind force and the parapet wind force will be summed.

Floor Level	qz	Windward Pres.	Leeward Pres. X	Leeward Pres. Z	Force X	Force Z
Floor Plan 4	.023	.016	.005	.01	2.436	7.27
Base Parapet	.024				3.024	8.006
Add'l Parapet					1.841	3.887
Total					7.302	19.162

If this level is sloping then information regarding the sloping area will be given as well and the total force due to both the wind force below the eave and the wind force above the eave will be summed.

Floor Level	qz	Windward Pres.	Leeward Pres. X	Leeward Pres. Z	Force X	Force Z
Floor Plan 4	.023	.016	.006	.01	2.449	7.328
Sloped Roof	.024	.017	.006	.01	2.056	4.494
Total					4.505	11.822



Note:

- The **Total** value will be the magnitude actually applied in RISA-3D to this level.
- For more information on parapets see the [Parapet Wind Loading](#) section.
- For more information on sloping roof wind loads see the [Sloping Roof](#) section.

Parapet Wind Loading

When RISA-3D is integrated with RISAFloor then it is possible to define parapets for both wall panels and columns. After parapets are defined the wind load generator will use these parapet heights to add additional wind loading to the roof levels of your structure to include this parapet wind with the

roof wind load. The total wind force, including all parapets and the non-parapet wind, will be applied at the center of wind location in RISA-3D. Here we will give some details as to how this works.

Display

The reporting was outlined above. However, the program will also give a Parapet Summary as well.

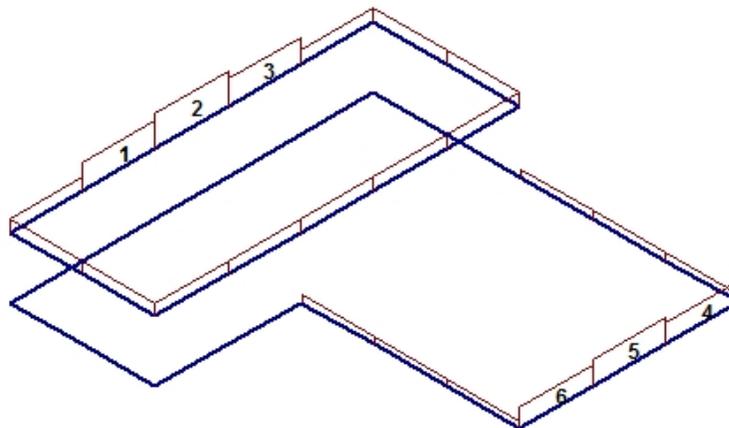
Wind Generation Floor Geometry Results

Floor Level	Height (ft)	Kz	Width (X) (ft)	Length (Z) (ft)	Leeward Cp(X)	Leeward Cp(Z)
Roof	20	.902	20.5	50.5	.5	.277
Base Parapet	25	.945	20.5	50.5		
Second Floor	10	.849	52	52	.5	.5
Base Parapet	13	.849	52	52		

Wind Generation Floor Force Results

Floor Level	qz	Windward Pres.	Leeward Pres. X	Leeward Pres. Z	Force X	Force Z
Roof	.019	.013	.008	.004	5.218	1.755
Base Parapet	.02				12.251	4.9
Add'l Parapet					4.583	0
Total					22.051	6.655
Second Floor	.018	.012	.008	.008	10.357	10.357
Base Parapet	.018				2.376	3.928
Total					12.734	14.285

Parapet Summary (heights are base parapet height unless noted below)



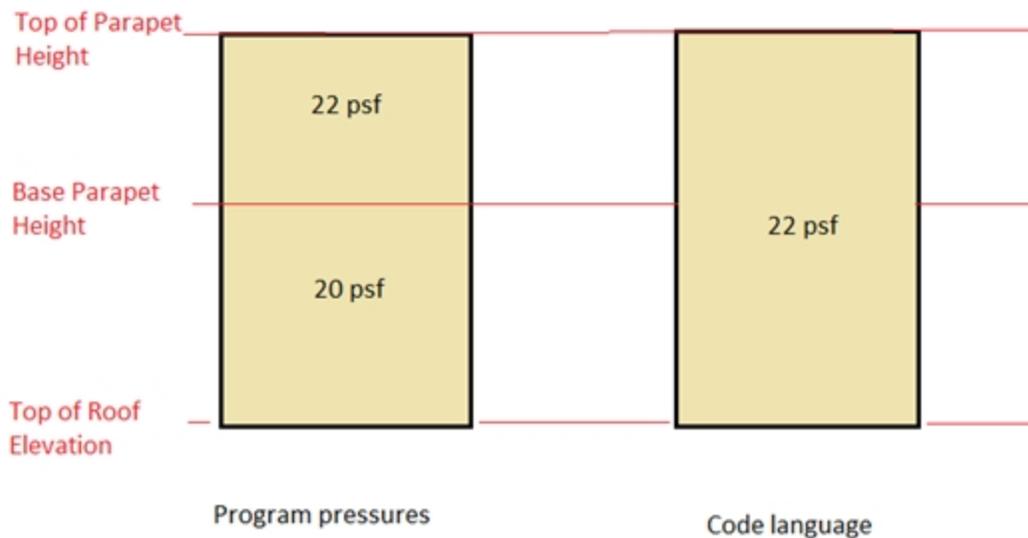
Parapet	Height Above	X Length	Z Length	qp (ksf)	Force X (k)	Force Z (k)
1	5	0	10	.02	1.528	0
2	8	0	10	.021	2.494	0
3	5	0	10	.02	1.528	0
4	5	0	10	.018	1.372	0
5	7	0	10	.019	1.964	0
6	5	0	10	.018	1.372	0

Note: $GC_{pn} = 1.5$ for these portions of the parapet in both horizontal directions.

In this image you can see (to scale) where all the parapets in your structure are located. The base parapet information has already been considered in the Base Parapet, so the only parapet information listed below the image are for the parapets that extend above the Base Parapet. The GC_{pn} for this portion of the parapets will be assumed to always be windward. Each of these parapets is shown with the extra force above and beyond the Base Parapet Height. The Force X & Z columns are once again added to the Base Parapet and the regular building wind to give the total wind applied at that floor level.

Note:

- Parapet widths do not include deck overhangs.
- A simplifying assumption is made for these extended parapets that is summarized in the image below.



Parapet Height Determination

Parapet height information is located in three places: the Floors, Columns, and [Wall Panels](#) spreadsheets. If a floor has a **Default Parapet Height** set then all walls and columns at that level will have their parapet values set to that value. This is considered the **Base Parapet**.

	Label	Elev...	Area L...	Floor Type	Deck Defa...	Deck A...	Parent	Inacti...	Splice Dist...	Splice Type	No ...	Parapet Height Default(ft)
1	Second Floor	10	Retail	Floor Beam	Wood	90	None	<input type="checkbox"/>	10	Shear	<input type="checkbox"/>	3
2	Roof	20	Roof	Floor Beam	Wood	90	None	<input type="checkbox"/>	7.5	Moment	<input type="checkbox"/>	5

The walls and columns are physically extended. From here you can go to the Columns or Wall Panels spreadsheets if you need to extend parapets above this base level. These values are the **Top of Parapet Height**. If this value is higher than the **Default Parapet Height** then this component of win loading will show up in the **Add'l Parapet** entry.

	Label	Point	Shape	Material	Function	An...	O.	De...	Splice	Distance B...	Splice Type	Splice ...	Parapet Height (ft)	Top Fix...
10	CS13 (C-3)_L1	N29	Pipe	A500 Gr.42	Lateral			DR1	<input checked="" type="checkbox"/>	Default (10)	Default (Shear)	None	NA	Fixe...
11	CS11_L1	N10	Wide Flange	A36 Gr.36	Lateral			Typical	<input checked="" type="checkbox"/>	Default (10)	Default (Shear)	None	(Default) 3	Fixe...
12	CS12_L1	N11	Wide Flange	A36 Gr.36	Lateral			Typical	<input checked="" type="checkbox"/>	Default (10)	Default (Shear)	None	(Default) 3	Fixe...
13	CS13_L1	N12	Wide Flange	A36 Gr.36	Lateral			Typical	<input checked="" type="checkbox"/>	Default (10)	Default (Shear)	None	NA	Fixe...
14	CS14_L1	N9	Wide Flange	A36 Gr.36	Lateral			Typical	<input checked="" type="checkbox"/>	Default (10)	Default (Shear)	None	(Default) 3	Fixe...

	Label	Top Floor	Bottom Floor	Start P...	End P...	Material ...	Material Set	Thicknes...	Function	Design Rules	Distance B...	Parapet Height (ft)
14	WP17	Roof	Second Floor	N18	N17	Wood	DF/SPine	5.5 (stud)	Lateral	Typical	10	10
15	WP18	Roof	Second Floor	N19	N18	Wood	DF/SPine	5.5 (stud)	Lateral	Typical	10	13
16	WP19	Roof	Second Floor	N20	N19	Wood	DF/SPine	5.5 (stud)	Lateral	Typical	10	10
17	WP20	Roof	Second Floor	N1	N20	Wood	DF/SPine	5.5 (stud)	Lateral	Typical	10	(Default) 5
18	WP21	Roof	Second Floor	N3	N25	Wood	DF/SPine	5.5 (stud)	Lateral	Typical	10	(Default) 5
19	WP22	Roof	Second Floor	N25	N29	Wood	DF/SPine	5.5 (stud)	Lateral	Typical	10	(Default) 5
20	WP23	Roof	Second Floor	N29	N12	Wood	DF/SPine	5.5 (stud)	Lateral	Typical	10	(Default) 5
21	WP24	Second Floor	Base	N1	N20	Wood	DF/SPine	5.5 (stud)	Lateral	Typical	10	NA
22	WP25	Second Floor	Base	N20	N19	Wood	DF/SPine	5.5 (stud)	Lateral	Typical	10	NA
23	WP26	Second Floor	Base	N19	N18	Wood	DF/SPine	5.5 (stud)	Lateral	Typical	10	NA

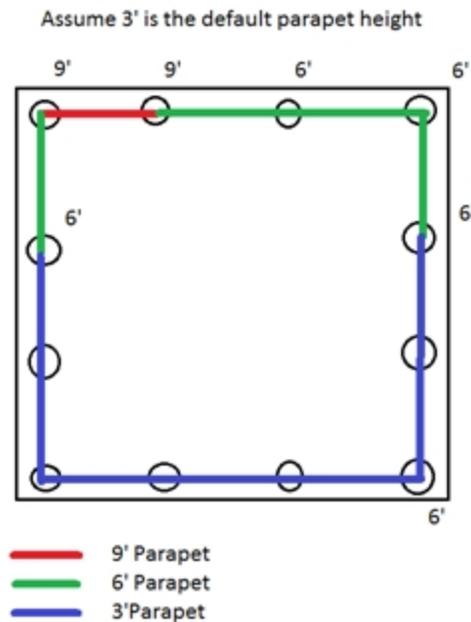


Note:

Limitations for slab-supported floors

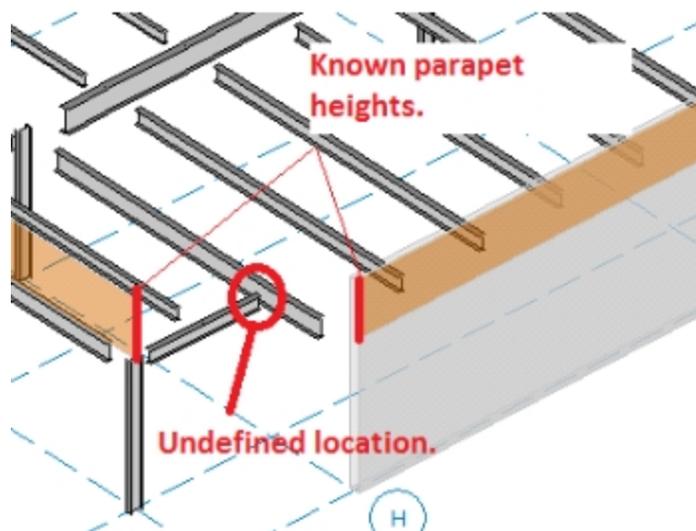
- Only the highest elevation floor in the structure can have a parapet.
- Parapets must be a uniform height around the entire slab edge and the height is defined in the Floors spreadsheet in RISAFloor.

For walls a parapet is defined along a line that defines the top of the wall. Thus, the area of the parapet is trivial. For parapets defined by columns, however, we now have parapet heights at finite locations. In this case the program will go "column to column" around the structure to determine the parapet height. The parapet height used between columns is the **SMALLER** of the two values at each column. Below is an example of what this would look like.



Note:

- A parapet can only be defined if it is along a deck edge. Any columns or walls that are either not along the deck edge or are not the topmost element in the stack will not be allowed to define a parapet and NA will be shown.
- If there are re-entrant corners there may be a case where there is not a column at a corner location. In this case a parapet height equal to the adjacent column/wall will be used in this area.



Code Criteria

Per the ASCE 7-16 Section 27.3.4, the velocity pressure at the top of the parapet shall be multiplied by a GC_{pn} of +1.5 for windward parapets and -1.0 for leeward parapets.

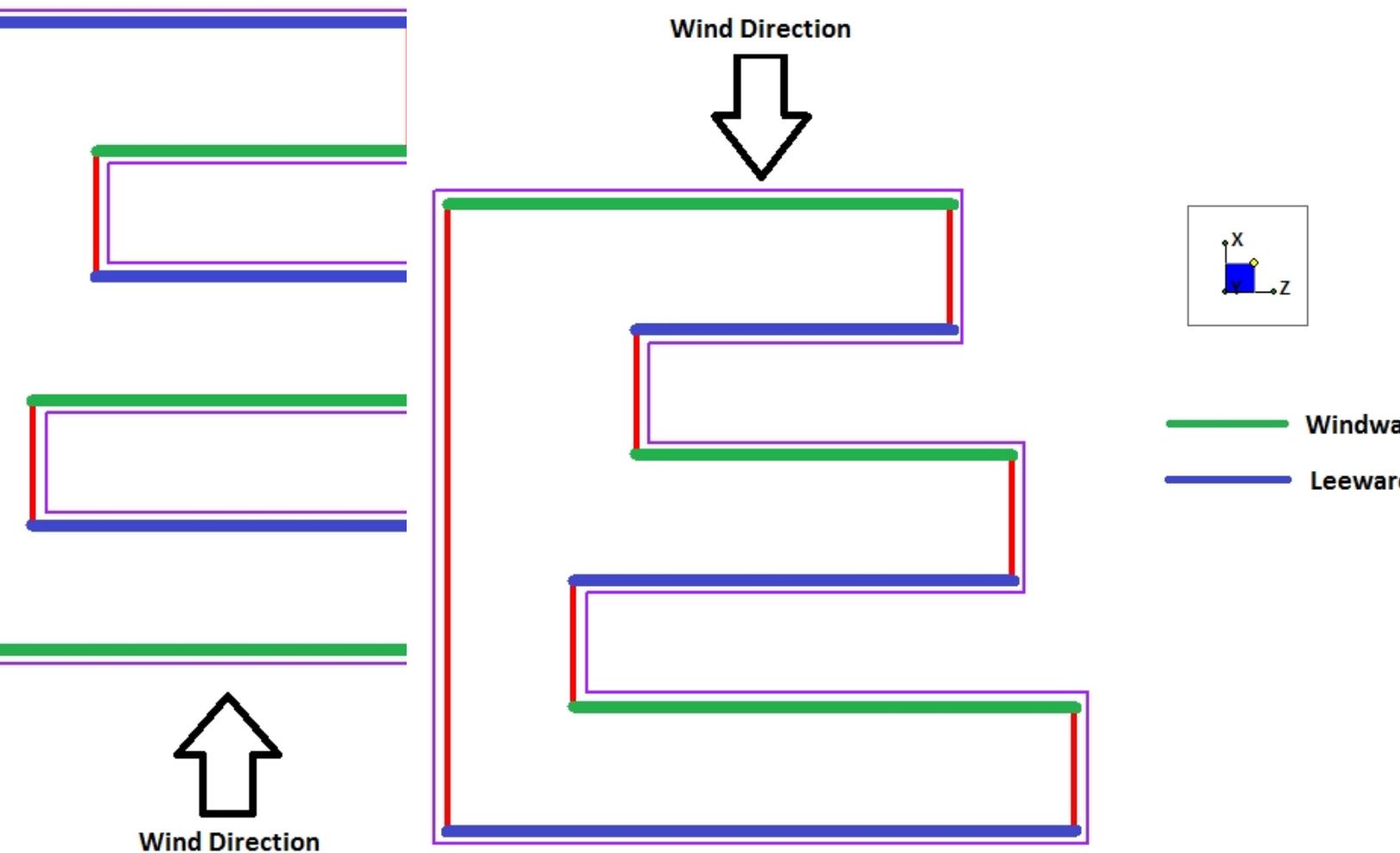


Note:

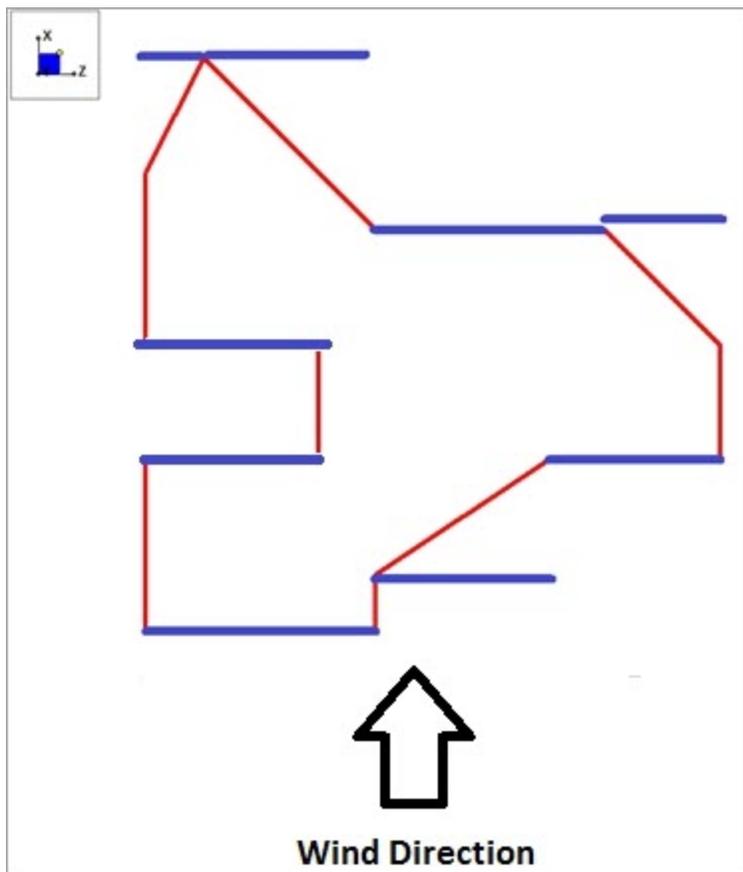
- Older versions of the ASCE 7 have identical parapet considerations.
- The Canadian codes (NBC 2015/10/05) do not have specific parapet treatment. The wind load section is also very different from the ASCE 7 methodology. Thus, we do not consider additional parapet height for Canadian codes.
- The Mexican code (NTC 2004) gives nothing specific to parapets. However, the Mexican code lays out very similarly to the ASCE 7 code. Thus, we are using the ASCE 7 methodology for this code, with a +1.5 GC_{pn} factor for windward parapets and -1.0 GC_{pn} for leeward.
- The Indian code (IS 875: 1987) specifies a parapet load factor of 2.25 GC_{pn} , but they do not specify windward and leeward. We assume a +1.25 factor for windward and a -1.0 factor for leeward.

Windward/Leeward Pressure Considerations

Most codes prescribe a higher pressure on the windward parapet than the leeward parapet. Thus, parapet wind loads may or may not be symmetric in each direction. The program will calculate the wind loads in both the positive and negative directions and use the larger of the two. For rectangular structures it is very simple to determine windward/leeward directions. However, a more complex structure can be more difficult. The image below gives one example of how windward/leeward parapets are considered.



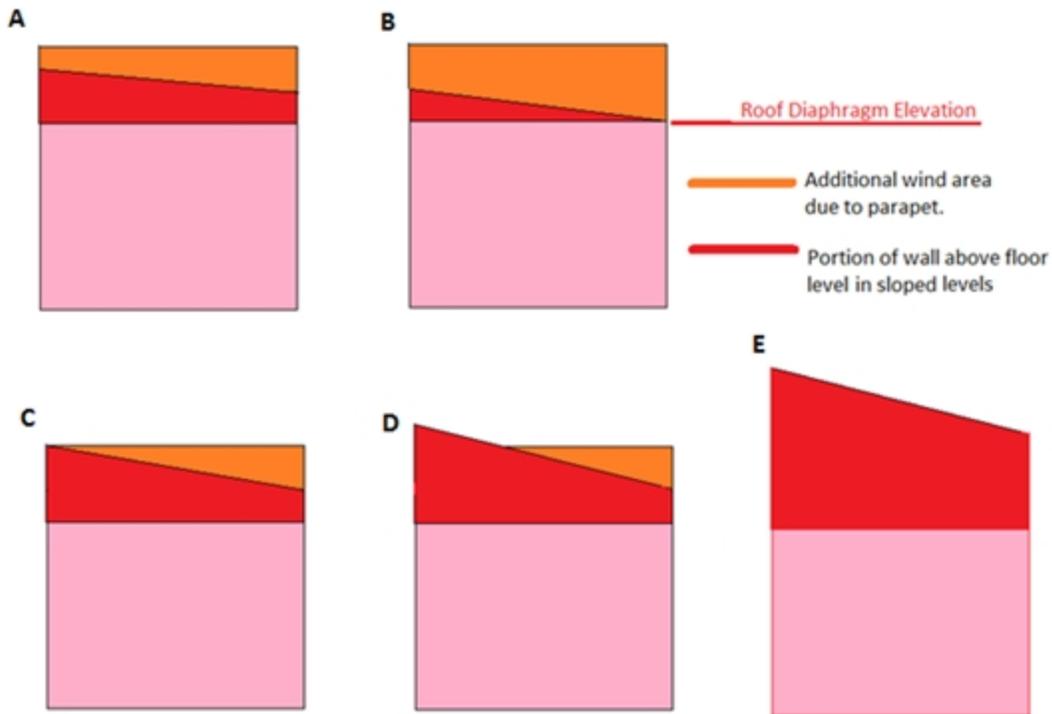
 **Note:** For parapets that are non-orthogonal to the global axes, the program properly takes the projected area when calculating wind areas.



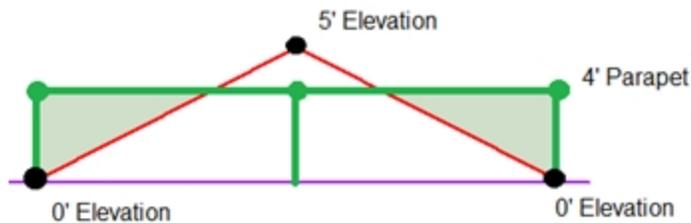
Sloping Floors with Parapets

In RISAFloor it is possible to slope the highest level in a structure. This can create "wind faces" due to these slopes. Think of a monosloped structure. Above the eave you have triangular areas that are wind faces that the program already calculates appropriately. If there are additional parapets to consider that occur as well the geometry can become challenging.

Note that the program will look at both the +/- X and +/- Z directions to determine which direction has the higher force and that is the force that will be used in the program.

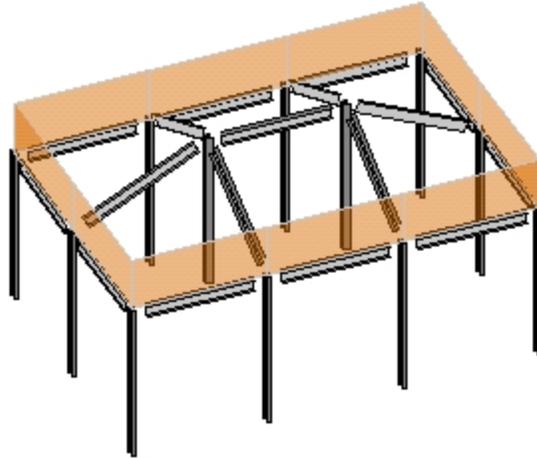


Note: All cases above are handled properly, except Case D. Because this would be a very odd scenario for a parapet on a sloped roof, the orange area above would not be included in parapet wind load areas.



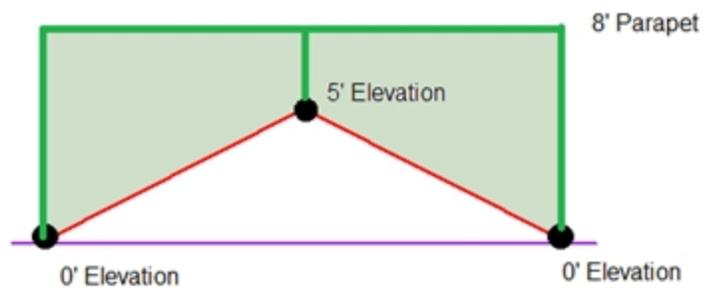
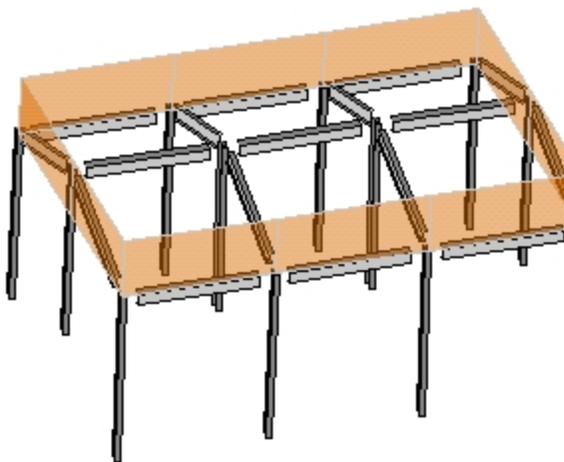
Here are some common conditions:

Gabled Roof with No Vertical Wind Faces and Parapets



Here we have a sloping roof which will have perpendicular wind loads applied. This will occur. In addition there will be additional horizontal loads due to the parapet.

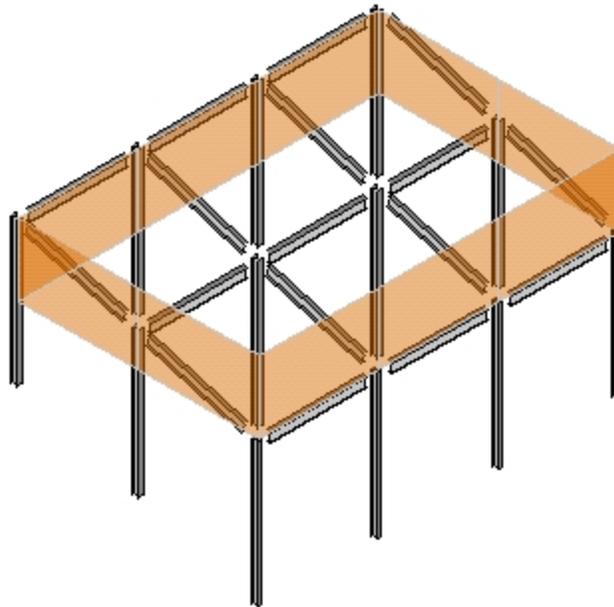
Gabled Roof with End Vertical Faces and Parapets



In this case the triangular face is taken as a regular wind load and the additional triangular/polygon loads that are due to the parapet (similar to Case A,B, or C) have their areas calculated and included as parapet loads.

Wind Generation Floor Force Results

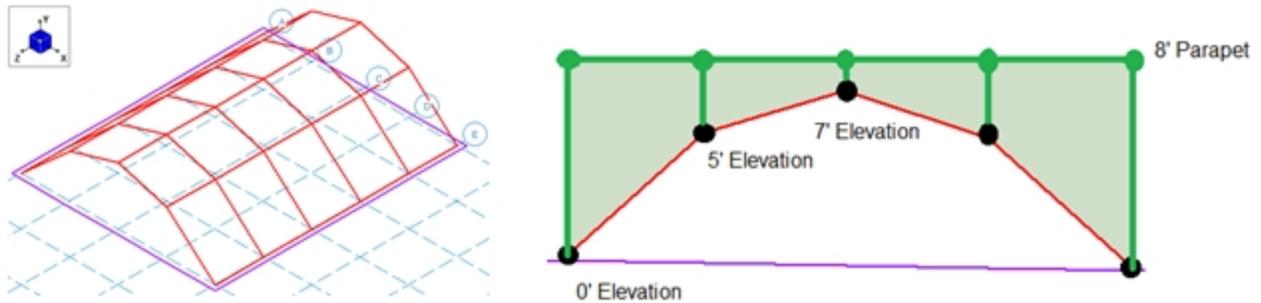
Floor Level	qz	Windward Pres.	Leeward Pres. X	Leeward Pres. Z	Force X	Force Z
Floor Plan 1	.017	.011	.007	.006	8.563	4.778
Base Parapet	.017				5.054	1.965
Sloped Roof	.017	.011	.007	.006	0	.569

Monoslope Roof With Parapet

In this case there are triangular parapet wind areas on two sides (similar to Case C above), a full parapet area on one side, and no parapet on the backside. In the triangular parapet direction the wind areas are equal in each direction thus the same value will occur in both directions. In the perpendicular direction the parapet is windward and the sloped area is leeward in the positive direction, and vice-versa in the negative direction. In this case the program will take the largest magnitude and apply it to your roof.

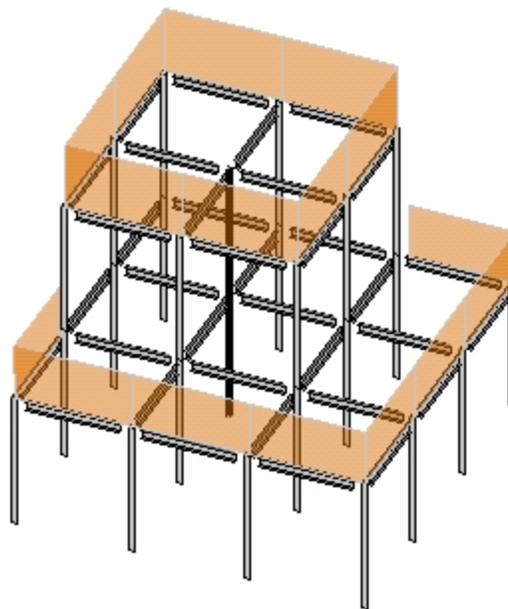
Gambrel Roof With Parapet

This is similar to the gabled with a slightly different geometry.



Partial Roof Levels

It is also possible to define partial roof parapets. The program will handle this scenario properly. Below we can see that we have a lower level with parapets defined around the edges. Thus, for some of the length of the parapet there is not a leeward parapet included.



Mezzanine Levels (No Wind Loads)

You can designate any floor level as a mezzanine or "No Wind" level in the Diaphragms spreadsheet.

	Node Label	Plane	Inactive	No Wind/Drift
1	N14	ZX	<input type="checkbox"/>	<input type="checkbox"/>
2	N419	ZX	<input type="checkbox"/>	<input checked="" type="checkbox"/>

By checking this check box you are telling the program that the floor is to be ignored in the wind load calculations.



Note: When integrating a RISAFloor model into RISA-3D, this designation may only be set in the RISAFloor model. It will show up as greyed out/unavailable in RISA-3D for integrated models, so you must go back to RISAFloor if you want to change the selection.

Sloped Roof Wind Loads

When RISA-3D is integrated with RISAFloor the program is capable of automatically calculating the perpendicular roof wind loads per ASCE 7-16, Figure 27.3-1. These loads are calculated using the mean roof height for each diaphragm, the standard Gust Effect Factor (0.85), and a Roof Pressure Coefficient (C_p) for each roof plane.

The angle (θ) for each roof plane is taken as the angle between the roof plane and the horizontal plane, as projected along the Global X and Z axes. To determine a Roof Pressure Coefficient the wind load is always assumed to act "Normal to Ridge" with θ not less than 10° . Roof planes that do not meet these conditions will still be treated as though they do.

Only the most positive values of C_p are used from the table in ASCE 7-16, Figure 27.3-1. The more negative values typically result in less base shear within the Main Wind Force Resisting System, so they are ignored. Interpolation is used between values given in the table.

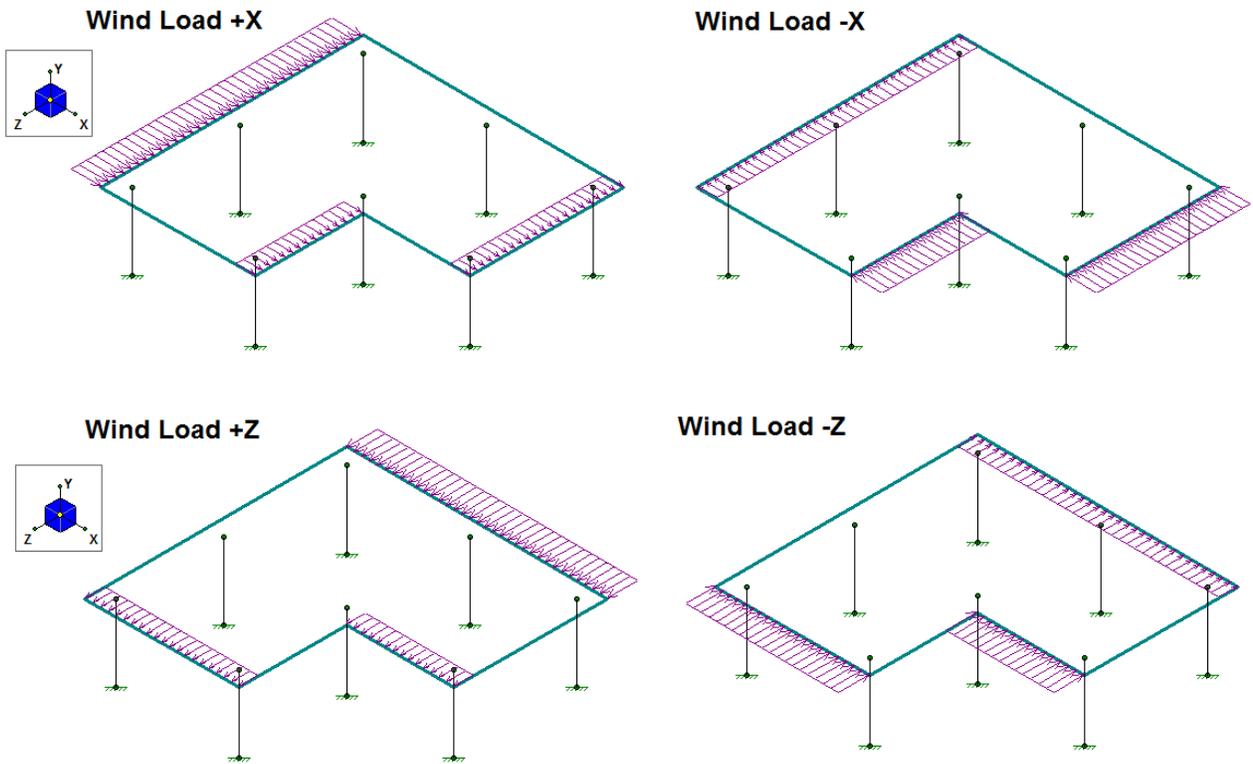
The sloped roof wind loads are applied as two-way Member Area Loads to each roof plane, and are created within roof wind load Basic Load Cases.

Semi-Rigid Wind Loads

When RISA-3D is integrated with RISAFloor the program has the ability to create Semi-Rigid wind loads and apply them to the diaphragm. The wind load is calculated by taking the Windward and Leeward forces and distributing it directly into the diaphragm as a linear load along the edge of the diaphragm. The Semi-Rigid Wind loads are placed into the Basic Load Cases shown below with distributed loads. The distributed loads are displayed per slab and the magnitude is shown but not editable. An example of the load distribution is shown on this L-shaped building.

Basic Load Cases											
	BLC Description	BLC Factor	Category	X Gravity	Y Gravity	Z Gravity	Nodal	Point	Distributed	Area(Member)	Surface(Plate/Wall)
28	Semi-Rigid Wind Load +X		WL+X								
29	Semi-Rigid Wind Load -X		WL-X								
30	Semi-Rigid Wind Load +Z		WL+Z								
31	Semi-Rigid Wind Load -Z		WL-Z								

Diaphragm Distributed Loads					
Member	Wall Panel	Diaphragm			
BLC 28: Semi-Rigid Wind...			Prev BLC	Next BLC	
	Diaphragm Label	Direction	Start Magnitude [k/ft]	End Magnitude [k/ft]	Inactive
1	S1	PX	0.035	0.035	
2	S1	PX	0.035	0.035	
3	S1	PX	0.056	0.056	



**Note:**

- In order to use these loads you must create Load Combinations using these Semi-Rigid categories. The Load Combination Generator will automatically create the LC's if the "Generate Semi-Rigid Loads?" is checked. Check out [Load Combination Generator](#) for more information.
- For a RISA-3D only model you will need to manually apply your loads and build your diaphragm with plates. See the [Semi-Rigid Diaphragms](#) section for more information on this..
- Parapet wind loads on semi-rigid diaphragms will also be added into these wind loads. Different height parapets are possible in different locations, however, the total magnitude of wind from the parapets is added uniformly to the non-parapet wind.

Other Considerations and Limitations

General

- Each diaphragm receives a full wind load, with no shielding effects considered from other diaphragms.
- The building is assumed to be fully enclosed with non-air-permeable cladding.
- Internal wind pressure is ignored. Because it must apply outwardly or inwardly to every surface simultaneously, it would cancel itself out for wall wind loading.
- Vertical uplift loads on flat roofs and flat portions of Mansard roofs are not included.
- Roof overhangs are not addressed.
- The same exposure is used for all wind directions.

ASCE 7-16

- The Directional MWFRS Procedure (Chapter 27) is used for all wind load calculations
- The "low rise" provisions of Part 2 of Chapter 27 are ignored
- Components and Cladding loads are not calculated or applied.
- The building is assumed to be "Regular Shaped" per the definition in Chapter 26
- The building is assumed to not have response characteristics making it subject to across wind loading, vortex shedding, or instability due to galloping or flutter.
- The building is assumed to be located in a site for which channeling effects and buffeting in the wake of upwind obstructions do not warrant special consideration.

ASCE 7-10

- The Directional MWFRS Procedure (Chapter 27) is used for all wind load calculations
- The "low rise" provisions of Part 2 of Chapter 27 are ignored
- Components and Cladding loads are not calculated or applied.
- The building is assumed to be "Regular Shaped" per the definition in Chapter 26
- The building is assumed to not have response characteristics making it subject to across wind loading, vortex shedding, or instability due to galloping or flutter.
- The building is assumed to be located in a site for which channeling effects and buffeting in the wake of upwind obstructions do not warrant special consideration.

ASCE 7-05

- Method 1 (Section 6.4) is not considered.
- Method 3 (Section 6.6) is not considered.
- The building is assumed to be "Regular Shaped" per the definition in Chapter 6
- The building is assumed to not have response characteristics making it subject to across wind loading, vortex shedding, or instability due to galloping or flutter.
- The building is assumed to be located in a site for which channeling effects and buffeting in the wake of upwind obstructions do not warrant special consideration.
- Special provisions for "Low-Rise Buildings" are not considered.
- Components and Cladding loads are not calculated or applied.

NBC 2015

- Both the static and dynamic wind procedures are considered.
- C_p is calculated from Figure I-15 of the User's Guide - NBC 2015 Structural Commentary.
- C_e is calculated from section 4.1.7.3.5 of the 2015 Building Code. Interpolation between low and rough terrain is not supported.
- C_g is calculated from section 4.1.7.3.8 of the 2015 Building Code. Note that for the static procedure $C_g = 2.0$ always (for buildings as a whole and main structural members).
- In the 2015 specification there is an experimental procedure as well. If your wind input criteria directs you to this procedure we will give a note letting you know this and then continue with either the static or dynamic procedure.

NBC 2010/2005

- Both the static and dynamic wind procedures are considered.
- C_p is calculated from Figure I-15 of the User's Guide - NBC 2010/05 Structural Commentaries.
- C_e is calculated from section 4.1.7.1.5 of the 2010 Building Code. Interpolation between low and rough terrain is not supported.
- C_g is calculated from section 4.1.7.1.6 of the 2010 Building Code. Note that for the static procedure $C_g = 2.0$ always.
- In the 2010 specification there is an experimental procedure as well. If your wind input criteria directs you to this procedure we will give a note letting you know this and then continue with either the static or dynamic procedure.

Material Properties

Material properties are defined on the **Material Properties Spreadsheet** and then are referred to as you build section sets, members, and plates. You can perform analysis using any type of material; simply define the properties for the material here. You can use up to 500 materials in a single model although most models will only have one or two. For example, your model might be made up of members of various grades of steel along with different concrete materials, timber or aluminum.

Material Properties Spreadsheet

The **Material Properties** spreadsheet records the material properties to be used in the model and can be accessed by selecting **Materials** from the 'Data Entry' section of the 'Explorer' panel. The entries are explained below.

Hot Rolled Steel Properties										
Hot Rolled	Cold Formed	Wood	Concrete	Masonry	Aluminum	Stainless	General			
	Label	E [ksi]	G [ksi]	Nu	Therm. Coeff. [1e ⁻⁵ *F ⁻¹]	Density [k/ft ³]	Yield [ksi]	Ry	Fu [ksi]	Rt
1	A36 Gr.36	29000	11154	0.3	0.65	0.49	36	1.5	58	1.2
2	A572 Gr.50	29000	11154	0.3	0.65	0.49	50	1.1	58	1.2
3	A992	29000	11154	0.3	0.65	0.49	50	1.1	58	1.2
4	A500 Gr.42	29000	11154	0.3	0.65	0.49	42	1.3	58	1.1
5	A500 Gr.46	29000	11154	0.3	0.65	0.49	46	1.2	58	1.1

The values for A36 steel are the default material set. You of course don't have to use the A36 properties; you can change these and also add as many other materials as you need. You can then save your preferred materials as the default materials by clicking the **Save as Defaults**  button on the **Spreadsheets** tab.

Material Properties Spreadsheet - Columns

Column	Description
Label	The Label column displays the material label you wish to use to describe the entered material properties. This label is how you will reference the set of properties later when defining section sets, members, and plates.
E	The 'E' column displays Young's modulus that describes the material stiffness.
G	The 'G' column displays the shear modulus and can be left blank if you would like it calculated automatically. The equation for "G" is: <div style="text-align: center; margin-top: 10px;"> $G = \frac{E}{2.0 * (1.0 + Poisson's\ Ratio)}$ </div>

Column	Description
	 Note: <ul style="list-style-type: none"> 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. The default Plywood material has a much lower G value than the formula above would suggest. This is because plywood is NOT an isotropic material. Also, the generic value (from the NDS or APA's Plywood Design Specification) has been reduced somewhat to account for assumed nail slip.
Nu	The 'Nu' column displays Poisson's ratio. Besides being used for the "G" calculation, this value is also used for shear deformation calculations. The value of Poisson's ratio may not exceed '0.5'.
Therm	The Therm column displays the coefficient of thermal expansion and is entered per 10 ⁵ (100,000) degrees. This coefficient is used in the calculation of thermal loads.
Density	The Density column displays the material density and is used in the calculation of the member and plate self weight. This density times the area gives the self weight per unit length for the members; density times plate volume gives the total weight of a given plate.
Yield	The Yield column displays the yield stress and is used only for Hot Rolled and Cold Formed steel design.

Hot Rolled Specific Material Data

The **Hot Rolled** tab records a number of hot rolled steel specific material properties that do not exist for the other materials. These entries are described below:

Hot Rolled Steel Properties										
Hot Rolled	Cold Formed	Wood	Concrete	Masonry	Aluminum	Stainless	General			
	Label	E [ksi]	G [ksi]	Nu	Therm. Coeff. [1e ⁵ F ⁻¹]	Density [k/ft ³]	Yield [ksi]	Ry	Fu [ksi]	Rt
1	A36 Gr.36	29000	11154	0.3	0.65	0.49	36	1.5	58	1.2
2	A572 Gr.50	29000	11154	0.3	0.65	0.49	50	1.1	58	1.2
3	A992	29000	11154	0.3	0.65	0.49	50	1.1	58	1.2
4	A500 Gr.42	29000	11154	0.3	0.65	0.49	42	1.3	58	1.1
5	A500 Gr.46	29000	11154	0.3	0.65	0.49	46	1.2	58	1.1

Hot Rolled Specific Material Data

Column	Description
Ry	The 'Ry' column displays the ratio of the expected yield stress to the specified minimum yield stress. This value is used in determining the required strength of an element for the seismic detailing checks.
Fu	The 'Fu' column displays the specified minimum tensile strength. This value is used per <i>AISC 358</i> to calculate C_{pr} which is in turn used to calculate the probable maximum moment at a plastic hinge in the seismic detailing calculations.
Rt	The 'Rt' column is currently not used by the program. In a future version of the program this value will be used to calculate the expected tensile strength for the seismic detailing checks.

Cold Formed Specific Material Data

The **Cold Formed** tab records a cold formed material property that may not exist for the other materials. This entry is described below:

Cold Formed Steel Properties								
Hot Rolled	Cold Formed	Wood	Concrete	Masonry	Aluminum	Stainless	General	
	Label	E [ksi]	G [ksi]	Nu	Therm. Coeff. [$10^{-6}F^{-1}$]	Density [k/ft ³]	Yield [ksi]	Fu [ksi]
1	A570_33	29500	11346	0.3	0.65	0.49	33	52
2	A607_C1_55	29500	11346	0.3	0.65	0.49	55	70

Cold Formed Specific Material Data

Column	Description
Fu (Ultimate)	The 'Fu' column displays the ultimate tensile stress.

Wood Specific Material Data

The **Wood** tab records a number of wood specific material properties that do not exist for the other materials. These entries are described below:

Wood Material Properties											
Hot Rolled	Cold Formed	Wood	Concrete	Masonry	Aluminum	Stainless	General				
Label	Type	Database	Species	Grade	Cm	Ci	Emod	Nu	Therm. Coeff. [1e ⁻⁵ F ⁻¹]	Density [k/ft ³]	
1	DF/SPine	Solid Sawn	Visually Graded	Com Species Group I DF,SP	No.1	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035
2	HF/Spruce Fir	Solid Sawn	Visually Graded	Com Species Group II HF,SPF	No.1	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035
3	DF	Solid Sawn	Visually Graded	Douglas Fir-Larch	No.1	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035
4	SP	Solid Sawn	Visually Graded	Southern Pine	No.1	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035
5	HF	Solid Sawn	Visually Graded	Hem-Fir	No.1	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035
6	SPF	Solid Sawn	Visually Graded	Spruce-Pine-fir	No.1	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035
7	24F-1.8E DF...	Glulam	NDS Table 5A	24F-1.8E_DF_BAL	na	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035
8	24F-1.8E DF...	Glulam	NDS Table 5A	24F-1.8E_DF_UNBAL	na	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035
9	24F-1.8E SP...	Glulam	NDS Table 5A	24F-1.8E_SP_BAL	na	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035
10	24F-1.8E SP...	Glulam	NDS Table 5A	24F-1.8E_SP_UNBAL	na	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035
11	LVL-PRL Co...	Custom	N/A	LVL_PRL_1.5E_2250F	na	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035
12	LVL-PRL Co...	Custom	N/A	LVL_PRL_2.0E_2900F	na	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035

Wood Specific Material Data

Column	Description
Type	The Type column displays the available wood types; currently, there are four: Solid Sawn, Glulam, SCL (this includes LVLs), and Custom.
Database	The Database column displays the appropriate Database menu, filtered by your Type choice.
Species	The Species column displays the appropriate Species menu, based on your Database selection. This is the wood species designation from your selected design code.
Grade	The Grade column displays the wood grade designation from your selected design code. This has a drop down list where you can select the appropriate grade.
Cm	The 'Cm' check-box determines if the wet service / moisture content factor should be applied. If you put a check in the Cm field, the appropriate factors will be applied to the allowable stresses and Young's Modulus (E).
Ci	The 'Ci' check-box determines if the incision factor should be applied. If you put a check in the Ci field, the appropriate factors will be applied to the allowable stresses and Young's Modulus (E).
Emod	The Emod column displays the factor that is applied to the Young's modulus modifier to reflect the NDS Appendix F criteria. This is not applicable to the CSA O86 design code.



Note: Refer to [Custom Wood](#) to specify a custom wood material.

Commercial Species Groups

When either the NDS 05/08, 12, 15, or 2018 wood design code is selected, the Solid Sawn Species menu lists two new species listed that are not specifically shown in the NDS code. These are the *Commercial Species Group I - DF/SP* and *Commercial Species Group II - HF/SPFe Fir*. These are meant to be simplified groupings of the most commonly used wood species. It is meant to simplify the selection procession for wood member design in the United States.

The design values for Group I take the NDS allowable stress values for two of the most widely used species (Doug Fir-Larch and Southern Pine) and uses the lower bound allowable stress value for each size and grade. To verify exactly what base allowable stress values were used, you can review the properties in the Shape Selection window.

Similarly, the design values for Group II take the allowable stress values for Hem Fir and Spruce-Pine-Fir and use the lower bound allowable stress values for each size and grade. To verify exactly what base allowable stress values were used, you can review the properties in the Shape Selection window.

Concrete Specific Material Data

The **Concrete** tab records a few concrete specific material properties that do not exist for the other materials. These entries are described below:

Concrete Properties										
Hot Rolled	Cold Formed	Wood	Concrete	Masonry	Aluminum	Stainless	General			
	Label	E [ksi]	G [ksi]	Nu	Therm. Coeff. [$1e^{-6}F^{-1}$]	Density [k/ft ³]	f'c [ksi]	Lambda	Flex Steel [ksi]	Shear Steel [ksi]
1	Conc3000NW	3156	1372	0.15	0.6	0.145	3	1	60	60
2	Conc3500NW	3409	1482	0.15	0.6	0.145	3.5	1	60	60
3	Conc4000NW	3644	1584	0.15	0.6	0.145	4	1	60	60
4	Conc3000LW	2085	907	0.15	0.6	0.11	3	0.75	60	60
5	Conc3500LW	2252	979	0.15	0.6	0.11	3.5	0.75	60	60
6	Conc4000LW	2408	1047	0.15	0.6	0.11	4	0.75	60	60

Concrete Specific Material Material Data

Column	Description
f'c	The 'f'c' column displays the concrete compressive strength used for concrete design.
Lambda	The 'Lambda' column displays the lightweight concrete modification factor. This factor only applies to the <i>ACI 318-14</i> , <i>ACI 318-11</i> , <i>ACI 318-08</i> and <i>CSA A23.3-04</i> codes. For all other codes the Density value of the material determines any strength reduction. The program automatically calculates the correct value if it is left blank. Only values between 0.75 and 1.0 will be considered.
Flex Steel	The Flex Steel column displays the reinforcement yield strength for

Column	Description
	flexural bars in members and vertical bars in walls. Shear Steel is the reinforcement yield strength for shear bars in members and horizontal bars in walls.

Masonry Specific Material Data

The **Masonry** tab records masonry specific material properties. Here we describe items that do not exist for other materials.

Masonry Properties									
Hot Rolled Cold Formed Wood Concrete Masonry Aluminum Stainless General									
	Label	E [ksi]	G [ksi]	Nu	Therm. Coeff. [1e ⁻⁵ *F ⁻¹]	Self Weight [k/ft ³]	f'm [ksi]	Flex Steel [ksi]	Shear Steel [ksi]
1	Concrete Matl	1350	540	0.25	0.6	Custom	1.5	60	60
2	Clay Matl	1050	420	0.25	0.6	Custom	1.5	60	60

Masonry **Self Weight** can be accounted for by two different methods. Entering a number for the density will result in masonry walls which have a self weight equal to that density multiplied by the wall cross-sectional area. Otherwise, click within the cell to launch the masonry self weight dialog:

3D Masonry Self Weight

Calculate Self Weight of Wall Panels

Using a Uniform Density k/ft³

Using Block and Grout Properties

Block Material

Grout Weight

Set the Using **Block and Grout Properties** option to have the program automatically calculate the self weight of the wall using the weights from tableB3 of the Reinforced Masonry Engineering Handbook (RMEH). The self weight will be listed as **Custom**.

Masonry Specific Material Data

Column	Description
f'm	The 'f'm' column displays the masonry compressive strength used for masonry design.
Flex	The Flex displays the rebar yield strengths for flexural bars used to reinforce the masonry.
Shear Steel	The Shear Steel displays the rebar yield strengths for shear bars used to reinforce the masonry.



Note:

- Masonry Shear Steel is automatically set to the Flex steel strength.
- Masonry materials are used only for walls in wall footings. See the Wall Footing Definitions spreadsheet for more information.

Aluminum Specific Material Data

The **Aluminum** tab records a number of aluminum specific material properties that do not exist for the other materials. These entries are described below:

Hot Rolled	Cold Formed	Wood	Concrete	Masonry	Aluminum	Stainless	General							
Label	E [ksi]	G [ksi]	Nu	Therm. Coeff. [1e ⁻⁵ *F ⁻¹]	Density [k/ft ³]	Table B.4	kt	Ftu [ksi]	Fty [ksi]	Fcy [ksi]	Fsu [ksi]	Ct		
1	3003-H14	10100	3787.5	0.33	1.3	0.173	Table B.4-1	1	19	16	13	12	141	
2	6061-T6	10100	3787.5	0.33	1.3	0.173	Table B.4-2	1	38	35	35	24	141	
3	6063-T5	10100	3787.5	0.33	1.3	0.173	Table B.4-2	1	22	16	16	13	141	
4	6063-T6	10100	3787.5	0.33	1.3	0.173	Table B.4-2	1	30	25	25	19	141	
5	5052-H34	10200	3787.5	0.33	1.3	0.173	Table B.4-1	1	34	26	24	20	141	
6	6061-T6...	10100	3787.5	0.33	1.3	0.173	Table B.4-1	1	24	15	15	15	141	

Aluminum Specific Material Data

Column	Description
Table B.4	<p>The Table B.4 column displays the table that refers to the Buckling Constants for a series of Temper designations. This column has a drop down list where you can pick from Table B.4.1 equations or Table B.4.2 equations.</p> <div style="border: 1px solid black; padding: 5px; margin-top: 10px;"> <p>Note: This table is named Table 3.3 in ADM 05 and this column has a drop down list where you can pick from Table 3.3-3 equations or Table 3.3-4 equations.</p> </div>
k_t	The 'k _t ' column refers to coefficient for tension members listed in the ADM 10 Table A.3.3.

Column	Description
Ftu	The 'Ftu' column displays the specified tensile ultimate strength.
Fty	The 'Fty' column displays the specified tensile yield strength.
Fcy	The 'Fcy' column displays the specified compressive yield strength.
Fsu	The 'Fsu' column displays the specified shear ultimate strength.
Ct	The 'Ct' column refers to Buckling Constant Intersection for Axial Compression in Curved Elements per Table B.4.1 and Table B.4.2 ADM 10.

Stainless Steel Specific Material Data

The **Stainless** tab records stainless steel material properties. These entries are described below:

Stainless Steel Properties									
Hot Rolled	Cold Formed	Wood	Concrete	Masonry	Aluminum	Stainless	General		
	Label	E [ksi]	G [ksi]	Nu	Therm. Coeff. [$1e^{-5}F^{-1}$]	Density [k/ft ³]	n	Yield [ksi]	Fu [ksi]
1	SS1	1.931e+5	74325.484	0.3	1.674	8.009	5.6	206.843	517.107

Stainless Steel Specific Material Data

Column	Description
n	The 'n' column displays the specified Ramberg Osgood parameter used for determining secant modulus. See AISC DG27 Table 6-1.
Yield	The Yield column displays the specified yield stress.
Fu	The 'Fu' column displays the specified ultimate tensile stress.

General Material Data

The **General** tab records general material properties. These entries are described below:

General Materials Properties							
Hot Rolled	Cold Formed	Wood	Concrete	Masonry	Aluminum	Stainless	General
	Label	E [ksi]	G [ksi]	Nu	Therm. Coeff. [$10^{-5}F^{-1}$]	Density [k/ft ³]	Plate Methodology
1	gen_Conc3NW	3155	1372	0.15	0.6	0.145	Isotropic
2	gen_Conc4NW	3644	1584	0.15	0.6	0.145	Isotropic
3	gen_Conc3LW	2085	906	0.15	0.6	0.11	Isotropic
4	gen_Conc4LW	2408	1047	0.15	0.6	0.11	Isotropic
5	gen_Alum	10600	4077	0.3	1.29	0.173	Isotropic
6	gen_Steel	29000	11154	0.3	0.65	0.49	Isotropic
7	RIGID	1e+6		0.3	0	0	Isotropic

Cold Formed Specific Material Data

Column	Description
Plate Methodology	The Plate Methodology column displays the behavior to use for plate analysis: isotropic or orthotropic. For more information, see Orthotropic Behavior .

Material Take-Off

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

Material Takeoff					
	Material	Size	Pieces	Length[ft]	Weight[K]
1	General Members				
2	gen_Conc3NW	RE24X24	1	12	6.96
3	RIGID		14	49.2	0
4	Total General		15	61.2	6.96
5					
6	Hot Rolled Steel				
7	A36 Gr.36	LL8X8X8X3	9	156.5	8.363
8	A36 Gr.36	W18X71	1	12	0.853
9	A992	W10X12	15	268.5	3.234
10	A992	W10X17	6	108	1.834
11	A992	W12X40	7	84	3.344
12	Total HR Steel		38	629.1	17.629
13					
14	Cold Formed Steel				
15	A607_C1_55	8Z52.75X105	1	9	0.048
16	A607_C1_55	2-4CS2X059-...	1	9	0.033
17	Total CF Steel		2	18	0.081
18					
19	Wood				
20	24F-1.8E SP Bala...	2.5X5.5FS	1	18	0.06
21	DF/SPine	2-4X4	3	54	0.322
22	DF/SPine	2-2X4	3	54.7	0.14
23	DF/SPine	3X3	30	73.7	0.112
24	DF/SPine	2X8	10	90	0.238
25	Total Wood		47	290.5	0.871
26					
27	Masonry Walls			Volume (y...	
28	Concrete Matl		2	11.6	25.025
29	Total Masonry		2	11.6	25.025
30					
31	Aluminum				
32	6061-T6	12.000ODX0....	4	97.3	1.599
33	6061-T6	RT1X1X0.065	2	20	0.006
34	6061-T6	RT1X1X0.125	2	20	0.011
35	6061-T6	RT2.75X2.75X...	1	3.3	0.005
36	6061-T6	RT3X3X0.125	1	3.3	0.006
37	6061-T6	RT3X4X0.188	2	6.7	0.02
38	6061-T6	RT4X4X0.125	4	75.5	0.176
39	6061-T6	RT6X6X0.125	4	49.9	0.176
40	Total Aluminum		20	276	1.998
41					
42	Concrete Memb...			Volume (y...	
43	Conc3000NW	CRECT12X10	7	2.6	10.15
44	Conc4000NW	CRECT12X8	6	3.2	12.615
45	Conc4000NW	CRECT8X8	8	2.4	9.28
46	Concrete Walls				
47	Conc3000NW		1	4.8	18.85
48	Total Concrete		22	13	50.895
49					
50	Plate Elements	Thickness (in)		Volume (y...	
51	gen_Conc4NW	8	320	31.6	123.733
52	Total Plates		320	31.6	123.733

This spreadsheet shows material takeoff information for each material and shape in the model. For each material the shapes are listed with the total length and weight. The total length and weight for each material is summed beneath the listing of shapes.

The length listed is the sum of the lengths of all the members assigned to this shape. Member end offsets are deducted from the length. The weight is the sum of the self-weight for all elements assigned to section set. This weight is calculated as $\text{Area} * \text{Density} * \text{Length}$ for each member, again with offset distances deducted from the length.

For Concrete members and plates, the volume is shown rather than the length of the members.

**Note:**

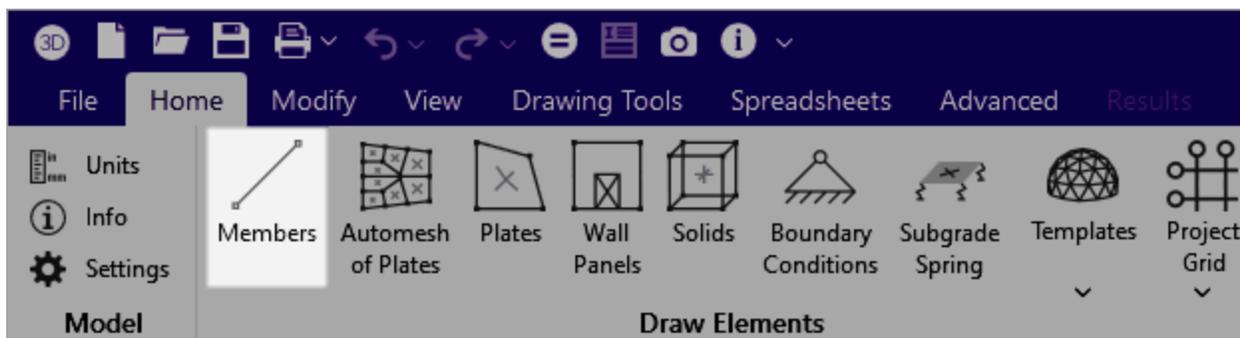
- This material takeoff report is independent of the loads applied to the model, i.e. the applied loads do not influence this report.
- The number of pieces is reflective of the number of members of a particular size. For example, one double angle will be counted as one piece, with the length being the member length, and the weight being that of the double angle.
- Plates are also included in the Material Take-Off.

Members

RISA-3D uses a [Physical Member](#) that is automatically sub-meshed into general-purpose beam elements. With the use of the member [End Releases](#) you may define truss members or any other end condition for that member. Member data may be viewed and edited in three ways: graphically in a model view, in the **Member Properties Panel**, or in the **Members Spreadsheet**. See [Members Spreadsheet](#) for descriptions of the member data. Design parameters for steel, wood or concrete design are recorded on the material tabs of the Members spreadsheet and are discussed in the [Hot Rolled Steel - Design](#), [Cold Formed Steel - Design](#), [Stainless Steel - Design](#), [Aluminum - Design](#), [Concrete - Design](#), and [Wood - Design](#) sections.

Draw Members

To create new members you can draw them using a [Drawing Grid](#), [Project Grid](#), or drawing "dot to dot" clicking existing joints. The member properties are specified in the Properties panel. You can set all of the member properties up front or you can modify these properties after you draw the member. Graphically modifying properties is discussed in the next section.



Properties	
Member ^	
Label Prefix	M
Draw Option	Node to Node v
General Properties ^	
Member Type	Beam v
Material Type	Hot Rolled Steel v
I Release	...
J Release	...
Section/Shape	Shape Database v
Shape	W8X10 ...
Material	A36 Gr.36 v ...
Design List	Wide Flange v ...
Design Rule	Typical v ...

The Member Drawing Options are as follows:

Member Drawing Options

Option	Description
Draw Node to Node	Use the first mouse click to select the start node of the new member. Use the second mouse click to select the end node of the new member. The tool will continue to draw node to node until you right-click to disengage.
Draw Node to Member/Wall	Use the first mouse click to select the start node of the new member. Use the second mouse click to select an existing member which would contain the end node of the new member. 'Offset' is the offset of the end point of the new member from the start node of the existing member.
Draw Member to Member	Use the first mouse click to select an existing member which would contain the start node of the new member. Use the second mouse click to select an existing member which would contain the end point of the new member. '1st Offset' and '2nd Offset' are the location offsets of the start and end nodes of the new member from the start nodes of the existing members.
Draw Wall to Wall	Use the first mouse click to select an edge of an existing wall panel which will contain the start node of the new member. Use the

Option	Description
	second mouse click to select an edge of an existing wall panel which will contain the end node of the new member. '1st Offset' and '2nd Offset' are the location offsets of the start and end nodes of the new member from the corner of the selected wall panel edge.
Draw Member/Wall to Node	Use the mouse click to select the existing member which would contain the start node of the new member. The new member would have a length equal to 'Length'. The new member would be created at an angle 'Relative Angle' to the projection of the existing member on the horizontal plane (plane perpendicular to the vertical axis).

To Draw Members

To draw Members:

1. If there is not a model view already open then go to the **View** ribbon.
2. Click the **Open 3D Views** icon in the 'Window' section to open a new view.
3. If you are not drawing between existing nodes, you need to create a drawing grid or define nodes in the **Node Coordinates** spreadsheet.
4. Go to the **Home** ribbon.
5. Click the **Members** icon in the 'Draw Elements' section.
6. In the **Properties** panel, set the member properties .

You can specify the General Properties, Additional Properties, RISAConnection Properties, and Design Properties.

7. To start drawing members, click on nodes or grid intersections with the left mouse button. The coordinates of the closest node or grid intersection to your cursor are displayed on the screen near your cursor.

The first click defines the **I-end** of the first member. The second click, and each click thereafter, defines the **J-end** of the first member and also the **I-end** of the next member so that you can continue to draw. To stop drawing, click the right mouse button. You can then start drawing somewhere else with the left button.

The new members are shown on screen and recorded in the **Members** spreadsheet.

8. To stop drawing altogether click the right mouse button or press the **Esc** key.



Note:

- You may also specify or edit members in the **Members** spreadsheet.
- You may undo any mistakes by clicking the  **Undo** icon in the Quick Access toolbar.

Modify Members

There are a number of ways to modify members.

- You can view and edit the member data in the [Members Spreadsheet](#).
- You can also click a single member or select multiple members to view and edit its properties in the **Member Properties Panel**.
- You can use the tools in the **Modify** ribbon to graphically modify the members in the 3D View.

The **Properties Panel** for members is discussed here which lets you modify the properties of members that already exist in your model. To use this, you will select the members that you want to modify, select the member type from the **Selection Properties** dropdown, then specify in the **Properties** panel the parameters you want changed. See the [Graphic Selection](#) topic for more on selecting.

MemberProperties

The parameters in the **MemberProperties** section are the member label and length. The member length is defined by the start and end node and cannot be edited from the Properties panel. You can modify members graphically in the 3D View using tools on the **Modify** ribbon to modify the length.

Properties	
Member	^
Label	F1_M36
Length, ft	30

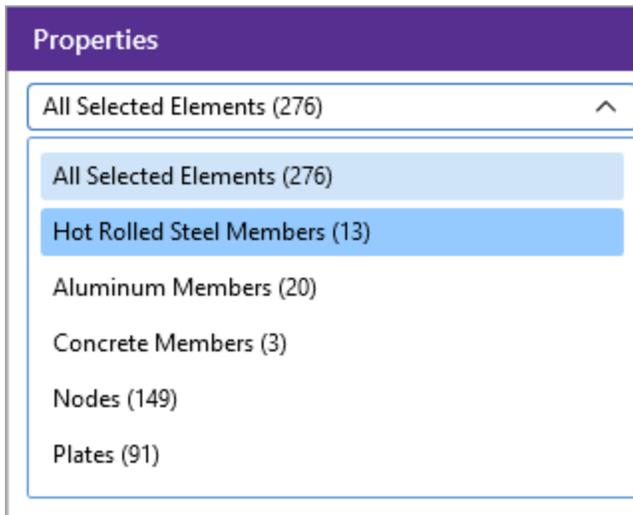


Note: To relabel members first sort the **Members** spreadsheet into the desired order, then select the **Modify** tab ► **Relabel All** (or **Relabel Selected**) button ► **Members**.

To Modify Member Properties

To modify Members properties from the Properties panel:

1. If there is not a model view already open then go to the **View** ribbon.
2. Click the **Open 3D Views** icon to open a new view.
3. Click a single member or select multiple members that you wish to modify.
4. If you have multiple elements selected, specify which material type to view properties for using the **Selection Properties** dropdown in the **Properties** panel.



5. Set the parameters for the members within the **Properties** panel.
If certain properties vary within your selection of members, the field will say Various.
6. You may choose to modify a single member at a time or an entire selection of members.
To modify a single member, click on the member in the **3D View** to pull up the properties in the **Properties** panel.
To modify a selection of members, select multiple elements in the **3D View** and specify which type to view properties for using the **Selection Properties** dropdown at the top of the **Properties** panel.



Note: You may undo any mistakes by clicking the **Undo**  button.

General Properties

In the **General Properties** section of the **Properties** panel, you can specify the member parameters such as member end release, material, and shape size. The parameters shown can be found in the **Primary** and **Advanced** tab of the **Members** spreadsheet.

You can assign a shape by either using a Section Set or choosing a specific shape from the Shape Database.

If you assign a shape using a Section Set, several parameters like shape size and material will gray out and the displayed values will be based on the Section Set. These parameters can be found in the **Section Sets** spreadsheet.

You can modify the general properties based on material type for a single member by clicking on a member or for multiple members by selecting multiple members and selecting a material type from the **Selection Properties** dropdown in the **Properties** panel.

General Properties		^
Member Type	Beam	▼
Material Type	Hot Rolled Steel	▼
I Release	BenPIN	⋮
J Release	BenPIN	⋮
Section/Shape	Shape Database	▼
Shape	W27X84	⋮
Material	A992	▼ ⋮
Design List	Wide Flange	▼ ⋮
Design Rule	Typical	▼ ⋮

**Note:**

- You may also modify members in the **Members** spreadsheet.
- You can add a new Section Set by setting the Assign Shape parameter to Section Set and clicking the triple dot next to the Section Set dropdown.

Additional Properties

In the **Additional Properties** section of the **Properties** panel, you can specify the member parameters such as member start and end nodes, physical or non physical member properties, and analysis offset. The parameters shown can be found in the **Primary** and **Advanced** tab of the **Members** spreadsheet.

You can modify the general properties based on material type for a single member by clicking on a member or for multiple members by selecting multiple members and selecting a material type from the **Selection Properties** dropdown in the **Properties** panel.

Additional Properties	
I Node	F1_N22
J Node	F1_N23
K Node	
Rotate (deg)	
T/C Only	Both Ways <input type="checkbox"/>
I Offset, in	
J Offset, in	
Physical	<input checked="" type="checkbox"/>
Analysis Offset, in	<input type="text"/>
Activation	Active <input type="checkbox"/>
Start Support	Default <input type="checkbox"/>
End Support	Default <input type="checkbox"/>



Note: You may also specify or edit design parameters in the **Advanced** tab of the **Members** spreadsheet.

RISAConnection Properties

In the **RISAConnection Properties** section of the **Properties** panel, you can view and define the connection rules for your hot-rolled steel connection design to be used in conjunction with RISAConnection. The connection rule parameters can be found in the **RISAConnection** tab of the **Members** spreadsheet. See the [RISAConnection Integration](#) topic for more information.

RISAConnection Properties	
Start Conn	Col/Bm Clip Angle <input type="checkbox"/>
End Conn	Col/Bm Shear Tab <input type="checkbox"/>

Design Properties

In the **Design Properties** section of the **Properties** panel, you can specify the Hot Rolled, Cold Formed, Wood, Concrete, Aluminum, or Stainless design parameters such as unbraced lengths and K factors. See [Steel Design](#), [Cold Formed Steel Design](#), [Stainless Design](#), [Concrete Design](#), [Aluminum Design](#), and [Wood Design](#) for information on the design parameters themselves. The parameters shown are the same as those on the Hot Rolled, Cold Formed, Wood, Concrete,

Aluminum, or Stainless tab of the Members spreadsheet. The Seismic Design Rule can be found on the [Advanced](#) tab of the Members spreadsheet and only pertains to hot rolled steel members.

You can modify the design parameters based on material type for a single member by clicking on a member or for multiple members by selecting multiple members and selecting a material type from the **Selection Properties** dropdown in the **Properties** panel.

Design Properties	
Lb y-y, ft	Segment
Lb z-z, ft	
Lcomp top, ft	1
Lcomp bot, ft	
L-Torque, ft	
K y-y	...
K z-z	...
Channel Conn.	N/A
a, ft	N/A
Cb	
Function	Lateral
Seismic DR	None



Note: You may also specify or edit design parameters in the **Material Design Parameters** tab of the **Members** spreadsheet.

Material and Cross Section Properties

Member material and cross section properties may be assigned to a member in one of two ways – either directly by specifying the shape and material explicitly, or by assigning a Section Set to the member. Section Sets allow you to control properties for a group of members that share the same properties. Combining section sets with the Member Redesign feature gives you great control over how new member sizes are picked and what members get updated. Section sets must be used when the desired shape is not in the database or when steel redesign (optimization) is desired. See [Section Sets](#) for more information.

All member properties may be assigned in the Properties panel either as you draw or later as a modification to the members.

Members Spreadsheet

Primary Data

The **Members Spreadsheet** records the properties for the member elements and may be accessed by selecting **Members** on the **Spreadsheets Menu**.

Member Primary Data											
Primary	Advanced	Hot Rolled	Cold Formed	Wood	Concrete Beam	Concrete Column	Aluminum	Stainless	RISAConnection		
	Label	I Node	J Node	K Node	Rotate(deg)	Section/Shape	Type	Design List	Material	Design Rule	
1	F2_M35	F2_N20	F2_N21			W27X84	Beam	Wide Flange	A992	Typical	
2	F2_M36	F2_N22	F2_N23			W30X90	Beam	Wide Flange	A992	Typical	
3	F2_M37	F2_N24	F2_N25			W24X62	Beam	Wide Flange	A992	Typical	
4	F2_M38	F2_N26	F2_N27			W24X76	Beam	Wide Flange	A992	Typical	
5	F1_M35	F1_N20	F1_N21			W24X62	Beam	Wide Flange	A992	Typical	
6	F1_M36	F1_N22	F1_N23			W27X84	Beam	Wide Flange	A992	Typical	
7	F1_M37	F1_N24	F1_N25			W24X62	Beam	Wide Flange	A992	Typical	
8	F1_M38	F1_N26	F1_N27			W27X84	Beam	Wide Flange	A992	Typical	
9	F1_CS...	N33	N78			W14X68	Column	WF 14	A992	Typical	
10	F1_CS...	N34	F2_N21			W14X61	Column	WF 14	A992	Typical	
11	F1_CS...	N35	N79		90	W14X61	Column	WF 14	A992	Typical	

The following data columns hold the **Primary** data for the members:

Members Spreadsheet - Primary Data

Column	Description
Member Labels	<p>You can assign a unique Label to any or all of the members. You can then refer to the member by its label. Each label has to be unique, so if you try to enter the same label more than once, you get an error message.</p> <p>You can relabel members at any time with the Relabel Members option on the Tools Menu.</p>
Member End Joints	<p>The I-Joint and J-Joint entries define the start (I-joint) and end (J-joint) locations of the member. The member local axes are defined based on these joints. See Member Local Axes for more information.</p>
K-Joint and X-axis rotation	<p>These two parameters may be used separately or together to define the rotation of a member. See Member Local Axes and Defining Member Orientation for more information.</p>
Section / Shape	<p>If you are explicitly assigning shapes to each member, then enter the database shape you wish to use for the member. You can select this by clicking on the arrow in the cell. Alternatively, you may choose a section set to represent the section properties, material properties, and re-design parameters.</p>

Column	Description
Member Type	<p>If you are explicitly assigning shapes to each member, then you may enter the member type that you wish to use. The choices are <i>Column</i>, <i>Beam</i>, <i>Vertical Brace</i>, and <i>Horizontal Brace</i>.</p> <p>Here are the main effects that the member type will have on your structure:</p> <ul style="list-style-type: none"> • If you are using concrete, this defines the rebar layout (column vs beam) . • If you are using design lists, they specifically reference the member type. • If you are using member area loads, loads will not be attributed to members defined as Hbraces or Vbraces. • If you are using flexible diaphragms, transient loads will not be attributed to members defined as None, Columns or Vbraces. • If you are using the RISA-Revit link the link will not work properly, unless you use member types. • If you are integrating with RISAConnection, the connection validation requires proper use of member types. • If you are using the seismic detailing provisions, you must use proper member types. <div data-bbox="493 1161 1409 1304" style="border: 1px solid black; padding: 5px; margin-top: 10px;">  Note: If you are using Section Sets to define your member, the information in this field is generated automatically based on the referenced section set. </div>
Design List	<p>If you are explicitly assigning shapes to each member, then you may enter the design list type that you wish to use. This entry will affect the members that are available to program when it is suggesting alternate or optimized shapes. Refer to Design Optimization for more information on the member optimization procedure. Also refer to Appendix A – Redesign Lists for information on creating or editing these lists.</p> <div data-bbox="493 1549 1409 1692" style="border: 1px solid black; padding: 5px; margin-top: 10px;">  Note: If you are using Section Sets to define your member, the information in this field is generated automatically based on the referenced section set. </div>
Material	<p>If you are explicitly assigning shapes to each member, then you may enter the material that you wish to use.</p> <p>You can select this by clicking on the arrow in the cell.</p>

Column	Description
	 Note: If you are using Section Sets to define your member, the information in this field is generated automatically based on the referenced section set.
Design Rules	If you are explicitly assigning shapes to each member, you can enter the design rules type that you wish to use. When the program is checking alternate or optimized shapes, it will restrict its selections to members that obey the chosen design rules. Refer to Design Rules– Size / U.C. for more information.
	 Note: If you are using Section Sets to define your member, then the information in this field will be generated automatically based on the referenced section set.

Members Spreadsheet - Advanced Data

The **Advanced** tab records the properties for the member elements and may be accessed by selecting **Members** on the **Spreadsheets Menu**.

Member Advanced Data												
Primary	Advanced	Hot Rolled	Cold Formed	Wood	Concrete Beam	Concrete Column	Aluminum	Stainless	RISAConnection	BRB		
	Label	I Release	J Release	Col-Wall Vert Release	I Offset [in]	J Offset [in]	T/C Only	Physical	Deflection Ratio Options	Analysis Offset [in]	Activation	Seismic DR
4	F2_M38	BenPIN	BenPIN	<input type="checkbox"/>	6.95	6.95		<input checked="" type="checkbox"/>	N/A			None
5	F1_M35	BenPIN	BenPIN	<input type="checkbox"/>	7	6.95		<input checked="" type="checkbox"/>	N/A			None
6	F1_M36	BenPIN	BenPIN	<input type="checkbox"/>	6.95	6.95		<input checked="" type="checkbox"/>	N/A			None
7	F1_M37	BenPIN	BenPIN	<input type="checkbox"/>	6.95	6.95		<input checked="" type="checkbox"/>	N/A			None
8	F1_M38	BenPIN	BenPIN	<input type="checkbox"/>	6.95	6.95		<input checked="" type="checkbox"/>	N/A			None
9	CS20 (C-5)_L1			<input checked="" type="checkbox"/>				<input checked="" type="checkbox"/>	** NA **			None
10	CS21 (E-5)_L1			<input checked="" type="checkbox"/>				<input checked="" type="checkbox"/>	** NA **			None
11	CS22 (F-1)_L1			<input checked="" type="checkbox"/>				<input checked="" type="checkbox"/>	** NA **			None
12	CS23 (F-2)_L1			<input type="checkbox"/>				<input checked="" type="checkbox"/>	** NA **			None
13	CS24 (F-6)_L1			<input type="checkbox"/>				<input checked="" type="checkbox"/>	** NA **			None
14	CS25 (F-8)_L1			<input type="checkbox"/>				<input checked="" type="checkbox"/>	** NA **			None

The following data columns hold the **Advanced** data for the members:

Member Spreadsheet - Advanced Data

Column	Description
I and J Releases	The I and J Releases control the forces that may be resisted by a member. You can use these to define pinned connections, truss members, and any other end condition. See Member End Releases for more information.

Column	Description
Col-Wall Vert Release	This option is only visible and applicable for columns transferred from RISAFloor. It defines the vertical connectivity between the column and the connected wall panel. If checked, the column and wall panel will be decoupled in the vertical direction during analysis. See Col-Wall Vert Release for more information.
I and J Offsets	The I and J End offsets may be used to model a rigid end zone for a member. See Member End Offsets for more information.
T/C Only	The T/C Only field is used to indicate that a member is to be Tension or Compression only. When a member is flagged as C , any members it will only be able to take compressive loads. The member will have no stiffness to resist tensile loads. When a member is flagged as T , the member will only be able to take tension loads. When a section is flagged as E , the member will primarily take only tension loads, however it will also take some compression load, up to its Euler buckling load.
Physical	The Physical box is checked if the member is a Physical Member. See Physical Members for more information.
Deflection Ratio Options	The Deflection Ratio Options are only applicable to members designated as beams. This field is used to designate the member end behavior either as supported or as a cantilever for the deflection ratio. If this field is blank, deflection ratio will be based on original member behavior. See Deflection Ratio Options to learn more.
Analysis Offset	Members may be offset from their centerline for analysis. This is useful for members such as single angles bolted or welded on only one leg, where axial compression also needs to induce bending. See Analysis Offset to learn more.
Activation (Inactive Members)	Members may be removed from the solution without deleting them from your model by making them inactive. See Inactive and Excluded Elements for more information.
Seismic DR (Design Rules)	Seismic Design Rules can be used to assign a Seismic Design Rule to each member individually. This can be left as None if you are not including seismic detailing in your design. This entry will only apply to Hot Rolled Steel members.

Members Spreadsheet - Detailing Data

The **Detailing** tab records the detailing data for the members that are necessary for full 2-way data transfer between RISA and steel detailing packages. For more information on this subject refer to the Help file for the RISA CIS/2 Translator which can be downloaded from our [website](#).

Member Detailing Data									
Primary	Advanced	Hot Rolled	Cold Formed	Wood	Concrete Beam				
Concrete Column	Aluminum	Stainless	RISACONNECTION			Detailing			
Label	I Cardinal Point	I x Offset [in]	I y Offset [in]	I z Offset [in]	J Cardinal Point	J x Offset [in]	J y Offset [in]	J z Offset [in]	
1	F2_M35	10- geometric centroid	0	0	0	10- geometric centroid	0	0	0
2	F2_M36	10- geometric centroid	0	0	0	10- geometric centroid	0	0	0
3	F2_M37	10- geometric centroid	0	0	0	10- geometric centroid	0	0	0
4	F2_M38	10- geometric centroid	0	0	0	10- geometric centroid	0	0	0
5	F1_M35	10- geometric centroid	0	0	0	10- geometric centroid	0	0	0
6	F1_M36	10- geometric centroid	0	0	0	10- geometric centroid	0	0	0
7	F1_M37	10- geometric centroid	0	0	0	10- geometric centroid	0	0	0
8	F1_M38	10- geometric centroid	0	0	0	10- geometric centroid	0	0	0
9	F1_CS20 (C-5)_L1	10- geometric centroid	0	0	0	10- geometric centroid	0	0	0

Tension/Compression-Only Members

Members in RISA by default are allowed to take both tension and compression. However, some design or modeling practices may require you to define specific members as either *Compression Only* or *Tension Only*. Here we will define what each of the T/C Only options means.

Both Ways

This is the default behavior. In this scenario a member will be allowed to take either compression or tension and the program will base the combined bending and axial code checks on these forces.

Tension Only/Euler Buckling

Tension-only behavior is just as it sounds. When a member is defined as tension-only the program will allow these members to only resist tension forces. If the load path introduces compression forces into the member, then it will be removed from the stiffness matrix and another solution will be run.



Note:

- To avoid stability issues, the program will allow the Tension Only member to resist a very small percentage of compression in the member. By allowing the member to take $1e-8$ of the member's stiffness in resistance to compression, the program can avoid model instabilities which would be caused by removing the Tension Only member all together.
- Tension-only member code checks will ONLY consider axial tension. Bending forces will NOT be used when computing code checks.

Euler Buckling

An option similar to tension-only is the Euler buckling option. This option is used as a replacement to tension-only and allows a member to take up to the Euler buckling capacity of the member in compression before it is removed.

The reason for the Euler buckling option is that allowing tension-only members to take a little compression helps model convergence greatly. Generally speaking, tension-only members are generally very slender members that have a small Euler buckling capacity. The intent of this option is to allow a very small amount of compression to get into your members so you don't get unstable models.

**Note:**

- Euler buckling member code checks will **ONLY** consider axial tension. Bending and compression forces will **NOT** be used when computing code checks.
- You can control the amount of compression that an Euler buckling 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 Hot Rolled, Cold Formed or Wood tabs of the **Members** spreadsheet.
- Using this option for members with high Euler buckling capacities is generally not appropriate and may lead to unconservative results.

Compression Only

Compression-only behavior is just as it sounds. When a member is defined as compression-only the program will allow these members to only resist compression forces. If there are tension forces then the member will be removed and another solution will be run.

**Note:**

- To avoid stability issues, the program will allow the Compression Only member to resist a very small percentage of tension in the member. By allowing the member to take 1e-6% of the member's stiffness in resistance to tension, the program can avoid model instabilities which would be caused by ignoring the Compression only member all together.
- Compression-only member code checks will **ONLY** consider axial compression. Bending forces will **NOT** be used when computing code checks.

Iterative Solution Considerations

When a model contains T/C only members, the program must iterate the solution until it converges. Each iteration is a solution of the model. After each solution the program checks T/C only member forces. If a tension-only member has compression then this member has it's stiffness reduced down to 1e-8% of it's original stiffness so it will take no forces. If an Euler buckling member has a compression greater than its Euler buckling load this member has it's stiffness reduced down to 1e-8% of it's original stiffness. If a compression-only member has tension then this member has it's

stiffness reduced in a similar fashion. After these member stiffnesses are reduced the stiffness matrix is re-built and another solution is run.

In subsequent solutions members can have their stiffnesses both removed and added back in. Reducing the stiffness of members is the identical process just explained. Adding member stiffness back in can occur for members that have been previously reduced. The program will look at the distance between the end nodes which define the previously reduced member. From this distance the program can calculate whether that member would now be in compression or tension. Let's take a tension-only member that had it's stiffness reduced due to the presence of compression in the first iteration. In the 2nd iteration if the distance between the nodes defining the member got longer than it was in iteration one, this would signify that the member would now be in tension. Thus, for the next iteration the program would place this member back in the stiffness matrix.

Because of this iterative solution, models with T/C only members can take a bit longer to solve than a regular static solution.

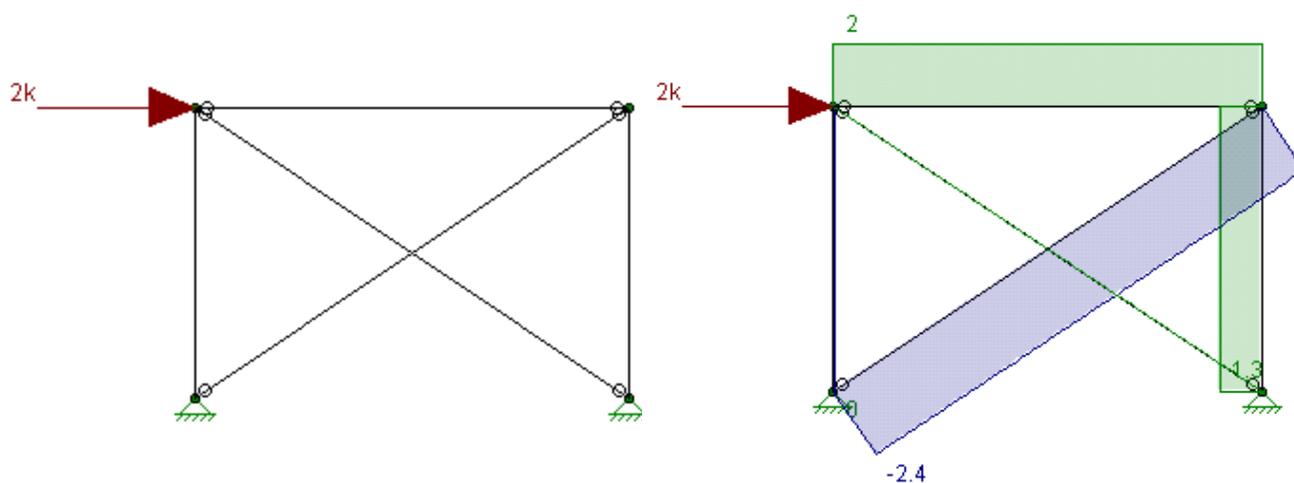


Note:

- If a member has it's stiffness reduced, it will still show up in the results. However, the forces and code checks for this member will all be zero.
- For X-bracing that is set to either tension-only or Euler buckling, do not place a node at the intersection of these braces. You will likely have convergence problems in this case and receive error code 1162.

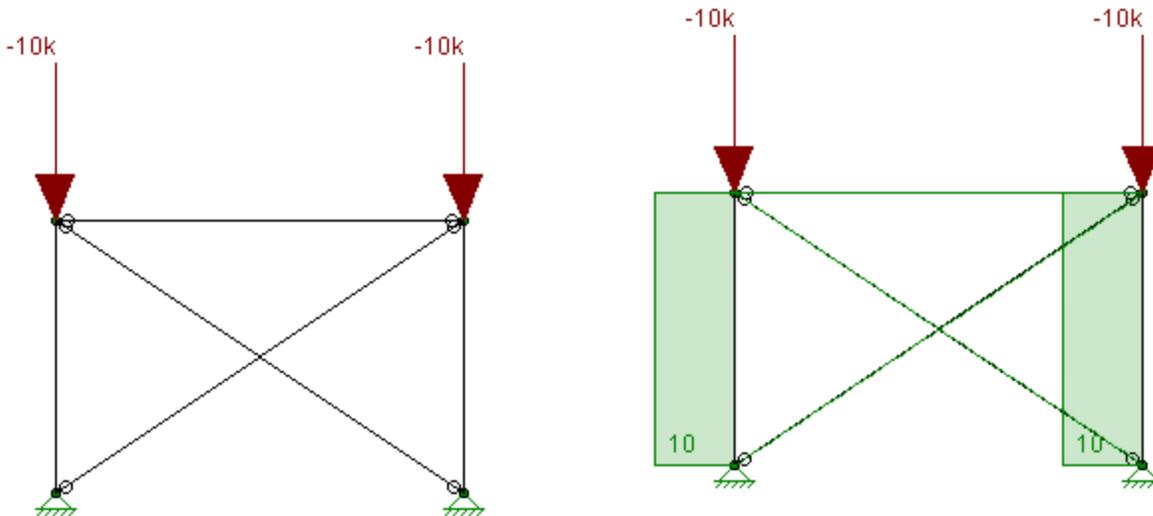
Instability Considerations

For members defined as tension-only you can run into problems with instabilities for gravity-only LC's. For example, let's take a look at an x-braced frame where the x-bracing is defined as tension-only. For a lateral load combination, the tension-only feature works great.

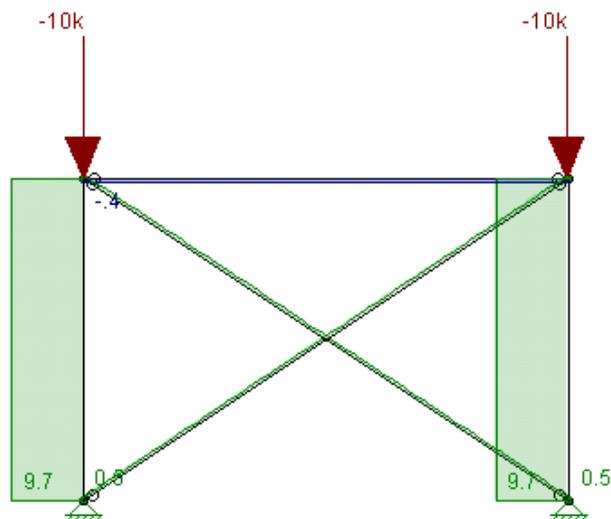


In the image above we can see that the "compression" brace has no force in it because it has it's stiffness reduced in the stiffness matrix and that all of the load is passing through the "tension" brace.

However, if we are looking at a gravity load combination, this results in an instability.



In this scenario the columns in this model undergo a small amount of axial shortening. This induces a small amount of compression and both of the tension-only braces have their stiffnesses reduced. This results in a frame that is unstable and illustrates the problem. The solution here is to use Euler buckling braces.



In this case these braces are taking about 500 lbs of compression, a relatively small amount. As long as this value is small then we can still consider these as tension-only members.



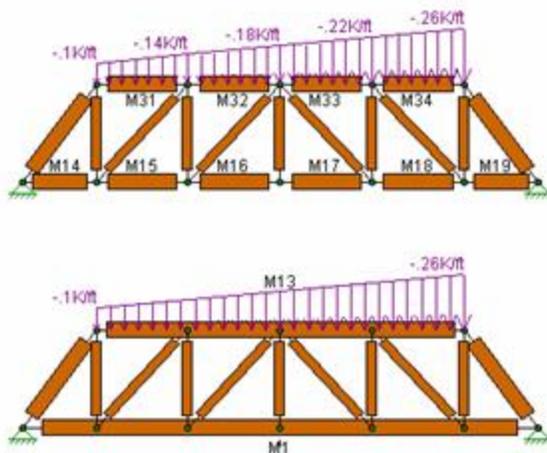
Note: Shear effects are considered for all T/C Only member options in separate code checks for shear.

Physical Members

Physical Members provide fixity to all joints that occur along the length of the member, without breaking that member into multiple smaller members. You may use the physical member feature to avoid defining one 'field' member with multiple members in your model. This saves time in building and editing your model and in understanding your results.

To define a Physical Member, check the Physical Member box in the **Set Member Properties** settings or in the **Modify Member Parameters** settings. You may also specify Physical Members in the **Members Spreadsheet** and in the **Member Information** settings.

To understand the benefits of Physical Members see the trusses in the following figure. The first truss shows the chords modeled without physical members and thus with **multiple** members modeling each chord. The second truss models each chord with **one** Physical Member. Both models will yield the same results. However the Physical Member model is more intuitive and easier to work with because you don't have to work with multiple members when you create the chords, load them, edit them or evaluate their results. Notice that the distributed load may be defined from start to end for the Physical Member. In the other model the distributed load end magnitudes have to be specified for each of five members in order for the entire load to be defined.



Continuous beams, multi-story columns and truss chords are examples of continuous 'field' members that can be modeled with one Physical Member. You may define them from start to finish without having to explicitly define their connection to other elements or boundaries through intermediate joints. Subsequently, when it comes time to make changes to a member you can edit the properties of one Physical Member rather than the multiple members that might otherwise represent it.

Physical Members are also effective in managing results because the results for one Physical Member are reported together in the results spreadsheets and the member detail report. With the truss example above the Physical Member model allows you to look at results for one member making it easier to track maximum forces and other values.

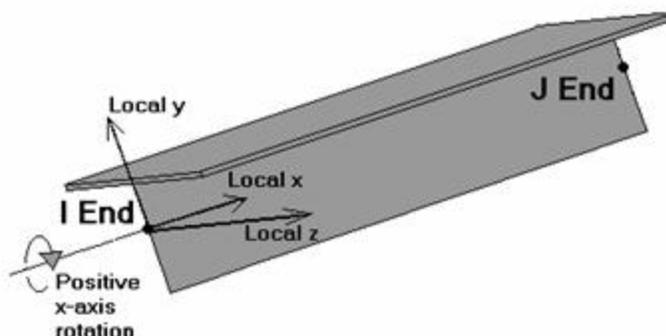
**Note:**

- You may convert a Physical Member into multiple members by removing the Physical Member property and then performing a Model Merge to split the member for connectivity. See [Model Merge](#) for more information.
- You cannot convert multiple members into one Physical Member. To achieve this you must delete the multiple members and define a new Physical Member.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **What are Physical Members**.

Local Axes

The following diagram illustrates the directions of the member's local axes that are used to define member forces, stresses, and deflections as well as loads defined in local axes directions:



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

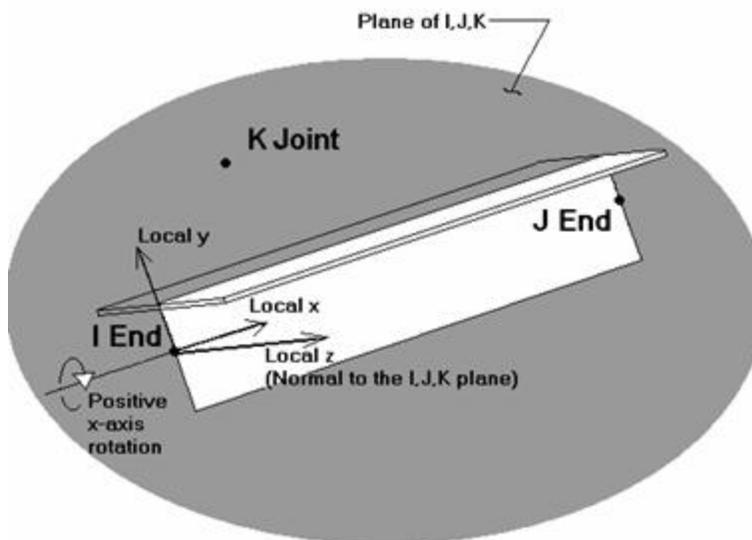
If you do not explicitly define the orientation for a member, the default is for the member's local z-axis to lie in the global X-Z plane or as near as possible. If the member is defined in the global Y-direction, the member's local y and z axes both lie in the global X-Z plane, so the local z-axis is made parallel to the global Z-axis. This works well for models with the Y-axis as the vertical axis because any beam members are typically oriented such that vertical loads are resisted by the strong axis bending of the member. You can change the default orientation of the members with the [Model Settings](#). For

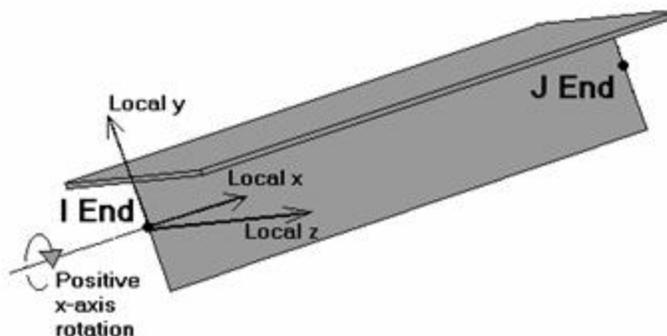
example if your vertical axis is the Z-axis then you can specify the default so that vertical loads are resisted by the strong axis bending of the member.

Member Orientation

RISA-3D provides two ways to explicitly set the orientation of the y-axis. The first is by rotating the member about the local x-axis. This member rotation is entered in the x-Axis Rotate column on the Member spreadsheet or may be specified in the graphic editing tools. For this rotation, positive is counter-clockwise about the x-axis, with the x-axis pointing towards you.

The second way to explicitly define the orientation is by specifying a K joint for the member. If a K joint is defined the three joints (I,J,K) entered for the member are used to define a plane. This plane is the plane of the member's x and y-axes. The z-axis is defined based on the right hand rule using the x and y-axes. See below:





Note: To be sure of a member's orientation you can always view the rendered shape of a member by clicking the **I** button on the **Quick View** toolbar on the **Home** tab.

Member End Releases

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

The partial fixity member end release option reduces the joint rotational stiffness about the member local y and z axes. The linear stiffness of the connection is reduced by the rotational spring constant to utilize a partially restrained connection, as opposed to a fully released or fixed connection.

To specify member releases go to the **I Release** or **J Release** field for the member in the Members Spreadsheet on the Advanced Tab, click the  button, and specify the condition.

Alternatively, you may specify the end condition by directly typing in the field. To indicate that a force component is released, put a **X** for that component in the release field. You can move within the release field using the space bar which will result in a **O** for no release.

RISA-3D has two special "keyword" release configurations built-in. These are:

AllPIN => Mx, My, Mz (all moments) released (OOOXXX)

BenPIN => My, Mz (bending only) released (OOOOXX)

If a Partial Fixity Option is selected, the keyword will show a **P** for the rotational restraint followed by the rotational stiffness input.

These keyword entries are included because 99% of the release configurations you'll ever want to define will be one of these two (98% will be BenPIN). You can call out the keyword entry by just

entering the first letter of the keyword. So if you go to a release field and enter "bp", the keyword "BenPIN" will be filled in automatically.

**Note:**

- RISA-3D will not allow you to release the member torsion at both ends. This is because it will be unstable as it would be free to spin about its centerline. For this reason, pinned end conditions should be modeled using the "BenPIN" entry instead of "AllPIN".
- A partial fixity end release cannot be combined with a member end offset.
- Shear deformations are not considered for members with partial fixity end releases.
- Tapered WF members cannot be used in conjunction with partial fixity end releases.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **End Release**.

Simpson Yield-Link

The Yield-Link is a prequalified connection developed and manufactured by Simpson Strong-Tie for steel special moment frames. For more information about the Yield-Link Connections, click the link to visit the Simpson Strong-Tie's website [here](#).

Currently in the program Yield-Links can be added to steel beams as Member End Releases to include their partial rotational stiffness in the analysis. The program calculates the rotational stiffness automatically based on the sizes of the Yield-Link and the beam.

To add a Yield-Link to a beam, click on the ellipsis button next to the **I Release** or **J Release** field to open the Set Member End Releases dialog. The Yield-Link size can be selected from the dropdown box under Simpson Yield-Link.

When choosing a specific Yield-Link to use, the nomenclature for the definition of the release is as follows: **YL4-2.25**, where YL4 represents the Yield-Link stem thickness in eighths of an inch (ex. 0.5" = 4 x 1/8") and 2.25 is the width of the yield link (in inches). For new series of Yield-Link with longer lengths, the nomenclature can be as follows: **YL4-2.25-10**, where the last number 10 is the longer length than standard links (in inches)

Users can also type in the Yield-Link size in the **I Release** or **J Release** field from either the **Properties Panel** or the **Members -> Advanced Spreadsheet**. The nomenclature of Yield-Link (e.g. YL4-2.25) needs to be followed for the program to recognize the input.

Member Advanced Data											
Concrete Column			Aluminum			Stainless		RISACConnection		Detailing	
Primary		Advanced			Hot Rolled		Cold Formed		Wood		Concrete Beam
	Label	I Release	J Release	I Offset [in]	J Offset [in]	T/C Only	Physical	Deflection Ratio Options	Analysis Offset [in]	Activation	Seismic DR
1	M71						<input checked="" type="checkbox"/>	N/A			None
2	M72	YL6-5-13	YL6-5-13				<input checked="" type="checkbox"/>	Default			None
3	M83	BenPIN	BenPIN				<input checked="" type="checkbox"/>	N/A			None

The selection of Yield-Link size needs to meet the following requirements:

- Yield-Link is only available for Hot-Rolled Wide Flange Beams.
- Currently the maximum beam depth allowed for Yield-Link is W36. The minimum beam depth allowed for Yield-Link range from W12, W16, and W24 depending on which Yield-Link series is used:
 - YL4 series: Min. W12, Max. W18
 - YL4 longer series: Min. W12, Max. W27
 - YL6 series: Min. W16, Max. W27
 - YL6 longer series: Min. W16, Max. W36
 - YL8 series: Min. W24, Max. W36
 - YL8 longer series: Min. W24, Max. W36
- The permitted beam flange thickness needs to be greater than 0.4 inch.
- The permitted beam flange width and column flange width for any specific Yield-Link size can be found in [Simpson's Yield-Link Design Guide](#).
- If Yield-Link is used on one end of a beam, the other end cannot have Fully Fixed or Custom Releases.
- If Yield-Links are used on both ends of a beam, the link size must be the same at each end.

**Note:**

If the input Yield-Link size and beam size do not meet the above requirements, the program will ignore the Yield-Link rotational stiffness during solution. A warning message will be given to indicate which requirement was not met.

Once users have input the Yield-Link size, together with the Beam Size, the program calculates the rotational stiffness of the Yield-Link and displays it in the I and J Release Edit dialog. This partial stiffness will be used in solution for the model. The rotational stiffness calculation follows the following equations:

$$P_{ye_link} = A_{yield} \times R_{y_link} \times F_{y_link}$$

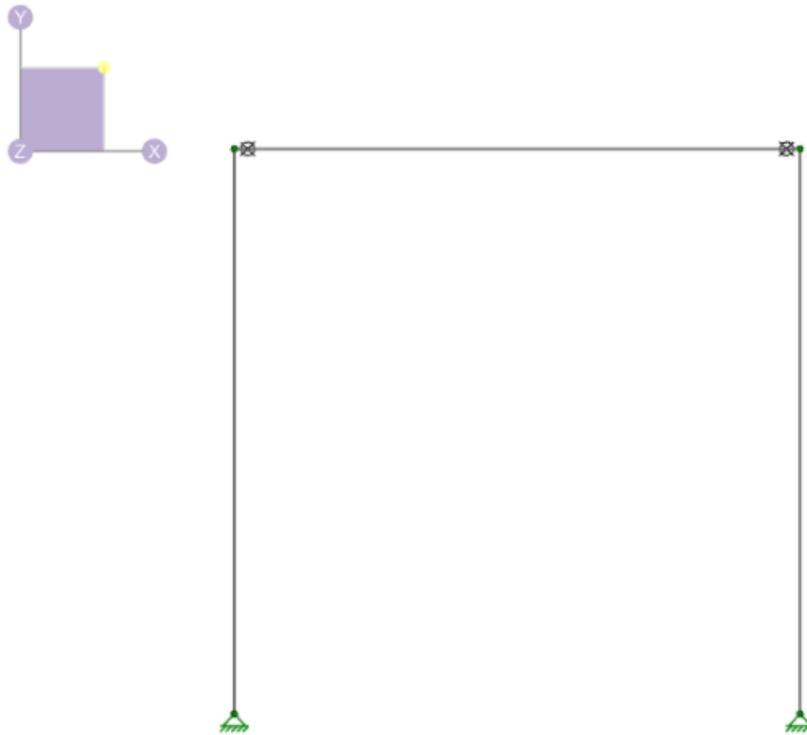
$$M_{ye_link} = P_{ye_link} \times (d_b + t_{stem})$$

$$\Delta_y = \frac{P_{ye_link}}{K_{eff}}$$

$$\theta_y = \frac{\Delta_y}{\frac{(d_b + t_{stem})}{2}}$$

$$K_{rot} = \frac{M_{ye_link}}{\theta_y}$$

With the Yield-Link defined, the wireframe model shows the member end release condition applied. The display looks the same as custom end releases in the Mz direction, which is the same direction where the Yield-Link rotational stiffness is applied.



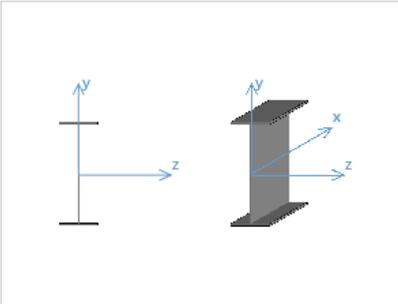
After solution, the partial stiffness from Yield-Links can be viewed in the member Detailed Report under the I/J releases.

Detail Report for - M72 — □ ×

< Member Label M72 >
LC 1: IBC 16-1 ▾
Options

Detail Report: M72

Load Combination: LC 1: IBC 16-1 Code check: 0.017 (axial/bending)



Input Data			
Shape:	W27X84	I Node:	N418
Member Type:	Beam	J Node:	N414
Length (ft):	18	I Release:	Custom
Material Type:	Hot Rolled Steel	J Release:	Custom
Design Rule:	Wide Flange	I Offset:	N/A
Internal Sections:	97	J Offset:	N/A
Design Code:	AISC 14th (360-10): LRFD	T/C Only:	Both Way

Material Properties			
Material:	A992	Therm. Coeff. (/1E5 F):	0.65
E (ksi):	29000	Density (k/ft ³):	0.49
G (ksi):	11154	F _y (ksi):	50
Nu:	0.3	R _y :	1.1
		F _u (ksi):	58
		R _t :	1.2

Shape Properties			
d (in):	26.7	Area (in ²):	24.7
b _f (in):	10	Z _{yy} (in ³):	33.2
t _f (in):	0.64	Z _{zz} (in ³):	244
t _w (in):	0.46	C _w (in ⁶):	17900
I _{yy} (in ⁴):	106	W _{no} (in ²):	65.2
I _{zz} (in ⁴):	2850	S _w (in ⁴):	104
		r _T (in):	2.49
		J (in ⁴):	2.81
		k _{det} (in):	1.875
		k _{des} (in):	1.24

Design Properties			
L _{b y-y} :	Segment	K _{y-y} :	1
L _{b z-z} (ft):	18	K _{z-z} :	1
L _{comp top} (ft):	1	y sway:	No
L _{comp bot} (ft):	18	z sway:	No
L _{torque} (ft):	18	Function:	Lateral
C _b :	1	Seismic DR:	None
		Max Defl Ratio:	L/10000
		Max Defl Location:	0
		Span:	N/A
		τ _b :	1

Print
 Add to Full Report
Close

Deflection Ratio Options

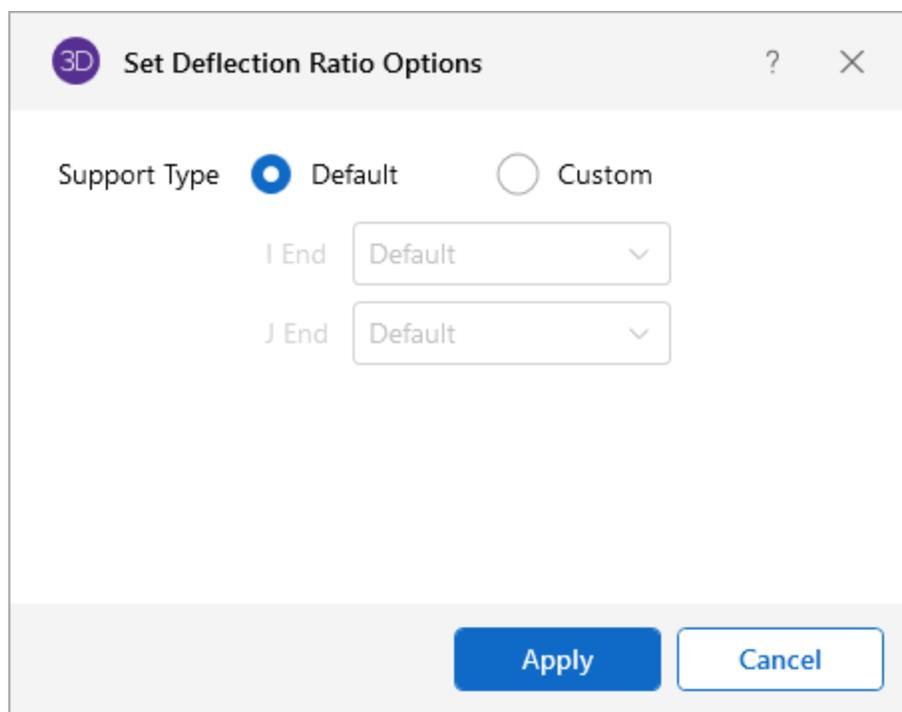
The **Beam Deflection Ratio** can be used during member design to check the deflection criteria. By default, the program determines whether a member is a cantilever or is supported on both ends. In

certain cases where the beam should be considered otherwise, more control over these support types for a beam can be designated in the **Deflection Ratio Options**. This feature is only applicable to members designated as "beams" and only affects the beam deflection ratios located in the [Beam Deflection](#) tab of the **Member Deflections** spreadsheet.

The option to set the support type for the beam deflection ratio can be applied from any of the following locations:

- Additional Properties section of the Properties Panel
- Members spreadsheet \ Advanced Tab

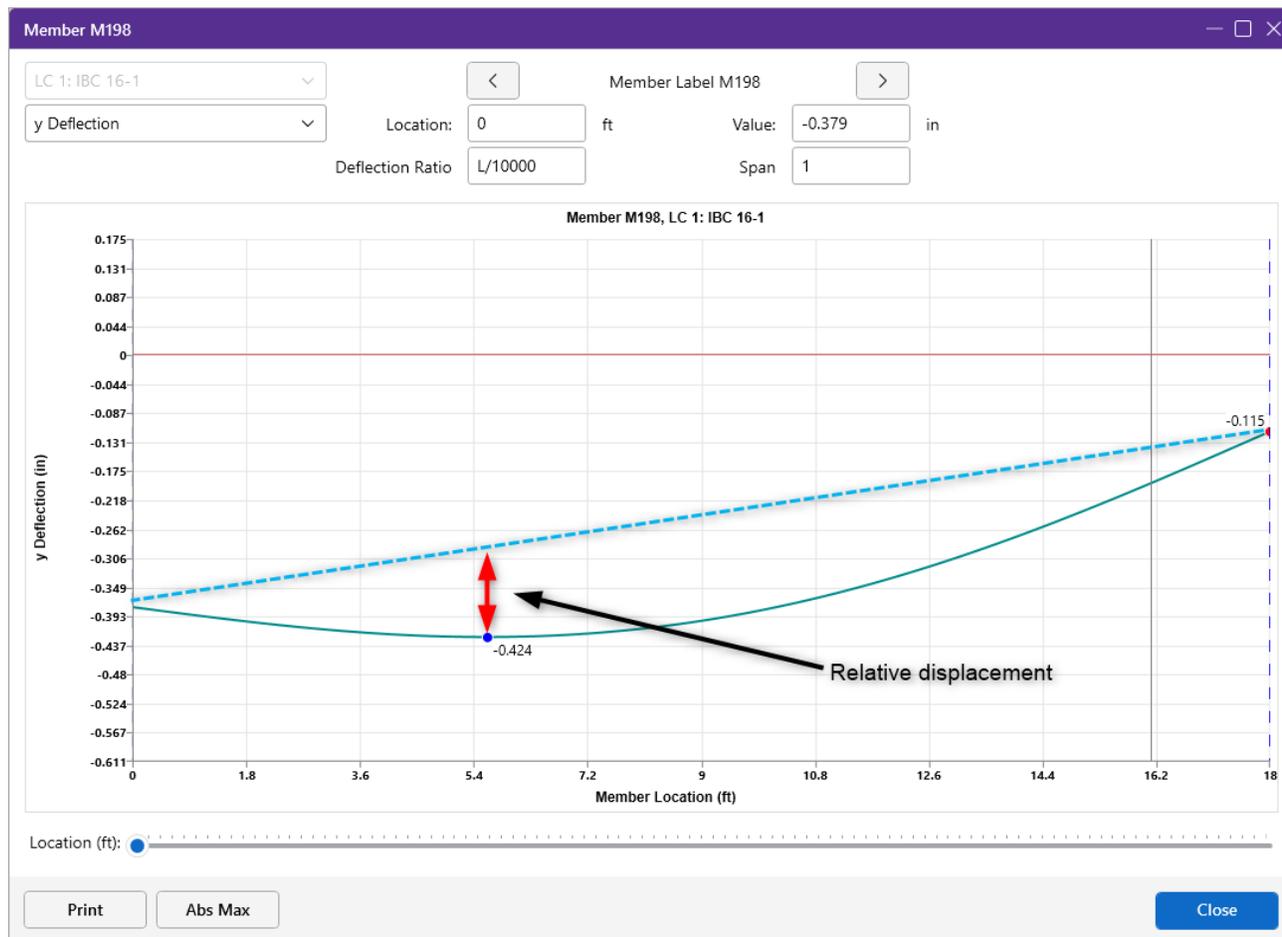
Clicking the  button in the spreadsheet brings up the following dialog box:



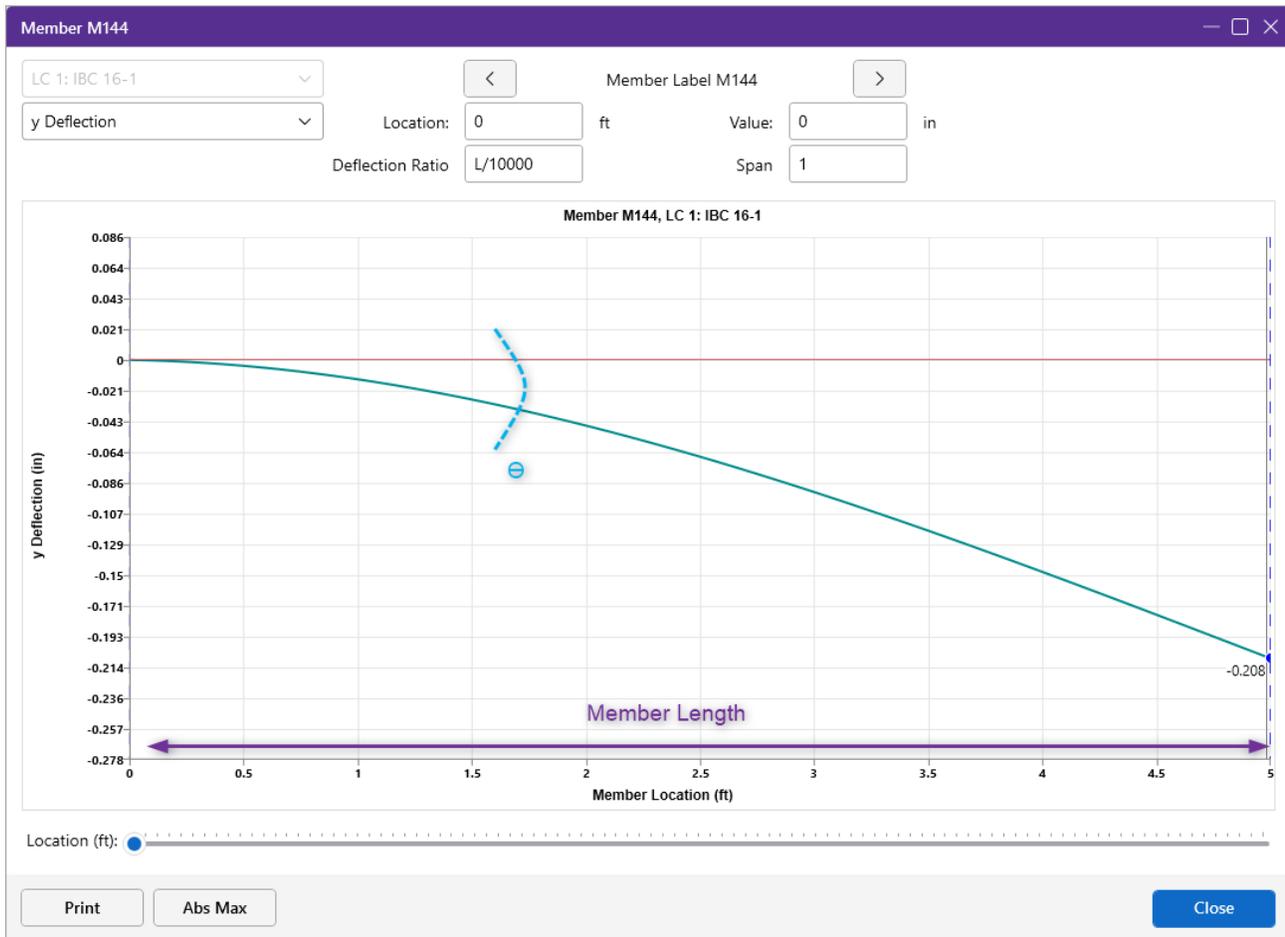
The dialog box is titled "Set Deflection Ratio Options" and features a 3D icon in the top left corner. It contains two radio buttons for "Support Type": "Default" (selected) and "Custom". Below this, there are two dropdown menus labeled "I End" and "J End", both currently set to "Default". At the bottom of the dialog, there are two buttons: "Apply" and "Cancel".

Single span beams can be designated as supported on both ends, or cantilever on either I- or J- end. Multi-span beams have an additional option to be cantilever on both I- and J- end. A double cantilever option is available for multi-span beams since internal supports will support the beam. Designating member ends as either supported or cantilever will affect only the beam relative deflection and also determine which equation is used when calculating beam deflection ratios. See [Beam Deflections](#).

For a member supported on both ends, the maximum relative displacement used in the beam deflection ratio will be the largest straight line distance (in the vertical y direction) between the deflected shape and the original undeflected shape.



For members with a cantilever end, the location of the maximum relative deflection will be at the free end of the member. The displacement value is equal to the rotation at the supported end multiplied by the length of the member.

**Note:**

- Designating specific member end support types for the beam deflection ratio will not affect global deflection or the member deflection ratios located in the [Service](#) and [Strength](#) tabs of the Member Deflections spreadsheet.
- For additional information on this topic, please see the RISA Tips & Tricks web page at risa.com/post/support. Type in Search keywords: **Deflection Ratio**

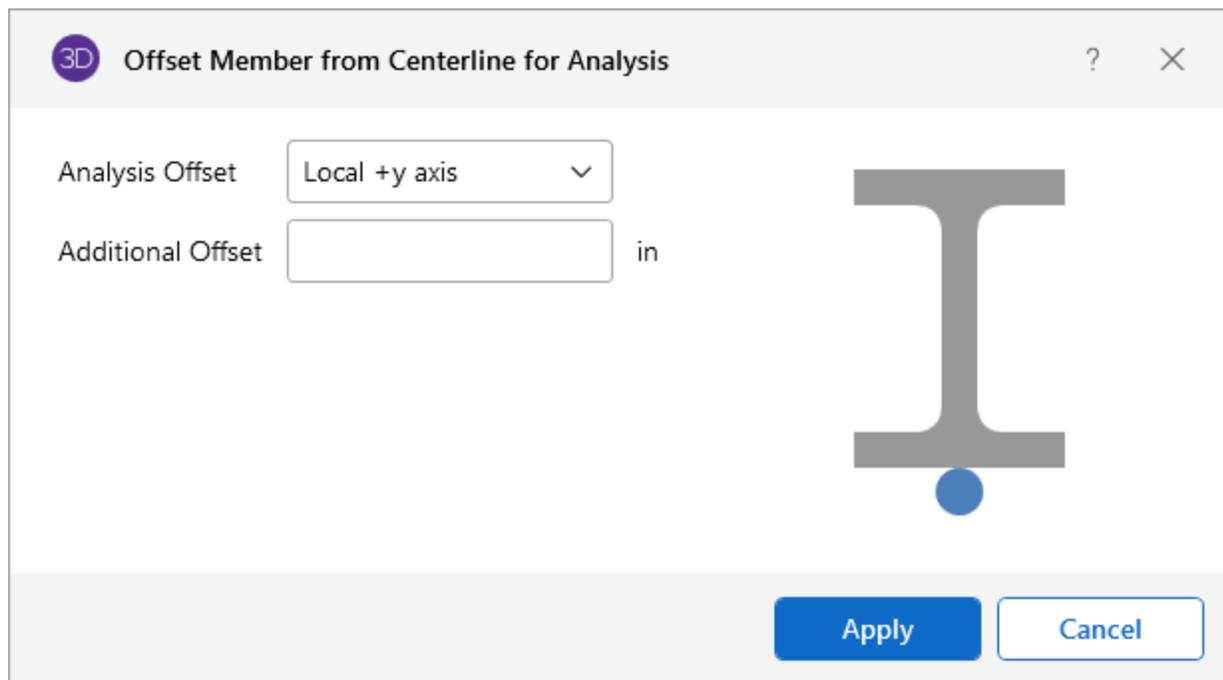
Analysis Offset

The Analysis Offset feature allows a member to be offset from its centerline for analysis. When it is enabled the member is actually shifted to the offset location for analysis. In order to provide connectivity the program places hidden rigid links to connect the member to the nodes that define the member's I/J Joints. Joints which fall along a physical member are also automatically connected to the member's actual analysis location via hidden rigid links.

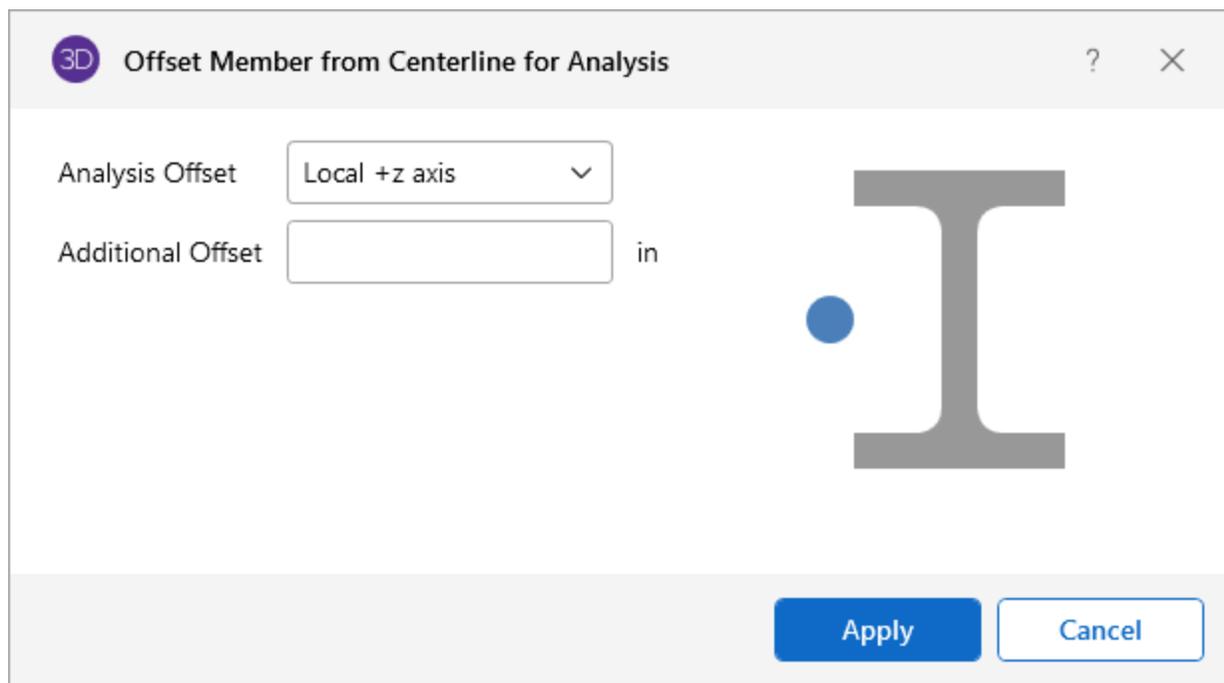
An analysis offset can be applied from any of the following locations:

- Additional Properties section of the Properties Panel
- Members Spreadsheet \ Advanced Tab

Clicking the  button in the panel or spreadsheet to bring up the following dialog box:



The blue dot represents where the member is drawn in the model, the shape represents where the shape will be offset relative to the centerline. Therefore in the above image the wide flange is offset in a way to account for the connections on this member being on the "vertical" (local y-axis) leg. In the below image the angle is offset in a way to account for the connections on this member being on the "horizontal" (local z-axis) leg.

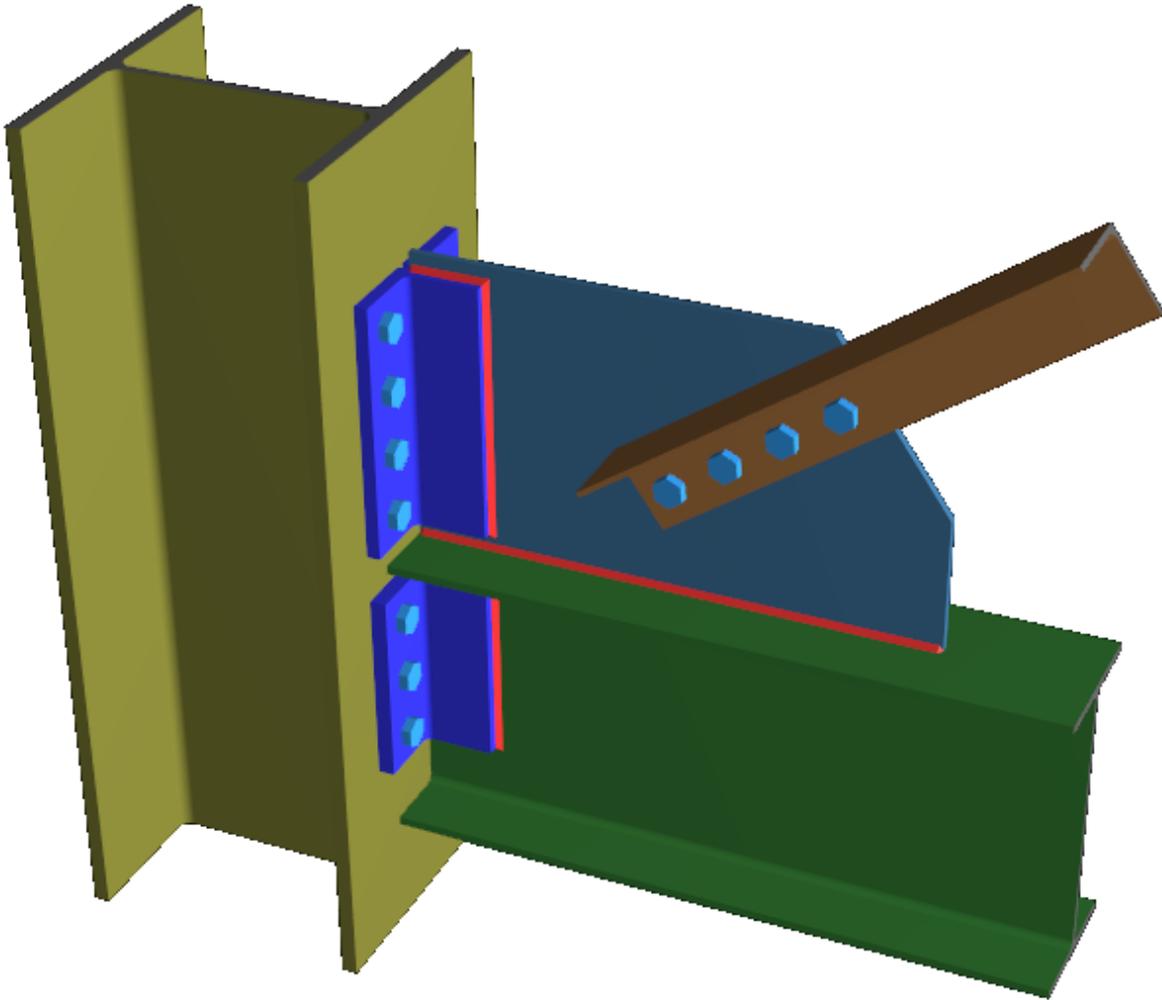


In this case there would be no practical application for using a -z axis or -y axis setting since neither leg would be near the line that the member was modeled on.

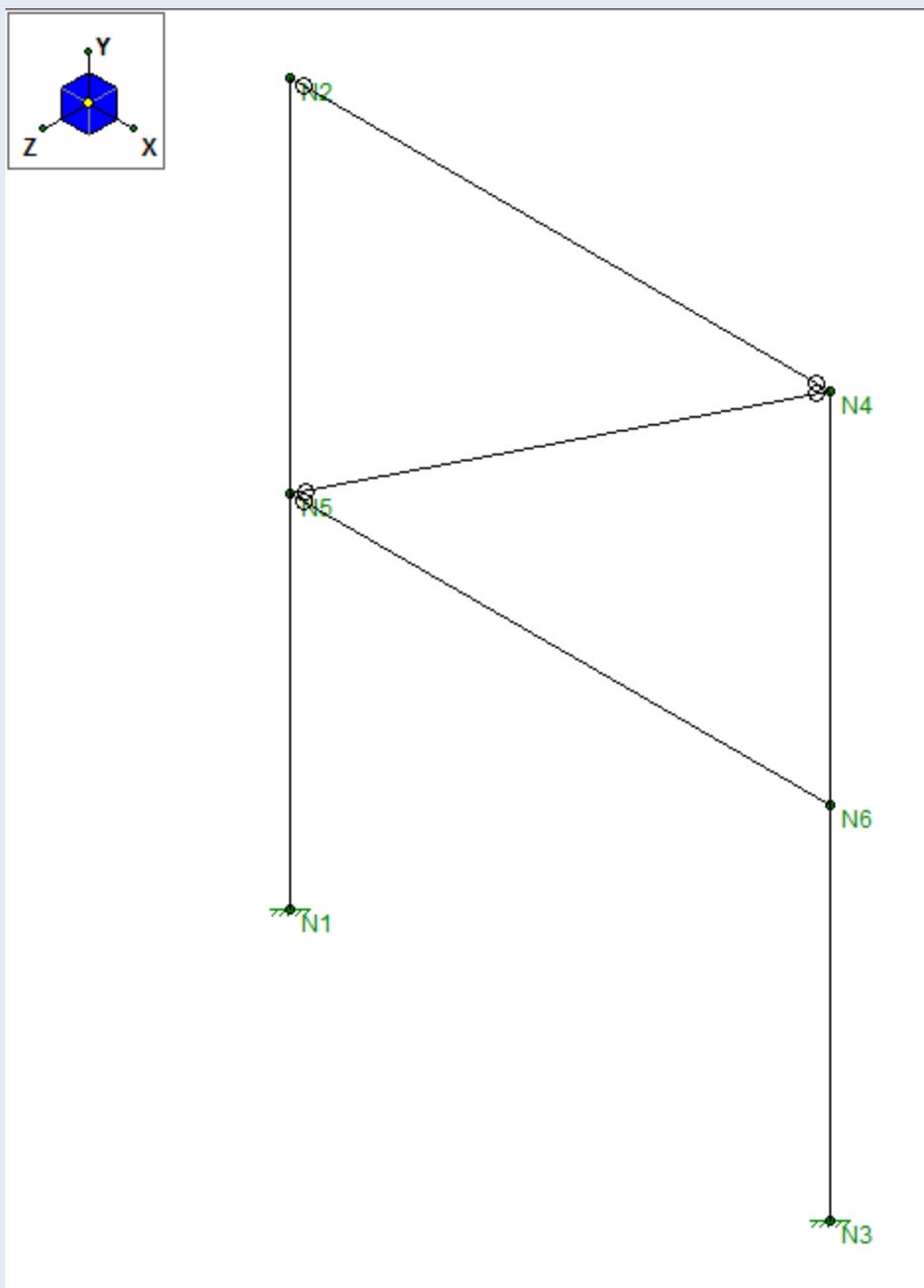
The most common application for this feature is modeling non-symmetric braces where the centroid of the brace does not align in the same plane as the columns/beams that the brace connects to. See the example below.

Example:

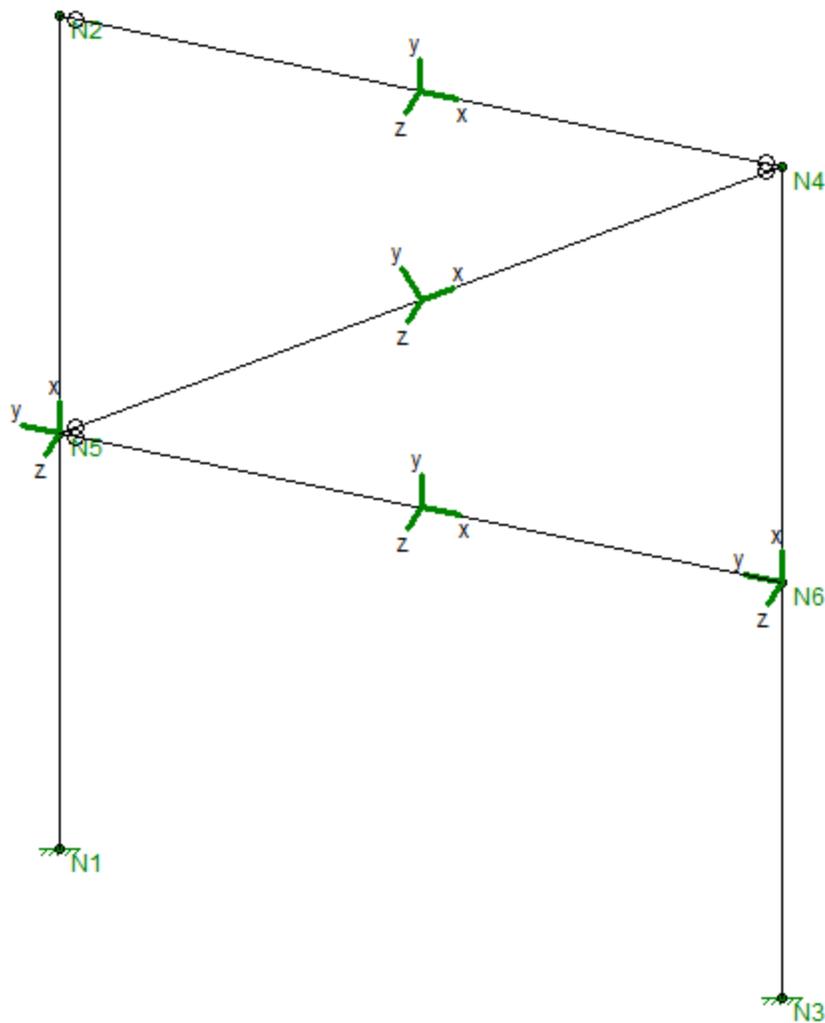
Consider a single angle brace that frames into a beam and column



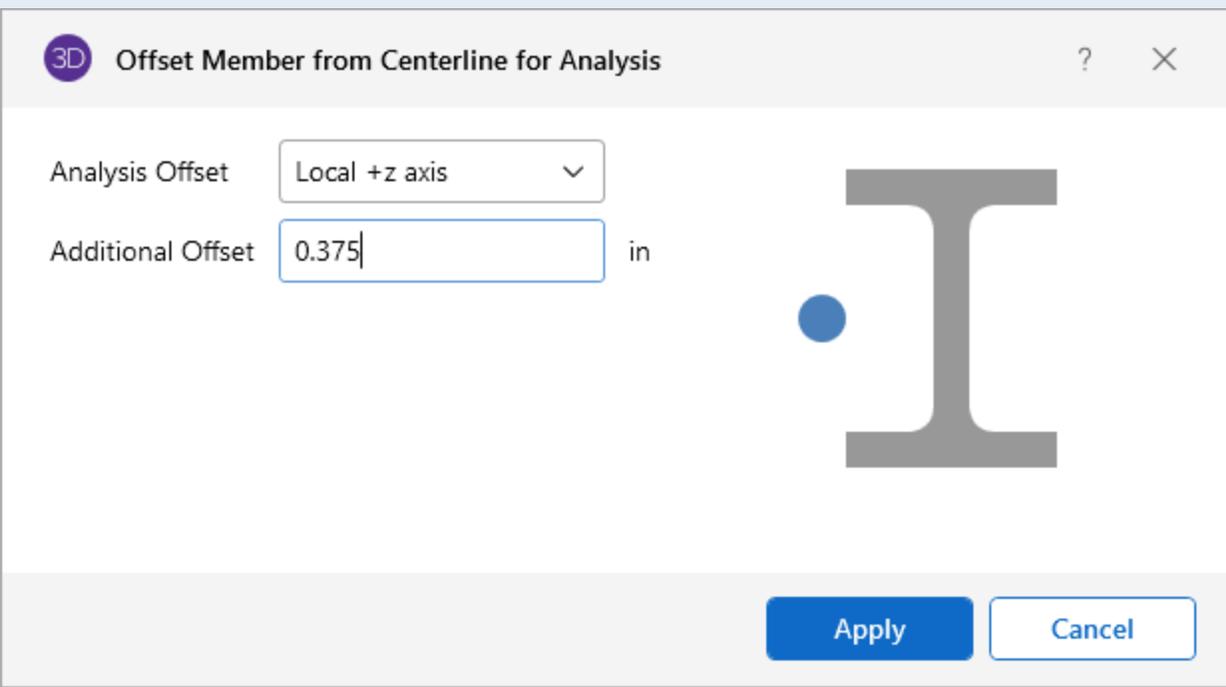
In RISA-3D this would be modeled so that both the brace and the beam end connect to the same Joint, which the column also passes through



When lateral load is applied to this frame with no analysis offset, the brace has a concentric axial force since its centerline aligns with the centerline of the beam and column. This would be accurate for a double-angle brace, but for a single angle brace the centroid of the brace is actually offset from the plane of the frame by a small but significant distance. That offset eccentricity should cause a bending in the brace which accompanies an axial compression force in the brace. This can be accounted for by shifting the location of the brace to its true centroid location.



In this case the brace should be shifted along its positive Local z-axis by a distance of \bar{z} (distance from centroid to connected leg) plus half the gusset plate thickness. For a 3/4" gusset plate the analysis offset should be set to the +z axis plus 3/8":



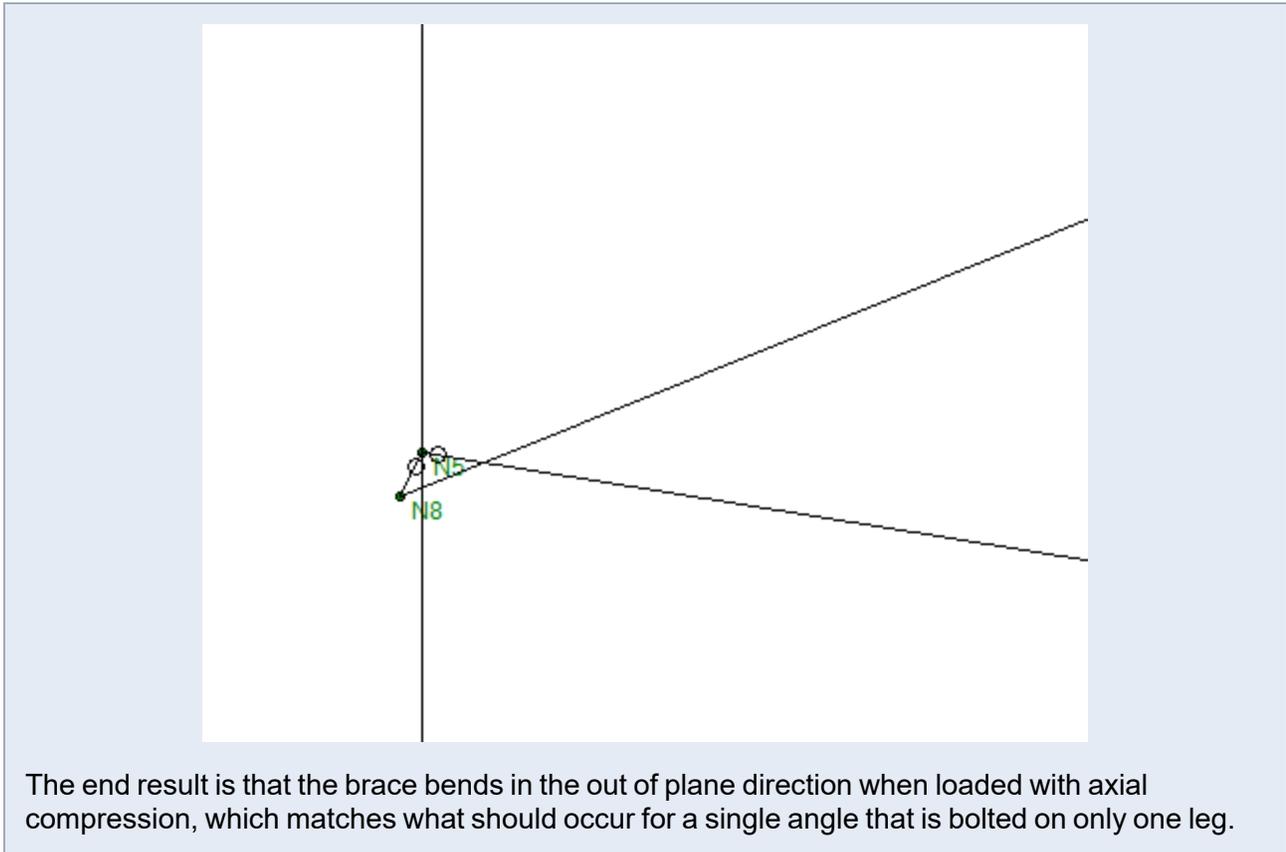
3D Offset Member from Centerline for Analysis

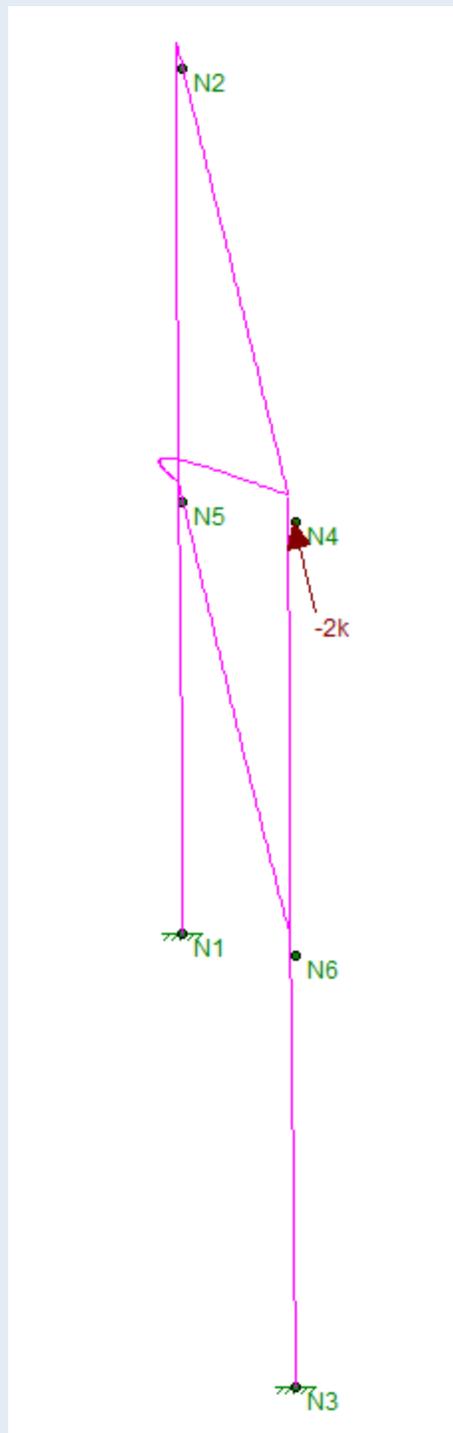
Analysis Offset Local +z axis

Additional Offset 0.375 in

Apply Cancel

For analysis the program will move the brace outward and put in a rigid link. This rigid link is not visible in the wireframe view, although the rendered view and deflected shape will both illustrate the offset location of the member. Below is a reconstructed view of what the program actually uses for analysis in this example.





Unless both legs of the single angle are attached to the gusset in this example it is actually *unconservative* to omit the analysis offset in the model, since bending will be ignored without it.

Col-Wall Vert Release

This option is only visible for models transferred from RISAFloor and is applicable solely to columns transferred from RISAFloor. It defines the vertical connectivity between the column and the connected wall panel. If this option is selected, the column and wall panel will be decoupled in the vertical direction during analysis. This implies that the axial load in the column will not be transferred to the wall panel, and vice versa. However, it's important to note that connectivity for other degrees of freedom, including in-plane and out-of-plane bending/shear, remains intact. As a result, forces associated with these degrees of freedom will continue to be transferred between the column and the wall panel.

The program implements this decoupling behavior by generating two sets of nodes along the column axis: one set for the column and another for the wall panel. It then defines the connectivity between these two sets of nodes. When users select the 'Col-Wall Vert Release' checkbox, the program releases the connectivity between the two node sets in the vertical direction while maintaining connectivity in other degrees of freedom.



Note:

- Since there are two sets of nodes along the column axis, the program must determine whether any loads applied to these nodes should apply to the column node set or the wall panel node set. A similar determination must be made for the reactions from any elements connected to these nodes. Nodal loads applied directly on a column, and members connected to a column, are resisted and supported directly by the column member. All other load types (line loads, area loads, etc.) and element types (plates, wall panels, diaphragms, etc.) are considered to be connected to the wall panels.

Member End Offsets

Member offsets reflect the fact that the member ends may not be attached at the centerline of the member being attached to. For example, a beam connected to the flange of a column is offset from the centerline of the column. The distance of the offset would be $\frac{1}{2}$ 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 you use a member label to define the end offset, this value is ALWAYS taken as simply the depth of the member. Thus, if you are framing at an angle into a member or into the weak axis of a member you should not use this feature.
- When the model is solved the member length is adjusted in the stiffness matrix by the offset distance resulting in a shorter, stiffer member. Also the results listed for members with offsets do take into account the offset distances. The I-end and J-end results are the results at the offset locations, and the report locations are determined by dividing the member length minus the offset distances by the **Number Of Sections** on the **Model Settings**.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Offsets**.

Inactive and Excluded Elements

Making an element, such as a member, plate or wall panel, 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.

Making an element **Inactive** means the item is not included when the model is solved or plotted.

Making an element **Excluded** means the item is in the solution, but excluded from the results list. An excluded item will be treated like any other member in the solution and plotting of the model, but the item will not be listed in the solution results (forces, stresses, deflections, etc.). This is useful if there are certain items whose results you're not interested in. You don't have to clutter up the results with these items and can concentrate on the items you're most interested in. See [Printing](#) for more limiting printed results.



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

Shear Deformations

Including shear deformation models the effects of shearing forces on the lateral displacements of the members. Shear deformation is included in the analysis by checking the **Consider Shear Deformation** box in the **Solution** tab of the **Model Settings**.

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

Including shear deformation causes the member stiffness matrix to be modified by the term M, where:

$$M := \frac{12 \cdot E \cdot I}{G \cdot A_s \cdot L^2}$$

Where A_s = Area * Shear Deformation Factor

These shear area coefficients will be automatically calculated for all standard members, but must be entered for arbitrary sections. While these shear deformation factors are always less than 1.0, they can vary from 0.85 for solid rectangular shapes, to 0.88 for solid circular shapes, to 0.53 for thin pipes. These coefficients are used strictly for shear deformation effects and should not to be confused with shear stress factors described in the next section. A good reference for the calculation of both of these coefficients is **Stress, Strain, and Structural Matrices** by Walter D. Pilkey.

For members whose length is much greater than the depth, shear deformation has a relatively minor impact. When the length of the member is less than 10 times its depth, shear deformation begins to have a significant impact on the solution. Keep this in mind if you are creating models where members are being broken up into several pieces because the length used to calculate the term M is the joint-to-joint member length.



Note:

- Shear deformation effects are included for the joint deflections only and not for the internal member deflections.
- Shear deformation effects are included for joint loads and not for member point loads.
- Shear deformation can play a significant role in the stiffness of the member and thus the results.

Shear Areas

The equation for calculating shear area varies for different shape types:

- For WT, LL, L, $A_v = (\text{depth} \cdot \text{thickness}) / 1.2$
- For channel, $A_v = (\text{depth} \cdot \text{thickness}) / 1.0$
- For rectangular tube, $A_v = (2 \cdot \text{depth} \cdot \text{thickness}) / 1.0$ for AISC 360-05 (13th edition) and AISC 360-10 (14th edition), but $(2 \cdot (\text{depth} - 3 \cdot \text{web thickness}^2)) / 1.0$ for AISC 360-16 (15th edition).
- For round tube, $A_v = A / 2$

Member Shear Stresses

Since shear stress is not equally distributed over a cross section, it is unconservative to take the maximum shear stress as simply the beam shear (V) divided by the beam's cross sectional area. To achieve more accurate results you must reduce the shear area by a shear stress factor,

$$\alpha_{\text{stress}} := \frac{\tau_{\text{avg}}}{\tau_{\text{max}}}$$

where:

$$\tau_{\text{max}} := \frac{V \cdot Q}{I \cdot t}$$

and

$$\tau_{\text{avg}} := \frac{V}{\text{Area}}$$

These shear stress factors will be automatically calculated for all standard members, but must be entered for arbitrary sections. While these shear area coefficients are always less than 1.0, they can vary significantly. Common values include 0.67 for solid rectangular shapes, and 0.75 for solid circular shapes.



Note: For wide flange and channel shapes, RISA will replace this calculation with the shear stress calculations required by the Steel Code specifications.

Torsion

A twisting of the member induces torsional forces and stresses. The primary reference used in the development of RISA-3D's torsional calculations was **Design Guide #9, 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 from the torsional reference.



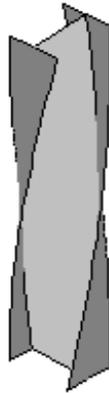
Note: Torsion considerations for any other cases can be grossly over or underestimated. RISA's calculation of torsion capacity is based on torsion due to racking of the structure, not point or line torques on an individual member. That type of check is not supported.

It is more accurate to consider warping effects when calculating member stiffnesses and stresses, but there is a way you can turn off these effects. In the **Model Settings** on the **Solution** tab, you'll see the checkbox **Consider Torsional Warping**. If this box is not checked warping effects will not be considered, i.e. stress and stiffness calculations for wide flanges and channels will be done just like all the other shapes ($k = GJ/L$). You may wish to do this to compare the RISA-3D results with and then without warping, or to compare RISA-3D results with a program that does not include warping.

Warping

A primary consideration in the calculation of torsional properties and stresses is whether the cross section is subject to warping. Solid cross sections are NOT subject to warping. For RISA-3D, all

closed shapes such as pipes and tubes are considered to be NOT subject to warping. Cross sections composed of rectangular elements whose centerlines all intersect at a common point are NOT subject to warping. Examples are Tee shapes and angle shapes. For simplification, double angle cross sections are also assumed to be not subject to warping. So, the only shapes RISA-3D considers subject to warping effects are wide flanges and channels (I's and C's). The importance of this extends beyond the stress calculations, however. Warping considerations also impact the calculation of torsional stiffness for these shapes.



For a nonwarping member or a warping member with warping unrestrained, the member's torsional stiffness is given by:

$$k := \frac{G \cdot J}{L}$$

G = Material Shear Modulus

J = Cross Section Torsional Stiffness

L = Member Length

For a member subject to warping, if the warping of the member is restrained its torsional stiffness is:

$$k := \frac{G \cdot J}{A \cdot \left[\tanh\left(\frac{L}{2A}\right) \cdot \cosh\left(\frac{L}{A}\right) - \tanh\left(\frac{L}{2A}\right) + \frac{L}{A} - \sinh\left(\frac{L}{A}\right) \right]}$$

$$A := \sqrt{\frac{E \cdot C_w}{G \cdot J}}$$

C_w = Cross Section Warping Constant

E = Material Modulus of Elasticity

L = Torque Length

Thus restraining the warping effects for a cross section subject to warping (I's or C's) makes the shape much stiffer in torsion. Think of it this way; if you twist a wide flange, the flanges want to warp. If you restrain the flanges from warping its much harder to twist the wide flange as it is stiffer in torsion.

Member Releases

If a member is released for any rotational degree of freedom at either end, warping is not considered for that member. For example, if you model a wide flange member with a "BenPIN" release code (at either or both ends), warping would not be considered for that member. This is because any connection that doesn't resist bending moments is certainly not going to restrain warping. RISA-3D does not consider the effect of warping "pins" at this time.

Warp Length

The "warp length" is the length between points of torsional restraint (or release). This may be equal to or greater than the member's actual joint to joint length. This warp length is calculated automatically by RISA-3D and is used for the member's torsional stiffness and stress calculations. Each member's warp length is shown on the member detail report (see [Member Detail Report](#)).

Physical members that are subject to torsion will always have their warp length be at least the length of the physical member. Members framing in along the length of the physical member are assumed to not reduce the warp length.

Note that the calculation of the warp length by RISA-3D can be "fooled" by beams that are modeled by several segments. The warp length used for each member will be the length of each straight-line segment rather than the whole length. Using Physical Members rather than member segments will avoid this. This is not possible however when modeling curved members with straight line segments.

Warping Pins

A member that is subject to warping effects, like a WF or channel shape, will still experience warping stresses, even if warping restraint is not provided at the ends of the member. RISA-3D currently does NOT consider any warping effects for members that have warping "pinned" end conditions. The addition of warping effects for members with warping pins will be addressed in a future program version.

Torsional Stresses

RISA-3D calculates and lists the torsional stresses for the members of the structure, including the warping stresses.

Pure torsional shear is calculated for all non-warping "open section" shapes based on the equation:

$$\tau := \frac{M_x \cdot t}{J}$$

M_x = Torsion Moment

t = Maximum Thickness of Any Part of the Cross Section

This is the only torsional stress calculated for non-warping shapes. Shape types that are not “open cross sections” will have their shear stresses calculated with equations that are appropriate for each type.

For warping shapes (I's and C's), three separate stresses are calculated: pure torsion shear, warping shear and warping normal (bending) stresses. These results are all listed for review. The equations used to calculate these values won't be listed here but they are contained in the reference.

Code Check

These torsional stresses are included when the AISC code check (ASD or LRFD) is calculated for the member.

The ‘shear stresses’ (pure torsion and warping) are included in the required shear force that is then used in the **Shear UC** check:

$$V_r = (\tau + \tau_t + \tau_w) * A_v$$

Where,

V_r = required shear force

τ = shear stress (from the **Member Stresses** spreadsheet)

τ_t = pure torsional shear stress (from the **Member Torsion** spreadsheet)

τ_w = shear stress due to warping (from the **Member Torsion** spreadsheet)

A_v = shear area

The warping normal stresses are added to the ‘weak axis bending stresses’ for calculation of the required weak axis bending moment that is then used in the **Max UC check**. By “weak axis bending stresses”, we mean the bending stresses produced by moments about the local y-axis. When including shear stresses from torsion in the shear code check, the program uses the worst case of the torsional shear stresses (top and bottom flanges or web) and combines that with the actual shear stresses due to pure flexure. This is intentionally conservative in cases where the worst case torsional shear occurs in the flange, but the worst case flexural shear occurs in the web.

$$M_{ry} = (\sigma_b + \sigma_w) * S_y$$

Where,

M_{ry} = required weak axis bending moment

σ_b = bending stress (from the **Member Stresses** spreadsheet)

σ_w = warping stress (from the **Member Torsion** spreadsheet)

S_y = elastic modulus of elasticity about the weak axis

Applied Torsional Loads

You can apply member point torques along the length of the member. However, these loads are intended to work only for non-warping members. The reason is that there are many derivatives necessary to properly handle these member point torques when applied to warping members. The calculations for non-warping members is quite simple, but for warping members they're complex. This may be added in a future version.

Cardinal Points

Overview

A detailing layer is available in RISA-3D that lets you set the "true" elevations and locations start/end, top of steel, etc...) for all members. For each member you can describe the connection point at member ends.

The areas where member detailing parameters can be added/viewed are:

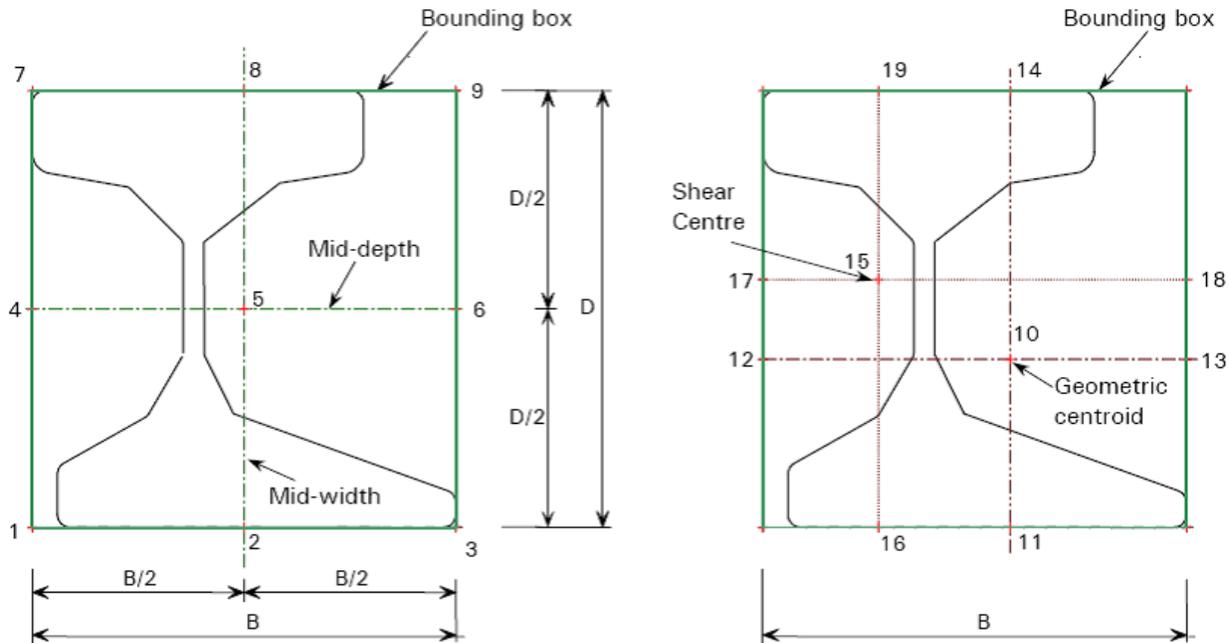
- RISA-3D
 - Drawing members ▶ **Properties** panel ▶ Detailing Properties
 - Click member ▶ **Properties** panel ▶ Detailing Properties
 - **Data Entry** toolbar ▶ **Members** spreadsheet ▶ Detailing tab
 - **View** tab ▶ **Model View Settings** button ▶ **Members** tab ▶ Detailing Info



Note: The Member Detailing information listed above is only visible if the setting "Show Detailing Info for members (CIS/2 Translator only)" is checked. It is accessible from the **3D Button** ▶ **Application Settings** button ▶ **General** tab.

For each end of the member, both cardinal point positions and decimal local offsets are used to described the connection location. The location of the cardinal point is plotted as follows:

Cardinal point Location and Local Axis



Cardinal point 1: lower left corner of the member section bounding box
Cardinal point 2: lower center point in the member section bounding box
Cardinal point 3: lower right corner of the member section bounding box
Cardinal point 4: mid-depth left point in the member section bounding box
Cardinal point 5: mid-depth center point in the member section bounding box
Cardinal point 6: mid-depth right point in the member section bounding box
Cardinal point 7: upper left corner of the member section bounding box
Cardinal point 8: upper center point in the member section bounding box
Cardinal point 9: upper right corner of the member section bounding box
Cardinal point 10: Geometric centroid of the member section



Note: Only cardinal point 1-10 are supported by RISA. You can instead use an offset.

The x, y, and z offsets are based on the local axis of the member. The x local axis is defined along the member from I to J. It coincides with the geometric centroid of the member section. The standard cardinal point positions (1-10), as well as the decimal local offsets are both supported in current detailing definition. If the cardinal point is set, the y and z offset values are automatically calculated and filled in the data structure. If there is a situation that doesn't match a cardinal point (angle brace as an example) you can also set the y and z offsets directly.



Note: The Analysis Offset and Rigid End Offset are completely different concepts from the detailing offset. The purpose of the detailing offsets is more realistic visualization and plotting of the model. The detailing offsets are not considered during the analysis. In the analysis, all members are still connected at their geometric centroid. Analysis Offset and Rigid End Offsets, on the other hand, affect the load distribution and design of the member. They are analysis parameters. In our program, the Analysis Offset / Rigid End Offset are completely separate data structure from the detailing offsets. When Analysis Offset is set for a model, for example, the detailing information is NOT automatically updated to cardinal point 8, accordingly.

Detailing Input and Modification

There are two ways for setting and modifying the detailing layer information. You can use the **Properties** panel or the **Members** spreadsheet to set the detailing information.

Default

When a member is drawn, by default, x offsets are set to be 0 on both ends. If the member type is beam, by default the y and z detailing offsets are on cardinal point 8 (top center) for both ends. If the member type is column, by default the detailing offset are on cardinal point 10 for both ends.

Properties panel

When drawing or modifying a member, the **Detailing Properties** section in the **Properties** panel contain member detailing information. You can choose to “set y, z offset using cardinal point” or “set y, z offsets directly”. When a cardinal point is picked, the value in the “y Offset” and “z Offset” text box are automatically updated.

Spreadsheets

A detailing tab is available on the **Members** spreadsheet. The spreadsheet shows a label and 8 values for each member. The “I cardinal point” and “J Cardinal Point” are dropdown lists with numbers and a description. When you pick a cardinal point, the y and z offset values update automatically based on the current shape. When a new y and z offset are input by you and it doesn't fall on any cardinal point, the “Cardinal Point” automatically changes to “None”. In the **Detailing** tab of the **Members** spreadsheet, the Recalculate button recalculates values based on new shapes.



Note: When the design shape of the member is changed, the detailing offsets of the member is not automatically updated. The Recalculate button needs to be clicked to get new y, z offset with respect to the cardinal point picked.

Visualization of the Detailing Layer

To visualize the model with the detailing layer, go to the **View** tab ▶ **Model View Settings** button ▶ **Members** tab and check on Detailing Info.



Note: When the **Detailing Info** box is checked, the members are plotted solely using the x, y and z offsets. The “Analysis Offset” and “Rigid End Offset” are not considered in the plot.

File I/O

[.MEMBERS_DETAILING_DATA] data section is added to the .R3D file to hold the member true location data. [..RF_COLUMNS_DETAILING_DATA] and [..RF_BEAMS_DETAILING_DATA] data sections are added to the .RFL file to hold member true location data. The CIS2 translator is updated to read/write this new data section.

Members - Results

When the model is solved, the results are separated into material specific design results and generic results. The generic member results are discussed in this section. The material specific design results are discussed in the following sections of the manual: [Hot Rolled Steel - Design](#), [Concrete - Design](#), [Cold Formed Steel - Design](#), and [Wood - Design](#). For information on **Member Detail Reports** see the [Results](#) section.

Number of Reported Sections

Note that the member results (forces, stresses, code checks) are only reported at the section locations. For example, if you set the **Number Of Reported Sections** in the **Model Settings Dialog** to be '2', you will not get any results for the middle of your member, you will only get results for the end points. If you have a point load applied to your member at a location that is not a section location, you will probably not report the maximum moment in the section if it does not occur at an endpoint.

Adjust Number of Sections

To adjust the number of Sections:

1. Go to the **Model Settings** window.
2. Click on the **Solution** tab, if it's not already open.
3. Adjust the **No. of Reported Sections** as needed.



Note: Adjusting the number of sections affects the amount of output.

Number of Internal Sections

Internally, the program subdivides the member into equally spaced sections to calculate forces, stresses, code checks, etc. The **Number of Internal Sections** can be adjusted in the **Model Settings Dialog**. If this value were set to 100, this would mean that for a member that is 100ft long, RISA-3D will calculate these values at approximately every foot. These values are then used in the member steel, wood, and concrete code checks, the diagrams in the model view, and in the detail reports. The locations of the maximum code checks are reported at a distance from the I-joint.

Adjust Number of Internal Sections

To adjust the number of internal Sections:

1. Go to the **Model Settings** window.
2. Click on the **Solution** tab, if it's not already open.
3. Adjust the **No. of Internal Sections** as needed.

All other results are reported at the **Number of Reported Sections** that you specify in the **Model Settings Dialog**. This controls how many places you receive **reported** or **printed** member force, stress, torsion, and deflection results. These locations are also equally spaced so setting the value to 5 will give you 5 equally spaced sections; at the ends, the middle and the quarter points.



Note:

- Adjusting the **Number of Internal Sections** will not affect the amount of output in the results spreadsheets.
- You may want to stick with odd numbers for the **Number of Reported Sections**. Setting the **Number of Reported Sections** to an even number will not report forces/stresses at the midpoint of the member, which is often the location of maximum moment.
- Setting the **Number of Reported Sections** to '2' will only report end forces which, might be desirable for connection calculation but not when looking for maximum forces along a members length. There is a printing option for end forces that is the equivalent to setting the **Number of Reported Sections** to '2' but allows you to see more results on screen while printing only the end forces.

Member Force Results

Access the **Member Section Forces** spreadsheet by going to the **Results** Menu and then selecting **Member Forces** then clicking the **Sections** tab ▶ .

Envelope Member Section Forces															
Sections	Member	Sec		Axial[k]		y Shear[k]		z Shear[k]		Torque[k-ft]		y-y Moment[k-ft]		z-z Moment[k-ft]	
					LC		LC		LC		LC		LC		LC
1	M70	1	max	0	17	3.478	9	0	17	0.001	1	0	17	0	17
2			min	0	1	2.884	2	0	1	0	5	0	1	0	1
3		2	max	0	17	1.739	9	0	17	0.001	1	0	17	-9.734	17
4			min	0	1	1.442	2	0	1	0	5	0	1	-11.739	3
5		3	max	0	17	0	17	0	17	0.001	1	0	17	-12.978	17
6			min	0	1	0	1	0	1	0	5	0	1	-15.651	3
7		4	max	0	17	-1.442	17	0	17	0.001	1	0	17	-9.734	17
8			min	0	1	-1.739	3	0	1	0	5	0	1	-11.739	3
9		5	max	0	17	-2.884	17	0	17	0.001	1	0	17	0	17
10			min	0	1	-3.478	3	0	1	0	5	0	1	0	1
11	M71	1	max	0.001	11	2.04	8	0	17	0	15	0	15	2.209	8

Envelope Maximum Member Section Forces																				
Sections	Member	Maximums	End Reactions		2nd/1st Moment Ratios															
			Axial[k]	Loc[ft]	LC	y Shear[k]	Loc[ft]	LC	z Shear[k]	Loc[ft]	LC	Torque[k-ft]	Loc[ft]	LC	y-y Moment[k-ft]	Loc[ft]	LC	z-z Moment[k-ft]	Loc[ft]	LC
1	M70	max	0	18	17	3.478	0	9	0	18	17	0.001	18	1	0	18	17	0	18	17
2		min	0	0	1	-3.478	18	3	0	0	1	0	0	5	0	0	1	-15.651	9	3
3	M71	max	0.001	6	11	2.04	0	8	0	18	15	0	18	11	0	12	11	11.871	12	4
4		min	0	12	11	-3.68	12	4	0	12.188	11	-0.001	0	11	0	18	11	-5.519	6	4
5	M72	max	0	6	11	4.927	0	4	0	6	11	0	6	15	0	12	11	16.221	0	4
6		min	0	12	11	-4.782	18	8	0	12	11	0	0	11	0	18	11	-10.076	9.188	3
7	M73	max	0	18	17	3.478	0	9	0.25	0	10	0	18	15	1.124	9	10	0	18	17
8		min	0	0	1	-3.478	18	3	-0.25	0	14	0	0	5	-1.124	9	14	-15.651	9	3
9	M74	max	0	6	11	6.058	0	3	0	6	15	0	18	15	0	0	11	21.747	0	3
10		min	0	12	17	-3.645	18	4	0	0	11	0	0	5	0	6	11	-17.184	12	4
11	M75	max	0	17.008	17	3.352	0	9	0	17.008	17	0	17.008	15	0	17.008	17	0	17.008	17
12		min	0	0	1	-3.352	17.008	3	0	0	1	0	0	11	0	0	1	-14.253	8.504	7

Envelope Member End Reactions															
Sections	Member	Member End	End Reactions		2nd/1st Moment Ratios										
			Axial[k]	LC	y Shear[k]	LC	z Shear[k]	LC	Torque[k-ft]	LC	y-y Moment[k-ft]	LC	z-z Moment[k-ft]	LC	
1	M70	I	max	0	17	3.478	9	0	17	0.001	1	0	17	0	17
2			min	0	1	2.884	2	0	1	0	5	0	1	0	1
3		J	max	0	17	-2.884	17	0	17	0.001	1	0	17	0	17
4			min	0	1	-3.478	3	0	1	0	5	0	1	0	1
5	M71	I	max	0.001	11	2.04	8	0	17	0	15	0	15	2.209	8
6			min	0	15	1.393	13	0	1	-0.001	11	0	11	0.038	13
7		J	max	0	15	-1.354	2	0	15	0	11	0	15	5.886	5
8			min	0	11	-1.843	4	0	11	0	15	0	11	3.959	2
9	M72	I	max	0	11	4.927	4	0	11	0	15	0	15	16.221	4
10			min	0	15	3.693	1	0	15	0	11	0	11	11.235	1
11		J	max	0	15	-3.59	1	0	15	0	15	0	15	14.917	8
12			min	0	11	-4.782	8	0	11	0	11	0	11	10.311	1

Envelope Member 2nd/1st Moment Ratios											
Sections	Member	Maximums	End Reactions	2nd/1st Moment Ratios							
				y-y Moment [k-ft]	2nd/1st Ratio	Loc [ft]	LC	z-z Moment [k-ft]	2nd/1st Ratio	Loc [ft]	LC
1	M70	max	NC	NC				-9.803	1	9	18
2		min	NC	NC				-15.651	1	9	3
3	M71	max	NC	NC				9.981	1.016	12	13
4		min	NC	NC				9.214	0.989	12	17
5	M72	max	NC	NC				11.669	1	0	16
6		min	NC	NC				11.586	1	0	12
7	M73	max	1.124	1	9	10		-9.803	1	9	18
8		min	-1.124	1	9	14		-13.724	1	9	1
9	M74	max	NC	NC				15.834	1	0	2
10		min	NC	NC				15.739	1	0	16
11	M75	max	NC	NC				-14.253	0.894	8.504	7
12		min	NC	NC				-8.816	0.893	8.504	18

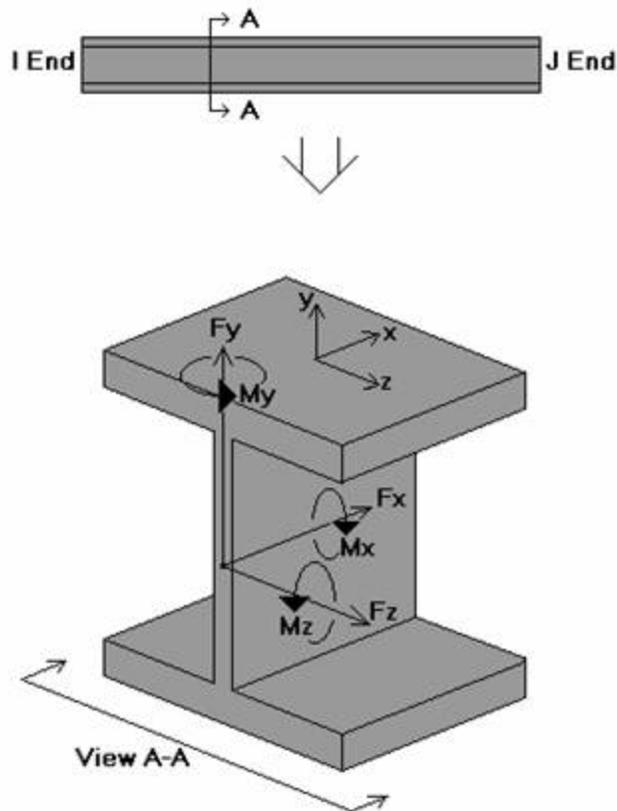
These are the member forces calculated along each active member taking into account any member offsets. The number of sections for which forces are reported is controlled by the **Number Of Reported Sections** specified in the **Model Settings Dialog**.

The **Maximums Tab** shows the maximum force that occurred in the member based on the **Number of Internal Sections**. Because the **Number of Internal Sections** is typically much larger than the **Number of Reported Sections**, this means that the **Maximums Tab** will always display the highest/lowest force in the member whereas the **Sections Tab** will miss the highest/lowest force under all but the most simple load diagrams.

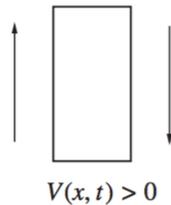
The **End Reactions Tab** shows the member forces at the ends of the member. This may not be useful for continuous members or cantilever members where the i-end and j-end may not be at a support.

The units for the forces are shown at the top of each column. As for the sign convention, the signs of these results correspond to the member's local axes, using the right hand rule. The left side forces at each section location are displayed. There are six force values for each section location.

These are axial, shear parallel to the local y axis (Shear y-y), shear parallel to the local z axis (Shear z-z), torque moment, moment about the member's local y axis (Moment y-y) and moment about the member's local z axis (Moment z-z). Please see the diagram below:

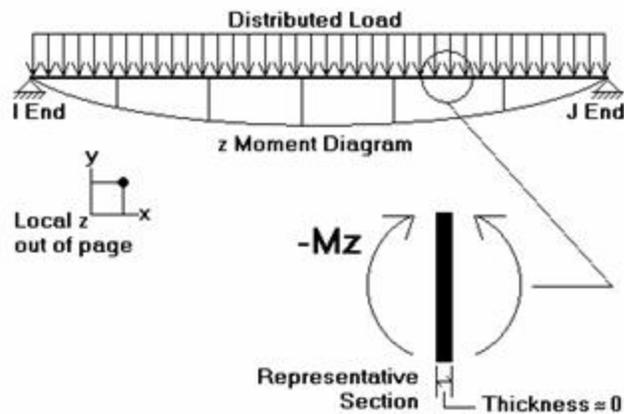


This diagram shows a member section location with all positive section forces. As can be seen, the section forces listed at any given section are the left side forces. For axial forces, compression is positive. For moments, counter-clockwise around the member axis is positive.



The sign convention for shear is positive when the free body diagram causes the member to spin clockwise (with I End on the left and J End on the right).

These section forces may also be displayed graphically. Remember that the section forces used for the plot are the left side forces. For an example of what you would see for the graphic plot of the moment diagram for a member, please see below:



RISA-3D uses the right hand rule **joint convention** and is always consistent with this convention. Since the left side moment is being used, a member under negative Mz moment would have the "holds water" deflected shape, which is contrary to some **beam conventions**. The opposite is true for M_y moments which will tend to "hold water" under a positive moment and "shed water" under a negative moment.

The **2nd/1st Moment Ratios Tab** shows the ratio of maximum moment with and without secondary P-Delta effect. In general, the larger the ratio is, the more significant the secondary effect is. Notice that some design codes may have a limit on this ratio (e.g., ACI has this limit as 1.4), but this limit is not checked by the program.

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.

The moving load results are enveloped and will display the Load Combinations with maximum and minimum values shown for each section location, for each active member. The governing load

combination and step location is shown for each result value under the "LC" column. The first number is the load combination, the second is the step number: (load combination - step number). See [Moving Loads](#) to learn more.

**Note:**

- See [Spreadsheet Operations](#) to learn how to use **Find**, **Sort**, and other options.
- See [Results View Settings - Members](#) to learn how to plot member results.

Member Stress Results

Access the **Member Stresses** spreadsheet by going to the **Results Menu** and then selecting the **Member Stresses** spreadsheet.

Member	Sec		Axial[ksi]	LC	y Shear[ksi]	LC	z Shear[ksi]	LC	y-Top[ksi]	LC	y-Bot[ksi]	LC	z-Top[ksi]	LC	z-Bot[ksi]	LC
1	M70	1	max	0	17	1.855	9	0	-7	0	17	0	17	0	17	0
2			min	0	1	1.538	2	0	-	0	1	0	1	0	1	0
3		2	max	0	17	0.927	9	0	-7	12.921	9	-10.714	17	0	17	0
4			min	0	1	0.769	2	0	-	10.714	2	-12.921	3	0	1	0
5		3	max	0	17	0	17	0	-7	17.228	9	-14.286	17	0	17	0
6			min	0	1	0	1	0	-	14.286	2	-17.228	3	0	1	0
7		4	max	0	17	-0.769	17	0	-7	12.921	9	-10.714	17	0	17	0
8			min	0	1	-0.927	3	0	-	10.714	2	-12.921	3	0	1	0
9		5	max	0	17	-1.538	17	0	-7	0	17	0	17	0	17	0
10			min	0	1	-1.855	3	0	-	0	1	0	1	0	1	0
11	M71	1	max	0	11	0.842	8	0	-7	-0.028	13	1.635	8	0	15	0
12			min	0	15	0.575	13	0	-	-1.635	8	0.028	13	0	11	0

These are the member stresses calculated along each active member. The number of sections for which stresses are reported is controlled by the **Number Of Reported Sections** specified in the **Model Settings Dialog**.

There will be seven stress values listed for each section location along the member taking into account any member offsets. The units for the stresses are shown at the top of each column. As for the sign convention, the signs of these results correspond to the signs of the forces. These line up as positive or negative according to the member local axis directions. Compression is positive and tension is negative.

The axial stress is the ratio P/A , where P is the section axial force. A positive stress is compressive, since the sign of the stress follows the sign of the force.

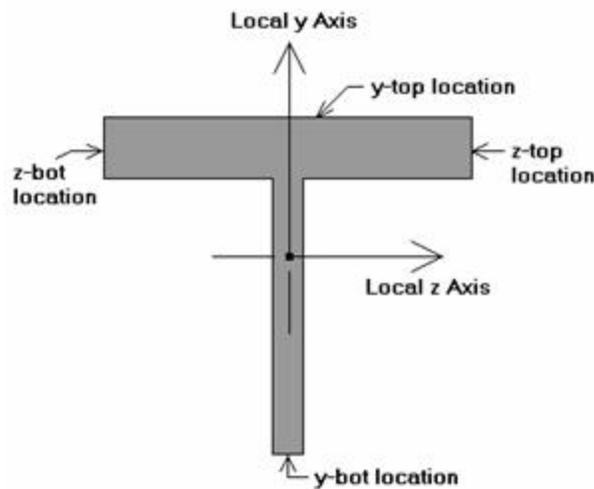
The shear stresses are calculated as V/A_s , where A_s is the effective shear area. The program obtains A_s by multiplying the total area by the shear stress factor. This factor is calculated automatically for most cross sections, but must be entered for Arbitrary members. Refer to [Member Shear Stresses](#).

The bending stresses are calculated using the familiar equation $M * c / I$, where "M" is the bending moment, "c" is the distance from the neutral axis to the extreme fiber, and "I" is the moment of inertia. RISA-3D calculates and lists the stress for the section's extreme edge with respect to the positive and

negative directions of the local y and/or z axis. A positive stress is compressive and a negative stress is tensile.

Note that two stress values are listed for each bending axis. This is because the stress values for a bending axis will not be the same if the shape isn't symmetric for bending about the axis, as with Tee and Channel shapes. The y-top and y-bot values are the extreme fiber stress for the + or - y-axis locations. The same is true for the z-top and z-bot stresses.

The locations for the calculated stresses are illustrated in this diagram:



So, the y-top location is the extreme fiber of the shape in the positive local y direction, y-bot is the extreme fiber in the negative local y direction, etc. The y-top,bot stresses are calculated using M_z and the z-top,bot stresses are calculated using M_y .

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.

The moving load results are enveloped and will display the Load Combinations with maximum and minimum values shown for each section location, for each active member. The governing load combination and step location is shown for each result value under the "LC" column. The first number is the load combination, the second is the step number: (load combination - step number). See [Moving Loads](#) to learn more.

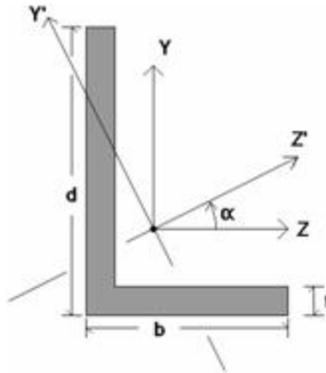


Note:

- A special case is bending stress calculations for single angles. The bending stresses for single angles are reported for bending about the principal axes.
- Torsional stress results are listed separately on the Torsion spreadsheet.
- See [Spreadsheet Operations](#) to learn how to use **Find**, **Sort**, and other options.
- See [Results View Settings - Members](#) to learn how to plot member results.

Single Angle Results

Depending on whether a single angle has been fully restrained against rotation or not, it will either behave about its geometric axes or its principal axes. This behavior can be controlled by correctly specifying the [unbraced lengths](#) for the angle. In the diagram below the z and y axes are the geometric axes. The z' and y' are the principal axes. The y' axis is considered to be the weak axis for principal behavior, and the z' is considered to be the strong axis.



The orientation of the shape is defined using the local y and z axes shown in the above diagram, but for principal axis behavior the bending calculations are done with respect to the y' and z' axes shown (the principal axes). The y' axis is the axis of minimum 'I' and the z' axis is the axis of maximum 'I'. RISA calculates the angle "α" and transposes the moments as shown below:

$$Mz' = Mz * \cos(\alpha) + My * \sin(\alpha)$$

$$My' = -Mz * \sin(\alpha) + My * \cos(\alpha)$$

The My' and Mz' moments are the moments shown as My and Mz respectively in the member forces results. Likewise, the y-top and y-bot bending stresses are relative to the extreme fibers along the y' axis (for the Mz' bending moment). The z-top, z-bot stresses are for My' bending at the extreme fiber locations along the z' axis.



Note: If both LcompTop and LcompBot have been set to zero then the angle will behave about its geometric axes and the member forces and stresses will be displayed relative to the geometric axes. Alternatively, setting the L-torque value to zero will also constrain the single angle to behave about its geometric axes.

Member Torsion Results

Access the **Member Torsions** spreadsheet by going to the **Results Menu** and then selecting **Member Torsions**.

Envelope Member Section Torsion															
Member	Sec		Torque[k-ft]	LC	Torsion Shear[ksi]	LC	y-y Warp Shear[ksi]	LC	z-z Warp Shear[ksi]	LC	z-Top Warp Bend[ksi]	LC	z-Bot Warp Bend[ksi]	LC	
11	M71	1	max	0	15	0	17	0	17	0	15	-0.016	15	-0.016	15
12			min	-0.001	11	0	1	0	1	-0.001	11	-0.027	11	-0.027	11
13		2	max	0	15	-0.006	15	0	17	0	15	-0.004	15	-0.004	15
14			min	-0.001	11	-0.01	11	0	1	0	11	-0.006	11	-0.006	11
15		3	max	0	15	-0.007	15	0	17	0	15	0	17	0	17
16			min	-0.001	11	-0.012	11	0	1	0	11	0	1	0	1
17		4	max	0	11	0.003	11	0	17	0	11	0.002	15	0.002	15
18			min	0	15	-0.003	15	0	1	0	15	-0.002	11	-0.002	11
19		5	max	0	11	0	17	0	17	0	11	0.009	15	0.009	15
20			min	0	15	0	1	0	1	0	15	-0.009	11	-0.009	11
21	M72	1	max	0	15	0	17	0	17	0	15	0.002	15	0.002	15
22			min	0	11	0	1	0	1	0	11	-0.002	11	-0.002	11
23		2	max	0	15	0.001	15	0	17	0	15	0.001	15	0.001	15

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 Reported Sections** option in the **Model Settings** window.

The units for the torsion stresses are shown at the top of each column. RISA-3D calculates pure torsion shear for any shape type; this value is based on the maximum thickness of any part of the cross section. Closed shapes such as tubes and pipes do not warp, nor do solid rectangular or circular shapes. For these shapes, there are no warping stresses to report. Warping only occurs in open cross sections where the rectangular pieces that make up the cross section do not all intersect at a single point. For example, a Tee shape could be thought of as two rectangular pieces, the flange and the stem. These two pieces intersect at the midpoint of the flange, so there is no warping. A channel, on the other hand, is comprised of three pieces, the two flanges and the web. These three pieces do NOT share a common point, so a Channel will warp. The same is true for a Wide Flange, so warping stresses are calculated only for I shapes (WF,S,H) and Channel shapes with warping restrained.

The shear and bending stresses caused by torsion are integrated into the code check and shear check calculations for the member, so your final code check (and final shear check) values DO include torsional effects. Warping shear is a shear stress acting parallel to the member's local y-and z-axis. Warping bending stress is a triangular stress normal to the cross section acting on the flanges, with the maximum stress at the outer edges of the cross section, the z-top and z-bot locations. As for the sign convention, the signs of these results correspond to the signs of the forces. These line up as positive or negative according to the member local axis directions. Compression is positive and tension is negative. See [Torsion](#) for more information on these calculations.

For enveloped results, the maximum and minimum value at each location is listed. The load combination producing the maximum or minimum is also listed, in the "LC" column.

The moving load results are enveloped and will display the Load Combinations with maximum and minimum values shown for each section location, for each active member. The governing load combination and step location is shown for each result value under the "LC" column. The first number is the load combination, the second is the step number: (load combination - step number). See [Moving Loads](#) to learn more.



Note:

- See [Spreadsheet Operations](#) to learn how to use **Find**, **Sort**, and other options.
- See [Results View Settings - Members](#) to learn how to plot member results.

Member Deflection Results

Access the **Member Section Deflections** spreadsheet by going to the **Results Menu** and then selecting **Member Deflections**

Service

Envelope Member Section Deflections Service															
Service	Strength	Beam Defl	Beam Check												
	Member	Sec		x [in]	LC	y [in]	LC	z [in]	LC	x Rotate [rad]	LC	(n) L/y' Ratio	LC	(n) L/z' Ratio	LC
1	M311	1	max	-0.093	19	-0.201	19	0.05	21	0.001	21	NC	21	NC	21
2			min	-0.095	18	-0.214	21	0.017	19	0.001	18	NC	18	NC	18
3		2	max	-0.057	19	-0.225	19	0.037	21	0.001	21	3566.3	18	NC	21
4			min	-0.059	18	-0.235	21	0.008	19	0.001	18	3460.047	21	NC	18
5		3	max	-0.022	19	-0.234	19	0.006	21	0.001	21	2268.41	18	NC	21
6			min	-0.023	18	-0.242	21	-0.014	19	0.001	18	2215.808	21	NC	18
7		4	max	0.014	19	-0.201	19	-0.027	21	0.001	21	3961.147	18	NC	21
8			min	0.012	18	-0.205	21	-0.038	19	0.001	18	3838.151	19	NC	18
9		5	max	0.05	19	-0.159	19	-0.049	18	0.001	19	NC	21	NC	21
10			min	0.048	18	-0.161	18	-0.054	19	0.001	18	NC	18	NC	18
11	M312	1	max	-0.049	18	-0.159	19	-0.048	18	0.001	21	NC	21	NC	21

The **Service** tab will contain deflection results only for load combinations marked as **Service** in the **Load Combinations - Design** tab.

Strength

Envelope Member Section Deflections Strength															
Service	Strength	Beam Defl	Beam Check												
	Member	Sec		x [in]	LC	y [in]	LC	z [in]	LC	x Rotate [rad]	LC	(n) L/y' Ratio	LC	(n) L/z' Ratio	LC
1	M70	1	max	0	17	-0.018	15	0	13	0.003	7	NC	17	NC	17
2			min	0	13	-0.02	2	0	17	0.002	12	NC	1	NC	1
3		2	max	0	17	-0.386	13	0	13	0.002	7	624.899	17	NC	17
4			min	0	13	-0.459	3	0	17	0.002	12	518.176	7	NC	1
5		3	max	0	17	-0.547	10	0	13	0.002	7	445.24	13	NC	17
6			min	0	13	-0.649	3	0	17	0.001	12	369.201	7	NC	1
7		4	max	0	17	-0.43	10	0	13	0.001	7	624.899	13	NC	17
8			min	0	13	-0.503	3	0	17	0.001	1	518.176	3	NC	1
9		5	max	0	17	-0.106	4	0	13	0.001	7	NC	17	NC	17
10			min	0	13	-0.123	1	0	17	0	1	NC	1	NC	1
11	M71	1	max	0	11	-0.104	5	0	12	0	15	NC	17	NC	17

The **Strength** tab will contain deflection results only for load combinations NOT marked as **Service** in the **Load Combinations - Design** tab.

The **Service** and **Strength** tabs are the member deflections calculated along each active member. The number of sections for which deflections are reported is controlled by the **Number Of Reported Sections** specified on the **Model Settings Dialog**.

The member section deflections are comprised of 3 translations in the member local axis directions, the rotation (x Rotate) about the local x-axis (the twist), and the relative deflection to length ratios for the y and z deflections. The units for the deflections are shown at the top of each column. As for the sign convention, the signs of these results correspond to the member's local axes, using the right hand rule.

The L/y' and L/z' ratios are the total member length (minus member offsets) divided by the relative deflection. The deflection in this calculation is not the deflection shown in the columns to the left, which are the absolute deflections. The deflection used is relative to the straight line between the deflected positions of the end joints. For cantilevers, the deflection is relative to the original position of the member.

Expressed as an equation, $n = L/\text{deflection}$, where **n** is what is tabulated in the spreadsheet. The smaller the deflection, the larger the value. If 'NC' is listed, that means the 'n' value is greater than 10000 which is a very small deflection. The minimum value that will be shown is '1'. For example, if the deflection criteria is L/360, check here to make sure no tabulated values are **less than 360**. Greater than 360 is OK.

For enveloped results the maximum and minimum value at each location is listed. The load combination producing the maximum or minimum is also listed, in the "LC" column.

The moving load results are enveloped and will display the Load Combinations with maximum and minimum values shown for each section location, for each active member. The governing load combination and step location is shown for each result value under the "**LC**" column. The first number is the load combination, the second is the step number: (load combination - step number). See [Moving Loads](#) to learn more.

Note

- For members that have not been designated as a beam, the maximum member deflection ratio from the **Service** or **Strength** tab will be reported as the Max Defl Ratio at the end of the [member detail report](#).
- See [Spreadsheet Operations](#) to learn how to use **Find**, **Sort**, and other options.
- If you wish, you can go to the model view to plot and animate the deflected structure. The amount of deflection shown on the plot is controlled by the magnification factor. The joints are plotted based on the joint displacements, and these member deflections are used to plot the member's curvature between the joints. See [Results View Settings - Members](#) to learn how to plot member results.

Beam Deflections

Enveloped spreadsheet

Batch spreadsheet

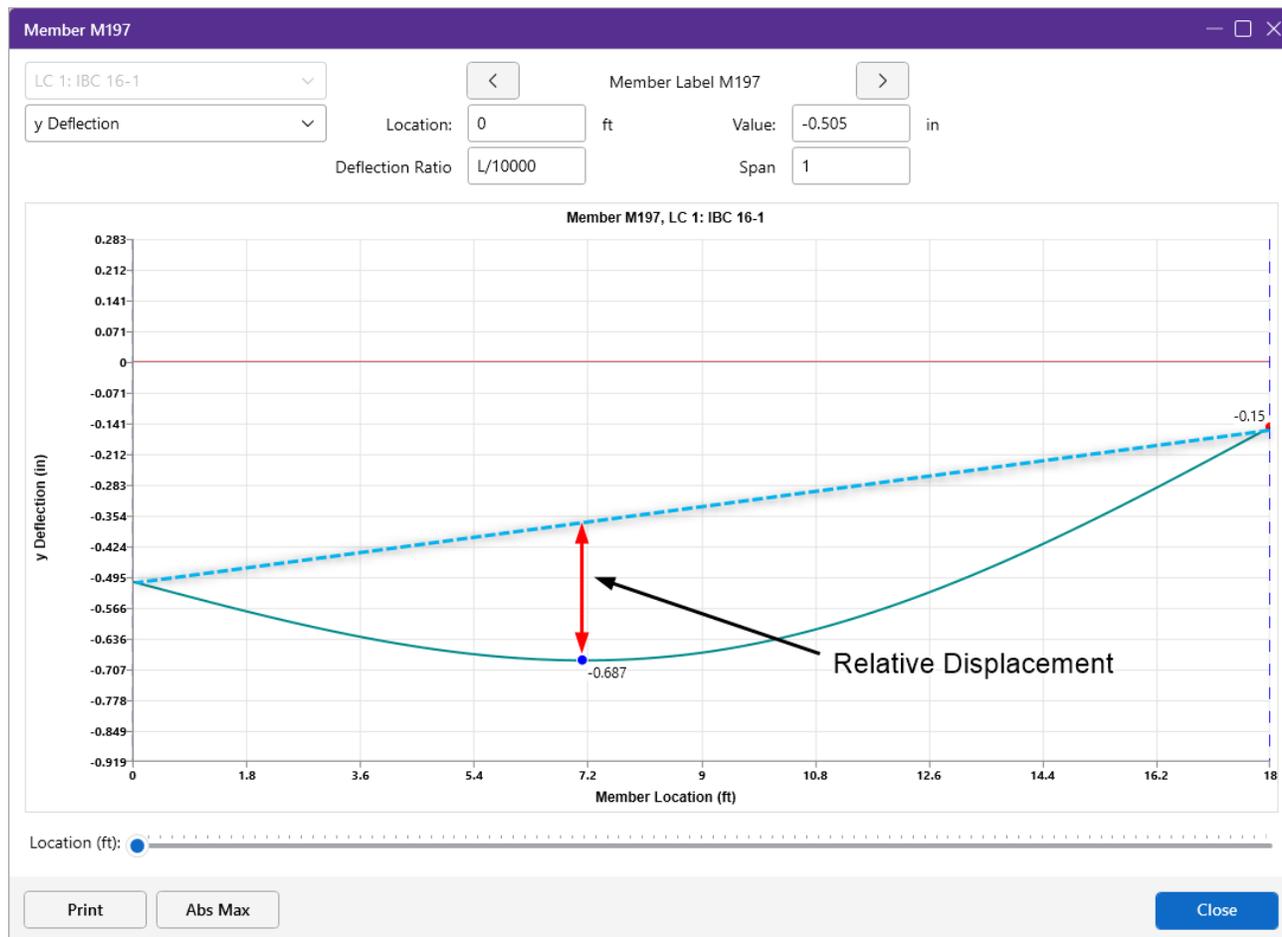
Service	Strength	Beam Defl		Beam Check			
	Member Label	Span		Location [ft]	y' [in]	(n) L'/y' Ratio	LC
1	M70	1	max	17.813	-0.016	NC	13
2		1	min	9	-0.585	369	7
3	M71	1	max	10.5	0.001	NC	7
4		1	min	5.25	-0.044	3293	4
5		2	max	17.813	0.001	NC	12
6		2	min	15.75	0.008	9241	8
7	M72	1	max	0.188	-0.002	NC	1
8		1	min	9	-0.236	917	3
9	M73	1	max	17.813	-0.02	NC	15
10		1	min	9	-0.731	295	7
11	M74	1	max	0.188	-0.003	NC	2

Service	Strength	Beam Defl		Beam Check			
	LC	Member Label	Span	Location [ft]	y' [in]	(n) L'/y' Ratio	
1	1	M70	1	9	-0.513	421	
2	1	M71	1	4.688	-0.026	5625	
3			2	18	0	NC	
4	1	M72	1	9.188	-0.159	1354	
5	1	M73	1	9	-0.641	336	
6	1	M74	1	10.125	-0.284	760	
7	1	M75	1	8.504	-0.516	418	
8	1	M76	1	7.125	-0.198	1091	
9	1	M77	1	8.812	-0.442	489	
10	1	M78	1	9	-0.153	1414	
11	1	M79	1	9.938	-0.185	1167	

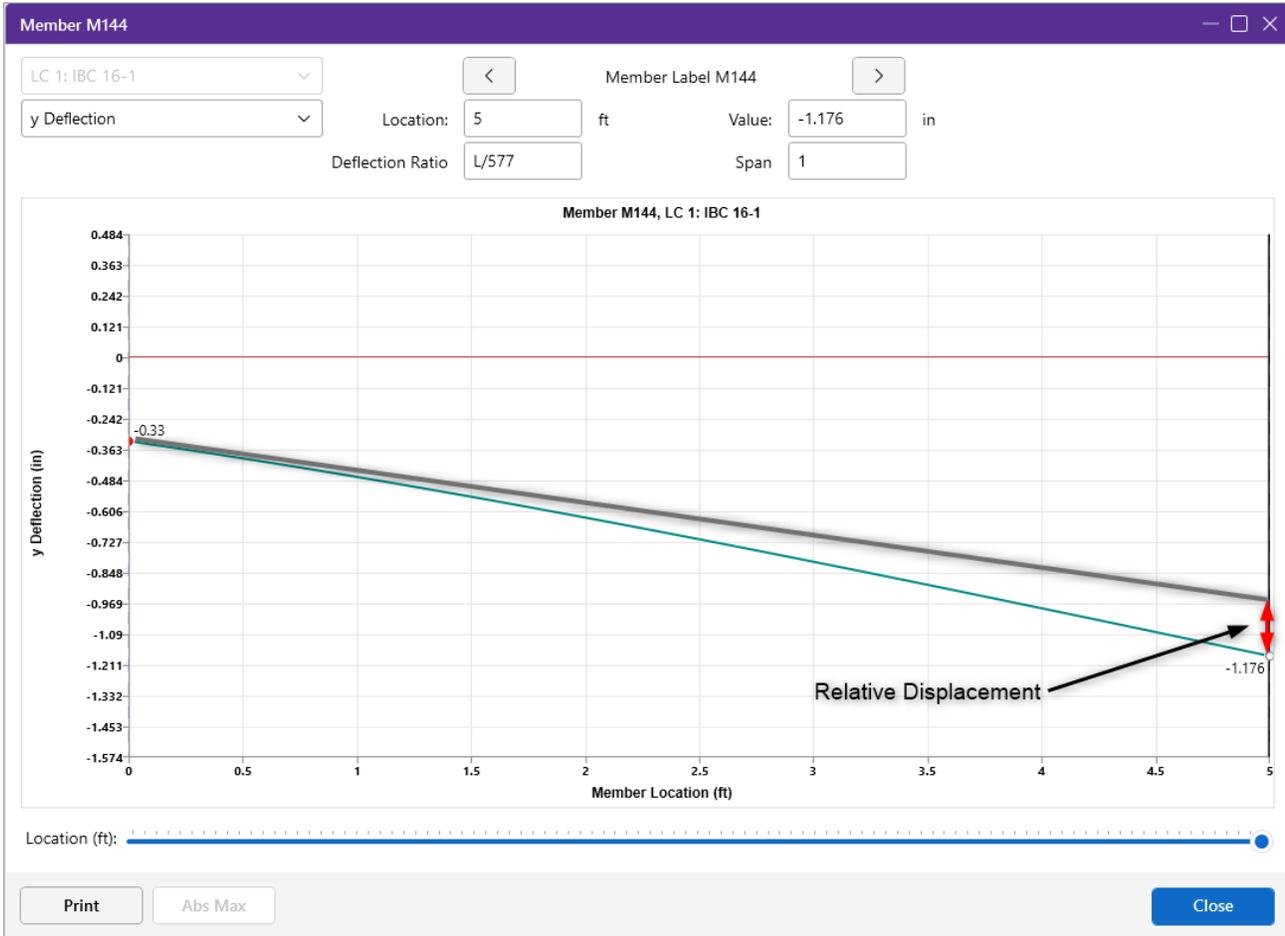
The **Beam Defl** spreadsheet shows the strong-axis (y) relative deflections, y' , for **Beam** member types. This spreadsheet will base the L'/y' Ratio on the actual span for multi-span beams. If there are multiple span beams, each span will have its own entry. The L'/y' Ratio uses the span length of the beam, denoted by L' (as opposed to the full member length, L) compared to the relative deflection, y' .

The program will detect the presence of cantilever spans. If an entire member or the end span(s) of a member are unsupported at one end, the program will consider those spans a cantilever and use 2*Span Length for the L'/y' Ratio calculations.

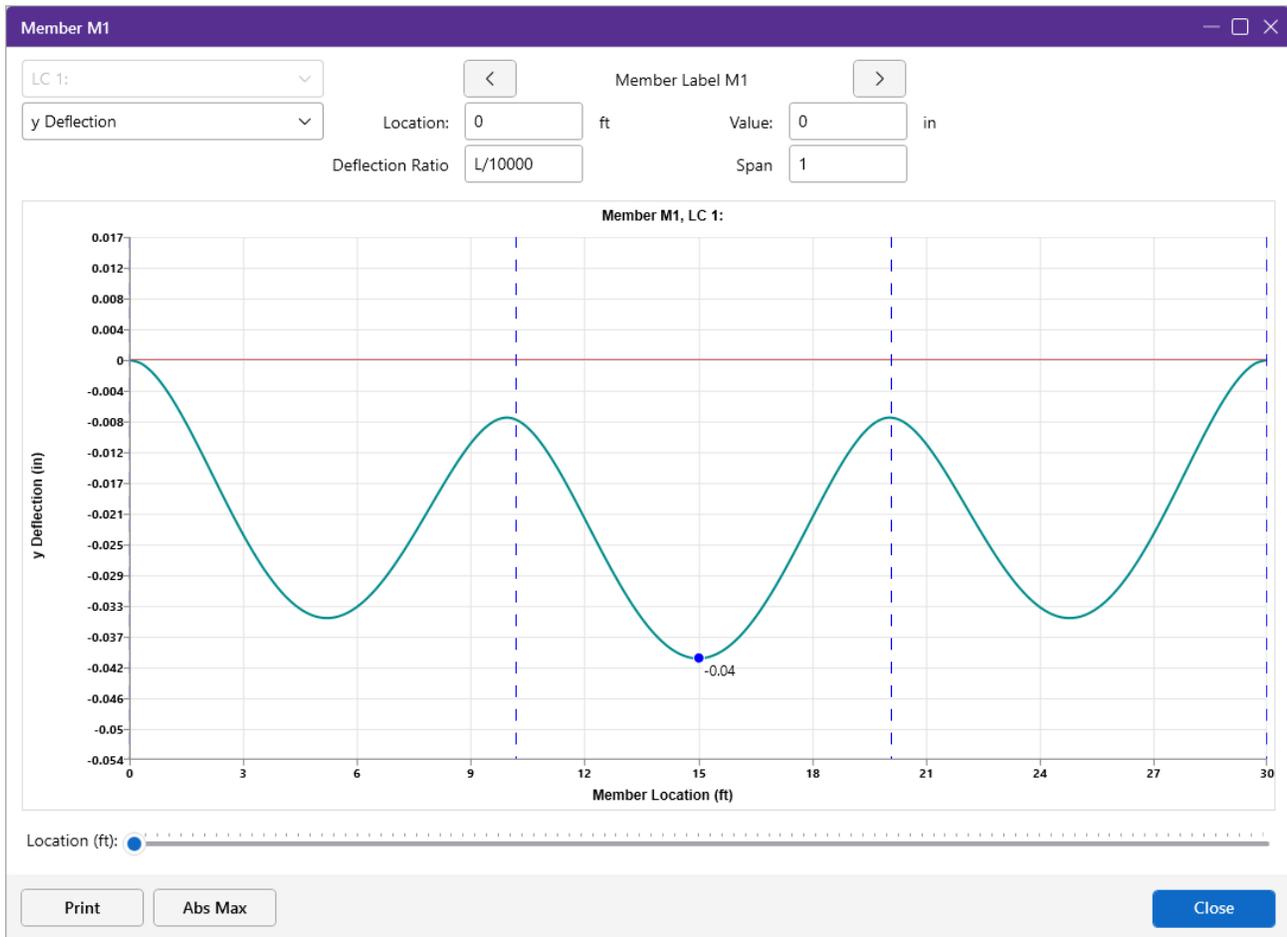
The location and value for the relative deflection, y' , is dependent on the beam being either supported or cantilever. For a member supported on both ends, the maximum relative displacement used in the beam deflection ratio will be the largest straight line distance (in the vertical y direction) between the deflected shape and the original undeflected shape.



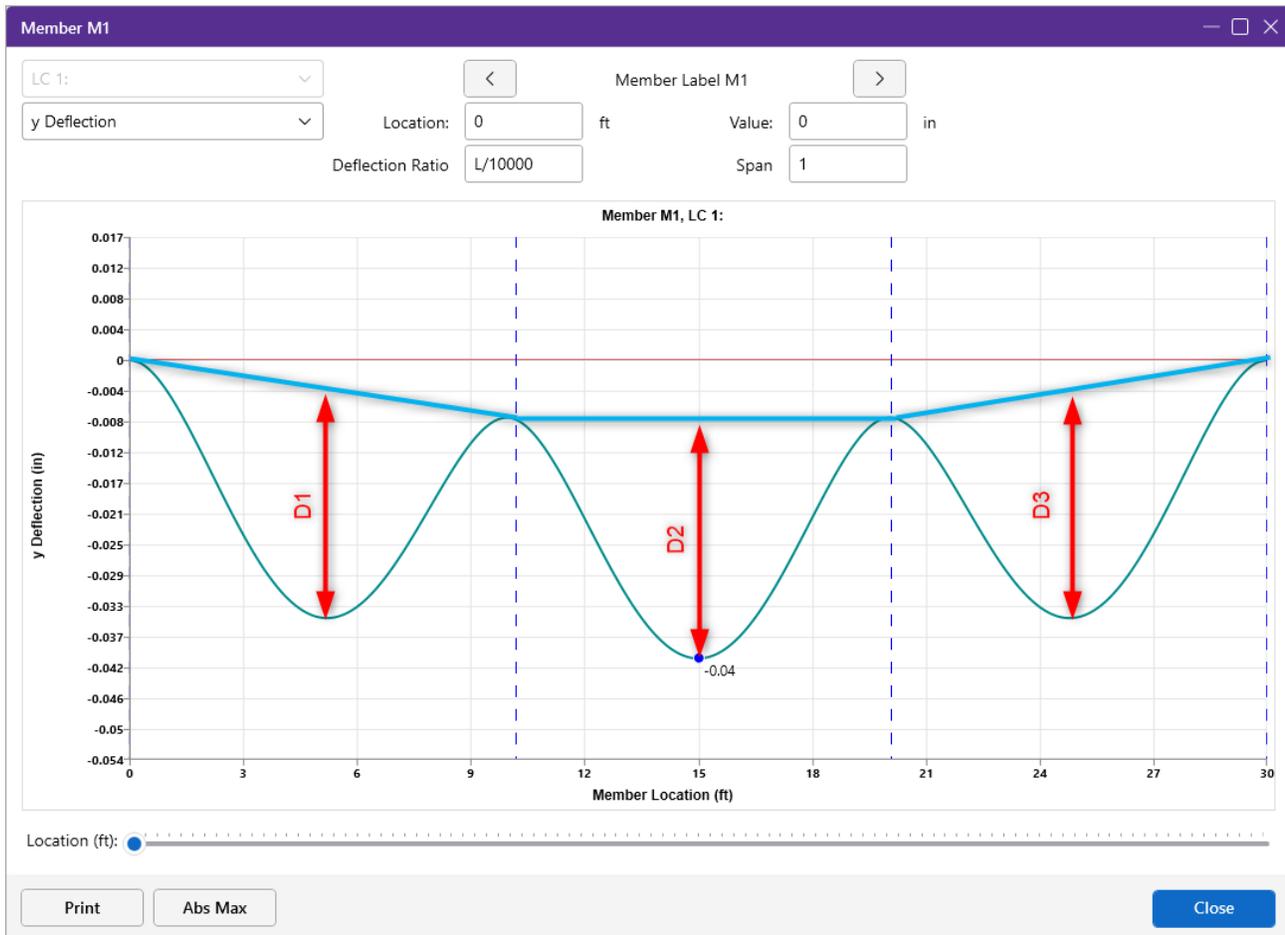
For members with a cantilever end, the location of the maximum relative deflection will be at the free end of the member. The displacement value is equal to the rotation at the supported end multiplied by the length of the member.



For multi-span members, the program will give span information in the expanded interactive strong-axis absolute deflection diagram. A vertical blue dashed line will be displayed at support locations which will give the proper span information. Note that any vertical reaction boundary conditions along the beam, and any columns (specifically members with their Member Type property set as Column) framing into the beam will be treated as support locations for the sake of determining the deflection ratio.



The L'/y' Ratio is based on relative deflection and member span length. The relative deflection for each span takes into account the span end deflections and is the maximum vertical deflection component measured from the absolute displacement to a straight line drawn from one end of the span to the other. In the image below, the relative deflection, y' , per span is called out as D1, D2, and D3.



The green line is the actual deflected shape.

The black vertical line (not shown in image) is the cursor marker in the expanded interactive strong-axis absolute deflection diagram.

The blue vertical dotted lines are the support locations.

The dark blue lines are straight line connectors between supports at the deflected location.

The dark red vertical lines are the relative strong-axis (y) deflection used in the L/'y' Ratio.

While the program determines if a member is a cantilever or supported based on member behavior and modeling configuration, there may be cases where the beam is actually considered otherwise. In this case, the [Deflection Ratio Options](#) can be used to designate a member end specifically as supported or cantilever. Any members with designated support ends using the Deflection Ratio Options will be denoted by a star suffix on the L/'y' Ratio listed in the **Beam Defl** spreadsheet.

Service	Strength	Beam Defl	Beam Check			
	LC	Member Label	Span	Location [ft]	y' [in]	(n) L'/y' Ratio
1	1	M1	1	5	-0.031	3879
2			2	15	-0.033	3681
3			3	25	-0.031	3879



Note:

- For members that have been designated as a beam, the maximum beam deflection ratio will be reported as the Max Defl Ratio at the bottom of the [member detail report](#).
- Overstrength load combinations will not be considered in this spreadsheet.
- There are no weak-axis (local z) deflections considered.

Beam Check

Service	Strength	Beam Defl	Beam Check									
	Beam	Design Rule	Span	Defl [in]	Ratio	LC	Defl [in]	Ratio	LC	Defl [in]	Ratio	LC
1	M1	Typical	1	-0.031	3879	1(1)	N/A	N/A	N/A	N/A	N/A	N/A
2			2	-0.033	3681	1(1)	N/A	N/A	N/A	N/A	N/A	N/A
3			3	-0.031	3879	1(1)	N/A	N/A	N/A	N/A	N/A	N/A

The **Beam Check** spreadsheet shows the local strong-axis (y) deflections for **Beam** member types for load combinations defined in the [Member Design Rules - Deflection tab](#). If a deflection value exceeds the limit from the design rule then the value will show in RED. This spreadsheet will also base the L/y ratios on the actual span for multi-span beams. If there are multiple span beams each span will have it's own entry.



Note:

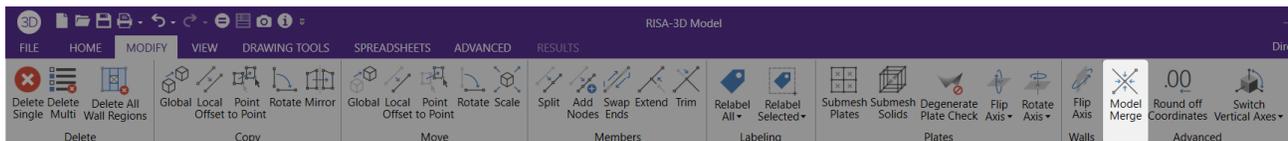
- If you're used the [Suggested Design](#) feature to optimize members in your model the program will consider these deflections in this optimization.
- Overstrength load combinations will not be considered in this spreadsheet.
- There are no weak-axis (local z) deflections considered.

Model Merge

Model Merge is a feature located on the **Modify** ribbon that scans through your model and automatically merges elements in the model. Model Merge detects unconnected nodes along member spans, unconnected crossing members and duplicate nodes, members and plates. You can use Model Merge to build models faster, as well as to detect and fix modeling errors.



Note: Model Merge will "connect" Physical Members by inserting a node at their intersection but will not break them up into smaller members. Model Merge also eliminates duplicate Physical Members so you should still use model merge to detect errors when working with these types of members. See [Physical Members](#) to learn more.



Knowing what Model Merge does allows you to skip modeling steps as you build your model and let the software perform these steps for you. You can take advantage of Model Merge in modeling your structure in many ways, some of which are discussed in the following links.

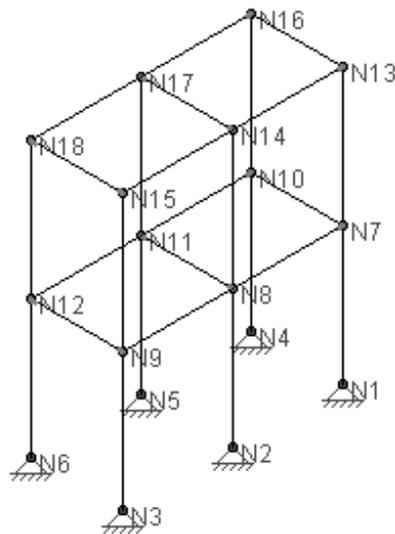
Model Merge Options

There are four main options for the model merge.

- If the **Merge Crossing Members** box is checked then as part of the merge process, all members will be scanned along their lengths for crossing members. Crossing members will be merged together at their intersection points. If you have cross bracing you may or may not want them to be merged. A finite element crossing a physical member will cause only the finite element to be broken up at the point of intersection. If two physical members are crossing, a node will be placed at the intersection point.
- **Merge Inactive Members** if left unchecked allows you to limit the merge process to the parts of your model that are selected. This allows you to prevent the program from merging portions of your model where you may have intentionally put nodes at the same location or have two members next to each other.
- **Trim/Extend Crossing Beams** can help connect member ends that are within the merge tolerance from an adjacent beam. Checking this box will move the end node of that member so that it lies on the adjacent beam.
- **Trim/Extend Crossing Wall Panels** can help when you have two wall panels that are overlapping or intersecting by a distance smaller than the merge tolerance. This allows you to easily correct minor modeling flaws. This will also correct the condition where a defined wall panel is non-coplanar.

A 'Merge Tolerance' feature that defines the maximum distance 2 nodes can be apart and still be merged together, affects the overall Model Merge feature. It is also used when scanning for crossing members and for unattached nodes along the spans of members. See [Merge Tolerance](#) under 'Model Settings' for more information.

Model Merge Examples



Looking at this frame, consider the column line on the right side, members 1-7 and 7-13. If you did not use Physical Members you would define this just that way, as two separate members. With the model merge capability you could instead enter a single member definition, 1-13, and let the model merge function break it up for you.

Other convenient uses of the model merge function are laying out floor plans and being able to draw all the nodes right over the main girders or defining truss chords as full length rather than specifying each panel point. The model merge will take care of breaking up the members at all the intersecting points. Of course the Physical Member feature goes a step farther for these situations because these members **never** need to be split to model a connection – allowing you to make edits and understand results more readily. See [Physical Members](#) to learn more.

Model Merge Limitations

Certain types of shape types and certain load types can cause members to not get broken up by the model merge function. In particular, members that are Tapered WF shapes will not get broken up by the model merge. Even if such members have intermediate unattached nodes, or crossing members within their spans, they will not be broken up.

Model Merge Process

1. Duplicate nodes are merged together.
2. All the members are scanned for other members crossing along their span. If a crossing member is found, a nodes is created at the intersection point.
3. All the members are scanned for nodes along their span. If found, the member is broken up into pieces to incorporate the nodes.
4. Duplicate members are merged together. Physical members take precedence over finite element members. Longer members take precedence over shorter members that are fully coincident with the longer member. When duplicate members with different section sets are merged, the set listed first in the Sections Sets spreadsheet are used.
5. Duplicate plates are merged together.

To better understand how the model merge function works, please refer to this figure:

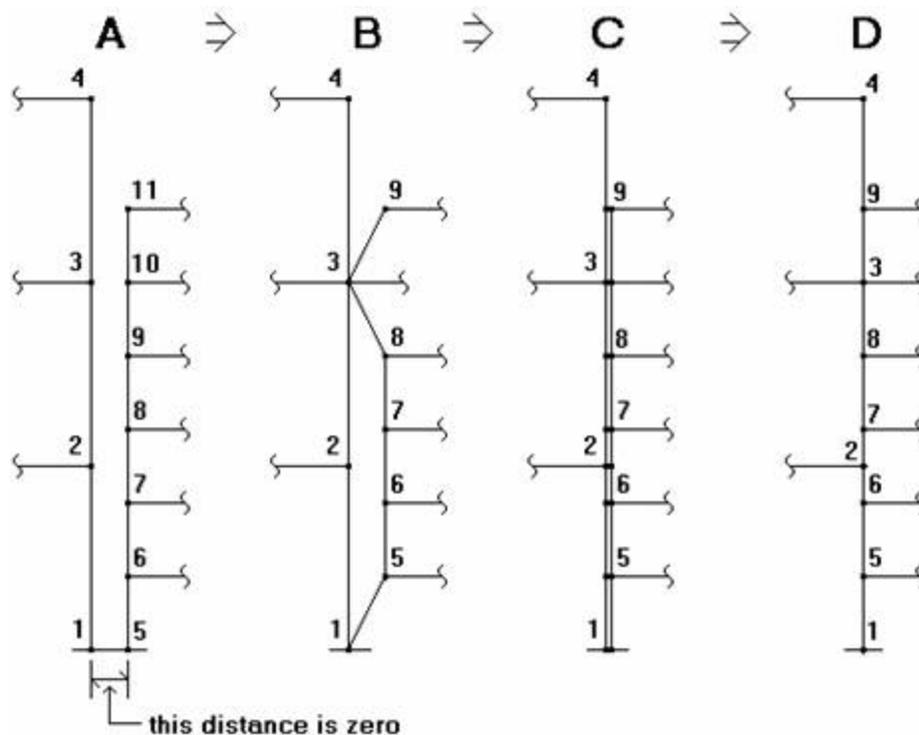


Diagram A shows the model before a merge. The two column lines are separated in diagrams A and B strictly for ease of viewing, they should be considered to be right on top of each other. Nodes 1 and 5 (in diagram A) have exactly the same coordinates, as do nodes 3 and 10. None of these members are Physical Members.

Step 1 of the merge eliminates duplicate joints changing the model from diagram A to diagram B. On diagram A, nodes 1 and 5 are duplicates (same coordinates), as are nodes 3 and 10. Nodes 5 and 10

are merged into nodes 1 and 3 respectively. This means any loads applied to nodes 5 and 10 are now applied to nodes 1 and 3. Any members connected to 5 and 10 are now connected to nodes 1 and 3 (these members are shown with the inclined lines in diagram B).

Step 2 looks for crossing members, however, there aren't any for this particular example. Members that are parallel to each other aren't treated as "crossing" since the end 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. The members with intermediate joints are broken up, shown in diagram C.

Step 4 eliminates duplicate members, in this case those that were created in step 2. This takes us from diagram C to diagram D. Looking at diagram C, the duplicate members are shown as the double lines. The first member listed on the **Member** spreadsheet is maintained and the other member is deleted. Any loads applied to the deleted member are transferred to the remaining member.

The final merged model is shown in diagram D. The column line is now comprised of 8 members, 1-5, 5-6, 6-2, etc. up to member 9-4.



Note: The direction code of loads merged for duplicate members and plates is kept the same. Elements with loads in local directions and different orientations may result in a load direction that is not be the same as the original direction.

Perform a Model Merge

To perform a Model Merge:

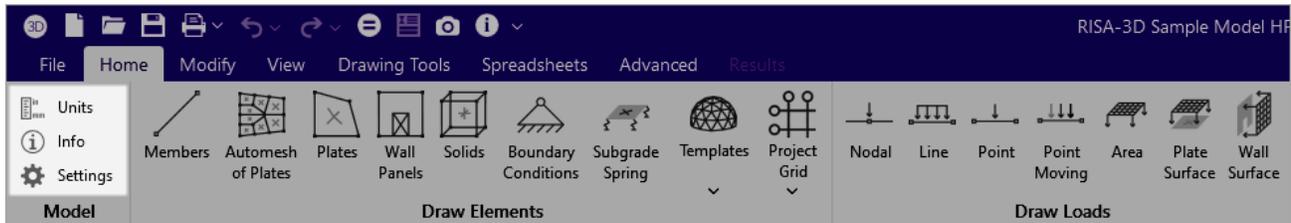
1. In the current model view window, select the items you wish to merge.
Typically you will want everything to be selected.
2. Go to the **Modify** ribbon.
3. Click on the **Model Merge** icon to bring up the model merge parameter window.
4. Set the parameters for the new merge.
For help on an item, click the **?** **Help** icon and then click the item.
5. Click **Apply All** or **Apply Selected** to apply the model merge to the entire model or items you selected.



Note: You may undo any mistakes by clicking the **Undo**  icon.

Model Settings

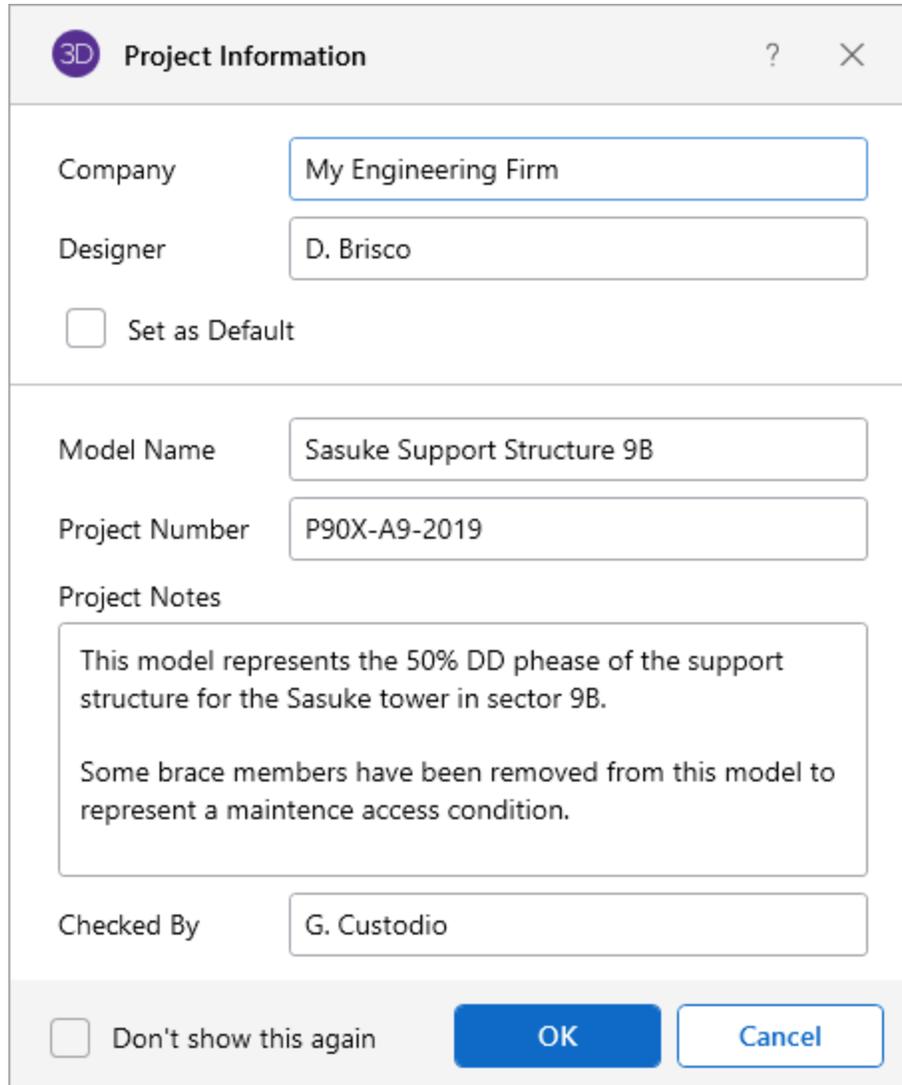
The model settings may be accessed on the **Home** ribbon in the **Model** section. The settings consist of three main options (**Units**, **Info** and **Settings**) that are highlighted in the following image.



- **Units:** The 'Units' option allows you to set the type of units (imperial, metric or both) you want to use in your model. Refer to the [Units](#) topic for more details on the units settings.
- **Info:** The 'Info' option allows you to enter descriptive information such as a title for the particular model being defined, the name of the designer and a job number. The title may then be printed at the top of each sheet of the output, and on the graphic printouts of the model.
- **Settings:** The 'Settings' option provides control for most of the model specific control settings. These Model Settings are specific to the model which is currently open. The information stored here is retained with the model, and will be retained even if the model is opened on a different computer.

You can save any of the settings in the Model section as the default setting, such that new models you create will begin with the settings you saved as Defaults. To do this, simply enter the information that you want to save and check the **Save as Defaults** box (see the image under [Project Information](#) for the location of this check box).

Project Information



The screenshot shows a dialog box titled "Project Information" with a 3D icon in the top-left corner and help and close buttons in the top-right. The dialog is divided into several sections:

- Company:** A text box containing "My Engineering Firm".
- Designer:** A text box containing "D. Brisco".
- Set as Default:** An unchecked checkbox.
- Model Name:** A text box containing "Sasuke Support Structure 9B".
- Project Number:** A text box containing "P90X-A9-2019".
- Project Notes:** A large text area containing two paragraphs:
 - "This model represents the 50% DD phase of the support structure for the Sasuke tower in sector 9B."
 - "Some brace members have been removed from this model to represent a maintenance access condition."
- Checked By:** A text box containing "G. Custodio".
- Don't show this again:** An unchecked checkbox.
- Buttons:** "OK" (blue) and "Cancel" (white with blue border) buttons.

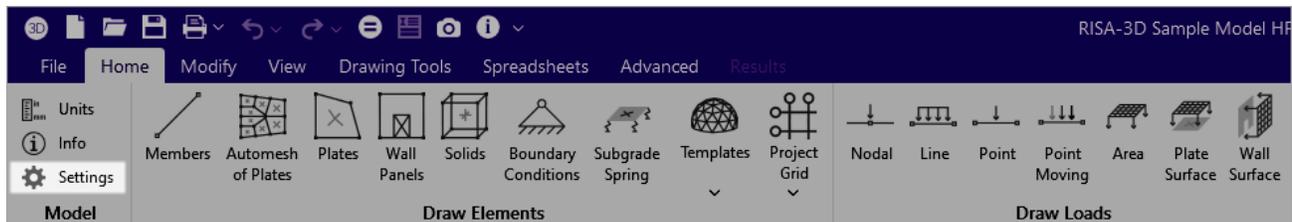
Project Notes is intended for information about the model that may be useful for the engineer or reviewer to refer to. This box is limited to 4000 characters.

Checked By is intended for the model reviewer to initial upon completion of a model review. Their initials are then printed on each sheet of the output.

Settings

The entries available from **Settings** provide most of the control for model specific settings. The following sections provide more details about these settings.

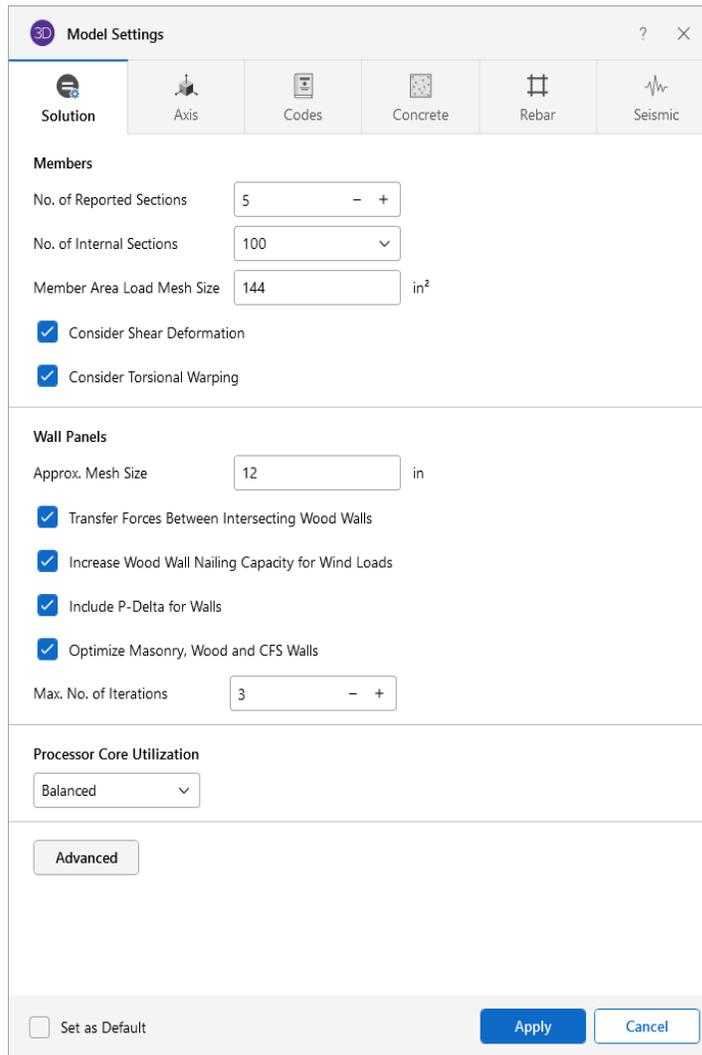
To open the 'Model Settings' window, click the **Settings** option from the 'Home' ribbon, as shown in the following image.



Note: You can check the **Save as Defaults** check box, to have you settings saved for the next time you open the program.

Solution

The **Solution** tab is the first tab in the Model Settings window. The entries on this tab are used to control settings that affect the general solution of the model ([Members](#), [Wall Panels](#), [Processor Core utilization](#) and [Advanced](#)).



Members

Solution tab - Members section

Option	Description
Number of Reported Sections	Number of Reported Sections controls how many places along each member you receive reported member force, stress, torsion, and deflection results. This only affects the amount of data displayed in the results spreadsheets, and has no effect on the solution of the model or the code checks. See Printing and Member Results for more information.
Number of Internal Sections	Number of Internal Sections controls how many places along each member the software calculates and stores results such as deflections and code checks. The member force diagrams

Option	Description
	<p>displayed in the model view and the detail plot are also drawn from these results. Increasing this value means that the program will make more "cuts" along the beam's length, which means it is more likely to hit the theoretical maximum and minimum values for code checks.</p> <div data-bbox="553 562 1409 1234" style="border: 1px solid black; padding: 10px; background-color: #e6f2ff;"> <p> Note:</p> <ul style="list-style-type: none"> Number of Sections cannot exceed 20. Also, Number of Internal Sections cannot be less than twice the Number of Sections. If unacceptable values are entered for either of these fields the program will automatically reset them to acceptable values. In the embedded version of RISAFoundation in RISAFloor or RISA-3D, the Beam Section Options will not appear. In these cases, the Model Settings from RISA-3D will control here. The Number of Sections and INTERNAL sections are remembered between RISAFloor and RISA-3D. If either of these is changed in RISA-3D, the RISAFloor results will be cleared as soon as RISAFloor is entered again. This is to keep results consistency between the programs. </div>
Member Area Load Mesh Size	The Member Area Load Mesh Size is used to determine the maximum size when meshing an area load and attributing the load to members. See Area Loads to learn more about this.
Consider Shear Deformation	Check the Consider Shear Deformation check box if shear deformation considerations are to be included in the model solution. See Member Shear Deformations for more information.
Consider Torsional Warping	The Consider Torsional Warping option considers torsional warping effects when calculating stiffness and stress values for shape types that warp. See Member Warping for more information.

Wall Panels

Solution tab - Wall Panels section

Option	Description
Approximate Mesh Size	Approximate Mesh Size is the base mesh size that is used when wall panels solved. If there is a constraint smaller than this mesh size, the mesh will be refined to accommodate this constraint.

Option	Description
	<p>You can see the mesh size graphically on the wall panels when you solve your model.</p> <div data-bbox="553 464 1409 569" style="border: 1px solid black; padding: 5px;">  Note: The mesh size is also used for plate sub-mesh for RISAFloor Concrete floor Semi-Rigid diaphragms. </div>
Transfer Forces Between Intersecting Wood Walls	<p>Transfer Forces Between Intersecting Wood Walls lets you decide whether wall panels framing into each other will transfer loads. Wall panels that are parallel to and touching each other will always transfer loads.</p> <p>This setting was added for times when intersecting perpendicular walls are not actually attached to each other, thus they may separate if loading conditions cause this.</p>
Increase Wood Wall Nailing Capacity for Wind Loads	<p>Increase Wood Wall Nailing Capacity for Wind Loads automatically increases the shear capacity of wood wall and diaphragm panels by 1.4, for all load combinations that contain wind loads. Load combinations that have both wind and seismic loads acting simultaneously will not receive this increase.</p>
Run a P-Delta Analysis for Wall Panels	<p>Run a P-Delta Analysis for Wall Panels is used to enable P-Delta analysis for wall panels. Even if this box is checked, the P-Delta analysis will only be performed on load combinations that have P-Delta enabled.</p>
Optimize Masonry and Wood Walls	<p>Optimize Masonry and Wood Walls defines whether you want to automatically iterate the solution for masonry and wood walls.</p>
Maximum Numbers of Iteration	<p>Maximum Numbers of Iterations defines how many automatic iterations will occur.</p>

Option	Description
	 <p>Note:</p> <ul style="list-style-type: none"> • This automatic iteration procedure will only happen for Batch or Envelope solutions. This is because we do not want to update your wall panel thicknesses if you are only solving a DL combination. In that case the program would then downsize your panel unnecessarily. • For more information on masonry wall optimization see the Masonry Wall - Design topic. • For more information on wood wall optimization see the Wood Wall - Design topic. • Concrete wall design will be done automatically and the selections here will not affect this design. This is because concrete wall reinforcement changes will have very little effect on the stiffness of the walls. • The program will stop the iteration procedure if the results from the previous solution match the results from the current solution.

Processor Core Utilization

Solution tab - Processor Core Utilization section

Option	Description
Single	<p>Single lets you use a single core to run the model solution.</p> <p>This option is useful when you have several other computer programs that are CPU intensive running in conjunction with RISA-3D.</p>
Multiple (Optimum)	<p>Multiple (Optimum) lets you use half of the available cores on your computer to run the solution.</p> <p>This option provides a faster solution time compared to using a single core but still allows half of your computer's available cores to be used for other programs and computer processes.</p>
Maximum	<p>Maximum lets you use all of the available cores on your computer minus one.</p> <p>For instance, if you have 8 available cores and select the Maximum option the program uses 7 cores in the processing of the solution.</p>

Option	Description
	This option provides the fastest solution time but still apportions a single core to be used for other programs and computer processes.

Advanced

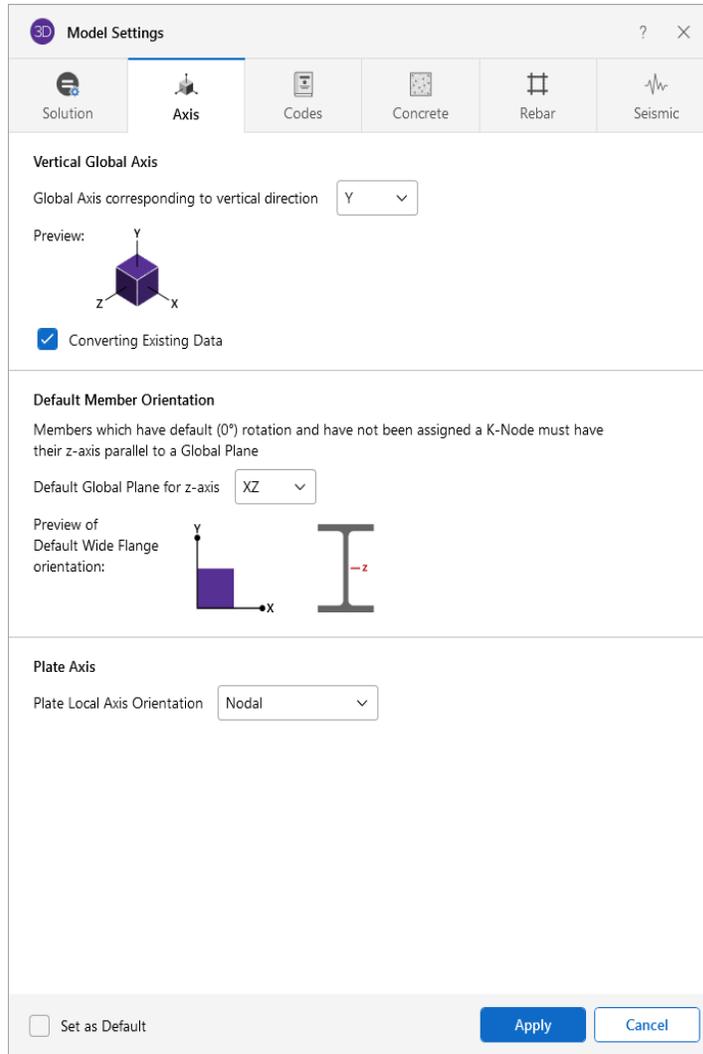
Advanced Options

Option	Description
P-Delta Convergence Tolerance	P-Delta Convergence Tolerance is used to adjust the tolerance used to determine convergence of the P-Delta analysis. Be sure to enter this value as a <i>percentage</i> ! The default for this is ½ of 1 percent (.5%). See P-Delta Analysis to learn more about this.
Eigensolution	Eigensolution Convergence Tolerance is used to set how close a

Option	Description
Convergence Tolerance	subsequent solution must be to the previous solution for a mode to be considered converged. See Dynamic Analysis for more information.
Gravity Acceleration	<p>Gravity Acceleration is used to convert loads into masses for the purpose of a dynamic analysis.</p> <p>In previous versions, the default Subgrade Modulus and Allowable Bearing were entered in the Model Settings. In version 10.0 and higher, the default subgrade modulus and allowable bearing pressure are entered in the Soil Definitions spreadsheet. There is a check box in the Soil Definitions spreadsheet to determine which soil definition will be used as the default. The default soil definition properties will be applied to the entire model except where another soil region is drawn in the model. It is not necessary to draw a soil region for the default soil definition.</p>
Merge Tolerance	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 joints along the spans of members.
Static Solver	<p>Static Solver lets you define the solver to be used during the solution.</p> <p>This setting can be selected by clicking on the Static option and then choosing Standard Skyline or Sparse Accelerated from the drop down arrow.</p> <p>See Solution for more information on these two options.</p>
Dynamics Solver	<p>Dynamics Solver lets you to choose between the Standard Solver, Accelerated Solver and Ritz Vector Solver solutions. Refer to the Solution topic for a more details.</p> <ul style="list-style-type: none"> • Standard Solver uses a simple sub-space iteration to solve for the natural frequencies. This solver has been used for years and the accuracy of the results is very well established. It has been included only for comparative / verification purposes. • Accelerated Solver uses an accelerated sub-space iteration with a Lanczos starting vector. The accelerated solver is the default and should produce solution in a fraction of the time that the standard solver would take to produce them. • Ritz Vector Solver does not solve for true mode shapes and natural frequencies. However, it is generally the best choice when running a response spectra analysis. The use of Load Dependent Ritz vectors provides a more accurate

Option	Description
	response spectra analysis for the same number of modes.
Time History	Time-History lets you choose between the Modal Superposition and Direct Integration solution methods. It also allows the user to set the damping preferences for each of these solution methods. Refer to the Solution and the Dynamic Analysis - Time History topics for more details on these settings.
P-Delta Analysis	The P-Delta Analysis options allow the user to choose between the Standard Nodal Shear method and Geometric Nonlinear Stiffness method. Refer to P-Delta Analysis for more details on these two methods.
Diaphragm Stiffness	Diaphragm Stiffness controls the stiffness of the rigid diaphragm. This is set to a unitless value of 1×10^7 by default. This value has been calibrated as providing the best behavior for most models. For more information refer to the Diaphragm topic.

Axis



Axis tab

Option	Description
<p>Vertical Axis</p>	<p>Vertical Axis can be set here as well. Setting the vertical axis may require you to set the Default Member Orientation also shown. As you specify new members RISA-3D will try to orient them correctly.</p> <p>The member local z-axis is typically the strong axis for a member and RISA-3D will orient members such that when you draw non-vertical members (beams) they will automatically be oriented such that loads in the vertical direction are resisted by the strong axis bending of the member. See Member Local Axes and Defining Member Orientation for more information.</p>

Option	Description
Convert Existing Data	Check the Convert Existing Data check box to convert the existing orientation data to the newly selected vertical global axis.
Plate Local Axis Orientation	Plate Local Axis Orientation can be designated here as 'Global' or 'Nodal'. This can be changed at any time, and modifies the axes of all plates. If set to 'Global', the local axes of all plates will be oriented in the same orthogonal direction, regardless of plate geometry. However, the positive out-of-plane axis is dependent on the configuration in which the plate or mesh was drawn. If set to Nodal, the positive orientation of the plate axes depend on the manner in which the plate was created. See Drawing Plates and Submeshing Plates for more information.

Codes

The screenshot shows the 'Model Settings' dialog box with the 'Codes' tab selected. The dialog is organized into a table with columns for 'Materials', 'Codes', and 'Stiffness Adjustment'. The 'Codes' column contains dropdown menus for various material types, and the 'Stiffness Adjustment' column contains checkboxes for iterative analysis.

Materials	Codes	Stiffness Adjustment
Hot Rolled Steel	AISC 14th (360-10): LRFD	Yes (Iterative)
	Seismic Detailing: AISC 341-10 and AISC 358-10	
Connections	AISC 13th (360-05): LRFD	
Cold Formed Steel	AISI S100-07: ASD	Yes (Iterative)
	CFS Walls: AISI S400-20 & AISI S240-20	
Wood	AF&PA NDS-05/08: ASD	
	Temperature: < 100F	
Concrete	ACI 318-11	
Masonry	ACI 530-08: ASD	
Aluminum	AA ADM1-05: ASD	Yes (Iterative)
	Structure Type: Building	
Stainless	AISC 14th (360-10): ASD	Yes (Iterative)

At the bottom of the dialog, there is a checkbox labeled 'Set as Default', and two buttons: 'Apply' and 'Cancel'.

The settings under the **Codes** tab control which design code is used for code checks for each material type.

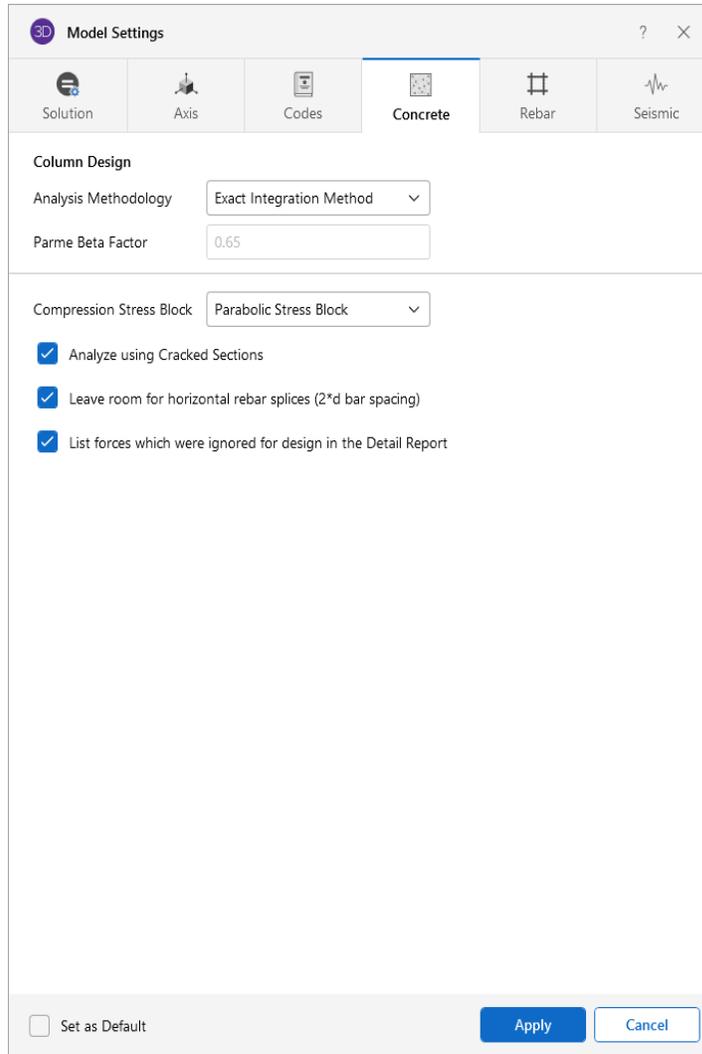
Codes tab

Option	Description
HR Steel	<p>HR Steel indicates which code is to be used for the design of hot rolled steel. For more information see Hot Rolled Steel Design.</p> <div data-bbox="553 604 1409 1297" style="border: 1px solid black; padding: 10px; background-color: #e6f2ff;"> <p> Note:</p> <ul style="list-style-type: none"> • If the AISC 360 code is selected (either ASD or LRFD), the Adjust Stiffness menu will appear. The options in this menu control the stiffness reduction provisions listed in the code. See Hot Rolled Steel - Design for more information on how the reduction is calculated and applied. • If the EN 1993-1-1:2005 (Eurocode) is selected, the National Annex menu will appear. If you select "None" the design will be performed per the generic Eurocode specifications. If you select a National Annex from the list then the provisions relating to that specific National Annex will also be considered in the design. Where the National Annex and the generic Eurocode differ, the National Annex provisions are used. See the Hot Rolled Steel - Design topic for more information on the differences in design. </div>
Connection	<p>Connection indicates which code is to be used for connection design. This entry is only considered if you are also using RISACONNECTION. See the RISACONNECTION Integration topic for more information.</p>
CF Steel	<p>CF Steel indicates which steel code is to be used for the design of cold formed steel. For more information see Cold Formed Steel Design.</p>
Wood	<p>Wood indicates which wood code is to be used for the design of wood, including wood walls and structural composite lumber. For more information see Wood Design.</p>

Option	Description
	 Note: <ul style="list-style-type: none"> The temperature menu is used to calculate the temperature factor (C_t) for the NDS code design. This factor is used in the design of wood members for all editions of the AF&PA code. See NDS-2012, Section 2.3.3 for more information. This selection is ignored for the CSA O86 code design. Rather than applying a separate temperature factor, the CSA O86 code suggests that the designer apply the wet properties reduction factor when the member is subject to prolonged exposure to temperatures higher than 50°C (122°F). See Chapter 11 (Reference Information) of the Wood Design Manual (2010) for more information.
Concrete	<p>Concrete indicates which concrete code is to be used in the design of concrete members and walls. For more information see Concrete Design.</p>  Note: ACI 318-08 and ACI 318-11 behave identically in the program, as there were no relevant changes between these two editions of the code.
Masonry	<p>Masonry indicates which masonry code is to be used for the design of masonry walls. For more information see Masonry Design.</p>  Note: If the UBC 1997 ASD code is chosen, there is an option for whether the construction of the wall is subject to special inspection. This affects some values in design under that code.
Aluminum	<p>The Aluminum entry indicates which aluminum code is to be used in the design of aluminum. For more information see Aluminum Design.</p>

Option	Description
	<div data-bbox="553 365 1409 808" style="border: 1px solid black; padding: 5px;">  Note: <ul style="list-style-type: none"> • The Building and Bridge options refer to the Safety Factors the code requires to be used for Building or Bridge type structures. • If the 2010 code is selected (either ASD or LRFD), the Adjust Stiffness menu will appear. The options in this menu control the stiffness reduction provisions listed in Chapter C (Design for Stability) of the code. See Aluminum- Design for more information on how the reduction is calculated and applied. </div>
LL Reduction	<p data-bbox="553 821 1386 951">LL Reduction check boxes for columns and beams indicate whether LL Reduction will be considered. These check boxes are only available for models that are integrated with RISAFloor. And, they only affect elements that originated in the RISAFloor model.</p> <div data-bbox="553 984 1409 1092" style="border: 1px solid black; padding: 5px;">  Note: Wall results are not currently affected by LL Reduction. </div>

Concrete



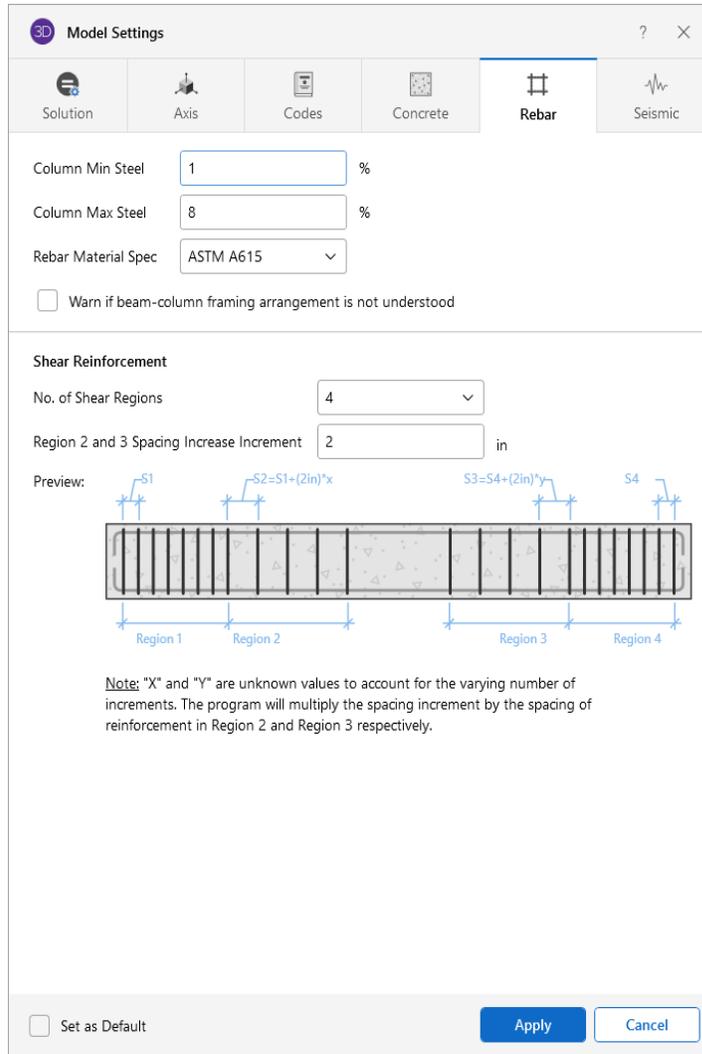
The entries under the **Concrete** tab contain options related to the analysis and design of concrete members.

Concrete tab

Option	Description
Analysis Methodology	Analysis Methodology controls which method is used to determine the biaxial column capacity. The options are the PCA Load Contour Method and the Exact Integration Method .
Parme Beta Factor	Parme Beta Factor is used to approximate the column's 3D interaction surface when using the PCA Load Contour

Option	Description
	Method. See Biaxial Bending of Columns for more information.
Compressions Stress Block	Compression Stress Block lets you choose what type of stress block to consider in your analysis. The options are the constant Rectangular Stress Block and the Parabolic Stress Block . See Parabolic vs. Rectangular Stress Blocks for more information.
Analyze using Cracked Sections	Check the 'Analyze using Cracked Sections' box if you want to modify the member and wall stiffnesses by the lcr Factor as described in both the Concrete - Design and Wall Panels topics.
Leave room for horizontal rebar splices (2*d bar spacing)	Check the 'Leave room for horizontal rebar splices (2*d bar spacing)' box, if you want to default to a two bar diameter or one inch clear spacing, whichever is greater, to allow for lap splices and continue to maintain adequate spacing between parallel bars. Otherwise a minimum spacing of one bar diameter between parallel bars will be allowed.
List forces which were ignored for design in the Detail Report	Check the 'List forces which were ignored for design the Detail Report' box to see force warnings in the detail report.

Rebar



Rebar tab

Option	Description
<p>Minimum % Steel for Columns</p>	<p>Minimum % Steel for Columns lets you choose the minimum percentage of reinforcement steel to be used in a concrete column section. The percentage entered is multiplied times the gross area of the column section to determine the minimum amount of reinforcement required in each column.</p> <p>It should be noted that the minimum percentage allowed by ACI 318-14 Section 10.6.1.1 (ACI 318-11 Section 10.9.1) is 1%.</p>

Option	Description
Maximum % Steel for Columns	<p>Maximum % Steel for Columns lets you choose the maximum percentage of reinforcement steel to be used in a concrete column section. The percentage entered is multiplied times the gross area of the column section to determine the maximum amount of reinforcement allowed in each column. It should be noted that the maximum percentage allowed by ACI 318-14 Section 10.6.1.1 (ACI 318-11 Section 10.9.1)1 is 8%.</p> <div data-bbox="683 663 1409 884" style="border: 1px solid black; padding: 5px;">  Note: Concrete cover for beams and columns is specified under Design Rules under the Concrete Rebar tab. Concrete cover for beams and columns is specified under Design Rules under the Concrete Rebar tab. </div>
Rebar Material Spec	<p>Rebar Material Spec lets you choose from the standard ASTM A615 (imperial), ASTM A615M ("hard" metric, i.e. #8M is an 8mm bar), BS 4449 (British), prENV 10080 (Euro), CSA G30.18 (Canadian), IS 1786 (Indian), and AS/NZS 4671:2001 (Australian/New Zealand) reinforcement standards.</p> <div data-bbox="683 1125 1409 1314" style="border: 1px solid black; padding: 5px;">  Note: The rebar specified in this section is for concrete design only. Masonry walls only support ASTM A615 rebar specification by default. See the Masonry Wall Design help topic. </div>
Warn if beam-column framing arrangement is not understood	<p>Check the 'Warn if beam-column framing arrangement is not understood' box to see bad framing warnings in the Warning Log.</p>
Shear Reinforcement Options	<p>Shear Reinforcement Options lets you control the Number of Shear Regions that is used when detailing a beam or column span. You can also specify a Region 2 & 3 Spacing Increase Increment that you'd like the program to use when increasing or reducing the spacing of the shear ties.</p>

Seismic

The options on the **Seismic** tab list settings that are specifically related to calculation of code prescribed Seismic Loads. This information can be used by RISA-3D to automatically generate the Lateral Loading on your structure. Depending on which Seismic Code you select will depend on the input options that you have.

3D Model Settings
?
×

Solution
Axis
Codes
Concrete
Rebar
Seismic

RISA-3D Seismic Load Options

Code: ▼

Risk Category: ▼

Drift Category: ▼

Base Elevation: ft

Story number above critical section: - +

Include the weight of the structure in base shear calculations

Site Parameters

S_1 : g SD_s : g

SD_1 : g T_L : sec

Structure Characteristics

	X Direction	Z Direction	
T	<input type="text"/>	<input type="text"/>	sec
C_t	<input type="text" value="0.02"/>	<input type="text" value="0.02"/>	
C_t Exp.	<input type="text" value="0.75"/>	<input type="text" value="0.75"/>	
R	<input type="text" value="6.5"/>	<input type="text" value="6.5"/>	
Ω_0	<input type="text" value="3"/>	<input type="text" value="3"/>	
C_d	<input type="text" value="4"/>	<input type="text" value="4"/>	
ρ	<input type="text" value="1"/>	<input type="text" value="1"/>	

Set as Default
Apply
Cancel

Most of the input fields on this tab are discussed in the [Seismic Load](#) topic. Below are a few inputs which are specific to this tab. See the section Drift Calculations to see how Rho, Cd, Drift Category and Risk Category affect the calculation of [story drift results](#).

Seismic tab

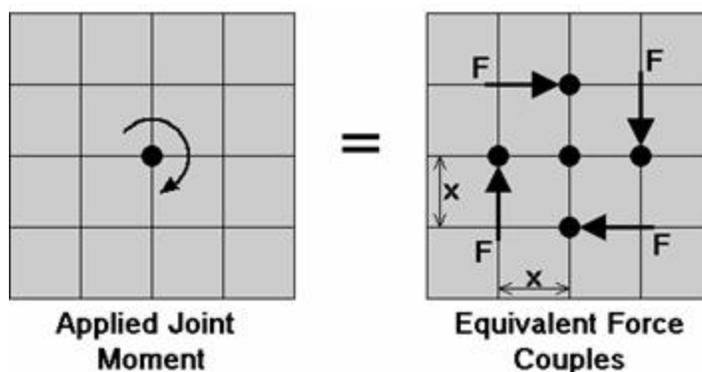
Option	Description
Story No. Above Critical Section	Story No. Above Critical Section establishes the critical section location for shear force amplification consideration in special concrete walls as per ACI 318-19. More information regarding shear force amplification can be found in the Concrete Wall - Seismic Design.
Ω (Overstrength Factor)	(Ω) is the “Overstrength Factor”. This can be included in the seismic load combinations generated by the LC Generator .
ρ	“Redundancy Factor” (ρ) is based on the extent of redundancy in

Option	Description
(Redundancy Factor)	your structure. This can be included in the seismic load combinations generated by the LC Generator .

Modeling Tips

Applying In-Plane Moment to Plates

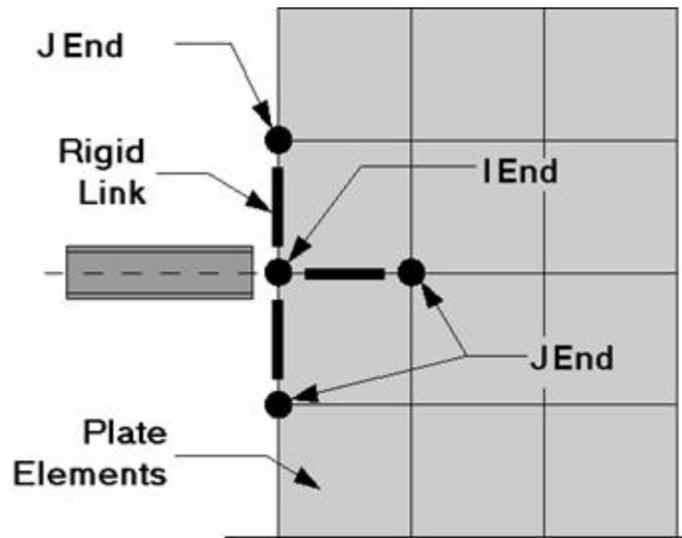
Occasionally you may need to model an applied in-plane moment at a node connected to plate elements. The plate/shell element cannot directly model in-plane rotations. One way around this is to model the in-plane moment as a force couple of in-plane forces. You would replace the applied in-plane node moment at 1 node with 2 or 4 in-plane forces at 2 or 4 nodes, 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 node and will be used to transfer the moment, then you will want to look at the topic **Modeling a Beam Fixed to a Shear Wall** below.

Modeling a Beam Fixed to a Shear Wall

Occasionally you may need to model the situation where you have a beam element that is fixed into a shear wall. A situation where this may occur would be a concrete beam that was cast integrally with the shear wall or a steel beam that was cast into the shear wall. The beam cannot just be attached to the node at the wall because the plate/shell element does not model in-plane rotational stiffness. A fairly simple work around is to use rigid links to transfer the bending moment from the node at the wall as shear force to the surrounding nodes in the wall. See [Rigid Links](#) in the Modeling Tips section to learn how to create rigid links. This modeling method provides a more accurate analysis than trying to use a plate/shell element with a "drilling degree of freedom" which attempts to directly model the in-plane rotation. See the figure below:



The only trick to this method is getting the proper member end releases for the rigid links. We want to transfer shear forces from the wall 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 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**, **Mx**, **My**, and **Mz** degrees of freedom released. Only the **y** and **z** degrees of freedom (local axes shears) should be connected from the J-ends to the interior wall nodes. This release configuration will allow the shears to be transferred into the wall, but the wall stiffness will not be adversely affected by the presence of the rigid links.

Modeling a Cable

While there is not a true “cable element”, there is a tension only element. A true cable element will include the effects of axial pre-stress as well as large deflection theory, such that the flexural stiffness of the cable will be a function of the axial force in the cable. In other words, for a true cable element the axial force will be applied to the deflected shape of the cable instead of being applied to the initial (undeflected) shape. If you try to model a cable element by just using members with very weak I_{yy} and I_{zz} properties and then applying a transverse load, you will not get cable action. What will happen is that the beam elements will deflect enormously with NO increase in axial force. This is because the change in geometry due to the transverse loading will occur after all the loads are applied, so none of the load will be converted into an axial force.

Guyed Structure (“Straight” Cables)

You can easily model cables that are straight and effectively experience only axial loading. If the cable is not straight or experiences force other than axial force then see the next section.

When modeling guyed structures you can model the cables with a weightless material so that the transverse cable member deflections are not reported. If you do this you should place all of the cable self-weight elsewhere on the structure as a point load. If you do not do this then the cable deflections (other than the axial deflection) will be reported as very large since it is cable action that keeps a

guyed cable straight. If you are interested in the deflection of the cable the calculation is a function of the length and the force and you would have to calculate this by hand.

The section set for the cable; should be modeled as a tension only member so that the cable is not allowed to take compression. See [T/C Members](#) for more on this.

To prestress the cable you can apply a thermal load to create the pre-tension of the cable. See [Prestressing with Thermal Loads](#) to learn how to do this.

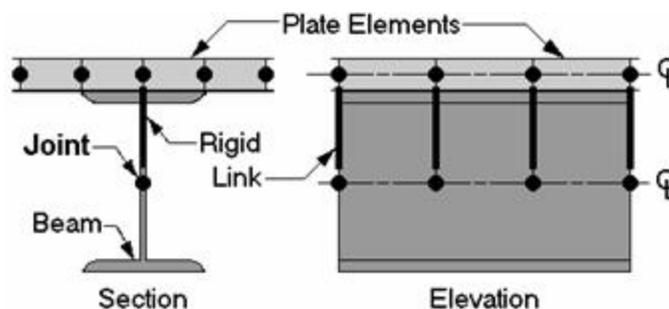
Modeling Composite Behavior

Occasionally you may want to model a structure with composite behavior. A practical situation where this arises is with composite concrete floor slabs, which have concrete slabs over steel or concrete beams. Another common case where composite behavior may be considered is where you have a steel tank with stiffeners. The stiffeners might be single angles or WT shapes.

The most accurate way to model this behavior for composite floor systems is to use a program (such as RISAFloor) that was created explicitly for this purpose. If that is not an option you may want to use an arbitrary member with an effective moment of inertia, I_{eff} , calculated according to the AISC or Canadian code provisions. This method has the advantage of being able to account for the effects of partial composite behavior, while the methods described below assume perfect connectivity between the steel beam and the concrete slab.

If you have already modeled the concrete deck with plate elements, then the situation can be quickly modeled for horizontal beams by using one beam modeled as a physical member and also using Analysis Offsets. The beam would simply be drawn along the plates/nodes for the composite section. This method has the advantage of being very fast to model but it also neglects the depth of the concrete in computing the total vertical offset between the centerline of the beam and the centerline of the slab. Note that you have to have an appropriate number of nodes along the span of the beam to model the shear transfer between the slab and the beam. A more refined (and complicated) method using rigid links is described next.

An example of a plate/beam model with composite action included using rigid links is shown here:



Note that beams and plates are each modeled at their respective centerlines. It is this offset of the beam and plate centerlines that causes the composite behavior. The distance between the centerlines is typically half the depth of the beam plus half the thickness of the plate elements. If the beam is an unsymmetrical shape, like a WT about the z-z axis or a single angle, then you would use the distance from the flange face to the neutral axis.

As shown above, a rigid link is used to connect each set of nodes between the beam and the plates. This rigid link is fixed to each node and therefore has no member end releases. See [Rigid Links](#) in the Modeling Tips section to learn how to create rigid links.

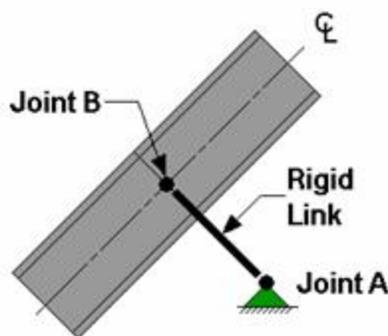
Graphical editing offers the fastest way to model composite action. It is usually best to model the plates with an appropriately fine mesh first. Then you would copy the plate nodes that are directly over the beam down to the centerline elevation of the beams. Next you would draw your beam to the two outer nodes at the beam centerline elevation.

There are a number of ways to build your rigid link member between all the corresponding plate and beam nodes. A good way to connect all the links is to generate grid members ([Grid Member Generation](#)). You can also draw the first one and then copy it along the length of the member.

Now perform a model merge. If you have used finite members rather than Physical Members the merge Model Merge will break up finite element beams at all the intermediate locations. See [Model Merge](#) for more information. If you're using physical members, performing the model merge will clean up duplicate nodes and members but not break up the beam member since the links are automatically attached to Physical Members as long as their node lie on the member.

Modeling Inclined Supports

You can 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:

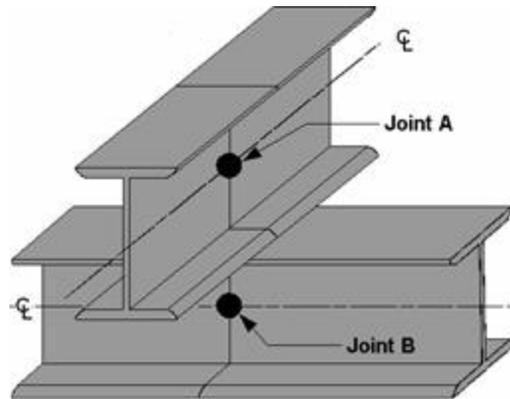


The rigid link should be “short”, say no more than 0.1 ft. The member end releases for the rigid link at node “B” are used to control which degrees of freedom are pinned or fixed in the inclined directions. This works because the member end releases are in the local member axes. See [Rigid Links](#) in the Modeling Tips section to learn how to create rigid links.

The section forces in the rigid link **are** the inclined reactions. Note that you need to make sure the rigid link is connected to the members/plates at the correct inclined angle. You can control the incline of the angle using the coordinates of nodes “A” and “B”. You can also rotate the rigid link to the proper angle.

Modeling One Member Over Another

Occasionally you may need to model the situation where one member crosses over another member. A common situation where this occurs is in the design of framing for crane rails, where the crane rail sits on top of, or is hanging beneath, the supporting beam. See the figure below:



The two beams are each modeled at their correct centerline elevations. Both the top and bottom members need to have a 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. The method to model this is to connect node A to node B with a rigid link. The member end releases at the A or B end can be used to control which degrees of freedom get transferred between the beams. For example, if Beam A is free to pivot over Beam B then you would apply a Pin end release to the top of the rigid link, and a Fixed end release to the bottom of the rigid link. Don't pin both ends of the rigid link though, because then there can be no shear transfer between the beams. See [Rigid Links](#) in the Modeling Tips section to learn how to create rigid links.

Reactions at 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. See Rigid Links below to learn how to create rigid links.

Rigid Links

Rigid links are used to rigidly transfer the forces from one point to another and to also account for any secondary moments that may occur due to moving the force. This is in contrast to using the tether feature for nodes where the forces are shared by 2 or more node degrees of freedom (DOF), but any secondary moments are lost when tethering the nodes. Tethered nodes actually share common DOF and so do not account for the distances between them. Rigid links do not have any practical internal deformation, i.e. there is no differential movement between the I-node and the J-node. Rigid links may be used to model situations such as composite behavior or beams fixed to walls modeled

with plate elements. They are also useful for getting information such as reactions at inclined supports or reactions at nodes with enforced displacements.

Make a Rigid Link

To make a Rigid Link:

1. On the **General** tab of the **Materials** spreadsheet:
 - a. Create a material **Label** called "LINK".
 - b. Enter "1e6 (ksi)" for the value of **E**.
 - c. Blank out the value for **G** by going to that column and pressing the space bar.
 - d. Double check that the **Density** is set to zero.
 - e. Leave all the other values as their defaults.
2. On the **General** tab of the **Section Sets** spreadsheet, create a section set **Label** called "RIGID" with the "LINK" **Material**.
3. Move the cursor to the **Shape** column and hit your space bar to erase the information in this field.
4. Set the **A**, and **Iy**, **Iz** and **J** values to "1e6 (based on units of inches)".
5. To make a rigid link, on the **Members** spreadsheet create a member that references the RIGID section set created above.

You can control which DOF are transferred through the link by using the member end releases.



Note:

- The density is set to zero in case the self-weight is used in a loading condition. If the density is left as the default then any gravity loading would cause the rigid link to add a very large load into your model due to its large area.
- To keep the model merge from deleting your rigid links, be careful not to create links whose lengths are less than your merge tolerance.

The weight density should be set to zero in case self-weight is used as a loading condition. If the material used is not weightless, then any gravity loading would cause the rigid link to add a very large load into your model. (Gravity load is applied as a distributed load with a magnitude equal to the member area times the weight density).

For models with very stiff elements, like concrete shear walls, the rigid link may not be rigid in comparison. If you see that the rigid link is deforming, then you may have to increase the stiffness of the link. The easiest way to do this is to increase the A, Iy, Iz, and J values for the RIGID section set. Make sure that the combination of **E*I** or **E*A** does not exceed 1e17 because 1e20 and 8.33e18 are the internal stiffnesses of the translational and rotational

Reaction boundary conditions. If you make a member too stiff, you may get ghost reactions, which tend to pull load out of the model. (The total reactions will no longer add up to the applied loads.)

Solving Large Models

Large models are those where the stiffness matrix size greatly exceeds the amount of available free RAM on your computer. Solving large models can take a long time, so it is useful to have an understanding of what steps can help speed up the solution. The time it takes to solve a model is dependent on several things; these include the bandwidth of the stiffness matrix, the number of terms that need to be stored for the stiffness matrix, and the amount of RAM in your computer.

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

If you are getting a long solve time and therefore a large stiffness matrix and you don't think that you have any modeling errors, you can try a few things to reduce the bandwidth. The first thing you can try is to sort your 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. The order of sorting depends on the model, so you might want to try a couple of different combinations and check the model each time. Sometimes, sorting the 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 minimization starts on the small model and then jumps to the big model. Instead, you will probably want to split these into 2 separate files.

The amount of "address space" available to solve your model is based on several things: the amount of RAM in your computer, the amount of free hard disk space, the operating system, the Virtual memory settings in the Windows Control Panel, and the internal limitations of your operating system.

If you get an error that states "You have run out of memory, try increasing your virtual memory..." you will want to note the amount of memory that was requested at the time versus the amount that was available. This amount should be displayed along with the error message. This amount will give a starting point from which you can increase the available address space. You may need to increase the amount of Virtual Memory so that you have enough address space to run the model and your other applications. (You typically do this by double clicking on "My Computer", then "Control Panel", then "System". Within the System options, you would click on the "Performance" tab and then you click on the "Virtual Memory" button.) Make sure that you are specifying more Virtual Memory than is needed to solve your model.

There is an internal limitation to the amount of memory that Windows will allocate to the RISA program. Within a 32 bit addressing space, Windows has a basic limit of 4 Gigabytes. Of that, they reserve 2 Gigs for the operating system. Therefore, RISA can only use a maximum of 2 Gigabytes with large models (where the stiffness matrix alone is well in excess of 1.0 Gigs), your only option may be to re-model your structure using fewer degrees of freedom.

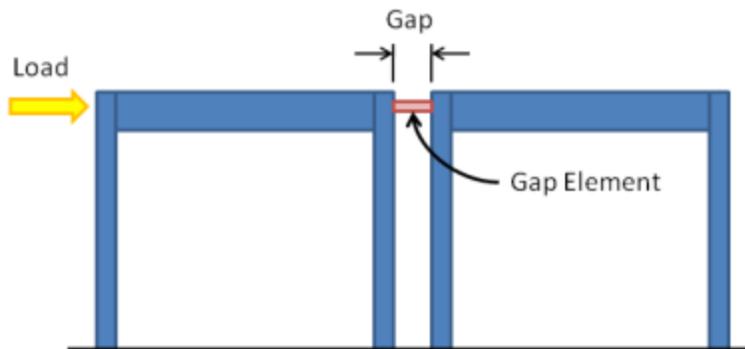
Ways to Minimize the Memory Use

Keep in mind that many of these solutions may result in a decreased accuracy of your solution

- Set the **Number of Sections** and **INTERNAL Sections** to a minimal amount. This can be done via the [Model Settings - Solution](#) tab.
- Set the **Mesh Size** for wall panels (if they exist in your model) to a larger value. This can be done via the [Model Settings - Solution](#) tab.
- Solve fewer load combinations. It is almost always possible to reduce the total number of combinations by inspection. Engineering judgment can help determine which load combinations will not control for your structure.
- Check to see that you are not causing an overly tight mesh for wall panels (if they exist in your model) by your modeling practices. This can cause a large amount of internal plates to be created and tax the solution. See the [Wall Panels - Tips For Ensuring a Healthy Mesh](#) section for more information.

Modeling a "Gap" (Expansion Joint) Between Structures

A gap element is a member that mimics the behavior of a gap or expansion joint between adjacent structures.



While RISA does not offer the capability to directly create a gap element, one may be indirectly created using the properties of the member and an applied thermal load. The concept is to place a 'shrunk' member between adjacent structures. The shrinkage of the member is achieved with a negative thermal load, in the form of a member distributed load.

The amount of shrinkage should be equal to the width of the gap, such that the structures act independently until they move close enough to each other to 'touch' and thereby transmit loads to each other. To calculate the thermal load required for a gap use the following formula:

$$\Delta T = \frac{-1 * (Gap)}{L * (\alpha)}$$

Where:

ΔT = Applied thermal load

Gap = Distance between two structures

L = Length of gap element

α = Coefficient of thermal expansion

In order to prevent the gap element from 'pulling' its connected structures towards it due to shrinkage it must be defined as a 'compression only' member under the advanced options tab. It is also advisable to define the gap element as a rigid material such that the amount of load it transfers once the gap is closed is not affected by elastic shortening.

Lastly, in cases where the applied temperature would need to be of an extraordinary magnitude, it might be useful to increase the coefficient of thermal expansion of the material such that a smaller temperature load would achieve the same shrinkage.

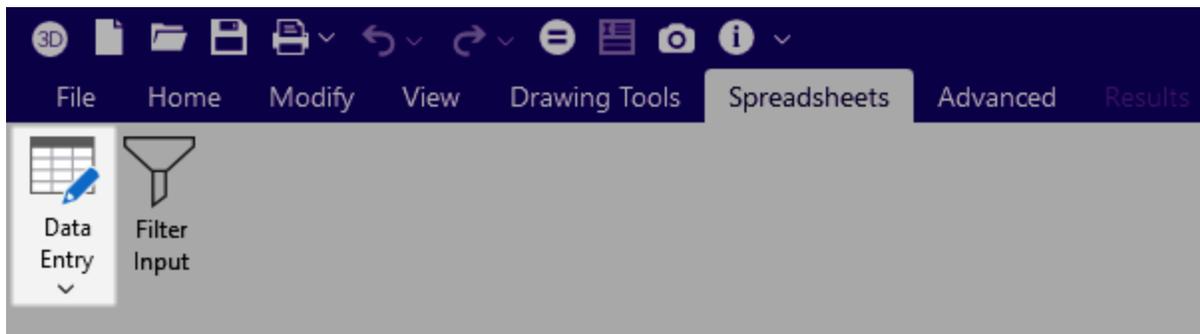
Nodes

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 nodal 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.



Note: The terms "node" and "joint" are interchangeable and are both used in this manual and in the program.

Nodes can be input manually, or they can be created automatically as you draw new members and plates on the drawing grid. Once defined, the nodes can be edited in three ways: graphically, in the **Node Properties Panel**, or in the **Node Coordinates** spreadsheet.



Define Nodes

To define Nodes:

1. Select the **Node Coordinates** spreadsheet from the **Spreadsheets** tab ▶ **Data Entry** button or from the **Data Entry** portion of the **Explorer** panel.
2. 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 nodes.
- To modify one node you may click that node in the 3D View to view and edit its properties in the Properties panel.

Relabel Nodes

To relabel Nodes:

1. After sorting the **Node Coordinates** spreadsheet into the desired order, go to the **Modify** ribbon.
2. Click the **Relabel All** or **Relabel Selected** icon.
3. Choose **Nodes**.
4. Alternatively, you can right-click in the **Node Coordinates** spreadsheet and select either **Relabel All Nodes** or **Relabel Selected Nodes**.

Alternatively, you can right-click in the **Node Coordinates** spreadsheet and select either **Relabel All Nodes** or **Relabel Selected Nodes**.

Round-Off Node Coordinates

To round-off Node Coordinates

1. go to the **Modify** ribbon.
2. Click the **Round Off Coordinates** icon.

Alternatively, you can right-click in the **Node Coordinates** spreadsheet and select **Round Off Node Coordinates**.



Note: This utility rounds off all the node coordinates to 1, 2, or 3 decimal places (user selected). It's sometimes useful for models that have been generated or created via DXF import and have node coordinates with a large number of decimal places that aren't significant. Rounding off those coordinates makes the data more consistent and can help avoid problems such as non-coplanar plate/wall nodes. Also, the default member orientations can change based on whether the member is exactly parallel to a global axis and slight unintended coordinate differences can have an impact here as well.

Node Coordinates Spreadsheet

The **Node Coordinates** spreadsheet records the node coordinates and ambient temperatures for the nodes and may be accessed from the **Spreadsheets** tab ▶ **Data Entry** icon ▶ **Node Coordinates**.

Node Coordinates						
	Label	X [ft]	Y [ft]	Z [ft]	Temp [F]	Detach From Diaphragm
1	N1	54	0	0		<input type="checkbox"/>
2	N2	54	0	18		<input type="checkbox"/>
3	N3	54	12	18		<input type="checkbox"/>
4	N4	54	12	0		<input type="checkbox"/>
5	N5	36	0	36		<input type="checkbox"/>
6	N6	36	12	36		<input type="checkbox"/>
7	N7	0	0	0		<input type="checkbox"/>
8	N8	18	0	0		<input type="checkbox"/>
9	N9	18	12	0		<input type="checkbox"/>
10	N10	0	12	0		<input type="checkbox"/>
11	N11	36	0	0		<input type="checkbox"/>

Node Coordinates - columns

Column	Description
Label	Label is used to assign a unique name (label) to every node. You can then refer to the node by its label. Each label has to be unique, so if you try to enter the same label more than once you receive an error message. As you create new lines, the program automatically creates a new, unique label for each node.
X, Y and Z columns	These three columns contain the coordinates of the node in each of the global directions. These represent the relative offsets of the nodes from the origin of the coordinate system (0, 0, 0). The appropriate units are listed at the top of each column.
Temp	Temp (Temperature) 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. See Loads - Thermal Loads for more information.
Detach From Diaphragm	Detach From Diaphragm is used to detach nodes from a diaphragm. See Diaphragms - Modeling Tips to learn more about this.
Round Off Coordinates	Another option is 'Round Off Coordinates', which when selected, rounds off all the node coordinates to 1, 2, or 3 decimal places. It's useful for models that have been created using generation functions or DXF files where a high number of decimal places is present for some of the coordinates.

Several options on the **Modify** ribbon help you work with nodes. Selecting **Relabel All** or **Relabel Selected** lets you define a prefix to be used for the node labels before creating a new label for each node by using this prefix with a sequential number. For example, if you were to enter a prefix of "FLR", the first node would get label FLR1, the second one would get FLR2, etc.

Node Information

Just as with the members and plates you can click any node to view it's properties. All of the same information that is stored in the **Nodes** spreadsheet is displayed for the node you choose, and can be edited. This is a quick way to view and change node properties.

Properties	
Node ^	
Label	<input type="text" value="N419"/>
Coordinates ^	
X, ft	<input type="text" value="0"/>
Y, ft	<input type="text" value="24"/>
Z, ft	<input type="text" value="36"/>
Temp, F	<input type="text"/>
Detach From Diaphragm	<input type="checkbox"/>
Boundary Conditions ^	
X, k/in	<input type="text"/> ...
Y, k/in	<input type="text"/> ...
Z, k/in	<input type="text"/> ...
X Rot, k-ft/rad	<input type="text"/> ...
Y Rot, k-ft/rad	<input type="text"/> ...
Z Rot, k-ft/rad	<input type="text"/> ...
Dynamic Spring Damping ^	
X, k sec/ft	<input type="text"/>
Y, k sec/ft	<input type="text"/>
Z, k sec/ft	<input type="text"/>
X Rot, k-ft sec/rad	<input type="text"/>
Y Rot, k-ft sec/rad	<input type="text"/>
Z Rot, k-ft sec/rad	<input type="text"/>

Properties Options

Option	Description
Label	Label lets you view and edit the joint's label.
Coordinates	Coordinates lets you view and edit the node coordinates.
Temperature	Temperature lets you set the ambient temperature at that node for

Option	Description
Boundary Conditions	<p>Thermal Loading.</p> <p>Boundary Conditions lets you view and edit the node boundary conditions. The edit boxes to the right of the boundary condition list boxes are where you would enter additional data for a boundary condition such as a spring stiffness, a Story number, or a primary node label for a Tethered node.</p>

 **Note:** It's more efficient to use the Properties panel to change the properties for many nodes at once. Simply select multiple nodes in the 3D View and choose Nodes from the dropdown list in the Properties panel.

Nodes - Results

When the model is solved, there are several groups of results specifically for the nodes. Story Drift calculation results are discussed in [Drift Results](#), the others are discussed here.

Node Deflections Results

Access the **Node Deflections Spreadsheet** by selecting it from the **Results Menu**.

	Node Label		X [in]	LC	Y [in]	LC	Z [in]	LC	X Rotation [rad]	LC	Y Rotation [rad]	LC	Z Rotation [rad]	LC
4		min	0	19	0	11	0	15	0	1	9.144e-7	19	-1.889e-3	2
5	N3	max	0.002	10	-0.003	19	0.01	11	1.48e-2	2	4.499e-5	11	6.022e-3	2
6		min	-0.009	14	-0.009	2	-0.007	15	3.987e-3	19	-5.005e-5	15	6.287e-4	19
7	N4	max	0.009	13	0	17	0.01	11	5.574e-6	11	4.499e-5	11	7.294e-5	17
8		min	-0.016	17	-0.004	13	-0.007	15	-4.3e-5	15	-5.005e-5	15	-2.093e-5	13
9	N5	max	0	12	0	12	0	5	1.805e-4	18	9.048e-6	5	0	21
10		min	0	16	0	16	0	1	-1.64e-3	7	1.523e-6	1	0	1
11	N6	max	0.007	12	-0.012	13	0.019	11	-1.519e-3	19	4.499e-5	11	2.165e-3	2
12		min	-0.015	16	-0.027	2	-0.018	15	-5.392e-3	2	-5.005e-5	15	1.286e-3	19
13	N7	max	0	4	0	19	0	5	-4.55e-4	11	4.499e-5	11	1.109e-3	2
14		min	0	2	0	1	0	2	-1.208e-3	15	-5.005e-5	15	4.609e-4	19
15	N8	max	0	2	0	19	0	5	-3.621e-4	5	4.499e-5	11	3.638e-5	17
16		min	0	8	0	1	0	2	-1.09e-3	2	-5.005e-5	15	-2.166e-4	13

These are the deflections, or displacements, for every node in the structure. Each node has 6 values calculated, 1 for each of the 6 global degrees of freedom. Those being 3 translations and 3 rotations. The units for the displacements are shown at the top of each column. The rotations are shown in units of radians (1 radian = 57.3 degrees, approximately).

For enveloped results the maximum and minimum value for each displacement is listed. The load combination producing the maximum or minimum is also listed, in the "lc" column. To include a particular Load Combination in the envelope analysis, open the **Load Combinations Spreadsheet** and check the box in the **Solve** column.

The moving load results are enveloped and will display the Load Combinations with maximum and minimum values shown for each section location, for each active member. The governing load combination and step location is shown for each result value under the "LC" column. The first number is the load combination, the second is the step number: (load combination - step number). See [Moving Loads](#) to learn more.

**Note:**

- See [Spreadsheet Operations](#) to learn how to use **Find**, **Sort**, and other options.
- See [Results View Settings - Reactions](#) to learn how to plot node results. You will NOT be able to plot the deflected shape for an envelope analysis. This is because the various maximum and minimum displacements probably correspond to different load combinations, so a deflected shape based on these values would be meaningless.

Node Reaction Results

Access the **Node Reactions** spreadsheet by selecting it from the **Results Menu**.

Envelope Node Reactions														
	Node Label		X [k]	LC	Y [k]	LC	Z [k]	LC	MX [k-ft]	LC	MY [k-ft]	LC	MZ [k-ft]	LC
1	N22	max	0.018	7	48.8	2	-0.412	7	0	21	0	21	0	21
2		min	-0.059	2	30.782	19	-0.823	2	0	1	0	1	0	1
3	N35	max	-0.905	19	272.341	2	0.976	15	0	21	0	21	0	21
4		min	-2.128	2	176.247	19	-1.712	11	0	1	0	1	0	1
5	N19	max	-0.358	7	89.184	2	0.342	2	0	21	0	21	0	21
6		min	-0.44	18	35.086	19	0.112	5	0	1	0	1	0	1
7	N45	max	0	1	1.101	5	0	11	0	21	0	21	0	21
8		min	-0.001	2	-0.355	18	0	2	0	1	0	1	0	1
9	N13	max	0.98	2	28.352	2	-0.463	7	0	21	0	21	0	21
10		min	0.553	5	18.564	19	-0.804	2	0	1	0	1	0	1
11	N508	max	0.001	2	1.489	5	0	2	0	21	0	21	0	21
12		min	-0.003	3	-0.049	18	0	7	0	1	0	1	0	1
13	NR	max	-0.071	8	171.745	1	0.707	2	0	21	0	21	0	21

These are the reactive forces **applied to the structure** at its points of support. A positive reaction is applied in the direction of the global axis and a negative reaction is applied in the direction opposite the global axis. A positive moment is given according to the right hand rule assuming the thumb is pointing in the positive global axis direction. Assuming a reaction has been calculated at all points of support, the total of the reactive forces in each direction should equal the total applied force in each direction. The units for the reactions are listed at the top of each column, and a total reaction for each direction is summed at the bottom of each translation column.

Wall Panel Reactions represent the total reactions for the wall about the bottom center of the wall. This location does not change even if the center of the wall is within an opening, or if the edge of the wall is restrained as well as the base.

The last line provides the center of gravity (COG) for the applied loads. This "COG" is based on the load components acting in the VERTICAL direction. If there are no vertical loads in the combination a "COG" will not be calculated.

For enveloped results the maximum and minimum reaction value is listed. The load combination producing the maximum or minimum is also listed, in the "**LC**" column.

The moving results loads are enveloped and will display the Load Combinations with maximum and minimum values shown for each section location, for each active member. The governing load combination and step location is shown for each result value under the "**LC**" column. The first number is the load combination, the second is the step number: (load combination - step number). See [Moving Loads](#) to learn more.

Overstrength Tab

If your solution included a load combination that included overstrength load factors, a second tab will be included in the Node Reactions spreadsheet. This tab contains all the reactive forces per the overstrength load combinations.



Note:

- See [Spreadsheet Operations](#) to learn how to use **Find**, **Sort**, and other options.
- See [Results View Settings – Reactions](#) to learn how to plot node results.
- An 'NC' listing means "No Calculation". This occurs for boundary conditions defined with the 'Fixed' option and joints with an enforced displacement in that degree of freedom.

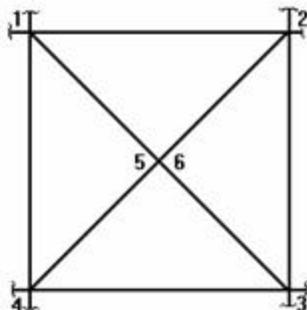
Nodes - Tethering Nodes

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



Note: Since the advent of [rigid links](#) there are not many circumstances where nodal tethering is the best solution for modeling.

An example of the use of nodal tethering is in X bracing, where the bracing members overlay and are pinned to each other. Modeling this connection makes the analysis far more complex than simply modeling the braces as separate pieces, and is **generally not recommended**. However, if it is desired to specifically account for interaction between the braces then this pinning will force the brace midpoints to deflect together for translations, but still leave them free to rotate independently.



In this diagram the braces are NOT modeled as physical members. Brace 1 to 3 is modeled with non physical members from nodes 1 to 5 and 5 to 3. Brace 2 to 4 is modeled with non physical members from nodes 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 tether node 6 to node 5 for the translation directions. To define a tethered node, use the **Boundary Conditions** spreadsheet. Enter the label of the node to be tethered as the "Node No.", and for the directions in which it is tethered, enter **TETHER nnn**, where **nnn** is the label of the primary node.

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

Node Boundary Conditions							
Primary	Advanced						
	Node Label	X [k/in]	Y [k/in]	Z [k/in]	X Rot [k-ft/rad]	Y Rot [k-ft/rad]	Z Rot [k-ft/rad]
1	N6	Tether N5	Tether N5	Tether N5			

Here we have "TETHER 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:** Tethered nodes should NOT be used to build rigid diaphragms. Tethering the translations will not give correct diaphragm behavior. If you need a rigid diaphragm, you should use the Rigid Diaphragm feature.

P-Delta - Analysis

When a model is loaded, it deflects. The deflections in the members of the model may induce secondary moments due to the fact that the ends of the member may no longer be co-linear in the deflected position. These secondary effects, for members (not plates), can be accurately approximated through the use of P-Delta analysis. This type of analysis is called "P-Delta" because the magnitude of the secondary moment is equal to "P", the axial force in the member, times "Delta", the distance one end of the member is offset from the other end.

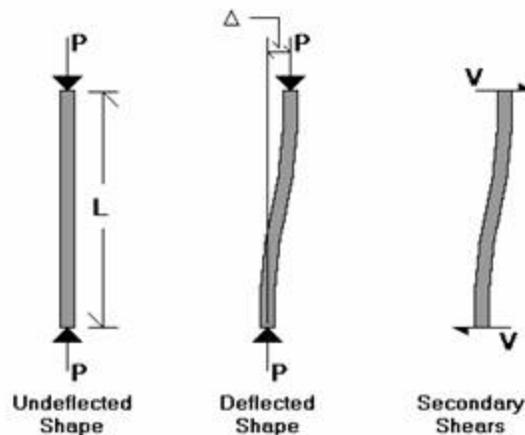
Since RISAFloor is designing entirely for gravity loads it does not need to account for the P-Delta effect. However, elements of the lateral force resisting system do need to consider this effect when being designed for lateral forces.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **P-Delta**.

P-Delta Procedure

Standard Nodal Shear Method

The actual modeling of these secondary moments is done through the calculation of secondary shears (shown as V in the diagram):



$$P * \Delta = V * L$$

$$\text{so, } V = P * \Delta / L$$

These shear forces are applied at the member ends. For a 3D model, this P-Delta calculation is done for the member's local y and local z directions.

The solution sequence is as follows:

1. Solve the model with original applied loads
2. Calculate the shears (V's) for every member in the model

3. Add these the shears (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.

Geometric Nonlinear Stiffness Method

Unlike the Standard Nodal Shear method, the Geometric Nonlinear Stiffness Method does not apply any nodal shear load to the member, instead, the stiffness matrix of the structure is directly modified to consider reduction of stiffness due to axial loads. This method is based on approach by Chen, W. F. and Lui, E. M., *Structural Stability: Theory and Implementation* (1987).

Similar as the Standard Nodal Shear Method, this method also requires an iterative procedure. In Step one, the linear static equilibrium equations are solved for the original model. In step two, the internal axial loads obtained from step one are used to form the geometric stiffness matrices as shown below, where P is axial load and L is the member length. The geometric stiffness matrix includes both the P-Delta and P-Small Delta effects. The geometric stiffness matrix is then assembled with the original stiffness matrix to obtain an updated stiffness matrix which considers P-Delta effect. The equilibrium equations are re-solved to obtain the new internal forces/new geometric stiffness matrices, and this procedure is repeated until it converges.

$$[K_g] = P \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 \\ & 6/5L & 1/10 & 0 & -6/5L & 1/10 \\ & & 2L/15 & 0 & -1/10 & -L/30 \\ & \text{sym.} & & 0 & 0 & 0 \\ & & & & 6/5L & -1/10 \\ & & & & & 2L/15 \end{bmatrix}$$

P-Delta Divergence

For both methods, if the P-Delta process is diverging dramatically, it will be stopped before numerical problems develop and an error will be displayed. If this error is displayed, the P-Delta displacements have reached a level where they are more than 1000 times greater than the maximum original displacements. If this happens with your model, the model may be unstable under the given loads, or there may be local instabilities present. See [P-Delta Troubleshooting](#) and [Testing Instabilities](#) to learn how to solve these problems.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **P-Delta**.

Perform a P-Delta Analysis

To Perform a P-Delta Analysis

1. Go to the Home ribbon.
2. Click on Settings to open the Model Settings window.
3. Select the **Solution** tab if not already selected.
4. Click the **Advanced** button to open the Set Advanced Options window.

5. Click on the **P-Delta Analysis** arrow.
6. Choose one of the two P-Delta Analysis methods (**Standard Nodal Shear Method** or **Geometric Nonlinear Stiffness Method**).
7. Click **OK**.
8. Open the **Load Combination** spreadsheet from the Explorer panel.
9. In the **P-Delta** column, place a "Y" to indicate the P-Delta combinations.



Note: You may indicate a compression only P-Delta analysis by placing a "C" in the **P-Delta** column.

P-Delta Limitations on Standard Nodal Shear Method

The P-Delta algorithm in the Standard Nodal Shear Method is based on end-joint displacements and will not automatically account for the effect of interior span forces on members, or member end rotations. As such the RISA implementation of P-Delta can be referred to as a P-Big Delta analysis.

In addition to the P-Big Delta effect, there can also be a P-Little Delta effect caused by end node rotation or transverse loading between node points. Please refer to the [P-Little Delta](#) topic for more information.



Note:

- P-Small Delta effect is considered in Geometric Nonlinear Stiffness Method.
- P-Delta effects are not calculated for plates.
- P-Delta effects are not currently include in Dynamic / Modal Analysis.

Compression Only P-Delta

The P-Delta effect can be thought of as decreasing the flexural stiffness of members in compression and increasing the flexural stiffness of members in tension. It is possible that if you have members with extremely large tensile forces, and intermediate nodes that are not connected to supports or other "stiff" members, the P-Delta algorithm could cause a node displacement to reverse direction, instead of converging to zero. This is an incorrect result. (A practical example where this could happen would be a truss chord with a large amount of tension that also has extra 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 0.5%. This means that the displacements from one solution to the next must vary by no more than ½ of 1 percent for the solution to be considered converged. You

may adjust this tolerance on the **Model Settings**. If you have a model that does converge but takes a lot of iterations, you may want to increase this tolerance so convergence is faster. Be careful! If you set this value too high, unstable models may falsely converge. It is not advisable to set this value above 2 or 3 percent.

P-Delta Troubleshooting

The first step in troubleshooting a P-Delta model that won't converge is to run the load combination without P-Delta analysis and make sure there are no instabilities. If it turns out that degrees of freedom are being locked, this indicates instabilities that you will want to fix.

In some ways, a P-Delta divergence may indicate an elastic buckling failure. Therefore, any P-Delta instabilities should be taken seriously and investigated thoroughly.

Local Instabilities

By far, the most common cause of P-Delta convergence problems is *local* instabilities. A local instability is when one part of the model is unstable causing the P-Delta analysis to diverge. To locate local instabilities, run the solution with P-Delta analysis turned OFF. Now plot the exaggerated deflected shape and animate it. Any local instabilities should be apparent. See [Testing Instabilities](#) for more information.

If you are trying to model P-Delta effects on a 2D frame, you will want to make sure that you restrain the out-of-plane degrees of freedom. See to learn how to do this.

Flexible Structures

In some cases, a model may be so flexible that it is not possible to run a P-Delta analysis. A situation where this might occur would be a wood frame where all the connections were modeled as pins, but the boundary conditions did not provide positive lateral support. In the real world, the connections will take some moment and the structure would be fine, but in the idealized model, there is zero moment resistance at each connection. The total lateral stiffness would be very small and this would make convergence of a P-Delta analysis unlikely.

Stiffness Reduction for the Direct Analysis Method

The Direct Analysis method requires a reduction in the axial and flexural stiffness of some members. This is done to account for the non-linear material effects caused by residual stresses. This [stiffness reduction](#) can be problematic during the early stages of design when your members may be smaller than they will be after basic drift and stress criteria are met. Therefore, RISA gives you the option (on the [Codes](#) tab of the Model Settings window) to turn off this stiffness reduction or set it to a constant value ($\tau = 1.0$).

P-Delta Amplification for AISC ASD

In order to ensure that using LRFD load factors does not result in a penalizing effect for P-Delta analysis, the AISC specification requires that the forces be multiplied by 1.6 during ASD analysis. Therefore, if AISC (360-05, 360-10, or 360-16) ASD is specified as the hot rolled code,

RISA automatically multiplies all load combinations by 1.6 during solution, then divides the force results by 1.6 prior to displaying the results.

P-Delta for ACI Concrete

See the [P-Little-Delta Topic](#).

Wall Panels

The P-Delta effect is handled the same way for Wall Panels. The main difference is that the shears for a wall panel are only generated at the top, story, or the diaphragm locations. The average deflection at the story height (or diaphragm location) is used to calculate the P-Delta shears for the top and bottom of the story. These shears are then distributed over the width of the wall at those locations.



Note: P-Delta effect is NOT considered for wall panels if Geometric Nonlinear Stiffness Method is selected.

Leaning Column Effect

Gravity columns and walls that are modeled in RISAFloor, but which are NOT included in the Lateral analysis will still have an effect on the P-Delta analysis in RISA-3D. This is because RISA-3D automatically includes this leaning column / leaning wall effect. This is done only for columns or walls that are contained within the a rigid diaphragm slab edge at that floor level. The program then uses the RISAFloor column / wall axial loads (without considering LL Reduction) along with the rigid diaphragm displacements (projected to each column / wall) to come up with equivalent leaning column shears for the diaphragm.

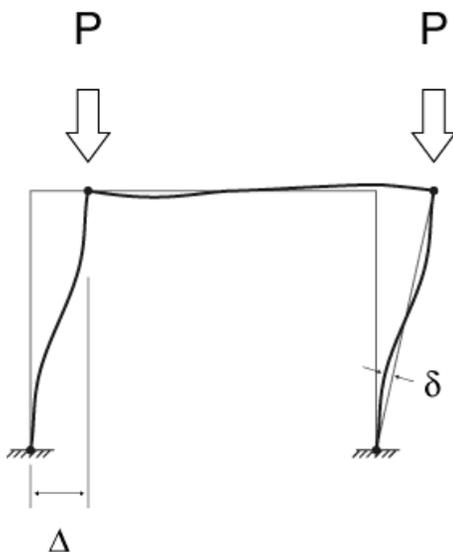


Note:

- The leaning column effect does not get accounted for columns or walls which do not connect to a diaphragm.
- The leaning column effect does not get accounted for columns or walls that connect to flexible or semi-rigid diaphragms.

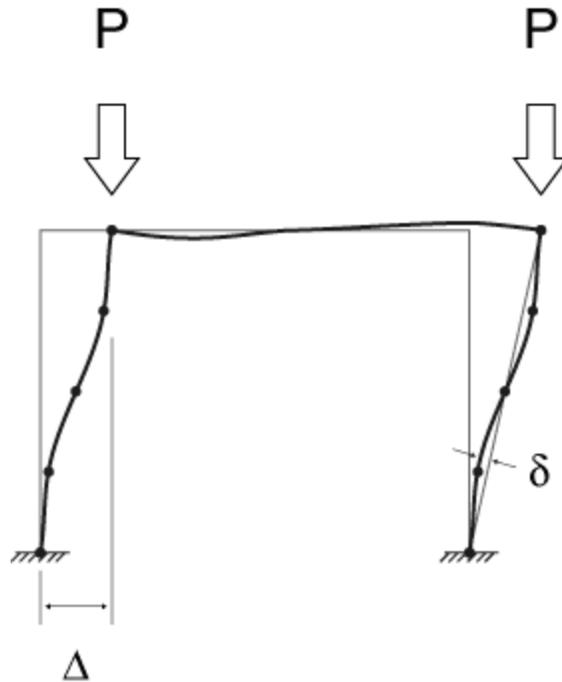
P-Little Delta Analysis

The P-Little Delta effect is essentially the destabilizing effect of individual member curvature on the axially loaded members. In practical applications, the effect of P-Little Delta will normally be significantly smaller than the P-Big Delta effect. The following figure demonstrates the displacements that cause the P-Delta and P-Little Delta effect in a typical moment frame.



P-Little Delta Procedure

The current implementation of P-Delta analysis does not directly account for the Little Delta effect. While this is true, the model can be easily adjusted to account for the effect as shown in the figure below.



Because RISA's P-Delta method is based entirely on nodal deflections, the introduction of nodes at the locations along the column where member displacement effects are at their maximum will adequately account for this effect.

There are a number of "benchmark" tests given in various publications to determine if a program is capable of properly considering this effect. Some of these benchmark problems (from the AISC Commentary) have been posted to our website with comparisons to theoretically correct solutions. These comparisons can be used as a basis for when the P-Little Delta effect is significant enough to consider in the analysis. They can also be used to determine how many intermediate nodes are required to adequately account for the effect at a given load level.



Note: P-Little Delta effects will have more of an impact as a member approaches its elastic euler buckling load. Special attention should be paid to cantilevered compression members, or members with significant weak axis bending moments.

AISC Direct Analysis Method

The AISC manual specifically requires the consideration of the P-Little Delta effect. However, the Direct Analysis Method (AISC 360-16, Chapter C) inherently acknowledges that P-Little Delta may not be important by stating that it may be neglected when, "the axial loads in all members whose flexural stiffnesses are considered to contribute to the lateral stability of the structure" are less than 15% of the Elastic / Euler buckling load of the member.

This may seem like it would be a low axial force, but it can actually be a very large force greater than the total axial capacity of the column. Situations where this code provision may justify ignoring the P-

Little Delta effect would include strong axis bending of a wide flange column with equal unbraced lengths in the strong and weak axes.

ACI Concrete Design

As of the 2014 edition, ACI 318 offers three options to account for slenderness and curvature in columns and walls (Section 6.2.6):

- Moment Magnification (6.6.4.5)
- Elastic Second Order Analysis (6.7)
- Nonlinear Second Order Analysis (6.8)

Both forms of second order analysis are more accurate than the moment magnification procedure, which is a hand calculation method intended to be used when computer analysis is not available. RISA does not have a true nonlinear solver though, so the program performs an Elastic Second Order Analysis using cracked section properties.

Columns in RISA-3D

In order to meet the requirements of Section 6.7 (thereby meeting the requirements of Section 6.2.6) you must turn on P-Delta in the [Load Combinations](#) spreadsheet, and possibly add intermediate joints along the length of the columns. The [Split Member](#) tool can be used to add joints along the physical columns.

When ACI 318-08 and newer is used the slenderness effects are *not* neglected per the provisions of Section 10.10.1 (ACI 318-14 Section 6.2.5)

Walls in RISA-3D

In order to meet the requirements of Section 6.7 (thereby meeting the requirements of Section 6.2.6) you must turn on P-Delta in the [Load Combinations](#) spreadsheet and include P-Delta for walls in the [Model Settings](#). The effects of P-Little-Delta are accounted for in walls using the Non-Sway Moment Magnification Procedure of ACI 318-11 Section 10.10.6 or ACI 318-14 Section 6.6.4.5.

Limitations

The provisions of ACI 318-11 Section 10.10.2.1 (ACI 318-14 Section 6.2.6) which state that P-Delta moments should not exceed 1.4 times non-P-Delta moments are not considered.

Plates/Shells

The plate/shell finite element allows you to easily model shear walls, diaphragms, shells, tanks and many other surface structures. We refer to the elements as plate elements, but they are actually plate/shell elements. Plate data may be viewed and edited in three ways: graphically, in the **Information** dialog or in the **Plates** spreadsheet.

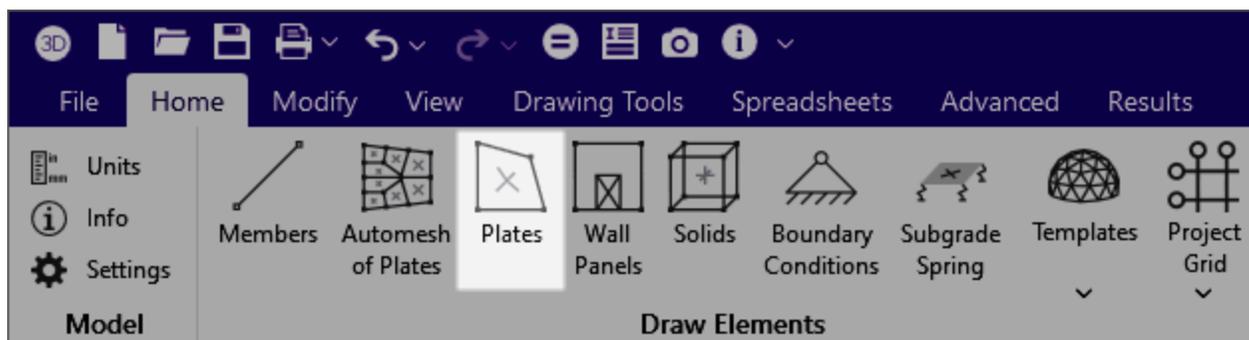
Draw Plates

There are several graphic-editing features that make the creation and modification of models quite easy. Use the Plates button in the Home tab and Modify menu to use these features in the model view. To create new members or plates, you can draw them using a drawing grid or draw "dot to dot" from existing nodes. Once you have created these items, you can use other graphic features to load the model and set boundary conditions.

Creating plate models requires more forethought than beam models. See [Plate Modeling Tips](#) and [Plate Modeling Examples](#) for tips on building plate models. To create new plates you can draw them using a drawing grid, a project grid, or draw "dot to dot" from existing nodes. You can set all of the plate properties up front or you can modify these properties after you draw them. Modifying properties is discussed in the next sections. See [Plate Spreadsheet](#) for information on plates and their properties.

The **Plates** tool lets you graphically draw plates in your model. Enter the appropriate plate parameters, and draw plates between existing nodes or on the drawing grid. You will also notice that the coordinates of the node or grid point that is closest to your cursor are displayed next to your cursor. The new plates will be shown on screen and will be recorded in the **Plates Spreadsheet**.

To actually draw a plate, you have two options. The fastest way is to use the **Create Plates by Clicking on Grid Areas** option, and then create plates by clicking on the grid areas formed by the intersecting grid lines. As you click on an area, a plate will automatically be created in that area. The second option is to create plates by drawing them one node at a time. You click on the grid point or node that you want to be the "A" node for the plate, then you click on the "B" node, "C" node, and then the "D" node in either clockwise or counter clockwise order. The plate will "stretch" like a rubber band as you draw from node to node.



The parameters shown are the same parameters that you would enter on the **Plates** spreadsheet.

To Draw Plates

To draw Plates:

1. If there is not a model view already open:
 - a. Go to the **View** ribbon.
 - b. Click on the **Open 3D Views** icon to open a new view.
2. If you are not drawing between existing nodes, you need to create a drawing grid or define nodes on the **Node Coordinates** spreadsheet.
3. Click the **Plates** icon.
4. Set the plate properties:
 - a. You must click four points in a clockwise or counter-clockwise order.
To create a triangular plate click on the third joint twice.
 - b. If you chose to click in grid areas, create plates by clicking between the drawing grids.
5. To stop drawing altogether right click or press the **Esc** key.
6. To exit Plates mode:
 - a. Go to the **Home** ribbon.
 - b. Click on the **Plates** icon.



Note:

- The default plate local axis is Global. To set this to Nodal, see [Axis](#) settings in the **Model Settings**. For more information on plate local axes, see [Plate Local Axes](#).
- To draw more plates with different properties, press CTRL-D to recall the **Plate Properties** settings.
- You can also specify or edit plates in the **Plates** spreadsheet.
- You can also view and edit plate properties by clicking on a plate.
- You can undo any mistakes by clicking  **Undo** on the Quick Access toolbar. If you want to undo a previous modification, click the 'Undo' icon's arrow and choose the modification you want to undo. Keep in mind that all modifications made after the one you've chosen to undo, will also be undone.

Modify Plates

There are a number of ways to modify plates. You can view and edit the member data in the [Plates spreadsheet](#). You can click a plate to view and edit its properties. You can select multiple plates to graphically modify a possibly large selection of members.

The **Plates** section in the Properties panel lets you modify the properties of plates that already exist in your model. To do so, you select the plates you want to modify, specify the properties you want to change, and the program automatically updates the properties. You can modify plates one at a time by clicking them individually and then changing the properties in the **Properties** panel. See the [Graphic Selection](#) topic for more on selecting.

The parameters shown are the same as those used to define new plates.

Properties	
Plate ^	
Material	gen_Conc4NW v ...
Thickness, in	8
Label	P19
Additional Properties ^	
A Node	N88
B Node	N14
C Node	N107
D Node	N106
Advanced Properties ^	
Plane Stress	<input type="checkbox"/>
Local Axis Angle	NA
A Release	...
B Release	...
C Release	...
D Release	...
Activation	Active v
D Release	...
Activation	Active v

To Modify Plates

Plates can be modified by changing their properties. You can change properties for one or more plates at a time.

To modify Plates:

1. If there is not a model view already open:
 - a. Go to the **View** ribbon.
 - b. Click on the **Open 3D Views** icon to open a new view.
2. Click an individual plate or select multiple, then change the values in the **Properties** panel as desired.

For more details, refer to [Plate Information](#), [Corner Releases](#) and [Inactive Excluded](#).

As you make changes, the program automatically updates the plates.



Note:

- You can also modify plates in the **Plates** spreadsheet.
- You can undo any mistakes by clicking  **Undo** on the Quick Access toolbar. If you want to undo a previous modification, click the 'Undo' icon's arrow and choose the modification you want to undo. Keep in mind that all modifications made after the one you've chosen to undo, will also be undone.
- It may not be possible to perfectly align the plate local axes.

Plate Information

Just as with the node and members, you can click any plate to view it's properties in the 'Properties' panel. All of the same information that is stored in the **Plates** spreadsheet is displayed for the plate you choose, and can be edited. This is a quick way to view and change plate properties.



Note: For modifying a large selections of plates, using the spreadsheet and graphic editing tools may be a faster method.

Properties	
Plate ^	
Material	gen_Conc4NW v ...
Thickness, in	8
Label	P19
Additional Properties ^	
A Node	N88
B Node	N14
C Node	N107
D Node	N106
Advanced Properties ^	
Plane Stress	<input type="checkbox"/>
Local Axis Angle	NA
A Release	...
B Release	...
C Release	...
D Release	...
Activation	Active v

The following table includes descriptions of the options available for modification.

Properties Panel - Plate Options

Section	Description
Material	Display the material used for the plate, and lets you edit it.
Thickness	Displays the plate thickness (in the current dimension units), and lets you edit them.
Label	Display the current label of the selected plate, and lets you edit that label.
Corner Nodes (A, B, C and D)	Displays the corner node labels, and lets you edit them.
Local Axis Angle	Displays the angle that determines the orientation of the local coordinate system of a plate element in relation to the global coordinate system.

Section	Description
Plane Stress	This box should be checked when you want to change the plate formulation to indicate that the plate is a Plane Stress plate. This option is useful for creating diaphragms that have only in-plane stiffness and will not attract out of plane moments. The default is unchecked which allows for both in-plane and out of plane behavior.
Plate Corner Releases (A, B, C and D)	Displays the corner release for the selected plate, and lets you edit them. See Plate Corner Releases for more details.
Activation	<p>Display the current activation state of the plate, and lets you change it.</p> <p>If the plate is made inactive, you will need to activate the plate from the Plates spreadsheet, or by using the Criteria Select feature to find and select inactive plates.</p> <p>See Inactive and Excluded Plates for more details.</p>

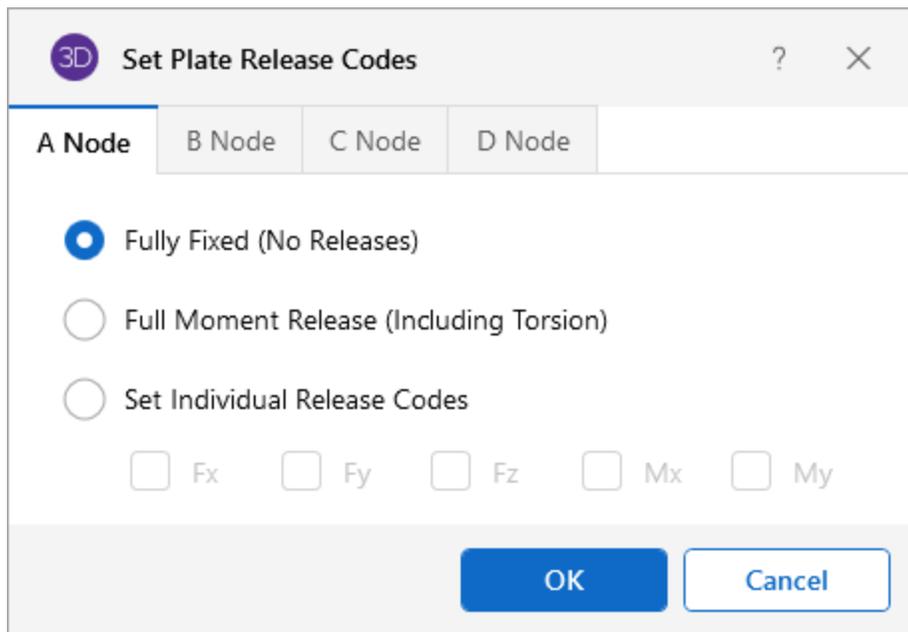
Plate Corner Releases

The **A, B, C, & D Release** fields are used to designate whether the forces and moments at the corners of the plate are considered fixed to or released from the plates's points of attachment (the A, B, C, and D nodes). Each plate has 5 force components at each corner (Fx, Fy, Fz, Mx, and My). Any or all of these force components can be released from the plate's point of attachment. If a force component is released, that force is not transferred between the node and the plate.

To Specify Plate Corner Releases

To specify plate corner releases:

1. Expand the **Advanced Properties** section of the Properties panel, if not already expanded.
2. Click on the **A, B, C, or D Release**  (ellipsis) button for the plate.
3. Specify the condition.



RISA-3D has a special "keyword" release configuration built-in. That is:

PIN => Mx and My (all moments) released (OOOXX)

This keyword entry is included because 99% of the release configurations you'll ever want to define will be "PIN". You can call out the keyword entry by just entering the first letter of the keyword, "p". So if you go to a release field and enter "p", the keyword "PIN" will be filled in automatically.

**Note:**

Alternatively, you can specify the corner condition by directly typing in the A, B, C or D Release box.

Advanced Properties	
Plane Stress	<input type="checkbox"/>
Local Axis Angle	NA
A Release	<input type="text"/> ...
B Release	<input type="text"/> ...
C Release	<input type="text"/> ...
D Release	<input type="text"/> ...
Activation	Active ▾

- To indicate that a force component is released, type an '**X**' for that component in the release box.
- To indicate that a force component is not released, type an '**O**' for that component in the release box.



Note: It's generally more efficient to use the **Graphic Editing** features if you want to change the properties for many plates at once.

Inactive and Excluded Plates

Making an item such as a member or plate inactive allows you to analyze the structure without the item, without having to delete the information that defines it. This leaves data intact so the item may be easily reactivated. This is handy if you want to try a model with and then without certain items, without having to actually delete the data.

- Choosing **Inactive** means the item is not included when the model is solved or plotted.
- Choosing **Excluded** means include the item in the solution, but exclude it from the results list.

So, an item set to **Excluded** is treated like any other plate in the solution and plotting of the model, but the plate will not be listed in the solution results (forces, stresses etc.). This is useful if there are certain items whose results you're not interested in. You don't have to clutter up the results with these items and can concentrate on the items you're most interested in. See [Printing](#) for more limiting printed results.

Rotate Plates

The Rotate Axis options let you perform a clockwise, or counter clockwise rotation of plate local axes so that they can better align with the surrounding plate local axes. It also lets you flip the local z-axis so that it is headed in the other direction. The degree increment of the clockwise or counter clockwise rotation depends on whether the [Local Axis](#) setting in Model Settings is 'Global' or 'Nodal'. When set to Global, rotation is based on whatever degree you set, and the plate's local axes are rotated by that degree. When 'Nodal' is set however, rotation is automatically set at 90°, where each click rotates the plate(s) local axes 90° at a time. Global is the Local Axis default, though you can change it to Nodal in the [Axis](#) tab of the 'Model Settings' window. If you would like to view the rotation of the plate(s) in the 3D View panel, [turn on the display of Local Axis Labels](#).

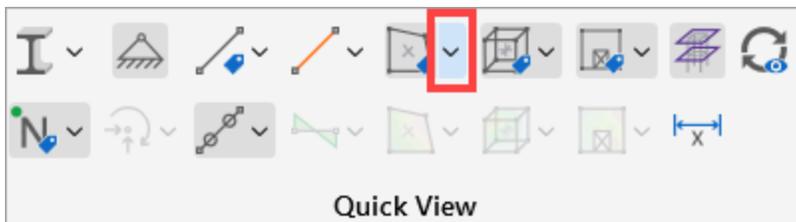


Note: While you can use the Rotate Axis feature to flip the local z-axis so that it is headed in the other direction, you may find the [Flip Axis](#) feature more convenient to use.

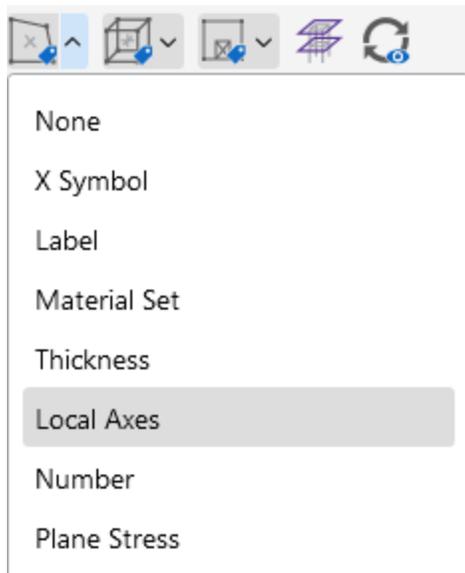
Turn on Display of Local Axis Labels

To turn on the display of local axis labels on the plates:

1. Open to the **Home** ribbon, if not already open.
2. Click the **Plate Labels** down arrow in the Quick View section.

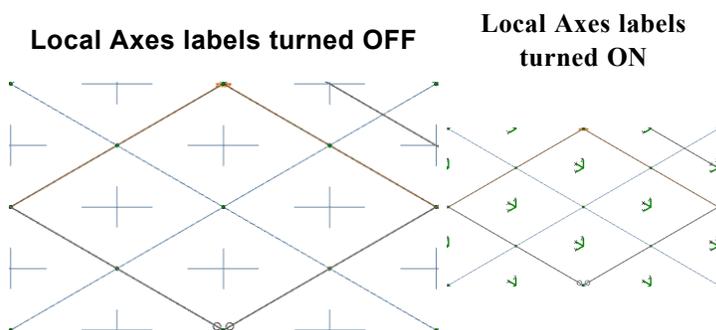


A list of labels opens.



3. Choose **Local Axes** from the list.

The plates display the Local Axes labels. The following images depict the local axes labels turned off and on.

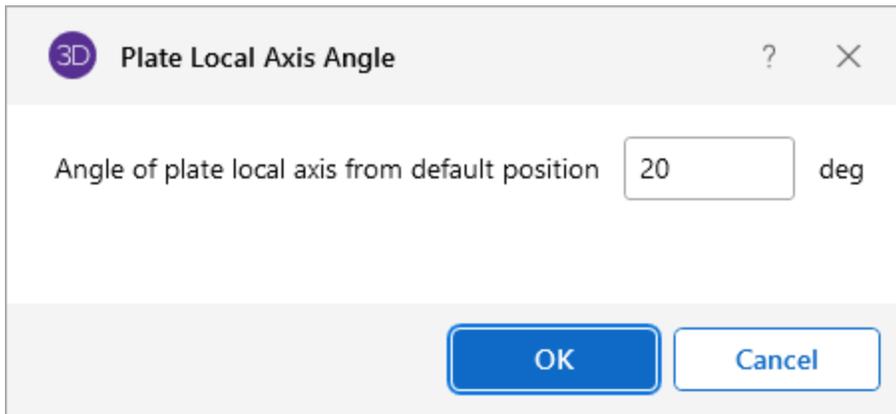


Rotate a Single Plate

To rotate a single Plate:

1. Go to the **Modify** ribbon, if it isn't already open.
2. Click the **Rotate Axis** icon in the Plates section.
3. Choose the **Single** option.

The Plate Local Axis Angle window opens, if 'Global' is set as the 'Plate Local Axis Orientation' in [Model Settings](#). If set to 'Nodal' you can ignore the remaining steps, as the plates are automatically rotated at 90° increments.



4. Enter the degrees you want to move the plate from its default position.
5. Click **OK** to close the window.
6. Click on the plate you want to rotate, for the action to take place.



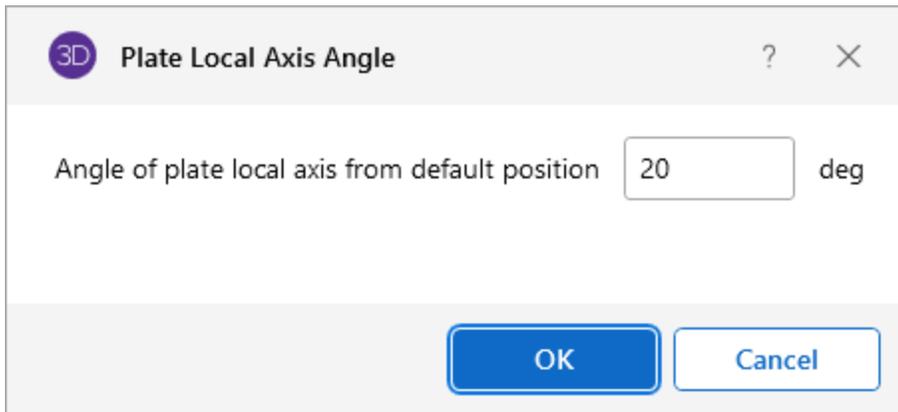
Note: You can hold down the **Ctrl** key while clicking on more plates, to apply the same degree of rotation to each plate. Release the Shift key after all plates you want have been selected and rotated.

Rotate Multiple Plates At Once

To rotate multiple plates the same degree, all at the once:

1. Hold the **Ctrl** key while clicking on each plate you want to rotate.
2. Go to the **Modify** ribbon, if it isn't already open.
3. Click the **Rotate Axis** icon in the Plates section.
4. Choose the **Selected** option.

The Plate Local Axis Angle window opens, if 'Global' is set as the 'Plate Local Axis Orientation' in [Model Settings](#). If set to 'Nodal' you can ignore the remaining steps, as the plates are automatically rotated at a 90° increment.



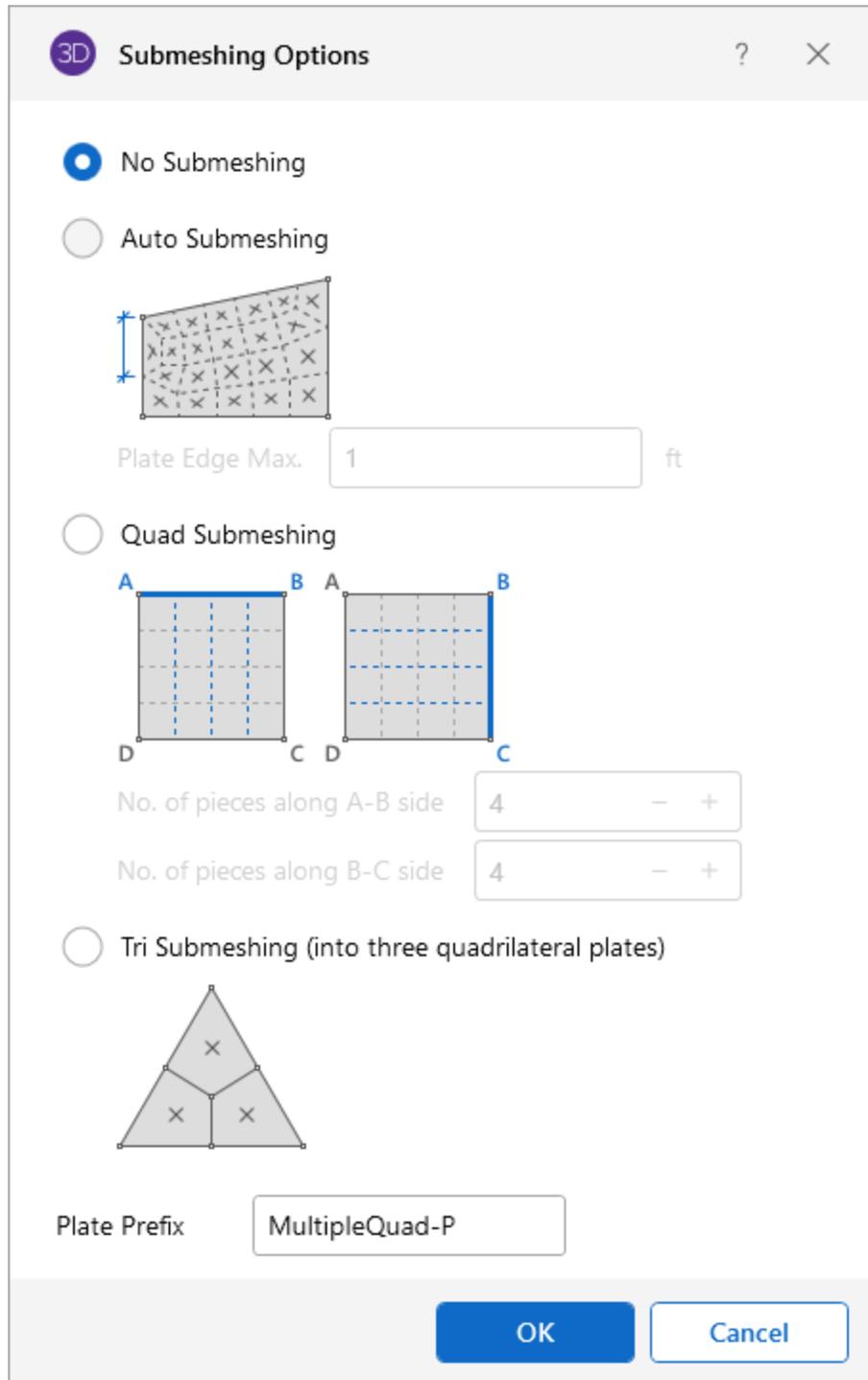
5. Enter the degrees you want to move the plate from its default position.
6. Click **OK** to close the window.

Submesh Plates

You can choose to submesh a plate [during the drawing process](#) or submesh [one or more existing plates](#) after they have been drawn. The second method is useful when you have several plates that you want to submesh with the same properties. When working with submeshing, you can choose from three different submeshing options, further explained in the following sections:

- [Auto Submesh Plates](#)
- [Quadrilateral Plates](#)
- [Triangular Plates](#)

There is a “Submeshing Options” window that provides a visual of each of the submesh options.



This window can be accessed by either clicking the **Meshing**  (ellipsis) button when the 'Plates' mode has been activated (from the 'Home' ribbon), or by clicking the  (ellipsis) button next to any one of the properties in the 'Properties' panel when **Submesh Plates** mode has been activated (from

the 'Modify' ribbon). Note that you can also choose a submesh option and set the same properties in this window as from you can in the Properties panel.

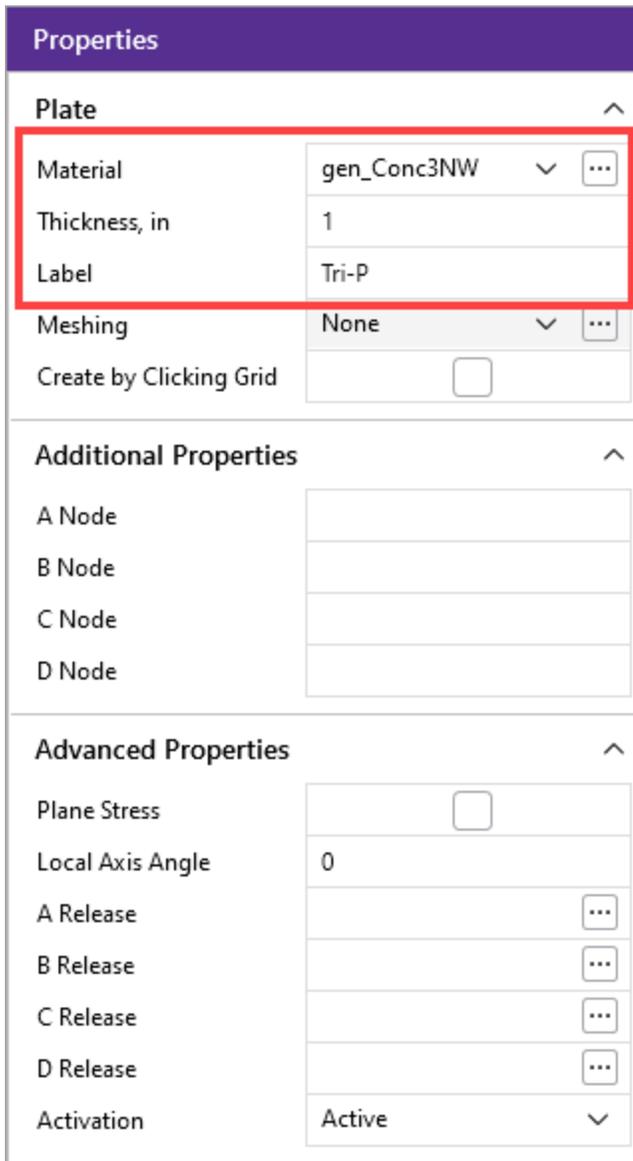
To Submesh a Plate When Drawing It

With this submeshing method, you can both create a plate and submesh it at the same time. When you finish clicking on the last node of the plate, the submeshing option and properties you've chosen, are automatically applied to the plate. Note that this method can be used only when drawing a new plate.

To submesh a plate when you first draw it:

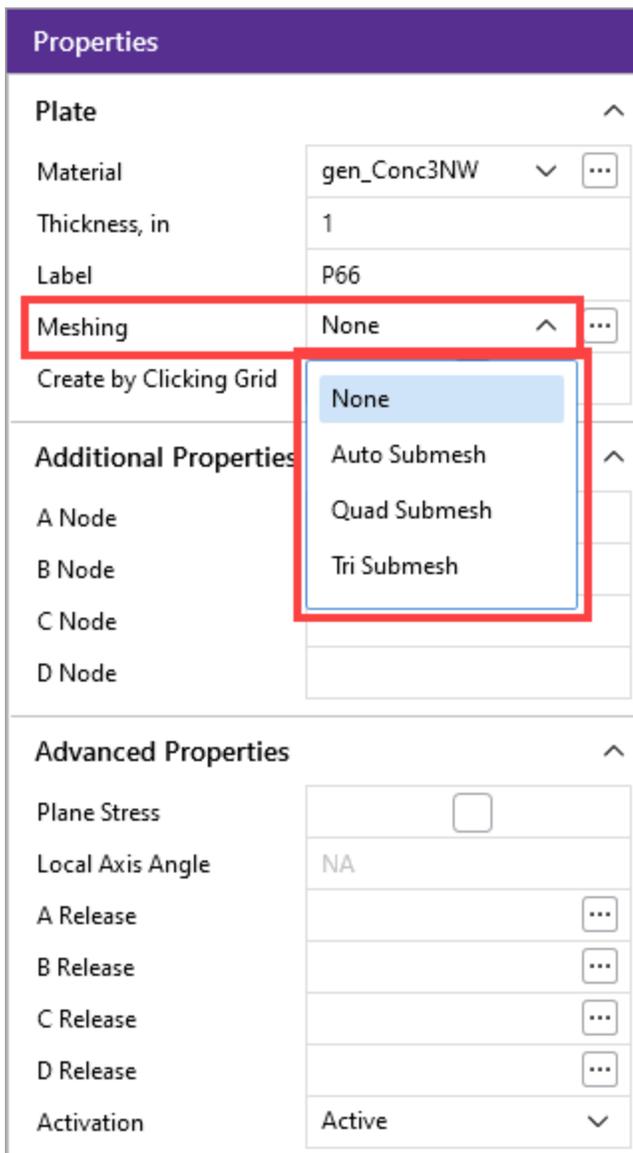
1. Go to the **Home** ribbon.
2. Click the **Plates** icon in the Draw Elements section, to turn on the Plates mode.

The Properties panel displays the properties for Plates, as shown in the following image.



Properties	
Plate ^	
Material	gen_Conc3NW v ...
Thickness, in	1
Label	Tri-P
Meshing	None v ...
Create by Clicking Grid	<input type="checkbox"/>
Additional Properties ^	
A Node	
B Node	
C Node	
D Node	
Advanced Properties ^	
Plane Stress	<input type="checkbox"/>
Local Axis Angle	0
A Release	...
B Release	...
C Release	...
D Release	...
Activation	Active v

3. Enter the plate information (**Material**, **Thickness** and **Label**).
4. Click the **Meshing** arrow and choose the submesh option you want used for the plate.



There are three options for submeshing plates: [Auto Submesh](#), [Quad Submesh](#), and [Tri Submesh](#), as indicated by the three options available from the drop down list in the image above.

Regardless of which option you chose, the Properties panel refreshes and displays the related submeshing properties for that option, directly below Meshing.



Note: Click on the **Meshing**  (ellipsis) button (in the Properties panel) to open the Submeshing Options window, which provides a visual of each available submesh option. You can also choose the option and its properties from the window, and then save your choices by clicking the **OK** button. [Click here](#) to view an image and short description of the Submeshing Options window.

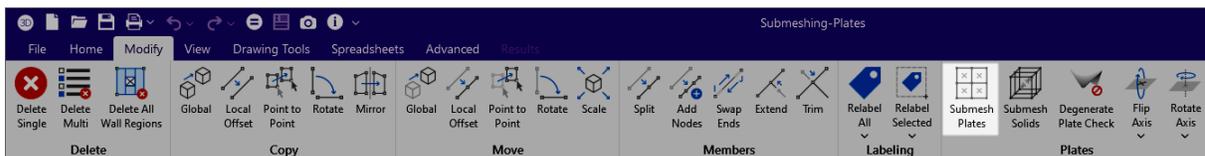
5. Enter the meshing properties you want used.
6. Enter an **Additional Properties** (if available) you want used.
7. Enter any **Advanced Properties** you want used.
8. Draw the plate in the model editing panel.

Immediately after click on the last connecting node, the plate is automatically updated with the submeshing option and settings you've chosen.

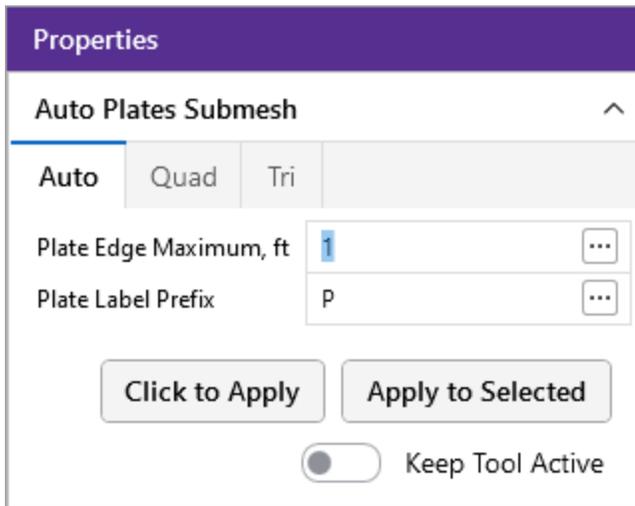
To Submesh Existing Plates

With this submeshing method, you can submesh any plates that already exist in the model and also modify existing submesh into more, thereby creating new submeshing.

1. Select the plate(s) you want to submesh.
To select multiple plates, hold down the **Ctrl** key while clicking on each plate.
2. Go to the **Modify** ribbon.



3. Click the **Submesh Plates** icon.
The Properties panel displays 'Meshing' under the Plate section.



There are three options for submeshing plates: **Auto** (automatic submesh), **Quad** (submesh quadrilateral plates) and **Tri** (submesh triangular plates).

The following sections provide more detail on each of the submesh options.

- [Auto Submesh Plates](#)
- [Quadrilateral Plates](#)
- [Triangular Plates](#)

4. Click on the tab that coincides with the submesh option you want to choose.



Note: Alternatively, you can click the ellipsis button next to any setting found under any of the tabs, to open the “Submeshing Options” window. The Submeshing Options window let you choose the meshing option and set its related properties as well. Clicking the **OK** button saves your choices. [Click here](#) to see an image and short description of the Submeshing Options window.

5. Set the submesh properties for your selected option.

You can submesh the plates one at a time by selecting the **Click to Apply** option and then clicking on the plates you wish to submesh.

6. Do one of the following:

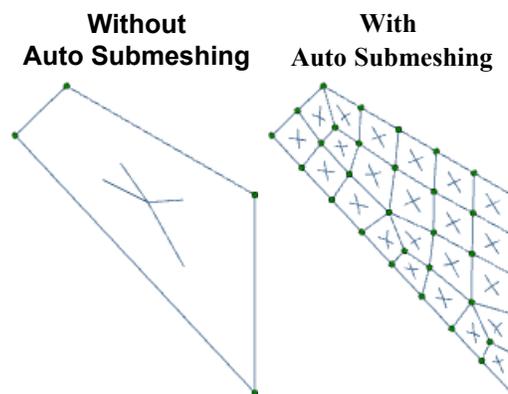
- Click **Click to Apply** button to submesh the plates one at a time by selecting the **Click to Apply** option and then clicking on the plates you wish to submesh.
- Click **Apply to Selected** button to modify entire selections of plates by selecting the plates and then using the **Apply to Selected** option.

Auto Submesh Plates

The **Auto Submesh** option in the Meshing drop down menu in the **Plates** section of the Properties panel lets you draw a polygon that RISA-3D will automatically submesh into smaller quadrilateral

plate elements. Just as with [Drawing Plates](#), the material set and plate thickness of the plates within the mesh may be indicated prior to drawing the polygon. In addition to these parameters, a plate edge maximum can also be provided. Polygons of virtually any size and shape may be drawn provided the drawing lines do not cross. The polygon points must be on the same plane, either one of the three global planes (XY, XZ, or YZ) or co-planar. Polygons may be drawn in either a clockwise or counter-clockwise direction. To complete a polygon, simply double click on the last node/grid intersection, or click on the starting node/grid intersection. Once a polygon is drawn, RISA-3D will create a submesh of quadrilateral plate elements, limited by the edges of the polygon, and of a size corresponding to the plate edge maximum indicated by the user.

The following is an example of a quadrilateral plate without and with Auto submeshing.



Note: Only quadrilateral plate elements are created with the Auto Submesh tool.

When drawing a polygon with the Auto Submesh tool, any existing nodes within the boundary and in the plane of the polygon will be considered control points. These **Control Points** will be considered "fixed" points within the mesh and will dictate the layout of individual plates surrounding them. It is important to note that only currently selected nodes at the time the polygon is drawn will be used as control points.

To AutoMesh a Polygon

To automatically submesh a Polygon when you draw it:

1. If there is not a model view already open:
 - a. Go to the **View** ribbon.
 - b. Click on the **Open 3D Views** icon to open a new view.
2. If you are not drawing between existing nodes, you need to create a drawing grid or define nodes on the **Node Coordinates** spreadsheet.
3. Make sure that all nodes that are to be considered **Control Points** are selected.
4. Click the **Automesh of Plates** icon on the 'Home' ribbon.

5. Set the **Plate Edge Maximum** setting in the Properties panel.
6. Start drawing a polygon by clicking on the nodes or grid points with the left mouse button.
7. To complete the drawing of the polygon, double click the last point or click on the original starting point.
It is at this point that the plate is automatically submeshed.
8. Right click or press the **Esc** key to stop drawing altogether.

**Note:**

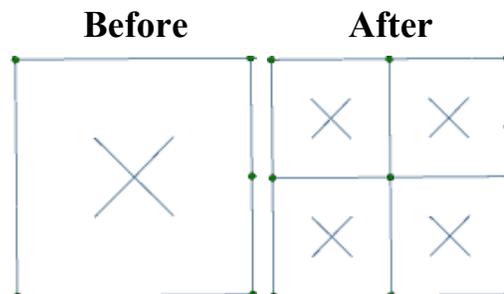
- You can also specify or edit plates in the **Plates** spreadsheet.
- You can also view and edit plate properties by clicking on a plate.
- You can undo any mistakes by clicking the  **Undo** button in the Quick Access toolbar. If you want to undo a previous modification, click the 'Undo' icon's arrow and choose the modification you want to undo. Keep in mind that all modifications made after the one you've chosen to undo, will also be undone.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Plate Mesh**.

Quadrilateral Plates

You can submesh quadrilateral (4 sided) plate elements into a mesh of smaller elements by using the **Quad** submesh option from the 'Meshing' drop down menu prior to drawing a plate or after the fact, by clicking the **Submesh Plates** icon in the **Modify** ribbon (see [To Submesh Existing Plates](#) for complete steps.) The new mesh can be any size up to the program limits for nodes and/or plates. The Quad option is very useful for refining a coarse mesh of elements; just make sure that all adjacent plate elements (elements sharing an edge) maintain connectivity.

The following is an example of a plate before and after applying Quad submeshing to it.

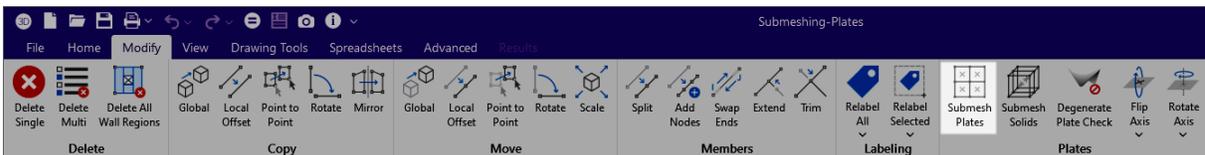


You can define different submesh increments in each direction. The A,B,C and D nodes for each plate are displayed in the plates spreadsheet. The A node is the first node clicked on when you created the plate. The B node is the second and so on. In addition, if the plate local axes is set to Nodal, you can determine which side is which by displaying the plate local axes and realizing that the local x-axis is parallel to the D-C edge of the plate.

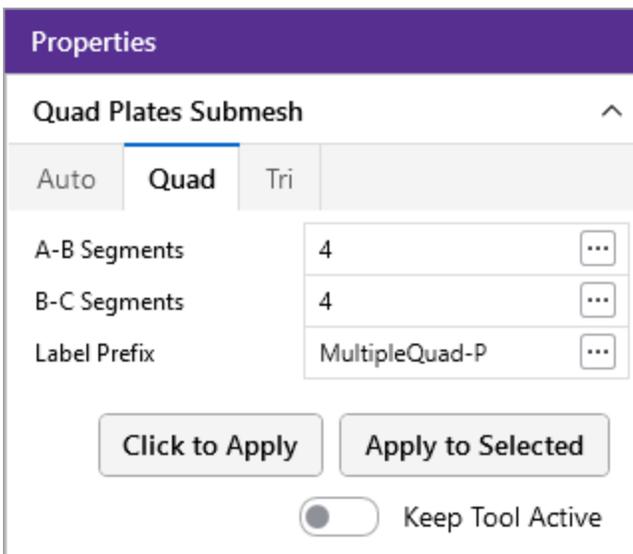
To Submesh Quadrilateral Plates

To submesh Quadrilateral Plates:

1. If there is not a model view already open:
 - a. Go to the **View** ribbon.
 - b. Click on the **Open 3D Views** icon to open a new view.
2. Select the plates you want to sub mesh.
3. Go to the **Modify** ribbon, if it isn't already open.



4. Click the **Submesh Plates** icon.
The Properties panel displays the Plate submeshing options.



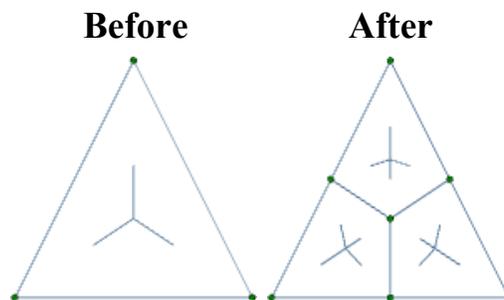
5. Click the **Quad** tab to display its property settings.
6. In the **A-B Segments** and **B-C Segments** boxes, specify the number of plates.
7. (Optional) In the **Label Prefix** box, change the prefix if you don't want to use the default (default is "P").
8. Click the **Apply to Selected** button.

**Note:**

- To submesh more plates with different parameters, press CTRL-D to recall the **Submesh Plates** action.
- You can undo any mistakes by clicking the  **Undo** button in the Quick Access toolbar. If you want to undo a previous modification, click the 'Undo' icon's arrow and choose the modification you want to undo. Keep in mind that all modifications made after the one you've chosen to undo, will also be undone.

Triangular Plates

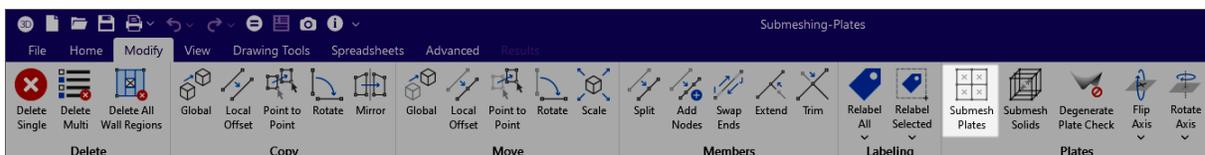
This is used to sub-mesh the selected triangular (3 sided) elements into a mesh of 3 quadrilaterals. This is done by first creating a new node at the center of each selected triangular element and also at the center point along each edge of the triangular element. These new nodes are then used to create three quadrilateral elements that replace the triangular element.



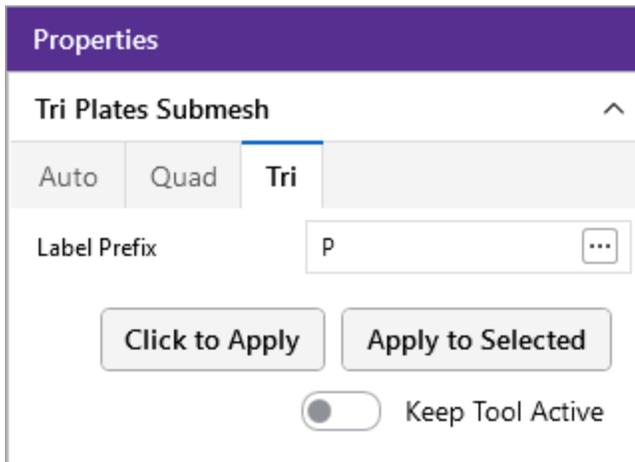
To Submesh Triangular Plates

To submesh Triangular Plates:

- If there is not a model view already open:
 - Go to the **View** ribbon.
 - Click on the **Open 3D Views** icon to open a new view.
- Select the plates you want to sub mesh.
- Go to the **Modify** ribbon if it doesn't automatically open.



- Click the **Submesh Plates** icon.
The Properties panel displays the Plate submeshing options.



5. Click the **Tri** tab to display its property settings.
The only property you can change is the label, which is optional.
6. (Optional) In the **Label Prefix** box, change the prefix if you don't want to use the default (default is "P").

**Note:**

- To submesh more plates with different parameters, press CTRL-D to recall the **Submesh Plates** action.
- You can undo any mistakes by clicking the  **Undo** button. If you want to undo a previous modification, click the 'Undo' icon's arrow and choose the modification you want to undo. Keep in mind that all modifications made after the one you've chosen to undo, will also be undone.

Plates Spreadsheet - Primary Data

The **Plates** spreadsheet records the properties for the plate/shell elements of the model. The spreadsheet can be accessed by selecting **Plates** from the 'Data Entry' section of the 'Explorer' panel.

Plate Primary Data							
Primary	Advanced						
	Label	A Node	B Node	C Node	D Node	Material	Thickness [in]
1	P2	N6	N77	N90	N89	gen_Conc4NW	8
2	P3	N77	N78	N91	N90	gen_Conc4NW	8
3	P4	N78	N31	N92	N91	gen_Conc4NW	8
4	P5	N31	N79	N93	N92	gen_Conc4NW	8

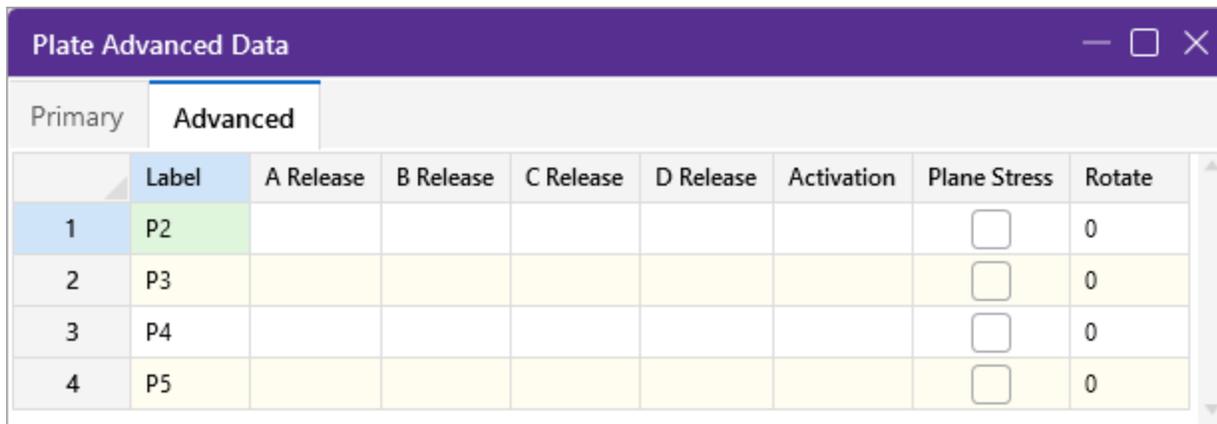
The following table provides descriptions for the columns in the Plate Primary Data spreadsheet.

Plates Spreadsheet - Primary Data

Column(s)	Description
Plate Labels	<p>Displays the label currently assigned to each plate. These labels allow you to refer to a plate by using its label. Note that each label must be unique. If you try to enter the same label more than once, you get an error message.</p> <p>In this spreadsheet, you can assign a different unique label to any or all of the plates using one of the following methods.</p> <ul style="list-style-type: none"> • Spreadsheet: Use this method to change one or more plate labels while in the Plate spreadsheet. Simply double-click on the plate's current Label and type the new label. • Relabel All: Use this method to change the Prefix of all plate labels at once. Go to the 'Modify' ribbon and click the Relabel All icon. • Relabel Selected: Use this method to relabel only specific plates that you have selected in the 3D View panel. Go to the 'Modify' ribbon and click the Relabel Selected icon. <div style="border: 1px solid black; padding: 5px; margin-top: 10px;"> <p> Note: Using the Relabel All or Relabel Selected icons in the 'Modify' ribbon, you can relabel plates at any time.</p> </div>
Plate Nodes (A, B, C and D)	<p>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.</p> <p>The direction and sequence in which you define the nodes determines how the elements local coordinate system is set up. This is discussed in</p>

Column(s)	Description
	the section on Plate Local Axes .
Plate Material	<p>The material set label links the plate with the desired material defined on the Materials spreadsheet.</p> <div style="border: 1px solid black; padding: 5px; margin-top: 10px;">  Note: Plates are always defined with general materials. This is because the other material sets (Hot Rolled, Cold Formed, Wood, and Concrete) are used to designate member code checking specifications. Since plates are only used for analysis, no code checking is provided and the material must be designated as a general material. </div>
Plate Thickness	The thickness field on the Plates spreadsheet is the thickness of the element. This thickness is constant over the entire element.

Plates Spreadsheet - Advanced Data



	Label	A Release	B Release	C Release	D Release	Activation	Plane Stress	Rotate
1	P2						<input type="checkbox"/>	0
2	P3						<input type="checkbox"/>	0
3	P4						<input type="checkbox"/>	0
4	P5						<input type="checkbox"/>	0

The following table provides descriptions for the columns in the Plate Advanced Data spreadsheet.

Plate Spreadsheet - Advanced Data

Column(s)	Description
Plate Labels	<p>Displays the label currently assigned to each plate. These labels allow you to refer to a plate by using its label. Note that each label must be unique. If you try to enter the same label more than once, you get an error message.</p> <p>In this spreadsheet, you can assign a different unique label to any or all of the plates using one of the following methods.</p> <ul style="list-style-type: none"> • Spreadsheet: Use this method to change one or more plate labels while in the Plate spreadsheet. Simply double-click on the plate’s current Label and type the new label.

Column(s)	Description
	<ul style="list-style-type: none"> • Relabel All: Use this method to change the Prefix of all plate labels at once. Go to the 'Modify' ribbon and click the Relabel All icon. • Relabel Selected: Use this method to relabel only specific plates that you have selected in the 3D View panel. Go to the 'Modify' ribbon and click the Relabel Selected icon. <div style="border: 1px solid black; padding: 5px; margin-top: 10px;">  Note: Using the Relabel All or Relabel Selected icons in the 'Modify' ribbon, you can relabel plates at any time. </div>
Plate Nodes (A, B, C and D)	The Plate Corner Releases for nodes A, B, C, and D of each plate may be set in these four data columns. See Plate Corner Releases for more information.
Activation	Allows for plates to be set to 'Active', 'Inactive', or 'Excluded'. These choices can be made by selecting them from the drop down list or by entering an 'I' for 'Inactive' or an 'E' for 'Excluded'. If the cell is left blank the plate will be 'Active' by default. See Inactive and Excluded Plates for more information.
Plane Stress	<p>Allows you to change the plate formulation to a 'Plane Stress' plate, by checking the box. When checked, only in-plane stiffness behavior is allowed. The default is unchecked, which allows for both in-plane and out of plane behavior.</p> <p>This option is useful for creating diaphragms that have only in-plane stiffness and will not attract out of plane shear or moment.</p>
Rotate	<p>Displays the angle that determines the orientation of the local coordinate system of a plate element in relation to the global coordinate system.</p> <p>What appears in this column depends on the option selected as the 'Plate Local Axis Orientation' option in the Model Settings window.</p> <ul style="list-style-type: none"> • If the 'Plate Local Axis Orientation' option is set to "Global" (the default for all new models), a numerical value indicating the angle appears in this column. The value will appear as "0" for brand new models. • If the 'Plate Local Axis Orientation' option is set to "Nodal", "NA" appears in this column, indicating that rotation is not applicable.

Plate Local Axes

The A, B, C, and D nodes are used to define the corners of a quadrilateral element. (To define a 3-node element, just leave the D node 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 direction.

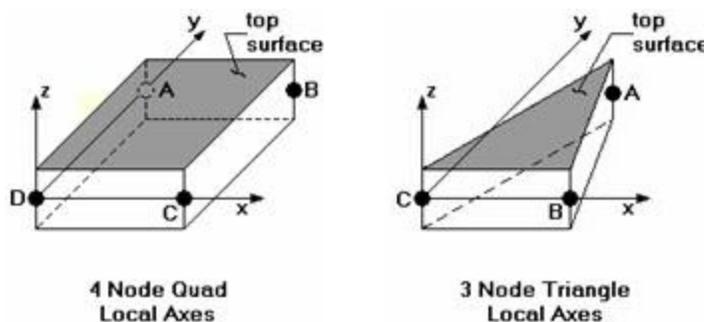
The direction and sequence in which you define the joints determines how the elements local coordinate system is set up. The plate local axes can be set to 'Global' or 'Nodal' in the [Axis](#) settings of the 'Model Settings' window. The following images depict the difference between Global and Nodal plate local axes, which are shown graphically in green.

To view Local Axis labels on plates, as shown in the following images, see [Turn on Display of Local Axis Labels](#) in this chapter.

When the plate local axis is set to 'Global', the local axes of all plates are oriented in the same orthogonal direction, regardless of plate geometry. However, the positive out-of-plane z-axis are dependent on the configuration in which the plate or mesh was drawn. The plate local out-of-plane axis is always the z-axis, but the positive direction of this axis is dependent on the direction in which you define the nodes of the plate. Defining the nodes in a clockwise pattern results in the out-of-plane z-axis pointing in the positive direction according to the global axes. For example, in the global XZ-plane, a plate drawn in the positive clockwise direction results in the plate local out-of-plane z-axis pointing along the global positive Y-axis. Similarly, a counter-clockwise pattern results in the out-of-plane z-axis pointing in the negative global axes direction. Plates do not follow the right-hand rule that is common in three-dimensional vector orientation. For more information on the interpretation of plate results based on local axis orientation, see [Plate Force Results](#).

When the plate local axis is set to Nodal, the local x-axis is defined as positive from the D node towards the C joint 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-joint and perpendicular to the x-axis. Once the x and y axes are defined, the positive local z-axis is found using the right hand rule.

The following diagrams illustrate how the elements local coordinate system is related to the node numbering sequence and direction:



The default plate local axis is set to 'Global'. To change this to 'Nodal', see [Axis](#) settings in the [Model Settings](#) section.

Plate/Shell Element Formulation

The element used is a mixed interpolation 4 node quadrilateral element. By mixed interpolation, we mean that the in-plane and transverse shear strain components are derived independently. This

allows the element to be easily simplified into a plane stress element in cases where transverse shear and bending are not desired. A reference for this element is *Finite Element Procedures*, by K.J. Bathe, Prentice-Hall, 1996. The book also provides many references for papers on the elements convergence and other characteristics. In brief, the element can model isotropic behavior for plane stress, plate bending and out-of-plane transverse shear.

This is accomplished by starting with the Mindlin-Reissner plate assumptions and adding interpolating functions for the out-of-plane transverse shear. This approach is analogous to incorporating shear deformation with flexural effects in beam theory. This results in an element that can be used for thin and thick plate applications. Traditional plate elements do not model out-of-plane transverse shear well (if at all) and cannot be used for thick plate applications. The element is also very insensitive to distortion.



Note: The linear-elastic behavior of the program implicitly means that non-linear (2nd order) behavior is not generally supported. Tension-structures such as fabric structures would fall into this non-linear category. Please see the [Modeling Tips - Modeling a Cable](#) section for some ways to approximate this behavior. This section is applicable to members but the principles illustrated are also applicable to plate modeling as well.

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. In fact, RISA-3D's [Auto Submesh](#) tool will only create quadrilateral plate elements for this very reason. RISA-3D provides a way to convert your triangular plates to quadrilaterals, see [Submeshing Triangular Plates](#).

Orthotropic Behavior

The RISA plate element can use two types of plate methodologies for analysis: Isotropic and Orthotropic. The plate material and methodology can be defined in the Materials spreadsheet under the General tab.

General Materials Properties							
Hot Rolled	Cold Formed	Wood	Concrete	Masonry	Aluminum	Stainless	General
	Label	E [ksi]	G [ksi]	Nu	Therm. Coeff. [$10^{-5}F^{-1}$]	Density [k/ft ³]	Plate Methodology
1	gen_Conc3NW	3155	1372	0.15	0.6	0.145	Isotropic
2	gen_Conc4NW	3644	1584	0.15	0.6	0.145	Isotropic
3	gen_Conc3LW	2085	906	0.15	0.6	0.11	Isotropic
4	gen_Conc4LW	2408	1047	0.15	0.6	0.11	Isotropic
5	gen_Alum	10600	4077	0.3	1.29	0.173	Isotropic
6	gen_Steel	29000	11154	0.3	0.65	0.49	Isotropic
7	RIGID	1e+6		0.3	0	0	Isotropic
8	gen_Ortho	1e+6		0.3	0.65	0.49	Orthotropic ^ ...

Isotropic
 Orthotropic

By default the material properties shown in the Materials/General spreadsheet are for Isotropic material. Isotropic materials can be used by all element types in the program such as Members, Wall Panels, Solids, and Plates. By using the Isotropic plate methodology, the plate element does allow a limited degree of orthotropic material behavior. Specifically, the In-Plane shearing of the plate will be almost entirely controlled by the G value for the material whereas the direct In-Plane compressive stiffness will be controlled by the E value of the material.

If you toggle the Plate Methodology to Orthotropic, the detailed orthotropic material properties can be defined in the Edit Orthotropic Material window after clicking the ... (ellipsis) button in the Plate Methodology cell.

Property	Value	Unit
E1	29000	ksi
E2	29000	ksi
G12	11154	ksi
G13	11154	ksi
G23	11154	ksi
Nu12	0.3	
Nu21	0.3	

The orthotropic material properties can be defined per plate local axes. E1 and E2 correspond to E_x and E_y , which are Young's modulus in plate's local x and y directions. Similarly, G12, G13, and G23 correspond to G_{xy} , G_{xz} , and G_{yz} , which are shear modulus per plate's local axes. Nu12 corresponds to ν_{xy} , which is the Poisson's ratio between plate's local x and y direction.

All plates need to have their local axes orientation set as "Global" instead of "Nodal" for the program to analyze orthotropic behavior. This can be set in the Plate Local Axis Orientation setting under the Global Model Setting.

The following stress-strain relationship is used in the program for orthotropic material:

$$\begin{bmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{bmatrix} = \begin{bmatrix} \frac{E_1}{1-\nu_{12}\nu_{21}} & \frac{\nu_{12}E_2}{1-\nu_{12}\nu_{21}} & 0 \\ \frac{\nu_{12}E_2}{1-\nu_{12}\nu_{21}} & \frac{E_2}{1-\nu_{12}\nu_{21}} & 0 \\ 0 & 0 & \frac{1}{G_{12}} \end{bmatrix} \left(\begin{bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \gamma_{12} \end{bmatrix} + (T-T_0) \begin{bmatrix} \alpha_1 \\ \alpha_2 \\ 0 \end{bmatrix} \right)$$

where $\nu_{21} = \nu_{12} \frac{E_2}{E_1}$

$$\begin{bmatrix} \tau_{13} \\ \tau_{23} \end{bmatrix} = \begin{bmatrix} G_{13} & 0 \\ 0 & G_{23} \end{bmatrix} \begin{bmatrix} \gamma_{13} \\ \gamma_{23} \end{bmatrix}$$



Note: Currently orthotropic plate methodology can only be applied to Plate elements in the program. If an orthotropic material is applied to Members, Wall Panels, or Solids, it will be treated as an isotropic material, with E, G, and Nu taken as E1, G12, and Nu12 from the orthotropic material input. Warning messages will be given after solution to explain this behavior.

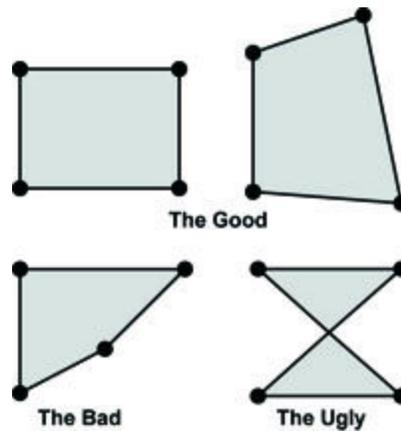
Plate Modeling Tips

A word of caution is in order if you are new to plate modeling. Unlike modeling with beam elements, plate elements require some understanding of finite element behavior to successfully obtain meaningful results. It is easy to build a finite element model using the powerful generators and graphic editing tools. However, without understanding the limitations of the analysis method used, you can end up with an impressive looking but very inaccurate model. Even if you've been engaged in structural engineering for years, modeling with plates is not something most engineers do frequently. It is therefore not realistic to have the expectation that you should be able to perform complicated analysis with plate elements in a short amount of time. Good plate modeling takes time, knowledge of plate and finite element behavior, and experience.

The first tip is to read all the Plate documentation before embarking on an ambitious modeling project. This will save you much aggravation down the road.

Plate Distortion

The finite elements in a model should be as undistorted as possible. See the following figure:



Non-Planar Plates

The plate element formulation used in RISA is not very sensitive to distortion within the plane of the plate itself (see the plate Distortion topic above). However, the plate element formation is particularly sensitive to non-planar plates. This means that it is critical that all 4 nodes that define a quadrilateral plate remain in the same plane. There is an internal tolerance of 0.01 inches built into the plate element formulation for maximum allowable planar distortion. This is appropriate for concrete slabs and shear walls and such. But, extra care should be given to avoiding out-of-plane plates for extremely small and thin plates.

Plate Generation

A fast way to build a new mesh of finite elements is with the generation features. RISA-3D currently provides several generation features to quickly build common structures providing an easy way to create cylinders, cones, grids, radial grids and disks of plates. The best way to see what these features do is to experiment with them. See [Generation](#) to learn more about generation.

Another time-saving method is to draw large elements to represent continuum such as slabs and shear walls, and then use the submesh features mentioned below to refine the mesh.



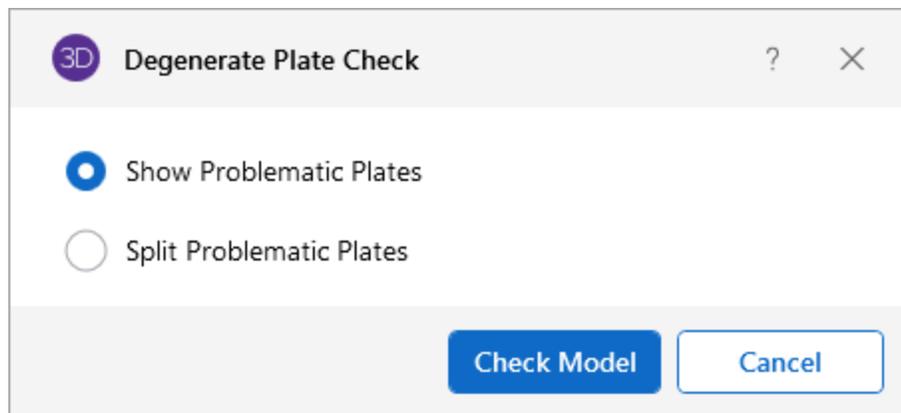
Note: Before sub-meshing, make sure that any adjacent “large” elements connect at their corner nodes. That way, any subsequent sub-mesh operations will produce element meshes that automatically connect at the intermediate nodes.

Automatic Plate Sub-Meshing

What if you’ve already built a model and you now decide that your finite element mesh is too coarse? To submesh elements see [Submeshing Plates](#). Performing a Model Merge after will insure that all the new elements get connected to existing beam elements and that duplicate nodes get merged. See [Model Merge](#) for more information.

Plate Error Checking

The program cannot provide a solution for plates which are poorly shaped or not planar. If any such "degenerate" plates exist, an error is produced during the solution. Click the **Degenerate Plate Check** icon in the 'Plates' section of 'Modify' ribbon to open the Degenerate Plate Check window to fix these issues.



The Degenerate Plate Check utility includes two options that are used to either show problematic plates or split them.

- The **Show Problematic Plates** option provides a model view showing all plates that were non-planar or poorly shaped. A summary report lists the number of degenerate plates, and the warning log is populated with an exact list.
- The **Split Problematic Plates** option splits each non-planar plate into four triangular plates. By definition a triangular plate is always planar, so this will allow the model to solve. The force and stress results for these triangular plates are not accurate, and should not be used for design. However, the stiffness is adequate for force transfer, so this splitting tool provides a convenient method to force a solution in a model where manually fixing the plates to be planar would be too time consuming, and where the plate stresses are not a concern.

Finite Element Basics

While this will not be a comprehensive treatment of plate and finite element fundamentals, a review of certain key basic concepts and terminology will be valuable to the engineer who has not worked with finite elements, or has not had the opportunity to use them recently.

A place to start is with the types of forces or stresses that can occur in a plate. One term that is commonly used is "plane stress". This term is used to describe a state of stress in a plate where all the stresses occur in the plane of the plate. A real world example would be a shear wall with forces applied only in the plane of the wall. The resulting plate forces would be just the normal stresses (F_x , F_y) and the in-plane shear stresses (F_{xy}). There would be no plate moments or out-of-plane shears generated.

Stress vs. Force

It should be pointed out that the results for a plate are always a stress. These stresses are multiplied by the plate thickness and the width or length to obtain a force. Note that this force obtained is just an average value for the plate, since the stress was for a point on the plate and it undoubtedly will vary throughout the plate area. The fact that the stresses vary within a plate is why a good finite element mesh is so critical to obtain accurate results. Stresses tend to vary more around point loads and supports, and less in regions that are far from supports and have a uniform load.

A different example of plate forces would be a horizontal diaphragm that is loaded only in the out-of-plane direction. The plate results would be plate moments, out-of-plane shears, but no membrane (plane stress) stresses. The reason for no membrane stresses is that there was no in-plane loading.

Sign Convention

One other comment on plate results is to point out the convention used for moments in plates. With beams, the 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. For more information, see [Plate Force Results](#).

Why Meshing Is Required

In a nutshell, finite elements tend to work by trying to approximate the correct deflected shape of the real world item being modeled. For example, if we are trying to model a horizontal diaphragm, simply supported on all edges, and loaded out-of-plane, our finite element model must be 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 gauge of “enough” for common structural elements is addressed in the Plate Model Examples section of the Reference Manual.

Distortion

Finite elements are also affected by geometric distortion. The best shape for the 4-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 degrees, and preferably shouldn’t even approach 180 degrees.

Drilling Degree of Freedom

One last item is that the element used by RISA-3D, like other plate/shell elements, cannot accurately model in-plane rotations. I.e., a plate/shell element will not provide resistance to a moment applied about the plate's local z-axis. For example, let's say you have a 4x4 grid of elements, simply supported about the edges, and you apply a node 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 node reactions and the element stresses. See [Applying In-Plane Moments to Plates](#) to learn how to work around this limitation.

Plates/Shells

The plate/shell finite element allows you to easily model shear walls, diaphragms, shells, tanks and many other surface structures. We refer to the elements as plate elements, but they are actually plate/shell elements. Plate data may be viewed and edited in three ways: graphically, in the **Information** dialog or in the **Plates** spreadsheet.

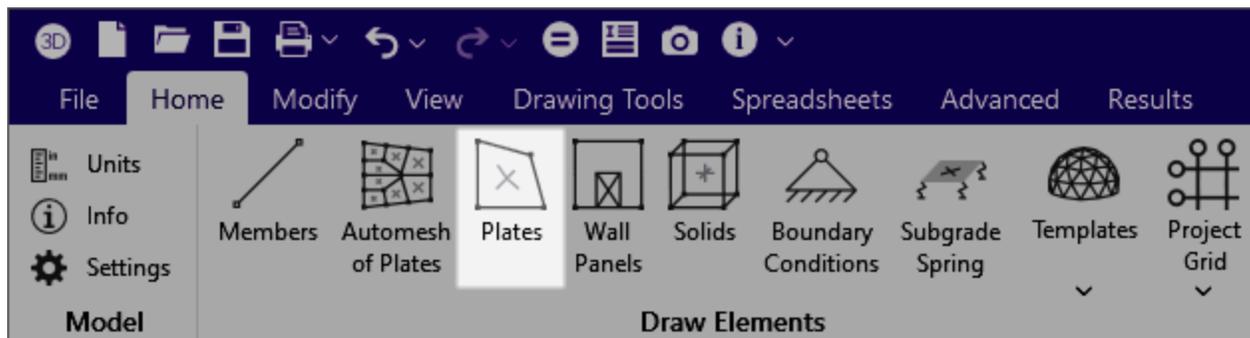
Draw Plates

There are several graphic-editing features that make the creation and modification of models quite easy. Use the Plates button in the Home tab and Modify menu to use these features in the model view. To create new members or plates, you can draw them using a drawing grid or draw "dot to dot" from existing nodes. Once you have created these items, you can use other graphic features to load the model and set boundary conditions.

Creating plate models requires more forethought than beam models. See [Plate Modeling Tips](#) and [Plate Modeling Examples](#) for tips on building plate models. To create new plates you can draw them using a drawing grid, a project grid, or draw "dot to dot" from existing nodes. You can set all of the plate properties up front or you can modify these properties after you draw them. Modifying properties is discussed in the next sections. See [Plate Spreadsheet](#) for information on plates and their properties.

The **Plates** tool lets you graphically draw plates in your model. Enter the appropriate plate parameters, and draw plates between existing nodes or on the drawing grid. You will also notice that the coordinates of the node or grid point that is closest to your cursor are displayed next to your cursor. The new plates will be shown on screen and will be recorded in the **Plates Spreadsheet**.

To actually draw a plate, you have two options. The fastest way is to use the **Create Plates by Clicking on Grid Areas** option, and then create plates by clicking on the grid areas formed by the intersecting grid lines. As you click on an area, a plate will automatically be created in that area. The second option is to create plates by drawing them one node at a time. You click on the grid point or node that you want to be the "A" node for the plate, then you click on the "B" node, "C" node, and then the "D" node in either clockwise or counter clockwise order. The plate will "stretch" like a rubber band as you draw from node to node.



The parameters shown are the same parameters that you would enter on the **Plates** spreadsheet.

To Draw Plates

To draw Plates:

1. If there is not a model view already open:
 - a. Go to the **View** ribbon.
 - b. Click on the **Open 3D Views** icon to open a new view.
2. If you are not drawing between existing nodes, you need to create a drawing grid or define nodes on the **Node Coordinates** spreadsheet.
3. Click the **Plates** icon.
4. Set the plate properties:
 - a. You must click four points in a clockwise or counter-clockwise order.
To create a triangular plate click on the third joint twice.
 - b. If you chose to click in grid areas, create plates by clicking between the drawing grids.
5. To stop drawing altogether right click or press the **Esc** key.
6. To exit Plates mode:
 - a. Go to the **Home** ribbon.
 - b. Click on the **Plates** icon.

**Note:**

- The default plate local axis is Global. To set this to Nodal, see [Axis](#) settings in the **Model Settings**. For more information on plate local axes, see [Plate Local Axes](#).
- To draw more plates with different properties, press CTRL-D to recall the **Plate Properties** settings.
- You can also specify or edit plates in the **Plates** spreadsheet.
- You can also view and edit plate properties by clicking on a plate.
- You can undo any mistakes by clicking  **Undo** on the Quick Access toolbar. If you want to undo a previous modification, click the 'Undo' icon's arrow and choose the modification you want to undo. Keep in mind that all modifications made after the one you've chosen to undo, will also be undone.

Modify Plates

There are a number of ways to modify plates. You can view and edit the member data in the [Plates spreadsheet](#). You can click a plate to view and edit its properties. You can select multiple plates to graphically modify a possibly large selection of members.

The **Plates** section in the Properties panel lets you modify the properties of plates that already exist in your model. To do so, you select the plates you want to modify, specify the properties you want to change, and the program automatically updates the properties. You can modify plates one at a time by clicking them individually and then changing the properties in the **Properties** panel. See the [Graphic Selection](#) topic for more on selecting.

The parameters shown are the same as those used to define new plates.

Properties	
Plate ^	
Material	gen_Conc4NW v ...
Thickness, in	8
Label	P19
Additional Properties ^	
A Node	N88
B Node	N14
C Node	N107
D Node	N106
Advanced Properties ^	
Plane Stress	<input type="checkbox"/>
Local Axis Angle	NA
A Release	...
B Release	...
C Release	...
D Release	...
Activation	Active v
D Release	...
Activation	Active v

To Modify Plates

Plates can be modified by changing their properties. You can change properties for one or more plates at a time.

To modify Plates:

1. If there is not a model view already open:
 - a. Go to the **View** ribbon.
 - b. Click on the **Open 3D Views** icon to open a new view.

2. Click an individual plate or select multiple, then change the values in the **Properties** panel as desired.

For more details, refer to [Plate Information](#), [Corner Releases](#) and [Inactive Excluded](#).

As you make changes, the program automatically updates the plates.

**Note:**

- You can also modify plates in the **Plates** spreadsheet.
- You can undo any mistakes by clicking  **Undo** on the Quick Access toolbar. If you want to undo a previous modification, click the 'Undo' icon's arrow and choose the modification you want to undo. Keep in mind that all modifications made after the one you've chosen to undo, will also be undone.
- It may not be possible to perfectly align the plate local axes.

Plate Information

Just as with the node and members, you can click any plate to view its properties in the 'Properties' panel. All of the same information that is stored in the **Plates** spreadsheet is displayed for the plate you choose, and can be edited. This is a quick way to view and change plate properties.



Note: For modifying a large selections of plates, using the spreadsheet and graphic editing tools may be a faster method.

Properties	
Plate ^	
Material	gen_Conc4NW v ...
Thickness, in	8
Label	P19
Additional Properties ^	
A Node	N88
B Node	N14
C Node	N107
D Node	N106
Advanced Properties ^	
Plane Stress	<input type="checkbox"/>
Local Axis Angle	NA
A Release	...
B Release	...
C Release	...
D Release	...
Activation	Active v

The following table includes descriptions of the options available for modification.

Properties Panel - Plate Options

Section	Description
Material	Display the material used for the plate, and lets you edit it.
Thickness	Displays the plate thickness (in the current dimension units), and lets you edit them.
Label	Display the current label of the selected plate, and lets you edit that label.
Corner Nodes (A, B, C and D)	Displays the corner node labels, and lets you edit them.
Local Axis Angle	Displays the angle that determines the orientation of the local coordinate system of a plate element in relation to the global coordinate system.

Section	Description
Plane Stress	This box should be checked when you want to change the plate formulation to indicate that the plate is a Plane Stress plate. This option is useful for creating diaphragms that have only in-plane stiffness and will not attract out of plane moments. The default is unchecked which allows for both in-plane and out of plane behavior.
Plate Corner Releases (A, B, C and D)	Displays the corner release for the selected plate, and lets you edit them. See Plate Corner Releases for more details.
Activation	<p>Display the current activation state of the plate, and lets you change it.</p> <p>If the plate is made inactive, you will need to activate the plate from the Plates spreadsheet, or by using the Criteria Select feature to find and select inactive plates.</p> <p>See Inactive and Excluded Plates for more details.</p>

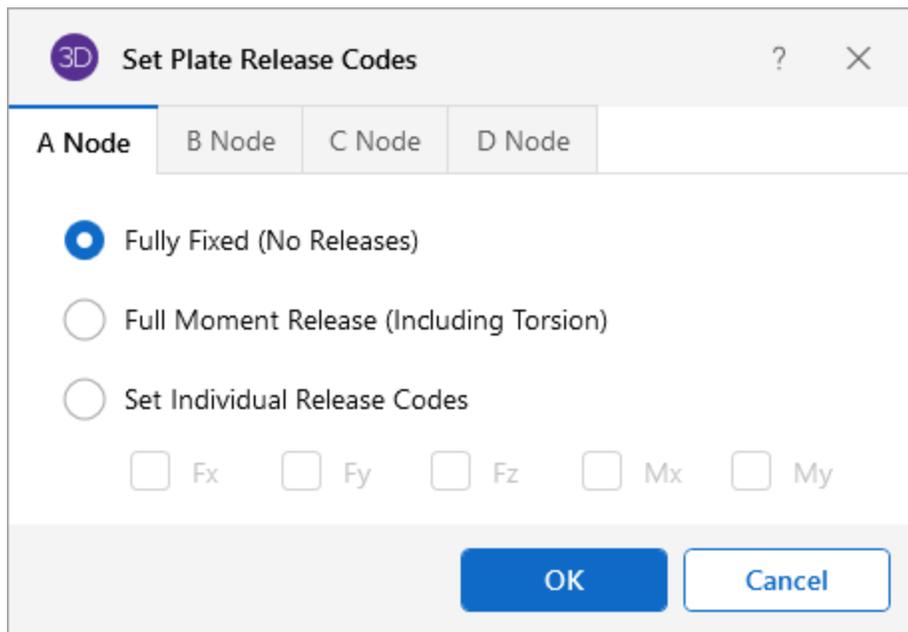
Plate Corner Releases

The **A, B, C, & D Release** fields are used to designate whether the forces and moments at the corners of the plate are considered fixed to or released from the plates's points of attachment (the A, B, C, and D nodes). Each plate has 5 force components at each corner (Fx, Fy, Fz, Mx, and My). Any or all of these force components can be released from the plate's point of attachment. If a force component is released, that force is not transferred between the node and the plate.

To Specify Plate Corner Releases

To specify plate corner releases:

1. Expand the **Advanced Properties** section of the Properties panel, if not already expanded.
2. Click on the **A, B, C, or D Release**  (ellipsis) button for the plate.
3. Specify the condition.



RISA-3D has a special "keyword" release configuration built-in. That is:

PIN => Mx and My (all moments) released (OOOXX)

This keyword entry is included because 99% of the release configurations you'll ever want to define will be "PIN". You can call out the keyword entry by just entering the first letter of the keyword, "p". So if you go to a release field and enter "p", the keyword "PIN" will be filled in automatically.

**Note:**

Alternatively, you can specify the corner condition by directly typing in the A, B, C or D Release box.

Advanced Properties	
Plane Stress	<input type="checkbox"/>
Local Axis Angle	NA
A Release	<input type="text"/> ...
B Release	<input type="text"/> ...
C Release	<input type="text"/> ...
D Release	<input type="text"/> ...
Activation	Active ▾

- To indicate that a force component is released, type an '**X**' for that component in the release box.
- To indicate that a force component is not released, type an '**O**' for that component in the release box.



Note: It's generally more efficient to use the **Graphic Editing** features if you want to change the properties for many plates at once.

Inactive and Excluded Plates

Making an item such as a member or plate inactive allows you to analyze the structure without the item, without having to delete the information that defines it. This leaves data intact so the item may be easily reactivated. This is handy if you want to try a model with and then without certain items, without having to actually delete the data.

- Choosing **Inactive** means the item is not included when the model is solved or plotted.
- Choosing **Excluded** means include the item in the solution, but exclude it from the results list.

So, an item set to **Excluded** is treated like any other plate in the solution and plotting of the model, but the plate will not be listed in the solution results (forces, stresses etc.). This is useful if there are certain items whose results you're not interested in. You don't have to clutter up the results with these items and can concentrate on the items you're most interested in. See [Printing](#) for more limiting printed results.

Rotate Plates

The Rotate Axis options let you perform a clockwise, or counter clockwise rotation of plate local axes so that they can better align with the surrounding plate local axes. It also lets you flip the local z-axis so that it is headed in the other direction. The degree increment of the clockwise or counter clockwise rotation depends on whether the [Local Axis](#) setting in Model Settings is 'Global' or 'Nodal'. When set to Global, rotation is based on whatever degree you set, and the plate's local axes are rotated by that degree. When 'Nodal' is set however, rotation is automatically set at 90°, where each click rotates the plate(s) local axes 90° at a time. Global is the Local Axis default, though you can change it to Nodal in the [Axis](#) tab of the 'Model Settings' window. If you would like to view the rotation of the plate(s) in the 3D View panel, [turn on the display of Local Axis Labels](#).

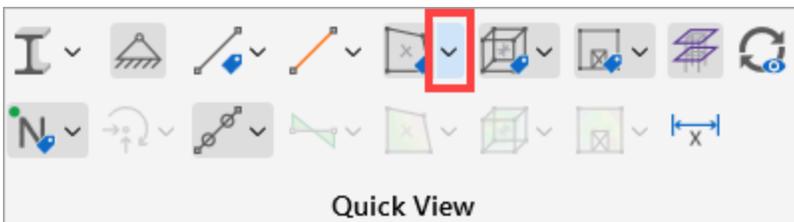


Note: While you can use the Rotate Axis feature to flip the local z-axis so that it is headed in the other direction, you may find the [Flip Axis](#) feature more convenient to use.

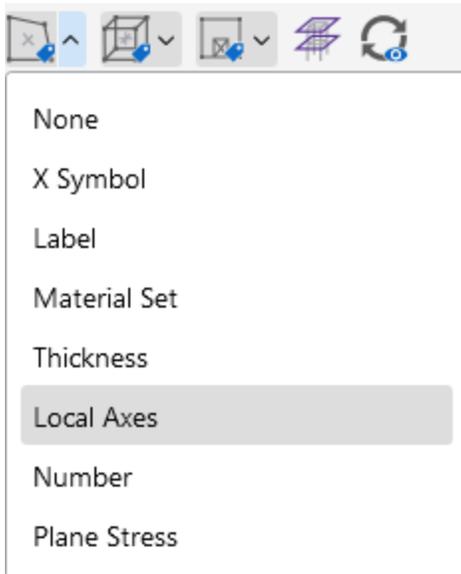
Turn on Display of Local Axis Labels

To turn on the display of local axis labels on the plates:

1. Open to the **Home** ribbon, if not already open.
2. Click the **Plate Labels** down arrow in the Quick View section.

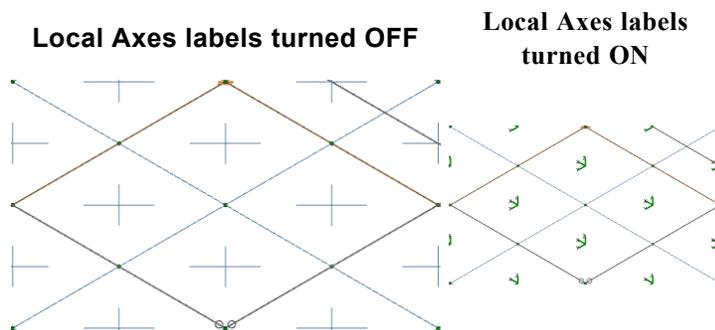


A list of labels opens.



3. Choose **Local Axes** from the list.

The plates display the Local Axes labels. The following images depict the local axes labels turned off and on.

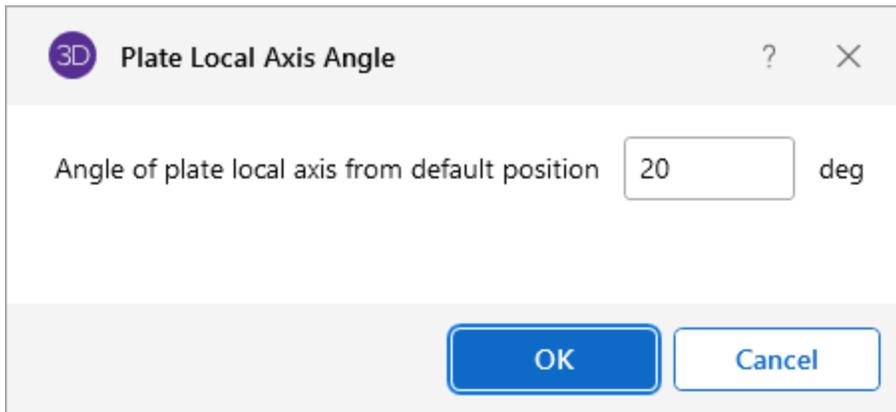


Rotate a Single Plate

To rotate a single Plate:

1. Go to the **Modify** ribbon, if it isn't already open.
2. Click the **Rotate Axis** icon in the Plates section.
3. Choose the **Single** option.

The Plate Local Axis Angle window opens, if 'Global' is set as the 'Plate Local Axis Orientation' in [Model Settings](#). If set to 'Nodal' you can ignore the remaining steps, as the plates are automatically rotated at 90° increments.



4. Enter the degrees you want to move the plate from its default position.
5. Click **OK** to close the window.
6. Click on the plate you want to rotate, for the action to take place.



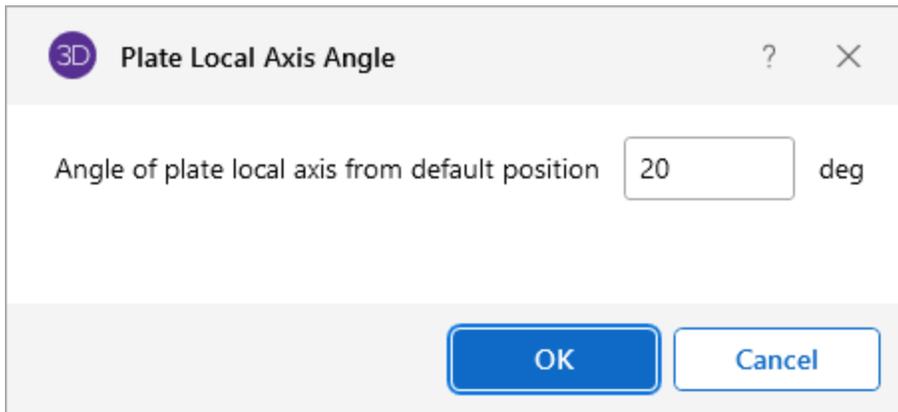
Note: You can hold down the **Ctrl** key while clicking on more plates, to apply the same degree of rotation to each plate. Release the Shift key after all plates you want have been selected and rotated.

Rotate Multiple Plates At Once

To rotate multiple plates the same degree, all at the once:

1. Hold the **Ctrl** key while clicking on each plate you want to rotate.
2. Go to the **Modify** ribbon, if it isn't already open.
3. Click the **Rotate Axis** icon in the Plates section.
4. Choose the **Selected** option.

The Plate Local Axis Angle window opens, if 'Global' is set as the 'Plate Local Axis Orientation' in [Model Settings](#). If set to 'Nodal' you can ignore the remaining steps, as the plates are automatically rotated at a 90° increment.



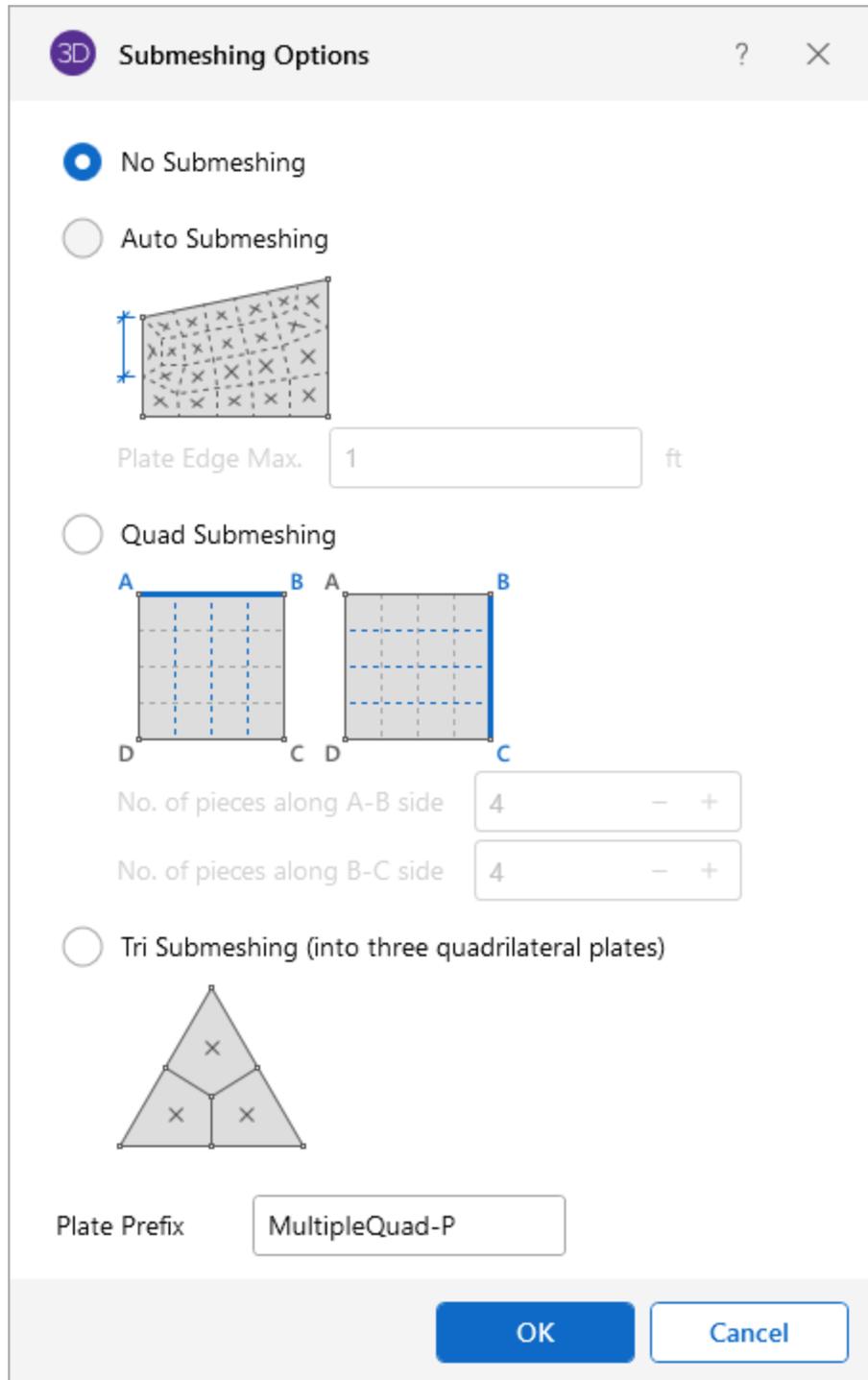
5. Enter the degrees you want to move the plate from its default position.
6. Click **OK** to close the window.

Submesh Plates

You can choose to submesh a plate [during the drawing process](#) or submesh [one or more existing plates](#) after they have been drawn. The second method is useful when you have several plates that you want to submesh with the same properties. When working with submeshing, you can choose from three different submeshing options, further explained in the following sections:

- [Auto Submesh Plates](#)
- [Quadrilateral Plates](#)
- [Triangular Plates](#)

There is a “Submeshing Options” window that provides a visual of each of the submesh options.



This window can be accessed by either clicking the **Meshing**  (ellipsis) button when the 'Plates' mode has been activated (from the 'Home' ribbon), or by clicking the  (ellipsis) button next to any one of the properties in the 'Properties' panel when **Submesh Plates** mode has been activated (from

the 'Modify' ribbon). Note that you can also choose a submesh option and set the same properties in this window as from you can in the Properties panel.

To Submesh a Plate When Drawing It

With this submeshing method, you can both create a plate and submesh it at the same time. When you finish clicking on the last node of the plate, the submeshing option and properties you've chosen, are automatically applied to the plate. Note that this method can be used only when drawing a new plate.

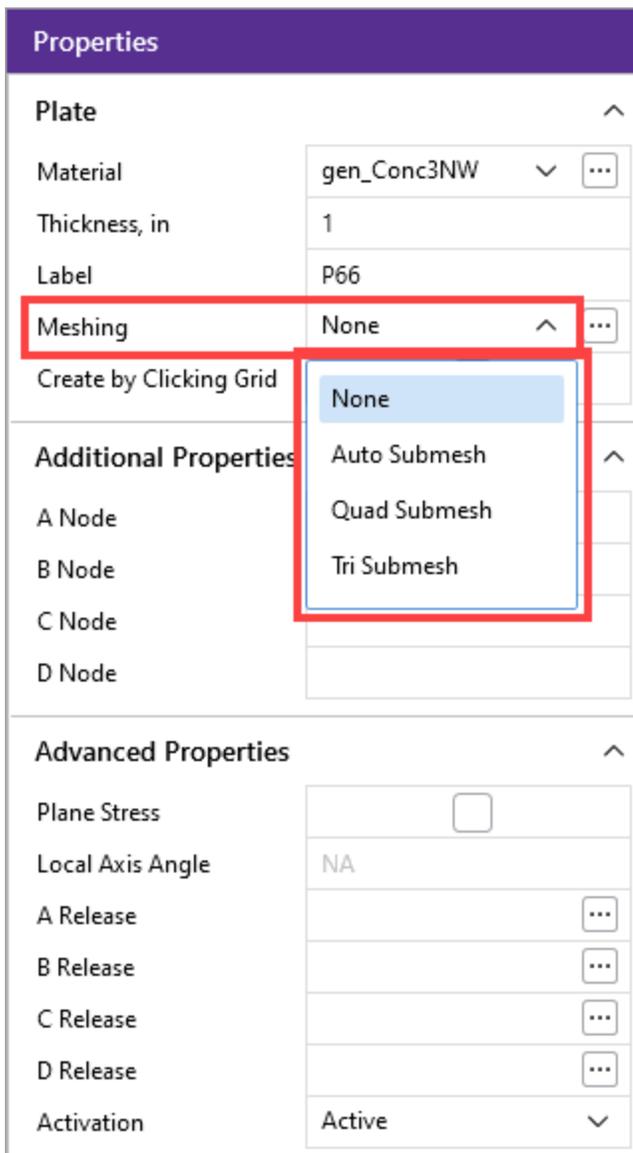
To submesh a plate when you first draw it:

1. Go to the **Home** ribbon.
2. Click the **Plates** icon in the Draw Elements section, to turn on the Plates mode.

The Properties panel displays the properties for Plates, as shown in the following image.

Properties	
Plate ^	
Material	gen_Conc3NW v ...
Thickness, in	1
Label	Tri-P
Meshing	None v ...
Create by Clicking Grid	<input type="checkbox"/>
Additional Properties ^	
A Node	
B Node	
C Node	
D Node	
Advanced Properties ^	
Plane Stress	<input type="checkbox"/>
Local Axis Angle	0
A Release	...
B Release	...
C Release	...
D Release	...
Activation	Active v

3. Enter the plate information (**Material**, **Thickness** and **Label**).
4. Click the **Meshing** arrow and choose the submesh option you want used for the plate.



There are three options for submeshing plates: [Auto Submesh](#), [Quad Submesh](#), and [Tri Submesh](#), as indicated by the three options available from the drop down list in the image above.

Regardless of which option you chose, the Properties panel refreshes and displays the related submeshing properties for that option, directly below Meshing.



Note: Click on the **Meshing** (ellipsis) button (in the Properties panel) to open the Submeshing Options window, which provides a visual of each available submesh option. You can also choose the option and its properties from the window, and then save your choices by clicking the **OK** button. [Click here](#) to view an image and short description of the Submeshing Options window.

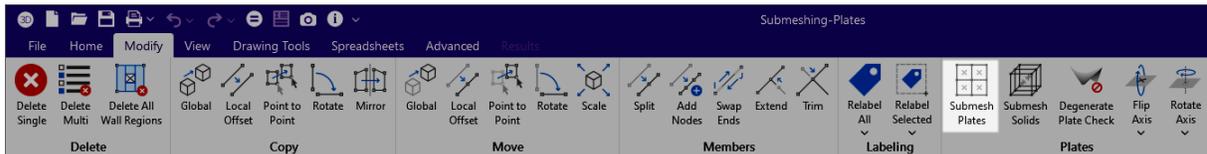
5. Enter the meshing properties you want used.
6. Enter an **Additional Properties** (if available) you want used.
7. Enter any **Advanced Properties** you want used.
8. Draw the plate in the model editing panel.

Immediately after click on the last connecting node, the plate is automatically updated with the submeshing option and settings you've chosen.

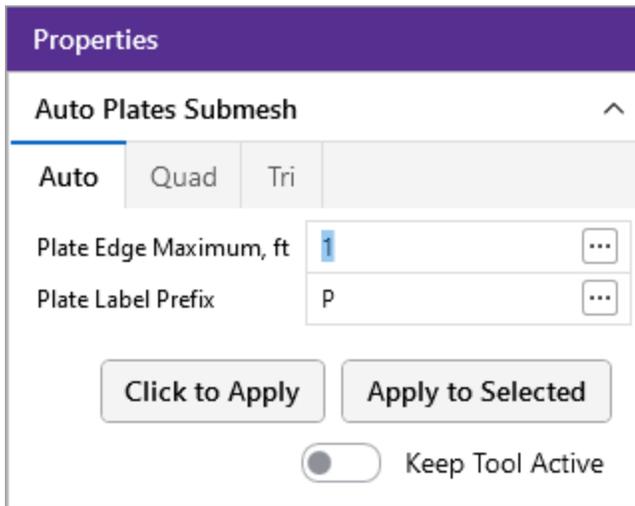
To Submesh Existing Plates

With this submeshing method, you can submesh any plates that already exist in the model and also modify existing submesh into more, thereby creating new submeshing.

1. Select the plate(s) you want to submesh.
To select multiple plates, hold down the **Ctrl** key while clicking on each plate.
2. Go to the **Modify** ribbon.



3. Click the **Submesh Plates** icon.
The Properties panel displays 'Meshing' under the Plate section.



There are three options for submeshing plates: **Auto** (automatic submesh), **Quad** (submesh quadrilateral plates) and **Tri** (submesh triangular plates).

The following sections provide more detail on each of the submesh options.

- [Auto Submesh Plates](#)
- [Quadrilateral Plates](#)
- [Triangular Plates](#)

4. Click on the tab that coincides with the submesh option you want to choose.



Note: Alternatively, you can click the ellipsis button next to any setting found under any of the tabs, to open the “Submeshing Options” window. The Submeshing Options window let you choose the meshing option and set its related properties as well. Clicking the **OK** button saves your choices. [Click here](#) to see an image and short description of the Submeshing Options window.

5. Set the submesh properties for your selected option.

You can submesh the plates one at a time by selecting the **Click to Apply** option and then clicking on the plates you wish to submesh.

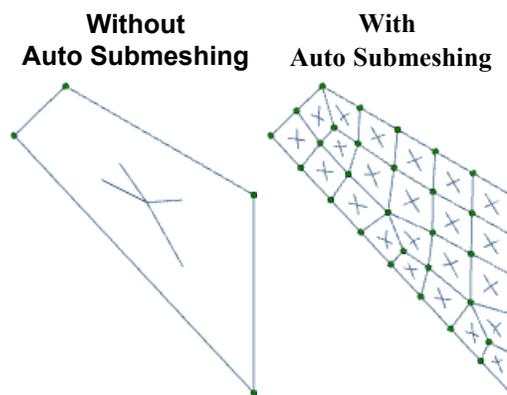
6. Do one of the following:

- Click **Click to Apply** button to submesh the plates one at a time by selecting the **Click to Apply** option and then clicking on the plates you wish to submesh.
- Click **Apply to Selected** button to modify entire selections of plates by selecting the plates and then using the **Apply to Selected** option.

Auto Submesh Plates

The **Auto Submesh** option in the Meshing drop down menu in the **Plates** section of the Properties panel lets you draw a polygon that RISA-3D will automatically submesh into smaller quadrilateral plate elements. Just as with [Drawing Plates](#), the material set and plate thickness of the plates within the mesh may be indicated prior to drawing the polygon. In addition to these parameters, a plate edge maximum can also be provided. Polygons of virtually any size and shape may be drawn provided the drawing lines do not cross. The polygon points must be on the same plane, either one of the three global planes (XY, XZ, or YZ) or co-planar. Polygons may be drawn in either a clockwise or counter-clockwise direction. To complete a polygon, simply double click on the last node/grid intersection, or click on the starting node/grid intersection. Once a polygon is drawn, RISA-3D will create a submesh of quadrilateral plate elements, limited by the edges of the polygon, and of a size corresponding to the plate edge maximum indicated by the user.

The following is an example of a quadrilateral plate without and with Auto submeshing.



Note: Only quadrilateral plate elements are created with the Auto Submesh tool.

When drawing a polygon with the Auto Submesh tool, any existing nodes within the boundary and in the plane of the polygon will be considered control points. These **Control Points** will be considered "fixed" points within the mesh and will dictate the layout of individual plates surrounding them. It is important to note that only currently selected nodes at the time the polygon is drawn will be used as control points.

To AutoMesh a Polygon

To automatically submesh a Polygon when you draw it:

1. If there is not a model view already open:
 - a. Go to the **View** ribbon.
 - b. Click on the **Open 3D Views** icon to open a new view.

2. If you are not drawing between existing nodes, you need to create a drawing grid or define nodes on the **Node Coordinates** spreadsheet.
3. Make sure that all nodes that are to be considered **Control Points** are selected.
4. Click the **Automesh of Plates** icon on the 'Home' ribbon.
5. Set the **Plate Edge Maximum** setting in the Properties panel.
6. Start drawing a polygon by clicking on the nodes or grid points with the left mouse button.
7. To complete the drawing of the polygon, double click the last point or click on the original starting point.
It is at this point that the plate is automatically submeshed.
8. Right click or press the **Esc** key to stop drawing altogether.

**Note:**

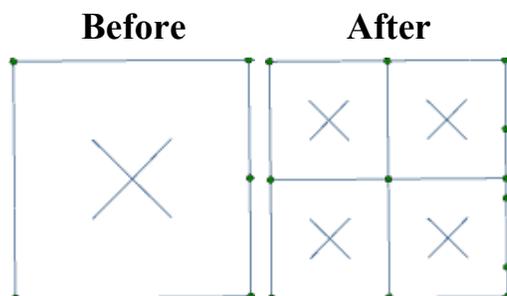
- You can also specify or edit plates in the **Plates** spreadsheet.
- You can also view and edit plate properties by clicking on a plate.
- You can undo any mistakes by clicking the  **Undo** button in the Quick Access toolbar. If you want to undo a previous modification, click the 'Undo' icon's arrow and choose the modification you want to undo. Keep in mind that all modifications made after the one you've chosen to undo, will also be undone.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Plate Mesh**.

Quadrilateral Plates

You can submesh quadrilateral (4 sided) plate elements into a mesh of smaller elements by using the **Quad** submesh option from the 'Meshing' drop down menu prior to drawing a plate or after the fact, by clicking the **Submesh Plates** icon in the **Modify** ribbon (see [To Submesh Existing Plates](#) for complete steps.) The new mesh can be any size up to the program limits for nodes and/or plates. The Quad option is very useful for refining a coarse mesh of elements; just make sure that all adjacent plate elements (elements sharing an edge) maintain connectivity.

The following is an example of a plate before and after applying Quad submeshing to it.

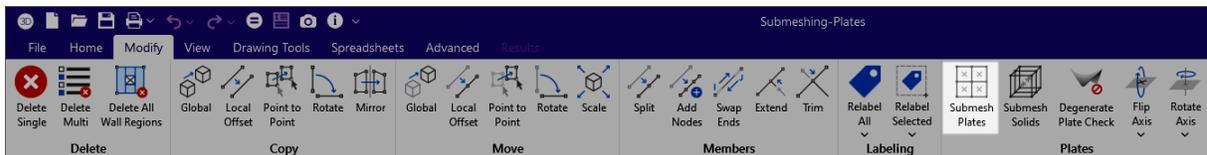


You can define different submesh increments in each direction. The A,B,C and D nodes for each plate are displayed in the plates spreadsheet. The A node is the first node clicked on when you created the plate. The B node is the second and so on. In addition, if the plate local axes is set to Nodal, you can determine which side is which by displaying the plate local axes and realizing that the local x-axis is parallel to the D-C edge of the plate.

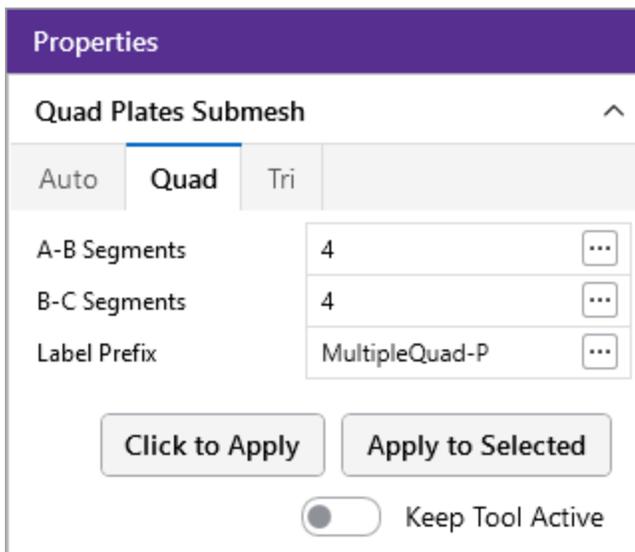
To Submesh Quadrilateral Plates

To submesh Quadrilateral Plates:

1. If there is not a model view already open:
 - a. Go to the **View** ribbon.
 - b. Click on the **Open 3D Views** icon to open a new view.
2. Select the plates you want to sub mesh.
3. Go to the **Modify** ribbon, if it isn't already open.



4. Click the **Submesh Plates** icon.
The Properties panel displays the Plate submeshing options.



5. Click the **Quad** tab to display its property settings.
6. In the **A-B Segments** and **B-C Segments** boxes, specify the number of plates.
7. (Optional) In the **Label Prefix** box, change the prefix if you don't want to use the default

(default is “P”).

8. Click the **Apply to Selected** button.

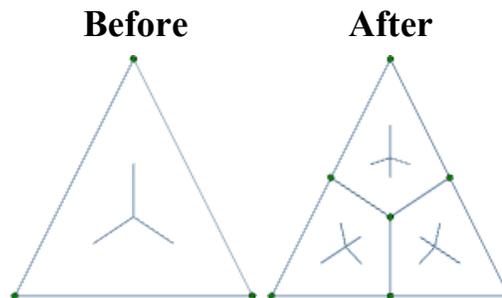


Note:

- To submesh more plates with different parameters, press CTRL-D to recall the **Submesh Plates** action.
- You can undo any mistakes by clicking the  **Undo** button in the Quick Access toolbar. If you want to undo a previous modification, click the ‘Undo’ icon’s arrow and choose the modification you want to undo. Keep in mind that all modifications made after the one you’ve chosen to undo, will also be undone.

Triangular Plates

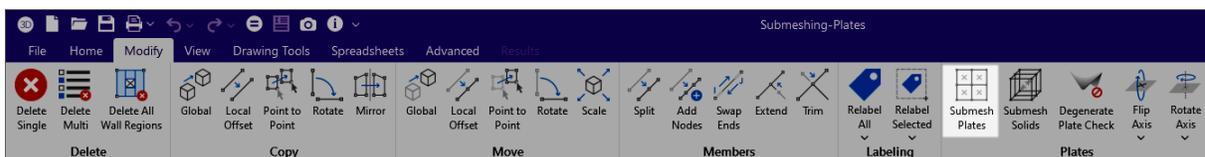
This is used to sub-mesh the selected triangular (3 sided) elements into a mesh of 3 quadrilaterals. This is done by first creating a new node at the center of each selected triangular element and also at the center point along each edge of the triangular element. These new nodes are then used to create three quadrilateral elements that replace the triangular element.



To Submesh Triangular Plates

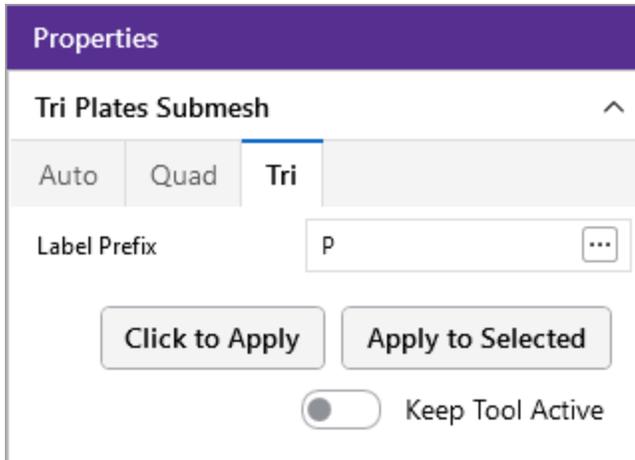
To submesh Triangular Plates:

1. If there is not a model view already open:
 - a. Go to the **View** ribbon.
 - b. Click on the **Open 3D Views** icon to open a new view.
2. Select the plates you want to sub mesh.
3. Go to the **Modify** ribbon if it doesn’t automatically open.



- Click the **Submesh Plates** icon.

The Properties panel displays the Plate submeshing options.



- Click the **Tri** tab to display its property settings.
The only property you can change is the label, which is optional.
- (Optional) In the **Label Prefix** box, change the prefix if you don't want to use the default (default is "P").



Note:

- To submesh more plates with different parameters, press CTRL-D to recall the **Submesh Plates** action.
- You can undo any mistakes by clicking the  **Undo** button. If you want to undo a previous modification, click the 'Undo' icon's arrow and choose the modification you want to undo. Keep in mind that all modifications made after the one you've chosen to undo, will also be undone.

Plates Spreadsheet - Primary Data

The **Plates** spreadsheet records the properties for the plate/shell elements of the model. The spreadsheet can be accessed by selecting **Plates** from the 'Data Entry' section of the 'Explorer' panel.

Plate Primary Data							
Primary	Advanced						
	Label	A Node	B Node	C Node	D Node	Material	Thickness [in]
1	P2	N6	N77	N90	N89	gen_Conc4NW	8
2	P3	N77	N78	N91	N90	gen_Conc4NW	8
3	P4	N78	N31	N92	N91	gen_Conc4NW	8
4	P5	N31	N79	N93	N92	gen_Conc4NW	8

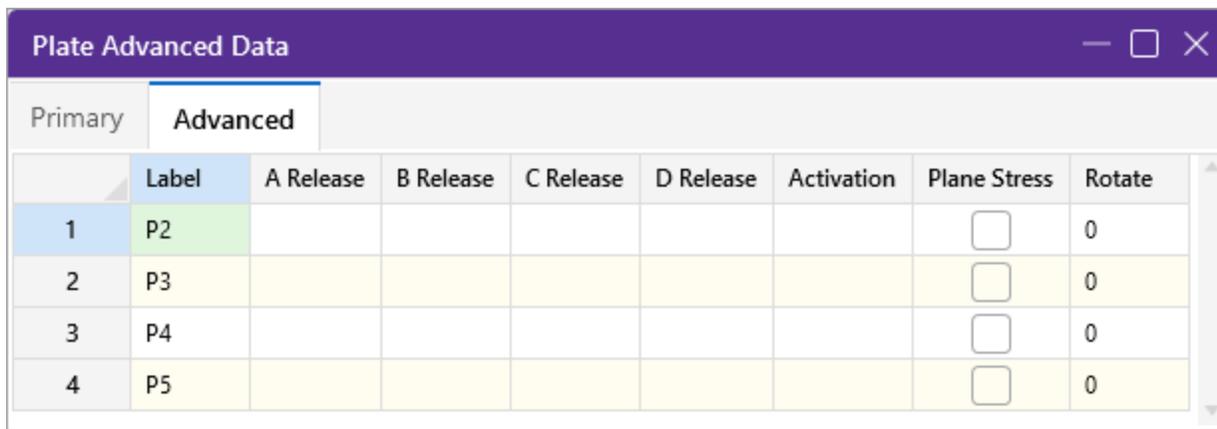
The following table provides descriptions for the columns in the Plate Primary Data spreadsheet.

Plates Spreadsheet - Primary Data

Column(s)	Description
Plate Labels	<p>Displays the label currently assigned to each plate. These labels allow you to refer to a plate by using its label. Note that each label must be unique. If you try to enter the same label more than once, you get an error message.</p> <p>In this spreadsheet, you can assign a different unique label to any or all of the plates using one of the following methods.</p> <ul style="list-style-type: none"> • Spreadsheet: Use this method to change one or more plate labels while in the Plate spreadsheet. Simply double-click on the plate's current Label and type the new label. • Relabel All: Use this method to change the Prefix of all plate labels at once. Go to the 'Modify' ribbon and click the Relabel All icon. • Relabel Selected: Use this method to relabel only specific plates that you have selected in the 3D View panel. Go to the 'Modify' ribbon and click the Relabel Selected icon. <div style="border: 1px solid black; padding: 5px; margin-top: 10px;"> <p> Note: Using the Relabel All or Relabel Selected icons in the 'Modify' ribbon, you can relabel plates at any time.</p> </div>
Plate Nodes (A, B, C and D)	<p>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.</p> <p>The direction and sequence in which you define the nodes determines how the elements local coordinate system is set up. This is discussed in</p>

Column(s)	Description
	the section on Plate Local Axes .
Plate Material	<p>The material set label links the plate with the desired material defined on the Materials spreadsheet.</p> <div style="border: 1px solid black; padding: 5px; background-color: #e6f2ff;"> <p> Note: Plates are always defined with general materials. This is because the other material sets (Hot Rolled, Cold Formed, Wood, and Concrete) are used to designate member code checking specifications. Since plates are only used for analysis, no code checking is provided and the material must be designated as a general material.</p> </div>
Plate Thickness	The thickness field on the Plates spreadsheet is the thickness of the element. This thickness is constant over the entire element.

Plates Spreadsheet - Advanced Data



	Label	A Release	B Release	C Release	D Release	Activation	Plane Stress	Rotate
1	P2						<input type="checkbox"/>	0
2	P3						<input type="checkbox"/>	0
3	P4						<input type="checkbox"/>	0
4	P5						<input type="checkbox"/>	0

The following table provides descriptions for the columns in the Plate Advanced Data spreadsheet.

Plate Spreadsheet - Advanced Data

Column(s)	Description
Plate Labels	<p>Displays the label currently assigned to each plate. These labels allow you to refer to a plate by using its label. Note that each label must be unique. If you try to enter the same label more than once, you get an error message.</p> <p>In this spreadsheet, you can assign a different unique label to any or all of the plates using one of the following methods.</p> <ul style="list-style-type: none"> • Spreadsheet: Use this method to change one or more plate labels while in the Plate spreadsheet. Simply double-click on the plate’s current Label and type the new label.

Column(s)	Description
	<ul style="list-style-type: none"> • Relabel All: Use this method to change the Prefix of all plate labels at once. Go to the 'Modify' ribbon and click the Relabel All icon. • Relabel Selected: Use this method to relabel only specific plates that you have selected in the 3D View panel. Go to the 'Modify' ribbon and click the Relabel Selected icon. <div style="border: 1px solid black; padding: 5px; margin-top: 10px;">  Note: Using the Relabel All or Relabel Selected icons in the 'Modify' ribbon, you can relabel plates at any time. </div>
Plate Nodes (A, B, C and D)	The Plate Corner Releases for nodes A, B, C, and D of each plate may be set in these four data columns. See Plate Corner Releases for more information.
Activation	Allows for plates to be set to 'Active', 'Inactive', or 'Excluded'. These choices can be made by selecting them from the drop down list or by entering an 'I' for 'Inactive' or an 'E' for 'Excluded'. If the cell is left blank the plate will be 'Active' by default. See Inactive and Excluded Plates for more information.
Plane Stress	<p>Allows you to change the plate formulation to a 'Plane Stress' plate, by checking the box. When checked, only in-plane stiffness behavior is allowed. The default is unchecked, which allows for both in-plane and out of plane behavior.</p> <p>This option is useful for creating diaphragms that have only in-plane stiffness and will not attract out of plane shear or moment.</p>
Rotate	<p>Displays the angle that determines the orientation of the local coordinate system of a plate element in relation to the global coordinate system.</p> <p>What appears in this column depends on the option selected as the 'Plate Local Axis Orientation' option in the Model Settings window.</p> <ul style="list-style-type: none"> • If the 'Plate Local Axis Orientation' option is set to "Global" (the default for all new models), a numerical value indicating the angle appears in this column. The value will appear as "0" for brand new models. • If the 'Plate Local Axis Orientation' option is set to "Nodal", "NA" appears in this column, indicating that rotation is not applicable.

Plate Local Axes

The A, B, C, and D nodes are used to define the corners of a quadrilateral element. (To define a 3-node element, just leave the D node 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 direction.

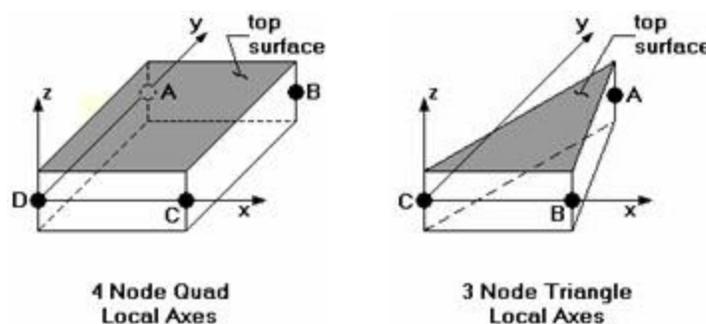
The direction and sequence in which you define the joints determines how the elements local coordinate system is set up. The plate local axes can be set to 'Global' or 'Nodal' in the [Axis](#) settings of the 'Model Settings' window. The following images depict the difference between Global and Nodal plate local axes, which are shown graphically in green.

To view Local Axis labels on plates, as shown in the following images, see [Turn on Display of Local Axis Labels](#) in this chapter.

When the plate local axis is set to 'Global', the local axes of all plates are oriented in the same orthogonal direction, regardless of plate geometry. However, the positive out-of-plane z-axis are dependent on the configuration in which the plate or mesh was drawn. The plate local out-of-plane axis is always the z-axis, but the positive direction of this axis is dependent on the direction in which you define the nodes of the plate. Defining the nodes in a clockwise pattern results in the out-of-plane z-axis pointing in the positive direction according to the global axes. For example, in the global XZ-plane, a plate drawn in the positive clockwise direction results in the plate local out-of-plane z-axis pointing along the global positive Y-axis. Similarly, a counter-clockwise pattern results in the out-of-plane z-axis pointing in the negative global axes direction. Plates do not follow the right-hand rule that is common in three-dimensional vector orientation. For more information on the interpretation of plate results based on local axis orientation, see [Plate Force Results](#).

When the plate local axis is set to Nodal, the local x-axis is defined as positive from the D node towards the C joint 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-joint and perpendicular to the x-axis. Once the x and y axes are defined, the positive local z-axis is found using the right hand rule.

The following diagrams illustrate how the elements local coordinate system is related to the node numbering sequence and direction:



The default plate local axis is set to 'Global'. To change this to 'Nodal', see [Axis](#) settings in the [Model Settings](#) section.

Plate/Shell Element Formulation

The element used is a mixed interpolation 4 node quadrilateral element. By mixed interpolation, we mean that the in-plane and transverse shear strain components are derived independently. This allows the element to be easily simplified into a plane stress element in cases where transverse shear

and bending are not desired. A reference for this element is *Finite Element Procedures*, by K.J. Bathe, Prentice-Hall, 1996. The book also provides many references for papers on the elements convergence and other characteristics. In brief, the element can model isotropic behavior for plane stress, plate bending and out-of-plane transverse shear.

This is accomplished by starting with the Mindlin-Reissner plate assumptions and adding interpolating functions for the out-of-plane transverse shear. This approach is analogous to incorporating shear deformation with flexural effects in beam theory. This results in an element that can be used for thin and thick plate applications. Traditional plate elements do not model out-of-plane transverse shear well (if at all) and cannot be used for thick plate applications. The element is also very insensitive to distortion.



Note: The linear-elastic behavior of the program implicitly means that non-linear (2nd order) behavior is not generally supported. Tension-structures such as fabric structures would fall into this non-linear category. Please see the [Modeling Tips - Modeling a Cable](#) section for some ways to approximate this behavior. This section is applicable to members but the principles illustrated are also applicable to plate modeling as well.

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. In fact, RISA-3D's [Auto Submesh](#) tool will only create quadrilateral plate elements for this very reason. RISA-3D provides a way to convert your triangular plates to quadrilaterals, see [Submeshing Triangular Plates](#).

Orthotropic Behavior

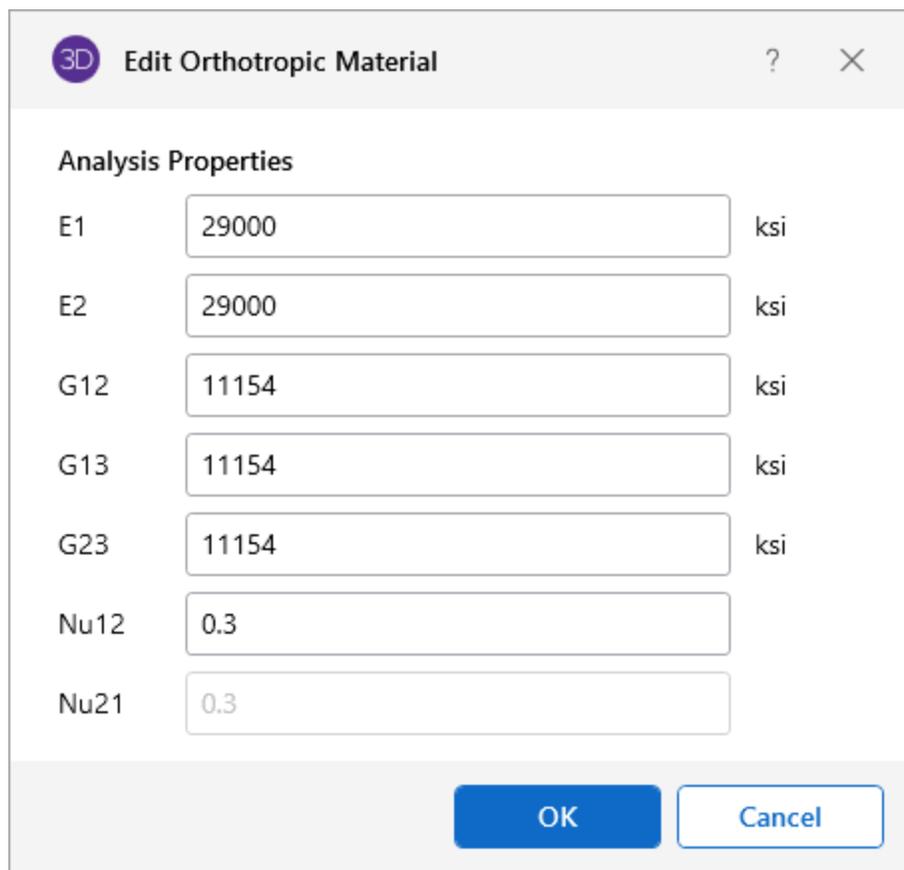
The RISA plate element can use two types of plate methodologies for analysis: Isotropic and Orthotropic. The plate material and methodology can be defined in the Materials spreadsheet under the General tab.

General Materials Properties							
Hot Rolled	Cold Formed	Wood	Concrete	Masonry	Aluminum	Stainless	General
	Label	E [ksi]	G [ksi]	Nu	Therm. Coeff. [10^{-5}F^{-1}]	Density [k/ft ³]	Plate Methodology
1	gen_Conc3NW	3155	1372	0.15	0.6	0.145	Isotropic
2	gen_Conc4NW	3644	1584	0.15	0.6	0.145	Isotropic
3	gen_Conc3LW	2085	906	0.15	0.6	0.11	Isotropic
4	gen_Conc4LW	2408	1047	0.15	0.6	0.11	Isotropic
5	gen_Alum	10600	4077	0.3	1.29	0.173	Isotropic
6	gen_Steel	29000	11154	0.3	0.65	0.49	Isotropic
7	RIGID	1e+6		0.3	0	0	Isotropic
8	gen_Ortho	1e+6		0.3	0.65	0.49	Orthotropic ^ ...

Isotropic
 Orthotropic

By default the material properties shown in the Materials/General spreadsheet are for Isotropic material. Isotropic materials can be used by all element types in the program such as Members, Wall Panels, Solids, and Plates. By using the Isotropic plate methodology, the plate element does allow a limited degree of orthotropic material behavior. Specifically, the In-Plane shearing of the plate will be almost entirely controlled by the G value for the material whereas the direct In-Plane compressive stiffness will be controlled by the E value of the material.

If you toggle the Plate Methodology to Orthotropic, the detailed orthotropic material properties can be defined in the Edit Orthotropic Material window after clicking the  (ellipsis) button in the Plate Methodology cell.



Property	Value	Unit
E1	29000	ksi
E2	29000	ksi
G12	11154	ksi
G13	11154	ksi
G23	11154	ksi
Nu12	0.3	
Nu21	0.3	

The orthotropic material properties can be defined per plate local axes. E1 and E2 correspond to E_x and E_y , which are Young's modulus in plate's local x and y directions. Similarly, G12, G13, and G23 correspond to G_{xy} , G_{xz} , and G_{yz} , which are shear modulus per plate's local axes. Nu12 corresponds to ν_{xy} , which is the Poisson's ratio between plate's local x and y direction.

All plates need to have their local axes orientation set as "Global" instead of "Nodal" for the program to analyze orthotropic behavior. This can be set in the Plate Local Axis Orientation setting under the Global Model Setting.

The following stress-strain relationship is used in the program for orthotropic material:

$$\begin{bmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{bmatrix} = \begin{bmatrix} \frac{E_1}{1-\nu_{12}\nu_{21}} & \frac{\nu_{12}E_2}{1-\nu_{12}\nu_{21}} & 0 \\ \frac{\nu_{12}E_2}{1-\nu_{12}\nu_{21}} & \frac{E_2}{1-\nu_{12}\nu_{21}} & 0 \\ 0 & 0 & \frac{1}{G_{12}} \end{bmatrix} \left(\begin{bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \gamma_{12} \end{bmatrix} + (T-T_0) \begin{bmatrix} \alpha_1 \\ \alpha_2 \\ 0 \end{bmatrix} \right)$$

where $\nu_{21} = \nu_{12} \frac{E_2}{E_1}$

$$\begin{bmatrix} \tau_{13} \\ \tau_{23} \end{bmatrix} = \begin{bmatrix} G_{13} & 0 \\ 0 & G_{23} \end{bmatrix} \begin{bmatrix} \gamma_{13} \\ \gamma_{23} \end{bmatrix}$$



Note: Currently orthotropic plate methodology can only be applied to Plate elements in the program. If an orthotropic material is applied to Members, Wall Panels, or Solids, it will be treated as an isotropic material, with E, G, and Nu taken as E1, G12, and Nu12 from the orthotropic material input. Warning messages will be given after solution to explain this behavior.

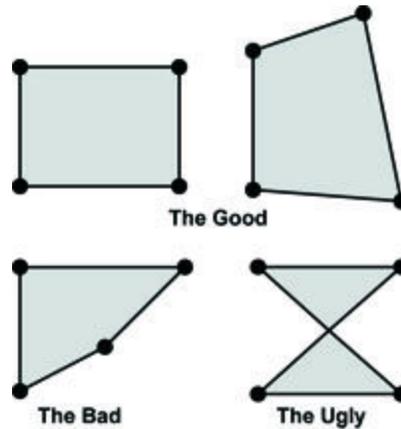
Plate Modeling Tips

A word of caution is in order if you are new to plate modeling. Unlike modeling with beam elements, plate elements require some understanding of finite element behavior to successfully obtain meaningful results. It is easy to build a finite element model using the powerful generators and graphic editing tools. However, without understanding the limitations of the analysis method used, you can end up with an impressive looking but very inaccurate model. Even if you've been engaged in structural engineering for years, modeling with plates is not something most engineers do frequently. It is therefore not realistic to have the expectation that you should be able to perform complicated analysis with plate elements in a short amount of time. Good plate modeling takes time, knowledge of plate and finite element behavior, and experience.

The first tip is to read all the Plate documentation before embarking on an ambitious modeling project. This will save you much aggravation down the road.

Plate Distortion

The finite elements in a model should be as undistorted as possible. See the following figure:



Non-Planar Plates

The plate element formulation used in RISA is not very sensitive to distortion within the plane of the plate itself (see the plate Distortion topic above). However, the plate element formation is particularly sensitive to non-planar plates. This means that it is critical that all 4 nodes that define a quadrilateral plate remain in the same plane. There is an internal tolerance of 0.01 inches built into the plate element formulation for maximum allowable planar distortion. This is appropriate for concrete slabs and shear walls and such. But, extra care should be given to avoiding out-of-plane plates for extremely small and thin plates.

Plate Generation

A fast way to build a new mesh of finite elements is with the generation features. RISA-3D currently provides several generation features to quickly build common structures providing an easy way to create cylinders, cones, grids, radial grids and disks of plates. The best way to see what these features do is to experiment with them. See [Generation](#) to learn more about generation.

Another time-saving method is to draw large elements to represent continuum such as slabs and shear walls, and then use the submesh features mentioned below to refine the mesh.



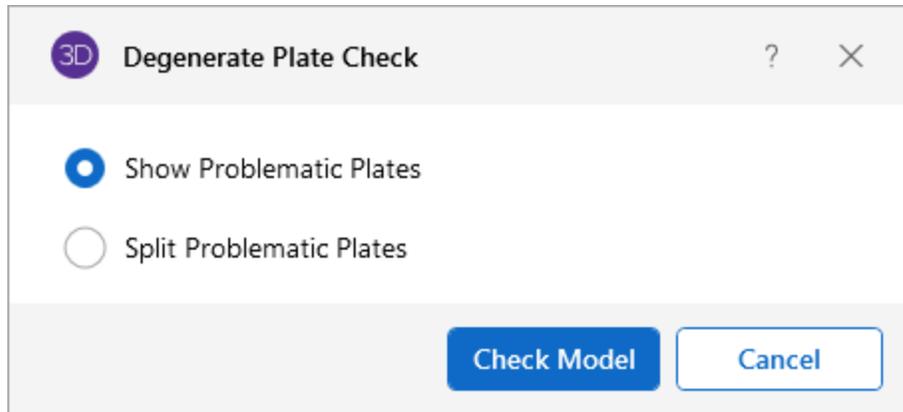
Note: Before sub-meshing, make sure that any adjacent “large” elements connect at their corner nodes. That way, any subsequent sub-mesh operations will produce element meshes that automatically connect at the intermediate nodes.

Automatic Plate Sub-Meshing

What if you’ve already built a model and you now decide that your finite element mesh is too coarse? To submesh elements see [Submeshing Plates](#). Performing a Model Merge after will insure that all the new elements get connected to existing beam elements and that duplicate nodes get merged. See [Model Merge](#) for more information.

Plate Error Checking

The program cannot provide a solution for plates which are poorly shaped or not planar. If any such "degenerate" plates exist, an error is produced during the solution. Click the **Degenerate Plate Check** icon in the 'Plates' section of 'Modify' ribbon to open the Degenerate Plate Check window to fix these issues.



The Degenerate Plate Check utility includes two options that are used to either show problematic plates or split them.

- The **Show Problematic Plates** option provides a model view showing all plates that were non-planar or poorly shaped. A summary report lists the number of degenerate plates, and the warning log is populated with an exact list.
- The **Split Problematic Plates** option splits each non-planar plate into four triangular plates. By definition a triangular plate is always planar, so this will allow the model to solve. The force and stress results for these triangular plates are not accurate, and should not be used for design. However, the stiffness is adequate for force transfer, so this splitting tool provides a convenient method to force a solution in a model where manually fixing the plates to be planar would be too time consuming, and where the plate stresses are not a concern.

Finite Element Basics

While this will not be a comprehensive treatment of plate and finite element fundamentals, a review of certain key basic concepts and terminology will be valuable to the engineer who has not worked with finite elements, or has not had the opportunity to use them recently.

A place to start is with the types of forces or stresses that can occur in a plate. One term that is commonly used is "plane stress". This term is used to describe a state of stress in a plate where all the stresses occur in the plane of the plate. A real world example would be a shear wall with forces applied only in the plane of the wall. The resulting plate forces would be just the normal stresses (F_x , F_y) and the in-plane shear stresses (F_{xy}). There would be no plate moments or out-of-plane shears generated.

Stress vs. Force

It should be pointed out that the results for a plate are always a stress. These stresses are multiplied by the plate thickness and the width or length to obtain a force. Note that this force obtained is just an average value for the plate, since the stress was for a point on the plate and it undoubtedly will vary throughout the plate area. The fact that the stresses vary within a plate is why a good finite element mesh is so critical to obtain accurate results. Stresses tend to vary more around point loads and supports, and less in regions that are far from supports and have a uniform load.

A different example of plate forces would be a horizontal diaphragm that is loaded only in the out-of-plane direction. The plate results would be plate moments, out-of-plane shears, but no membrane (plane stress) stresses. The reason for no membrane stresses is that there was no in-plane loading.

Sign Convention

One other comment on plate results is to point out the convention used for moments in plates. With beams, the 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. For more information, see [Plate Force Results](#).

Why Meshing Is Required

In a nutshell, finite elements tend to work by trying to approximate the correct deflected shape of the real world item being modeled. For example, if we are trying to model a horizontal diaphragm, simply supported on all edges, and loaded out-of-plane, our finite element model must be 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 gauge of “enough” for common structural elements is addressed in the Plate Model Examples section of the Reference Manual.

Distortion

Finite elements are also affected by geometric distortion. The best shape for the 4-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 degrees, and preferably shouldn’t even approach 180 degrees.

Drilling Degree of Freedom

One last item is that the element used by RISA-3D, like other plate/shell elements, cannot accurately model in-plane rotations. I.e., a plate/shell element will not provide resistance to a moment applied about the plate's local z-axis. For example, let's say you have a 4x4 grid of elements, simply supported about the edges, and you apply a node 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 node reactions and the element stresses. See [Applying In-Plane Moments to Plates](#) to learn how to work around this limitation.

Plates/Shells - Results

When the model is solved, there are several groups of results spreadsheets specifically for the plates.

Plate Stress Results

To access the Plate Principal Stresses results spreadsheet:

1. Click on **Results** in the Explorer panel to expand Env (Envelope) results.
2. Click on **Plate Stresses** to open the Plate Stresses spreadsheet.

	LC	Plate Label	Loc	Sigma1[ksi]	Sigma2[ksi]	Tau Max[ksi]	Angle[rad]	Von Mises[ksi]
1	1	P2	T	0.311	-0.366	0.339	0.826	0.587
2			B	0.366	-0.311	0.338	-0.745	0.587
3	1	P3	T	0.143	-0.489	0.316	0.917	0.573
4			B	0.489	-0.143	0.316	-0.654	0.573
5	1	P4	T	0.024	-0.444	0.234	1.063	0.456
6			B	0.444	-0.024	0.234	-0.507	0.457
7	1	P5	T	-0.041	-0.405	0.182	1.24	0.387
8			B	0.405	0.04	0.182	-0.33	0.387
9	1	P6	T	-0.072	-0.346	0.137	1.471	0.316
10			B	0.346	0.072	0.137	-0.1	0.316

The plate stresses are listed for the top and bottom of each active plate. The principal stresses sigma1 (σ_1) and sigma2 (σ_2) are the maximum and minimum normal stresses on the element at the geometric center of the plate. The Tau Max (τ_{max}) stress is the maximum shear stress. The Angle entry is the angle between the element's local x-axis, and the direction of the σ_1 stress (in radians). The Von Mises value is calculated using σ_1 and σ_2 , but not σ_3 which isn't available for a surface (plate/shell) element, so this Von Mises stress does not include any transverse shear forces.

The equations are:

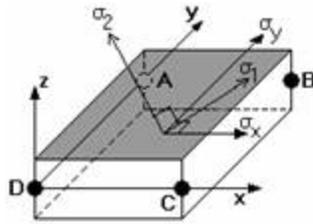


Plate Principal Stresses

$$\sigma_1 = \frac{(\sigma_x + \sigma_y)}{2} + \sqrt{\frac{(\sigma_x - \sigma_y)^2}{4} + \tau_{xy}^2}$$

$$\sigma_2 = \frac{(\sigma_x + \sigma_y)}{2} - \sqrt{\frac{(\sigma_x - \sigma_y)^2}{4} + \tau_{xy}^2}$$

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

$$\text{Von Mises} = \sqrt{\sigma_1^2 - \sigma_1\sigma_2 + \sigma_2^2}$$

The angle, ϕ , is the angle in radians between the maximum normal stress and the local x-axis. The direction of the maximum shear stress, τ_{\max} , is $\pm \pi/4$ radians from the principal stress directions.

The Von Mises stress is a combination of the principal stresses and represents the maximum energy of distortion within the element. This stress can be compared to the tensile yield stress of ductile materials for design purposes. For example, if a steel plate has a tensile yield stress of 36ksi, then a Von Mises stress of 36ksi or higher would indicate yielding of the material at some point in the plate.

The σ_x , σ_y , and σ_{xy} values used to calculate the stresses are a combination of the plate bending and membrane stresses, thus the results are listed for the top and bottom surfaces of the element. The "Top" is the extreme fiber of the element in the positive local z direction, and the "Bottom" is the extreme fiber of the element in the negative local z direction. The membrane stresses are constant through the thickness of the element, while the bending stresses vary through the thickness of the element, very similar to the bending stress distribution in a beam.

For enveloped results the maximum and minimum value at each location is listed. The load combination producing the maximum or minimum is also listed, in the "LC" column.

**Note:**

- See [Spreadsheet Operations](#) to learn how to use **Find**, **Sort** and other options.
- See [Results View Settings](#) to learn how to plot Plate Stress results.

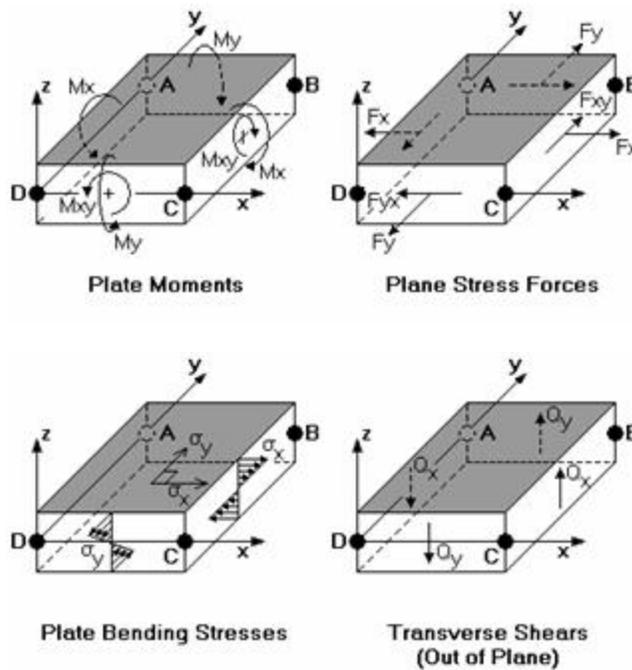
Plate Force Results

To access the Plate Forces spreadsheet:

1. Click on **Results** in the Explorer panel to expand Env (Envelope) results.
2. Click on **Plate Stresses** to open the Plate Stresses spreadsheet.

Plate Forces (per ft) (By Combination)										
	LC	Plate Label	Qx[k]	Qy[k]	Mx[k-ft]	My[k-ft]	Mxy[k-ft]	Fx[k]	Fy[k]	Fxy[k]
1	1	P2	-4.067	4.473	-0.586	-0.003	3.598	-0.01	-0.035	0.018
2	1	P3	-2.359	-0.639	-2.722	-0.968	3.25	0	0	0
3	1	P4	-1.566	0.314	-3.556	-0.921	2.12	0	-0.02	-0.011
4	1	P5	-0.697	0.091	-3.913	-0.844	1.194	0	-0.019	0
5	1	P6	0.125	0.102	-3.66	-0.799	0.289	0	0	0
6	1	P7	0.95	0.133	-2.805	-0.784	-0.582	0	-0.012	0
7	1	P8	1.828	0.145	-1.252	-0.762	-1.368	0	-0.011	0
8	1	P9	2.875	0.136	1.329	-0.816	-2.132	0	0	0
9	1	P10	5.213	3.623	6.808	-0.021	-1.317	0	-0.016	0
10	1	P11	-5.194	3.612	6.742	-0.019	1.253	0	-0.018	0.01
11	1	P12	-2.867	0.124	1.203	-0.8	2.06	0	0	0
12	1	P13	-1.829	0.136	-1.446	-0.731	1.317	0	0	0
13	1	P14	-0.962	0.121	-3.06	-0.73	0.568	0	0	0

The **Plate Forces** are listed for each active plate. Interpretation of output results is perhaps the most challenging aspect in using the plate/shell element. The results for the plates are shown for the geometric center of the plate.



The forces (Q_x and Q_y) are the out-of-plane (also called “transverse”) shears that occur through the thickness of the element. The Q_x shear occurs on the element faces that are perpendicular to the

local x-axis, and the Q_y shear occurs on the element faces that are perpendicular to the local y-axis. Q_x is positive in the z-direction on the element face whose normal vector is in the positive x-direction. This is also the σ_x face. Q_y is positive in the z-direction on the element face whose normal vector is in the positive y-direction. This is also the σ_y face. The total transverse shear on an element face is found by multiplying the given force by the width of the element face.

The plate bending moments (M_x , M_y and M_{xy}) are the plate forces that induce linearly varying bending stresses through the thickness of the element. M_x is the moment that causes stresses in the positive x-direction on the top of the element. Likewise, M_y is the moment that causes stresses in the positive y-direction on the top of the element. M_x can then be thought of as occurring on element faces that are perpendicular to the local x-axis, and the M_y moment occurs on faces that are perpendicular to the local y axis. To calculate the total M_x or M_y on the face of an element, multiply the given value by the length of the element that is parallel to the axis of the moment. For example, looking at the 'Plate Moments' figure above, the total M_x moment could be obtained by multiplying the given M_x force by the length of side BC (the distance from joint B to joint C). The total M_y force can be calculated in the same way by instead using the length of side DC.

The M_{xy} moment is the out-of-plane twist or warp in the element. This moment can be added to the M_x or M_y moment to obtain the 'total' M_x or M_y moment in the element for design purposes. This direct addition is valid since on either the top or bottom surface, the bending stresses from M_{xy} will be going in the same direction as the M_x and M_y moments.

**Note:**

- For the placement of concrete reinforcement, it is helpful to realize that laying reinforcement parallel to the local x-axis will resist the M_x moment.
- A positive M_x or M_y moment will put the top fiber of the plate in tension.

The plane stress forces (F_x , F_y and F_{xy}) are those forces that occur in the plane of the plate. These forces, which are also called “membrane” forces, are constant through the thickness of the element. F_x and F_y are the normal forces that occur respectively in the direction of the local plate x and y-axes, positive values indicating tension. These forces are reported as a force/unit length. To get the total force on an element, you would need to multiply the given value by the length of the element that is perpendicular to the normal force. For example, looking at the 'Plane Stress Forces' figure, the total F_x force could be obtained by multiplying the given F_x force by the length of side BC (the distance from joint B to joint C).

The F_{xy} force is the in-plane shear force that occurs along the side of the element. The subscript 'xy' indicates that the shear occurs on the face of the element that is perpendicular to the x-axis and is pointing in the y-direction. F_{yx} is the complementary shear force, where the subscript 'yx' indicates that the shear occurs on the face of the element that is perpendicular to the y-axis and is pointing in the x-direction. RISA-3D only gives values for F_{xy} because F_{xy} and F_{yx} are numerically equal. The total in-plane shear can be obtained by multiplying the given force value by the length of the element that is parallel to the shear force. For example, when looking at the 'Plane Stress Forces' figure, the total F_{xy} force which is parallel to the local y-axis could be obtained by multiplying the given F_{xy} force by the length of side BC.

Note that the plate bending (Q_x , Q_y , M_x , M_y , M_{xy}) and membrane (F_x , F_y , F_{xy}) results are forces per unit length. For example, a rectangular element with a B to C length of 10 feet showing a F_x force of 20K would have a total normal force on the B-C face of the element of 20K (per foot) times 10 feet, or 200K.

For enveloped results the maximum and minimum value is listed. The load combination producing the maximum or minimum is also listed, in the "LC" column.

**Note:**

- See [Spreadsheet Operations](#) to learn how to use **Find**, **Sort** and other options.
- See [Results View Settings](#) to learn how to plot Plate Force results.

Plate Corner Force Results

To access the Plate Corner Forces spreadsheet:

1. Click on **Results** in the Explorer panel to expand Env (Envelope) results.
2. Click on **Plate Stresses** to open the Plate Stresses spreadsheet.

	LC	Plate Label	Node	X[k]	Y[k]	Z[k]	MX[k-ft]	MY[k-ft]	MZ[k-ft]
1	1	P2	N6	-0.031	12.272	-0.072	3.33	0	-1.32
2			N77	-0.006	-3.028	0.001	5.609	0	-2.565
3			N90	0.026	-4.807	0.036	6.314	0	-6.741
4			N89	0.01	-3.839	0.035	2.637	0	-5.643
5	1	P3	N77	0.001	2.493	0.003	-5.299	0	2.903
6			N78	0	-3.472	0.003	2.085	0	-3.178
7			N91	-0.001	-0.947	-0.003	3.95	0	-6.984
8			N90	0	2.524	-0.003	-3.291	0	-2.177
9	1	P4	N78	0.002	2.845	0.002	-2.686	0	3.633
10			N31	0.021	-1.918	-0.041	1.474	0	-3.981
11			N92	-0.006	-0.915	0.019	3.219	0	-6.261
12			N91	-0.016	0.585	0.02	-0.749	0	0.347
13	1	P5	N31	-0.004	1.306	-0.036	-1.976	0	4.275

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, in the same way that joint reactions are reported. At any given joint, the corner forces for all plates connected to that joint should sum to zero (a requirement of equilibrium), assuming no members or boundary conditions are also present at the joint.

As an example of how to use these corner forces, you can obtain the total shear at a given level in a shear wall by adding the proper corner forces for the plates at that level. See [Plate Modeling Examples](#) to learn how to use the plate corner forces to get shear wall story shears and moments, as well as slab moments and shears.

For enveloped results the maximum and minimum value is listed. The load combination producing the maximum or minimum is also listed, in the "lc" column.

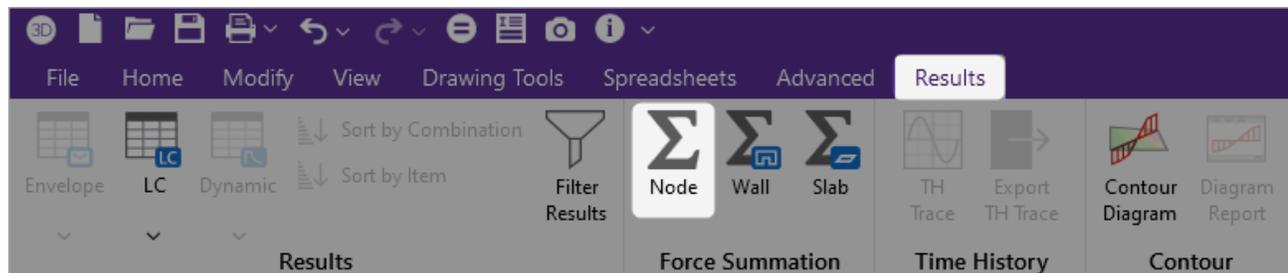


Note:

- See [Spreadsheet Operations](#) to learn how to use **Find, Sort, Label Marked Lines in Current View>Show Selected Lines in Current View** and other options.
- See [Results View Settings](#) to learn how to plot Plate Corner Force results.

Plates/Shells - Design Tools

Internal Force Summation Tool



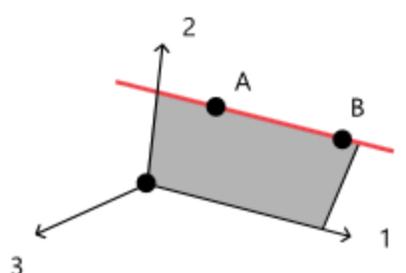
This tool may be activated by clicking the **Node** icon in the 'Force Summation' section of the 'Results' ribbon. The tool is only available from within a model view that has active results for a single load combination or a batch solution. This tool is not available for an envelope solution.

The Internal Force Summation Tool can be used to quickly come up with information on story shears in a building, or internal moments in a elevated or on grade slab.

When initiated, the tool requires the user to select two points (A and B). These points will be used to define the plane in which the internal force summation will be performed. That defines a "cutting plane" perpendicular to the screen.

Clicking on a third point locks the cutting plane to that exact location and provides a detailed summary of the internal forces at that location. These summary results are summarized below:

3D Internal Force Summation
? X



- The origin is the geometric center.
- The 1 axis is defined by the user selected points A and B.
- The 3 axis is normal to the model view.
- The shaded area is in the plane of the computer screen.

Resultant Forces/Moments

Load Combination: LC 1: IBC 16-1

F1	-226.321 k	M1	-584.901 k-ft
F2	469.977 k	M2	5846.772 k-ft
F3	322.936 k	M3	-7771.558 k-ft

Location Information

Geometric Center of Selected Items

X	24.317 ft	Y	12.505 ft	Z	1.231 ft
---	-----------	---	-----------	---	----------

Vector Normal to Cutting Plane

X	0 ft	Y	0.707107 ft	Z	-0.707107 ft
---	------	---	-------------	---	--------------

Print

OK

The Internal Force Summation report is separated into three regions. The first region gives the resultant forces and moments. The second region gives information needed to locate the reference plane and origin of local axes. The third region gives a basic reference for the orientation of the local axes and forces.

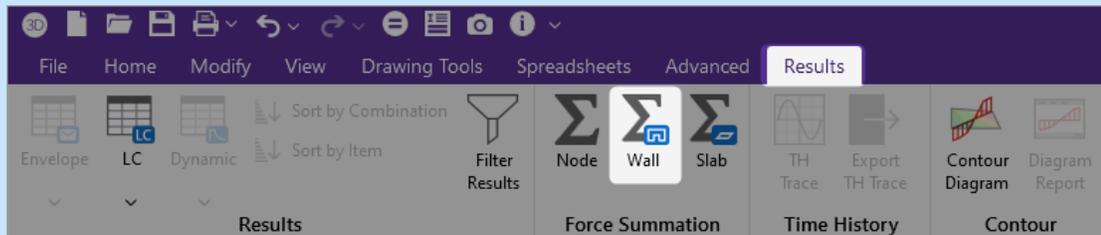
Since the **Forces** and **Moments** are summed only for the selected members, this tool can be easily used to determine the overall story forces in a shear wall or moment frame (if the whole model is selected), as well as the forces in an individual bent or pier (if only a portion of the model is selected).

The **Geometric Center of the Selected Items** defines the origin at which the forces (F1, F2, and F3) are reported. This also corresponds to the point used to define the moments (M1, M2 and M3). These forces and moment are all given with respect to the local 1, 2, 3 axes. The local 3 axis is always perpendicular to the current model view. The local 1 axis is always defined parallel to the points A and B which were selected by the user. The 2 axis is then defined by the right hand rule.

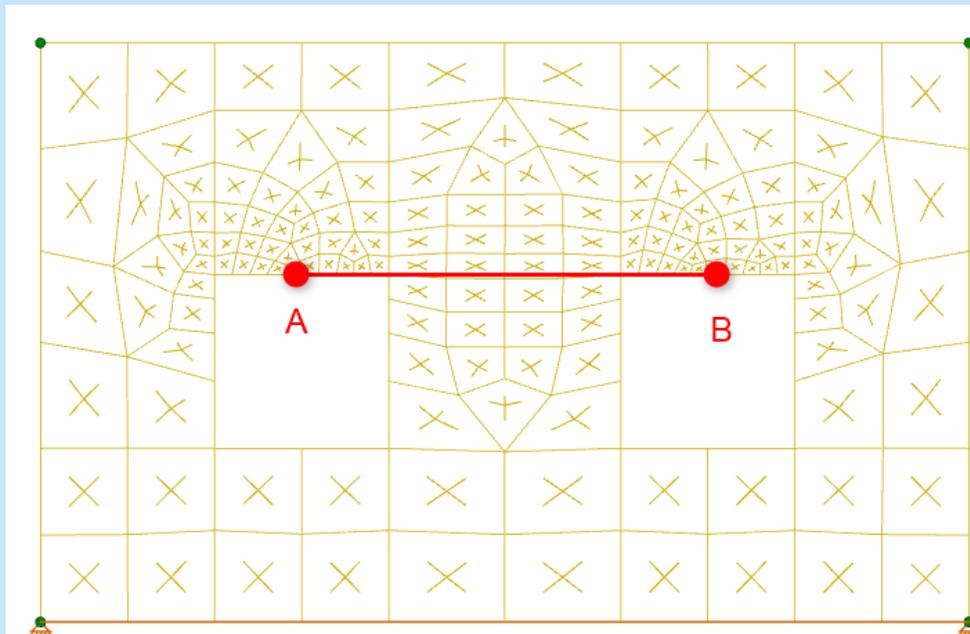
The **Vector Normal to the Cutting Plane** is used to define the plane where the internal force summation was performed. This was determined by the user selected points A and B.

**Note:**

- For wall panel specific internal force summation, click the **Wall** icon in the 'Force Summation' section of the 'Results' ribbon. This icon is intended to be used by clicking two points in the same plane as a wall panel (or a group of co-planar wall panels).

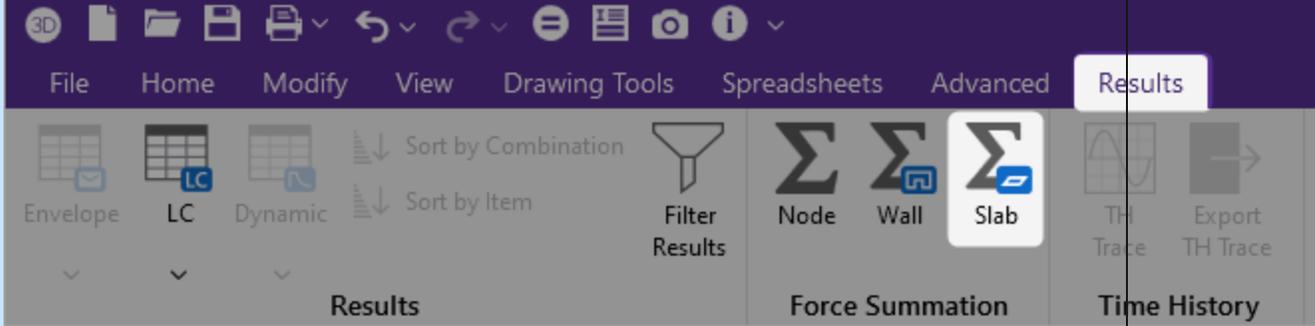


- Masonry wall panels with the "Transfer Load" option selected in the Wall Design Rules may show different values between the IFST and the Region Detail Report if you cut across a region next to an opening. This is because the "Transfer Load" load option will transfer in plane loads from regions above and below openings into the regions adjacent to the openings for wall panel design. This behavior is also true for concrete walls which have the "Transfer In" and "Transfer Out" options available in the Wall Design Rules. In order to see comparable results from the plates using IFST Wall, increase the cut length to include the region width plus half the distance across each opening.



When "Transfer Forces" is checked in the Wall Design Rules

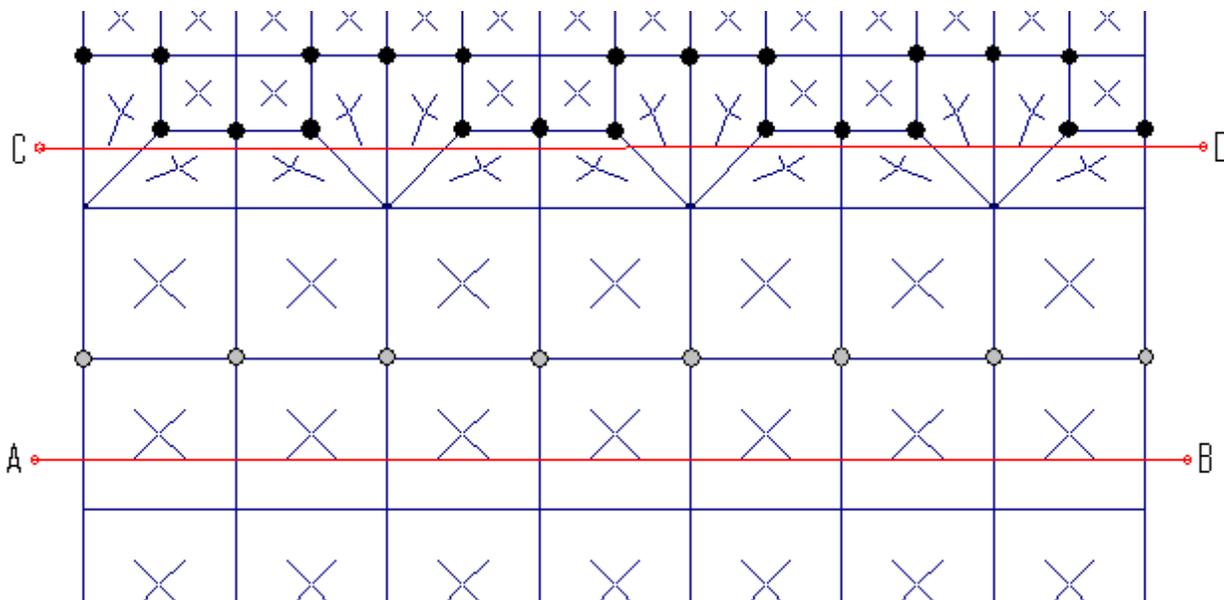

 • For semi-rigid slab specific internal force summation, press the **Slab Force Summation** button on the **Results** tab. This button is intended to be used by clicking two points in the same plane as a slab (or a group of co-planar slabs). Semi-rigid slabs are elements found in RISA-3D in a combined RISAFloor-RISA-3D model with slab floors.



For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Story Shears** or **IFST**.

Calculation and Theory

When cutting through a plate, the internal force summation tool uses the plate corner forces to derive the force in the cutting plane. When cutting through the interior of a plate, any corner node that is "above" the cutting plane is included in the summation. Moments are then interpreted based on the location of the forces compared to the centroid of the cutting plane. See the figure below:



Consider the cutting plane A-B defined above. This was created by clicking from left to right on the page (from A to B). Therefore, there are 8 nodes above cutting plane A-B. These are highlighted in a gray color and are the nodes whose corner forces will be used to create the base value of the cutting plane force. Because the cutting plane is below the row of nodes, and because there may be applied surface or self weight loading at the plates, some interpolation must be used between the values above the plane and the values below the plane. This consists of essentially a linear interpolation.

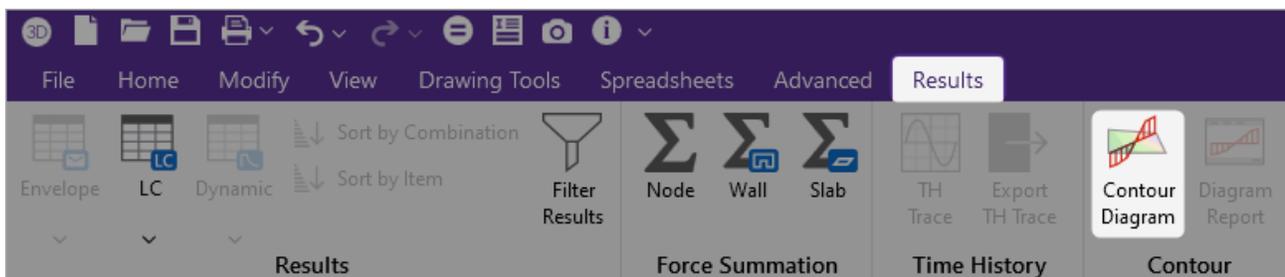
For cutting plan C-D there are 22 nodes above the cutting plane that will be used to determine the base value of the cutting plane force.



Note:

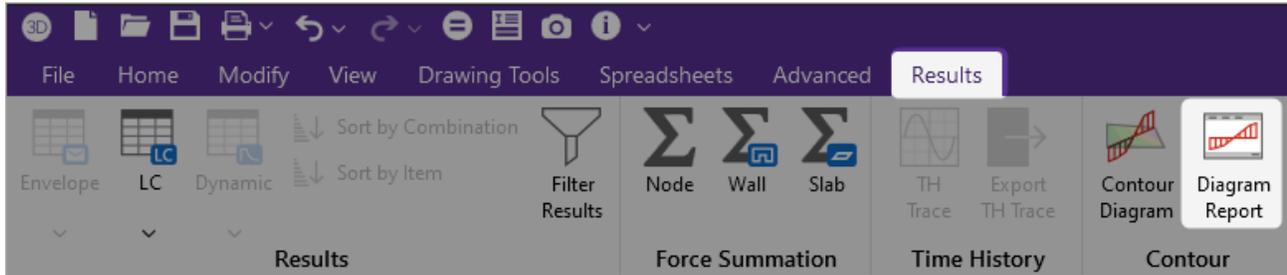
- This tool only computes forces for elements that are selected in your model. In the same sense, it will sum up forces for **everything that is selected** that is in the cutting plane. You may use the selection tools to unselect portions of the model for which you do not want force results for.
- Because these forces are reported with respect to the local 1,2,3 axes of the cutting plane it is NOT recommended that this tool be used from an isometric view of the structure. It will function best in a pure plan or a pure elevation view.
- The results for the internal force summation tool are always given for the currently displayed results. The currently displayed results show up in the “Results View Settings” window accessible by clicking the **Results** icon in the “View Settings” section of the ‘View’ ribbon. If the model has not been run and there are no results, then the tool is not available. Similarly, the tool is not available with Envelope Results.

Contour Display Details



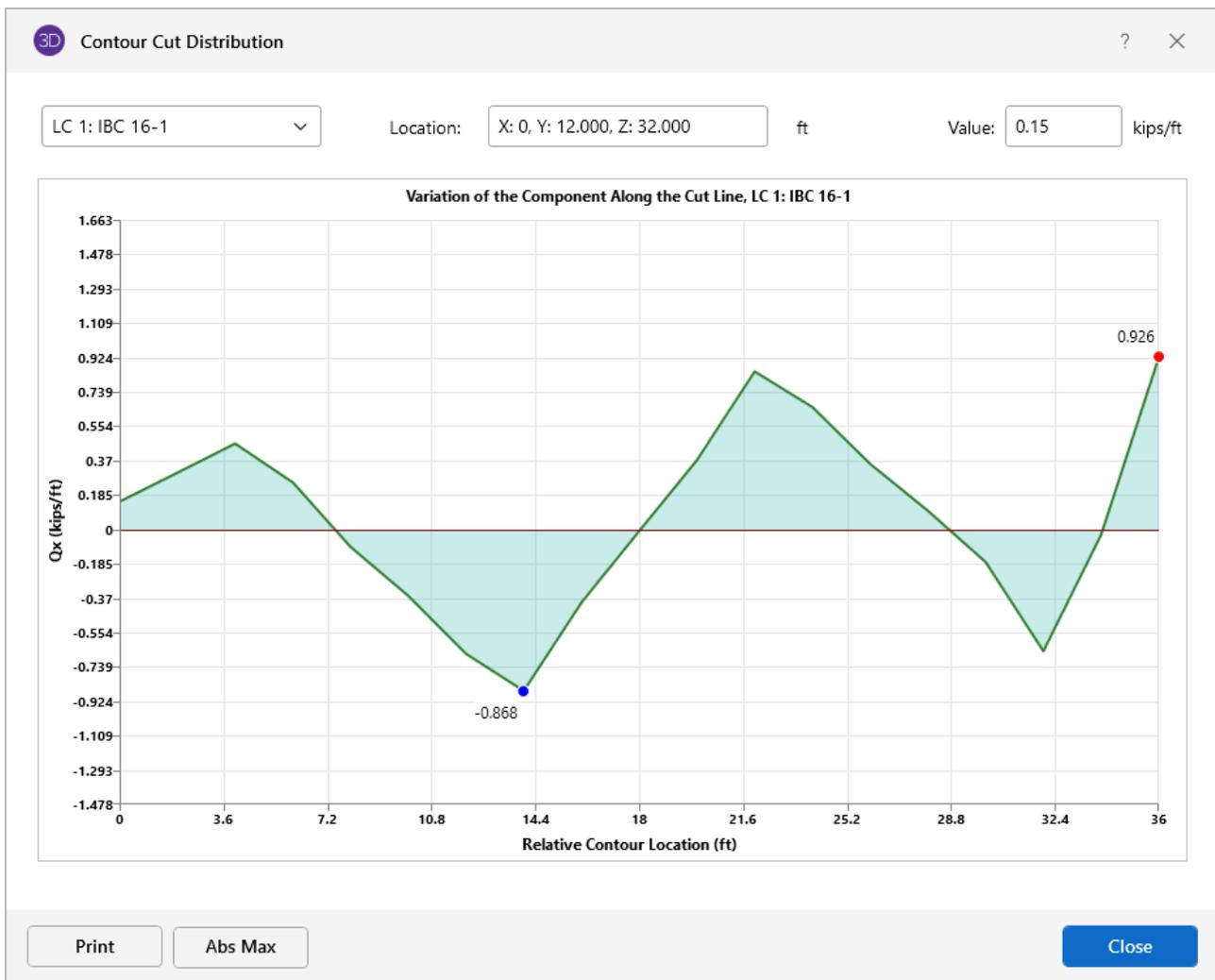
This tool is activated by clicking the **Contour Diagram** icon on the ‘Results’ ribbon. The tool is only available from within a model view that is currently plotting a plate contour.

This tool can also be used for curved plate models such as tanks and vessels. The integral of the force diagram is displayed in the Status Bar at the bottom of the screen. This may be useful for summing up shear forces in a wall or lintel.



Click the **Diagram Report** icon on the 'Results' ribbon to turn a visual plate contour into a more traditional shear or moment diagram. The tool merely transforms the visual contour display into actual numerical that you can view. In the image below, the contour display shows the numerical values associated with the contour at the cutting line.

When a Contour Display Detail is shown, the right click menu gives access to a Detailed Diagram of the contour cut as shown below:



This detailed diagram shows you all the results for every section along the length of the diagram. This information may even be copied to the clipboard for use in a spreadsheet program. This can be done by selecting **Copy Data** from the right click menu.



Note: Because this display is just a currently displayed plate contours, the units for the detailed diagrams are always the same as for the displayed contours.

Plates/Shells - Modeling Examples

The finite element method is an approximating method. This means that the results will never be exact and can be made better and better, within reason, by sub-dividing the element mesh. Accurate results are dependent on the modeling of the problem.

The following pages are studies of finite element mesh fineness and its relationship to accurate stress and deflection results. These studies are meant to be an aide to help you select appropriate mesh fineness for a structure you are trying to model. These studies will also answer the "why" many people ask when told they must use a "mesh" of elements to model a structural item (such as a shear wall) instead of using one giant element. Obviously these studies only give an overview of some basic elements and the engineer must be the final judge as to whether a specific finite element model is a good reflection of the "real" structure.

Shear Wall Modeling

Theoretical Deflection of Shear Wall with Point Load

	<p>Shear Wall Properties</p> <p>L = 240 in</p> <p>Area = 1440 in²</p> <p>B = 12 in</p> <p>H = 120 in.</p> <p>E = 4000 ksi</p> <p>$\nu = 0.30$</p> <p>G = 1538.5 ksi</p> <p>P = 15,000 kip</p>	<p>$I = BH^3/12$</p> <p>$= 12(120)^3 / 12$</p> <p>$= 1,728,000 \text{ in}^4$</p> <p>$\Delta = PL^3/3EI +$</p> <p>$1.2 PL / AG$</p> <p>$= 11.95 \text{ in}$</p> <p>$K = P / \Delta$</p> <p>$= 15,000\text{k} / 11.95\text{in}$</p> <p>$= 1255 \text{ kips/in}$</p>
--	---	--

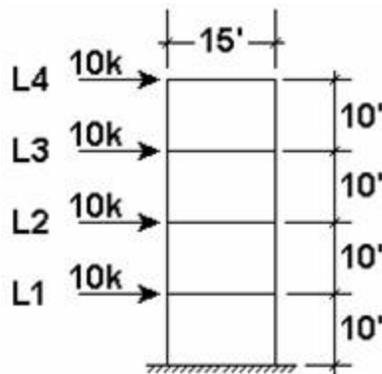
Stiffness as a Function of Mesh Fineness

Element Mesh	1x1	1x2	2x2	2x4	4x8
Deflection (in.)	4.54	8.07	8.26	10.43	11.2
Error	62%	33%	31%	13%	6%
Stiffness K (kip s/in)	3304.0	1858.7	1816.0	1438.2	1328.6



Note: On Methodology - Since the theoretical solution is based on an assumption that plane sections shall remain plane after deformation, the last model (4x8 mesh) had very stiff axial members included at the 2nd, 4th, 6th and top level across the width of the wall. This prevented horizontal differential joint movement and allows for a more meaningful comparison with the theoretical solution.

Shear Wall Design Forces



Floor	Shear	Moment	Elements
4	9.99k	100.05k-ft	P49-P52
3	20k	300.08k-ft	P33-P36
2	30k	600k-ft	P17-P20
1	40k	999.98k-ft	P1-P4

Shown above are the analysis results of a 4-story shear wall. This example is for a straight shear wall, however the method and results are valid for box, channel, or any other shear wall shapes.

The RISA-3D files that were used to obtain these results are included as “4X1WALL.R3D” and “4X4WALL.R3D”. The 10 kip story loads were applied uniformly across each story. This was done to more accurately model loads being applied to the wall from a rigid or semi-rigid floor.

The story shears at each level were calculated as the sum of the FX corner forces. The story moments at each level are calculated from the FY corner forces as shown below:

$$M_i = (F_{youter\ node} * 15ft) + (F_{yinner\ node} * 7.5ft)$$

Story Shears - Hand Calculation

The story shears were calculated as shown below from the corner forces. See the screen shot close up of the FX corner forces on the next page.

Story	Sum FX Corner Forcers at Story Level (k)	Shear	Moment
4	$[0.462 + 0.786 + 1.77 + 1.98] * 2 = 9.996$	9.9k	P49-P52
3	$[1.2 + 1.65 + 3.45 + 3.7] * 2 = 20.000$	20k	P33-P36
2	$[1.9 + 2.5 + 5.14 + 5.46] * 2 = 30.000$	30k	P17-P20
1	$[11.76 - 0.32 + 6.09 + 2.47] * 2 = 40.000$	40k	P1-P4

Level 4 Global FX Corner Forces

1.3	0.52	1.21	3	20.5	1.2
P61	P62	P63	P64		
-7	-1.1	-1.5	-1.8	-1.8	-1.5
0.7	0.8	1.8	1.8	1.8	0.7
P57	P58	P59	P60		
-7	-0.1	-1.7	-1.8	-1.7	-7
0.7	0.8	1.8	1.8	1.8	0.7
P53	P54	P55	P56		
-6	-0.1	-1.7	-1.9	-1.9	-6
0.6	0.6	1.9	1.9	1.9	0.6
P49	P50	P51	P52		
-5	-0.1	-1.8	-2	-1.8	-5
1.7	1.7	3.2	3.2	3.3	1.7
P45	P46	P47	P48		
-1.5	-1.9	-3.2	-3.4	-3.2	-1.5
1.5	1.7	3.4	3.4	3.4	1.5

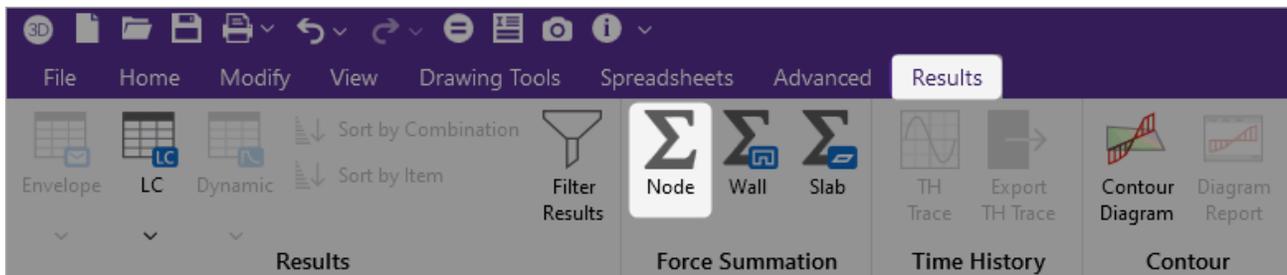
For the graphical display of the corner forces, there are 4 corner forces shown for each plate. This is similar to a beam element which has 2 member end forces. To get the story shear at any line, just sum up all the FX corner forces along the line.

**Level 4 Global FY
Corner Forces**

0	1.1	1.1	1.1	1.1	0
P61	0.2	P62	P63	P64	0
-1.2	1.6	1.4	0.6	1.6	-1.2
1.2	-7.1	P57	P58	P59	P60
-2.1	1.7	1.1	1.7	2.1	2.1
2.1	2.5	1.6	0.1	2.5	-2.1
P53	P54	P55	P56		
-3.1	-1.5	2.3	0.7	2.3	3.1
3.1	3.3	2.2	-5.3	3.3	-3.1
P49	P50	P51	P52		
-3.9	-2.5	0.6	32.5	3.9	3.9
3.9	5.2	3.1	3.1	-3.5	-3.9
P45	P46	P47	P48		
-6.2	-2.9	4.6	1.2	1.2	4.6
6.2	6.7	3.5	3.5	-9.6	-6.2

To get the story moment at any line, just sum the moments obtained by multiplying the Fy corner forces along a line, times the moment arm such as the distance of each Fy force to the center of the wall.

Story Shears - Summation Tool



A quicker way to calculate story forces is to use the [Internal Force Summation](#) tool to have the program automatically calculate the global forces that pass through a given elevation of the shear wall. To do this, just click on the **Node Force Summation** icon in the **Results** tab whenever you have a set of valid, non-envelope solution results. This tool will sum the forces for the displayed items at the desired elevation and will report them back to the user in terms of the Global X, Y and Z directions.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Story Shears**.

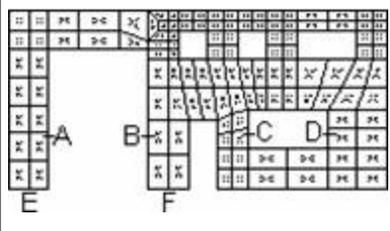
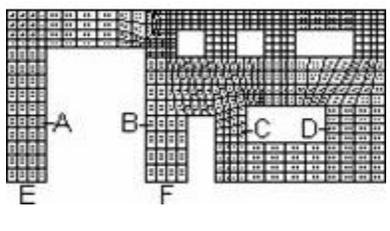
Story Moments

The story moments are calculated as the FY Corner forces times their moment arms. The forces are symmetrical so each force on one side is multiplied by twice the arm length.

Story	Sum FY Corner Forces * Moment Arm (k)	Story Moment	Elements
4	$[3.92 \text{ k} * 15' + (2.48 \text{ k} + 3.02 \text{ k}) * 7.5'] = 100.05 \text{ k}'$	100.1 kip-ft	P49-P52

Story	Sum FY Corner Forces * Moment Arm (k)	Story Moment	Elements
3	$[12.33 \text{ k} * 15' + (7.9 \text{ k} + 7.45 \text{ k}) * 7.5'] = 300.075 \text{ k}'$	300.1 kip-ft	P33-P36
2	$[24.93 \text{ k} * 15' + (16.5 \text{ k} + 13.64 \text{ k}) * 7.5'] = 600.00 \text{ k}'$	600 kip-ft	P17-P20
1	$[45.81 \text{ k} * 15' + (25.16 \text{ k} + 16.55 \text{ k}) * 7.5'] = 999.975 \text{ k}'$	999.9 kip-ft	P1-P4

Shear Wall Penetrations

		
Horizontal Deflection at Top	0.028 in.	0.033 in.
Shear @ A-A	10.82 kips	10.53 kips
Shear @ B_B	24.2 kips	23.08 kips
Shear @ C_C	33.74 kips	33.05 kips
Shear @ D-D	45.26 kips	47.35 kips
Reactions at E	10.8 kips	10.53 kips
Reactions at F	24.2 kips	23.06 kips

This is an example of a typical concrete shear wall with penetrations for windows and doors of various sizes. The files for the models are WALLPEN1.R3D (coarse mesh) and WALLPEN2.R3D (fine mesh). No theoretical solution results are given to compare with, however the two finite element densities are compared to observe the rate of convergence to the “true” answer. The shears at the various lines are computed by adding up the X corner forces for the element corners closest to the lines. The horizontal deflection is for the top of the wall. A very rigid link is added to the top of the wall to simulate the effect of a concrete horizontal diaphragm. This has the effect of stiffening the walls and spreading the load uniformly across the top of the wall. The load is applied as a uniform load of

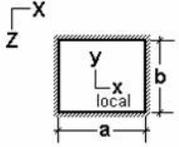
3.0 kips/ft. across the top of the wall. The total width of the wall is 38 ft, so the total applied load is 114 kips. The total height of the wall is 18 ft.

The “coarse” mesh on the left is an example of the minimum finite element mesh that should be used to model this type of wall. Notice that the coarse mesh gives good results for the wall shears and reactions. The overall deflection of the coarse mesh is off by about 15% from the “fine” mesh. The coarse mesh tends to give too much stiffness to the slender walls around the loading door opening on the left, this can be seen in the larger reactions at points E and F as well in the horizontal deflections. The fine mesh on the right shows that the slender wall sections are more flexible than shown by the coarse mesh and thus the reactions and wall shears are reduced for the slender wall sections.

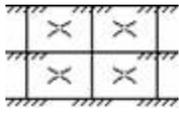
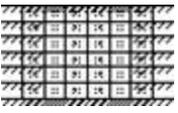
Diaphragm Modeling

Theoretical Solution for Plate with Fixed Edges

(Results from Roark’s Formulas for Stress and Strain, 5th Ed., pg. 392)

	<p>Plate Properties</p> <p>$a = 21$ ft.</p> <p>$b = 15$ ft.</p> <p>thickness = 8 in.</p> <p>$E = 3122$ksi</p> <p>$q = 60$ psi</p> <p>$\alpha = 0.0226$</p> <p>$\beta_1 = 0.4356$</p> <p>$\beta_2 = 0.2094$</p>	<p>Deflection at center (y):</p> <p>$= \alpha q b^4 / Et^3 = 0.891$ in.</p> <p>$\sigma_{Center} = \beta_2 q b^2 / t^2 = 6361$ psi</p> <p>$M_{Center} = \sigma_{Center} (t^2/6) (1 \text{ kip}/1000\#) = 67.8$ k-ft/ft</p> <p>$\sigma_{Max} = \beta_1 q b^2 / t^2 = 13,231$ psi</p> <p>$M_{Max} = \sigma_{Max} (t^2/6) (1 \text{ kip}/1000\#) = 141.1$ k-ft/ft</p>
--	---	--

Stiffness and Stress as a Function of Mesh Fineness

				
Element Mesh	2x2	4x4	5x5	6x6
Deflection @ Center Error (%)	0.032 96%	0.895 1%	0.774 13%	0.911 2.2%
My @ Center (K-ft / ft) Error (%)	80.45 19%	75.8 12%	70.3 4%	73.4 8%
Global MX Reaction @ Center of Long Side to Obtain Max. Local My	Joint 6 Reaction 844.8 K-ft	Joint 15 Reaction 717.2K-ft	Joint 18 Reaction 545.3K-ft	Joint 4 Reaction 489.4K-ft
Local My @ Center of Long Side (Mx Reaction divided by tributary length) Error (%)	80.5 k-ft / ft 43%	136.6 k-ft / ft 3%	129.8 k-ft / ft 8%	139.8 k-ft / ft 1%

The condition being modeled is a flat plate with fixed edges and a uniform load over the entire surface. The RISA-3D2DFloor files that were used to obtain these results are included as "2X2FIXED.R3D", "4X4FIXED.R3D", "5X5FIXED.R3D", and "6X6FIXED.R3D".

The plate moments at the center of the long side were calculated by dividing the global Mx reaction at the center of the long side by the tributary length. See the summary results below. (Note that the 5x5 mesh produces good results even though the Mx reaction is not at the exact center of the long side.) Remember that plates with perfectly fixed end conditions have their maximum moments at the center edge of their longest side.

Mesh	Mx Global Reaction	Tributary Length	Equation	My Local Moment
2x2	844.8 k-ft	21ft / 2 = 10.5ft	844.8 / 10.5	80.5 k-ft / ft
4x4	717.2 k-ft	21ft / 4 = 5.25ft	717.2 / 5.25	136.6 k-ft / ft
5x5	545.3 k-ft	21ft / 5 = 4.25ft	545.3 / 4.25	129.8 k-ft / ft
6x6	489.4 k-ft	21ft / 6 = 3.5 ft	489.4 / 3.5	139.8 k-ft / ft

The edge moments only need to be considered as the maximum moments when a plate is fixed at its edges, since the maximum moments will often occur in the center of the plate for most other support conditions. (The edge moments will still need to be considered for moment reversal if the plate is continuous across the supports).

For the situation of continuous slabs supported by beams between columns, the maximum moment will often occur at mid span and not at the edges. Thus a 3x3 or 5x5 mesh should be used to obtain correct moments. Even numbered meshes (e.g. 6x6, 4x4, or 2x2) should be used to obtain the best deflection information and odd numbered meshes (e.g. 3x3 or 5x5) should be used to obtain the best bending moment results. The 6x6 mesh could be used to obtain good moments and deflection results.

The internal M_y bending moments are obtained using the Global Corner Forces and the Internal Force Summation tool for the 2x2, 4x4, and 6x6 meshes. The total global MX moment on the side of an element was computed and then divided by the length of the element. (Global MX moments are parallel to local M_y moments in this model) The internal M_y @ Center are found using the Plate Forces for the odd plate example (5x5) because there is no node at the center.

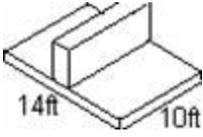
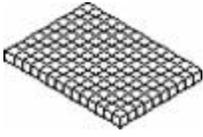
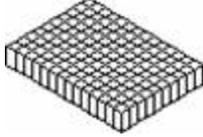
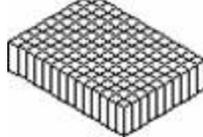
It should be noted that the deflection obtained from the 4x4 and 6x6 meshes is larger than that predicted by the Roark equations because the RISA-3D2DFloor finite element accounts for transverse shear deformation while the Roark equations ignore shear deformation.

These results are for a uniform load. If the loading is more localized, or approaches a point load, a much finer mesh in the vicinity of the load will be needed to model the loading itself and to get accurate results. Also note that RISA-3D2DFloor's finite element (like most commercial finite elements) is based on small strain theory. This means that the in-plane diaphragm stresses are not affected when the transverse deflections become large. According to Roark, (pgs. 405-409), this additional stress becomes significant when the transverse deflection is larger than half the plate thickness.

Spread Footing Modeling

Stress Accuracy as a Function of Distortion

Shown below are the analysis results for an axial wall load on a spread footing, which is then on soil springs. The files used for this parametric study are FLXFTNG.R3D for the "flexible" footing results and RGDFTNG.R3D for the rigid footing results. Note that the theoretical values shown are based on the assumption of an infinitely rigid footing.

			
Footing Thickness	12 in.	24 in.	36 in.
Element Ratio Thickness:Length	1:1	2:1	3:1
One way shear at "d" from the wall face, Flexible Footing	109.2 kips	92.4 kips	70 kips
Elements Used for Shear	9-135 (by 14) 10-136 (by 14)	10-136 (by 14) 11-137 (by 14)	11-137 (by 14) 12-138 (by 14)
Moment at Wall Face, Flexible Footing	368 k-ft	415.2 k-ft	421.2 k-ft
Elements Used for Moment	9-135 (by 14)	9-135 (by 14)	9-135 (by 14)
Theoretical 1-Way Shear at "d" from the Wall Face	117.9 kips	94.3 kips	70.7 kips
Theoretical Moment at Wall Face	424.3 k-ft	424.3 k-ft	424.3 k-ft

As can be seen in the table, the results are converging to the theoretical solution for a infinitely rigid footing as the footing thickness increases and begins to become "very rigid" when compared to the soil spring stiffness.

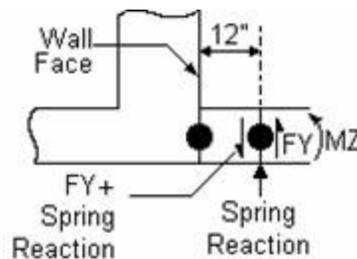
To obtain the One Way Shear values at a distance "d" from the wall face, simply sum the FY global corner force values for the elements on both sides of the appropriate row, and then take the average of these two values. You need to average the two values in this case, because the corner force results are on either side of a soil spring. For example, to obtain the one way shear for the 24" thick footing, sum all the FY corner forces for elements 9 to 135 by 14 (9, 23, 37, ...) and then 10 to 136 by 14 (10, 24, 38, ...). Then take the average of those two sums. If you don't have soil springs at the corner force locations, you don't have to average the two values. (The sums on each side in this case will be equal). The easiest way to add up the corner forces is to simply sum them from the graphics display. This way you don't have to track the element numbers.

To obtain the Moment values at the face of the wall, just add up the MZ global corner forces for the elements along the wall face. For this example these would be elements 9 to 135 by 14. Again, the easiest way to add up the corner forces is to sum them from the graphics display so you don't have to worry about element numbers.

The finite element corner forces work best when the footing is aligned with the global axes. That way the global corner forces line up with the desired footing shears and moments.

Since the theoretical values for the shear and moment are based on the assumption of an infinitely rigid footing, it is instructive to look at a finite element model where we use an artificially high value of "E" (Elastic Modulus) to approximate an infinitely rigid foundation.

Rigid Footing Results



Footing Thickness	12 in.
1-Way Shear at "d" from the Wall Face, Rigid Footing	117.9 kips
Elements Used for Shear	9-135 (by 14) 10-136 (by 14)
Moment at Wall Face, Rigid Footing	9-135 (by 14) 10-136 (by 14)
Elements Used for Moment	9-135 (by 14)

These results agree exactly with the theoretical values.

Computing Soil Spring Stiffness

Obtain the subgrade modulus for a 1' by 1' or .3m x .3m sample plate. A typical value for medium dense dry sand would be say, $k_1 = 500$ kcf. This value must first be modified to account for our actual footing size (10 ft by 14 ft). For this example we will use equations from Principles of Foundation Engineering, 3rd edition, by Braja Das, pgs. 263 - 264. We will assume a 1ft x 1ft sample plate.

$$K_{10 \times 10} = k_1 \left(\frac{B+1}{2B} \right)^2 = 500 \left(\frac{10+1}{2 \cdot 10} \right)^2 = 151 \text{ kcf}$$

$$K_{10 \times 14} = K_{10 \times 10} * \left(\frac{1+0.5 \cdot B/L}{1.5} \right) = 151 * \left(\frac{1+0.5 \cdot 10/14}{1.5} \right) = 136.6 \text{ kcf} = 0.07906 \text{ kip/in}^3$$

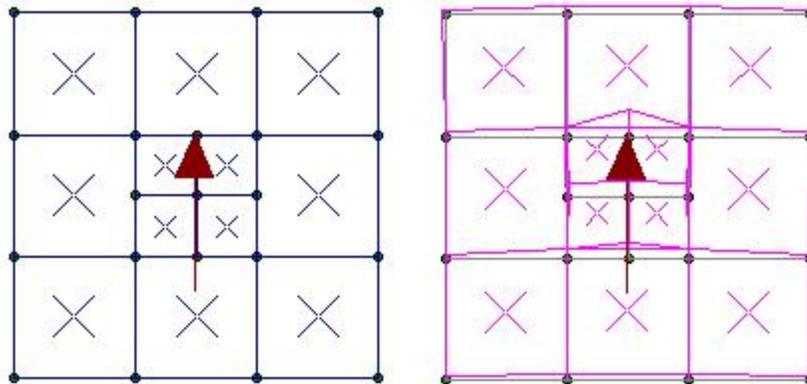
We can now calculate the spring stiffnesses for all the joints in the model based on tributary area. The program is capable of automatically generating these soil springs based on the tributary area of each node in the plate element mesh. Refer to [Generating Soil Springs](#) of the Boundary Conditions section for more information.

Tributary Area	K(10x14) k/in ³	K(spring) k/in	Example Joint
1 sq.ft = 144 sq.in.	0.07906	11.38	13
0.5 sq.ft = 72 sq.in.	0.07906	5.69	34
0.25 sq.ft = 36 sq.in.	0.07906	2.85	1

Although a little out of date, the ACI publication 336.2R-88, "Suggested Analysis and Design Procedures for Combined Footing and Mats", is another good reference for the modeling of mat foundations.

Plate Connectivity Problems

Shown below is a common modeling problem with plate elements. Since plates only have connectivity at their corner nodes, the applied load at middle joint connects to the plates below the joint, but not to the one above it. Because of this lack of connectivity, you see the joint "pushing through" the plate edge above in the plotted deflected shape.



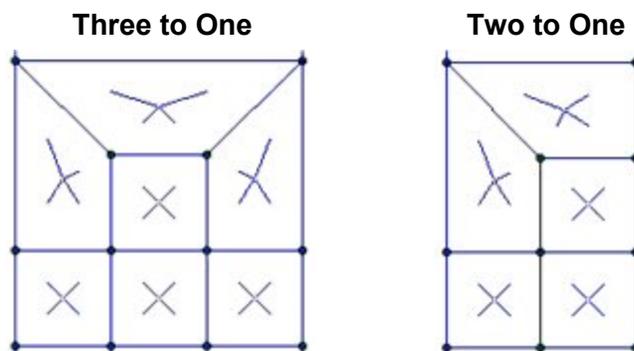
The proper way to hand this type of mesh is with one of the mesh transitions described in the following section.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Plate Connectivity**.

Mesh Transition Examples

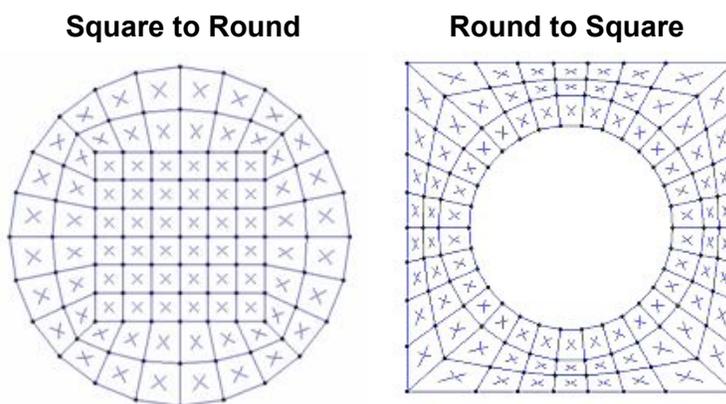
Coarse Mesh to Fine Mesh

Shown below are two methods for transitioning from an area with a fine mesh to an area with a larger / coarser mesh.



Rectangular Mesh to Radial Mesh

Shown below are two methods for transitioning from an area with a rectangular mesh into an area with a radial mesh.



For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Plate Mesh**.

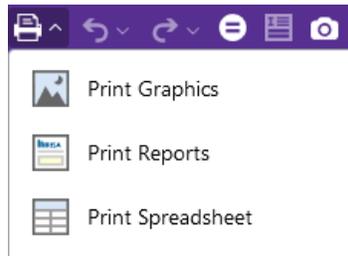
Printing

You can print graphic or tabulated information from your current model by using the Quick Access toolbar or the File menu.

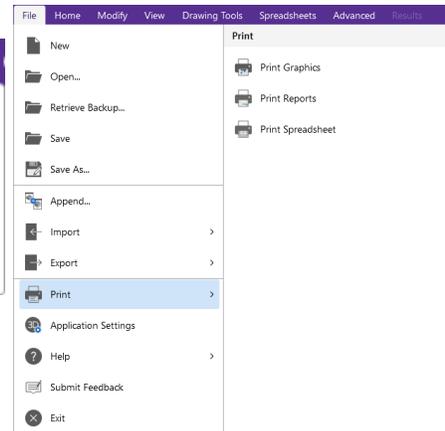
The following links provide instructions on using the various print options.

- [Print Graphics](#)
- [Print Reports](#)
- [Print Spreadsheet](#)
- [Print to a File](#)

Print - Quick Access Toolbar



File menu



The following provide more details on printing graphics and building custom reports.

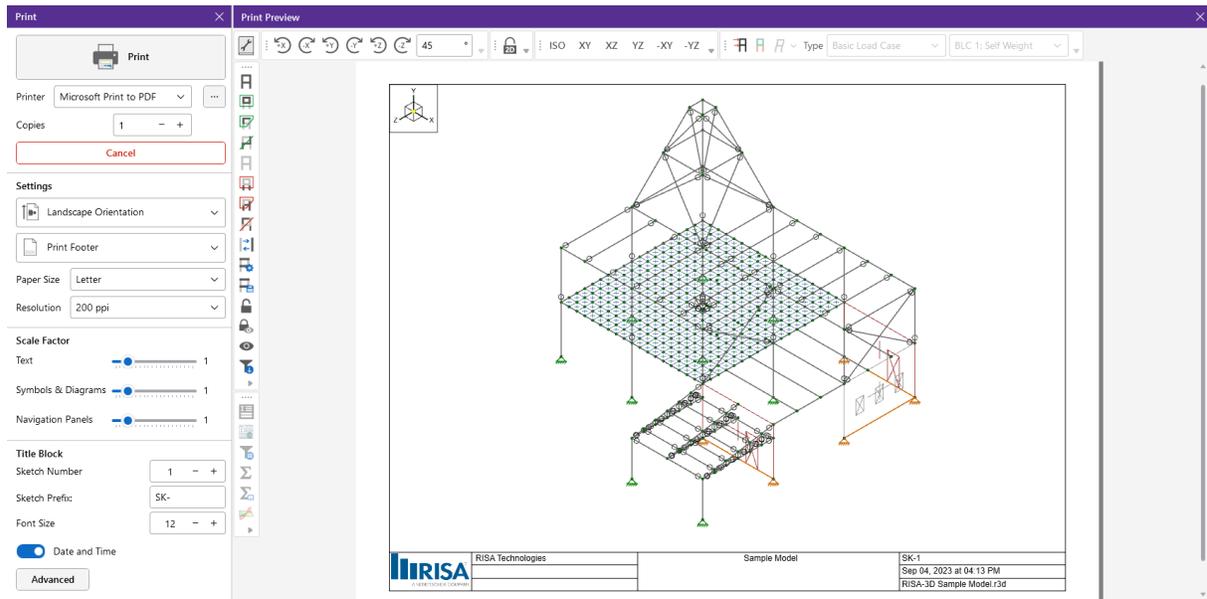
- [Print Options](#)
- [Print Graphics](#)
- [Build a Custom Report](#)
- [Save a Report as a Template](#)

Print Options

The available print options vary slightly, depending on whether a spreadsheet or a graphic (model) is active at the moment when you click the 'Print' icon on the Quick Access toolbar or the 'Print' option on the File menu.

- When a *graphic* (model) is currently active, the available print options from both the 'Quick Access' toolbar and 'File' menu are **Print Graphics**, **Print Reports** or **Print Spreadsheet**.

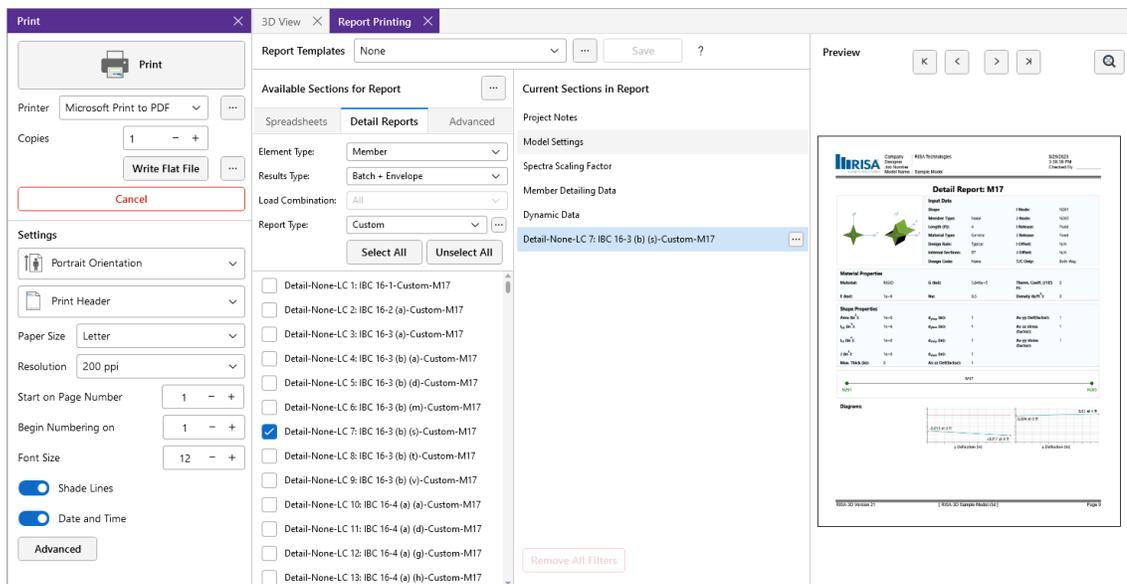
Once you've chosen to print a 'graphic', the 'Print' panel opens with print settings that apply to printing graphics, along with a 'Print Preview' panel that displays the graphic on a page (shown in the following image). See [Graphics Printing](#) for more information, and see [Print Graphics](#) for instructions.



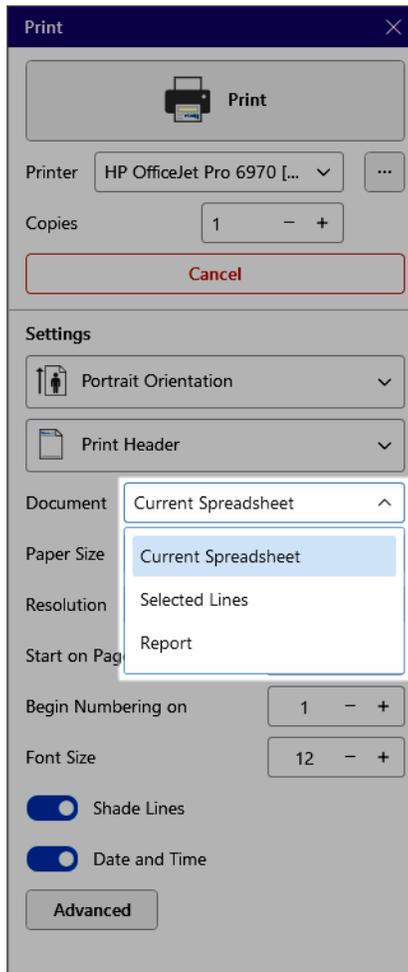
- When a *spreadsheet* is currently active, the available print options in both the ‘Quick Access’ toolbar and ‘File’ menu are **Print Reports** or **Print Spreadsheet**.

Once you’ve chosen your print option, the ‘Print’ panel opens with print settings that are applicable to that option.

- If you choose “Print Reports”, a ‘Print’ panel opens on the left, and a ‘Report Printing’ tab opens in the 3D View panel, along with a ‘Preview’ that displays a graphical view of the report’s pages as you choose the report sections and details to include in the report (shown in the following image). See [Print a Report](#) for more information.

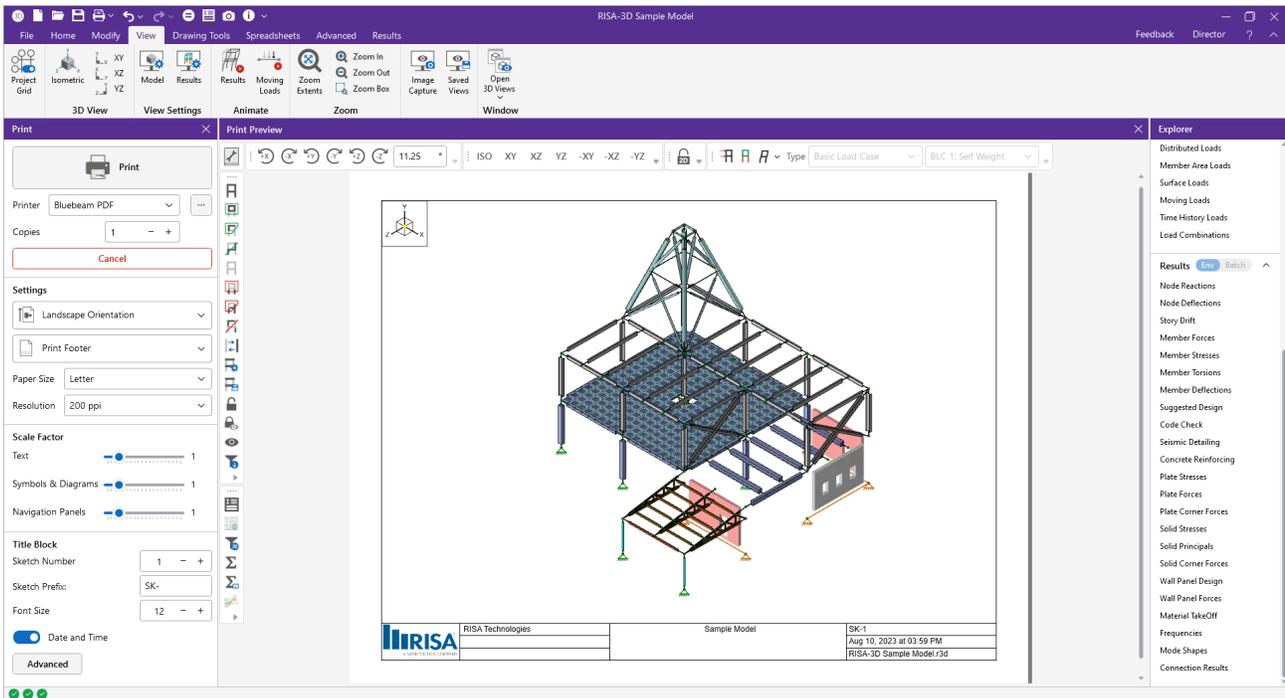


- If you choose “Print Spreadsheet”, a Print panel opens on the left. This panel provides three additional print options in the ‘Document’ box (as shown in the following image) that allow you to print the **Current Spreadsheet**, **Selected Lines** in the current spreadsheet, or switch to **Report** printing. Note that these print options are only available when a ‘spreadsheet’ is active. See [Print a Report](#) for instruction.



Print Graphics

At any point, while in a 3D View or Wall Panel Editor, you can print a graphic by choosing **Print Graphics** from any of the **Print** menus. The current window changes into a ‘Print Preview’ and allows all the same view settings to be adjusted, even when in the preview. Rotating, zooming and rendering are only a few of the settings that can be altered dynamically in the preview.



Many of the same options that exist in **Report Printing** are present in **Graphic Printing**, but some unique options available are **Scale Factors** and **Title Block Info**. The following table provides more information on the options that are unique to printing graphics.

Unique Options for Printing Graphics

Option	Description
<p>Scale Factor</p>	<p>Three different Scale Factors are defined. They make 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.</p> <p>Text: This option is used to increase/decrease the size of the text items.</p> <p>Symbols & Diagrams: This option is used to increase/decrease the size of symbol and diagram items.</p> <p>Navigation Panels: This option is used to increase/decrease the size of navigational panel items.</p>
<p>Title Block Info (Header)</p>	<p>This option lets you include the Title Block Info(Header) to display the Model Title, Designer, Company Name, Date and Time (All of which are defined in your Project Info). You can also enter a comment that will be included in the Title Block.</p> <p>Advanced Settings This option lets you to set the margins.</p>
<p>Sketch #</p>	<p>This option lets you define the starting number for the first sketch. The sketch number automatically increments with each sketch added.</p>

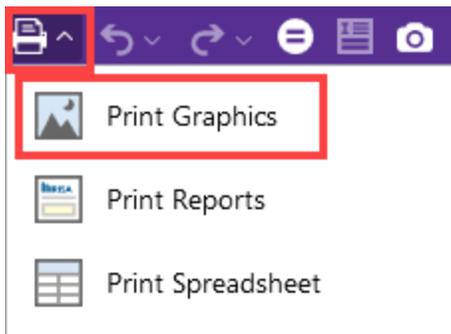
Option	Description
Sketch Prefix	This option lets you define a specific label to use in front of the sketch number (e.g., SK-). This prefix will be used for each sketch added.
Font Size	This option
Date and Time	This options lets you choose to show/hide the date and time that the print is created. Simply slide the dot to the left to hide the date and time. By default, the date and time is shown.
Advanced	This option lets you include a logo on the print and set the outside margins (top, bottom left and right.)

To Print Graphics

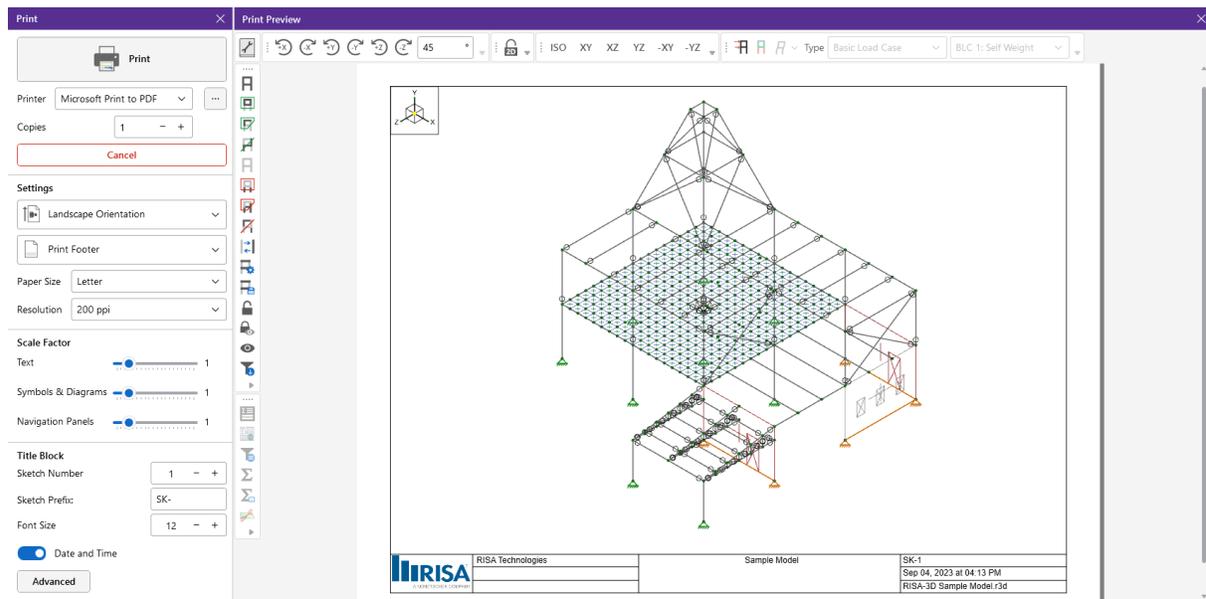
Print Graphics from the Quick Access Toolbar

To print graphics from the Quick Access toolbar:

1. While in a graphic (model), click on the **Print** icon in the 'Quick Access' toolbar.
A menu displays the print options.



2. Click on **Print Graphics**.
A 'Print' panel appears on the left and a Print Preview panel opens to its right.

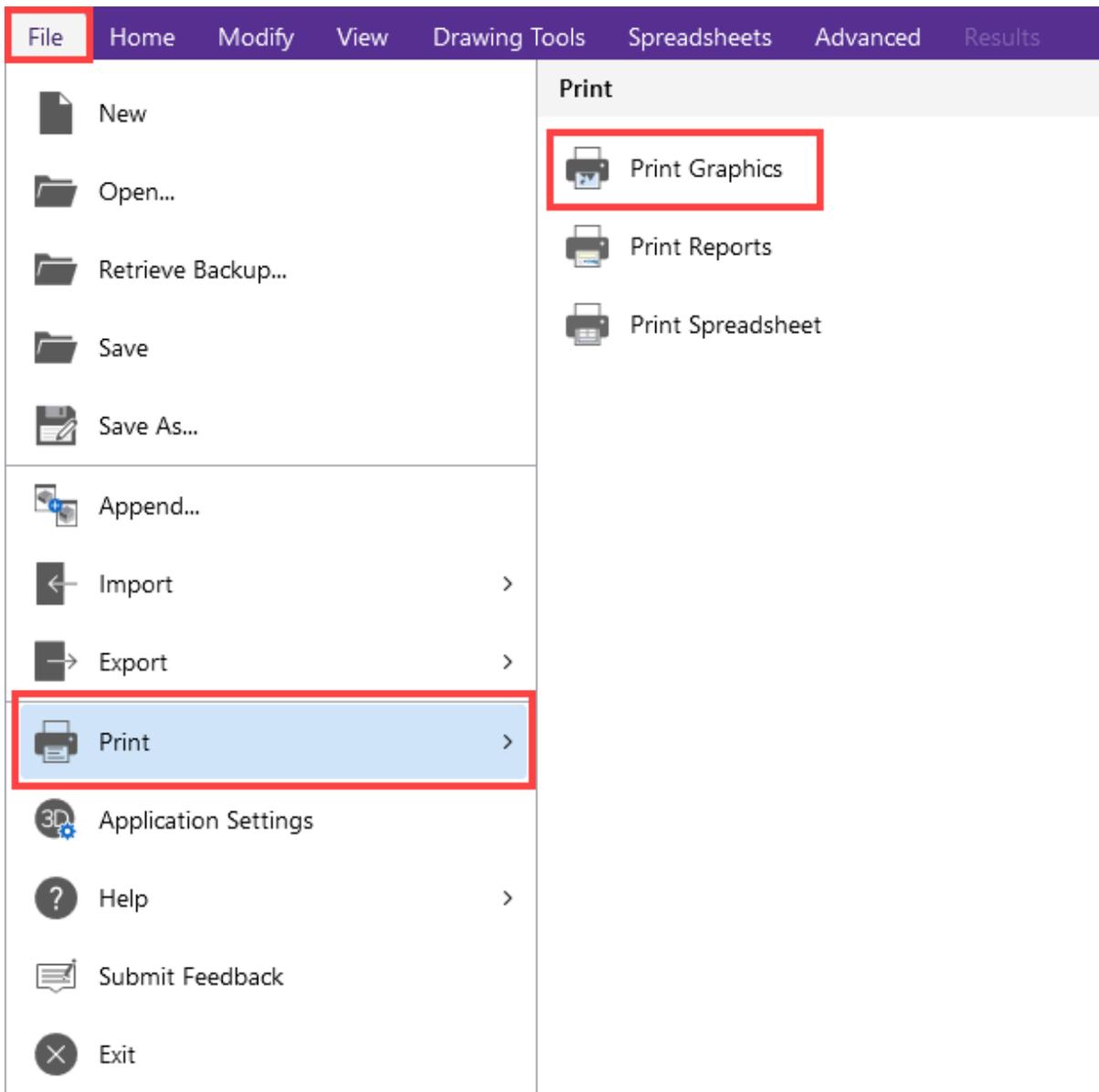


3. Choose the print **Settings** you want to use for printing the graphic.
These settings include page orientation, whether or not you want to print a footer, the paper size and the print resolution.
4. Choose the **Scale Factor** for text, symbols & diagrams and navigation panels within the graphic.
5. Choose **Title Block** settings.
Title Block settings are those that appear at the bottom of the graphic page. They represent what you want to use for a Sketch Prefix and the number to follow the prefix, the font size of all text that appears in the Title Block area and whether to include the date and time that the graphic is printed. See [Print Graphics](#) for additional information on these settings.
6. When finished with your choices, click the **Print** button at the top of the Print panel.

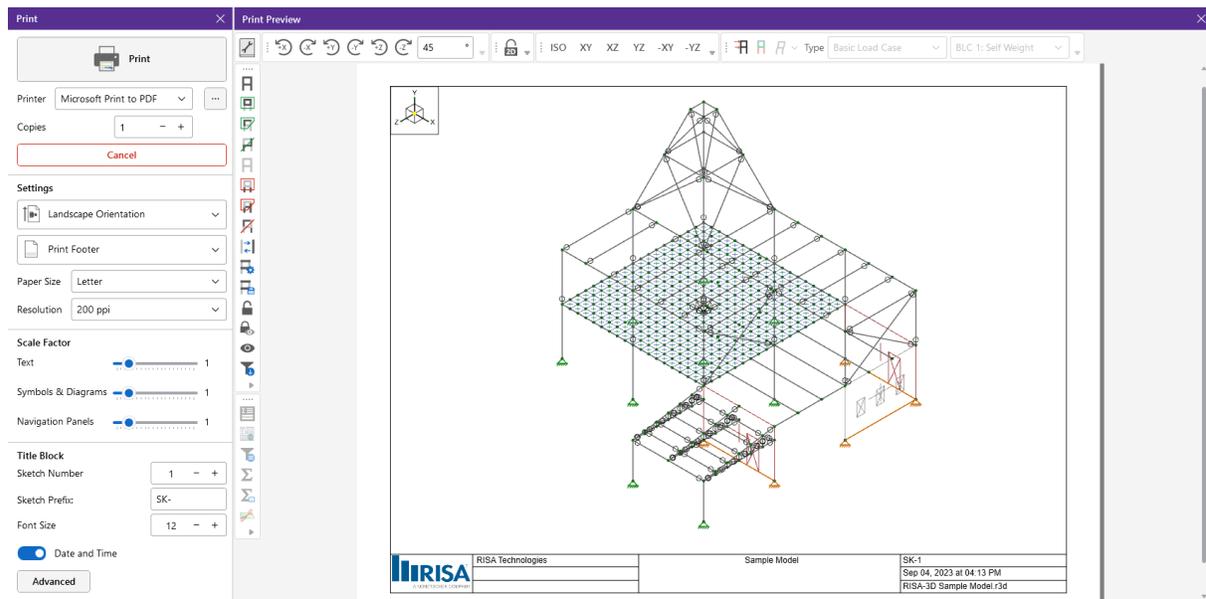
Print Graphics from the File menu

To print graphics from the File menu:

1. While in a spreadsheet, go to **File > Print > Print Graphics**.



A 'Print' panel appears on the left and a Print Preview panel opens to its right.



2. Choose the print **Settings** you want to use for printing the graphic.

These settings include page orientation, whether or not you want to print a footer, the paper size and the print resolution.

3. Choose the **Scale Factor** for text, symbols & diagrams and navigation panels within the graphic.
4. Choose **Title Block** settings.

Title Block settings are those that appear at the bottom of the graphic page. They represent what you want to use for a Sketch Prefix and the number to follow the prefix, the font size of all text that appears in the Title Block area and whether to include the date and time that the graphic is printed. See [Print Graphics](#) for additional information on these settings.

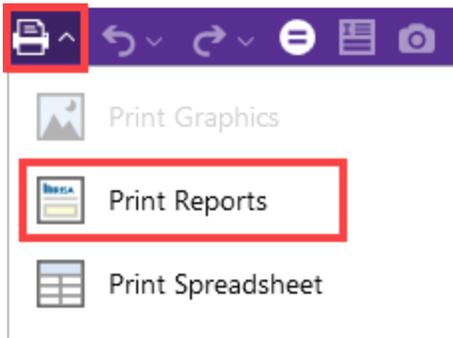
5. When finished with your choices, click the **Print** button at the top of the Print panel.

Print Reports

Print a Report from the Quick Access Toolbar

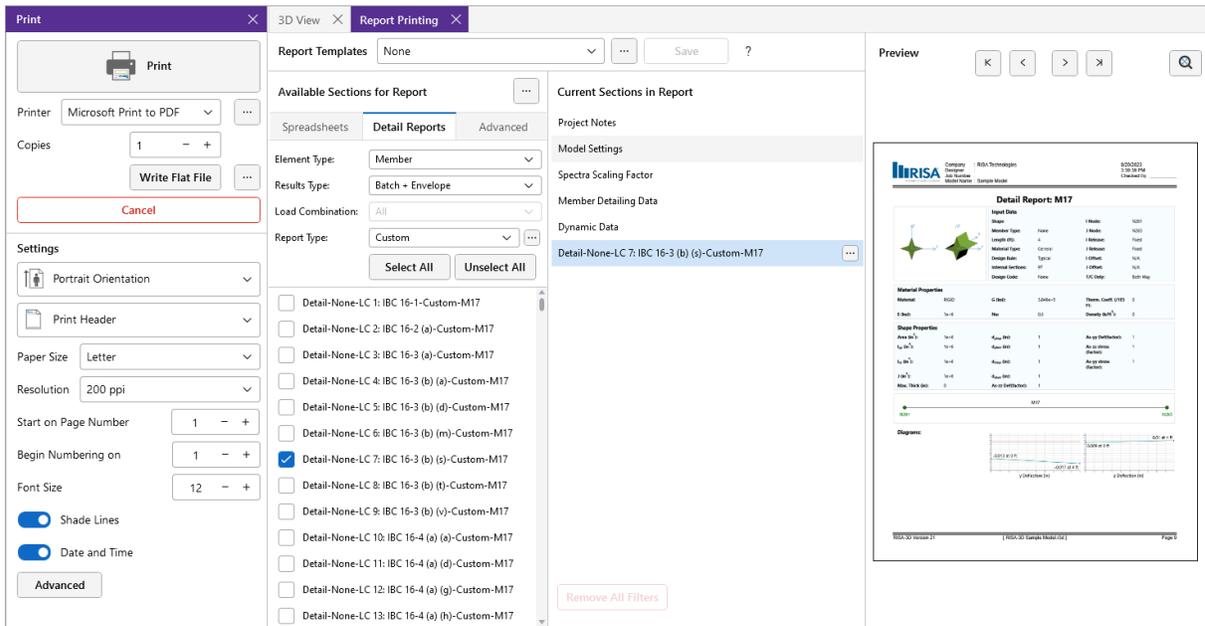
To print a report from the Quick Access toolbar:

1. While in a spreadsheet, click on the **Print** icon in the 'Quick Access' toolbar.
A menu displays the print options.



2. Click on the **Print Reports** option.

A 'Print' panel opens on the left, and a "Report Printing" tab opens in the 3D View panel, along with a 'Preview' panel that displays a graphical view of the report's pages, as you choose your Report Printing options (as shown in the following image).



3. In the **Print** panel on the left, choose how you want to print your report.
4. Move to the **Report Printing** tab in the 3D View panel and choose what you want to print.

From the **Report Printing** tab you can choose from a set of "standard" reports or create your own "custom" report, which can also be save as a template for reuse with other models.

- A "standard" report includes predefined report sections, and is included in the RISA-3D application. The report sections included in a standard report template are based on the report template you choose. Though, you can still use the customizing features to choose other report sections to print in the report.



Note: While you cannot save changes to any of the report templates that come with RISA-3D, you CAN save the customized report as a “new” template, which will include your changes. Any new templates you save are modifiable.

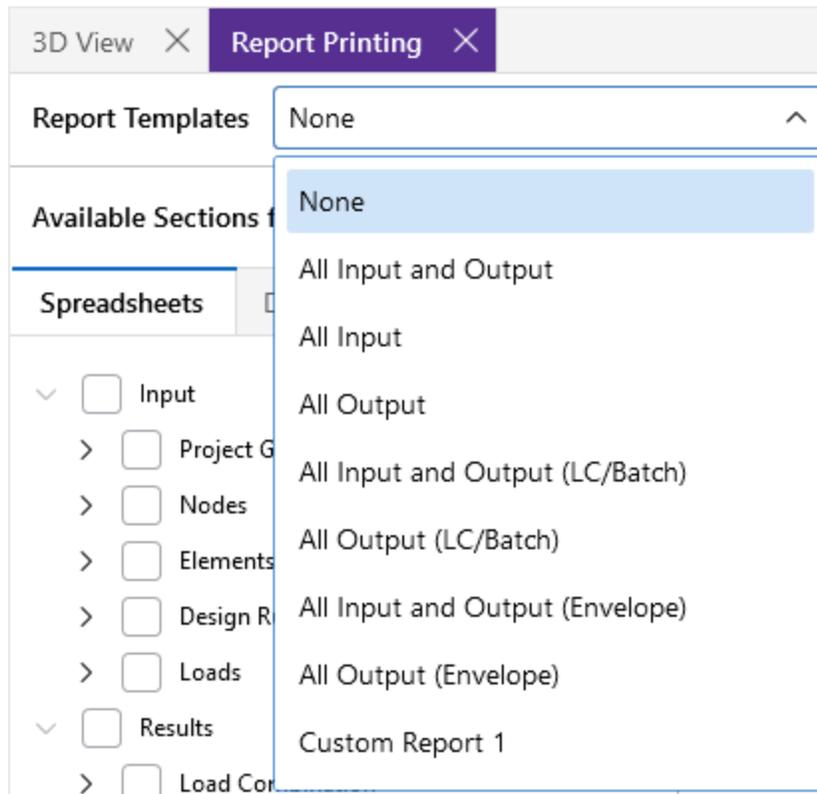
- A “custom” report is any report that you create using the customizing options found in the ‘Available Sections for Reports’ and ‘Current Sections in Report’ panels.

5. Choose the report type you want to print.

To choose a *standard* report or a *custom* report you’ve already created

- Click on the **Report Templates** arrow.

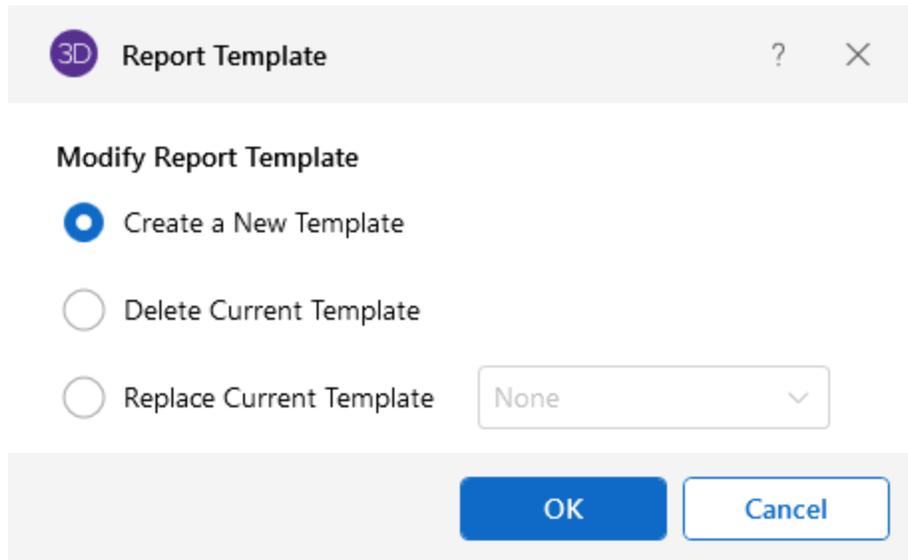
A list of all standard and saved custom reports appears.



- Click on the report name to choose it.

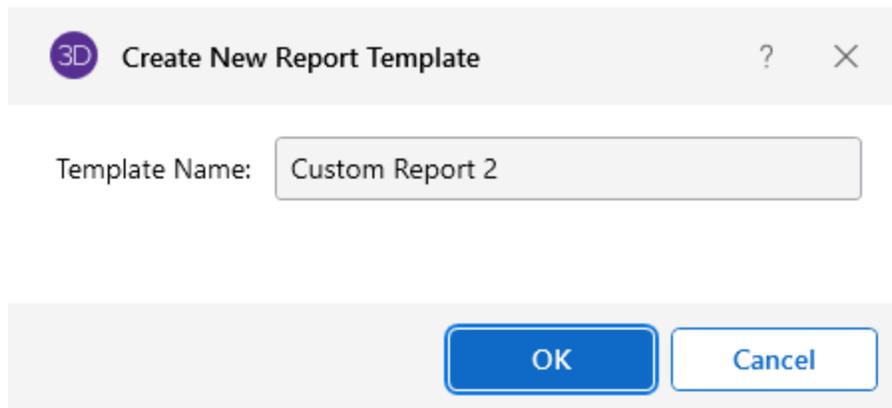
To create a *custom* report

- a. Click the  ellipsis button to the right of **Report Templates** option
A Report Template window opens.



- b. Click on **Create a New Template**, if not already selected.
c. Click **OK**.

A Create New Report Template window opens.



- d. In the **Template Name** box, type a name for your new, custom template.
e. Click **OK**.
Your custom report name is added to the Report Templates list and selected in the Report Template box.
f. Choose the sections to include in your custom report using the 'Available Sections for Report'  ellipsis button and *Spreadsheets, Detail Reports*

and *Advanced* tabs. See [Build a Custom Report](#) for more information.

As you choose the options, the 'Current Sections in Report' panel displays the report sections, and the 'Preview' panel displays the related pages.

The 'Current Sections in Report' panel displays all the report sections currently selected for the report. You can also filter report section's elements by their properties for any report section that displays an  ellipsis button next to its name.

6. (Optional) In the **Current Sections in Report** panel, click the  ellipsis button next to any section that you would like to filter its elements by their properties.

As you choose your report options, the 'Save' button at the top of the Report Printing panel becomes active, as shown in the following image.

7. Click the **Save** button if you want to save the report as a report template.

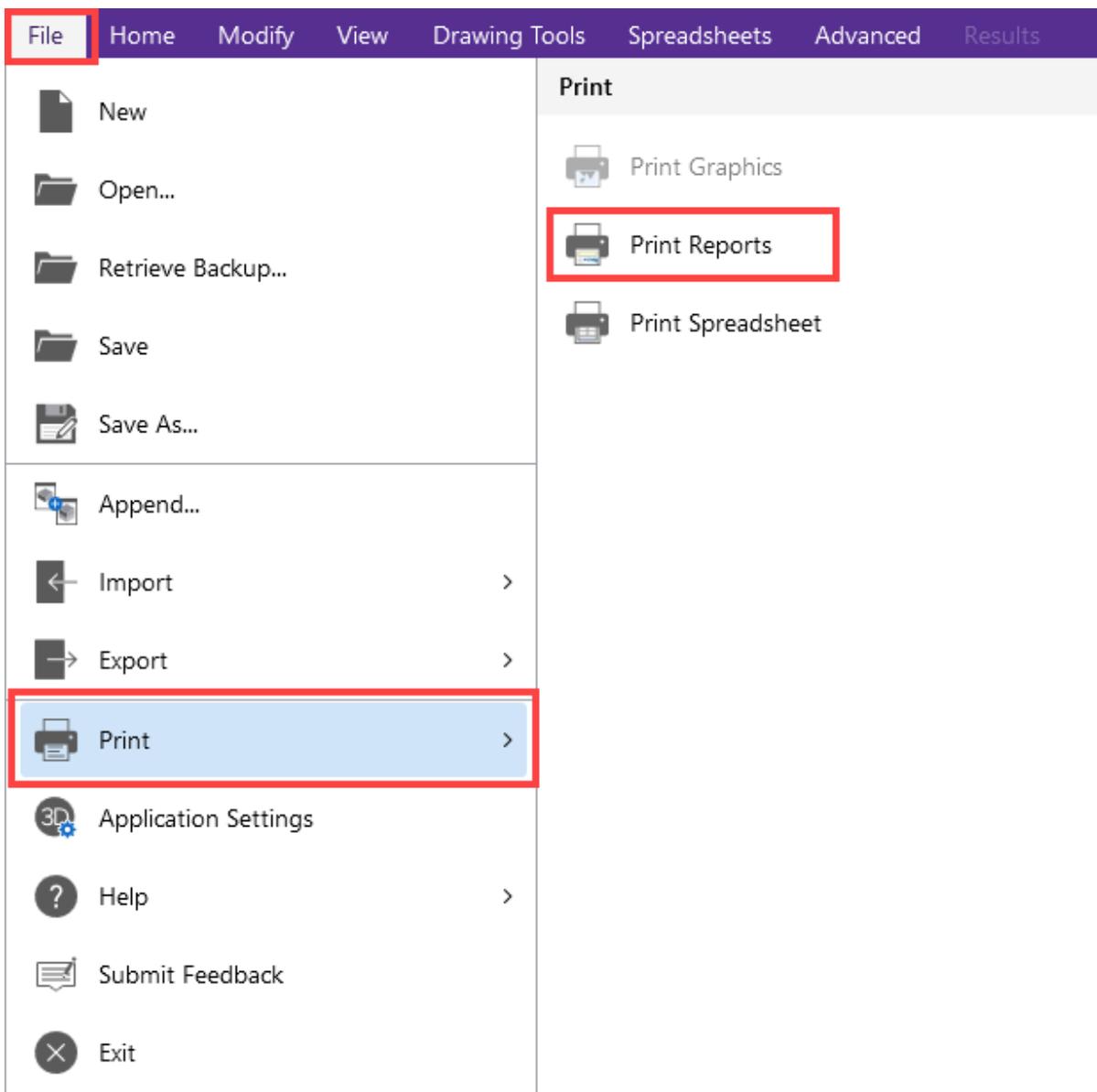


8. When finished with your choices, click the **Print** button at the top of the 'Print' panel.

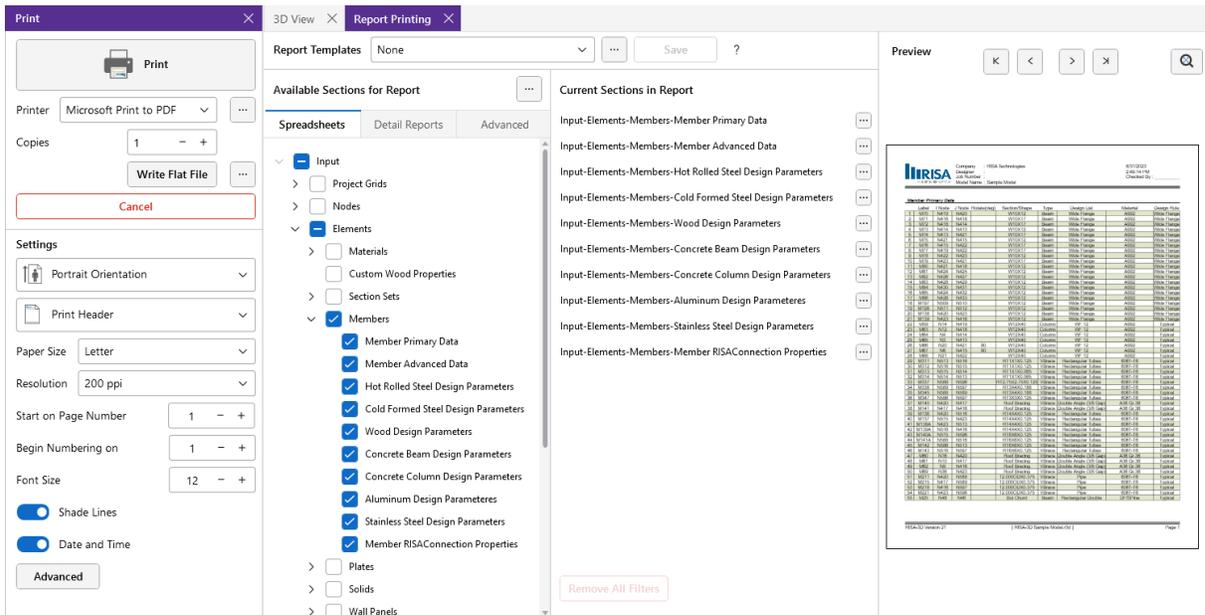
Print a Report from the File Menu

To print a report from the File menu:

1. While in a spreadsheet, go to **File > Print > Print Reports**.



A 'Print' panel opens on the left, and a "Report Printing" tab opens in the 3D View panel, along with a 'Preview' panel that displays a graphical view of the report's pages, as you choose your Report Printing options (as shown in the following image).



2. In the **Print** panel on the left, choose how you want to print your report.
3. Move to the **Report Printing** tab in the 3D View panel and choose what you want to print.

From the **Report Printing** tab you can choose from a set of “standard” reports or create your own “custom” report, which can also be save as a template for reuse with other models.

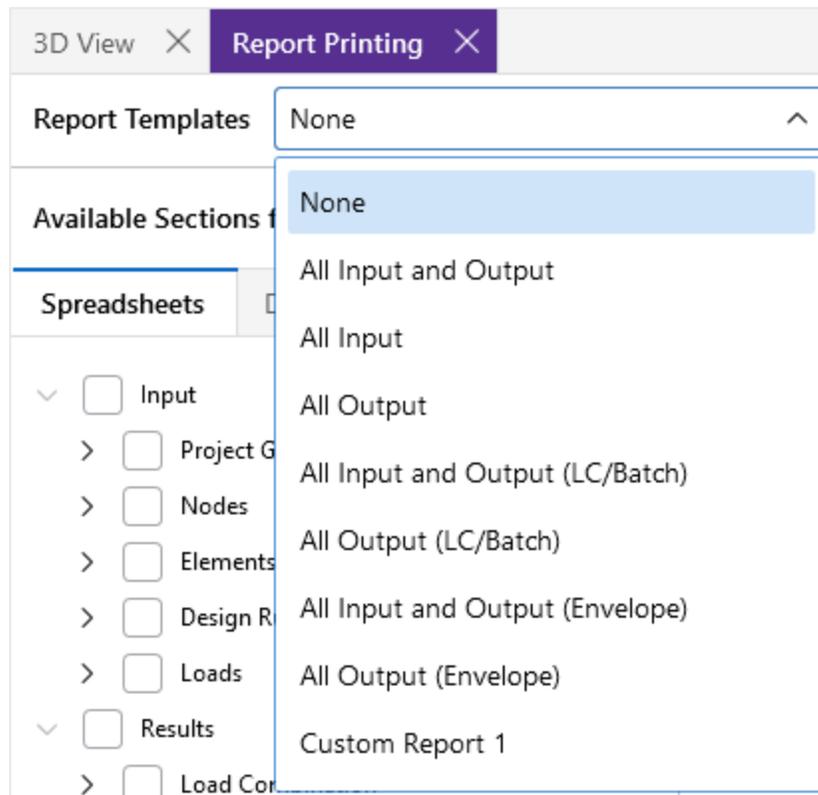
- A “standard” report includes predefined report sections, and is included in the RISA-3D application. The report sections included in a standard report template are based on the report template you choose. Though, you can still use the customizing features to choose other report sections to print in the report.

 **Note:** While you cannot save changes to any of the report templates that come with RISA-3D, you CAN save the customized report as a “new” template, which will include your changes. Any new templates you save are modifiable.

- A “custom” report is any report that you create using the customizing options found in the ‘Available Sections for Reports’ and ‘Current Sections in Report’ panels.
4. Choose the report type you want to print.

To choose a *standard* report or a *custom* report you’ve already created

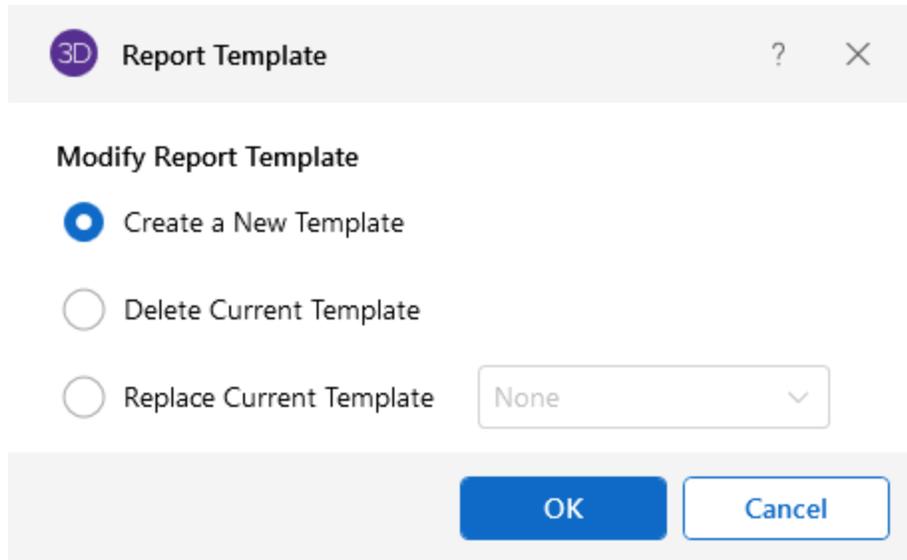
- a. Click on the **Report Templates** arrow.
A list of all standard and saved custom reports appears.



- b. Click on the report name to choose it.

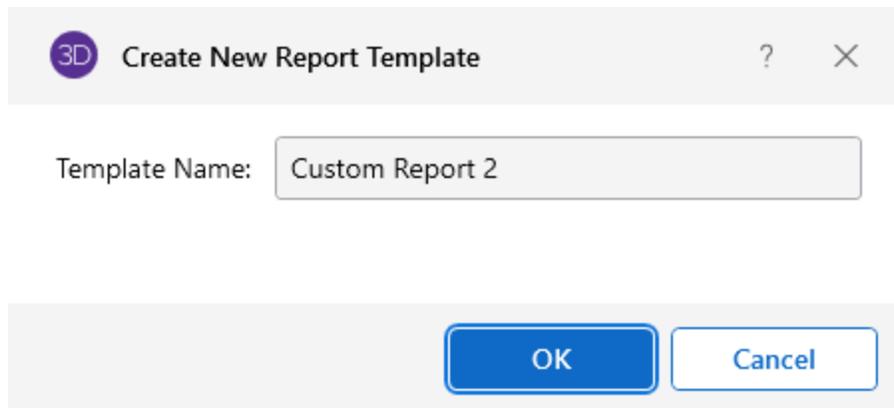
To create a *custom* report

- a. Click the ellipsis button to the right of **Report Templates** option
A Report Template window opens.



- b. Click on **Create a New Template**, if not already selected.
- c. Click **OK**.

A Create New Report Template window opens.



- d. In the **Template Name** box, type a name for your new, custom template.
- e. Click **OK**.

Your custom report name is added to the Report Templates list and selected in the Report Template box.

- f. Choose the sections to include in your custom report using the 'Available Sections for Report'  button and *Spreadsheets*, *Detail Reports* and *Advanced* tabs. See [Build a Custom Report](#) for more information.

As you choose the options, the 'Current Sections in Report' panel displays the report sections, and the 'Preview' panel displays the related pages.

The 'Current Sections in Report' panel displays all the report sections currently selected for the report. You can also filter report section's elements by their properties for any report section that displays an  ellipsis button next to its name.

- (Optional) In the **Current Sections in Report** panel, click the  ellipsis button next to any section that you would like to filter its elements by their properties.

As you choose your report options, the 'Save' button at the top of the Report Printing panel becomes active, as shown in the following image.

- Click the **Save** button if you want to save the report as a report template.



- When finished with your choices, click the **Print** button at the top of the 'Print' panel.

Print a Spreadsheet

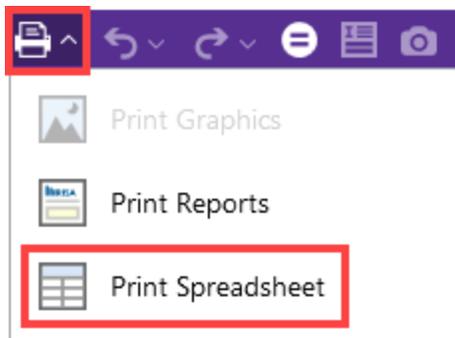
There are two main ways to print a spreadsheet; through the Quick Access toolbar or from the File menu. Each method is described in the following sections.

Print a Spreadsheet from the Quick Access Toolbar

To print a spreadsheet from the Quick Access toolbar:

- While in a spreadsheet, click on the **Print** icon in the 'Quick Access' toolbar.

A menu displays the print options.



- Click on **Print Spreadsheet** to open the 'Print' panel.

If you weren't in a spreadsheet when you clicked on Print Spreadsheet:

A "Print Spreadsheet" window opens, allowing you to choose the spreadsheet you want to print (as shown in the following image).

3D Print Spreadsheet ? X

Input

Data Entry

Result

Load Combination

Sort by: Combination Item

Envelope

Dynamic

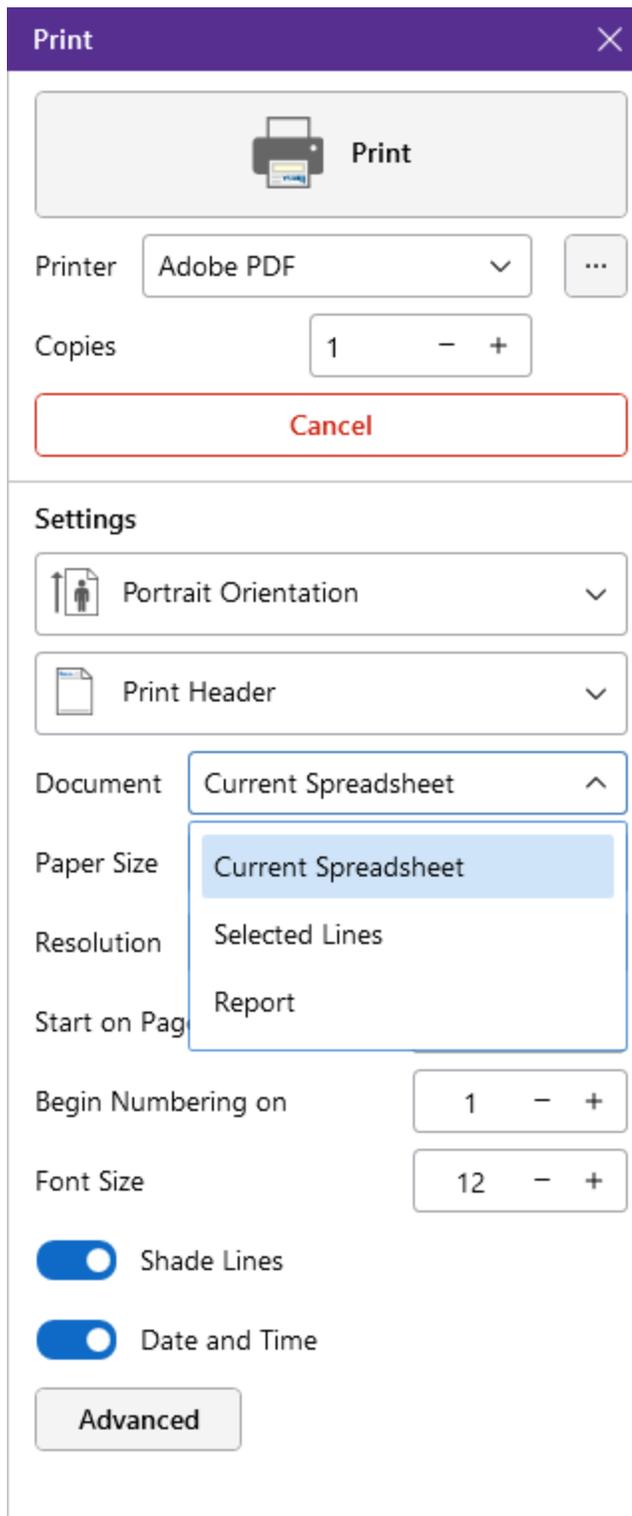
Print **Cancel**

If you have not “Solved” the model yet, you can only choose a **Data Entry** spreadsheets. If you have “solved” the model, you can choose from **Data Entry**, or one of the Results spreadsheets (**Load Combination**, **Envelope** or **Dynamic**), depending on what you chose to “solve”.

- a. Choose the spreadsheet you want, from the available options.
- b. Click **Print**.

The spreadsheet you chose is brought to focus in the 3D View panel, and the ‘Print’ panel opens.

The Print panel opens on the left side of the 3D View panel.



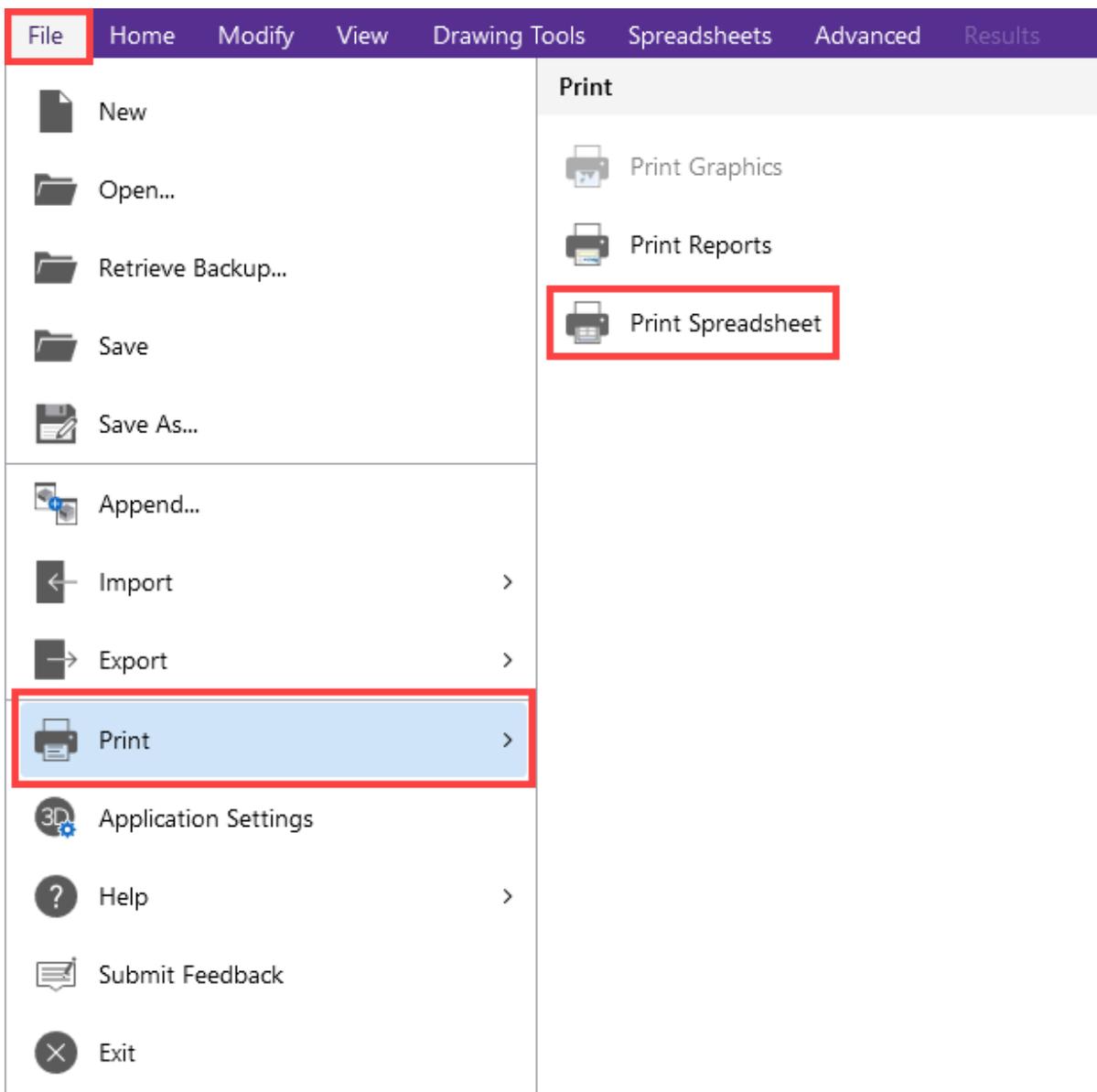
3. In the **Print** panel, choose your printing options.

4. Click the **Document** arrow and choose whether to print the **Current Spreadsheet**, **Selected Lines** from the current spreadsheet, or switch to **Report** printing options.
 - If you choose **Current Spreadsheet**, the whole spreadsheet is selected for printing.
 - If you choose **Selected Lines**, only the lines that you currently have highlighted in the spreadsheet are selected for printing. If you have not selected any lines, all lines are selected by default. If you don't want all lines, you can simply click on only those lines you want printed.
 - If you choose **Report**, the Report Printing tab opens in the 3D View panel, allowing you to choose from the various report printing options. See steps **4 - 6** under [Print a Report from the Quick Access Toolbar](#).
5. When finished with your print choices, click the **Print** button at the top of the Print panel.

Print a Spreadsheet from the File Menu

To print a spreadsheet from the File menu:

1. While in a spreadsheet, go to **File > Print > Print Spreadsheet**.



If you weren't in a spreadsheet when you clicked on Print Spreadsheet:

A "Print Spreadsheet" window opens, allowing you to choose the spreadsheet you want to print (as shown in the following image).

3D Print Spreadsheet ? X

Input

Data Entry

Result

Load Combination

Sort by: Combination Item

Envelope

Dynamic

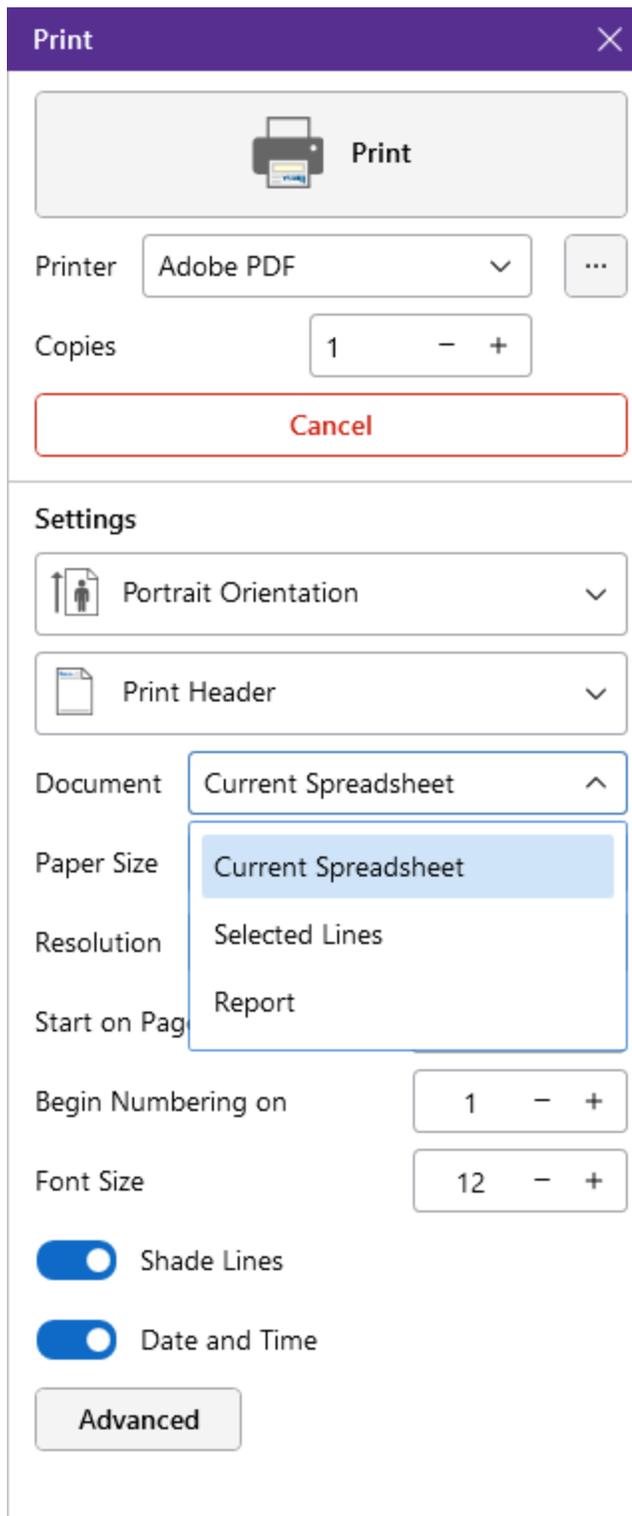
Print **Cancel**

If you have not “Solved” the model yet, you can only choose a **Data Entry** spreadsheets. If you have “solved” the model, you can choose from **Data Entry**, or one of the Results spreadsheets (**Load Combination**, **Envelope** or **Dynamic**), depending on what you chose to “solve”.

- a. Choose the spreadsheet you want, from the available options.
- b. Click **Print**.

The spreadsheet you chose is brought to focus in the 3D View panel, and the ‘Print’ panel opens.

The Print panel opens on the left side of the 3D View panel.



2. In the **Print** panel, choose your printing options.

3. Click the **Document** arrow and choose whether to print the **Current Spreadsheet**, **Selected Lines** from the current spreadsheet, or switch to **Report** printing options.
 - If you choose **Current Spreadsheet**, the whole spreadsheet is selected for printing.
 - If you choose **Selected Lines**, only the lines that you currently have highlighted in the spreadsheet are selected for printing. If you have not selected any lines, all lines are selected by default. If you don't want all lines, you can simply click on only those lines you want printed.
 - If you choose **Report**, the Report Printing tab opens in the 3D View panel, allowing you to choose from the various report printing options. See steps **4 - 6** under [Print a Report from the Quick Access Toolbar](#).
4. When finished with your print choices, click the **Print** button at the top of the Print panel.

Print to a File

A flat file is a file without column headings, print formatting, or graphical elements and is useful for importing and parsing into spreadsheets, database tables, or as post processor input data.

There are several options available to make the flat file easier to parse. Note that printing and then looking at a sample output file with all the options selected, makes it easier to understand what the options do.

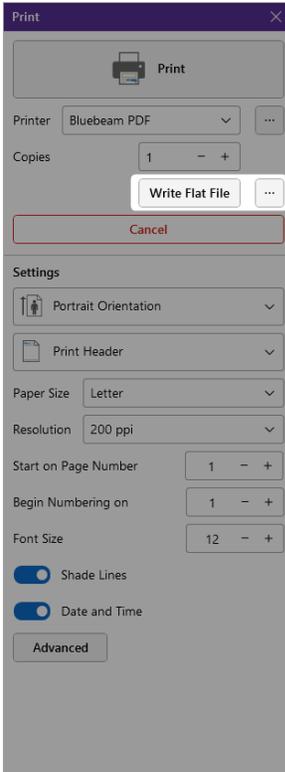
The current flat file options are saved each time the program is closed.

Print to a Flat File

The Print to a Flat File feature can be used only when printing a standard or customized report.

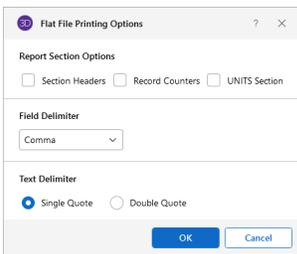
To print a report to a flat file:

1. While in the report, move to the **Print** panel.



2. (Optional) To make the flat file easier to parse, click the  ellipsis button to the right of the 'Write Flat File' option (as shown in the previous image).

A Flat File Printing Options window opens.



- a. Choose the parsing options (described in the following table) you want.

Flat File Printing Options

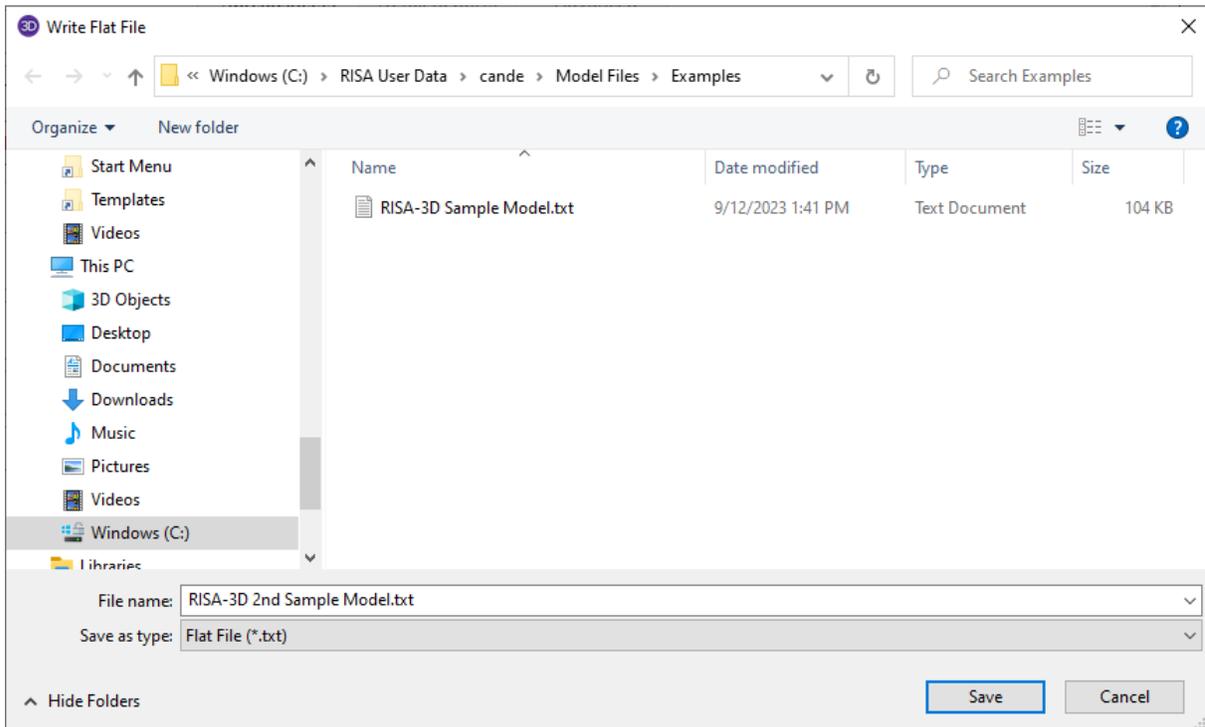
Option	Description
<p>Report Section Options</p>	<p>These options let you add section headers, record counters and a UNITS section to the flat file.</p> <p>Section Headers: This option adds a 'Section Header' area that prints a text description of each block of data. For example, the Node data would be preceded by a [NODE]</p>

Option	Description
	<p>header on it's own line before the data.</p> <p>Record Counters: This option adds a 'Record Count' area that is useful if you're writing looping code to read in the number of records within each data block. The number of records prints on it's own line.</p> <p>UNITS Section: This option adds a special 'UNITS Section' that shows the units used for all the values in the program.</p>
Field Delimiter	<p>This option lets you choose what character to use to separate each field of data in a record. For example, each coordinate value in the Node data would be separated by the character selected. Programs that you might be importing the file into, like MS Excel or MS Access, often have options to select what the field delimiter will be for records of data.</p>
Text Delimiter	<p>This option works like the field delimiter, except that it's used to set apart text labels. All text in the flat file is enclosed at the beginning and the end by the selected text delimiter character. This is very useful when trying to read in label strings that contain embedded spaces. As an example, the Node labels in the Node data would each be enclosed by a single or double quote.</p>

b. Click **OK** to save your selections.

3. Click on **Write Flat File**.

A Write Flat File window opens in which you can select where to save the file and give it a name.



By default, the flat file is named the same as the model file but with a .txt extension, and it is saved to the same folder in which the model resides.

4. Click **Save** to save the flat file and close the Write Flat File window.

Build a Custom Report

A “custom report” relates to any report that isn’t one of the standard reports included in the application. You can customize any report, be it standard, an existing custom report that you’ve saved as a template, or a new report that you are creating. For example, if you choose a standard report, but then find that it includes more sections than you want in your report, you can customize the standard report, view and print it, and also save it as a “new” template if you’d like to use the same report for other models.



Note: While you cannot save changes to any of the report templates that come with RISA-3D, you CAN save the customized report as a “new” template, which will include your changes. Any new templates you save are modifiable.

To build your own custom report you can select any of the spreadsheets, detail reports, or miscellaneous sections found in the **Available Report Sections** and they will be populated in the **Current Section in Report** panel. You can add entire categories by selecting the heading of a dropdown section in the **Spreadsheets** tab. You can re-order the report by dragging and dropping the sections in your report you would like to move. You can also remove sections from your report by

deselecting them similar to how they were selected, or you can drag the item out of the **Current Sections in Report** region.

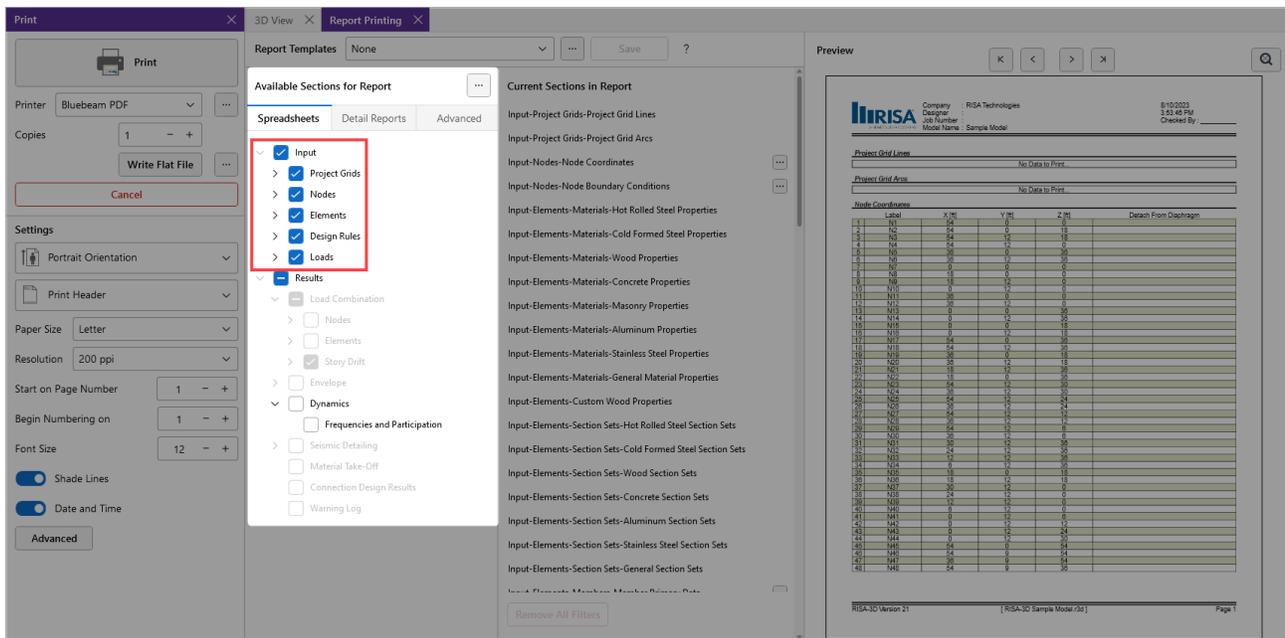
The  ellipsis button found next to applicable report sections in the **Current Sections Report** panel lets you filter the sections included in the report based on selected element properties. This applies to both spreadsheets and detail reports. Only those report sections that include elements having properties have the ellipsis button.

The **Current Sections in Report** portion of the panel shows which sections and the order the printed report will be built. The **Misc.** tab of the **Available Report Sections** contains all images that were added to the project as well as the other miscellaneous items not in the **Spreadsheets** for **DetailReports** tab. Images can be added either using the "Snapshot" option or by adding external images using the **Add Misc Item** button.



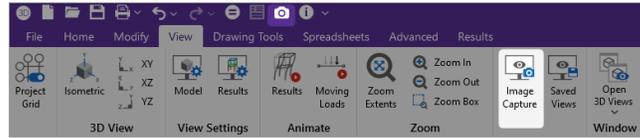
Note:

- Any saved reports from the Report Templates drop-down will currently only save spreadsheet information. Any snapshots, Detail Reports or added items from the Misc. tab will manually need to be added to any saved reports.
- The program will flag solution spreadsheets as to whether there is information available in them. If that type of solution has not been run you will see a prefix of ****NA**** ahead of that spreadsheet in the **Available Report Sections**.



Snapshot (Capture) Images

A snapshot image icon (shown in the following image) is on the **Quick Access** toolbar and the **View** tab of the ribbon.



When this icon is clicked on a folder is automatically created in the same location where the model file is located. This icon then provides some options regarding the print location and file name. Once you define this information, click **Save** and the program creates the image.



Note:

- If the model has yet to be solved the image will be created in the **Model Data Files** location from **File - Application Settings - File Locations**.
- The relative path of these image files will be saved with the model. If the model file is moved, move the associated **Images** folder as well and then all images will still be available.

This snapshot is a great way to get images of model views and detail reports that you want to add to your report. Once a snapshot is taken it will show up in the the **Images** section in the **Advanced** tab of the **Report Printing** dialog and can then be added to the **Current Sections in Report**.

Adding Images

Any additional images that you wish to add to your report that are not from RISA, can be added using the **Add Item** button. Press this button and navigate to any PNG, BMP, or JPEG file and you can select it. Once that happens it will then show up in the **Misc Items** section in the **Advanced** tab and can then be added to the **Current Sections in Report**.



Note: The complete path of these additional images are saved with the file. If the model file is moved it will still find these additional images.

Report Print Settings

Formatting settings allow you to specify your print options. The following table provides settings descriptions.

Report Print Settings

Setting	Description
Shade Lines?	This setting lets you shade every other line to enhance readability.
Start__at__Page#	This setting lets you select the starting page number. The number shown is 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.
Begin Numbering On	This setting lets you start your numbering at some page other than page one.

Setting	Description
	This can be useful if you have a title page or table of contents that you don't want to number.
Print Header	This setting lets you add a header that includes the Model Title specified in the Model Settings, company, designer, job number, and a place to initial any checking. Note that all reports include a footer with version information, the file name and path, and the page number.
Date/Time?	This setting lets you to turn off the Date/Time stamp so that this will not appear on your printed reports.
Item Options	This setting lets you to select member related options. You can specify that you want the member results to be listed for each member section (specified in the Model Settings) or just for the member ends, which can be useful for connection design.



Note: See the [Customizing](#) RISA-3D topic for more information about printing, including the ability to print out a RISA or custom company logo.

Save a Report as a Template

You can save any report you have created or modified so that it can be used again in the current model or in other models. This includes any existing custom report template, and any standard template you've modified and saved.

Save a “Customized” Standard Report

While you can customize and print a report based on a standard report template, you cannot save your customizations to that existing template. Instead, you must save the report as a “new” template with a name that differs from the original standard report's name.

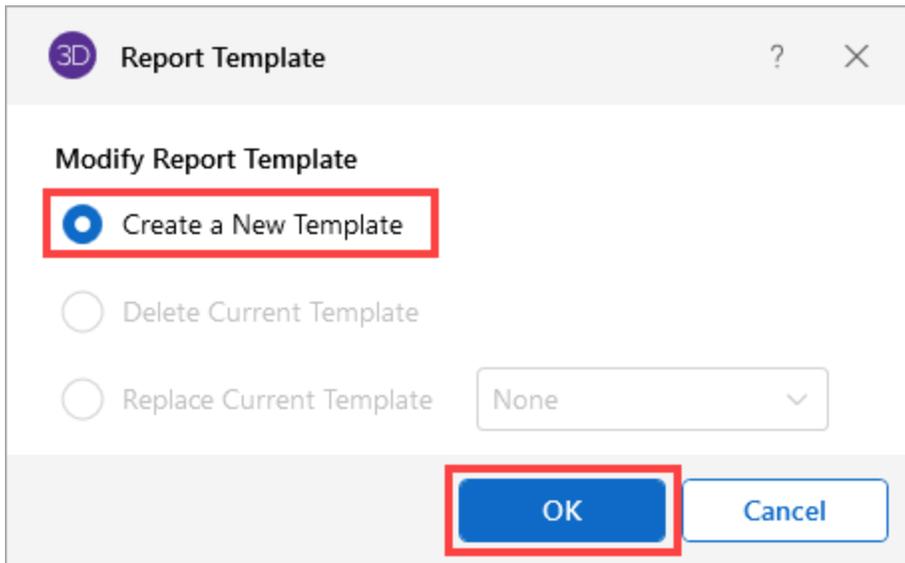
To save a customized standard report as a “new” template:

Ensure that the report includes all the pages and information you want, from the Spreadsheets, Detail Reports and Advanced tabs (if applicable).

1. Click the  ellipsis button next to the Save button.

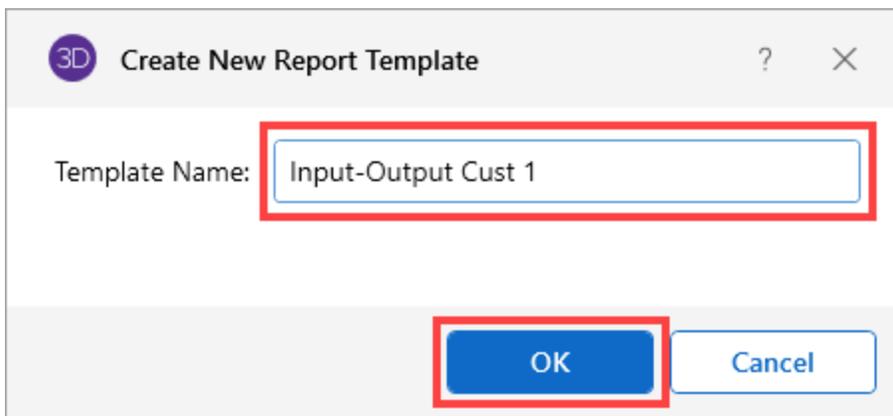


The Report Template window opens.



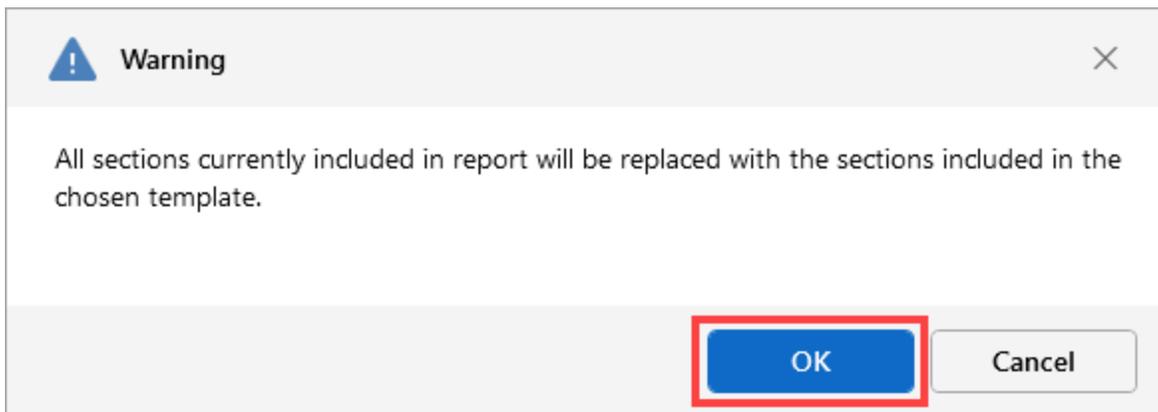
2. Choose **Create a New Template** from the Modify Report Template section.
3. Click **OK**.

A Create New Report Template window opens.



4. Click in the **Template Name** box and type a name for the customized report.
5. Click **OK**.

A warning appears.



6. Click **OK**.

The customized report is saved to the Report Templates list so you can use it for future versions of this model, or other models.

Save a New, Customized Report

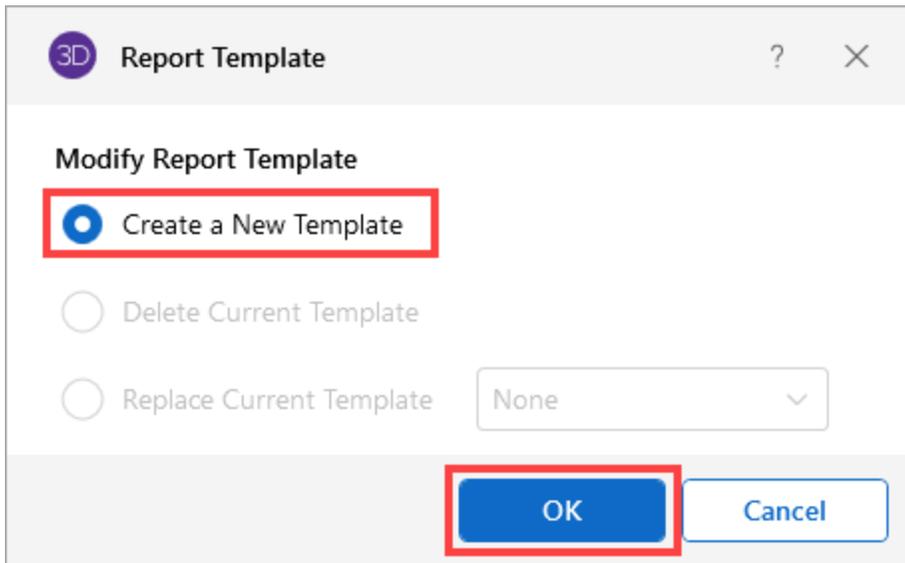
A "new" customized report is a report that you have built yourself and want to save as a template for future use on this model, or other models.

To save a new, customized report as a report template:

1. Click the  ellipsis button next to the Save button.

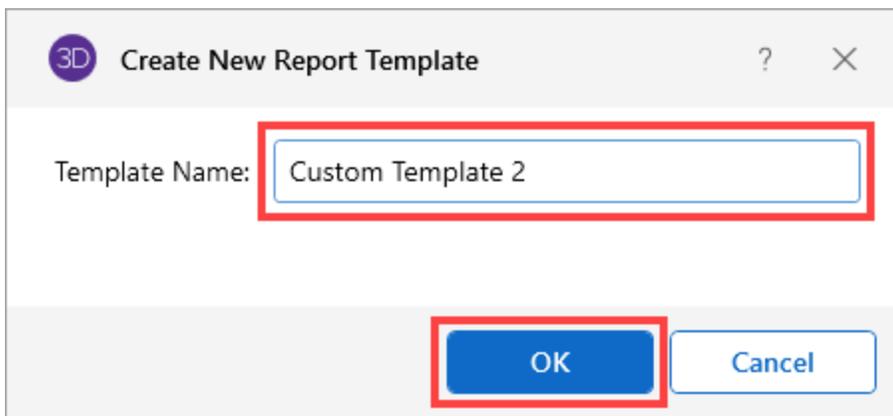


The Report Template window opens.



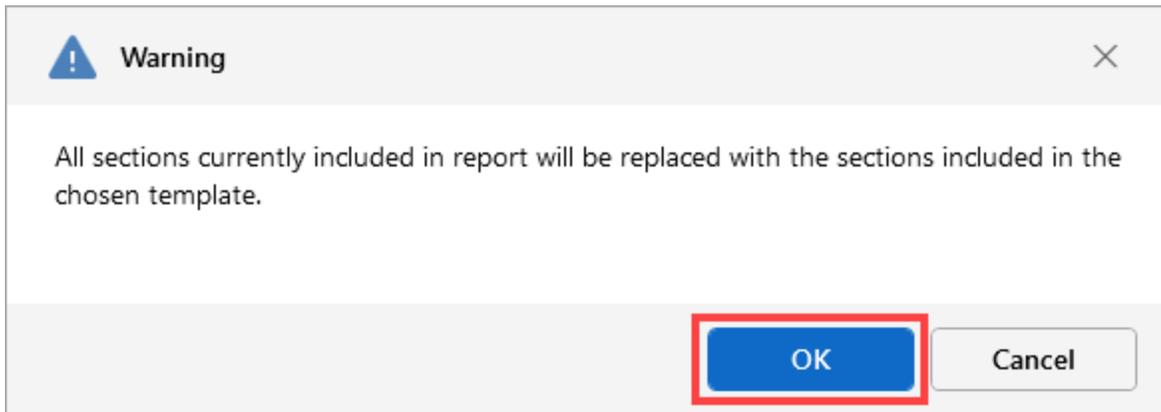
2. Choose **Create a New Template** from the Modify Report Template section.
3. Click **OK**.

A Create New Report Template window opens.



4. Click in the **Template Name** box and type a name for the customized report.
5. Click **OK**.

A warning appears.



6. Click **OK**.

A new, empty report is saved to the Report Templates list, and it's name is automatically selected in the 'Report Templates' box so you can begin to customize the report as you wish.

7. Customize the report (see [Build a Custom Report](#)) to include those items you want saved to this new report template.
8. Preview the pages of the report to ensure that all the information you want saved to the template, is included.
9. Click **Save**.



The new, empty customized report you created and is updated with your modifications and re-saved to the Report Templates list.

Save an Existing, Customized Report Template

You can customize an existing "Customized" report and then re-save it by simply choose the previously saved customized report template from the Report Templates list, modifying it and then re-saving it.

To re-save an existing customized report template that you have modified:

1. Click the **Report Templates** arrow and choose the existing customized report template you want to modify and re-save.
2. Modify the report (see [Build a Custom Report](#)) with the changes you want.
3. Preview the pages of the report to ensure you've made all the changes you want.
4. Click **Save**.



The customized report is updated with your modifications and re-saved to the Report Templates list.

Results

You can work with the results of a solution by viewing and sorting data in the spreadsheets, graphically plotting them with the model or by viewing detailed member reports. You can also print the results in any of these forms. To learn about printing results, see [Printing](#).

Upon the completion of a static solution, RISA opens the **Results** menu in the **Explorer** panel and the **Reactions Spreadsheet**. You can specify that other results be displayed automatically as well. You can then proceed to view any results and make any changes for further analysis.

If you make any changes to the model that would void the results, such as moving joints or adding members, the results will be purged and another solution will be required before the results can be viewed again.

Each of the result types is described in it's own section:

Results Spreadsheets

- [Node Deflection Results](#)
- [Node Reaction Results](#)
- [Story Drift Results](#)
- [Member Force Results](#)
- [Member Stress Results](#)
- [Member Torsion Results](#)
- [Member Deflection Results](#)
- [Hot-Rolled Code Check](#)
- [Cold-Formed Code Check](#)
- [Wood Code Check](#)
- [Concrete Design Results](#)
- [Aluminum Code Check](#)
- [Concrete Wall Results](#)
- [Masonry Wall Results](#)
- [Wood Wall Results](#)
- [Plate Stress Results](#)
- [Plate Force Results](#)
- [Plate Corner Force Results](#)
- [Modal Frequency Results](#)
- [Mode Shape Results](#)

Detail Reports

- [Member Detail Reports](#) (Hot Rolled Steel, Cold Formed Steel, & Wood)
- [Concrete Beam Detail Reports](#)
- [Concrete Column Detail Reports](#)

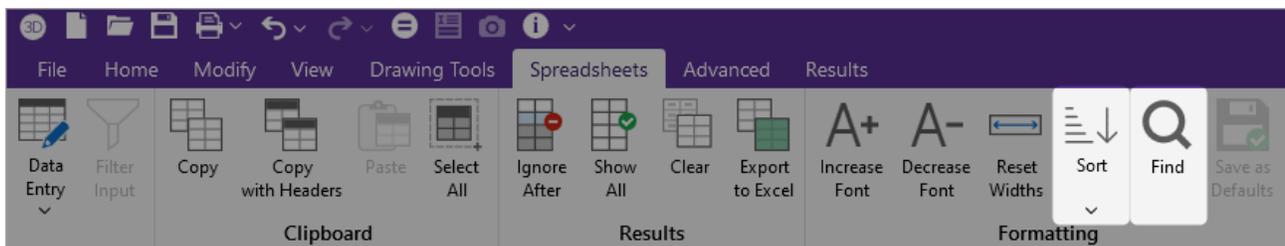
Save Results

When you save a file that has been solved you can also save the results. The next time that the file is opened the saved results will be opened as well. You can use the **Application Settings** on the **Tools Menu** to change the way that you are prompted to save results.

If changes are made to the model, any saved results are deleted. Saved results for models that no longer exist in the same directory are also deleted.

Results Spreadsheets

You can access the result spreadsheets by selecting them from the **Results Menu** in the **Explorer** panel, or from the **Results** ribbon. You can use the **Find**, **Sort**, and **Exclude** features to find the results you are interested in. For example you might sort the member stresses from high to low, bringing all of the highly stressed members to the top. You might then exclude members that do not have significant axial stresses so that they do not distract you or so that they are not printed.



Find Results

To go to a certain item while in a spreadsheet, click the **Find** icon on the **Spreadsheets** ribbon or select **Find** from the Right-Click menu, and type in the desired member, plate or joint label.

Sort Results

To sort the results, click on the column of results you wish to sort and then click the **Sort** icon on the **Spreadsheets Tab** or select **Sort** from the Right-Click menu to specify sorting settings. You can sort based on maximum, minimum, absolute maximum or input order.

Exclude Results

There are three ways to exclude results so that you can work with the results that are important to you.

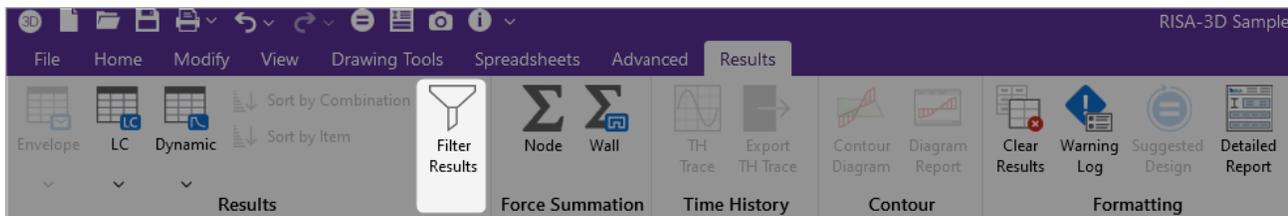
Exclude Results Before the Solution

Excluding items before the solution, lets you remove the items from the results while leaving them as part of the model. This exclusion is permanent for that solution and any exclusion changes then require another solution. This exclusion can be applied graphically in the **Properties** Panel for selected members or within the spreadsheets, by recording an “E” in the **Inactive** field of the **Members**, **Wall Panels**, and **Plates** spreadsheets.

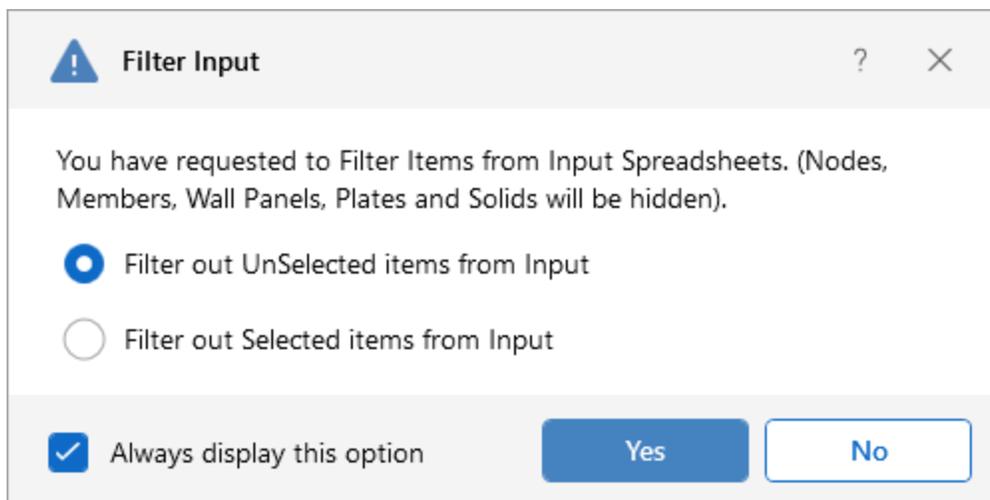
A member with an “E” in the “Inactive?” field is treated like any other member in the solution and plotting of the model, but the member is not listed in the solution results (forces, stresses, deflections, etc.). This is useful if there are certain members whose results aren't of interest. You don't have to clutter up the results with these members and can concentrate on the members you are most interested in.

Exclude Results After the Solution

You can run the solution and then graphically select the joints, members, and plates that are of interest. By clicking the **Filter Results** button and in the pop-up window, you can select **Filter Out UnSelected items from Results** or **Filter out Selected items from Results**. You can then update the spreadsheets and printed reports so that they will not have results for the selected or unselected items. All of the spreadsheets and reports will be controlled by this selection and you can adjust this selection at any time.



This option is also available by clicking the **Filter Results** button  from the Results toolbar in the 3D View.



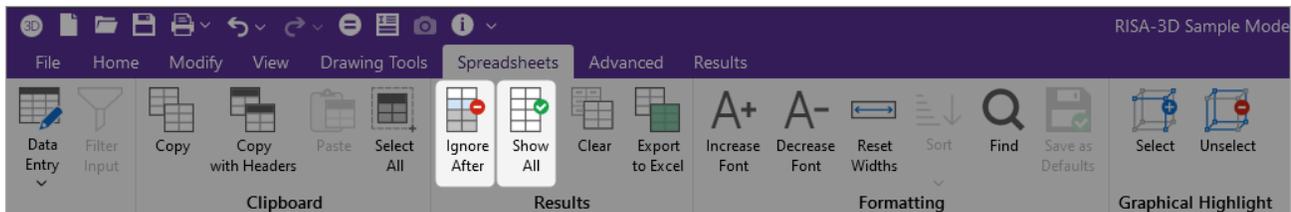
When you open any results spreadsheet, you can tell if it has been filtered by the filter icon in the spreadsheet header.

Member Section Forces (By Combination)									
Sections	Maximums	End Reactions	2nd/1st Moment Ratios						
	LC	Member Label	Sec	Axial[k]	y Shear[k]	z Shear[k]	Torque[k-ft]	y-y Moment[k-ft]	z-z Moment[k-ft]
1	1	M195	1	23.64	-0.65	-0.563	0	0	0
2			2	23.132	-0.65	-0.563	0	-1.689	1.95
3			3	22.625	-0.65	-0.563	0	-3.377	3.901
4			4	22.117	-0.65	-0.563	0	-5.066	5.851
5			5	21.61	-0.65	-0.563	0	-6.754	7.802

Exclude Results While in Spreadsheets

You can also select the results of interest on each spreadsheet. While viewing spreadsheet results you can select the last line of interest and exclude the rest. Exclusions are applied independently for each results spreadsheet. Any exclusion applied to the Member Forces spreadsheet will not affect the Member Stresses spreadsheet, etc. Excluding items graphically will reset **all** of the spreadsheets to match the graphic selection. If you wish to combine these two features to fine tune the results perform the graphic selection first.

To exclude results click on the last line of results you wish to keep and then click the **Ignore After** button under the **Spreadsheets** Tab. You can bring the results back by clicking on the **Show All** button. Alternatively, you can select these features from the Right-Click menu in any result spreadsheets.



Note: It might be best to first sort the results before excluding. As an example, let's say that you only wanted to view members with Code checks greater than or equal to 0.7. To do this you would open the Steel Code Checks spreadsheet and sort the members by Code Check magnitude. Then you would visually identify the last member with a Code Check of 0.7 and use the Exclude feature to exclude all members after that one.

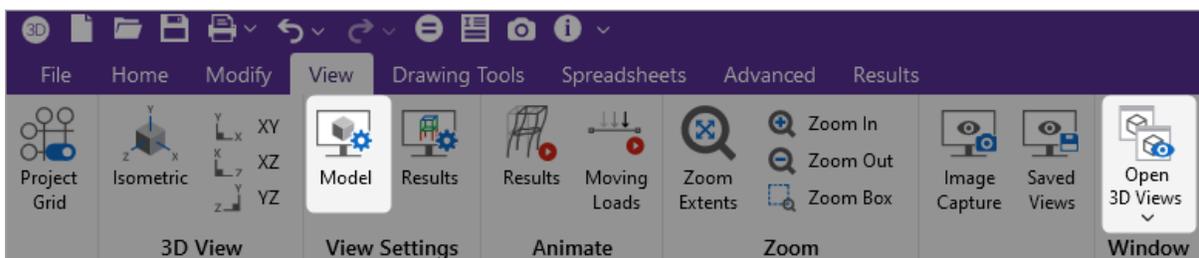
Graphic Results

Most of the analysis results may be viewed graphically as well as in the spreadsheets. For the joints you can plot the reactions. For the members you can plot force diagrams as well as color-code the plotted members by code check or stress levels. Plate stress contours and corner forces may also be viewed graphically. Deflected shapes and mode shapes may be viewed and animated. See [Model Display Options](#) for more information.

Plot Results Graphically

To plot results graphically:

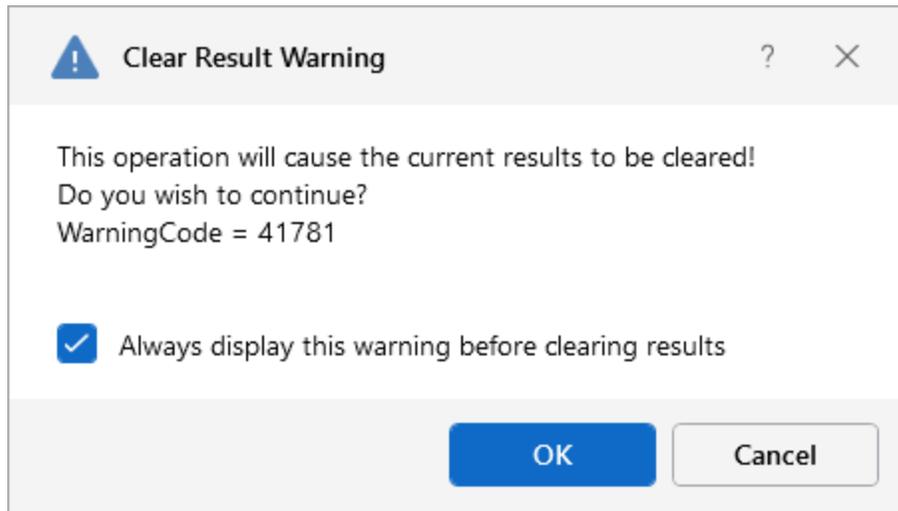
1. If you do not already have the model view open, click the **Open 3D Views** icon on the **View** ribbon.



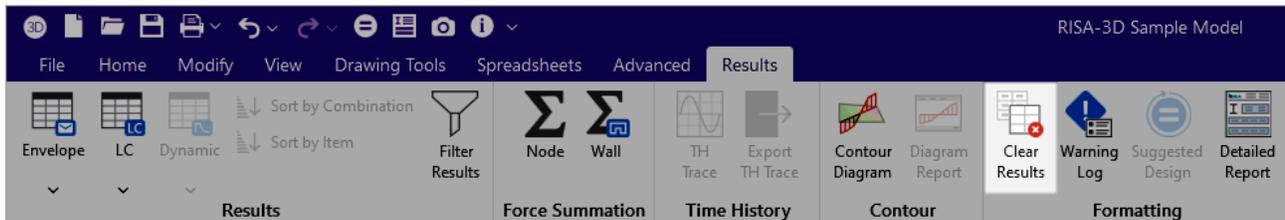
2. Click the **Model** icon on the **View** ribbon for model view settings.
3. Select the options you would like to view.
4. Click **Apply** or **OK**.

Clear Results

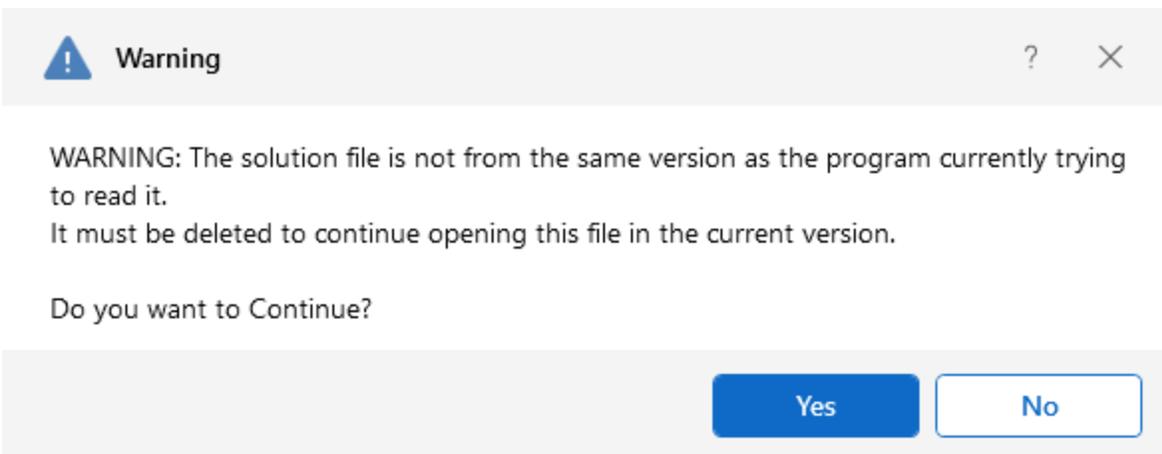
You will be provided with a warning if changes are made to the model that might invalidate the current results. Should you decide to proceed, the results will automatically be cleared and you will have to re-solve the model to get results once you are finished making changes. This warning may be disabled in the **Application Settings** under the **3D** button.



To manually clear results, click the **Clear Results** icon.

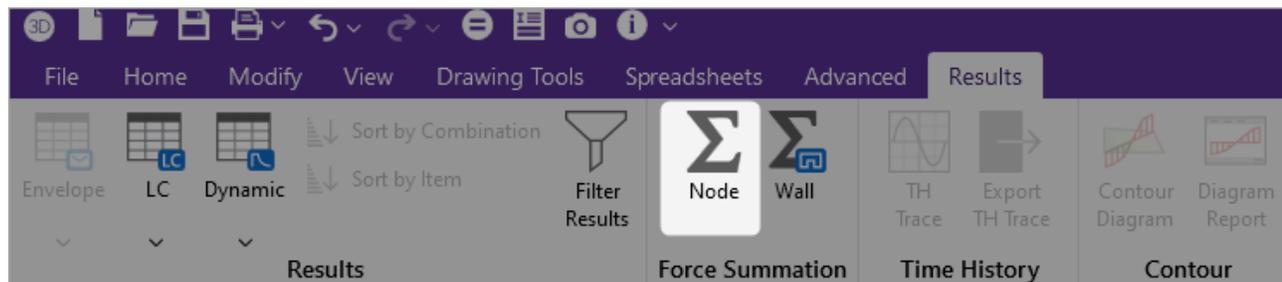


Opening a model with a solution file that was saved in a different version will also prompt you to delete the solution file in order to open the model.



Internal Force Summation Tool

This tool may be activated by clicking the icon under the Results tab. The tool is only available from within a model view that has active results for a single combination or a batch solution. This tool is not currently available for an envelope solution.



This tool was originally created to convert Plate Analysis results into design forces for concrete slabs and walls. However, the tool is equally useful for structure that have beams, columns and braces. It is frequently used to determine overall story shears and moments. For detailed information on the tool, please refer to the [Plate Design Tools](#) section.

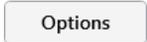


Note: The internal force summation tool does NOT currently work on Solid Elements.

Member Detail Report

The detail report allows you to see the overall force/stress/deflection state for any particular member. This report shows diagrams for all these quantities and also lists detailed information on the code checks for Hot Rolled Steel, Cold Formed Steel, Concrete, Aluminum and Timber design.

This detail report is available for any member following the solution. If you have run an envelope solution, then you will only be able to view an enveloped detail report. If you run a Batch + Envelope solution, you can scroll between individual (Batch) load combination results, and the Enveloped results.

- If there is a (y) or (z) value next to the code check, it defines whether the controlling case is in the local y or z axes respectively.
- If there is an (s) this tells you that the maximum check occurred at some other axes where a round shape is used and the cross-sectional properties for the member are the same in all directions.
- If you are viewing an enveloped detail report, the governing load combination number will be reported in parenthesis. Ex: (LC 10).
- When a force or stress shows up with multiple colors (see image below), this signifies the difference between the total value and the value when Live Load Reduction is considered.
- The viewing of Live Load Reduction within the diagrams or the consideration of LL reduction in the reported values can be turned on or off by clicking on the  button.
- For members designated as beams, the Max Defl Ratio will report the maximum beam deflection ratio (L'/y') as reported in the [Beam Defl](#) tab of the Member Deflections spreadsheet.

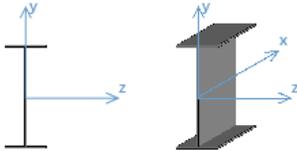
- For members that are not designated as beams, the Max Defl Ratio will report the maximum member deflection ratio (L/y') as reported in the [Service](#) or [Strength](#) tab of the Member Deflections spreadsheet.

Detail Report for - M70 — □ ×

< Member Label M70 >
LC 1: IBC 16-1 ▾
Options

Detail Report: M70

Load Combination: LC 1: IBC 16-1 Code check: 0.293 (axial/bending)



Input Data			
Shape:	W10X12	I Node:	N419
Member Type:	Beam	J Node:	N420
Length (ft):	18	I Release:	BenPIN
Material Type:	Hot Rolled Steel	J Release:	BenPIN
Design Rule:	Wide Flange	I Offset:	N/A
Internal Sections:	97	J Offset:	N/A
Design Code:	AISC 14th (360-10); LRFD	T/C Only:	Both Way

Material Properties			
Material:	A992	Therm. Coeff. (/1E5 F):	0.65
E (ksi):	29000	Density (k/ft ³):	0.49
G (ksi):	11154	Fy (ksi):	50
Nu:	0.3	Ry:	1.1
Fu (ksi):	58	Rt:	1.2

Shape Properties			
d (in):	9.87	Area (in ²):	3.54
bf (in):	3.96	Zyy (in ³):	1.74
tf (in):	0.21	Zzz (in ³):	12.6
tw (in):	0.19	Cw (in ⁶):	50.9
Iyy (in ⁴):	2.18	Wno (in ²):	9.56
Izz (in ⁴):	53.8	Sw (in ⁴):	1.99
rT (in):	0.955	J (in ⁴):	0.055
kdet (in):	0.75	kdes (in):	0.51

Design Properties			
Lb y-y:	Segment	Ky-y:	1
Lb z-z (ft):	18	Kz-z:	1
Lcomp top (ft):	1	y sway:	No
Lcomp bot (ft):	18	z sway:	No
Ltorque (ft):	18	Function:	Gravity
Cp:	1	Seismic DR:	None
Max Defl Ratio:	L/421	Max Defl Location:	9
Span:	1	τb:	1

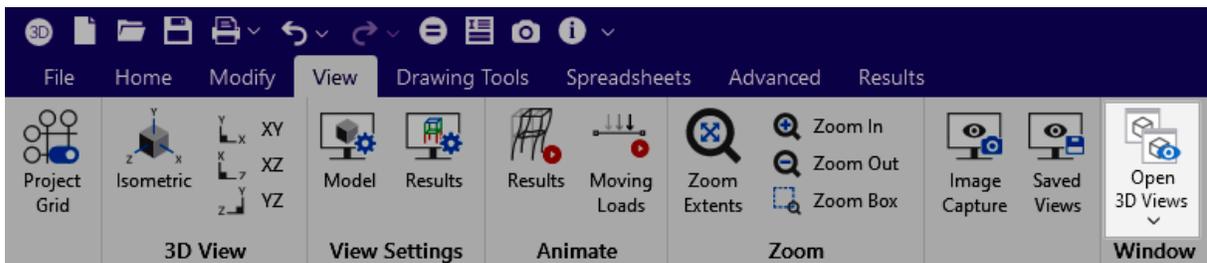
Print
 Add to Full Report
Close

View a Detail Report

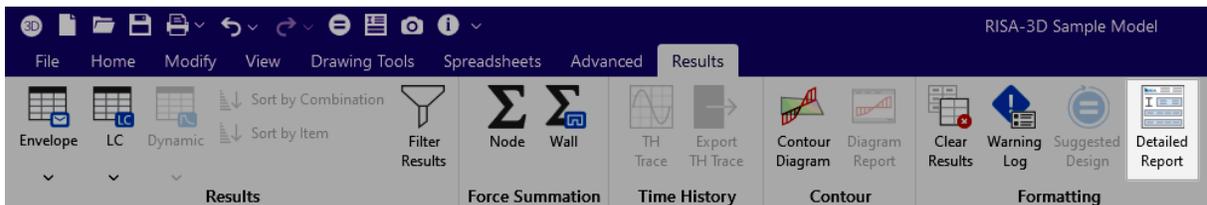
There are three ways in which you can view a Detail Report: from the Quick Access toolbar, from the right-click menu when in a spreadsheet and from the Results ribbon. Please note that you must run a solution prior to a Detail Report being available.

- **Quick Access toolbar:** Click on the Detailed Report  button on the Quick Access toolbar and then click on that member.

If you do not already have the model view open, click the **Open 3D Views** icon on the **View** ribbon.

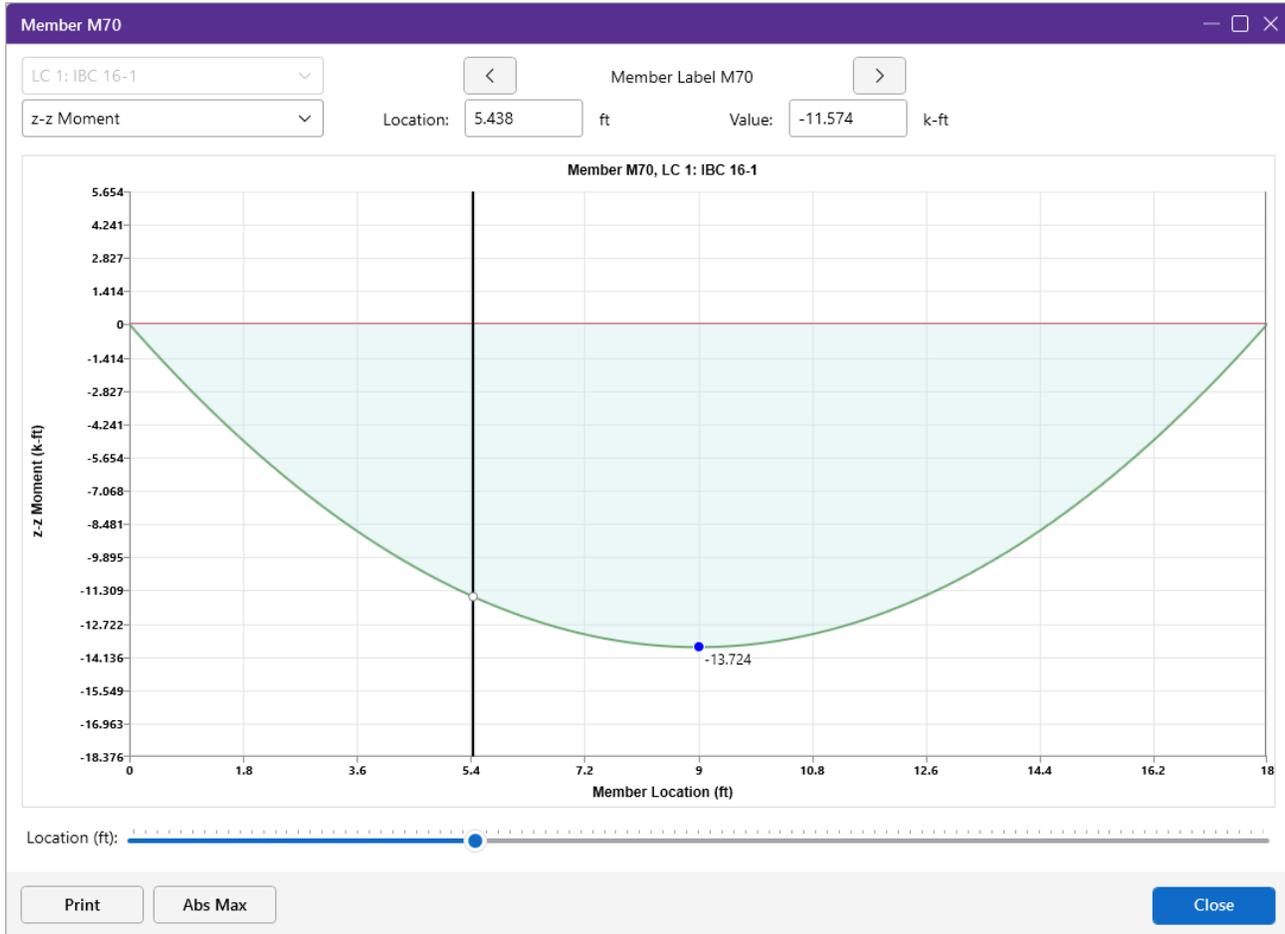


- **Right-click menu:** While viewing a member results spreadsheet, such as **Member Stresses**, click on the **Detailed Report** option from the Right-Click menu.
- **Results ribbon:** Click the **Detailed Report** icon on the **Results** ribbon and then click on a member to open the report for that member.



Interactive Diagrams

Once a detail report is open, you can also click on any of the force, stress, or deflection diagrams to open an interactive diagram. Click on any place along the length of the member to see how the value changes.



The diagrams provided are:

Plot Designation	Plotted Value
A	Axial Force
T	Torsional Moment
Vy	Shear Force Parallel to y-y Axis
Vz	Shear Force Parallel to z-z Axis
My	Bending Moment About y-y Axis
Mz	Bending Moment About z-z Axis
Dy	Deflection in Local y Direction
Dz	Deflection in Local z Direction
fa	Axial Stress
fc	Bending Compressive Stress
ft	Bending Tension Stress

Plot Designation	Plotted Value
f(y) and f(z)	(Envelope Only)

The "fa" stresses are the stresses resulting from the axial force. The "fc" and "ft" stresses incorporate both bending moment and torsional normal warping stresses. In the envelope detail report, the "f(y)" and "f(z)" stresses display both the maximum positive and negative bending stress located at the extreme fiber of the member in the local y- and z-axis. These stresses are listed in the Member Stresses spreadsheet as z-Top, z-Bot, y-Top and y-Bot.

The diagrams are scaled in groups to give a good representation of relative values. For example, the force diagrams (A, Vy and Vz) are scaled such that the force of maximum magnitude fills the diagram space. The other diagrams are then plotted using that same scale. The moments, deflections and stresses are similarly scaled together.

When an output file is saved, the program will discard much of the unneeded force, moment and deflection data used to create the detail report plots. When this saved file is later retrieved, these plots will appear more coarse and inexact. However, the maximums, minimums and controlling code checks are always maintained regardless of how coarse the plots appear.



Note:

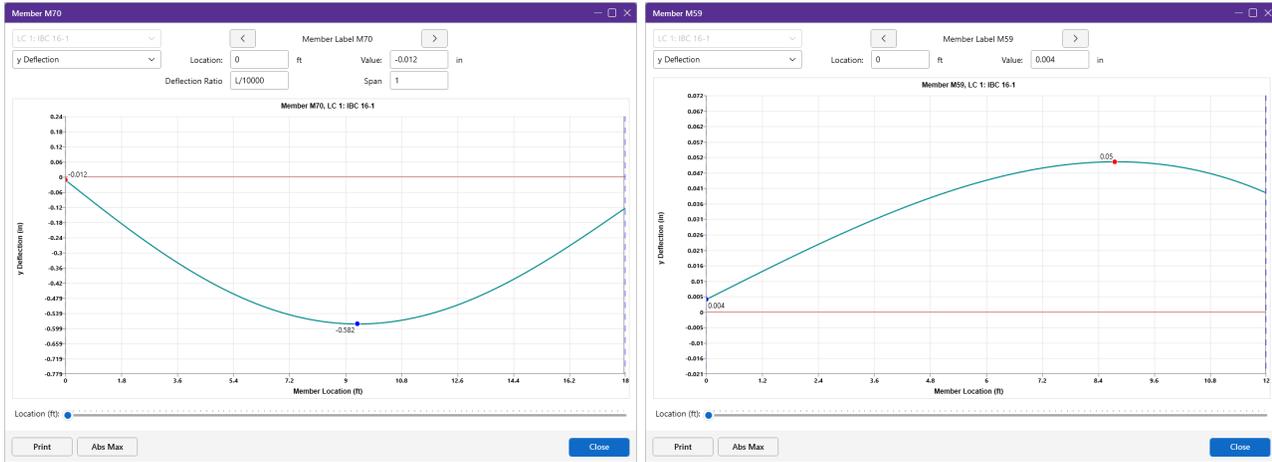
- When a force or stress shows up with multiple colors, this signifies the difference between the total value and the value when Live Load Reduction is considered.
- The consideration of LL Reduction is only included for models that are integrated with RISAFloor.
- The consideration of LL reduction in the displayed diagram may be turned on or off by toggling the **LL Reduction** check box.

Detailed View for Deflection Diagrams

Only members that have been designated as beams will also report the Defl Ratio in the detailed deflection diagram. This value is the beam deflection ratio which can also be seen in the [Beam Defl](#) tab of the Member Deflections spreadsheet and the summary at the end of the [Member Detail Report](#).

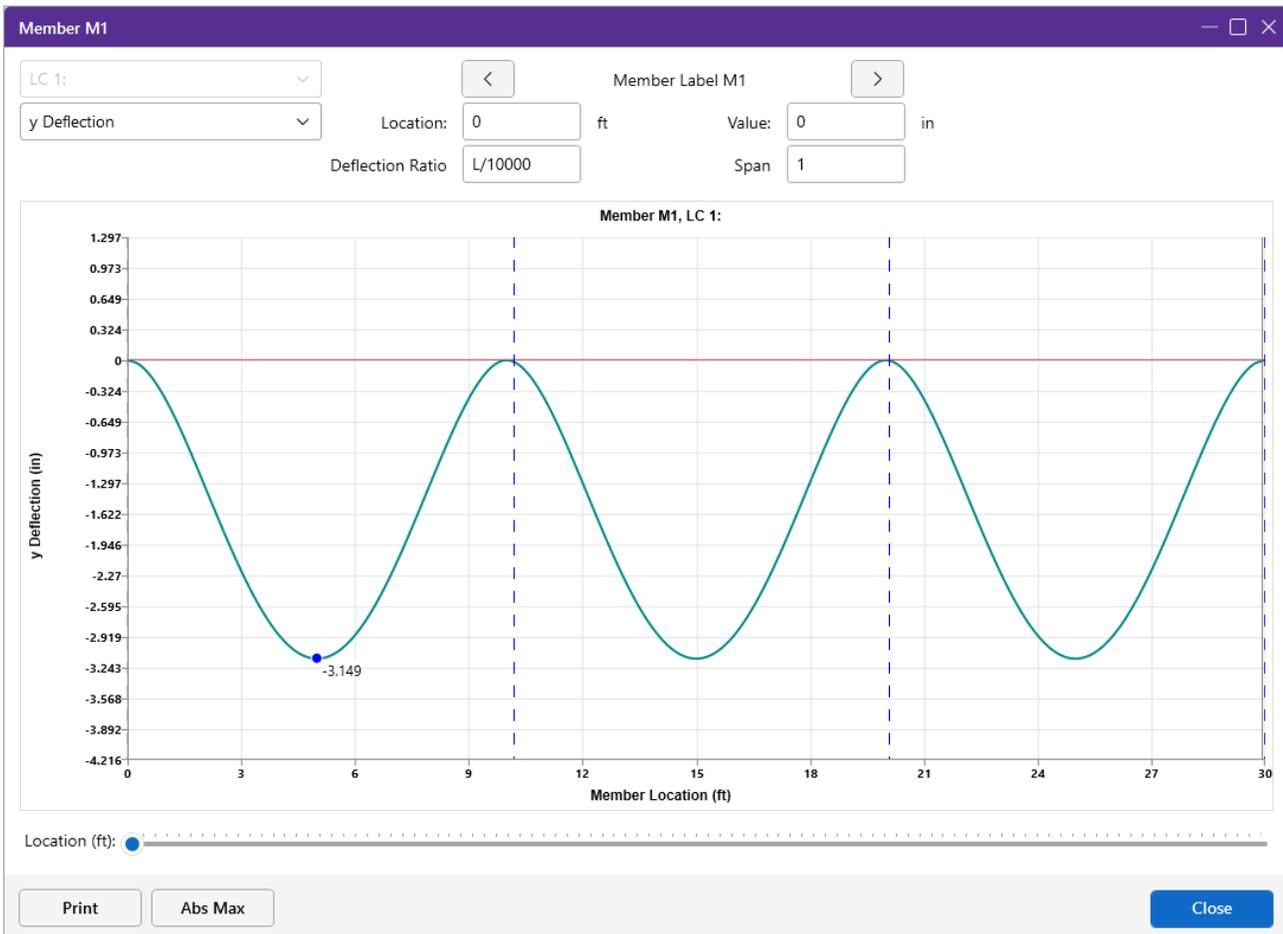
Beam Members

Other Members



Multi-Span Detailed View for Deflection Diagrams

For multi-span beams the detailed deflection diagram will give span information and show a dashed line at the support locations. The Defl Ratio is based on the Span Length of the current span.



Tapered Members

The Detail Reports for tapered members designed to AISC Design Guide 25 contain information not available in other Hot Rolled Steel Design Reports. Refer to the [Tapered Member Results](#) sub-topic of Hot Rolled Steel Design for more information.

Envelope Member Detail Reports

The Detail Reports can be viewed for single load combinations, or as an envelope. If you solve a Batch + Envelope solution, you can toggle between the overall enveloped results and the individual load combination results using the menu at the top left.

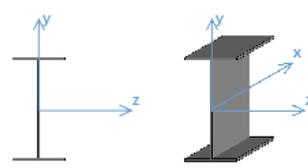
Detail Report for - M1 — □ ×

< Member Label M1 >

Envelope ^

- Envelope
- LC 1: Deflection 1
- LC 2: Deflection 2
- LC 3: Deflection 3
- LC 4: IBC 21/ASCE Strength 1
- LC 5: IBC 21/ASCE Strength 2 (a)
- LC 6: IBC 21/ASCE Strength 3 (b)
- LC 7: IBC 21/ASCE Strength 4 (a)
- LC 8: IBC 21/ASCE Strength 5

Options



check: 9.999 (LC 4)

I Node:	N1
J Node:	N2
I Release:	Fixed
J Release:	Fixed
I Offset:	N/A
J Offset:	N/A
T/C Only:	Both Way

Material Properties

Material:	A992	Therm. Coeff. (1/E5 F):	0.65	F_u (ksi):	65
E (ksi):	29000	Density (k/ft³):	0.49	R_t:	1.1
G (ksi):	11154	F_y (ksi):	50		
Nu:	0.3	R_y:	1.1		

Shape Properties

d (in):	7.89	Area (in²):	2.96	r_T (in):	1
b_f (in):	3.94	Z_{yy} (in³):	1.66	J (in⁴):	0.043
t_f (in):	0.205	Z_{zz} (in³):	8.87	k_{det} (in):	0.688
t_w (in):	0.17	C_w (in⁶):	30.9	k_{des} (in):	0.505
I_{yy} (in⁴):	2.09	W_{no} (in²):	7.57		
I_{zz} (in⁴):	30.8	S_w (in⁴):	1.53		

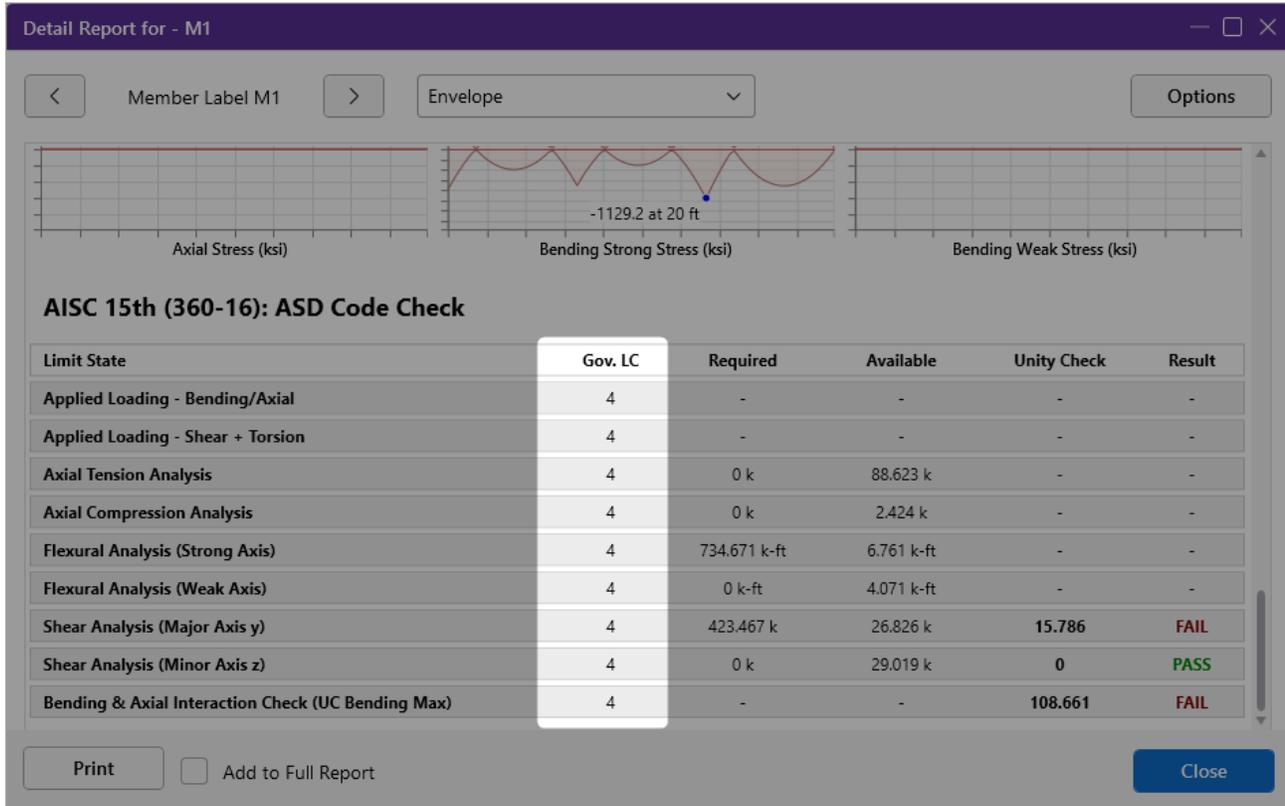
Design Properties

L_{b y-y} (ft):	30	K_{y-y}:	1	Seismic DR:	None
L_{b z-z} (ft):	30	K_{z-z}:	1	Max Defl Ratio:	L/27
L_{comp top}:	L _{b yy}	y sway:	No	Max Defl Location:	5
L_{comp bot} (ft):	30	z sway:	No	Span:	1
L_{torque} (ft):	30	Function:	Lateral	τ_b:	1

M1

Print
 Add to Full Report
Close

For the enveloped report, all diagrams will show enveloped data. Governing load combinations will be noted in the header of each load check.



 **Note:** If you solve a load combination with a Moving Load or a Time History Load, the results will only be available in the enveloped detail report..

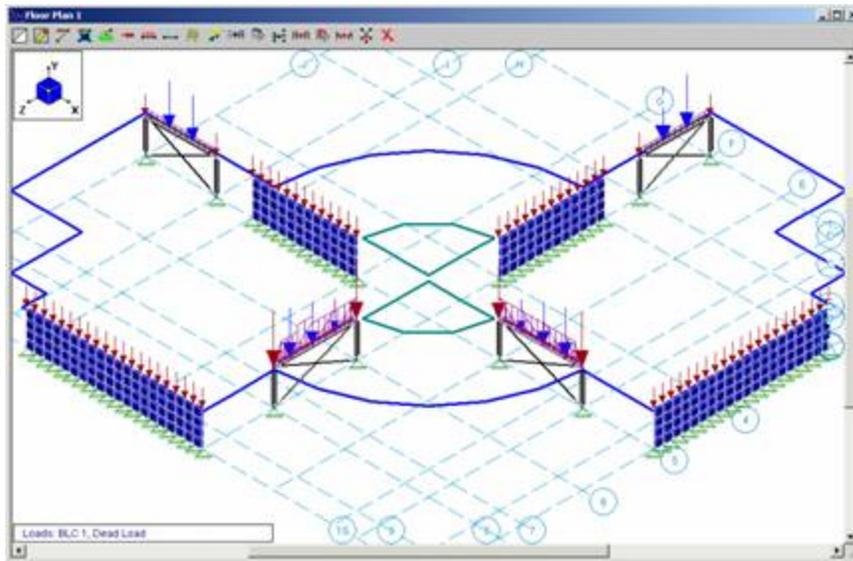
Concrete Member Detail Reports

Concrete member detail reports are similar in function to detail reports for other materials, but they are different in the type and amount of information they convey. One of the largest differences is that the force diagrams are always **envelope** force diagrams, because the majority of the concrete design results are based on the envelope forces.

The detail reports for concrete Beams and Columns are discussed in greater detail in the [Concrete - Design Results](#) section.

RISAFloor & RISA-3D Integration

While the primary function of RISAFloor is to create and optimize floor systems, another strength is that it can be used to automatically generate a model of the lateral force resisting system in RISA-3D.



Note: The features described in this section are only available to users who are running both RISAFloor and RISA-3D.

Lateral System Model Generation

Beams, columns, and walls whose function is set to "Lateral" on the **Primary Data Tab** of their respective spreadsheets in **RISAFloor** will automatically be generated in the RISA-3D model when accessed via the **Director** menu. To access this model, click the **Director** button on the far right end of the Main Menu and choose **RISA-3D**. The RISA Application Interface will then switch from RISAFloor to that of RISA-3D.

Once in RISA-3D you will notice that you can use the RISA-3D features to edit and solve the model. You can add braces, beams, columns, walls, and additional loads just as you would in a regular RISA-3D model. Refer to the RISA-3D General Reference Manual and Tutorials for documentation of RISA-3D's features.

The "gravity" model in RISAFloor and the "lateral" model in RISA-3D are fully linked. Subsequently, any changes made to RISAFloor generated members in the RISA-3D model will automatically update those same members in RISAFloor model.

**Note:**

- Beams, columns, and walls whose function is set to "Gravity" in RISAFloor will NOT be generated in RISA-3D for optimization. These members have been indicated as "gravity-only" members and will not collect any lateral load in the RISA-3D model. Therefore, these members would only "clutter" the lateral system model in RISA-3D and are subsequently not drawn.
- RISAFloor "Gravity" members may be viewed in RISA-3D via the general tab of the model view settings dialog. These members will be displayed for visual effect only in the model view but will not contribute to the stiffness of the RISA-3D model.
- The unbraced lengths of RISAFloor Lateral members are coordinated with those in the RISA-3D model.
- Members explicitly defined in RISAFloor will have a design list of 'None' in RISA-3D to prevent alternate shapes from being suggested.
- Member end releases in the RISA-3D model are controlled by RISAFloor, and cannot be changed in RISA-3D. To modify these, go to the Beams or Columns spreadsheet in RISAFloor.

Diaphragms

Diaphragms are created in RISA-3D for every floor with a defined diaphragm edge in RISAFloor. Although you cannot delete these diaphragms, you can make them **inactive** in RISA-3D. The **Mass**, **Mass Moment of Inertia** and **Center of Mass** are automatically calculated based on the RISAFloor loads and on the settings in [Model Settings](#). For more information on this interaction, see the [Diaphragms](#) topic.

Floor Diaphragms																	
General																	
	Elevation [ft]	Mass [k]	Mass MOI [k-ft ²]	Center Of Mass [ft]	Plus X Eccentricity [ft]	Minus X Eccentricity [ft]	Plus Z Eccentricity [ft]	Minus Z Eccentricity [ft]	Inactive	Diaphragm	Type	Region	Design Rule	SGAF	Material	Thickness [in]	No Wind/Drift
1	30	608.897	3.295e+6	195.586, 75.027	%5	%5	%5	%5	<input type="checkbox"/>	D2	Flexible		N/A	N/A	N/A	N/A	<input type="checkbox"/>
2	30	643.261	1.414e+6	133.374, 59.571	%5	%5	%5	%5	<input type="checkbox"/>	D4	Rigid		N/A	N/A	N/A	N/A	<input type="checkbox"/>
3	30	389.734	2.735e+5	24.54, 153.482	%5	%5	%5	%5	<input type="checkbox"/>	D5	Flexible		N/A	N/A	N/A	N/A	<input type="checkbox"/>
4	15	3830.637	3.333e+7	116.295, 116.047	%5	%5	%5	%5	<input type="checkbox"/>	D3	Semi-Rigid		N/A	N/A	gen_Conc3NW	12	<input type="checkbox"/>

The **X and Z eccentricities** are used in the equivalent lateral force method for calculation of seismic loads. This allows you to quickly and easily account for the effects of accidental torsion when calculating your seismic response.

The **Diaphragm** and **Region** columns are the names of diaphragms and regions in the model.



Note: Diaphragms defined as rigid are not designed, thus the Region column is blank.

The **Type** allows you to toggle between flexible, semi-rigid, and rigid for Beam Supported Floors (can only be toggled on RISAFloor side). The Concrete Supported floors can be Rigid or Semi-Rigid and the type is defined in the Slab Definitions spreadsheet in RISAFloor.

The **Design Rule** allows you to switch between design rules for the diaphragm regions. The [Design Rules spreadsheet](#) is where many parameters are defined for your diaphragm. For more information on diaphragm modeling and interaction, see the [Diaphragms](#) topic.

Concrete Floor Slabs

Concrete Floor Slabs (created with RISAFloor ES) are transferred into RISA-3D either as Rigid diaphragms or Semi-Rigid diaphragms.

Rigid diaphragms have no out-of-plane stiffness therefore it would be recommended that you model lateral beams in RISAFloor with the Rigid diaphragms. The interaction of slabs and beams is ignored with respect to the analysis and design of lateral concrete beams in RISA-3D.

Semi-Rigid diaphragms are automatically modeled in RISA-3D with a plate sub-mesh representing the slab. There is in-plane and out of plane stiffness for a Semi-Rigid diaphragm.

Gravity Loads

The gravity loads on the lateral members, including the beam reactions from the gravity only members, become part of the RISA-3D model. The Load Categories in RISAFloor are automatically converted into Basic Load Cases in RISA-3D. The exceptions to this are the Load Categories that deal specifically with construction loads (DL Const, and LL Const), which are not converted.

Wind Loads

Wind loads can be automatically generated per various codes.

See [Load Generation - Wind Loads](#) for more information.

Seismic Loads

Seismic loads can be automatically generated per various codes.

See [Load Generation - Seismic Loads](#) for more information.

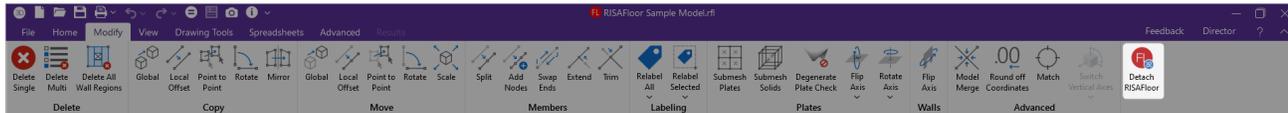
Buckling-Restrained Braced Frame (BRBF) System

Buckling-restrained brace (BRB) design is available for models that come from RISAFloor to RISA-3D. The gravity members can be drawn and designed in RISAFloor. Users can then direct the model to RISA-3D and add BRB members from specific databases (e.g. CoreBrace database). The BRB-related databases are only available for RISAFloor-RISA-3D integration. BRB members and design are not available for RISA-3D only models. If an existing model with BRB shape is detached in RISA-3D, the BRB shapes will be converted to a generic shape RE1x1.

The integration of RISAFloor and RISA-3D fits well with the split gravity and lateral design requirements for the BRBF system. The framing members such as columns and beams can be designed in RISAFloor under gravity loads to meet the requirements of them resisting gravity loads

without BRB braces. After the model is directed to RISA-3D, users can add BRB braces to perform the lateral load design for the whole system.

Detach



On the Modify ribbon, the **Detach RISAFloor** icon lets you detach the RISA-3D model from the RISAFloor model. This deletes any diaphragm information and generated loads.

RISAFoundation Interaction

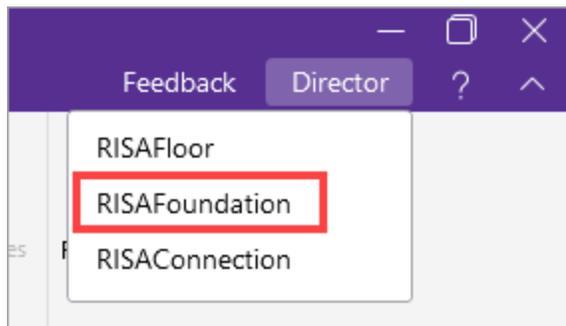
RISAFoundation has the ability to transfer load information from RISA-3D and RISAFloor and use this information to design foundations. The following entry will describe how these loads come across the design modules so you can accurately use RISAFoundation for your foundation design.

RISAFoundation Interaction with RISA-3D

RISAFoundation transfers joint reactions from RISA-3D into RISAFoundation by way of [load categories](#). This is not the same as load combinations. Load categories are defined in the Basic Load Cases spreadsheet of RISA-3D.

To get your reactions to transfer from RISA-3D to RISAFoundation:

1. Loads must be split into specific load categories (DL, LL, etc.) under the Basic Load Cases spreadsheet.
2. The model must be solved.
3. Click on **Director** and choose **RISAFoundation**, to transfer to RISAFoundation.



**Note:**

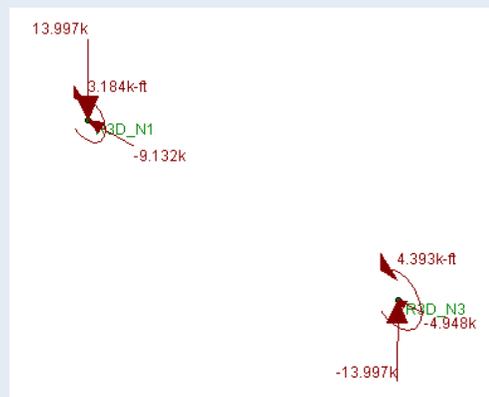
- If you do not define a load category for a basic load case, then the loads from that case will not be transferred into RISAFoundation.
- You must run at least a single load combination.
- When you are working between RISA-3D and RISAFoundation Model Settings will only be brought over to RISAFoundation the first time you transfer from the RISA-3D module. After this first transfer, any time you change something in RISA-3D from Model Settings that would affect RISAFoundation, then you would also have to go into RISAFoundation and make the change.
- Boundary conditions defined as **Reaction** or **Spring** will be transferred but **Fixed** boundary conditions will not.
- Wall reactions will be brought into RISAFoundation as a series of point loads along the base of the wall. The point reactions are taken from the location of the base of the wall mesh from the RISA-3D model.

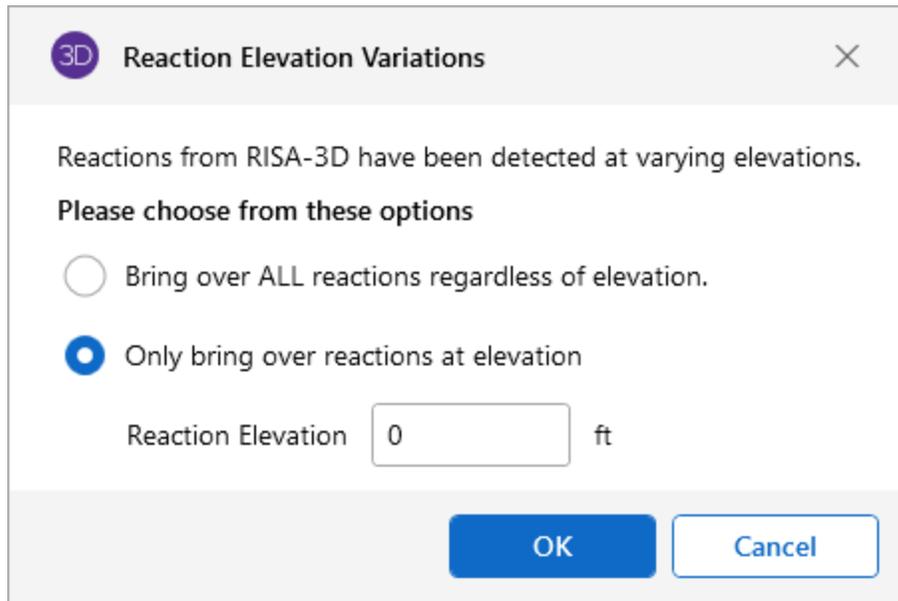
Example of RISA-3D and RISAFoundation Interaction

A simple example would be the braced frame structure seen below on the left. This has been created in RISA-3D. The load applied to this structure came from Load Category WL (wind load). Note that this structure had other loads applied as well. But the wind loads are used as an example.

RISA-3D Wind Loads Graphical Representation

RISAFoundation Wind Loads Graphical Representation





You can choose to bring over reactions at a single elevation of your choice. This may be helpful if there are some reactions that are due to an adjacent structure or support that is not a foundation.

You can choose to bring over all reactions regardless of elevation. This is a good option to have if there is a partial basement level or any condition where there truly are foundations at different levels.



Note: All of the differing elevations will be brought in at a single elevation.

RISAFoundation Interaction with RISAFloor

This interaction is a little more involved because it involves three programs (RISA-3D, RISAFloor and RISAFoundation). For RISAFoundation, the bases of columns and the bottom of walls will be considered boundary conditions. Thus, loads are applied at the bases of these elements for foundation design in RISAFoundation.

To transfer from RISAFloor to RISAFoundation you must pass through RISA-3D. Please see the [RISA-3D to RISAFoundation](#) procedure.

Gravity Elements in RISAFloor

Elements (columns, walls) defined as gravity only are the only elements which transfer load directly from RISAFloor into RISAFoundation. Wall reactions come in as line loads and column reactions come in as point loads.

Wall line load reactions are brought in according the overall moment, M , and axial force, P in the overall wall. The wall moment is taken as a summation of $\text{force} \times \text{distance}$ about one end of the wall. Once P and M are calculated, the magnitudes at either end of the wall are calculated as:

$$I_{end} := \frac{4 \cdot P}{L_{wall}} - \frac{6 \cdot M}{L_{wall}^2} \qquad J_{end} := \frac{6 \cdot M}{L_{wall}^2} - \frac{2 \cdot P}{L_{wall}}$$

Lateral Elements in RISAFloor

When applying loads to lateral members within RISAFloor, those loads are simply attributed to the members brought over into RISA-3D. Thus, RISAFoundation does not bring any information in from RISAFloor concerning lateral members. All of this information is taken directly from RISA-3D. From this point we are just working as if we only had RISA-3D and RISAFoundation working together. See [above](#).

Limitations

RISAFoundation can not recognize moving loads

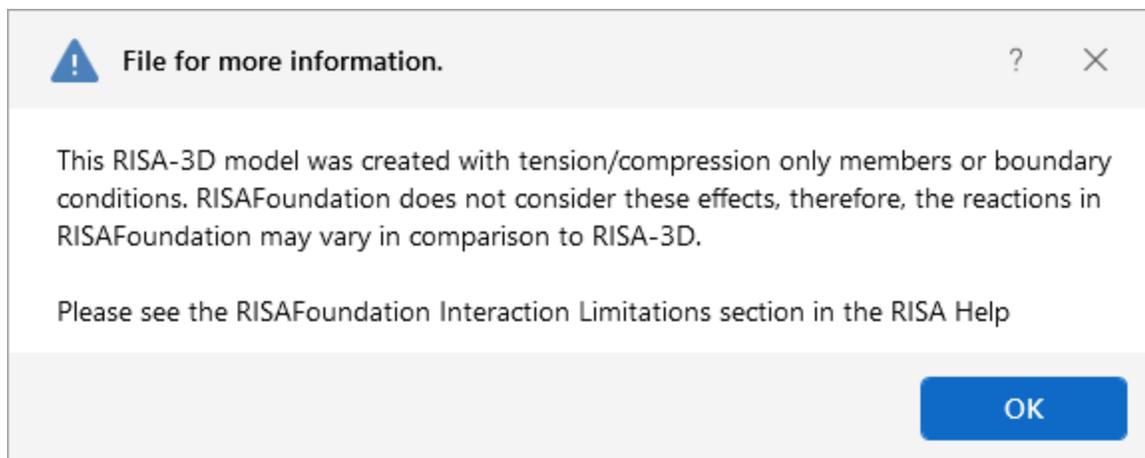
Moving Loads, not described as a load category, and their reactions can't be read as loading within RISAFoundation, thus loads will need to be applied manually to the foundations to account for these.

RISAFoundation requires your Y-axis to be vertical

Models coming from RISA-3D need to have the Y-axis to be vertical to be brought into RISAFoundation. If the Y-axis is not the vertical axis, then a dialog box will pop up when you try to link to RISAFoundation with this message.

RISAFoundation will not consider non-linear load effects

In a true Non-Linear analysis (with Tension Only or Compression Only members), the individual reactions would vary depending on which LC was used to create it because different members may be active for different LC's. In that manner the Load Category reactions that make their way into RISAFoundation are imperfect because a DL or LL reaction would technically be different for the WLX and WLZ cases.



RISAFoundation will not consider second order effects

In a true second order analysis, the P-Delta effect would vary depending on which LC was used to create it because of the non-linearity of the effect when multiple loads are combined together. In that manner the Load Category reactions that make their way into RISAFoundation are imperfect because the P-Delta effect on a DL or LL reaction would technically be different depending on whether the final Load Combination included WLX or WLZ cases.

Section Sets

Cross-section properties may be assigned to members in one of two ways; either by choosing a shape directly from the steel database or by using a section set. Section sets provide a way to group members so that they have the same properties. Adjusting the set properties, rather than selecting and adjusting each member can achieve changes to the properties for all of the members in the set. You must use section sets if you want to perform timber code checks or steel shape optimization.

The cross section data for the members is recorded on the **Section Sets** spreadsheet. Cross sectional properties can be entered manually or may be retrieved from one of the shape databases. Currently the databases include Hot Rolled Steel, Cold Formed Steel, Wood, Concrete, and Aluminum. Once the section is defined, it is then referenced on the **Members** spreadsheet when assigning properties to a member.

Define a New Section Set

To define a new Section Set:

1. Click on **Section Sets** in the 'Data Entry' menu, to open the spreadsheet.
2. Select the tab for the Material type that you want to use.
3. Enter the section set label and other information.

You can edit or create a new material by clicking on the browse button  in the cells.

4. (Optional) Click on the **Shape** column browse button  if you want to open the database.



Note:

- If you leave the Shape name blank for **General Sections**, then you may enter the rest of the properties (A, I, J) yourself without having to define an arbitrary shape.
- For rectangular, circular, and pipe shapes you can use the On-Line shapes to automatically define the shape properties. See [On-Line Shapes](#).

Section Sets Spreadsheet

The **Section Sets** spreadsheet records the section properties for the member elements and may be accessed by selecting **Section Sets** in the 'Data Entry' menu.

Hot Rolled Steel Section Sets										
Hot Rolled	Cold Formed	Wood	Concrete	Aluminum	Stainless	General				
	Label	Shape	Type	Design List	Material	Design Rule	Area [in ²]	Iyy [in ⁴]	Izz [in ⁴]	J [in ⁴]
1	Column	W10X33	Column	Wide Flange	A992	Typical	9.71	36.6	171	0.583
2	Int Girder	W24X55	Beam	Wide Flange	A992	Typical	16.2	29.1	1350	1.18
3	Ext Girder	W30X99	Beam	Wide Flange	A992	Typical	29	128	3990	3.77
4	Purlin	W10X26	Beam	Wide Flange	A992	Typical	7.61	14.1	144	0.402
5	Brace	LL4X4X4X3	Beam	Double Angle (No Gap)	A992	Typical	3.86	12.2	6	0.088

The following are input columns on the spreadsheet that may be used to specify cross section data for the members in the model.

Section Label

The **Section Label** is the label you'll reference on the 'Members' spreadsheet to assign properties to a member. This label can be anything you wish, so long as it's not the same as any other section set label.

Section Database Shape

The **Shape** field is used to obtain properties from the shape databases. To use the database, simply enter the name of the database shape and the shape properties (A, I, J) will be filled in automatically. Click  to pick from the database.

If you don't want to use a database shape and wish to enter the shape properties directly, just leave this field blank.



Note: For rectangular, circular, and pipe shapes you can use the On-Line shapes to automatically define the shape properties. See [On-Line Shapes](#).

Section Material

The **Material** field is used to enter the label of the material. The material must be defined on the 'Materials' spreadsheet or on the 'Wood Properties' spreadsheet.

Member Type

Enter the member type for the section set. The choices are *Column*, *Beam*, *Vertical Brace*, and *Horizontal Brace*.

Here are the main effects that the member type will have on your structure:

1. If using concrete members, this will define the rebar layout (column vs beam) .
2. If using design lists, they specifically reference the member type.

3. If using member area loads, loads will not be attributed to members defined as Hbraces or Vbraces.
4. If using the RISA-Revit link the link will not work properly unless you use member types.
5. If integrating with RISACONNECTION, the connection validation requires proper use of member types.
6. If using the seismic detailing provisions then you must use proper member types.

Design List

Enter the **design list** type that you wish to use for this section set. This entry will affect the members that are available to program when it is suggesting alternate or optimized shapes. Refer to [Design Optimization](#) for more information on the member optimization procedure. Also refer to [Appendix A – Redesign Lists](#) for information on creating or editing these lists.

Design Rules

Enter the **design rules** type that you wish to use for this section set. When the program is checking alternate or optimized shapes, it will restrict its selections to members that obey the chosen design rules. Refer to [Design Rules– Size / U.C.](#) for more information.

Cross Section Properties

The cross section properties will be filled in automatically if you use a database shape. For **General Materials**, you may leave the shape field blank and enter these directly. **lyy** and **lzz** are for bending about the respective member local axes. Note that it's not a good idea to edit these fields if you already have a database shape assigned. Any changes will not be saved and will be replaced with the original values when the file is reopened or the shape is reentered in some other manner. You should create a new shape in the database to avoid this situation.



Note: For Tapered WF shapes, only the shape properties at the I-end will be shown on the 'Section Sets' spreadsheet. To view properties for both ends use the **Edit** button in the 'Shape Selection' settings.

Seismic Detailing - Design Rules

Seismic Design Rules can be applied to Column, Beam and VBrace members in the model. The rules invoke various design or code check requirements according to AISC seismic design provisions (AISC 341, AISC 358). The program uses the 2005, 2010, or 2016 versions of these codes, depending on the HR steel design code settings in the Model Settings.

The design provisions primarily apply only to Hot Rolled steel members. However, the **Column Overstrength** design option applies to all members to which that seismic design rule has been applied. The reason for this is so that members (collectors or such) which require design to the overstrength provisions per the requirements of ASCE-7 can be automatically designed to the higher force requirements of the Overstrength load combinations.

The default entry for a member's seismic design rule is None which means that no special seismic detailing provisions will apply to the code checking provisions for that member.

General Frames and Columns

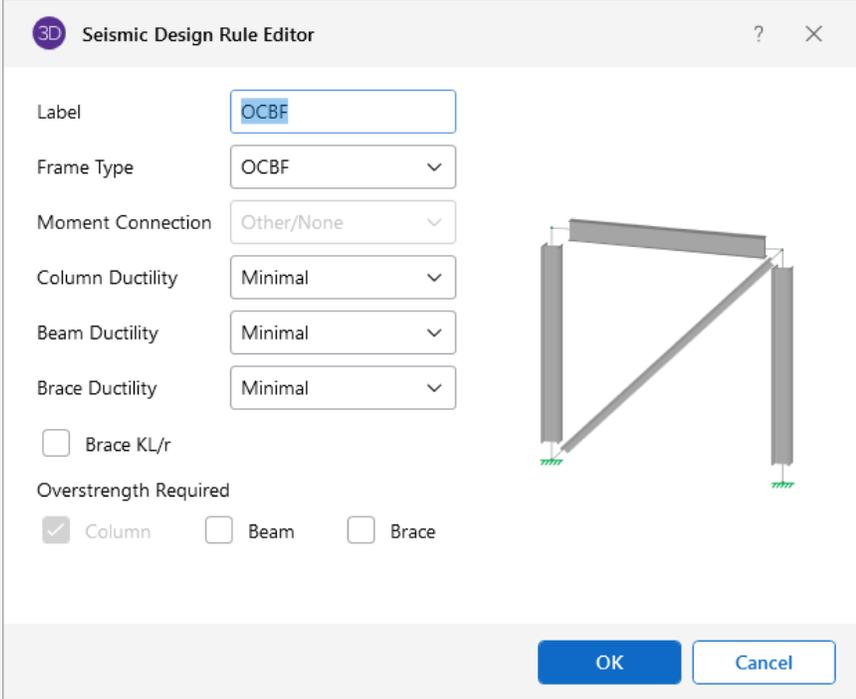
Seismic Design Rules: Hot Rolled Frame - General Columns

The first portion of the Seismic Design Rules under the Hot Rolled Frame tab applies to the steel frame in general and Hot Rolled columns.

Frame / HR Seismic Design Rule												
Hot Rolled Frame												
Label	Frame Type	Column Ductility	Column Overstrength	Beam Ductility	Connection	Beam Overstrength	Z Factor	Hinge Location [in]	Brace Ductility	Brace Overstrength	KL/r	
1	OCBF	Minimal	<input checked="" type="checkbox"/>	Minimal	Other/None	<input type="checkbox"/>			Minimal	<input type="checkbox"/>	<input type="checkbox"/>	
2	SCBF	High	<input checked="" type="checkbox"/>	High	Other/None	<input checked="" type="checkbox"/>			High	<input type="checkbox"/>	<input checked="" type="checkbox"/>	
3	OMF	Minimal	<input checked="" type="checkbox"/>	Minimal	BUEEP	<input type="checkbox"/>		12	N/A	<input type="checkbox"/>	<input type="checkbox"/>	
4	IMF	Moderate	<input checked="" type="checkbox"/>	Moderate	BFP	<input type="checkbox"/>		12	N/A	<input type="checkbox"/>	<input type="checkbox"/>	
5	SMF-RBS	High	<input checked="" type="checkbox"/>	High	RBS	<input type="checkbox"/>	0.685	14.625	N/A	<input type="checkbox"/>	<input type="checkbox"/>	
6	SMF-KaiserB	High	<input checked="" type="checkbox"/>	High	KBB-B	<input type="checkbox"/>		12	N/A	<input type="checkbox"/>	<input type="checkbox"/>	
7	SMF-KaiserW	High	<input checked="" type="checkbox"/>	High	KBB-W	<input type="checkbox"/>		12	N/A	<input type="checkbox"/>	<input type="checkbox"/>	
8	SMF-BSEEP	High	<input checked="" type="checkbox"/>	High	BSEEP	<input type="checkbox"/>		12	N/A	<input type="checkbox"/>	<input type="checkbox"/>	
9	SMF-WUF-W	High	<input checked="" type="checkbox"/>	High	WUF-W	<input type="checkbox"/>			N/A	<input type="checkbox"/>	<input type="checkbox"/>	

Seismic Design Rules: Hot Rolled Frame - General / Column Ductility

Column Header	Description
Label	<p>The label is a user defined text string which is used as a unique identifier for each of the seismic design rules defined for the structure. The program comes pre-loaded with number of generic seismic design rules based on AISC 341 seismic detailing specification.</p> <p>Click on the  ellipsis in the Label column to open the Seismic Design Rule Editor window. This window lets you further define the frame type, member ductility, and member overstrength</p>

Column Header	Description
	<p>requirements, etc. You can choose either to use this editor window or the spreadsheet, to edit a seismic design rule.</p> 
Frame Type	<p>Frame Type defines the frame type (e.g. SMF, SCBF) for the frame. The program is pre-loaded with several typical seismic-force-resisting-systems based on AISC 341 seismic detailing specification. The program uses this entry to determine which code sections of seismic detailing requirements should be applied to members with the current seismic design rule.</p>
Column Ductility	<p>Column Ductility defines the ductility requirements for the columns.</p> <ul style="list-style-type: none"> • High Ductility: Refers to a member which requires Seismically Compact sections for members such as a Special Concentrically Braced Frame (SCBF) or a Special Moment Frame (SMF). • Moderate Ductility: Refers to a member which require Compact sections for frame members, but which does not require the special Seismic Compactness defined in AISC 341. One example would be an Intermediate Moment Frame (IMF). • Minimal Ductility: Refers to a member which does not have specific compactness requirements beyond the normal AISC specification. This can even include members with slender elements. One example would be

Column Header	Description
	an Ordinary Moment Frame (OMF).
<div style="border: 1px solid black; padding: 10px; background-color: #e6f2ff;">  <p>Note:</p> <ul style="list-style-type: none"> • Ordinary Concentrically Braced Frames (OCBF) require Moderate or High ductility out of the brace members (depending on the version of the code), but allow for Minimal ductility out of the other members. The program will automatically account for this brace ductility requirement. Therefore, the Frame Ductility may still be entered as Minimal. • When checking the seismic compactness of members, the axial load used to calculate C_a is based on the worst case axial compression from the normal load combinations. It will not consider any axial force that occurs from an "overstrength" load combination. • The Frame Ductility setting is used by the program when it is checking some of the miscellaneous beam-column moment connection requirements per AISC 358. For example: if the beam to column connection is specified by the user as a Reduced Beam Section (RBS) then the span to depth ratio of the beam must be greater than 7.0 to be considered a highly ductile frame, and greater than 5.0 to be considered a moderately ductile frame. </div>	
Column Overstrength	In seismic design some members may be required to be designed to an elevated / overstrength load. In RISA, this is done by creating load combinations with the Omega overstrength factors applied to the earthquake loads. If the column members are required to be designed to these load combinations, then this check box will be checked.

Column Header	Description
	<p> Note:</p> <ul style="list-style-type: none"> • In many cases, it's only the axial effect of the overstrength loads that needs to be considered in the column overstrength code checks. In these cases, it could be overly conservative to consider the moment with these elevated axial forces. A future revision to the program will add an Axial Only option to this input field which would allow the program to ignore the effects of moment when the amplified axial force is taken into account. • Generally speaking the 2005 version of AISC 341 requires that moment frame columns be designed for the overstrength loads whenever the axial force for the regular load combinations exceed 40% of the column's axial capacity. However, the 2010 and newer versions of the same specification always require that these columns be designed to these overstrength loads. Therefore, this entry cannot be modified when the 2010 or newer code is selected. • The program does not consider any of the limitations related to the "sum of shears which can be transmitted to the column" or the "sum of the expected strength of the braces". • If a Seismic Design Rule is applied to a column member which is not hot rolled steel (such as a wood drag strut or collector), that member will ignore the design rules except for the Overstrength required flag. If this is selected, then the non-steel member's capacity will be checked against the forces derived from the overstrength seismic load combinations. • Refer to the LC Generator and the Set BLC Entry sub-topics in the Load Combinations section for more information regarding the creation of overstrength load combinations.

Beams

Seismic Design Rules - Hot Rolled Frame - Beams

This portion of the spreadsheet contains information pertinent mostly for the design of hot rolled beams that are part of moment frames. The only exception to this is the Beam Overstrength option which applies to any beam member which is assigned the seismic design rule.

Frame / HR Seismic Design Rule												
Hot Rolled Frame												
Label	Frame Type	Column Ductility	Column Overstrength	Beam Ductility	Connection	Beam Overstrength	Z Factor	Hinge Location [in]	Brace Ductility	Brace Overstrength	KL/r	
1	OCBF	OCBF	Minimal	<input checked="" type="checkbox"/>	Minimal	Other/None	<input type="checkbox"/>			Minimal	<input type="checkbox"/>	
2	SCBF	SCBF	High	<input checked="" type="checkbox"/>	High	Other/None	<input checked="" type="checkbox"/>			High	<input type="checkbox"/>	<input checked="" type="checkbox"/>
3	OMF	OMF	Minimal	<input checked="" type="checkbox"/>	Minimal	BUEEP	<input type="checkbox"/>		12	N/A	<input type="checkbox"/>	
4	IMF	IMF	Moderate	<input checked="" type="checkbox"/>	Moderate	BFP	<input type="checkbox"/>		12	N/A	<input type="checkbox"/>	
5	SMF-RBS	SMF	High	<input checked="" type="checkbox"/>	High	RBS	<input type="checkbox"/>	0.685	14.625	N/A	<input type="checkbox"/>	
6	SMF-KaiserB	SMF	High	<input checked="" type="checkbox"/>	High	KBB-B	<input type="checkbox"/>		12	N/A	<input type="checkbox"/>	
7	SMF-KaiserW	SMF	High	<input checked="" type="checkbox"/>	High	KBB-W	<input type="checkbox"/>		12	N/A	<input type="checkbox"/>	
8	SMF-BSEEP	SMF	High	<input checked="" type="checkbox"/>	High	BSEEP	<input type="checkbox"/>		12	N/A	<input type="checkbox"/>	
9	SMF-WUF-W	SMF	High	<input checked="" type="checkbox"/>	High	WUF-W	<input type="checkbox"/>			N/A	<input type="checkbox"/>	

Seismic Design Rules: Hot Rolled Frame - Beam Ductility

Column Header	Description
Beam Ductility	Beam Ductility defines the ductility requirements for the beams. The definitions of High Ductility , Moderate Ductility , Minimal Ductility are described under the Column Ductility section above.
Connection	<p>Connection designates which of the pre-qualified moment connections defined in AISC 358 or its supplement are being used for the beam to column moment connection. The options are:</p> <ul style="list-style-type: none"> • Bolted Flange Plate (BFP) as described in Chapter 7 of AISC 358. • Reduced Beam Section (RBS) as described in Chapter 5 of AISC 358. • Bolted Unstiffened Extended End Plate (BUEEP) and Bolted Stiffened Extended End Plate (BSEEP) as described in Chapter 6 of AISC 358. • Welded Unreinforced Flange - Welded Web (WUF-W) as described in Chapter 8 of AISC 358. • Kaiser Bolted Bracket (KBB) as described in Chapter 9 of AISC 358 • Other/None: There are times where AISC 341 and AISC 358 have conflicting provisions. This includes the definition of the probable maximum moment at the hinge and the definition of the Strong Column / Weak Beam Ratio. When one of the pre-qualified connections is selected then the program will enforce the AISC 358 version of the provisions. When the Other connection is selected then, the AISC 341 version of these provisions will be enforced instead. This "connection option" should normally be used when the beam will not have moment connections (i.e. it is part of a braced frame).

Column Header	Description
	 <p>Note: The moment connection setting is used (in combination with the Frame Ductility setting on the column tab), to check some of the miscellaneous beam-column moment connection requirements per AISC 358. For example: if the beam to column connection is specified by the user as an Reduced Beam Section (RBS) then the column depth must be limited to a maximum of W36, and the beam weight cannot exceed 300 lbs / ft.</p>
<p>Beam Overstrength</p>	<p>In seismic design some members may be required to be designed to an elevated / overstrength load. In RISA, this is done by creating load combinations with the Omega overstrength factors applied to the earthquake loads. If the beam members are required to be designed to these load combinations, then this box would be checked.</p>
	 <p>Note:</p> <ul style="list-style-type: none"> • Beams in a braced frame may act as some form of a collector or drag strut. As such, the ASCE-7 seismic provisions would require that they be designed for the overstrength load combinations. • If a Seismic Design Rule is applied to a beam member which is not hot rolled steel (such as a wood drag strut), that member will ignore the design rules except for the Overstrength required flag. If this is selected, then the non-steel member's capacity will be checked against the forces derived from the overstrength seismic load combinations. • If only an overstrength load combination was run then you will get no results for members that are not required to be designed to overstrength loading. • Refer to the LC Generator and the Set BLC Entry sub-topics in the Load Combinations section for more information regarding the creation of overstrength load combinations.
<p>Z Factor</p>	<p>This factor is used to define the reduction in plastic hinge moment expected for Reduced Beam Sections. Enter in the ratio between the plastic section modulus for the reduced beam section and the unreduced beam. For RBS connections this value will vary greatly, but will always be less than 1.0. The program will not allow a value of less than 0.1 to be entered in by the user. If this value is left blank, then no reduction in moment is considered.</p> <p>This factor will be used to determine the probable design strength and the strong column / weak beam moment ratio for the connection. It does NOT currently reduce the stiffness of the beam used in the analysis.</p>

Column Header	Description
	 Note: AISC 358 has some restrictions on the length and depth of cut that is allowed for the RBS section. RISA does not make any attempt to enforce these restrictions.
Hinge Location	<p>This entry defines the location of the assumed plastic hinge (in inches) from the face of the column. This is used to determine the design moment at the face of the column as well as the strong column / weak beam ratio.</p> <ul style="list-style-type: none"> For RBS connections, this entry should be the distance from the face of column to the center of the reduced beam section. For stiffened connections (such as BSEEP, or BFP), this will usually be the distance from the face of column to the end of the stiffener, haunch, or flange plate. For unstiffened end plate connections, this entry will usually be the lesser of 50% of the beam depth or 3 times the beam flange width. For WUF-W connections, this entry will normally be set to 0.0.

Braces

Seismic Design Rules - Hot Rolled Frame - Braces

This portion of the spreadsheet contains information pertinent to the design of hot rolled braces. The only exception to this is the Brace Overstrength option which applies to any VBrace member which is assigned the seismic design rule.

Hot Rolled Frame												
Label	Frame Type	Column Ductility	Column Overstrength	Beam Ductility	Connection	Beam Overstrength	Z Factor	Hinge Location [in]	Brace Ductility	Brace Overstrength	KL/r	
1	OCBF	Minimal	<input checked="" type="checkbox"/>	Minimal	Other/None	<input type="checkbox"/>			Minimal	<input type="checkbox"/>	<input type="checkbox"/>	
2	SCBF	High	<input checked="" type="checkbox"/>	High	Other/None	<input checked="" type="checkbox"/>			High	<input type="checkbox"/>	<input checked="" type="checkbox"/>	
3	OMF	Minimal	<input checked="" type="checkbox"/>	Minimal	BUEEP	<input type="checkbox"/>		12	N/A	<input type="checkbox"/>	<input type="checkbox"/>	
4	IMF	Moderate	<input checked="" type="checkbox"/>	Moderate	BFP	<input type="checkbox"/>		12	N/A	<input type="checkbox"/>	<input type="checkbox"/>	
5	SMF-RBS	High	<input checked="" type="checkbox"/>	High	RBS	<input type="checkbox"/>	0.685	14.625	N/A	<input type="checkbox"/>	<input type="checkbox"/>	
6	SMF-KaiserB	High	<input checked="" type="checkbox"/>	High	KBB-B	<input type="checkbox"/>		12	N/A	<input type="checkbox"/>	<input type="checkbox"/>	
7	SMF-KaiserW	High	<input checked="" type="checkbox"/>	High	KBB-W	<input type="checkbox"/>		12	N/A	<input type="checkbox"/>	<input type="checkbox"/>	
8	SMF-BSEEP	High	<input checked="" type="checkbox"/>	High	BSEEP	<input type="checkbox"/>		12	N/A	<input type="checkbox"/>	<input type="checkbox"/>	
9	SMF-WUF-W	High	<input checked="" type="checkbox"/>	High	WUF-W	<input type="checkbox"/>			N/A	<input type="checkbox"/>	<input type="checkbox"/>	

Seismic Design Rules: Hot Rolled Frame - Brace Ductility

Column Header	Description
Brace Ductility	Brace Ductility defines the ductility requirements for the braces. The definitions of High Ductility , Moderate Ductility , Minimal Ductility are described under the Column Ductility section above.
Brace Overstrength	In seismic design some members may be required to be designed to an elevated / overstrength load. In RISA this is done by creating load combinations with the (Ω_0) overstrength factors applied to the earthquake loads. If the brace members are required to be designed to these load combinations, then this box would be checked.

**Note:**

- If a Seismic Design Rule is applied to a brace member which is not hot rolled steel, that member will ignore the design rules except for the Overstrength required flag. If this is selected, then the non-steel member's capacity will be checked against the forces derived from the overstrength seismic load combinations.
- Refer to the [LC Generator](#) and the [Set BLC Entry](#) sub-topics in the Load Combinations section for more information regarding the creation of overstrength load combinations.

Max KL/r	<p>In seismic design some members may have the following restriction on the maximum slenderness (KL/r) value that they are allowed to have.</p> $\frac{KL}{r} \leq 4 \sqrt{\frac{E}{F_y}}$ <p>Examples would be AISC 341-05 braces that are part of a Special Concentrically Braced Frame (SCBF). This would also include AISC 341-05 and AISC 341-10 K, V, or inverted V braces in an Ordinary Concentrically Braced Frame (OCBF).</p>
-----------------	---

Limitations

Seismic Detailing Limitations

Limitation	Description
Panel Zone Capacity	Currently, the program uses the elastic shear capacity equations (AISC 360-2010, eqns J10-9 and J10-10) for its code check of the panel zone. When plastic panel zone deformation is considered (along with shear

Limitation	Description
	capacity of the flanges) less conservative equations (J10-11 and J10-12) may be more appropriate.
V and Inverted V Braced Frames & Gravity Loads	The requirement that the beam be analyzed as though the brace carried no dead or live load cannot be directly met in the RISA-3D analysis. However, this may be accomplished by integrating the model with RISAFloor, which automatically analyzes the beam for gravity loads as though the brace were not present.
Unbalanced Beam Force in Braced Frames	The effect of unbalanced brace forces is calculated only for beams in V or an inverted V frame configuration. This force is reported in the Seismic Detailing portion of the beam's detail report. However, this force is NOT used in the code checking of the beam UNLESS a capacity-limited (CL) load combination is used. For capacity-limited LCs (e.g. LC with ELX-CL), the program will apply brace expected strengths as seismic loads for columns and beams design. Under these load combinations, the beam unbalanced forces will be automatically considered in beam design by applying the capacity-limited forces.
Expected Strength of Braces	The effect of expected strength of braces is considered through the capacity-limited load combinations (e.g. ELX-CL, ELZ-CL) for columns and beams in braced frames such as SCBF and BRBF. Users need to select the supported design code (AISC 2010 or 2016), assign seismic design rules with appropriate frame type (e.g. SCBF or BRBF), and generate capacity-limited load combinations to perform capacity-limited design for columns and beams in braced frames.

Seismic Detailing - Design Rules

Seismic Design Rules can be applied to Column, Beam and VBrace members in the model. The rules invoke various design or code check requirements according to AISC seismic design provisions (AISC 341, AISC 358). The program uses the 2005, 2010, or 2016 versions of these codes, depending on the HR steel design code settings in the Model Settings.

The design provisions primarily apply only to Hot Rolled steel members. However, the **Column Overstrength** design option applies to all members to which that seismic design rule has been applied. The reason for this is so that members (collectors or such) which require design to the overstrength provisions per the requirements of ASCE-7 can be automatically designed to the higher force requirements of the Overstrength load combinations.

The default entry for a member's seismic design rule is None which means that no special seismic detailing provisions will apply to the code checking provisions for that member.

General Frames and Columns

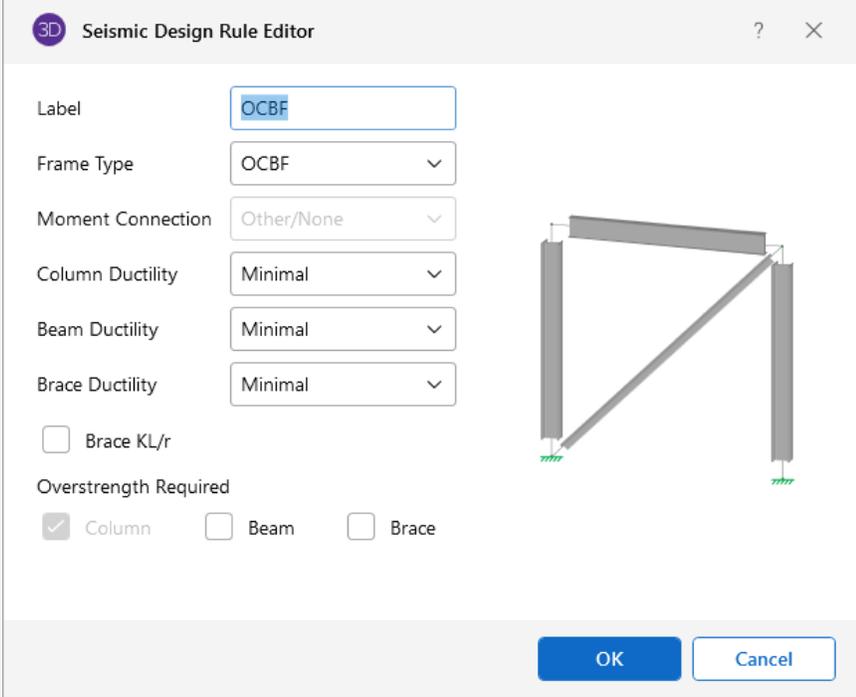
Seismic Design Rules: Hot Rolled Frame - General Columns

The first portion of the Seismic Design Rules under the Hot Rolled Frame tab applies to the steel frame in general and Hot Rolled columns.

Frame / HR Seismic Design Rule												
Hot Rolled Frame												
Label	Frame Type	Column Ductility	Column Overstrength	Beam Ductility	Connection	Beam Overstrength	Z Factor	Hinge Location [in]	Brace Ductility	Brace Overstrength	KL/r	
1	OCBF	OCBF	Minimal	<input checked="" type="checkbox"/>	Minimal	Other/None	<input type="checkbox"/>			Minimal	<input type="checkbox"/>	
2	SCBF	SCBF	High	<input checked="" type="checkbox"/>	High	Other/None	<input checked="" type="checkbox"/>			High	<input type="checkbox"/>	<input checked="" type="checkbox"/>
3	OMF	OMF	Minimal	<input checked="" type="checkbox"/>	Minimal	BUEEP	<input type="checkbox"/>	12		N/A	<input type="checkbox"/>	<input type="checkbox"/>
4	IMF	IMF	Moderate	<input checked="" type="checkbox"/>	Moderate	BFP	<input type="checkbox"/>	12		N/A	<input type="checkbox"/>	<input type="checkbox"/>
5	SMF-RBS	SMF	High	<input checked="" type="checkbox"/>	High	RBS	<input type="checkbox"/>	0.685	14.625	N/A	<input type="checkbox"/>	<input type="checkbox"/>
6	SMF-KaiserB	SMF	High	<input checked="" type="checkbox"/>	High	KBB-B	<input type="checkbox"/>	12		N/A	<input type="checkbox"/>	<input type="checkbox"/>
7	SMF-KaiserW	SMF	High	<input checked="" type="checkbox"/>	High	KBB-W	<input type="checkbox"/>	12		N/A	<input type="checkbox"/>	<input type="checkbox"/>
8	SMF-BSEEP	SMF	High	<input checked="" type="checkbox"/>	High	BSEEP	<input type="checkbox"/>	12		N/A	<input type="checkbox"/>	<input type="checkbox"/>
9	SMF-WUF-W	SMF	High	<input checked="" type="checkbox"/>	High	WUF-W	<input type="checkbox"/>			N/A	<input type="checkbox"/>	<input type="checkbox"/>

Seismic Design Rules: Hot Rolled Frame - General / Column Ductility

Column Header	Description
Label	<p>The label is a user defined text string which is used as a unique identifier for each of the seismic design rules defined for the structure. The program comes pre-loaded with number of generic seismic design rules based on AISC 341 seismic detailing specification.</p> <p>Click on the <input type="text" value="..."/> ellipsis in the Label column to open the Seismic Design Rule Editor window. This window lets you further define the frame type, member ductility, and member overstrength requirements, etc. You can choose either to use this editor window or the spreadsheet, to edit a seismic design rule.</p>

Column Header	Description
	
Frame Type	<p>Frame Type defines the frame type (e.g. SMF, SCBF) for the frame. The program is pre-loaded with several typical seismic-force-resisting-systems based on AISC 341 seismic detailing specification. The program uses this entry to determine which code sections of seismic detailing requirements should be applied to members with the current seismic design rule.</p>
Column Ductility	<p>Column Ductility defines the ductility requirements for the columns.</p> <ul style="list-style-type: none"> • High Ductility: Refers to a member which requires Seismically Compact sections for members such as a Special Centrically Braced Frame (SCBF) or a Special Moment Frame (SMF). • Moderate Ductility: Refers to a member which require Compact sections for frame members, but which does not require the special Seismic Compactness defined in AISC 341. One example would be an Intermediate Moment Frame (IMF). • Minimal Ductility: Refers to a member which does not have specific compactness requirements beyond the normal AISC specification. This can even include members with slender elements. One example would be an Ordinary Moment Frame (OMF).

Column Header	Description
 Note: <ul style="list-style-type: none"> • Ordinary Concentrically Braced Frames (OCBF) require Moderate or High ductility out of the brace members (depending on the version of the code), but allow for Minimal ductility out of the other members. The program will automatically account for this brace ductility requirement. Therefore, the Frame Ductility may still be entered as Minimal. • When checking the seismic compactness of members, the axial load used to calculate C_a is based on the worst case axial compression from the normal load combinations. It will not consider any axial force that occurs from an "overstrength" load combination. • The Frame Ductility setting is used by the program when it is checking some of the miscellaneous beam-column moment connection requirements per AISC 358. For example: if the beam to column connection is specified by the user as a Reduced Beam Section (RBS) then the span to depth ratio of the beam must be greater than 7.0 to be considered a highly ductile frame, and greater than 5.0 to be considered a moderately ductile frame. 	
Column Overstrength	In seismic design some members may be required to be designed to an elevated / overstrength load. In RISA, this is done by creating load combinations with the Omega overstrength factors applied to the earthquake loads. If the column members are required to be designed to these load combinations, then this check box will be checked.

Column Header	Description
	<p> Note:</p> <ul style="list-style-type: none"> In many cases, it's only the axial effect of the overstrength loads that needs to be considered in the column overstrength code checks. In these cases, it could be overly conservative to consider the moment with these elevated axial forces. A future revision to the program will add an Axial Only option to this input field which would allow the program to ignore the effects of moment when the amplified axial force is taken into account. Generally speaking the 2005 version of AISC 341 requires that moment frame columns be designed for the overstrength loads whenever the axial force for the regular load combinations exceed 40% of the column's axial capacity. However, the 2010 and newer versions of the same specification always require that these columns be designed to these overstrength loads. Therefore, this entry cannot be modified when the 2010 or newer code is selected. The program does not consider any of the limitations related to the "sum of shears which can be transmitted to the column" or the "sum of the expected strength of the braces". If a Seismic Design Rule is applied to a column member which is not hot rolled steel (such as a wood drag strut or collector), that member will ignore the design rules except for the Overstrength required flag. If this is selected, then the non-steel member's capacity will be checked against the forces derived from the overstrength seismic load combinations. Refer to the LC Generator and the Set BLC Entry sub-topics in the Load Combinations section for more information regarding the creation of overstrength load combinations.

Beams

Seismic Design Rules - Hot Rolled Frame - Beams

This portion of the spreadsheet contains information pertinent mostly for the design of hot rolled beams that are part of moment frames. The only exception to this is the Beam Overstrength option which applies to any beam member which is assigned the seismic design rule.

Frame / HR Seismic Design Rule												
Hot Rolled Frame												
Label	Frame Type	Column Ductility	Column Overstrength	Beam Ductility	Connection	Beam Overstrength	Z Factor	Hinge Location [in]	Brace Ductility	Brace Overstrength	KL/r	
1	OCBF	OCBF	Minimal	<input checked="" type="checkbox"/>	Minimal	Other/None	<input type="checkbox"/>			Minimal	<input type="checkbox"/>	
2	SCBF	SCBF	High	<input checked="" type="checkbox"/>	High	Other/None	<input checked="" type="checkbox"/>			High	<input type="checkbox"/>	<input checked="" type="checkbox"/>
3	OMF	OMF	Minimal	<input checked="" type="checkbox"/>	Minimal	BUEEP	<input type="checkbox"/>		12	N/A	<input type="checkbox"/>	
4	IMF	IMF	Moderate	<input checked="" type="checkbox"/>	Moderate	BFP	<input type="checkbox"/>		12	N/A	<input type="checkbox"/>	
5	SMF-RBS	SMF	High	<input checked="" type="checkbox"/>	High	RBS	<input type="checkbox"/>	0.685	14.625	N/A	<input type="checkbox"/>	
6	SMF-KaiserB	SMF	High	<input checked="" type="checkbox"/>	High	KBB-B	<input type="checkbox"/>		12	N/A	<input type="checkbox"/>	
7	SMF-KaiserW	SMF	High	<input checked="" type="checkbox"/>	High	KBB-W	<input type="checkbox"/>		12	N/A	<input type="checkbox"/>	
8	SMF-BSEEP	SMF	High	<input checked="" type="checkbox"/>	High	BSEEP	<input type="checkbox"/>		12	N/A	<input type="checkbox"/>	
9	SMF-WUF-W	SMF	High	<input checked="" type="checkbox"/>	High	WUF-W	<input type="checkbox"/>			N/A	<input type="checkbox"/>	

Seismic Design Rules: Hot Rolled Frame - Beam Ductility

Column Header	Description
Beam Ductility	Beam Ductility defines the ductility requirements for the beams. The definitions of High Ductility , Moderate Ductility , Minimal Ductility are described under the Column Ductility section above.
Connection	<p>Connection designates which of the pre-qualified moment connections defined in AISC 358 or its supplement are being used for the beam to column moment connection. The options are:</p> <ul style="list-style-type: none"> • Bolted Flange Plate (BFP) as described in Chapter 7 of AISC 358. • Reduced Beam Section (RBS) as described in Chapter 5 of AISC 358. • Bolted Unstiffened Extended End Plate (BUEEP) and Bolted Stiffened Extended End Plate (BSEEP) as described in Chapter 6 of AISC 358. • Welded Unreinforced Flange - Welded Web (WUF-W) as described in Chapter 8 of AISC 358. • Kaiser Bolted Bracket (KBB) as described in Chapter 9 of AISC 358 • Other/None: There are times where AISC 341 and AISC 358 have conflicting provisions. This includes the definition of the probable maximum moment at the hinge and the definition of the Strong Column / Weak Beam Ratio. When one of the pre-qualified connections is selected then the program will enforce the AISC 358 version of the provisions. When the Other connection is selected then, the AISC 341 version of these provisions will be enforced instead. This "connection option" should normally be used when the beam will not have moment connections (i.e. it is part of a braced frame).

Column Header	Description
	 <p>Note: The moment connection setting is used (in combination with the Frame Ductility setting on the column tab), to check some of the miscellaneous beam-column moment connection requirements per AISC 358. For example: if the beam to column connection is specified by the user as an Reduced Beam Section (RBS) then the column depth must be limited to a maximum of W36, and the beam weight cannot exceed 300 lbs / ft.</p>
<p>Beam Overstrength</p>	<p>In seismic design some members may be required to be designed to an elevated / overstrength load. In RISA, this is done by creating load combinations with the Omega overstrength factors applied to the earthquake loads. If the beam members are required to be designed to these load combinations, then this box would be checked.</p>
	 <p>Note:</p> <ul style="list-style-type: none"> • Beams in a braced frame may act as some form of a collector or drag strut. As such, the ASCE-7 seismic provisions would require that they be designed for the overstrength load combinations. • If a Seismic Design Rule is applied to a beam member which is not hot rolled steel (such as a wood drag strut), that member will ignore the design rules except for the Overstrength required flag. If this is selected, then the non-steel member's capacity will be checked against the forces derived from the overstrength seismic load combinations. • If only an overstrength load combination was run then you will get no results for members that are not required to be designed to overstrength loading. • Refer to the LC Generator and the Set BLC Entry sub-topics in the Load Combinations section for more information regarding the creation of overstrength load combinations.
<p>Z Factor</p>	<p>This factor is used to define the reduction in plastic hinge moment expected for Reduced Beam Sections. Enter in the ratio between the plastic section modulus for the reduced beam section and the unreduced beam. For RBS connections this value will vary greatly, but will always be less than 1.0. The program will not allow a value of less than 0.1 to be entered in by the user. If this value is left blank, then no reduction in moment is considered.</p> <p>This factor will be used to determine the probable design strength and the strong column / weak beam moment ratio for the connection. It does NOT currently reduce the stiffness of the beam used in the analysis.</p>

Column Header	Description
	 Note: AISC 358 has some restrictions on the length and depth of cut that is allowed for the RBS section. RISA does not make any attempt to enforce these restrictions.
Hinge Location	This entry defines the location of the assumed plastic hinge (in inches) from the face of the column. This is used to determine the design moment at the face of the column as well as the strong column / weak beam ratio. <ul style="list-style-type: none"> For RBS connections, this entry should be the distance from the face of column to the center of the reduced beam section. For stiffened connections (such as BSEEP, or BFP), this will usually be the distance from the face of column to the end of the stiffener, haunch, or flange plate. For unstiffened end plate connections, this entry will usually be the lesser of 50% of the beam depth or 3 times the beam flange width. For WUF-W connections, this entry will normally be set to 0.0.

Braces

Seismic Design Rules - Hot Rolled Frame - Braces

This portion of the spreadsheet contains information pertinent to the design of hot rolled braces. The only exception to this is the Brace Overstrength option which applies to any VBrace member which is assigned the seismic design rule.

Frame / HR Seismic Design Rule												
Hot Rolled Frame												
Label	Frame Type	Column Ductility	Column Overstrength	Beam Ductility	Connection	Beam Overstrength	Z Factor	Hinge Location [in]	Brace Ductility	Brace Overstrength	KL/r	
1	OCBF	OCBF	Minimal	<input checked="" type="checkbox"/>	Minimal	Other/None	<input type="checkbox"/>		Minimal	<input type="checkbox"/>	<input type="checkbox"/>	
2	SCBF	SCBF	High	<input checked="" type="checkbox"/>	High	Other/None	<input checked="" type="checkbox"/>		High	<input type="checkbox"/>	<input checked="" type="checkbox"/>	
3	OMF	OMF	Minimal	<input checked="" type="checkbox"/>	Minimal	BUEEP	<input type="checkbox"/>	12	N/A	<input type="checkbox"/>	<input type="checkbox"/>	
4	IMF	IMF	Moderate	<input checked="" type="checkbox"/>	Moderate	BFP	<input type="checkbox"/>	12	N/A	<input type="checkbox"/>	<input type="checkbox"/>	
5	SMF-RBS	SMF	High	<input checked="" type="checkbox"/>	High	RBS	<input type="checkbox"/>	0.685	14.625	<input type="checkbox"/>	<input type="checkbox"/>	
6	SMF-KaiserB	SMF	High	<input checked="" type="checkbox"/>	High	KBB-B	<input type="checkbox"/>	12	N/A	<input type="checkbox"/>	<input type="checkbox"/>	
7	SMF-KaiserW	SMF	High	<input checked="" type="checkbox"/>	High	KBB-W	<input type="checkbox"/>	12	N/A	<input type="checkbox"/>	<input type="checkbox"/>	
8	SMF-BSEEP	SMF	High	<input checked="" type="checkbox"/>	High	BSEEP	<input type="checkbox"/>	12	N/A	<input type="checkbox"/>	<input type="checkbox"/>	
9	SMF-WUF-W	SMF	High	<input checked="" type="checkbox"/>	High	WUF-W	<input type="checkbox"/>		N/A	<input type="checkbox"/>	<input type="checkbox"/>	

Seismic Design Rules: Hot Rolled Frame - Brace Ductility

Column Header	Description
Brace Ductility	Brace Ductility defines the ductility requirements for the braces. The definitions of High Ductility , Moderate Ductility , Minimal Ductility are described under the Column Ductility section above.
Brace Overstrength	In seismic design some members may be required to be designed to an elevated / overstrength load. In RISA this is done by creating load combinations with the (Ω_0) overstrength factors applied to the earthquake loads. If the brace members are required to be designed to these load combinations, then this box would be checked.

**Note:**

- If a Seismic Design Rule is applied to a brace member which is not hot rolled steel, that member will ignore the design rules except for the Overstrength required flag. If this is selected, then the non-steel member's capacity will be checked against the forces derived from the overstrength seismic load combinations.
- Refer to the [LC Generator](#) and the [Set BLC Entry](#) sub-topics in the Load Combinations section for more information regarding the creation of overstrength load combinations.

Max KL/r	<p>In seismic design some members may have the following restriction on the maximum slenderness (KL/r) value that they are allowed to have.</p> $\frac{KL}{r} \leq 4 \sqrt{\frac{E}{F_y}}$ <p>Examples would be AISC 341-05 braces that are part of a Special Concentrically Braced Frame (SCBF). This would also include AISC 341-05 and AISC 341-10 K, V, or inverted V braces in an Ordinary Concentrically Braced Frame (OCBF).</p>
-----------------	---

Limitations

Seismic Detailing Limitations

Limitation	Description
Panel Zone Capacity	Currently, the program uses the elastic shear capacity equations (AISC 360-2010, eqns J10-9 and J10-10) for its code check of the panel zone. When plastic panel zone deformation is considered (along with shear

Limitation	Description
	capacity of the flanges) less conservative equations (J10-11 and J10-12) may be more appropriate.
V and Inverted V Braced Frames & Gravity Loads	The requirement that the beam be analyzed as though the brace carried no dead or live load cannot be directly met in the RISA-3D analysis. However, this may be accomplished by integrating the model with RISAFloor, which automatically analyzes the beam for gravity loads as though the brace were not present.
Unbalanced Beam Force in Braced Frames	The effect of unbalanced brace forces is calculated only for beams in V or an inverted V frame configuration. This force is reported in the Seismic Detailing portion of the beam's detail report. However, this force is NOT used in the code checking of the beam UNLESS a capacity-limited (CL) load combination is used. For capacity-limited LCs (e.g. LC with ELX-CL), the program will apply brace expected strengths as seismic loads for columns and beams design. Under these load combinations, the beam unbalanced forces will be automatically considered in beam design by applying the capacity-limited forces.
Expected Strength of Braces	The effect of expected strength of braces is considered through the capacity-limited load combinations (e.g. ELX-CL, ELZ-CL) for columns and beams in braced frames such as SCBF and BRBF. Users need to select the supported design code (AISC 2010 or 2016), assign seismic design rules with appropriate frame type (e.g. SCBF or BRBF), and generate capacity-limited load combinations to perform capacity-limited design for columns and beams in braced frames.

Seismic Detailing - Results

The seismic detailing results are presented in two ways: the seismic detailing results spreadsheet and the seismic detailing sections of the [member detail reports](#). The seismic detailing results spreadsheet is intended to be a summary report. More detailed information can be found for each member on that member's detail report.

Seismic Results Spreadsheet - Columns

The first tab of the Seismic Design Rules spreadsheet applies to the Hot Rolled members which have been assigned to the Member Type of Column and which have a seismic design rule applied.

Label	Seismic Design Rule	Ductility Req'd	UC Max	LC	Slenderness Checks	Panel Zone Checks	Panel Zone Eqn	Cont. Plate Req'd	Cont. Plate Eqn	SC/WB Ratio	SC/WB Beam	Misc. Checks
1 M1	OMF	Minimal	0.346	1	Pass	N/A	N/A	N/A	N/A	N/A	N/A	Pass
2 M2	OMF	Minimal	0.478	1	Pass	N/A	N/A	N/A	N/A	N/A	N/A	Pass
3 M3	OMF	Minimal	0.154	1	Pass	N/A	N/A	N/A	N/A	N/A	N/A	Pass
4 M4	OMF	Minimal	0.376	1	Pass	N/A	N/A	N/A	N/A	N/A	N/A	Pass
5 M5	OMF	Minimal	0.57	1	Pass	N/A	N/A	N/A	N/A	N/A	N/A	Pass

Seismic Results Spreadsheet - Columns

Column Header	Description
Label	The Label column displays the user defined name for the column. It is used as a unique identifier for each of the seismic design rules defined for the structure.
Seismic Design Rules	Seismic Design Rule displays the specific rule applied to the column.
Ductility Req'd	<p><i>Ductility Required</i></p> <p>The Ductility Req'd column displays the ductility requirements for the columns. Possible requirements include:</p> <ul style="list-style-type: none"> • High Ductility: Refers to a member which requires Seismically Compact sections for members such as a Special Concentrically Braced Frame (SCBF) or a Special Moment Frame (SMF). • Moderate Ductility: Refers to a member which require Compact sections for frame members, but which does not require the special Seismic Compactness defined in AISC 341. One example would be an Intermediate Moment Frame (IMF). • Minimal Ductility: Refers to a member which does not have specific compactness requirements beyond the normal AISC specification. This can even include members with slender elements. One example would be an Ordinary Moment Frame (OMF).
UC Max	<p><i>Maximum Code Check</i></p> <p>The UC Max column summarizes the member code checks from all the load combinations that included seismic force. It does NOT include code checks related to connection design like panel zone shear checks or continuity plate checks.</p>
LC	<p><i>Load Combinations</i></p> <p>The LC column displays the load combination governing the column. If the load combination that controlled the design used the overstrength seismic forces (meaning it had an Ω factor in the load combination) then the governing load combination will be followed by an asterisk (*).</p>
Slenderness Checks	The Slenderness Checks column summarizes the worst case condition between local element compactness checks and KL/r slenderness restrictions (if any). Whether the member passed the required element compactness checks based on the Req'd Ductility setting. Highly ductile frames are considered to require a "seismically compact" member per the provisions of AISC 341 Table D1.1. Whereas moderately ductile frames are considered to

Column Header	Description
	<p>require a "compact" member per the requirements of the regular AISC 360 design standard (Table B4.1).</p> <p>This column also displays a warning message if the column fails any KL/r restrictions such as those given in AISC 341. Such requirements for special moment frames require that columns which are unbraced must have a $KL/r < 60$.</p> <div style="border: 1px solid black; padding: 5px; background-color: #e6f2ff;">  Note: For sections that do not have a Compact limit per Table B4.1 of the AISC 360 (i.e. Case 3), and who do not meet or exceed the non-compact limit, RISA will report as "non-slender". </div>
Panel Zone Checks	<p>Whenever the program detects a valid seismic moment connection between a beam and a column it checks the panel zone of the column per the regular AISC 360 chapter J checks and displays it in the Panel Zone Checks column.</p> <ul style="list-style-type: none"> • For frames that require high ductility (SMF), the required panel zone shear force is based on the maximum probable moment projected to the face of column per AISC 341. • For frames which do not require high ductility (IMF and OMF), the panel zone shear demand is based on the end moments from the solved load combinations. If the Overstrength Req'd flag has been set for the column, then the Ω_o load combinations is used as well. This is because AISC 341 provides little additional panel zone requirements beyond those given in the AISC 360 specification. <p>In addition, the program checks the panel zone for column beam connections per the extra seismic detailing checks of AISC 341. That is the requirement that the thickness of the column web is greater than the sum of the depth and width of the panel zone divided by 90.</p>
Panel Zone Eqn	<p><i>Panel Zone Equation</i></p> <p>Only the worst case / controlling code equation for the panel zone checks are reported in the Panel Zone Eqn column. However, detailed information on the other equations are reported in the column detail report.</p>

Column Header	Description
	<div style="border: 1px solid black; padding: 10px;">  Note: <ul style="list-style-type: none"> The regular panel zone checks are done using the formulas which assume that panel zone deformation is not included in the analysis. Currently, the effects of column story shear (which tend to decrease the panel zone shear demand) are not taken into account. Per the seismic detailing specification, the extra panel zone thickness checks are only checked for "highly ductile" frames (i.e. Special Moment Frames). These conditions are checked for each beam connected to the column with a valid seismic moment connection. However, the total panel zone shear demand is based on the sum of the panel zone shears from the individual beams. This may be overconservative for cases with an IMF or OMF frame where the maximum moments in the beams occur for different load cases. </div>
<p>Cont. Plate Req'd</p>	<p><i>Continuity Plate Required</i></p> <p>Whenever the program detects a valid seismic moment connection between a beam and a column, it automatically checks the column's capacity to resist the concentrated beam flange forces from the moment connection per section J10.6 of the regular AISC 360 specification. If any of these checks fail, then the program reports that a continuity plate is required at that connection location.</p> <p>When you have selected one of the pre-qualified moment connections from AISC 358, then the program enforces the additional continuity requirements of AISC 358-2005 section 2.4.4 or AISC 341-2010 section E3.6f. For bolted end plates, the equivalent equations from AISC 358 chapter 6 is used instead. Because the codes do not require these provision for frames with minimal ductility (OMF's) this check is not enforced in those cases.</p>
<p>Cont. Plate Eqn</p>	<p><i>Continuity Plate Equation</i></p> <p>Only the worst case / controlling code equation for the continuity plate checks are reported in the Cont. Plate Eqn column. However, detailed information on the other equations are reported in the column detail report.</p> <div style="border: 1px solid black; padding: 10px; margin-top: 10px;">  Note: The demand moment used to calculate the beam flange forces is based on the Req'd Moment reported on the Beams tab of the Seismic Detailing results. </div>
<p>SC/WB Ratio</p>	<p><i>Strong Column / Weak Beam Ratio</i></p> <p>The Strong Column / Weak Beam ratio is checked for all moment</p>

Column Header	Description
	connections between a beam(s) and the column. The worst case (lowest) ratio is reported here. If the frame ductility requirements dictate a minimum value for this ratio, then the program also lists a pass / fail check in this column.
SC/WB Beam	<p><i>Strong Column / Weak Beam Controlling Beam</i></p> <p>The controlling beam is reported to identify the location on the column which resulted in the lowest SC/WB ratio. If the worst case connection had beams framing in from either side then only one of the beams is reported, as only one is required to identify that location.</p>
Misc. Checks	<p><i>Miscellaneous Checks</i></p> <p>The Misc. Checks column relates to a number of miscellaneous checks required by AISC 358. For columns this is normally related to the maximum depth or weight of the column, or whether the member type (tube, wide flange, pipe) is valid with the moment connection assigned to it. Other connection limitations (non-orthogonal / skewed connections) are reported with the beam's Misc Checks.</p>

Seismic Results Spreadsheet - Beams

The second tab of the Seismic Design Rules spreadsheet applies only to Hot Rolled steel members which have been assigned a Member Type of Beam and which have a seismic design rule applied.

Columns	Beams	Braces											
	Label	Seismic Design Rule	Ductility Req'd	UC Max	LC	Slenderness Checks	Type	Req'd Shear[k]	Req'd Moment[k-ft]	SC/WB Ratio	SC/WB Col	Span/Depth	Misc. Checks
1	M6	OMF	Minimal	1.15	1	Pass	BUEEP	7.078	18.907	N/A	N/A	N/A	Fail
2	M7	OMF	Minimal	0.905	1	Pass	BUEEP	6.919	15.023	N/A	N/A	N/A	Fail
3	M8	OMF	Minimal	0.271	1	Pass	BUEEP	6.305	13.69	N/A	N/A	N/A	Fail
4	M9	OMF	Minimal	0.195	1	Pass	BUEEP	5.742	9.597	N/A	N/A	N/A	Fail

Seismic Results Spreadsheet - Beams

Column Header	Description
Label	The Label column displays the user defined name for the beam. It is used as a unique identifier for each of the seismic design rules defined for the structure.
Seismic Design Rules	Seismic Design Rule displays the specific rule applied to the beam.
Ductility Req'd	<p><i>Ductility Required</i></p> <p>The Ductility Req'd column displays the ductility requirements for the beams. Possible requirements include:</p>

Column Header	Description
	<ul style="list-style-type: none"> • High Ductility: Refers to a member which requires Seismically Compact sections for members such as a Special Concentrically Braced Frame (SCBF) or a Special Moment Frame (SMF). • Moderate Ductility: Refers to a member which require Compact sections for frame members, but which does not require the special Seismic Compactness defined in AISC 341. One example would be an Intermediate Moment Frame (IMF). • Minimal Ductility: Refers to a member which does not have specific compactness requirements beyond the normal AISC specification. This can even include members with slender elements. One example would be an Ordinary Moment Frame (OMF).
UC Max	<p><i>Maximum Code Check</i></p> <p>The UC Max column summarizes the member code checks from all the load combinations that included seismic force. It does NOT include code checks related to connection design like panel zone shear checks or continuity plate checks.</p>
LC	<p><i>Load Combination</i></p> <p>The LC column displays the load combination governing the beam. If the load combination that controlled the design used the overstrength seismic forces (meaning it had an Ω factor in the load combination) then the governing load combination is followed by an asterisk (*).</p>
Slenderness Checks	<p>The Slenderness Checks column summarizes whether the member passed the required element slenderness checks based on the Req'd Ductility setting.</p> <ul style="list-style-type: none"> • For the 2005 code, highly ductile frames are considered to require a "seismically compact" member per the provisions of AISC 341, whereas moderately ductile frames are considered to require a "compact" member per the requirements of the regular AISC 360 design standard. • For the 2010/2016 code seismically compact and compact limits are determine from AISC 341 table D1.1. AISC 360 limits are only used when reporting whether a member should be reported as non-compact, slender.

Column Header	Description
	 <p>Note: For sections that do not have a Compact limit per Table B4.1 of the AISC 360 (i.e. Case 3), and who do not meet or exceed the non-compact limit, RISA will report as "non-slender".</p>
Type	<p><i>Connection Type</i></p> <p>The Type column merely lists the type of moment connection that was specified for the member in the Seismic Design Rules applied to the member.</p>
Req'd Shear	<p><i>Required Shear</i></p> <p>The Req'd Shear column displays the required shear strength for connection design. For Highly ductile connections, this column displays the shear force required to develop a plastic hinge on either side of the beam plus the contribution of shears from the gravity loads. In the member's detail report we refer to these forces as V_{pr} and V_g.</p> <p>Connections which require Minimal ductility are designed for the lesser value of the required hinge shear force ($V_{pr} + V_g$) or the Ω shear developed when considering the overstrength load combinations.</p> <p>For Moderately ductile connections the required shear force for connection design is based on the connection type. Connections listed with a connection type of "other" use the same criteria as the minimally ductile connections. The pre-qualified connections (BUEEP, RBS, et cetera) typically require the same design shear values ($V_{pr} + V_g$) as required for the Highly ductile connections.</p>
 <p>Note:</p> <ul style="list-style-type: none"> • The Other connections assume a value of 1.1 to account for strain hardening (rather than the C_{pr} factor used in AISC 358). • The V_g values are obtained during a batch or envelope solution by looking at the beam shears at the hinge location for every solved load combination which does not include any wind or seismic loading. • If there were no gravity-only load combinations solved then V_g is approximated by taking half the difference in the shear force between the two ends of the beam member. • For minimally ductile moment frames, the required shear need not exceed $R \cdot V_u$ either. Though this is unlikely to ever govern over the $\Omega \cdot V_u$ calculation. 	
Req'd Moment	<i>Required Moment</i>

Column Header	Description
	<p>The Req'd Moment column displays the required Moment strength for connection design. Currently all highly and moderately ductile moment frames beam connections require that the connection be designed for the maximum probable moment projected to the face of the column. For minimally ductile moment frames, the required moment need not exceed $R \cdot \mu$ based on discussion in the AISC 341 commentary.</p>
<div style="border: 1px solid black; padding: 10px;"> <p> Note:</p> <ul style="list-style-type: none"> This moment is not used to code check the actual beam and column members. It is used, however, to determine if the column requires continuity plate stiffeners to resist the connection forces. The Beam Detail Report may also list the Overstrength Moments ($\Omega_0 \cdot \mu$) obtained from solving the overstrength load combinations. These are reported for reference only. </div>	
SC/WB Ratio	<p><i>Strong Column / Weak Beam Ratio</i></p> <p>The Strong Column / Weak Beam ratio is checked for all moment connections between a beam(s) and the column and is reported in the SC/WB Ratio column. The worst case (lowest) ratio is reported here. If the frame ductility requirements dictate a minimum value for this ratio, then the program also lists a pass / fail check in this column.</p>
SC/WB Col	<p><i>Strong Column / Weak Beam Controlling Beam</i></p> <p>The controlling column member is reported to identify which end of the beam resulted in the lowest SC/WB ratio. That information is displayed in the SC/WB Col column.</p>
Span /Depth	<p>The Span/Depth column reports the span to depth ratio for the beam. This value is important because it is used in testing for the AISC 358 pre-qualified connections. Each of the pre-qualified connections have limits on the minimum span to depth ratio for which that moment connection can be used. When the frame ductility requirements and connection type have limits on this value then the program also lists a pass / fail check in this column.</p> <div style="border: 1px solid black; padding: 10px; margin-top: 10px;"> <p> Note: The span used in this calculation is the clear span from column flange to column flange as specified in AISC 358.</p> </div>
Misc. Checks	<p><i>Miscellaneous Checks</i></p> <p>The Misc. Checks column relates to a number of miscellaneous checks required by AISC 358.</p>

Column Header	Description
	<ul style="list-style-type: none"> • Geometry Checks on the Beam (i.e. max. Weight, Depth, Flange Thickness, Flange) • Sloped connections: If beam and column webs are in the same plane, the connection need not be orthogonal. But, the program will not perform seismic moment connection calculations when the slope exceeds 15 degrees. • Strong Axis connections: The program currently only supports strong axis moment connections for seismic code checking. The tolerance for determining if a connected is skewed enough to be disqualified is 5 degrees.

Seismic Results Spreadsheet - Braces

The third tab of the Seismic Design Rules spreadsheet applies only to Hot Rolled steel members which have been assigned a Member Type VBrace and which have a seismic design rule applied.

Label	Seismic Design Rule	Ductility Req'd	UC Max	LC	Slenderness Checks	Req'd Tension[k]	Req'd Comp[k]	Unbalanced Beam	Brace Vert. Force[k]	Brace Hori. Force[k]	Misc. Checks	
1	M10	OCBF	Minimal	2.347	1	Pass	108.533	16.291	N/A	N/A	N/A	Pass
2	M11	OCBF	Minimal	0.213	1	Pass	108.533	17.931	N/A	N/A	N/A	Pass
3	M12	OCBF	Minimal	1.599	1	Pass	108.533	17.931	N/A	N/A	N/A	Pass
4	M13	OCBF	Minimal	0.103	1	Pass	108.533	19.645	N/A	N/A	N/A	Pass

Seismic Results Spreadsheet - Braces

Column Header	Description
Label	<p>The Label column displays the user defined text string which is used as a unique identifier for each of the seismic design rules defined for the structure.</p> <div style="border: 1px solid black; padding: 5px; margin-top: 10px;"> Note: For OCBF frames, the required frame ductility will be listed as minimal even though the element slenderness requirements will enforce the highly ductile (i.e. seismically compact) limits for the brace. </div>
Seismic Design Rules	Seismic Design Rule displays the specific rule applied to the brace.
Ductility Req'd	<p><i>Ductility Required</i></p> <p>The Ductility Req'd column displays the ductility requirements for the brace. Possible requirements include:</p> <ul style="list-style-type: none"> • High Ductility: Refers to a member which requires Seismically Compact sections for members such as a

Column Header	Description
	<p>Special Concentrically Braced Frame (SCBF) or a Special Moment Frame (SMF).</p> <ul style="list-style-type: none"> • Moderate Ductility: Refers to a member which require Compact sections for frame members, but which does not require the special Seismic Compactness defined in AISC 341. One example would be an Intermediate Moment Frame (IMF). • Minimal Ductility: Refers to a member which does not have specific compactness requirements beyond the normal AISC specification. This can even include members with slender elements. One example would be an Ordinary Moment Frame (OMF).
UC Max	<p><i>Maximum Code Check</i></p> <p>The UC Max column displays the combined bending and axial code check for the brace. This code check is based on non-overstrength load combinations only, unless the Seismic Design Rule for the brace has been designated as Overstrength Req'd. The Load Combination number which resulted in the highest UC value is reported. If the governing load combination contained overstrength seismic forces (Ω_0) it will be followed by an asterisk (*).</p>
LC	<p><i>Load Combination</i></p> <p>The LC column displays the load combination governing the brace. The Load Combination number that results in the highest UC value is reported. If the governing load combination contained overstrength seismic forces (Ω_0) it is followed by an asterisk (*).</p>
Slenderness Checks	<p>The Slenderness Checks column summarizes the worst case condition between local element compactness checks and KL/r slenderness restrictions (if any). This field summarizes whether the brace passed the element compactness checks based on the Req'd Ductility setting. Special Concentrically Braced Frames require a "seismically compact" member per the provisions of AISC 341.</p> <p>This also indicates when a brace fails the KL/r restrictions listed in AISC 341.</p>

Column Header	Description
	<div data-bbox="170 367 235 430" style="float: left; margin-right: 10px;">  </div> <p>Note:</p> <ul style="list-style-type: none"> • For OCBF frames (which have minimal required frame ductility) still require that the bracing member's element slenderness meet the highly ductile (i.e. seismically compact) limits of AISC 341. • For sections that do not have a Compact limit per Table B4.1 of the AISC 360 (i.e. Case 3), and who do not meet or exceed the non-compact limit, RISA will report as "non-slender".
<p>Req'd Tension</p>	<p><i>Required Tension</i></p> <p>The Req'd Tension field summarizes the axial forces tension required for the design of the brace connections. The connection design forces are separated into tension and compression design forces. These forces are determined based on the capacity of the brace itself.</p> <div data-bbox="560 934 625 997" style="float: left; margin-right: 10px;">  </div> <p>Note: For frames with Minimal ductility, where Overstrength is required for the brace, the maximum brace axial load from Overstrength (Ω_0) load combinations is reported if it exceeds the capacity-based connection design forces.</p>
<p>Req'd Comp</p>	<p><i>Required Compression</i></p> <p>The Req'd Tension field summarizes the axial forces compression required for the design of the brace connections. The connection design forces are separated into tension and compression design forces. These forces are determined based on the capacity of the brace itself.</p> <div data-bbox="560 1417 625 1480" style="float: left; margin-right: 10px;">  </div> <p>Note: For frames with Minimal ductility, where Overstrength is required for the brace, the maximum brace axial load from Overstrength (Ω_0) load combinations is reported if it exceeds the capacity-based connection design forces.</p>
<p>Unbalanced Beam</p>	<p>For brace configurations which the program identifies as V or Inverted-V (Chevron) the program automatically calculates the unbalanced beam forces per the provisions of AISC. The resultant unbalanced vertical force on the beam is reported in the Unbalanced Beam column.</p> <p>For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search</p>

Column Header	Description
keywords: Unbalanced .	
 Note: <ul style="list-style-type: none"> For braces that are non-symmetric the most unbalanced combination of tension/compression is reported. The horizontal and vertical components are shown relative to the beam's local axes. (Up) and (Down) refer to the direction of the resultant force on the beam relative to its positive local y axis. These unbalanced forces are not actually used in the design of the beam member. They are reported for reference only. The beam must have a seismic design rule assigned to it in order to view these forces. The requirement that the beam be analyzed as though the brace carried no dead or live load cannot be met by RISA-3D, since the program cannot force certain members to only carry certain components of a load. This can be accomplished by integrating the model with RISAFloor though, which will analyze the beam for dead and live loads as though the brace were not present. 	
Brace Vert. Force	<i>Brace Vertical Force</i> Axial force in the Vertical Brace.
Brace Hori. Force	<i>Brace Horizontal Force</i> Axial force in the Horizontal Brace.
Misc. Checks	<i>Miscellaneous Checks</i> There are no miscellaneous brace checks in the program at this time. The Misc. Checks column has been reserved for future additions. This column will always report "Pass".

 **Note:** reported if it exceeds the capacity-based connection design forces.

Seismic Detailing - Detail Reports

The seismic detailing results are presented in two ways: the seismic detailing results spreadsheet and the seismic detailing sections of the member detail reports. The seismic detailing results spreadsheet is intended to be a summary report. More detailed information can be found for each member on that member's detail report as described below. The information described here is based on enveloped information from a batch solution.

Design Forces for Moment Connections

The seismic portion of hot-rolled steel detail reports for beams in moment frames contain sections called **Required Connection Shear Strength** and **Required Connection Moment Strength**.

Required Connection Shear Strength				
Column	Vg (k)	Vpr/1.5 (k)	R*Vu(k)	Demand
M1	-0.969	9.828	3.708	2.292
M2	-0.969	9.828	-5.983	4.913

Required Connection Moment Strength					
Column	Hinge Loc(in)	Ze(in3)	Mpr/1.5 (k-ft)	R*Mu(k-ft)	Demand(k-ft)
M1	12	8.87	31.168	-4.194	4.194
M2	12	8.87	31.168	12.507	12.507

Required Connection Shear Strength

The Vg listed is the shear at the plastic hinge location in the beam due to gravity loads. RISA determines this value by taking the worst case shear from the load combinations which do NOT include any wind or seismic loads. If there are no solved LC's which meet this criteria, then the value is taken as half the difference in the end shears from the seismic LC's.

RISA uses the term Vpr to refer to the shear required to produce the maximum probable hinge moment (Mpr) on both sides of the beam.. This value will be equal to $2 * Mpr / \text{Distance between plastic hinge locations}$. If the connection is Other / None then the Mpr will be based on a Cpr of 1.1 per AISC 341 assumptions. If the connection is a WUF-W connection, this will be based on a Cpr of 1.4. Otherwise, the calculation will be based on the Cpr of the material as defined in AISC 358.

The $\Omega_o * Vu$ value listed in the detail report is the shear demand based on the worst case of the Overstrength Load Combinations. This is presented because AISC 341 (Section 10.2a in the 2005 code or Section E1.6b in the 2010/2016 code) says that moderately and minimally ductile frames need not have their shear connection designed to a force greater than this value.

Per AISC 341, the required shear strength Demand for a connection requiring high ductility will be based on the Vpr value plus the shear due to gravity loading (Vg). For minimal ductility, the required shear strength Demand will be based on the lesser of this shear (Vpr + Vg) or the shear demand from the overstrength load combinations.

**Note:**

- The V_g values are obtained during a batch or envelope solution by looking at the beam shears at the hinge location for every solved load combination which does not include any wind or seismic loading.
- If there were no gravity-only load combinations solved then V_g is approximated by taking half the difference in the shear force between the two ends of the beam member.
- In addition, the program will not allow the design shear to exceed $R \cdot V_u$. Though this is not likely to ever control over the $\Omega_o \cdot V_u$ loading.

Required Connection Moment Strength

The required Moment strength for connection design is listed here along with the values used to compute it.

The **Hinge Location** reported here is the Hinge Location from the assigned Seismic Design Rule. This represents the distance from the face of column to the center of the plastic hinge.

The **Ze** value is the effective plastic section modulus of the hinge. It is equal to the plastic section modulus of the beam multiplied by the Z Factor from the assigned Seismic Design rule.

The **Mpr** value is the probable plastic moment projected to the face of the column. This would be equal to $Cpr \cdot Ry \cdot Fy \cdot Ze$.

The **Demand Moment** is the Mpr moment projected to the face of the column. The projection of this moment must account for the extra moment due to the shear force at the hinge location times the distance from the hinge location to the face of column. This would be equal to $Mpr + (Vpr + Vg) \cdot \text{Hinge Location distance}$. This value will be used as the demand moment for calculating the beam flange force demand to determine if the column requires continuity plate stiffeners.

The $\Omega_o \cdot Mu$ is reported mostly for reference. However, it may be used to determine the shear demand on the column's panel zone. In addition, moment frames requiring minimal ductility will not allow the demand moment to exceed $R \cdot Mu$.

Column Panel Zone Capacity Calculations for Moment Frames

The seismic portion of hot-rolled steel detail reports for columns in moment frames contains a section called **Panel Zone Summary**.

Panel Zone Summary Fail (Web Reinf. Required)				
Beam M8	Demand	LC	Capacity	UC
360-16: Eqn J10-9	$V_u = 124.472 \text{ k}$	3	$V_n/O_m = 24.095 \text{ k}$	5.166
341-16: Eqn E3-7	$tcw > 0.166 \text{ in}$	N/A	$tcw = 0.17 \text{ in}$	0.978
Beam M10	Demand	LC	Capacity	UC
360-16: Eqn J10-9	$V_u = 124.472 \text{ k}$	3	$V_n/O_m = 24.095 \text{ k}$	5.166
341-16: Eqn E3-7	$tcw > 0.166 \text{ in}$	N/A	$tcw = 0.17 \text{ in}$	0.978

For frames that require high ductility (SMF), the Demand V_u is based on the maximum probable moment projected to the face of column. In addition, the program will check the panel zone for column beam connections per the extra seismic detailing checks of AISC 341 (eqn 9-2 in the 2005 code or E3-7 in 2010/2016). That is the requirement that the thickness of the column web is greater than the sum of the depth and width of the panel zone divided by 90.

For frames which do not require high ductility (IMF and OMF), the panel zone shear demand will be based on the end moments from the solved load combinations. If the Overstrength Req'd flag has been set for the column, then the $\Omega_o \mu_u$ values listed in the section above (Design Forces for Moment Connections) will be used instead.

**Note:**

- Currently, the effects of column story shear (which tend to decrease the panel zone shear demand) are not taken into account.
- These conditions are checked for each beam connected to the column with a valid seismic moment connection. However, the total panel zone shear demand is based on the sum of the panel zone shears from the individual beams. This may be over-conservative for cases with an IMF or OMF frame where the maximum moments in the beams occur for different load cases.

Continuity Plate Checks for Columns in Moment Frames

The seismic portion of hot-rolled steel detail reports for columns in moment frames contains a section called **Continuity Plate Summary**.

Continuity Plate Summary Fail (Flange Stiffener Required)					
Beam M8	Demand	LC	Capacity	UC	Eqn
Flange Bending	Puf = 53.216 k	N/A	N/A	N/A	358-16: 2.4.4(1) Eqn 6.8-14
Web Crippling	Puf = 53.216 k	N/A	8.543 k	6.229	358-16: 2.4.4(1) Eqn 6.8-23
Web Buckling	Puf = 53.216 k	N/A	5.159 k	10.315	358-16: 2.4.4(1) Eqn 6.8-20
Web Yielding	Puf = 53.216 k	N/A	10.328 k	5.153	358-16: 2.4.4(1) Eqn 6.8-17
Beam M10	Demand	LC	Capacity	UC	Eqn
Flange Bending	Puf = 65.529 k	N/A	N/A	N/A	358-16: 2.4.4(1) Eqn 6.8-14
Web Crippling	Puf = 65.529 k	N/A	8.543 k	7.671	358-16: 2.4.4(1) Eqn 6.8-23
Web Buckling	Puf = 65.529 k	N/A	5.159 k	12.701	358-16: 2.4.4(1) Eqn 6.8-20
Web Yielding	Puf = 65.529 k	N/A	10.328 k	6.345	358-16: 2.4.4(1) Eqn 6.8-17

These checks come from either the AISC 358, AISC 341 or the AISC 360 depending on the required ductility of the system and the type of moment connection used.

Required Flange Force Calculations

The flange force, P_{uf} , is the demand force that the column has to resist. This force is based on the probable maximum moment at the face of column (see the Demand Moment from the Design Forces

for Moment Connections section listed above). This will be used for the [AISC 358 Continuity Plate Checks](#).

**Note:**

- When the WUF-W connection is used the program will calculate the probable maximum moment at the face of the column based on a C_{pr} of 1.4. This connection assumes a large degree of strain hardening which artificially increases the probably maximum moment over the other pre-qualified connections with a similar beam and material.
- When an Other / None connection is used, the C_{pr} is always assumed to be equal to 1.1.
- When ASD design is used the required Flange Force for connection design is determined by dividing the probably maximum moment at the face of the column by a factor of 1.5.

AISC 341-05 Continuity Plate Checks

Column Flange Thickness Requirement (AISC 341: Section 11.5 for 2005, no equivalent in 2010/2016)

For wide flange columns in moment frames which require minimal ductility (OMF's), continuity plates are required if the equations from section 11.5 are not met. For connections which require higher ductility (SMF or IMF) see the AISC 358-05 Continuity Plate Checks listed below.

AISC 358/341 Continuity Plate Checks

Column Flange Thickness Requirement (AISC 358: Section 2.4.4 in 2005, E3.6f in 2010/2016)

For wide flange columns with a pre-qualified moment connection (except for Extended End Plates) continuity plates are required if column flange thickness does not meet the requirements of these sections. This check is enforced for only high and intermediate ductility requirements (SMFs and IMF's) in 2005, but is checked for all frames in 2010/2016.



Note: If the connection type is listed as Other / None then this check is not enforced even for frames with high ductility requirements.

End Plate Moment Connections (BUEEP and BSEEP from AISC 358: Section 6.10)

Moment connections requiring high or intermediate ductility (SMFs and IMFs) which use an end plate moment connections require the continuity plate checks described in Section 6.10 of AISC 358.

For End Plate Moment Connections (BUEEP and BSEEP) the program will check following concentrated force failure modes to determine if continuity plates are required:

- Column Web Yielding (per AISC 358-05: Equation 6.9-24)
- Column Web Crippling (per AISC 358-05: Equation 6.9-29 through 6.9-31): With the assumption that $N = 2 \cdot t_{bf}$
- Column Web Compression Buckling (per AISC 358-05: Equation 6.9-26 and 6.9-27): With the caveat that this check is only performed if the moment connection has beams framing in on both sides of the column.



Note:

- These checks are really just variations of the normal AISC 360 continuity plate checks. However, they have been customized to account for the presence of the end plates. For this reason, the BUEEP and BSEEP moment connections do NOT check the AISC 360 Continuity Plate requirements as they would be considered redundant.
- The Column Flange Bending checks of equation 6.9-21 are not performed. These checks require the calculation of the Y_c term which depends on the number of bolts and the geometry of the connection. Thus, it can not be calculated and an N/A is presented instead.
- The bolted end plate moment connection equations from AISC 358 are only formulated for LRFD design. Therefore, in cases where ϕ equals 0.75, an Ω value of 2.0 will be assumed. Similarly, in cases where ϕ equals 0.9, an Ω value of 1.67 will be used.

AISC 360 Continuity Plate Checks

For SMFs, IMFs and OMFs that are not using BUEEP and BSEEP connections AISC 341 requires that the connections still meet the requirements of AISC 360-05 Section J10. The program will check the following concentrated force failure modes to determine if continuity plates are required:

- Column Flange Bending (per AISC 360-05: Equation J10-1.)
- Column Web Yielding (per AISC 360-05: Equation J10-2 and J10-3.) With the assumption that $N = t_{bf}$
- Column Web Crippling (per AISC 360-05: Equation J10-4, J10-5a and J10-5b.) With the assumption that $N = t_{bf}$

- Column Web Compression Buckling (per AISC 360-05: Equation J10-8.) With the caveat that this check is only performed if the moment connection has beams framing in on both sides of the column.

Multiple Moment Connections Framing to a Column

When a column with seismic design rules has a beam framing in from each side, each beam will show up separately in the detail report with its own continuity plate checks. Since these checks are local to the beam flange to column flange force transfer, the program does not assume any interaction between the continuity plate checks of the two beams.

Strong Column / Weak Beam (SC/WB) Moment Ratios

The seismic portion of hot-rolled steel detail reports for columns in moment frames contains a section displaying the **Column Beam Moment Ratio** which is frequently referred to as the Strong Column / Weak Beam Moment ratio.

Column Beam Moment Ratio (Fail)				
Beam	Sum M^*_{pc} (k-ft)	Sum M^*_{pb} (k-ft)	Ratio	Eqn
M8	23.655	88.455	0.267 (Fail)	358-16 6.4(2)
M10	23.655	88.455	0.267 (Fail)	358-16 6.4(2)

This value is reported for reference only for frames that require Minimal or Moderate Ductility (OMF's and IMF's).

For frames that require High Ductility (SMF's), any value less than 1.0 is considered to fail the SC/WB code check per AISC 341(Section 9.6 in 2005 and E3.4 in 2010/2016). These frames report a warning that connection bracing is required anytime the SC/WB ratio is calculated as less than 2.0 per the AISC 341 section for unbraced connections.



Note:

- The Sum M^*_{pb} value is based on the maximum probable moment at the plastic hinge location projected to the centerline of the column.
- For two sided moment connections, this Sum M^*_{pb} value assumes that the shear force from gravity loads (V_g) adds to the plastic hinging shear force (V_{pr}) on one side of the connection and takes away from it on the other side.

Bracing Requirements for Beams in a Moment Frame

The seismic portion of hot-rolled steel detail reports for beams in moment frames contains a section called **Beam Flange Bracing**. This section reports the required beam flange bracing forces for moment connection. These values depend on ductility requirements of the connection.

Beam Flange Bracing

Reqd Strength	0.846 k	360-16: Eqn A-6-7
Reqd Strength	2.539 k	341-16: D1.2c
Reqd Stiffness	10.055 k/in	360-16: Eqn A-6-8
Max Spacing	84.181 in	341-16: D1.2a

For connections which require high or moderate ductility (SMF, or IMF), the L_b values used in Equation A-6-8 are based on Maximum Spacing values shown in the detail report. These maximum spacing for beam bracing is based on the requirements of AISC 341 (sections 9.8 / 10.8 in 2005 and E2.4a / E3.4b in 2010/2016).

For connections which require minimal ductility (OMF), L_b values used in Equation A-6-8 are based on the maximum of $L_{comp-top}$, $L_{comp-bottom}$ or L_p as defined in AISC 360-05 equation F2-5.

**Note:**

- The C_d values in equations A-6-7 and A-6-8 are always assumed to be 1.0 based on the commentary to AISC 341.
- For RBS connections, the M_r values used in equations A-6-7 and A-6-8 are assumed to be equal to the plastic section modulus at the reduced beam section.

Requirements for Braced Frames

In addition to the input echo and the Element Slenderness checks, braced frames display the following sections in the member detail report. These sections are displayed for the Brace Member and (in the case of V or inverted V braced systems) the beam member attached to the V bracing.

Brace Connection Strength**Required Connection Axial Strength**

Comp.(1.14*Fcre*Ag/1.5):	15.297 k	Tens.(Ry*Fy*Ag/1.5):	108.533 k
Pu:	0.331 k	Pu:	2.515 k

The three brace connection design forces are all presented for reference, although the actual design of the connection is outside of the scope of RISA-3D. The first two are determined based on the capacity of the brace itself. The third value is based on the maximum force in the brace from the solved load combinations. If the brace has been specified to be designed for Overstrength (Ω_0) then this value will be based on the Overstrength load combinations.



Note: When using the 2010/2016 code, the expected brace strength in compression is based on the normal AISC 360 chapter E requirements computed assuming a yield stress equal to $R_y * F_y$.

Unbalanced Forces

For brace configurations which the program identifies as V or Inverted-V (Chevron) the program automatically calculates the unbalanced beam forces per the provisions of AISC 341. The resultant unbalanced forces on the beam are reported.

Unbalanced Force on Beam (unfactored)--only considered in beam code check if Capacity-Limited LCs are used--			
Beam	M3		
Brace 1	M16	9.089 k	341-16: F1.4
Brace 2	M15	162.8 k	341-16: F1.4
Horizontal Component		70.537 k	
Vertical Component		140.172 k	(Down)

For more information see the [Seismic Detailing Results](#) topic; For additional advice on this topic, please see the RISA Tips & Tricks section of the website: www.risa.com/post/support. Type in Search keyword: **Unbalanced**.

Brace KL/r Restriction

$$KL/r = 156.602 > 96.333 \text{ Fail per 341-16: F1.5b}$$

The program will report the KL/r check for certain brace members based on the user input setting for the [Seismic Design Rules](#). Examples where a user would likely select this input would be braces that are part of a Special Concentrically Braced Frame (SCBF) as well as K, V, or inverted V braces in an Ordinary Concentrically Braced Frame (OCBF).

Buckling Restrained Braces

The adjusted brace strength and required connection strength for BRB are different than other HR members. The compression strength adjustment factor (β) and strain hardening adjustment factor (ω) are included in the calculation. See [Hot Rolled Steel - Design](#) for additional information on these parameters.

Adjusted Brace Strength		
ω	1.267	
β	1.161	
Ry=1.0 for CoreBrace BRB		
Tension	$P_{tmax} := \omega \cdot R_y \cdot F_{ymax} \cdot A_{sc} = 174.818 \text{ kips}$	341-16 F4.2a
Compression	$P_{cmax} := \beta \cdot \omega \cdot R_y \cdot F_{ymax} \cdot A_{sc} = 203.041 \text{ kips}$	341-16 F4.2a
Required Connection Axial Strength		
Tens. (Ptmax/1.5)	116.545 kips	341-16 F4.6c1
Comp. (Pcmax/1.5)	135.361 kips	341-16 F4.6c1

Miscellaneous Seismic Checks

Element Width-to-Thickness Ratios Checks

This section of the detail report displays the results of the element Width-to-Thickness ratios checks per the provisions of AISC 341.

Seismic Design Rule:	SCBF	Flange Width-to-Thickness Ratio:	Non-Slender (Fail)
Design to Ω_0 Loads:	Yes	Web Width-to-Thickness Ratio:	Seismically Compact

- AISC 341 Table D1.1 requires (based on the frame ductility and member type) that the Width-to-Thickness ratios of the cross section satisfy certain criteria. Members do not meet the criteria will report Fail using red text to indicate that the member has failed the AISC 341 criteria.

Miscellaneous AISC 358 Pre-Qualification Checks

This section of the detail report displays the results of some of the miscellaneous provisions of AISC 358. Specifically those sections related to pre-qualification of the beam-column moment connection. Some of these requirements are related to the beam or column dimensions (size, weight, et cetera). Some of them are related to the clear span to beam depth ratio.

Col-BM Moment Ratio	1.184 (For Reference Only)
Span to Depth Ratio	19.116 (Pass)

Column Beam Moment Ratio (Fail)				
Beam	Sum M^*_{pc} (k-ft)	Sum M^*_{pb} (k-ft)	Ratio	Eqn
M19	24.289	45.166	0.538 (Fail)	358-16 9.4(2)



Note: The program is not checking any of the AISC 358 weld restrictions.

Other Miscellaneous Checks

The program produces warning messages (in the detail report only) relating to the various AISC 341 restrictions. Among them are the provisions which require columns with a L/r ratio greater than 60 to be braced out of plane as shown below.

L/r = 142.808 > 60 per 341-16 E3.4c(2)(b) (For Reference Only)

Shape Databases

For each material, there are several databases of common structural shapes such as Hot Rolled Steel Wide Flanges, Cold Formed Shapes, Wood, Concrete Tees, etc. You may also choose from shapes created in RISASection. You may type in the names directly, select shapes from these databases or add your own shapes.

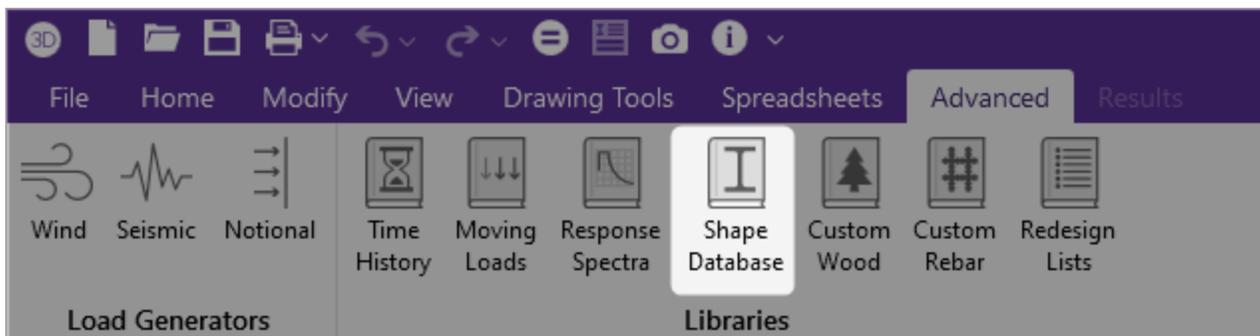
- [What Aluminum Shapes are available?](#)

Database Shape Types

There are different types of shapes for each material type including General shapes. Names for each shape type follow a syntax so that they can be typed directly into the **Shape** box on the **Section Sets** spreadsheet or on the **Primary** tab of the **Members** spreadsheet. Alternately you may click the  button to look up a shape and select it.

Hot Rolled Shapes

AISC, Canadian, Trade Arbed and custom Hot Rolled shapes are accessed by clicking the **Shape Database**  icon from the **Advanced** ribbon and then clicking the **Hot Rolled** tab from the **Shape Database Editor** window.



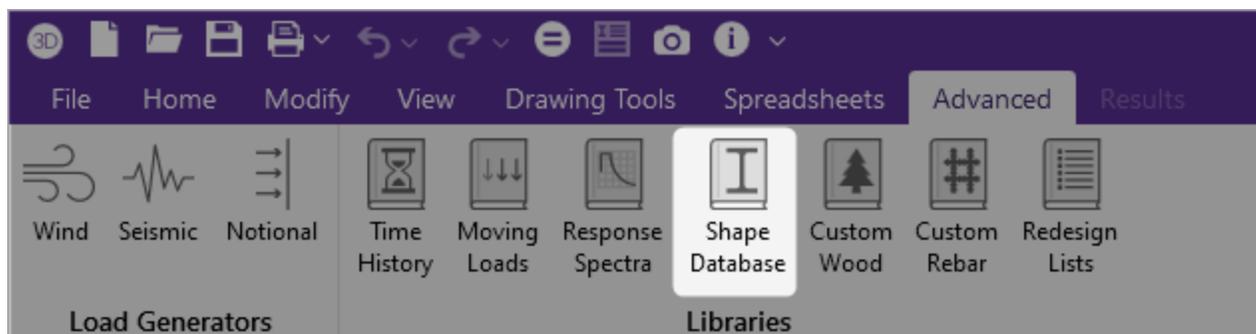
The hot rolled shapes and databases are more fully described in the Hot Rolled Steel Design section. See [Hot Rolled Steel Databases](#) for more information.

Virtual Joists and Virtual Joist Girders

The Steel Joist Institute (SJI) has put together Virtual Joist and Virtual Joist Girder tables which convert common joist and joist girder sizes into equivalent wide flange beams. These shapes are available by selecting **Virtual Joist Girder** or **Virtual Joist** from the **Database** menu. For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Virtual Joist Girders**

Cold Formed Shapes

Manufacturer and custom cold formed shapes are accessed by clicking the **Shape Database**  icon from the **Advanced** ribbon and then clicking the **Cold Formed** tab from the **Shape Database Editor** window.



The cold formed shapes and database are more fully described in the Cold Formed Design section. See [Cold Formed Steel Databases](#) for more information

Concrete Shapes

Concrete shapes do not have a predefined database like hot rolled and cold formed steel. Instead, they are defined using a parametric shape code that may be assigned any depth or width. There are two types of shapes currently supported: Rectangular and Round. See [Concrete Database](#) for more information.

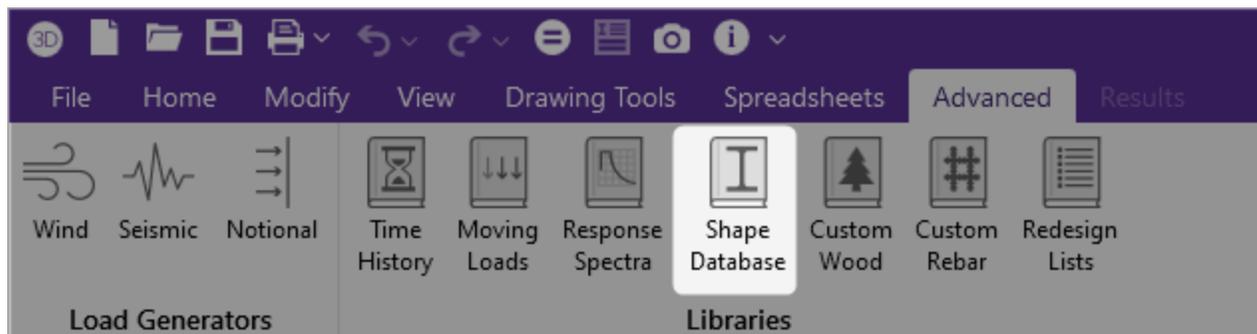
Wood Shapes

The available wood shapes are based on the dimension lumber and post and timber shapes given in the NDS or CSA O86, depending on what code you have chosen as your Wood design code. You may also design for multiple plies of these shapes. Note that the NDS dimension lumber shapes are all *nominal* sizes. CSA O86 shapes are *actual* sizes.

Allowable stress values for each shape are based on the species and grade information given in the selected design code. See [Wood Database](#) for more information.

Aluminum Shapes

US, Canadian and custom Aluminum shapes are accessed by clicking the **Shape Database**  icon from the **Advanced** ribbon and then clicking the **Aluminum** tab from the Shape Database Editor window.



The aluminum shapes and databases are more fully described in the [Aluminum - Databases](#) section.

General Shapes

Arbitrary Shapes

Arbitrary Shapes are a special, catch-all shape. This arbitrary shape type is provided so that any shape can be added to the shape database.

AISC code checks are not calculated for arbitrary shapes since their place in the specification is unknown. Everything else will be calculated for them (forces, deflections, stresses). The max thickness (Max thick) value for the cross section is used to determine the pure torsional shear stress for the shape. "J" is the torsional constant. The "d" values (the distances to the extreme fibers) allow the program to calculate stresses at the extreme fibers.



Note:

- These shapes will generally be rendered using a greenish cruciform shape. The center of the cruciform will reflect the centroid of the section with the tips of the cruciform representing the distances from the neutral axis to the extreme fiber.
- Refer to Member Shear Deformations and Member Shear Stresses for more information on the shear area factors (As-zz Def, As-yy Def, .As-zz Stress, & As-yy Stress) shown in the figure below.

View Shape
? X

Basic Properties

Shape Name

Area	<input type="text" value="4.25"/>	in ²	As-zz Def	<input type="text" value="1"/>	(factor)
I _{yy}	<input type="text" value="104"/>	in ⁴	As-yy Def	<input type="text" value="1"/>	(factor)
I _{zz}	<input type="text" value="15"/>	in ⁴	As-zz Stress	<input type="text" value="1"/>	(factor)
J	<input type="text" value="0.5"/>	in ⁴	As-yy Stress	<input type="text" value="1"/>	(factor)
d _{ytop}	<input type="text" value="1.158"/>	in	Max. Thickness	<input type="text" value="0.188"/>	in
d _{ybot}	<input type="text" value="2.642"/>	in			
d _{ztop}	<input type="text" value="6.5"/>	in			
d _{zbot}	<input type="text" value="6.5"/>	in			

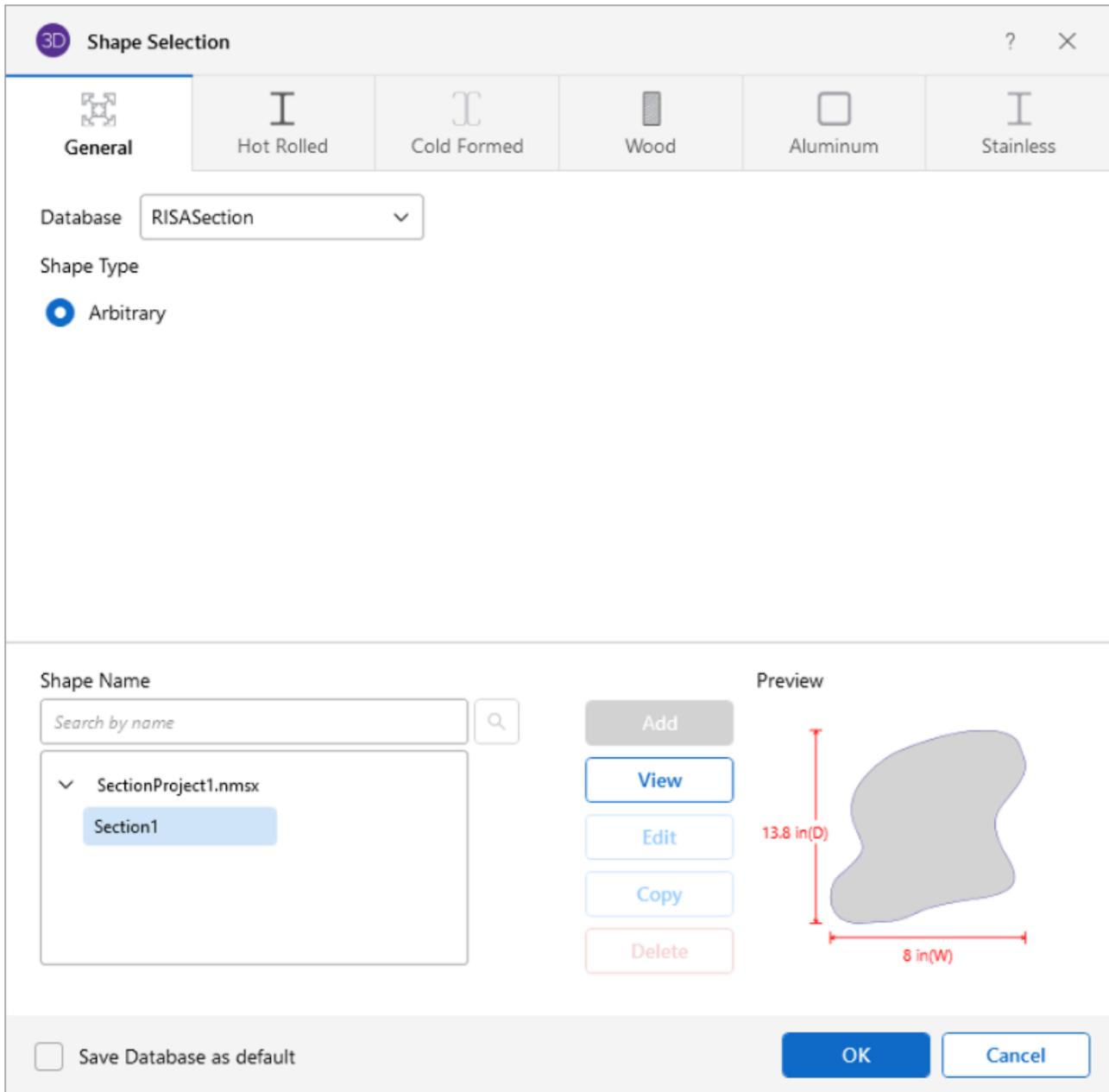
To Add an Arbitrary Shape to the Database

- To enter an arbitrary shape in the database, click the **Shape Database** icon from the **Advanced** ribbon and then clicking the **General** tab from the Shape Database Editor window. Set the shape type to Arbitrary, click **Add** and then enter the shape name and properties.

RISASection Files

You can create simple or complex sections in RISASection and then import those sections for use in your model. Sections that exist in RISASection (files located in the "RISASection" file specified in **File - Application Settings - File Locations**) will be available for use in the model. Each section must have a unique name for it to be available.

The shapes that are designated as General Material, Arbitrary Shape Type in RISASection will show up in the **General** tab under the "**RISASection**" Database.



The shapes that are designated as Hot Rolled Steel Material in RISASection will show up in the **Hot Rolled** tab under the appropriate Shape Type (Channel, Wide Flange, etc.) when "**RISASection**" is selected as the **Database**.

3D Shape Selection
? X


 General


Hot Rolled


 Cold Formed


 Wood


 Aluminum


 Stainless

Database: RISASection

Shape Type

<input type="radio"/> WF	<input type="radio"/> Double Angle
<input type="radio"/> Tube	<input type="radio"/> Single Angle
<input type="radio"/> Pipe	<input type="radio"/> Solid Rectangle
<input type="radio"/> Channel	<input type="radio"/> Solid Round Bar
<input type="radio"/> Double Channel	<input checked="" type="radio"/> Tapered WF
<input type="radio"/> WT	

Shape Name

W36x210MC18x42.7.nmsx

Section1A

Preview

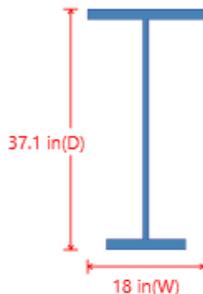
Add

View

Edit

Copy

Delete



Save Database as default

OK

Cancel



Note: Currently, RISASection can only import General and Hot Rolled Steel shapes. More material choices will be available in a future version.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **RISASection Integration**.

Troubleshooting RISASection / RISA- Interaction

There are a few common issues that arise when attempting to access RISASection files from within RISA-.

- When RISA-3D launches, it will only import RISASection files that exist in the directory specified in **File - Application Settings - File Locations**.
- RISA-3D only reads in database files when it is first launched. Therefore, if a RISASection file is created or modified, then RISA-3D must be closed and re-started before it will recognize the new or modified section.

There are some common mistakes that are made from within the RISASection program that may cause an issue when trying to read that shape from within RISA-3D.

- The name of the RISASection "File" is actually different from the name of the section itself. Both names will appear in the RISA-3D database, but it is important to give your sections unique names in order to properly tell them apart. Note: each RISASection file can contain many different sections. Hence the need to name each individual section.
- Unless specifically identified as a Hot Rolled Material type and shape (from within RISASection), the shapes will be imported as Arbitrary shapes with a General Material type
- RISASection can save files with a new *.nmsx file type (for RISASection 2.0 or higher), or an old *.nms (for RISASection version 1.1 or older). If saved as the old file type, the sections will only come in as Arbitrary members with a General material type.
- If the file is saved with a *.nmsx extension it will not be read into older versions of the RISA programs (prior to RISA-3D version 9.1.1).

Structural Desktop (SDT) Shapes

The SDT database is provided by the Structural Desktop software in a file called SDT.FIL. Structural Desktop automates drawing production of RISA models. The SDT database is provided for shapes that are not directly supported in RISA models (such as bar joists) but are available in Structural Desktop. For more information on Structural Desktop see www.structuraldesktop.com.

On-Line Shapes

On-Line shapes are shapes whose dimensions are defined directly in the syntax of the shape name. On-line shapes are not stored in the shape database because there is enough information from the label syntax to calculate all the shape properties. A pipe, for example, can be fully defined by specifying the thickness and diameter.

These shapes are treated just like database shapes for stress calculations. Currently, Pipes, Solid Rectangular and Solid Circular shapes are defined on-line as discussed below in Pipe Database Shapes, Solid Rectangular Shapes, and Solid Circular Shapes.

Pipe Database Shapes

Pipe shapes, which are hollow circular shapes, are entered as on-line shapes. The syntax for these shapes is "PIdiaXthick", where "dia" is the pipe outside diameter and "thick" is the pipe thickness (in inches or centimeters). For example (assuming US Standard units), PI10X.5 would be a 10" diameter pipe with a wall thickness of 1/2".

Solid Rectangular Shapes

These shapes can be defined as on-line shapes. The syntax is "REhtXbase", where "ht" is the rectangle height and "base" is the rectangle base (in inches or cm). For example, RE10X4 would be a 10" deep, 4" width rectangular shape (assuming US Standard units). These shapes can also be defined in the Shape Editor. When defined in the Shape Editor the depth of the solid rectangular section must always be greater than or equal to the width.

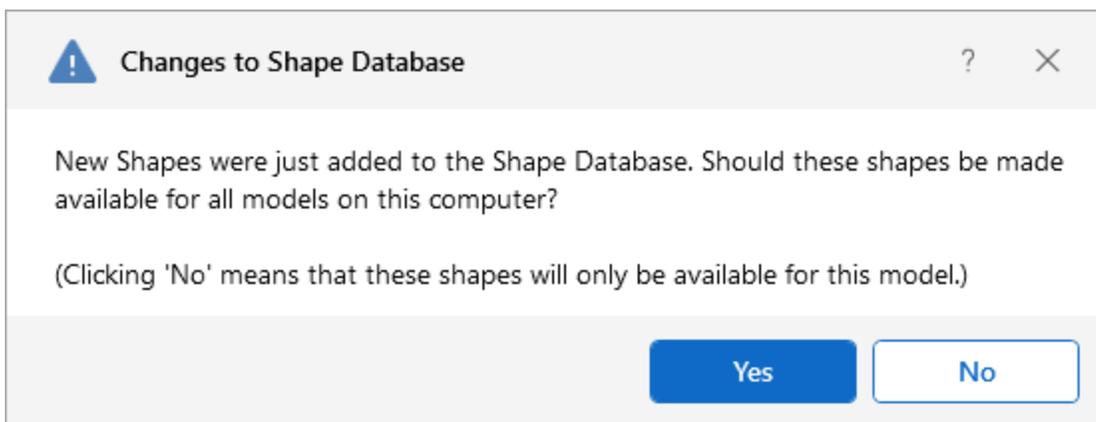
Solid Circular Shapes

These shapes are defined as on-line shapes. The syntax is "BARdia", where "dia" is the circle diameter. For example (assuming metric units), BAR2 would be a circular bar with a diameter of 2 cm.

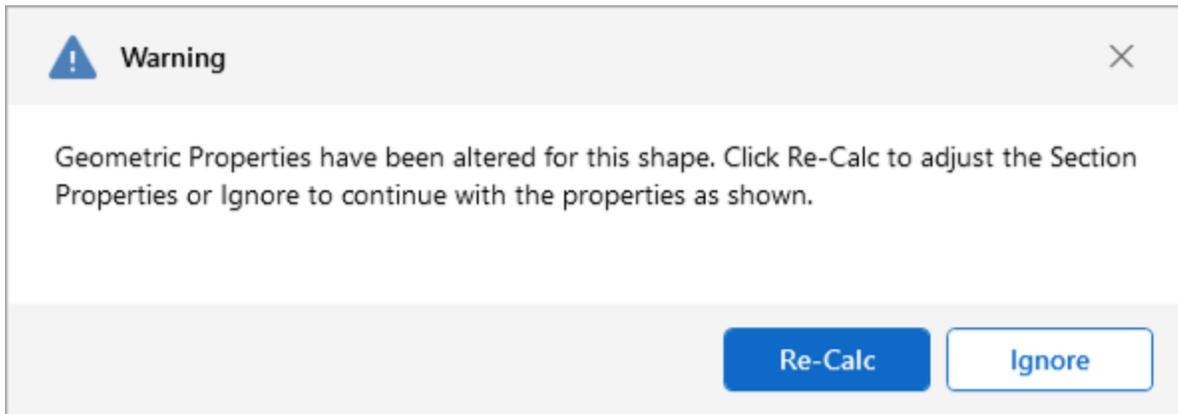
Database Files

The shape databases are stored in the database files (*.FIL). These files may only be edited through the program. The path to these files is set in the **File Locations** tab of **Application Settings**, which is found in the **File** menu.

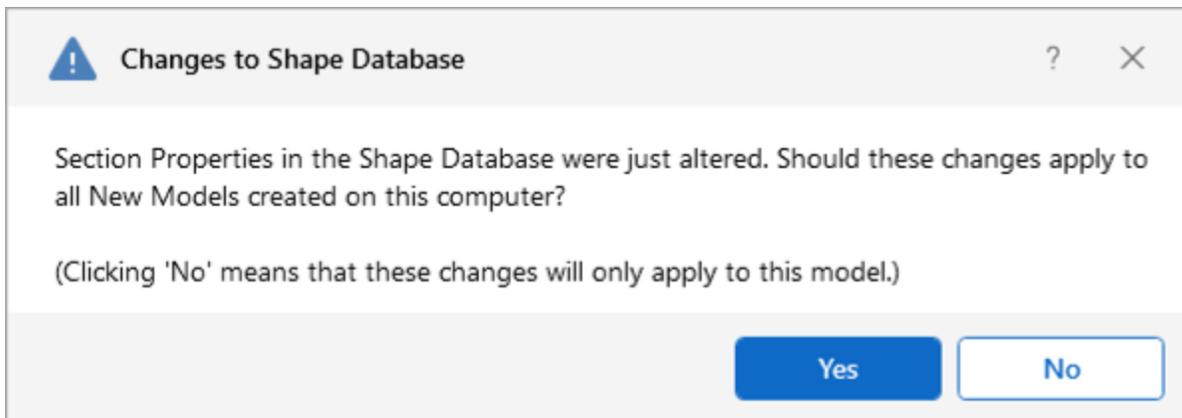
After adding a new shape in your model, you will be prompted with this Changes to Shape Database dialog. Click 'Yes' if you want to add the shape to your Shape Database to be available for all future models. Click 'No' if you do not want the new shape to be available for other models, this will not save the shape to your Shape Database.



You can also make edits to the geometric and section properties of the shapes in your Shape Database. If you modify geometric properties, you can also use the Re-Calc button for the program to automatically calculate the corresponding section properties.



After editing the geometric or section properties for a shape in your model, you will be prompted with this Changes to Shape Database dialog. Click 'Yes' if you want to add the shape modifications to your Shape Database to be available for all future models. Click 'No' if you do not want the changes to apply to other models, this will not save the changes in the Shape Database for other models.

**Note:**

- Alterations to the shape databases are not permanent unless you agree to save them. Changes that are not saved only remain valid for the current session and will not be present the next time you start RISA.
- New shapes are always added to the bottom of the database in blue text.
- 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. Only shapes that are not part of the default database are able to be edited. If the shape you would like to edit is a default database shape, the program will ask if you would like to make a copy of the shape. Geometric values can be edited here, and will allow you to recalculate the section properties if edits have been made.

Solid Elements

The Solid or 8 node brick element allows you to model structures that are too thick to be modeled by plate elements. Common applications could include dam models, extremely thick pile caps or vibrating equipment with extremely thick support slabs. Essentially, these elements should be used whenever the Mindlin – Reissner assumption of linear strain through the thickness of the element would not be appropriate.

Solid elements may be viewed / edited in a couple of ways, but can only be created by extruding existing plate elements.

Create Solids

Solid Elements are not drawn in RISA-3D. Instead, they must be extruded from a mesh of plate elements. Use the Insert menus or the Drawing Toolbar to create new solids. Once you have created these items, you can use other graphic features to load the model and set boundary conditions.

Creating solid models requires more forethought than either beam or plate elements. To create solids you must first create a plate element mesh, and then extrude that mesh into a series of solid elements. See [Plate Modeling Tips](#) and [Plate Modeling Examples](#) for tips on building plate element meshes. You can set all the element properties up front or you can modify these properties after you draw them. [Modifying Solids](#) is discussed in the next section.

The “Create Solids by Extruding Plates” window (shown below) lets you take any existing plate and extrude it out into a three dimensional object.

3D Create Solids by Extruding Plates ? X

Solid Material

Material

Extrusion Options

Extrude Axes

X ft

Y ft

Z ft

No. of Solid Elements - +

Extrude all selected plates

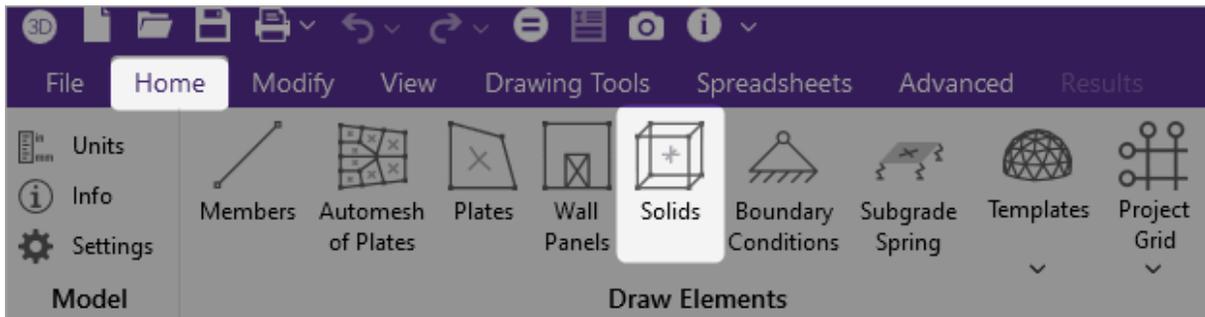
Extrude by clicking plates individually

OK **Cancel**

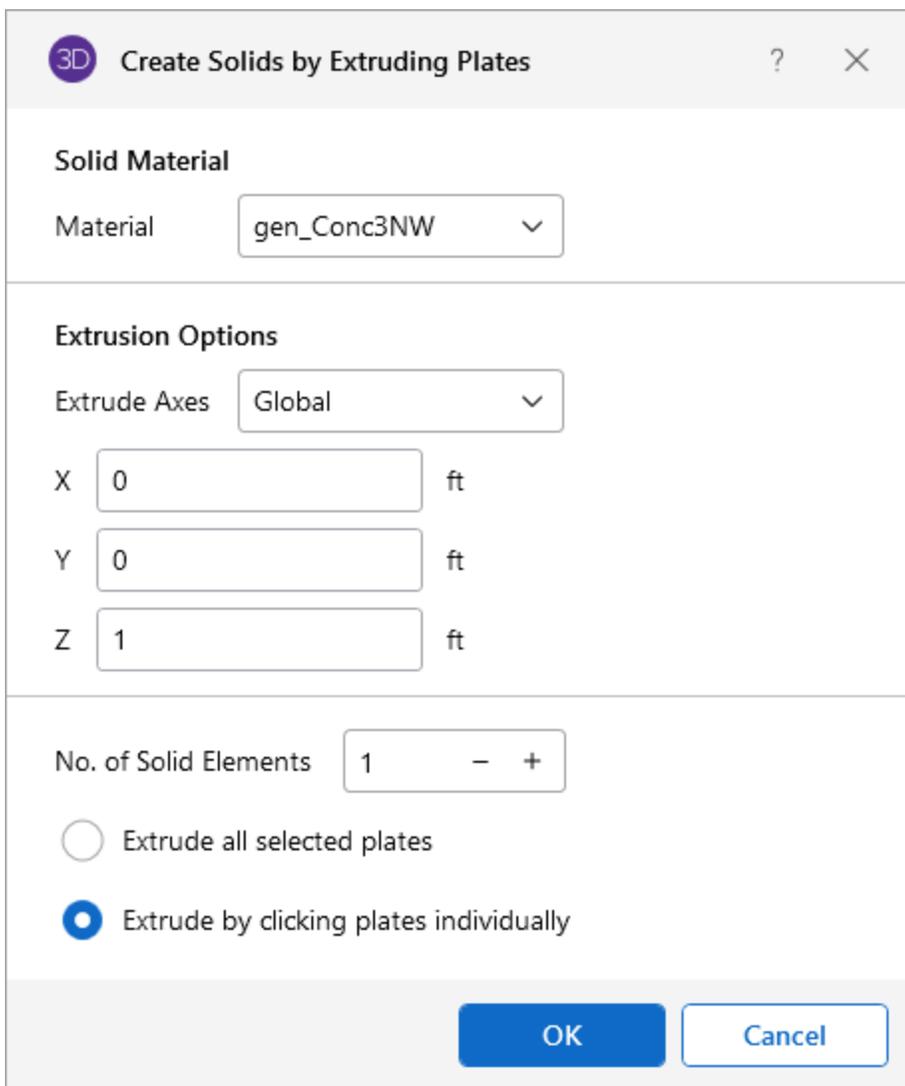
Extrude Solids

To extrude solid elements:

1. If there is not a model view already open, then click the **Open 3D Views** icon in the 'View' ribbon.
2. If you do not already have a plate element mesh, then you must create one.
See [Drawing Plates](#) for more information on this.
3. Click the **Solids** icon (as shown in the following image).and set the solid properties.



The Create Solids by Extruding Plates window opens.



The “Create Solids by Extruding Plates” window lets you take any existing plate and extrude it out into a three dimensional object.

4. Click the **Material** arrow and choose the solid material you want to extrude.

5. In the Extrusion Options section enter the extruding options:
 - **Extrude Axes** either globally or locally.
The **X**, **Y**, and **Z** components of the extruding vector and the thickness of the extrusion vector.
 - The **No. of Solid Elements** (number solid elements) along the extrusion vector.
6. Choose one of the following options:
 - **Extrude all selected plates** - Choose this option to extrude an entire selection of plates at once.
Select the plates in the 3D View panel first, then choose this option.
All plates currently selected in the 3D View panel will be extruded. If you don't have any plates selected, ALL plates of the same material in the model are extruded by default.
 - **Extrude by clicking plates individually** - Choose this option to extrude a single solid at a time or to select just a few plates to extrude.
Choose this option first, then click on the individual plate(s) you want to extrude.
7. Click **OK** to extrude the plates.

**Note:**

- Only Quadrilateral plates can be extruded.
- To extrude more solids with different properties, press CTRL-D to recall the Create Solids by Extruding Plates window.
- You can also view and edit solid properties by clicking on a solid.
- You can undo any mistakes by clicking the Undo  button.

Modify Solids

You can modify one or more solid elements at once using the Properties panel. The Properties panel lets you modify the properties of solids that already exist in your model. You can modify solids one at a time by selecting a single solid and modifying the properties, or you can modify entire selections of solids by selecting the solids first and then modifying the parameters for the entire set.

To modify solid elements using the Properties panel:

1. Click on one or more solids to view to display the properties in the Properties panel.

Properties	
Solid ^	
Material	gen_Conc3NW v ...
Solid Label	S1
Additional Properties ^	
A Node	N1
B Node	N2
C Node	N3
D Node	N4
E Node	N5
F Node	N6
G Node	N7
H Node	N8
Activation	Active v

2. In the **Properties** panel, edit the properties you want to modify.
your changes are immediately applied.

Sub-Meshing Solids

You can submesh solid plate elements into a mesh of smaller elements. This new mesh can be any size up to the program limits for joints and/or solids. This is very useful for refining a coarse mesh of elements; just make sure that all adjacent solid elements (elements sharing an edge) maintain connectivity.

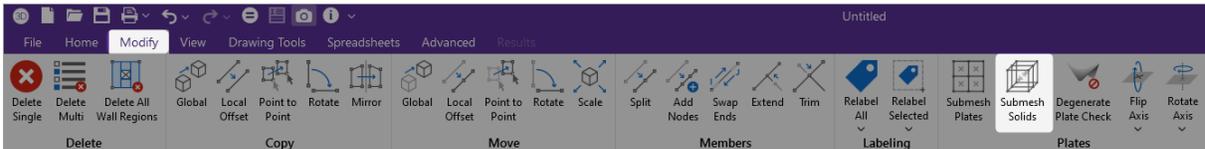
You can define different submesh increments in each direction.

You can submesh the solids one at a time by selecting the **Click to Apply** option and then clicking on the solids you wish to submesh. You may also modify entire selections of solids by selecting the plates and then using the **Apply to Selected** option.

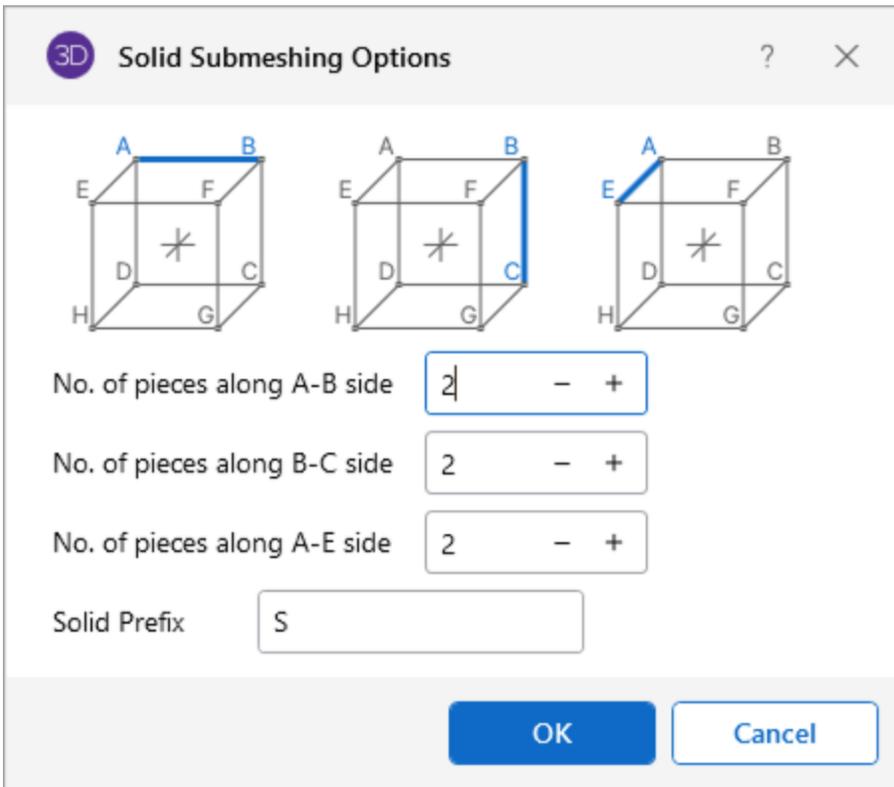
Submesh Solid Elements

To submesh solid elements:

1. If there is not a model view already open, click the **Open 3D Views** icon in the View ribbon.
2. Select the Solids you want to sub mesh.
3. Click the **Submesh Solids** icon in the 'Modify' ribbon (as shown below).



The Solid Submeshing Options window opens.



This window allows you to define different submesh increments in each direction.

4. Specify the number of pieces along the side.
5. Click **OK** to apply your options.

**Note:**

- You can submesh solids one at a time by specifying the number of pieces, clicking **OK** and then clicking on the solids you wish to submesh. Or, you can modify entire selections of solids by selecting the plates, opening the Solid Submeshing Options window, specifying the number or pieces, and then clicking **OK**.
- To submesh more solids with different parameters, press CTRL-D to recall the **Submesh Solids** settings.
- You may undo any mistakes by clicking the **Undo**  button.

Solids Spreadsheet

The **Solids** Spreadsheet records the properties for the solid elements of the model and may be accessed by selecting **Solids** on the **Data Entry** menu.

	Label	A Node	B Node	C Node	D Node	E Node	F Node	G Node	H Node	Material	Activation
1	S1	N1	N2	N3	N4	N5	N6	N7	N8	gen_Conc3NW	
2	S2	N9	N10	N11	N12	N13	N14	N15	N16	gen_Conc3NW	
3	S3	N17	N18	N19	N20	N21	N22	N23	N24	gen_Conc3NW	
4	S4	N25	N26	N27	N28	N29	N30	N31	N32	gen_Conc3NW	

The following data columns hold the 'Primary' data for the solids.

Solids Primary Data Columns

Column	Description
Label	Display the unique name of each solid. You can assign a unique label to any or all of the solids. You can then refer to the solid by its label. Each label has to be unique. If you try to enter the same label more than once, you receive an error message. You can relabel solids at any time using the Relabel All or Relabel Selected icons on the Modify ribbon.
Solid Nodes (A - H)	The A, B, C, D, E, F, G, and H node entries are used to define the 8 corner nodes of the solid element. These nodes are unable to be edited in the spreadsheet, since solids are created by extruding existing plates.
Material	The material label links the solid with the desired material defined on the Materials Spreadsheet .

Column	Description
	 Note: Solids are always defined with general materials. This is because the other material sets (Hot Rolled, Cold Formed, Wood, and Concrete) are used to designate member code checking specifications. Since solids are only used for analysis, no code checking is provided and the material must be designated as a general material.
Activation	The Activation data column allows for solids to be set to 'Active', 'Inactive', or 'Excluded'. These choices can be made by selecting them from the drop down list or by entering an 'I' for 'Inactive' or an 'E' for 'Excluded'. If the cell is left blank the plate is 'Active' by default. See Inactive and Excluded Solids for more information.

Solid Information

Just as with the nodes, members, and plates you can click any solid to view it's properties. All of the same information that is stored in the **Solids** spreadsheet is displayed for the solid you choose, and may be edited. This is a quick way to view and change solid properties.

Properties

Solid ^

Material gen_Conc3NW v ...

Solid Label S1

Additional Properties ^

A Node N1

B Node N2

C Node N3

D Node N4

E Node N5

F Node N6

G Node N7

H Node N8

Activation Active v

The following table describes the information provided in the Properties panel.

Solid Properties Panel

Property	Description
Material	The material set label links the solid with the desired material defined on the Material spreadsheet.
Solid Label	<p>You must assign a unique label to all of the solids. You can then refer to the solid by its label. Each label has to be unique, so if you try to enter the same label more than once you will get an error message. You can relabel solids at any time by clicking the Relabel All icon on the Modify ribbon and selecting Solids.</p> 
Additional Properties (Corner Nodes)	The A, B, C, D, E, F, G, and H node entries, under the Additional Properties section, are used to define the 8 corner nodes of a hexahedral element.
Activation	<p>Activation</p> <p>The activation state of the element may be changed. If the solid is made inactive, you will need to activate the solid from the solids spreadsheet, or by using the Criteria Select feature to find and select inactive solids.</p> <div data-bbox="492 1129 1409 1459" style="border: 1px solid black; padding: 10px; background-color: #e6f2ff;"> <p> Note:</p> <ul style="list-style-type: none"> • If you have a plate that has a side of a length that is less than the merge tolerance that you specify in the Model Settings under the Solutions tab, then we will not extrude a solid element from that plate. • If you come across this as being a problem, then your plates are probably not well meshed and you should think about cleaning up your mesh. </div>

Inactive and Excluded Solids

Making an item such as a member or solid inactive allows you to analyze the structure without the item, without having to delete the information that defines it. This leaves data intact so the item may be easily reactivated. This is handy if you want to try a model with and then without certain items, without having to actually delete the data.

- Choosing "Inactive" makes the item inactive, i.e. the item is not included when the model is solved or plotted.

- Choosing "Exclude" will exclude it from the results list. So, an item set as "Exclude" will be treated like any other solid in the solution and plotting of the model, but the solid will not be listed in the solution results (forces, stresses etc.). This is useful if there are certain items whose results you're not interested in. You don't have to clutter up the results with these items and can concentrate on the items you're most interested in.

See [Printing](#) for more limiting printed results.

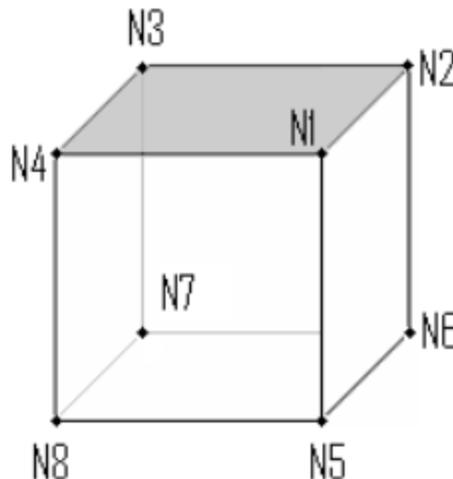
Solids Formulation

A reference for this element is Finite Element Procedures, by K.J. Bathe, Prentice-Hall, 1996. Although the book does not complete the element derivations, it does provide many references for papers on the family of elements. In brief, the element formulation is standard 8-node ISO-parametric formulation.

Node Connectivity

The node connectivity in our current solid formulation follows the "left hand rule". It means if the first 4 nodes are ordered *counterclockwise* in plane (as shown in the picture below), then node 5 has to be below node 1. However, if the first 4 nodes are ordered *clockwise* in plane, then N5 would have to be above node 1.

The node connectivity for solid element is listed in the following picture.



If the solid element is generated using the extrusion tool, the nodes will automatically be generated in the proper order. However, when you are modifying the location the nodes, it is important to keep the order the nodes to be consistent with the rules stated above. If not, then the local element matrix will be singular and an error message will be produced at solution time.

Degrees of Freedom

The Solid element activates the THREE translational degrees of freedom at each of its connected nodes. Rotational degrees of freedom are NOT activated. This element contributes stiffness to all of these translational degrees of freedom. If a rotational load or constraint is applied to a node that is only connected to the solid elements, it will be ignored. Modeling these types of rotations would be similar to modeling the "Drilling Degree of Freedom" for plates. Refer to the Modeling Tips section of the general reference manual for more information.

Coordinate System

For the time being, no local coordinate system is defined for the solid element. All the input and output, such as material properties, stresses, displacements are all defined in the global coordinate systems.

Solid Modeling Tips

Number of Elements

The standard ISO-parametric formulation needs at least 4 elements through the thickness in order to accurately simulate a bending dominant part (such as a thin beam or thin plate).

Aspect Ratio of Elements

Solid Elements are more sensitive to element distortion than plate elements. For this reason, it is a good idea to keep a solid elements relatively undistorted. The best formulation for a solid is a cube with equal length sides.

**Note:**

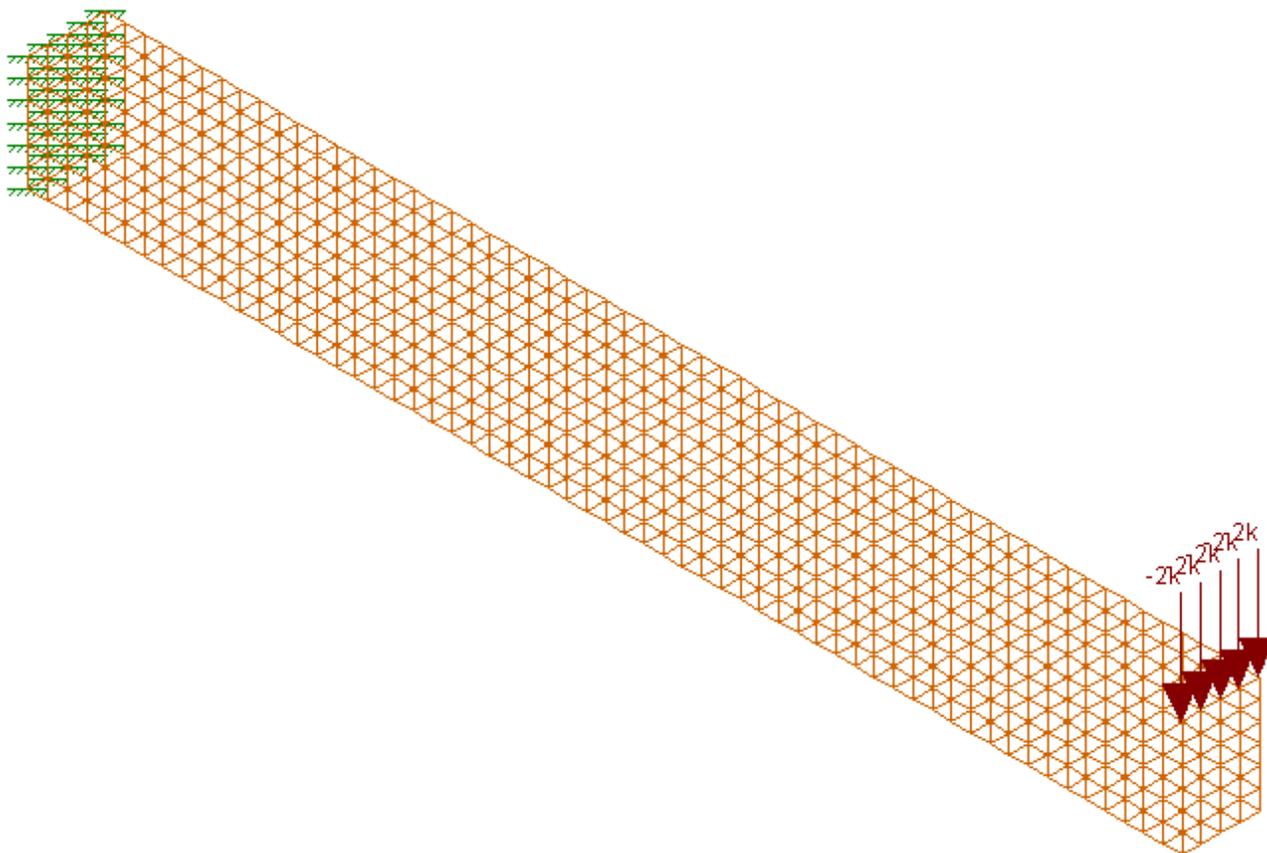
- Solids are always defined with general materials. This is because the other material sets (Hot Rolled, Cold Formed, Wood, and Concrete) are used to designate member code checking specifications. Since solids are only used for analysis, no code checking is provided and the material must be designated as a general material.
- It's generally more efficient to use the Graphic Editing features if you want to change the properties for many solids at once.

Loading

For the time being, only joint loads and self weight can be applied to the solid elements.

Verification Examples

Open the **Solid_Cantilever.r3d** from the Examples folder in the RISA directory.



In this example, a straight cantilever beam (modeled with solid elements) is subjected to a unit force at the tip in the three orthogonal direction and the unit moments at the tip about the three orthogonal directions, each in a different load case. The tip displacements are compared with hand calculations as shown below:

$$d_{beam} := 12 \cdot \text{in} \quad b_{beam} := 8 \cdot \text{in} \quad L_{beam} := 120 \cdot \text{in}$$

$$E_{beam} := 3644 \cdot \text{ksi} \quad A_{beam} := d_{beam} \cdot b_{beam} = 96 \text{ in}^2$$

$$I_{strong} := \frac{b_{beam} \cdot d_{beam}^3}{12} = 1152 \text{ in}^4 \quad I_{weak} := \frac{d_{beam} \cdot b_{beam}^3}{12} = 512 \text{ in}^4$$

$$P_{axial} := 175 \cdot \text{kip} \quad P_{strong} := 10 \cdot \text{kip} \quad P_{weak} := 14 \cdot \text{kip}$$

$$\Delta_{Axial} := \frac{P_{axial} \cdot L_{beam}}{A_{beam} \cdot E_{beam}} = 0.060030 \text{ in}$$

$$\Delta_{strongbend} := \frac{P_{strong} \cdot L_{beam}^3}{3 \cdot E_{beam} \cdot I_{strong}} = 1.372 \text{ in}$$

$$\Delta_{weakbend} := \frac{P_{weak} \cdot L_{beam}^3}{3 \cdot E_{beam} \cdot I_{weak}} = 4.322 \text{ in}$$

The values shown in the table for the RISA output is an average of the 35 nodes at the tip of the cantilever.

Case	Equation	Theory	RISA	% Difference
Axial Extension	Delta = PL/AE	0.060"	0.060"	0
Strong Axis Bending	Delta = PL ³ /3EI	1.372"	1.361"	0.80
Weak Axis Bending*	Delta = PL ³ /3EI	4.322"	4.203"	2.75

From this table we can see that solid element deflections are in very good agreement with the theoretical values.

Solid Elements - Results

When the model is solved, there are two types of results specifically for solid elements: Solid Stresses and Solid Principal Stresses.

Solid (Global) Stress Results

Access the **Solid Stress** results by selecting **Solid Stresses** from the **Results** section of the Explorer panel.

	LC	Solid Label	SigmaXX [ksi]	SigmaYY [ksi]	SigmaZZ [ksi]	SigmaXY [ksi]	SigmaYZ [ksi]	SigmaXZ [ksi]
1	1	S1	0	-0.001	0.001	0	-0.002	0
2	1	S2	0	-0.001	0.001	0	-0.002	0
3	1	S3	0	-0.001	0.002	0	-0.002	0
4	1	S4	0	-0.001	0.002	0	-0.002	0

The solid element stresses in the global directions are evaluated at the standard 2 x 2 x 2 Gauss integration points of the element and extrapolated/interpolated to the joints and the center of the element for graphical display of the stress contours. The **Solid Stresses** spreadsheet, however, only displays the center stress value for each element. These stresses are always reported with respect to the global X, Y and Z axes of the model.

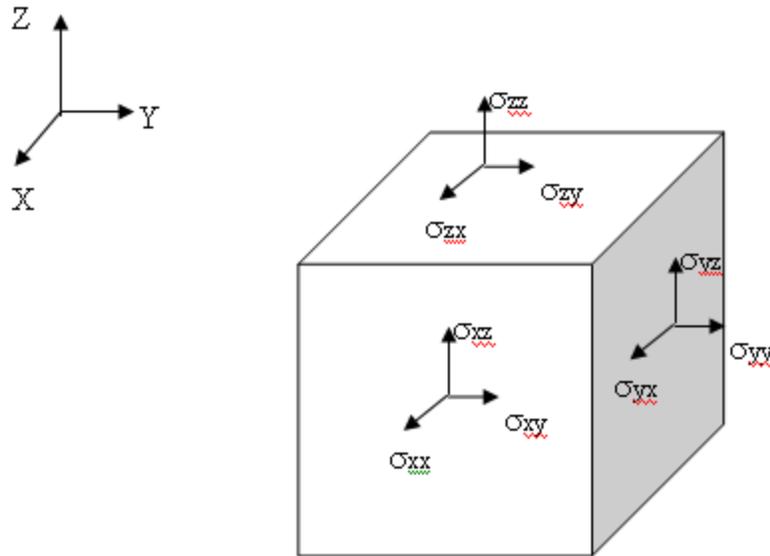
For enveloped results the maximum and minimum value at each location is listed. The load combination producing the maximum or minimum is also listed, in the "LC" column.

Sign Convention for Stresses

For solid element stresses, RISA uses a sign convention where tension is represented by a positive value of Sigma XX, Sigma YY, or Sigma ZZ.

Sign convention for the other stresses is as shown in the image below.

- The first subscript represents the plane on which the stress is acting. In the case of Sigma YZ, the plane is perpendicular to the Global Y plane.
- The second subscript represents the direction of the stress. In the case of SigmaYZ, a positive stress has an orientation in the positive Z direction.
- Since RISA's implementation uses only isotropic material, Sigma YZ = Sigma ZY, Sigma XY = Sigma YX, and Sigma XZ = Sigma ZX.



Note:

- See [Spreadsheet Operations](#) to learn how to use **Find**, **Sort** as well as other useful spreadsheet options.
- See [Results View Settings](#) to learn how to plot solids results.

Solid Principal Stress Results

Principal values and their associated principal directions are also computed. Access the **Solid Principal Stress** results by selecting **Solid Principals** from the **Results** panel.

Solid Principal Stresses (By Combination)															
	LC	Solid Label	Sigma1 [ksi]	Sigma2 [ksi]	Sigma3 [ksi]	Von Mises [ksi]	Angle1X	Angle1Y	Angle1Z	Angle2X	Angle2Y	Angle2Z	Angle3X	Angle3Y	Angle3Z
1	1	S1	0.003	0	-0.002	0.004	1.082	1.082	2.415	0.821	0.821	1.841	0.855	0.855	1.188
2	1	S2	0.003	0	-0.002	0.004	1.069	1.069	2.393	0.816	0.816	1.822	0.865	0.865	1.162
3	1	S3	0.003	0	-0.002	0.005	1.112	1.112	2.465	0.825	0.825	1.855	0.842	0.842	1.229
4	1	S4	0.003	0	-0.002	0.005	1.115	1.115	2.47	0.821	0.821	1.839	0.843	0.843	1.225

Principal Stresses

Sigma1, Sigma2, and Sigma3 represent the principal stresses for the element. The principal stresses are the three eigenvalues of the 3 by 3 stress matrix.

$$\begin{bmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{bmatrix}$$

Orientation of Principal Stresses

The direction of the Sigma1 principal stress is defined by Angle1X, Angle1Y, and Angle1Z. Where each of the angles defines the angle between that global axis and the principal stress direction. The same definitions hold true for the Sigma2 and Sigma3 stresses.

Von Mises Stress

The Von Mises stress is a combination of the principal stresses and represents the maximum energy of distortion within the element. This stress can be compared to the tensile yield stress of ductile materials for design purposes. For example, if a steel plate has a tensile yield stress of 36 ksi, then a Von Mises stress of 36 ksi or higher would indicate yielding of the material at some point in the plate.

$$\sigma_v = \sqrt{\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_1 - \sigma_3)^2}{2}}$$

For enveloped results the maximum and minimum value at each location is listed. The load combination producing the maximum or minimum is also listed, in the "LC" column.

**Note:**

- See [Spreadsheet Operations](#) to learn how to use **Find**, **Sort** as well as other useful spreadsheet options.
- See [Results View Settings](#) to learn how to plot solids results.

Solid Corner Force Results

Access the **Solid Corner Forces Spreadsheet** by selecting the **Results Tab** and then selecting **Envelope** (or **LC**) button ▶ **Solid Corner Forces**. Alternatively, on the **Results** toolbar you can select **Solid Corner Forces Spreadsheet**.

	LC	Solid Label	Node	X [k]	Y [k]	Z [k]
1	1	S1	N1	0.295	4.577	-0.53
2			N2	-0.436	4.31	-4.905
3			N3	-0.156	5.304	-4.449
4			N4	0.301	5.052	0.113
5			N5	0	2.584	0
6			N6	0	0	0
7			N7	-0.004	-0.077	-1.105
8			N8	0	0	0
9	1	S2	N9	0.237	4.536	-0.602
10			N10	-0.359	4.272	-4.836
11			N11	-0.116	5.343	-4.04
12			N12	0.239	5.131	0.208
13			N13	0	2.543	0
14			N14	0	0	0
15			N15	0	-0.074	-1.605
16			N16	0	0	0
17	1	S3	N17	0.243	4.561	-0.54
18			N18	-0.491	4.307	-4.902
19			N19	-0.108	5.412	-5.577
20			N20	0.349	4.963	-1.042
21			N21	0	2.595	0
22			N22	0	0	0
23			N23	0.011	-0.04	0.02
24			N24	-0.004	-0.048	1.165

The solid corner forces are the global forces at the corner of each plate and are listed for each active solid.

These are the forces calculated at the corners of the solids, in the GLOBAL directions. These values are obtained by multiplying the solid'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 solid at the corner in the given direction, in the same way that node reactions are reported. At any given node, the corner forces for all solids connected to that node should sum to zero (a requirement of equilibrium), assuming no members or boundary conditions are also present at the node.

For enveloped results, the maximum and minimum value is listed. The load combination producing the maximum or minimum is also listed, in the "LC" column.

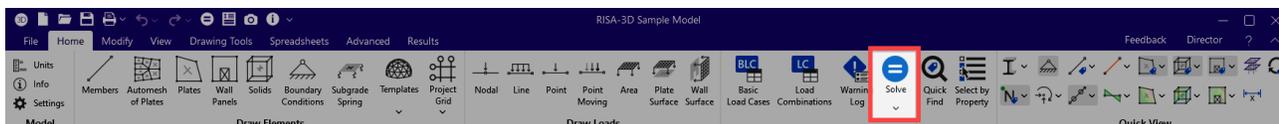


Note: See [Spreadsheet Operations](#) to learn how to use **Find, Sort, Label Marked Lines in Current View** **Show Selected Lines in Current View** and other options.

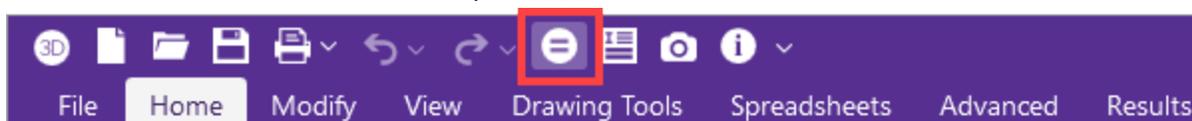
Solution

To solve the model, click the **Solve** icon on the Home ribbon or on the Quick Access toolbar. A progress bar displays information as the solution proceeds.

Home ribbon

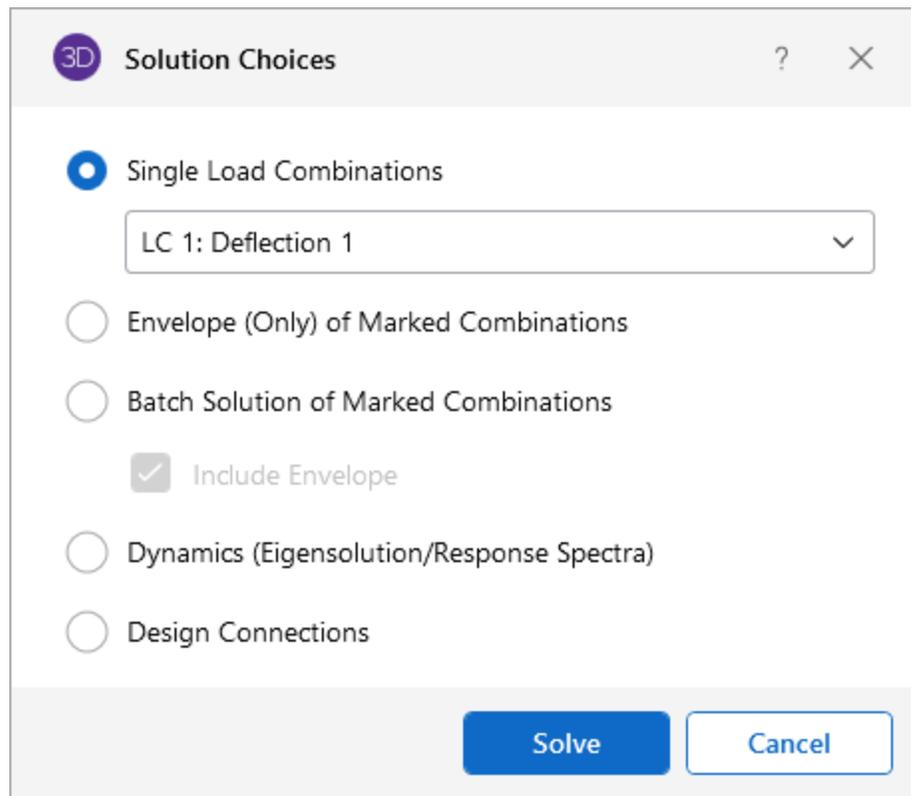


Quick Access toolbar



Note: The linear-elastic behavior of the program implicitly means that non-linear (2nd order) behavior is not generally supported. Tension-structures such as guy cables or fabric structures would fall into this non-linear category. Please see the [Modeling Tips](#) section for some ways to approximate this behavior. This note applies to all elements in the program (wall panels, plates, members, etc.).

Four solution options are presented when you click **Solve** on the menu. Some of the options require other solution types to be performed. See [Results](#) for information on solution results. For additional information on [Dynamic Analysis](#) and [Response Spectra Analysis](#) results refer directly to those sections. For connection design within RISA-3D see the [RISAConnection Integration](#) topic.

**What do you want to do?**

- [Perform a dynamic analysis.](#)
- [Perform a response spectra analysis.](#)
- [Perform a Time History solution.](#)
- [Perform a Static Solution](#)

What do you want to know?

- [What is an Envelope Solution?](#)
- [What is an Batch Solution?](#)

Differences Between RISA-3D and RISAFloor

RISAFloor performs an independent solution at each floor. The assumptions in the Floor solution are essentially the following:

- The vertical stiffness of walls are infinite and are not affected by the stiffness of the members supporting them.
- Vertical and rotational stiffness of columns are approximated based on the floor height and properties (EEI/L and AE/L).

- Lower floors will not affect the solution or loading of floors that are located above them.
- RISA-3D results do not currently account for Live Load Reduction coefficients in the analysis or design unless the model is linked to RISAFloor.

These assumptions are not always 100% accurate. Therefore, the RISA-3D solution can differ in cases where these assumptions are less than perfect.

Static Solutions

Static solutions are based on load combinations and may be performed on any defined load combinations. When a static solution has been performed and the results are available the  icon on the status bar in the lower left corner will change from  to . If any changes are made to the model that invalidate the results then the results will be cleared and another solution will be necessary.

The solution is based on the widely accepted linear elastic stiffness method for solution of the model. The stiffness of each element of the structure is calculated independently. These element stiffnesses are then combined to produce the model's overall (global) stiffness matrix. This global matrix is then solved versus the applied loads to calculate joint deflections. These joint deflections are then used to calculate the individual element stresses. The primary reference for the procedures used is **Finite Element Procedures**, by K. J. Bathe (Prentice-Hall, 1996).

Skyline Solver

This solution method is also sometimes referred to as an Active Column solution method. In finite element analysis, the nonzero terms of the stiffness matrix are always clustered around the main diagonal of the stiffness matrix. Therefore, the Skyline or Active Column solutions take advantage of this by condensing the stiffness matrix to exclude any zero stiffness terms that exist beyond the last non-zero term in that column of the matrix.

Since the majority of terms in a stiffness matrix are zero stiffness terms, this method greatly reduces the storage requirements needed to store the full stiffness matrix. However, for large models (+10,000 joints), the memory requirements even for a skyline solution can be problematic.

This solution method has been used successfully in RISA for more than 20 years, and has proven its accuracy continuously during that time.

Sparse Solver

The skyline solver described above is moderately efficient because it only stores and performs operations on the terms within the "skyline" of the stiffness matrix. However, that solution still contains a great number of zero stiffness terms within the skyline of the matrix. A Sparse Solver will reduce the matrix size to an absolute minimum by eliminating the storage of ALL zero stiffness terms.

A sparse solver is the most efficient solution methodology possible because it stores and performs operation only on the non-zero terms of the stiffness matrix. For this reason the sparse solution is preferred from both a solution speed and memory requirement standpoint.

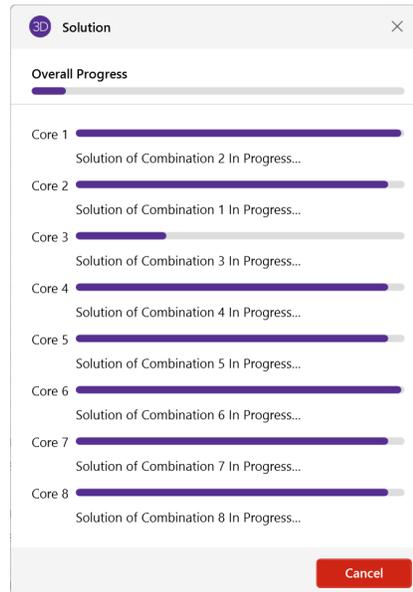


Note: The skyline solver is retained mostly for comparison and verification purposes.

Multi-Threaded Solutions

RISA products now have the ability to use multiple CPU cores if you set the Processor Core Utilization to Multiple (Optimum) or Maximum in [Model Settings](#). The program will split up the solving based on load combination and use each available CPU core to solve load combinations in parallel. Once this portion of the solution is complete the program then combines this data for enveloping and design purposes. This behavior will improve solution time for computers with multiple cores and models with large numbers of load combinations.

During the load combination portion of the solution the solution dialog will show progress for each core being used.



Single Combination Solutions

Choose this option to solve one load combination by itself.

Envelope Solutions

Static solutions may also be performed on multiple combinations and the results enveloped to show only the minimum and maximum results. Each of the results spreadsheets will contain minimum and maximum values for each result and also the corresponding load combination. The member detail report and deflected shape plots are not available for envelope solutions. See [Load Combinations](#) to learn how to mark combinations for an envelope solution.

Batch Solutions

Static solutions may be performed on multiple combinations and the results retained for **each** solution. When performing a batch solution, you have the option to also include a set of envelope results. This is useful when an envelope result is desired to quickly determine a controlling load combination, but when the investigation of that load combination required the greater details given with batch solution results. When using the Batch + Envelope Solution, to view the envelope results, click on a spreadsheet in the Results toolbar. To view the batch results, click on that same spreadsheet in the Results toolbar a second time. Both the envelope and batch solution spreadsheets should now be available simultaneously.

You may group the results by item or by load combination by choosing from the **Results** section on the **Results** tab. For example you can have all the combination results for member M1 together or you can have all the member results for Load Combination "D+L" together. See [Load Combinations](#) to learn how to mark combinations for a batch solution.

Dynamic Solutions

Eigenvalues & Mode Shapes

In this section, the term "Dynamic Solutions" refers to solving of the dynamic properties of a structure. This involves assembling the mass matrix, solving for the eigen values (natural periods / frequencies), as well as determining the mode shapes of vibration associated with each period / frequency. Dynamic analysis requires a load combination, but this combination is merely used to determine the mass of the model. See [Dynamic \(Modal\) Analysis](#) for much more information.

When a dynamic solution has been performed and the results are available the  icon on the status bar in the lower left corner will change from gray to . If any changes are made to the mode that invalidates the results then the results will be cleared and another solution is necessary.

Accelerated Solver

The Accelerated Solver is actually three solvers in one: A Direct Jacobian, an Accelerated Sub-Space solver, and a Lanczos Solver. Each of these solvers will have a certain range of models for which it is most efficient. Therefore, RISA will automatically detect which should be most efficient for the given model and use that solver for the eigen-solution. This solver is much more efficient than the standard solver and will find frequencies and mode shapes in a fraction of the time the standard solver. Therefore, it is the default solution option.

Standard Solver

This solver uses sub-space iteration to solve for the Eigen values. This solver has been used for years and the accuracy of the results is very well established. It has been included mostly for comparative / verification purposes.

Ritz Vector Solver

This solver uses Load Dependent Ritz (LDR) vectors to bias the dynamic solution. This means that the solution does not necessarily represent the true mode shapes or natural frequencies of the structure. As such, this solution should not be used in cases where the natural frequencies and modes shapes are the main goal.

Ritz vectors are derived by using a static displacement vector as the basis of the derived vector, only allowing for the solution of modes / vectors that will be excited by the initial loading or which will contribute to the total response. In RISA, the initial static displacement vectors are based on the load combination used for the definition of [dynamic mass](#). The mass defined in this load combination is converted into a static load in each direction for which a Response Spectra solution is requested. The solution of which forms the basis for the initial Ritz Vectors.



Note:

- For cases where the mode shapes and frequencies are used to estimate dynamic response (i.e. Response Spectra or Time History analysis), it has been demonstrated that using Ritz vectors can produce a more accurate response than would be obtained using the same number of true eigen modes.
- It is possible, however, for there to be some frequency shift in the mass participation. For the case of earthquake response of a typical building structure, this frequency shift should result in a more conservative base shear than would be produced by a true eigen solution producing the same amount of mass participation.
- In particular Ritz Vectors are useful for cases in which it is difficult to obtain the desired level of mass participation with a traditional eigen value analysis. This is because Ritz vectors are biased towards a presumed deflected shape.
- When Ritz Vectors are used for a Time History analysis the LC used as the basis for the dynamic mass should contain TH loading at the main joints of the structure. The program will detect this and then include the Time History Load function in the initial static displacement vector. This doesn't change the mass matrix at all. But, it will bias the Ritz Vectors to displacements at those joints which leads to a more accurate Time History response with fewer modes.

Response Spectra Solutions

Whenever a dynamic / eigen solution has been performed the  icon on the status bar in the lower left corner will change from gray to .

If any changes are made to the model that invalidate these dynamic results (even if they don't invalidate the mass matrix), the dynamic results will be cleared and need to be re-solved before a time history analysis can be performed.

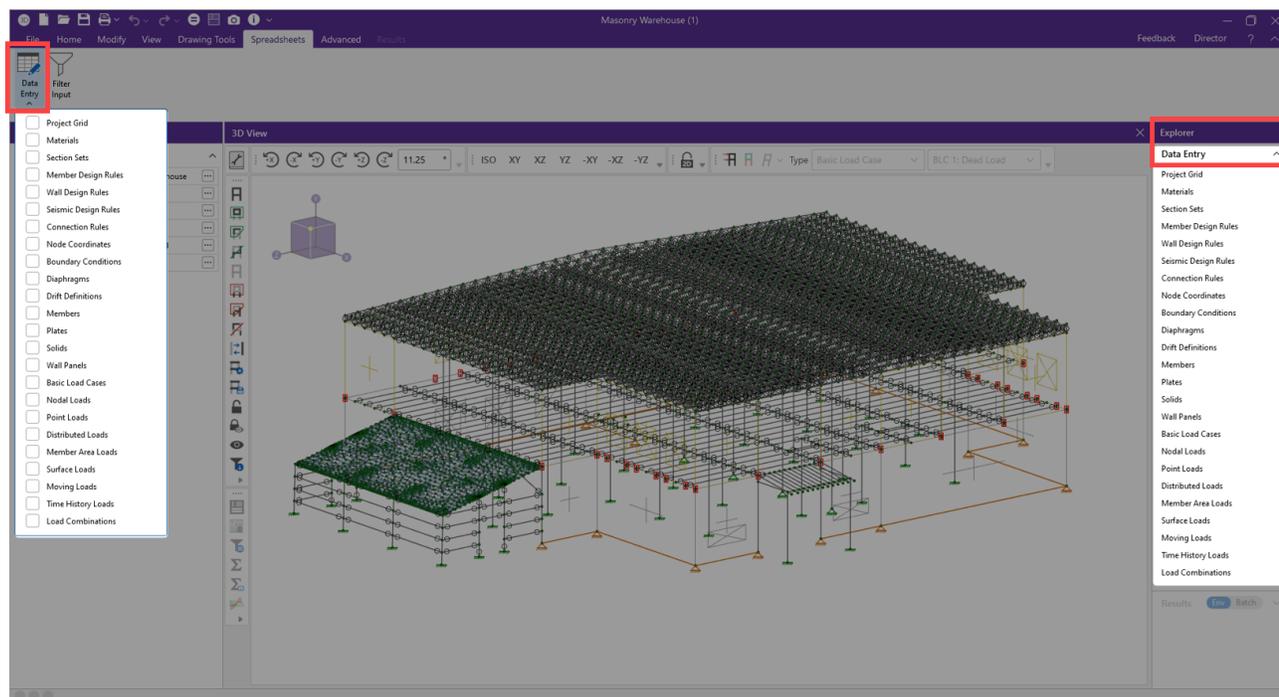
**Note:**

- Alpha will default to zero, at which point the solution is identical to the Newmark Beta method using integration constants $\beta = 0.25$ and $\gamma = 0.5$.
- A negative value for alpha is often used when the desire is to dissipate energy associated with higher frequency response. This makes the HHT solution methodology more stable than other solution methods, and may allow the user to set their integration time step slightly larger to improve time history solution speed. Refer to topic Dynamic Analysis - Time History for more information on setting time steps.
- RISA requires an Eigensolution or Ritz Vector solution even when Direct Integration is used. Refer to [Required Number of Modes](#) for a discussion on this topic.

Spreadsheet Operations

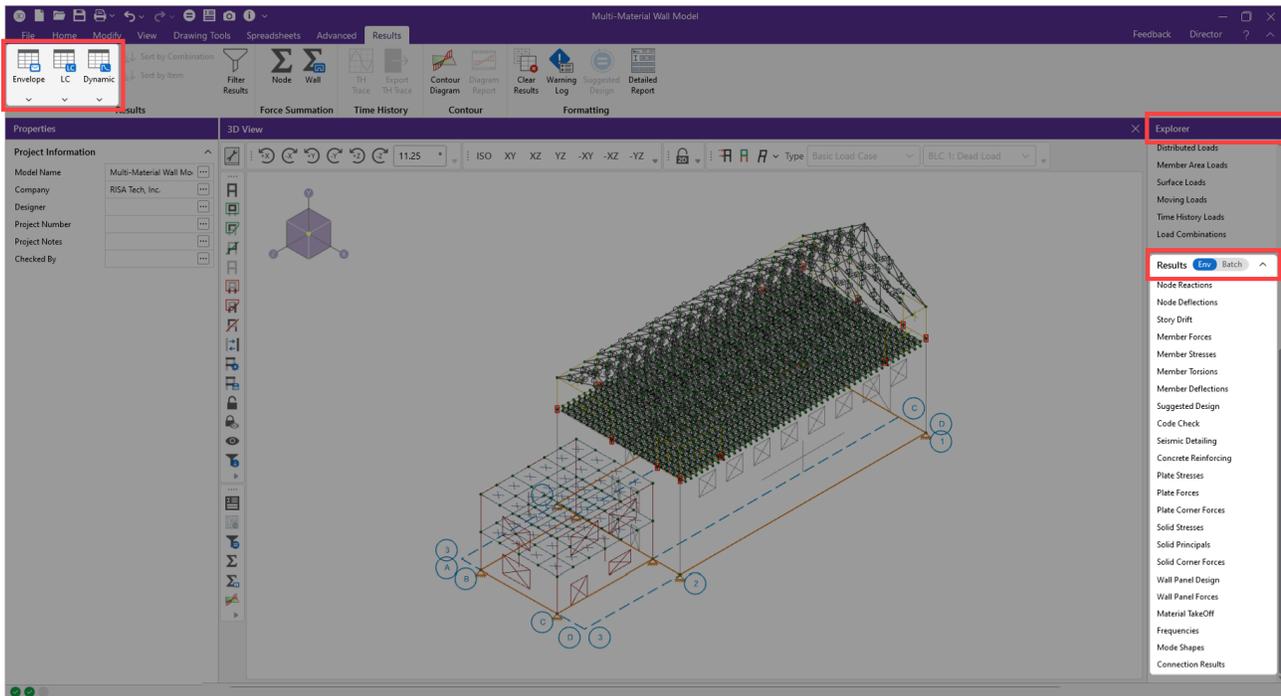
Powerful spreadsheets can be used to view, sort and edit the input data and results. The spreadsheets and model views are synchronized. As you edit a model graphically the spreadsheets are automatically updated. As you make changes in the spreadsheets the model views reflect these changes immediately.

The input data may be accessed from the **Explorer** panel or from the **Data Entry** icon (shown in the following image) on the **Spreadsheets** ribbon.



You can edit the data or you can add new data. You can also paste data from another application directly into the spreadsheet via the Windows clipboard. Any changes made to the input spreadsheets may also be viewed graphically.

After solving the model, results are recorded in spreadsheets for browsing. These spreadsheets can be accessed from the **Explorer** sidebar or the various result spreadsheet icons (**Envelope**, **LC**, and **Dynamic**) found on the **Results** ribbon (as shown in the following image).



You can sort the results in order to find maximums and exclude data that is not important. You can also copy this data to the Windows clipboard and use it in another application.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Advanced Spreadsheet**.

Move and Scroll

To move between cells in a spreadsheet, click any cell or use the keyboard arrow keys. When you move to a cell it becomes the active cell. To see a different area of the spreadsheet use the vertical scroll bars on the right side of the spreadsheet.

To Scroll	Do This
One row up or down	Click the arrows on the vertical scroll bar.
One column left or right	Click the arrows on the horizontal scroll bar.
One page up or down	Click above or below the scroll box in the vertical scroll bar.
One page left or right	Click to the left or right of the scroll box in the horizontal scroll bar.
A large distance	Drag the scroll box to the approximate relative position.

**Note:**

- The size of a scroll box indicates the proportional amount of the spreadsheet that is visible. The position of a scroll box indicates the current location relative to the spreadsheet.
- The mouse wheel is also an excellent tool for scrolling up and down in spreadsheets or graphics. Roll the wheel or click and drag to move around.

Spreadsheet Keyboard Commands

The following keyboard commands are available:

Key	Function
Arrow Keys	Move the active cell one location
TAB	Move right one cell
ENTER	Move to the first column of the next line. Adds new line if necessary.
PAGE UP	Move the active cell one full page up.
PAGE DOWN	Move the active cell one full page down.
HOME	Move to the first line of the spreadsheet.
END	Move to the last line of the spreadsheet.
F3	Insert new line below current line and repeat the current values in the new line.
F4	Delete current line.
F8	Add new row to end using values from row above.

Select Spreadsheet Cells

Before you can carry out commands or tasks in a spreadsheet, you must select the cells that you want to work with.

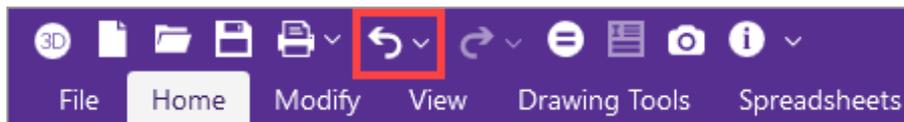
To Select	Do This
A single cell	Click the cell, or press the arrow keys to move to the cell.
A range of cells	Click the first cell of the range, and then drag to the last cell.
An entire row	Click the row heading.
An entire column	Click the column heading.
Adjacent rows or columns	Drag across the row or column headings



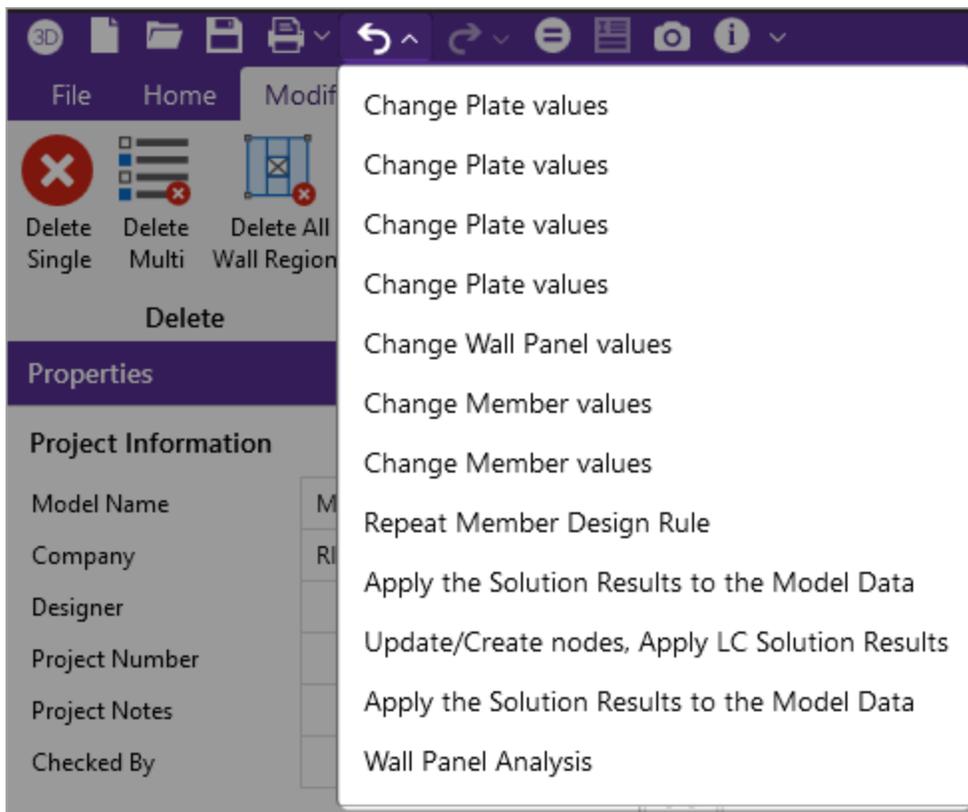
Note: To cancel a selection of cells, click any single cell in the spreadsheet.

Undo Operations

RISA-3D provides you with virtually unlimited 'Undo' capability so that you can easily correct mistakes or just back up to try different possibilities. Simply click the **Undo** button as many times as you wish to go back a step.



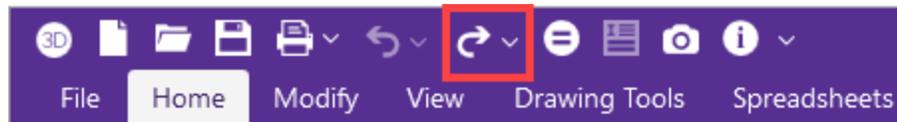
Another option is 'Multiple Undo', accessed by the down arrow next to the **Undo** button. This lets you go back to a specific previous action that you would like to Undo.



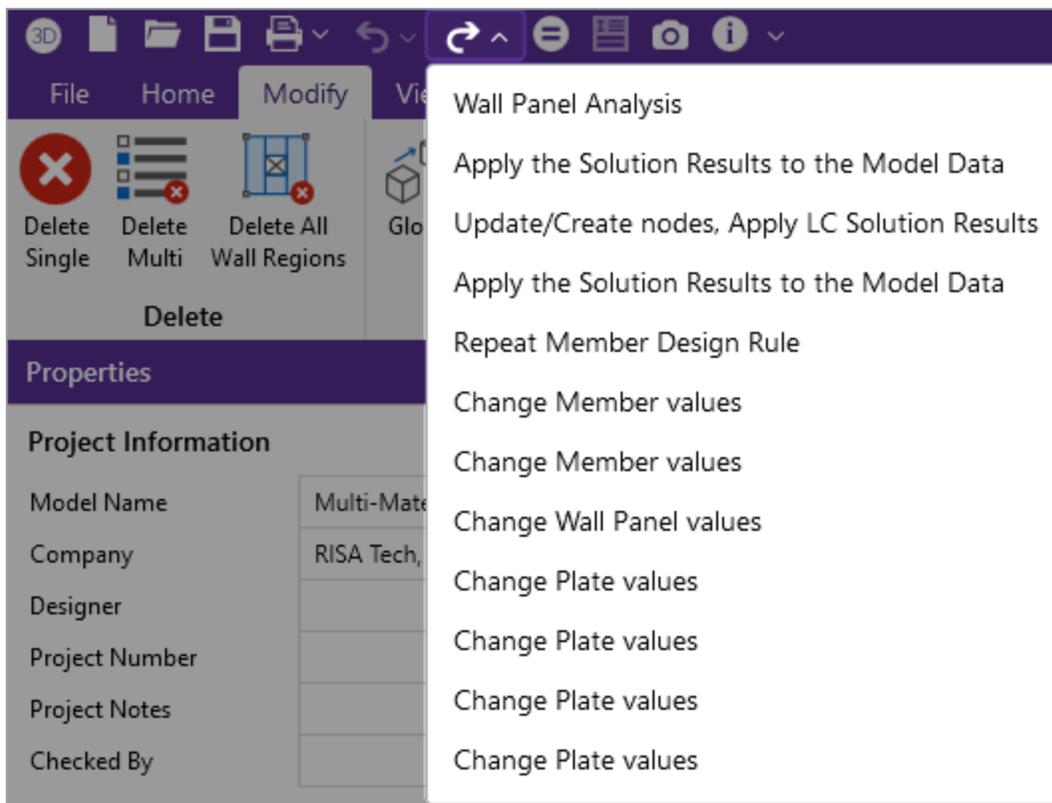
The model view and the spreadsheets visually display the "undoing". Remember that graphic edits are undone as well.

Redo Operations

RISA-3D provides you with virtually unlimited 'Redo' capability so that you may reapply actions that were previously undone. Simply click the **Redo** button as many times as the Undo button was used just prior.



Another option is **Multiple Redo**, accessed by the drop-down arrow next to the Undo button. This lets you go back to a specific previous action that you would like to Redo.



The model view and the spreadsheets visually display the "redoing". Remember that graphic edits are redone as well.

Edit Spreadsheets

The spreadsheets have been specifically developed for the input and editing of structural models. There are many ways to edit the spreadsheets allowing you to quickly build your model. You can copy and move data from other locations or other files. You can also fill large blocks of cells automatically and perform math on these cells.

Change BLC in a Load Spreadsheet

To change Basic Load Cases (BLC) in a Load spreadsheet:

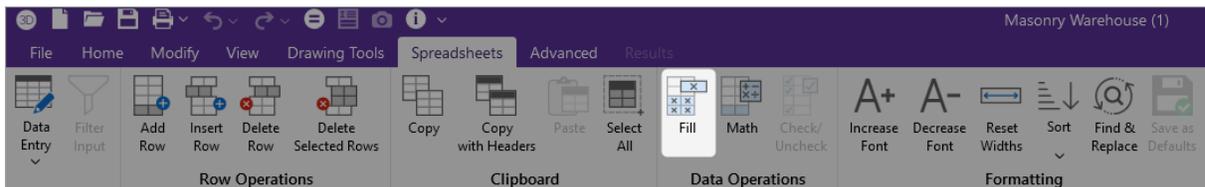
- Use the drop down load list (as shown in the following image) or the adjacent buttons on the window toolbar, to control the basic load case in a load spreadsheet.



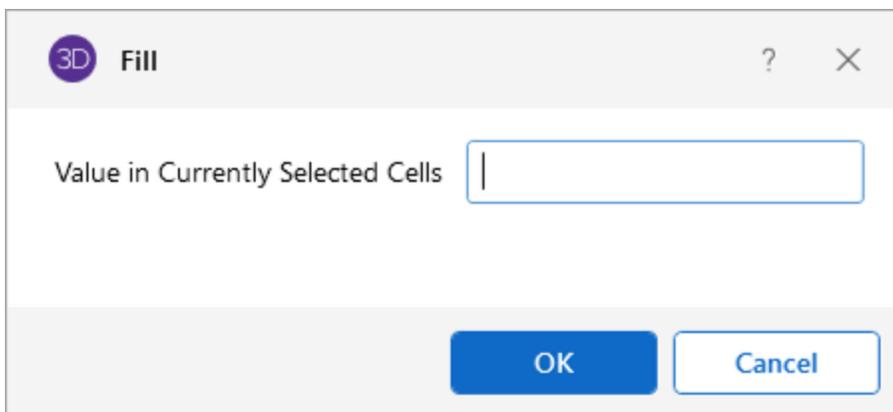
Fill Selected Cells

To fill selected cells:

- Select the cells you wish to fill.
- Click the **Block Fill** icon on the Spreadsheets ribbon.



- Type the value you wish to apply.

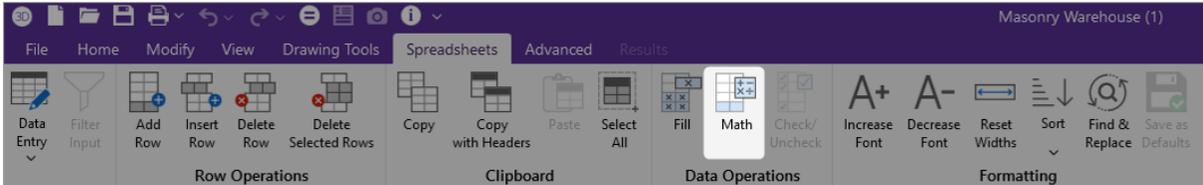


- Click **OK**.

Perform Math on Selected Cells

To perform math on selected cells:

1. Select the desired cells.
2. Click the **Block Math** button.



3. Specify the value and the operation you wish to apply.

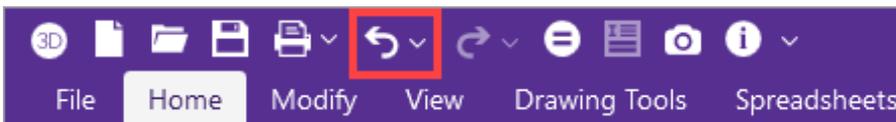
 A screenshot of the 'Math' dialog box. It has a title bar with the RISA-3D logo and the text 'Math'. Below the title bar, there are two input fields: 'Block Math Operand' and 'Function'. The 'Function' dropdown menu is set to 'ADD the operator to the block'. At the bottom right, there is a blue 'Apply' button.

4. Click **OK**.

Cancel or Undo Editing

To cancel or undo editing:

- To cancel an entry before you press ENTER, press **ESC**.
- To undo a completed entry, click the **Undo** button.



Edit Cell Contents

To edit cell contents:

1. Click the cell that contains the data you want to edit.
2. Make any changes to the cell contents.
3. To enter your changes, press **ENTER**.
4. To cancel your changes, press **ESC**.

**Note:**

- To replace the contents of a cell, click on the cell and type the new entry.
- To edit the contents of a cell, double-click on the cell and use the arrow keys to locate the cursor and perform the edit.

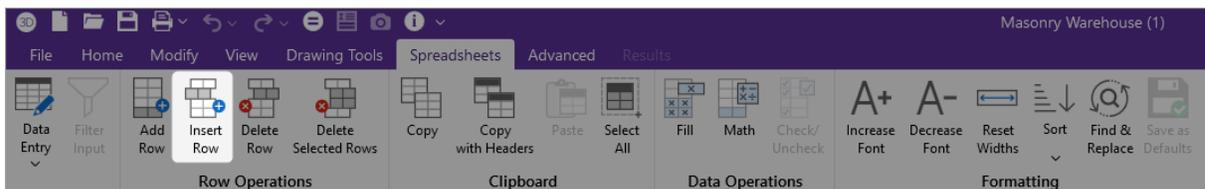
Inserting, Deleting and Clearing Cells

To insert cells you must insert an entire row. When you delete cells you must delete an entire row. The row is removed from the worksheet and shifts the lower cells to fill the space. When you clear cells, you remove the cell contents but leave the blank cells on the spreadsheet.

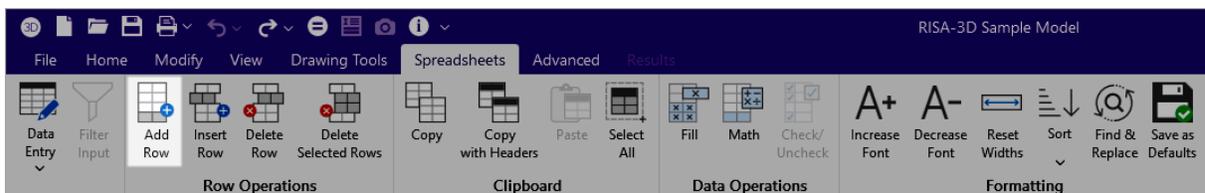
Insert Rows

To insert rows:

1. Click a cell in the row immediately above where you want the new row.
2. To insert a blank line, click the **Insert Row** icon.



3. To have the values of the current line copied in the new line, click the **Add Row** icon.



Clear Cell Contents

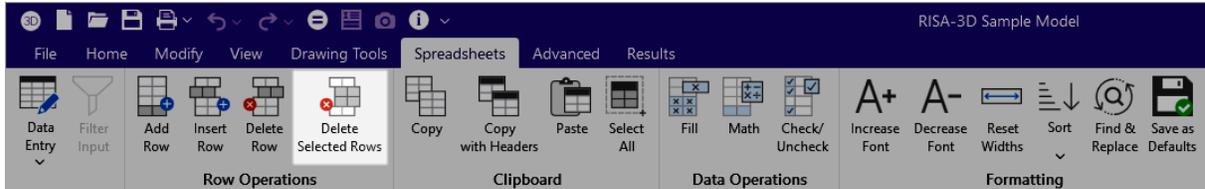
To clear cell contents:

1. Select the cells, rows or columns you want to clear.
2. Press the **DELETE** key.

Delete Rows

To delete rows:

1. Select the rows you want to delete.
2. Click the **Delete Selected Rows** icon .



The lower cells will shift up to fill the space.



Note: Some spreadsheets do not allow you to delete lines. For example the Member Design spreadsheet has one line for each member defined. You may leave these lines blank but can not delete them.

Move and Copy Cell Contents

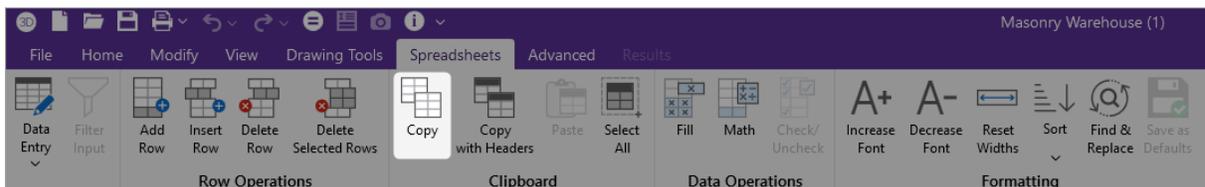
Standard Windows cut, copy and paste functions are fully supported. By using the clipboard you may move or copy any spreadsheet data to another location in the same spreadsheet, or to another spreadsheet (provided it fits). You may also copy data to or from another model or any other application that supports copy and paste such as a word processor or spreadsheet.

Move / Copy Cell Content

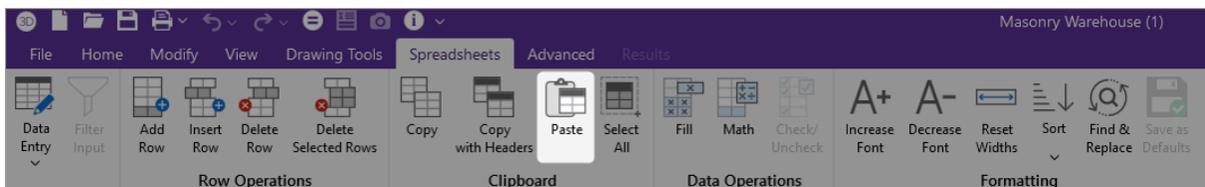
To move or copy cell content:

1. Select the cells you want to move or copy.
2. Do one of the following, depending on what you want to do.

To copy the cells: Click the **Copy** icon on the Spreadsheet ribbon.



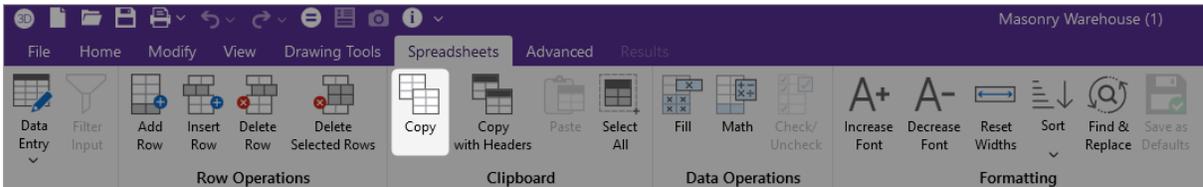
3. Click on the cell you wish to place the data in and click the **Paste** icon.



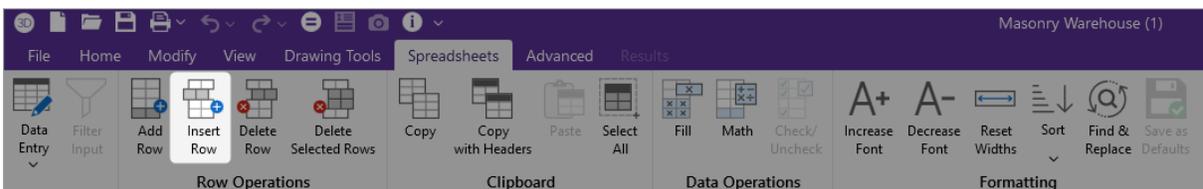
Move / Copy Cells between Existing Cells

To move or copy cells between existing cells:

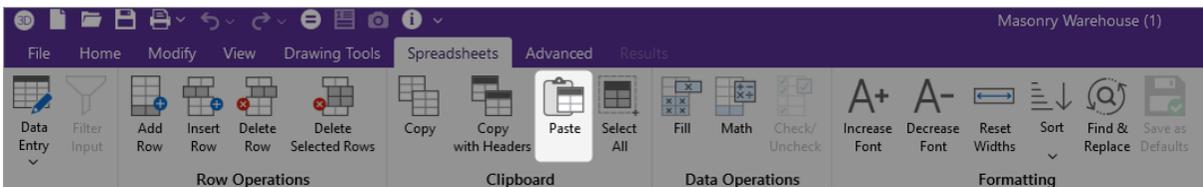
1. Select the cells you want to move or copy.
2. To copy the cells select the **Copy** icon from the Spreadsheet ribbon.



3. Select a cell on the row above where you want to place the data.
4. Click the **Insert Row** icon for each new line that is needed.



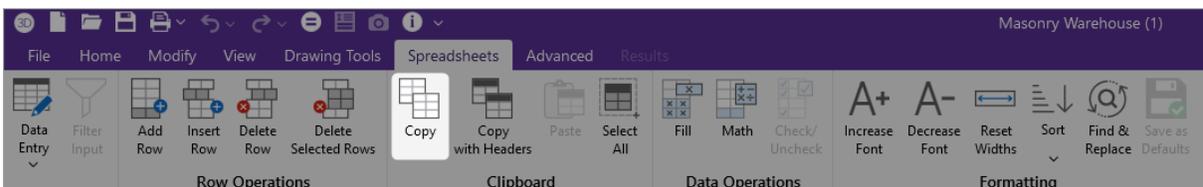
5. Click on the upper left cell you wish to place the data in and click the **Paste** icon.



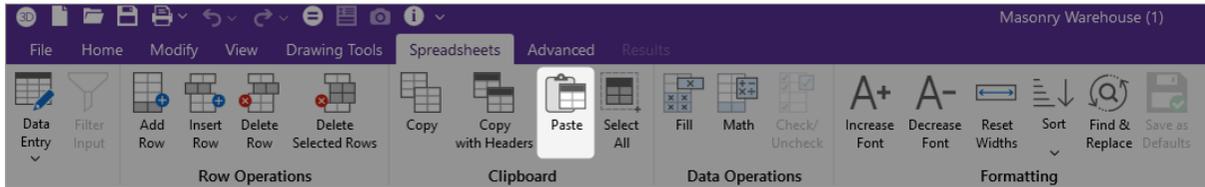
Move / Copy Cells to Another File

To move or copy cells to another file:

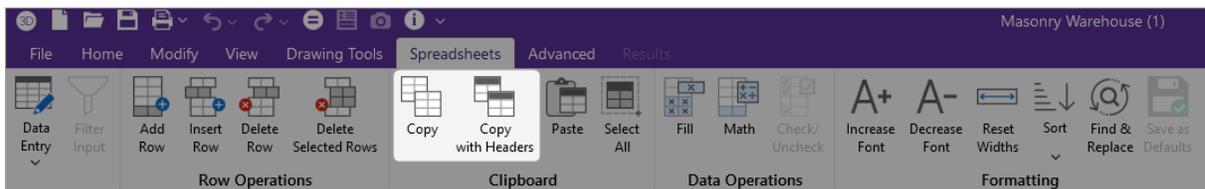
1. Select the cells you want to move or copy.
2. To copy the cells select the **Copy** icon from the Spreadsheet ribbon.



3. Open the file you wish to copy the data to.
4. Click on the cell you wish to place the data in and click the **Paste** button.

**Note:**

- Your data stays in the clipboard until you cut or copy new data. You may repeat step 4 to move or copy data to multiple locations.
- When you copy information, you have the option to **Copy** or **Copy with Headers**. This is helpful if you are copying information to a spreadsheet-type program. See the [Customizing RISA](#) topic for more information on this.



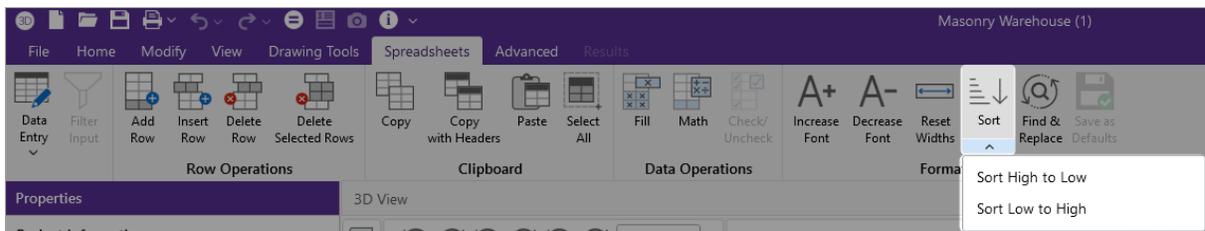
Sorting and Finding in Spreadsheets

Sort a Spreadsheet by a Column Value

You can sort spreadsheets by the values in most any column.

To sort a spreadsheet by a column value:

1. Click in the column you wish to sort.
2. Click on the **Sort** icon



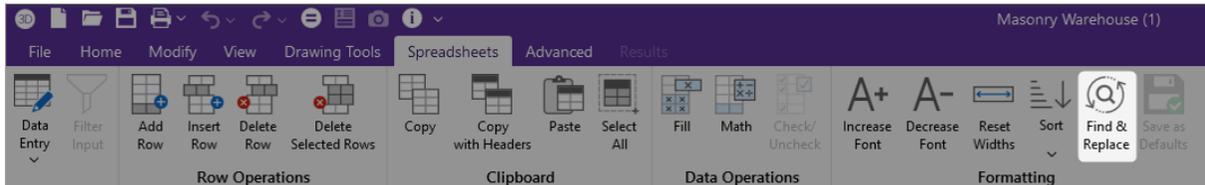
3. Choose the sorting method from the **Sort** menu.

You can relabel the nodes, members, plates, or solids after sorting them by using the options in the **Tools Menu**.

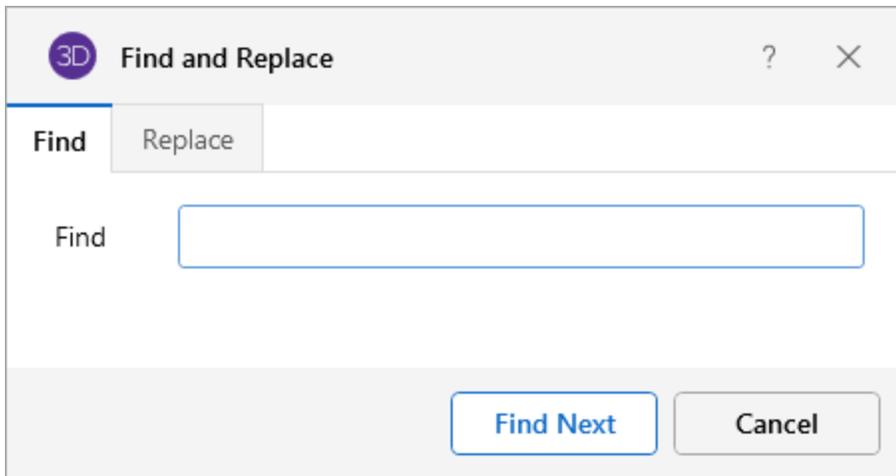
Find a Specific Element in a Spreadsheet

To locate or find a specific element while in a spreadsheet:

1. Click the **Find & Replace** icon on the Spreadsheets ribbon.



A Find and Replace window opens.

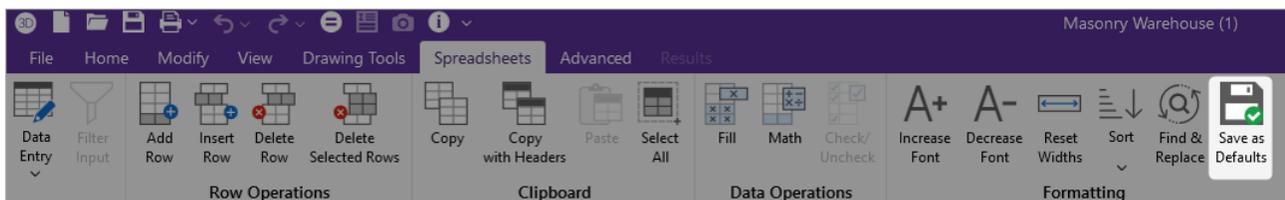


2. Enter the element .
3. Click **Find Next**.

Default Spreadsheet Data

Many of the spreadsheets provide the option to save the current data as the default and every new file subsequent to the save will already have that data. This way the office standards that you might use in most of your models are already entered and available in new models. This feature is available in the following spreadsheets: **Materials, Design Rules, Footings, and Load Combinations**.

To save default data simply click the **Save as Defaults** icon when you are ready and the current data in the spreadsheet will be used in each new file that you create.



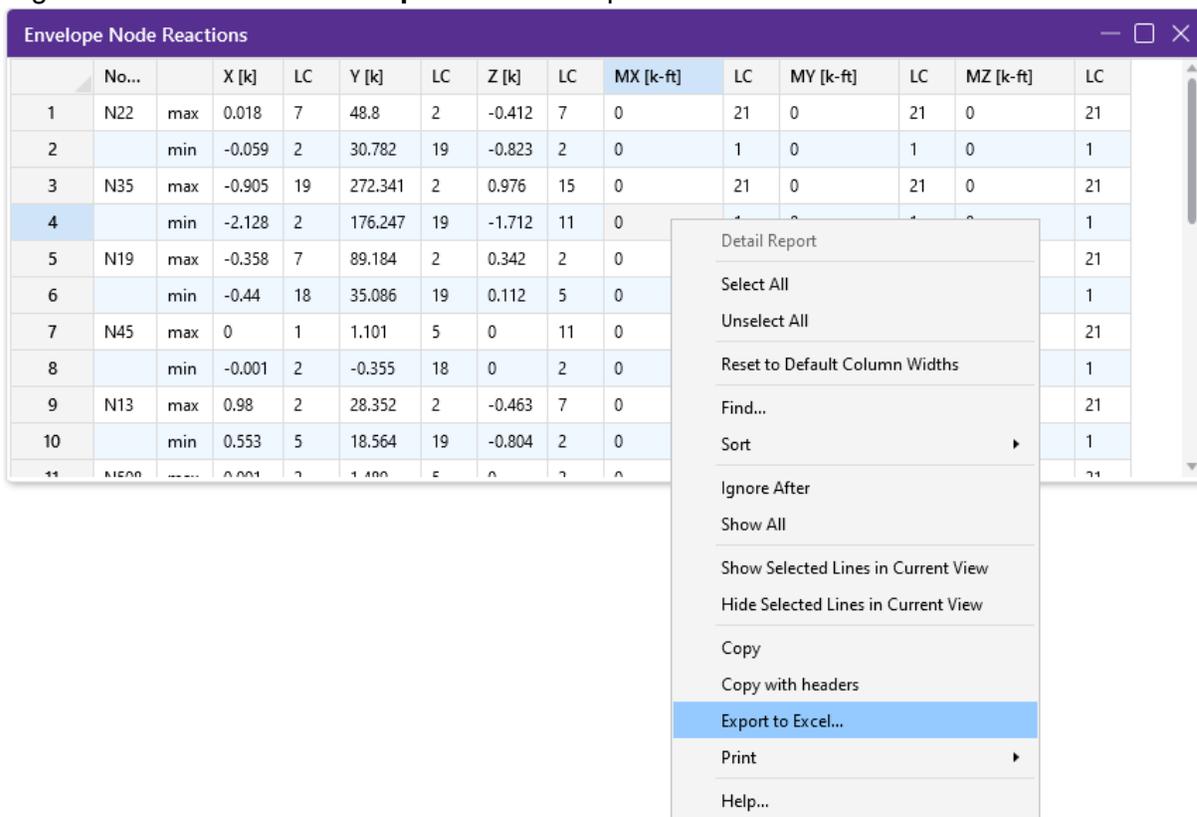
The data is saved for the current, active, spreadsheet only and affects no other open or closed spreadsheets.

Export to Excel Spreadsheet

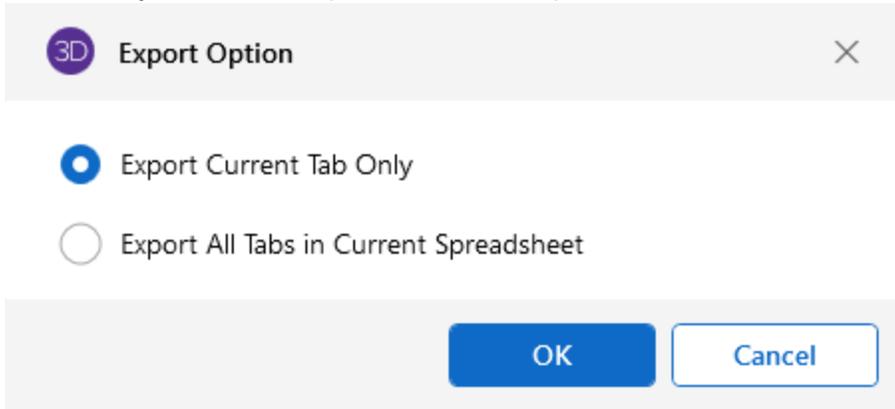
For all Results spreadsheets, the program is capable of exporting them to Excel Spreadsheets.

To export a Results spreadsheet to an Excel spreadsheet:

1. Right Click and choose the **Export to Excel** option.

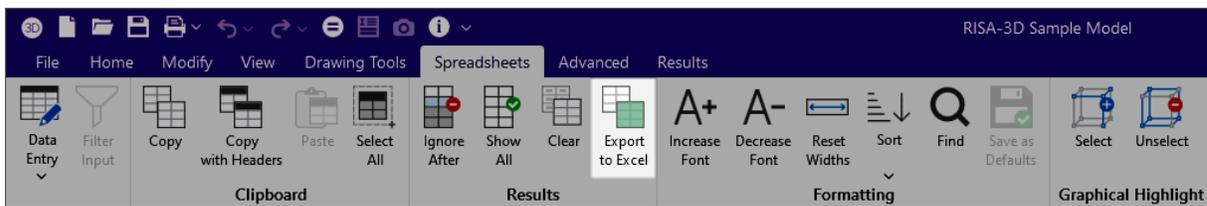


- Choose if you want to export the current spreadsheet tab or all tabs.



- Select Excel file name and file save location and export.

 **Note:** The Export to Excel can also be used by clicking the button on ribbon.



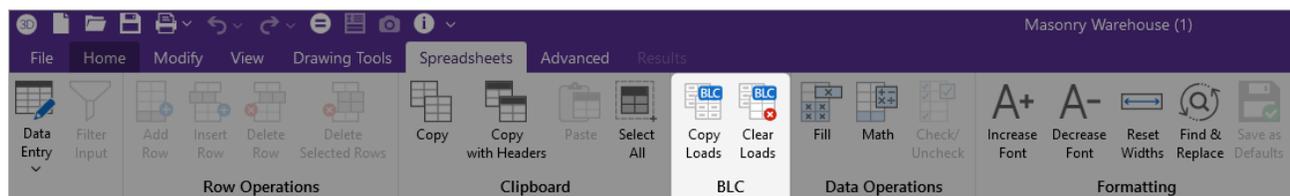
Special Spreadsheet Functions

There are special functions that help you with a particular spreadsheet. For example you may generate K factors for members on the **Member Design Parameters Spreadsheet**. These features may be accessed on the **Window Toolbar** or you may right click your mouse on the spreadsheet and choose the feature on the **Shortcut Menu**.

Member Design Spreadsheet

Click **K** to approximate the K factor for the current member or all of the members. See [K Factors](#) (Effective Length Factors) learn more about this feature.

Basic Load Case Spreadsheet

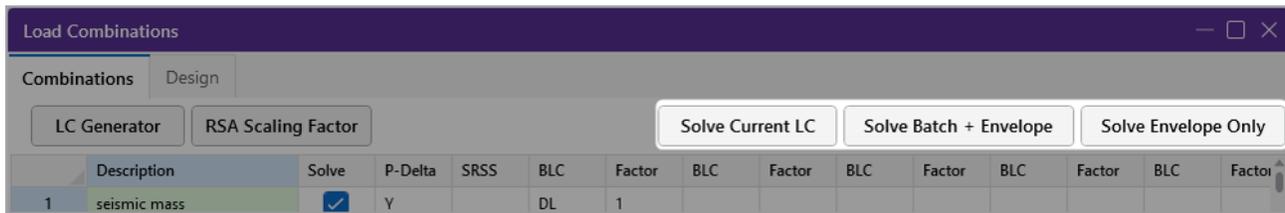


Click  to copy loads from one basic load case to another. You may choose certain load types such as distributed loads and point loads. Once you have created a copy you can use the spreadsheet tools to quickly modify the loads.

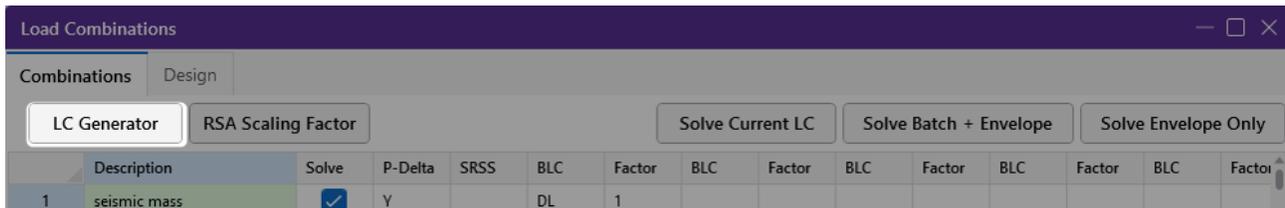
Click  to clear loads from a basic load case.

Load Combinations Spreadsheet

Solution options are easily available while working in the Load Combinations spreadsheet.



You can click LC Generator button to have the program generate load combinations based on a variety of Building Codes. See [Generating Building Code Combinations](#) for more information.

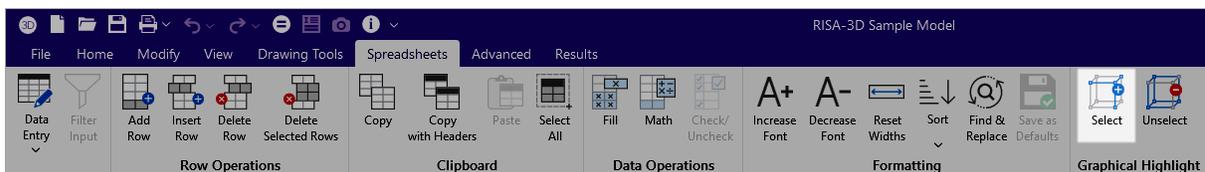


Selecting Items in the Model View

In certain results spreadsheets there is the capability to have elements selected graphically.

The process is as follows:

1. Open the desired results spreadsheet.
2. Highlight the lines from your spreadsheet that you wish to view graphically, by clicking on the number of those lines on the far left.
3. Click the **Select** icon in the Graphical Highlight section of the Spreadsheet ribbon.



4. Only those elements highlighted will be selected graphically.

The screenshot displays the RISA-3D interface. On the left, a 3D view of a wall panel is shown with a grid of elements. On the right, a spreadsheet titled 'Wall Panel ACI 318-11: Concrete Code Checks (In Plane)' is visible. The spreadsheet has columns for Wall Panel, Region, Max UC, LC, Shear UC, LC, Pn*phi[k], Mn*phi[k-ft], and Vn*phi[k]. A context menu is open over the spreadsheet, and the option 'Show Selected Lines in Current View' is highlighted with a red box.

Wall Panel	Region	Max UC	LC	Shear UC	LC	Pn*phi[k]	Mn*phi[k-ft]	Vn*phi[k]	
1	WP1	R1	0.001	1	0.003	1	NC	7162.04	493.651
2	WP2	R1	0.02	1	0.001	1	8853.459	7811.119	666.005
3	WP3	R1	0.035	1	0.02	1	901.183	288.682	84.694
4						1	NC	3267.582	336.58
5						1	NC	3267.582	336.58
6						1	812.089	570.609	115.352
7	WP4					1	901.183	210.939	115.421
8						1	NC	3030.377	314.141
9						1	NC	3053.179	252.328
10						1	901.183	276.343	91.025
11	WP6					1	4426.73	NC	427.541
12	WP7					1	2836.274	462.913	359.019
13						1	NC	443.405	112.193
14						1	2650.759	1268.591	263.817
15						1	NC	1555.514	224.387
16						1	NC	1555.514	224.387
17						1	1775.971	NC	192.144
18						1	NC	1555.514	224.387
19						1	NC	1540.603	224.387
20						1	1775.971	NC	224.387
21		R10	0.001	1	0.01	1	NC	1555.514	224.387
22		R11	0.003	1	0.003	1	NC	1540.603	224.387
23		R12	0.009	1	0.015	1	NC	1540.603	224.387
24	WP8	R1	0.023	1	0.005	1	2650.759	415.043	228.283
25		R2	0.002	1	0.006	1	NC	277.395	89.755
26		R3	0.021	1	0.013	1	1060.303	238.31	134.632



Note:

The inverse process can be used with the **Unselect** in the Graphical Highlight section.

The screenshot shows the software ribbon with the 'Graphical Highlight' section active. The 'Unselect' button is highlighted with a red box. Other buttons in the section include 'Select'.

Stability

Instabilities occur whenever a node can deflect or rotate without limit. Put another way; **a node is unstable if there is nothing to restrain it.**

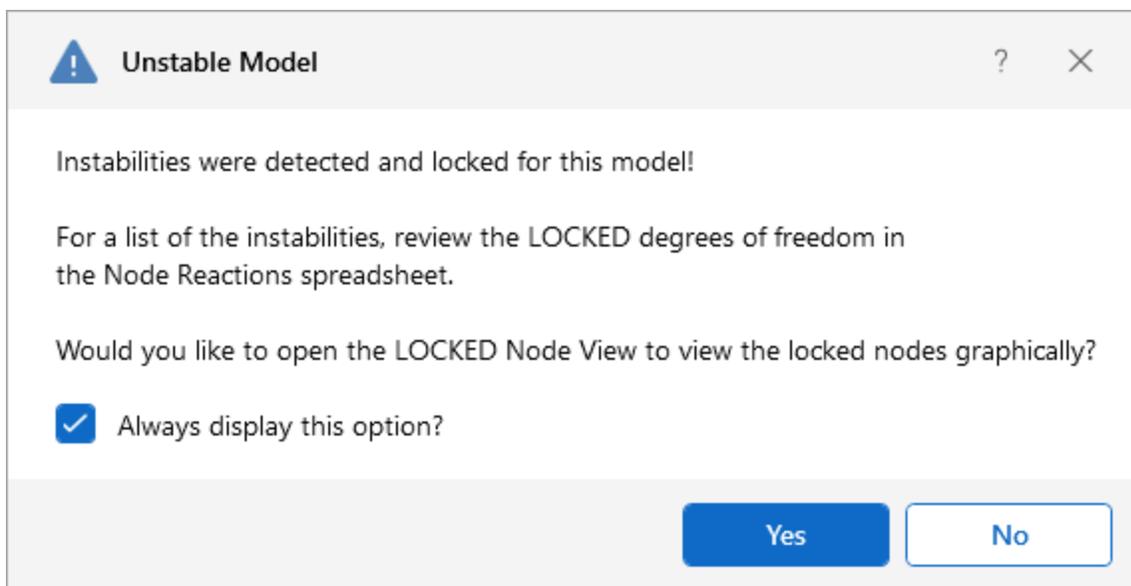
With that one statement you can understand and resolve any instability problem. Instabilities are easy to understand and easy to fix. The next section explains what RISA-3D does with instabilities. The following sections provide some simple examples of instabilities and their resolution.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Instabilities**.

Instability Procedure

Because many instabilities are inconsequential to the results yet they prevent a solution RISA-3D locks them as they are discovered and proceeds with the solution. This locking is a boundary condition that removes the degree of freedom from the solution. A reaction (if any) is not calculated and that is one of the dangers of ignoring instabilities. See [Testing Instabilities](#) to learn how to test if an instability is affecting the results.

At the end of the solution you will be notified that nodes have been locked and that you may view the affected nodes in a model view. These locks will also be reported in the **Reactions** spreadsheet.



Note: Isolated rotational instabilities do not produce a notification that nodes have been locked. You can adjust this in the **Output** section of the **Application Settings** from the **File** menu if you wish to be warned about all instabilities.

Instability Causes

Common causes of instabilities are briefly mentioned here and then highlighted in the following examples.

Member End Releases – Boundary Conditions

Overuse of member end releases and/or boundary conditions is by far the most common cause of instability as shown in the examples below. The solution is to either remove a member end release or change a boundary condition so that the node is restrained. **At least one member or boundary needs to be fixed to each node to prevent instability.** If you think of a node as the end of a member and specify no release for that member end this member still will not experience moment at the end if all other elements are left unfixed.

Unconnected Elements

Nodes that are not connected into the model cause instabilities. This is much more common in models without Physical Members as can be seen below. The solution may be to merge the model with the model merge feature.

Flexible Elements

Members or plates with relatively small properties such as a long member with a moment of inertia of 1.0 in^4 can cause instabilities. This is usually not a problem unless the members are not used properly.

Instability Examples

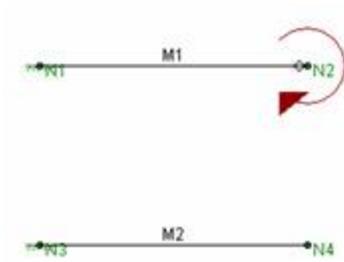
These simple examples are provided to directly address the common scenarios that occur in structural modeling. You will notice a recurring theme so once you understand one or two of them you will have a handle on the causes and resolutions for most instabilities, including those in more complex models.

Remember the golden rule as you look at each example: **A node is unstable if there is nothing to restrain it.**

Cantilever Beam/Column

If a member end release is specified at the free end of a cantilever the node becomes unstable because it is free to rotate without any resistance.

Cause: Specifying a member release at the free end of a cantilever member.



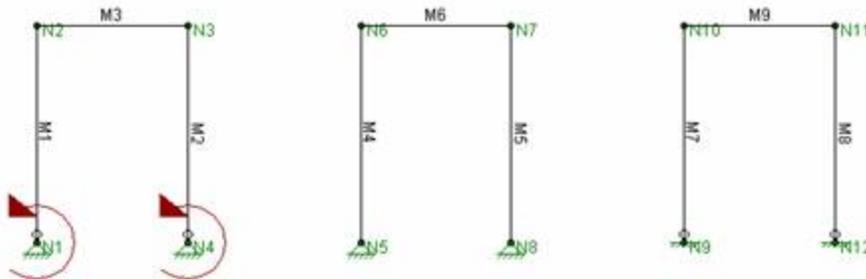
Resolution: Remove the member end release.

If the member is not released from the node then the member provides resistance to the node so that it cannot rotate without limit. The member end moves and rotates and the node goes along for the ride. There will be no moment at the free end of the member since there is nothing there to pass moment.

Column at a Support

If a pinned column base is modeled with a pinned boundary condition **AND** a member end release at the base of the column the node becomes unstable because it is free to rotate without any resistance.

Cause: Specifying a pinned boundary condition **AND** a member release.



Resolution: Either remove the member end release or specify a fixed boundary condition. Do not do both unless you want a fixed column base that resists moment.

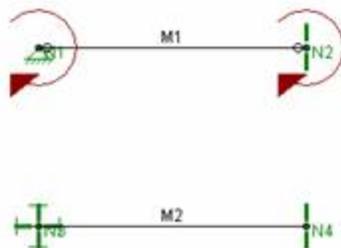
If the member is not released from the node then the member provides resistance to the node so that it cannot rotate without limit. The member end rotates and the node goes along for the ride. There will be no moment at the base of the member since the pinned boundary cannot resist moment.

If instead the boundary is specified as fixed then the boundary provides resistance to the node so that it cannot rotate. The member end release allows it to rotate while the node does not. There will be no moment at the base of the member since the fixed boundary cannot pass moment through the member end release.

Simply Supported Beam

If a pinned beam end is modeled with a pinned boundary condition **AND** a member end release the node becomes unstable because it is free to rotate without any resistance.

Cause: Specifying a pinned boundary condition **AND** a member release.



Resolution: Either remove the member end release or specify a fixed boundary condition. Do not do both unless you want a fixed member end that resists moment.

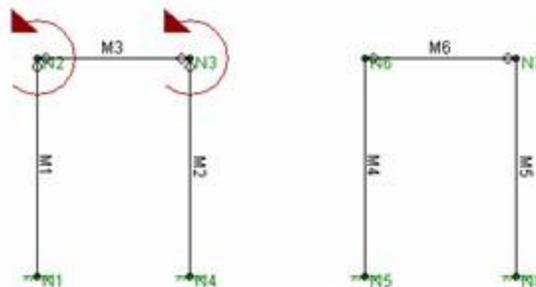
If the member is not released from the node then the member provides resistance to the node so that it cannot rotate without limit. The member end rotates and the node goes along for the ride. There will be no moment at the end of the member since the pinned boundary cannot resist moment.

If instead the boundary is specified as fixed then the boundary provides resistance to the node so that it cannot rotate. The member end release allows it to rotate while the node does not. There will be no moment at the end of the member since the fixed boundary cannot pass moment through the member end release.

Beam-Column Connection

If a pinned beam/column connection is modeled with a released column end **AND** a released beam end the node becomes unstable because it is free to rotate without any resistance.

Cause: Specifying a released column end **AND** a released beam end.



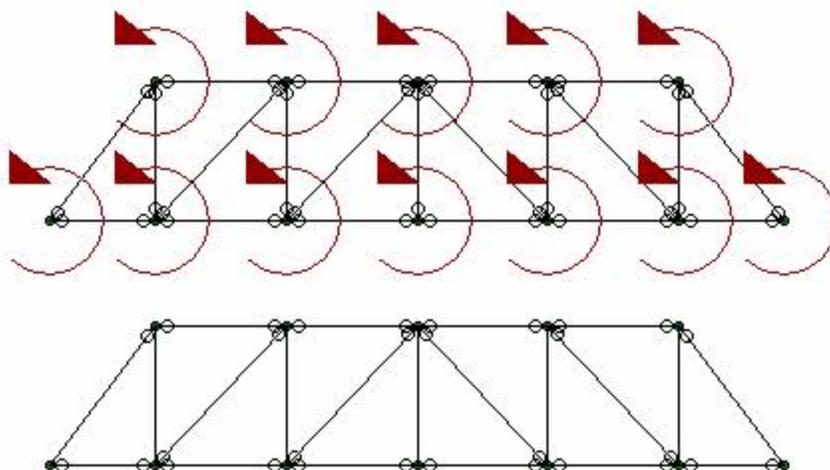
Resolution: Either remove the column end release or the beam end release. Do not do both unless you want a fixed connection that resists moment.

If the column is not released from the node then the column provides resistance to the node so that it cannot rotate without limit. There will be no moment at the connection since the beam end release cannot pass moment.

Simple Truss

If a truss panel point is modeled with releases at the ends of EVERY member connecting to that point the node becomes unstable because it is free to rotate without any resistance.

Cause: Specifying all members with released ends.



Resolution: Either remove one end release or add a rotational boundary condition at each node. Do not do both.

If one member end is not released from each node then the member provides resistance to the node so that it cannot rotate without limit. There will be no moment at the member end since the remaining members have end releases and cannot pass moment.

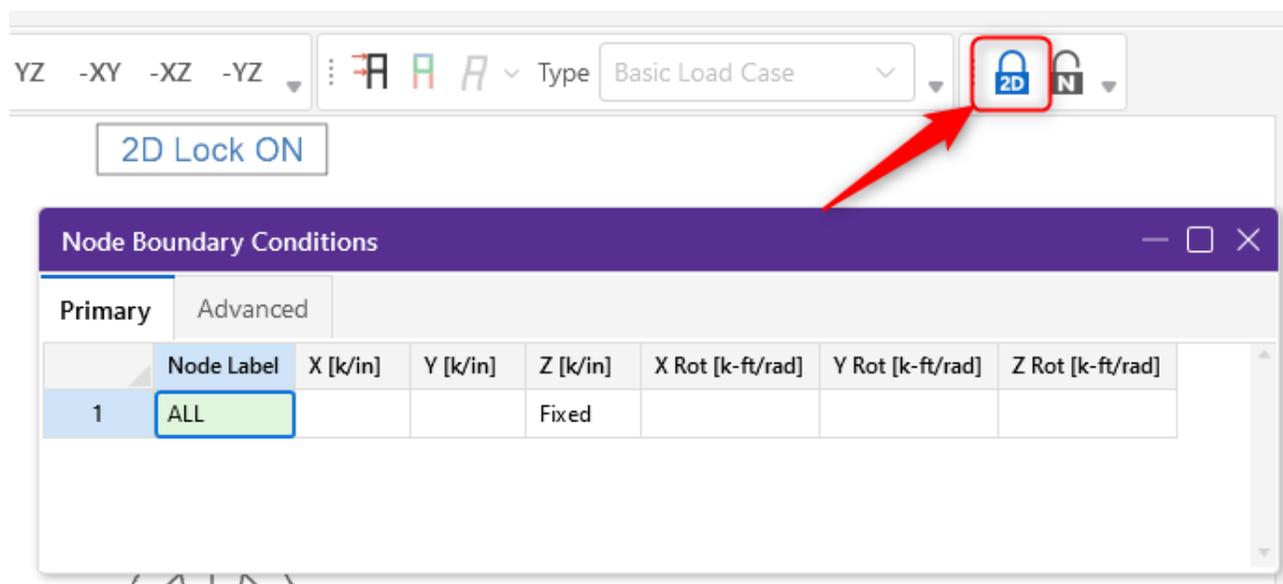
You can also solve the problem with a rotational boundary condition at each node. Using the ALL code you can restrain each node for rotation and proceed to use end releases at all members.

2D Models

If you are solving a 2D model defined in the XY plane and you're only interested in the planar action, you could enter "ALL" and put an "F" (for Fixed) for Z translation, X Rotation and Y Rotation. See the following figure:

Node Boundary Conditions							
Primary	Advanced						
	Node Label	X [k/in]	Y [k/in]	Z [k/in]	X Rot [k-ft/rad]	Y Rot [k-ft/rad]	Z Rot [k-ft/rad]
1	N4	Reaction					
2	N1		Reaction	Reaction			
3	ALL			Fixed	Fixed	Fixed	

There is a 2D Mode button which can quickly add Fixed boundary conditions to all nodes in the Z translation:

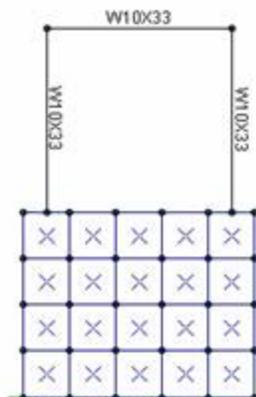


Note: If a node is explicitly listed with boundary conditions, those boundary conditions override the "ALL" conditions for all 6 directions. The "ALL" specified boundary codes apply only to those nodes NOT otherwise listed on the **Boundary** spreadsheet. This is why nodes 1 and 2 in the figure above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

Unconnected Elements

If a node, member or plate is not connected to the model then there will be instability. With the use of Physical Members this is rare in a model that only consists of beam elements however those that have plates or finite members can be defined in a way that they are not connected.

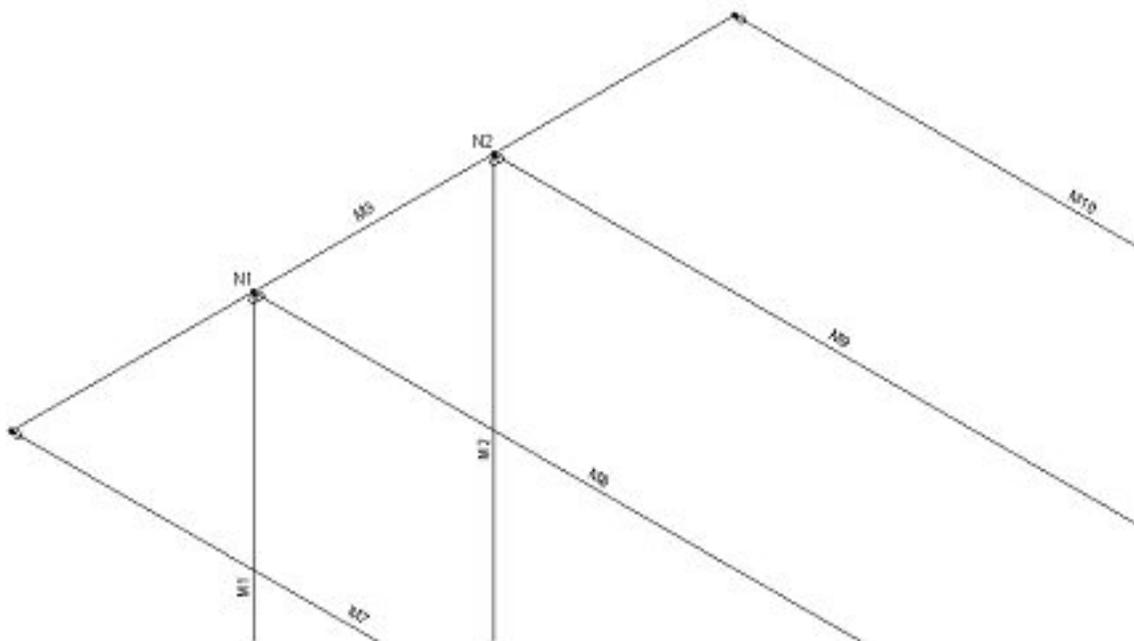
Cause: Unconnected elements. The portal frame is not connected to the plate elements because the bottoms of the columns do not fall on plate corners. The plates are stable however the portal frame is not.



Resolution: For beam models, run the [Model Merge](#) feature. For plate models redefine the mesh so that plates are connected at their corners. Model merge will not solve problems caused by lack of plate continuity.

3D Models

For three-dimensional models, torsional instabilities are not uncommon. A "torsional" instability is where a member, or a series of members, is free to spin about its centerline (local x) axis. This diagram illustrates such a situation:



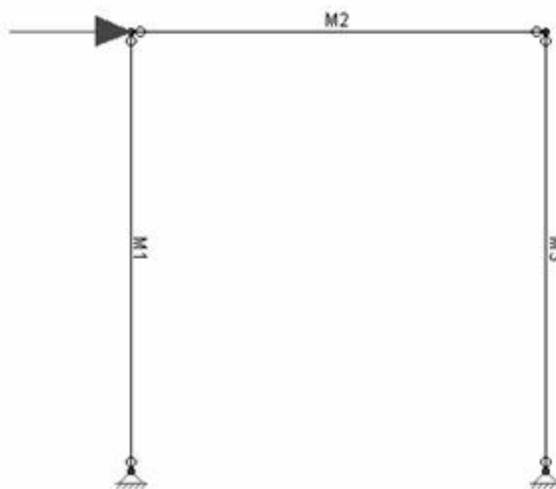
The member M3 as a whole is unstable because there is nothing to restrain it from spinning in torsion. At nodes N1 and N2 the columns and beams framing into the member (members M1, M2, M8, M9)

are pinned. The same can be said of Members M7 and M10 framing into the member ends. Therefore, there is nothing in the model that will restrict the torsional rotation of the M1 beam.

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

Testing Instabilities

Although some can be ignored, keep in mind that not all instabilities are necessarily inconsequential. Look at the following model:



This is an example of a single bent frame that is laterally unstable. To obtain a solution the lateral direction would be locked and a solution obtained, though not a correct one. The warning message may be annoying if you know the instabilities being locked are of no consequence, but there won't be any surprises.

The best way to test whether an instability is inconsequential or not is to apply a Reaction to the node in the unstable degree of freedom. Then re-run the model and examine the reactions. If the Reaction that is restraining the instability is showing a non-zero force or moment, then you have a problem with the model that must be corrected for you to get valid results. If the Reaction that is restraining the instability is showing a *ZERO* force or moment, then the instability is inconsequential to the results.

Unbraced Lengths

The member capacity for axial compression and for flexure is dependent on the spacing of elements which provide bracing along the length of a member. By default the program assumes that no bracing is provided along the length of the [Physical Member](#). This is represented by the condition where the unbraced length field is shown as empty/blank.



Note: If bracing is not provided at each end of the physical member then the assumption that the unbraced length equals the physical member length can be unconservative. The program allows you to specify an unbraced length greater than the physical member length to address this situation.

You may specify unbraced lengths as a fixed distance or by using RISA's [Unbraced Length Commands](#). The unbraced lengths are:

- **Lbyy**
- **Lbzz**
- **Lcomp-top**
- **Lcomp-bot**
- **L-torque**

The unbraced lengths that are used for member capacity calculations are listed in the **Member Detail** report after solution.



Note: Specifying bracing via unbraced lengths does not provide physical bracing to the model for analysis. It only affects the member capacity calculated for code checks. See the example below.

Hot Rolled Steel Design Parameters

Primary	Advanced	Hot Rolled	Cold Formed	Wood	Concrete Beam	Concrete Column	Aluminum	Stainless	RISAConnection				
Label	Shape	Length [ft]	Lb y-y [ft]	Lb z-z [ft]	Lcomp top [ft]	Lcomp bot [ft]	L-Torque [ft]	K y-y	K z-z	Cb	Function	Channel Conn.	a [ft]
1	M1	W8x10	12			Lbyy					Lateral	N/A	N/A

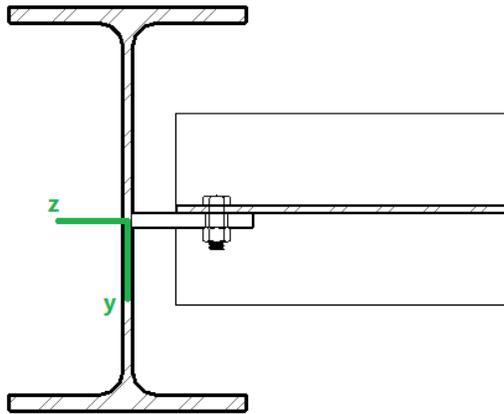
For compression capacity, M1 (above) is braced at N3 based on specified Lbyy and Lbzz

However, for analysis this member is not considered braced (see above)

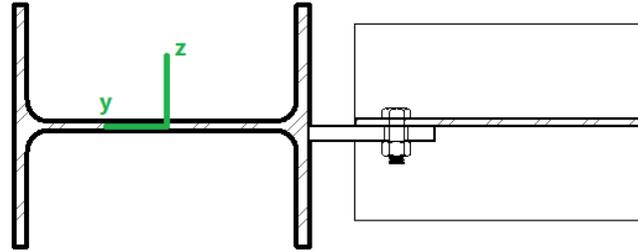
A bracing member (M2) must be modeled to provide bracing for analysis

Lb Values (Lb, Lu, Le)

The **Lb** values: **Lbyy** and **Lbzz**, represent the distance between points which brace the member against Flexural (column-type) Buckling *about* the member's local y and z axes, respectively. **Lb** bracing prevents the entire member from moving laterally (perpendicular to its own axis). These **Lb** values are used to calculate slenderness ratios (KL/r) for both directions, which are used in the calculation of member axial compression capacity.



**Lb-yy Bracing prevents column
from buckling about y-y axis**



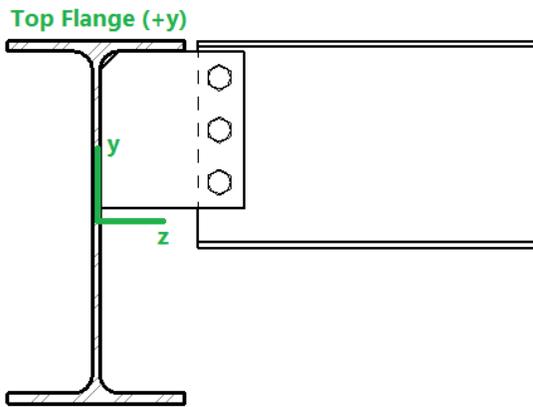
**Lb-zz Bracing prevents column
from buckling about z-z axis**

Lcomp Values (Lcomp, Le-bend)

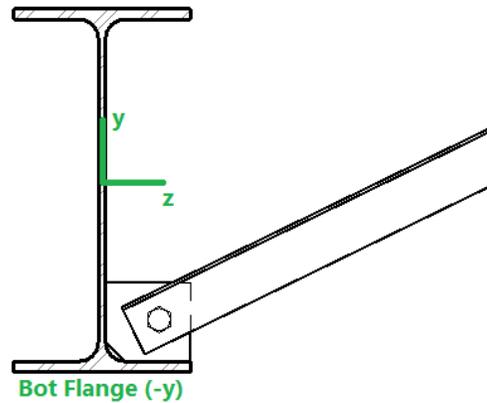
The **Lcomp** values, **Lcomp-top** and **Lcomp-bot**, represent the distance between points which brace the top or bottom flange of the member against Lateral-Torsional (beam-type) Buckling. These **Lcomp** values are used to calculate the member's flexural (bending) capacity. Where the top flange of the member is in compression due to bending, **Lcomp-top** is used. Where the bottom flange of the member is in compression due to bending, **Lcomp-bot** is used. **Lcomp** bracing prevents the member's flange from moving laterally (perpendicular to the member's axis).

Members which experience a moment reversal along their length (such as fixed-end or continuous members) have compression in both the top and bottom flange (although not simultaneously). The program uses the appropriate **Lcomp** (top or bottom) based on moment direction to calculate the bending capacity at each [internal section](#) along the member.

The top flange is the flange on the positive local y-axis side of the member. Therefore if a beam is rotated 180 degrees about its own axis (flipped upside down) the "top" flange will actually be facing downwards.



L_{comp-top} Bracing prevents lateral movement of beam's top flange



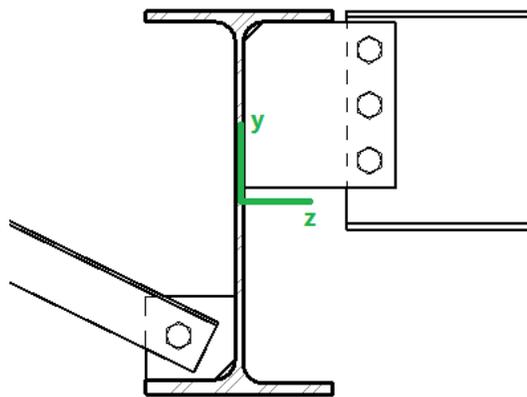
L_{comp-bot} Bracing prevents lateral movement of beam's bottom flange



Note: Concrete members are not checked for lateral-torsional buckling, so **L_{comp}** is not used for concrete.

L-torque

The **L-torque** value represents the distance between points which restrain the member against twisting about its own axis. This value is used to calculate the member's Torsional Buckling and Flexural-Torsional Buckling capacity. These limit states affect the member's axial compression capacity.



L-torque Bracing prevents beam from twisting about its own axis

**Note:**

- The L-torque value is NOT used for calculations of stiffness or stress for members subjected to warping. This is done using an internal "warping length" set by the program.
- Not all design codes check for Torsional Buckling or Flexural-Torsional Buckling. For those that do, these limit states will only be checked if **L-torque** > **L_{bzz}** and **L_{byy}**.
- Currently the AISC 360 code, CSA S16-14 code, Aluminum codes, and the 2010 and newer Cold Formed Steel codes support this unbraced length.

Automated Unbraced Length Commands

Aside from leaving the unbraced length blank or inputting a fixed distance, you can harness the program's ability to use a limited intelligence for determining unbraced lengths. In order to do this you can simply type in the name of the Unbraced Length Commands into the unbraced length field. If a valid command word is used then the program will accept and display that command in lieu of a distance value.

	Label	Length[ft]	Lbyy[ft]	Lbzz[ft]	Lcomp Top[ft]	Lcomp Bot...	L-torque[ft]
1	M1	10	7	3D	Framing	Lbyy	Segment

Some member types are pre-populated with an Unbraced Length Command when they are created. Below is a list of the commands which may be used:

Segment

When this command is used, all Nodes which fall along the length of the physical member are assumed to provide bracing. If Nodes are not evenly spaced along the member then an unbraced length is calculated for each "segment". A segment is defined as the distance between adjacent Nodes. In the example below, the member is defined as one physical member spanning from N1 to N2. The intermediate node N3 is considered as a brace point when Segment is specified for this member. Therefore the forces which occur in Segment 1 will be compared against a capacity calculated using L_{b1} , and the forces occurring in Segment 2 will be compared against a capacity calculated using L_{b2} .



When using the Segment command the program will assume that each node along the length of the member can act as a brace location regardless of whether any bracing or restraint exists. In the

example above this will result in an unconservative member capacity if no bracing will be supplied at N3 in real life.

Lbyy

This command is only available for Lcomp-top and Lcomp-bot. When this command is used, the value entered for Lbyy will also be used for Lcomp.

Properties

Hot Rolled Steel Members (1)
▼

Member ^	
Label	M1
Length, ft	12

General Properties ▼

Additional Properties ▼

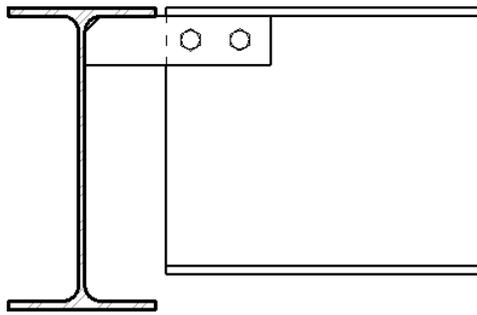
RISAConnection Properties ▼

Design Properties ^

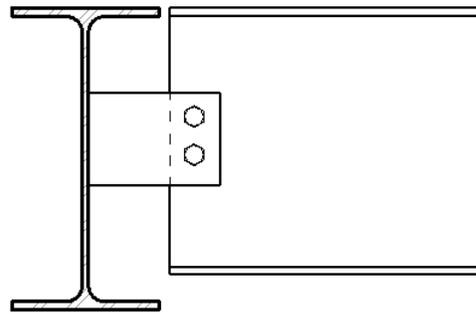
Lb y-y, ft	7
Lb z-z, ft	
Lcomp top, ft	Lbyy
Lcomp bot, ft	
L-Torque, ft	

When Lbyy is used for Lcomp, the value in the Lbyy field will also be used for Lcomp

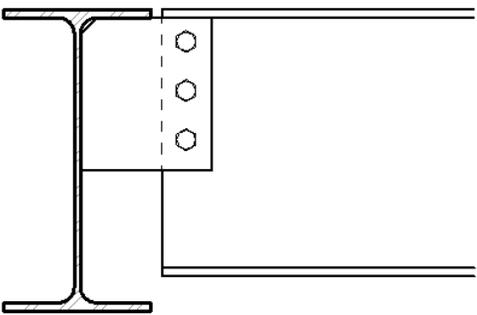
In the example below, Case C, the Lbyy command could be appropriately be used in the Lcomp-top field. In the example below, Case D, the Lbyy command could be appropriately used in both Lcomp fields. It would not be appropriate to use the Lbyy command for Case A or Case B below.



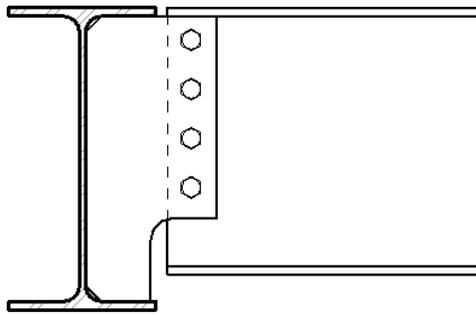
A) The beam is braced for $L_{comp-top}$ Only



B) The beam is braced for L_{by} Only



C) The beam is braced for $L_{comp-top}$ and L_{by}
 $L_{comp-top} = L_{by}$

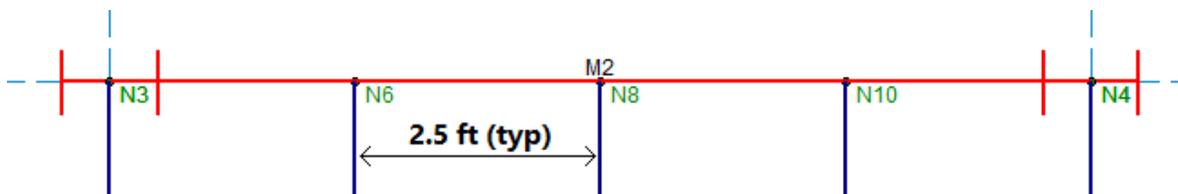


D) The beam is braced for L_{by} ,
 $L_{comp-top}$ and $L_{comp-bot}$
 $L_{comp-top} = L_{by}$
 $L_{comp-bot} = L_{by}$

Floor

For RISA-3D models which are linked to RISAFloor models, if an unbraced length is set in RISAFloor, it is listed as **Floor** in RISA-3D spreadsheets. This indicates that the unbraced length was set in RISAFloor for all locations along the length of the member, and those same values are being used in the RISA-3D model.

In the example below, the fixed-end beam M2 is using the Framing command for $L_{comp-bot}$ in RISAFloor.



	Label	Length[ft]	Lbyy[ft]	Lbzz[ft]	Lcomp Top[ft]	Lcomp Bot[ft]	L-torque[ft]
1	M3	10			Framing	Framing	
2	M2	10			Framing	Framing	
3	M4	10			Framing	Framing	

When this beam is brought into RISA-3D, the RISA-3D member properties show that Lcomp-bot is referencing RISAFloor (**Floor**).

After the model is solved you can confirm in the Member Detail Report that the RISAFloor unbraced length is being used. The maximum code check occurs at 0 feet from the i-end of the member. Because this is a fixed-fixed beam, the bottom flange is in compression there so Lcomp-bot is used instead of Lcomp-top.

Material Specific Behavior

Hot Rolled Steel

- For AISC ASD 9th Edition calculation of Kl/r for WTs and Double Angles, there is an effective Kl/r ratio that may be used when Flexural-Torsional buckling controls. This can be found in the commentary Section E3.
- Specifying LcompTop and LcompBot both as zero constrains a single angle to bend/buckle about its geometric axes. Otherwise it will behave about its principal axes. Alternatively, specifying L-torque as zero also constrains single angles to bend/buckle about their geometric axes. See [Member Results](#) for more information on Single Angle behavior.
- For single angles behaving about their principal axes, Lbyy specifies bracing against buckling about the minor principal axis. Lbzz specifies bracing against buckling about the major principal axis. See [Member Results](#) for more information on Single Angle behavior.

Cold Formed Steel

In general, the unbraced lengths for Cold Formed bending checks are based on the **Lcomp** value and axial checks are based on **Lb** values. Below describes more specifics based on the AISI 2012 code.

Lateral Torsional Buckling (AISI C3.1.2)

$$L_t = L_{comp}$$

Lcomp is the unbraced length for the flange that's in bending compression at that section (either Lcomp_top or Lcomp_bot).

This applies to all shapes and all AISI codes.



Note: Older Versions of RISA-3D (v12 and older)

$L_t = \text{smaller of } L_{by} \text{ and } L_{comp}$

When Lateral Torsional Buckling governs over local buckling, the display of the S_{eff} in the Detail Report will report the S_c which is the elastic section modulus of effective section calculate relative to the extreme compression fiber at F_c .

Distortional Buckling (AISI C3.1.4)

$L_m = L_{comp}$

L_{comp} is the unbraced length for the flange that's in bending compression at that section (either L_{comp_top} or L_{comp_bot}).

Axial Strength (AISI C4.1)

$L =$ Could be L_{by} or L_{bz} , whichever gives the larger of $(KL/r)_{yy}$ or $(KL/r)_{zz}$

Flexural Torsional Buckling or Torsional Buckling (AISI C4.1.2)

$L_t = L\text{-Torque}$



Note: Older Versions of RISA-3D (v12 and older)

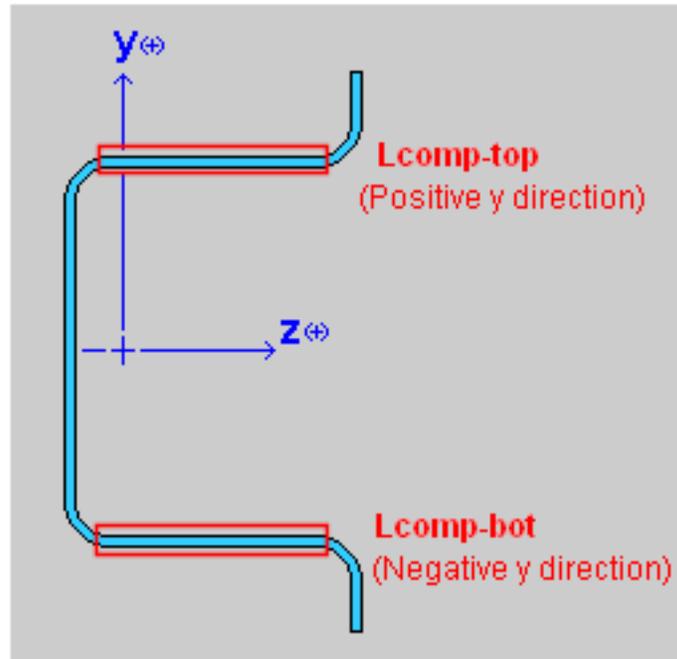
$L_t = \text{smaller of } L_{by} \text{ and } L_{comp}$

Distortional Buckling (AISI C4.2)

$L_m = L\text{-Torque}$

- The Torsional Warping constant for Back-to-Back Channels and Tracks, C_w is calculated per the AISI prescribed doubly symmetric shapes (AISI-08 Manual 3.3.3) that are continuously welded (all the unbraced lengths are 0). If the Back-to-Back shape is not continuously welded (any of the unbraced lengths larger than zero), C_w is twice the value of each individual section.
- For Hat Channel (HU) shape types, the **L_{comp_top}** and **L_{comp_bot}** values only apply to the flanges perpendicular to the local y axis (see image below). Therefore, if your loading is applied in the local z direction, these entries will not apply. This assumption was made in

reference to section *C3.1.2.1 of the AISI Specification with Commentary* whose footnote tells us that the limit state of Lateral-Torsional Buckling does not apply to these shapes.

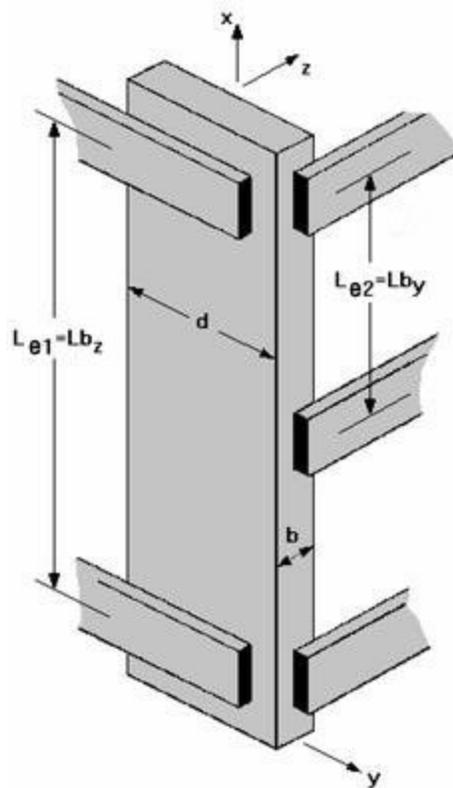


Concrete

- The unbraced lengths for Flexural Buckling (L_{byy} , L_{bzz}) are called L_{u-yy} and L_{u-zz} for concrete. Their behavior is the same as the behavior for L_{byy} and L_{bzz} .
- The unbraced lengths are used for the [Moment Magnification](#) procedure in older versions of the ACI code.

Wood

- The unbraced lengths for Flexural Buckling (L_{bzz} , L_{byy}) are called L_{e1} and L_{e2} for wood. Their behavior is the same as the behavior for L_{byy} and L_{bzz} .



- The unbraced lengths for Lateral-Torsional buckling ($L_{comp-top}$, $L_{comp-bot}$) are called $L_{e-bend Top}$ and $L_{e-bend Bot}$ for wood. Their behavior is the same as the behavior for $L_{comp-top}$ and $L_{comp-bot}$.
- See AF&PA NDS-2012, Table 3.3.3 for information on how to determine L_{e-bend} based on the unbraced length. This procedure is not followed automatically in RISA.

Aluminum

- Specifying $L_{compTop}$ and $L_{compBot}$ both as zero constrains a single angle to bend/buckle about its geometric axes. Otherwise it will behave about its principal axes. Alternatively, specifying L -torque as zero also constrains single angles to bend/buckle about their geometric axes. See [Member Results](#) for more information on Single Angle behavior.
- For single angles behaving about their principal axes, L_{byy} specifies bracing against buckling about the minor principal axis. L_{bzz} specifies bracing against buckling about the major principal axis. See [Member Results](#) for more information on Single Angle behavior.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Unbraced Lengths**.

K Factors (Effective Length Factors)

Effective Length Factors (K) are recommended or required for some design codes. The effective length factor allows you to adjust the unbraced length for Flexural Buckling as a simplified method of accounting for buckling effects. **Kyy** is a modifier factor for **Lbyy**. **Kzz** is a modifier factor for **Lbzz**.

If the K Factor field is left blank/empty then it is taken as 1.0, thereby not affecting the unbraced length.

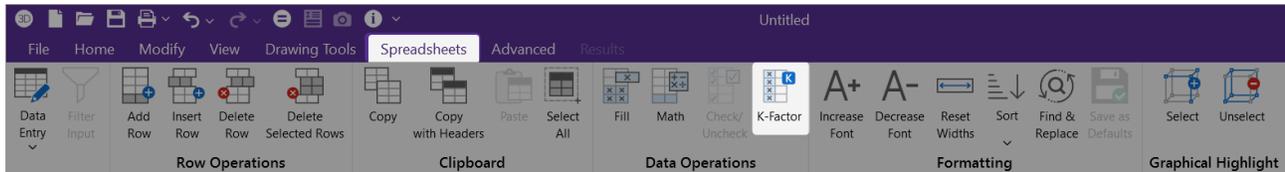
If a value is entered for a **K Factor**, that value will be used for the entire length of the physical member. If an entry is not made (left blank), the value will internally default to '1' for that member.



Note: K-torque for Torsional or Flexural-Torsional Buckling is always be assumed to be 1.0, except for Aluminum design where it is taken as the lesser of Kyy or Kzz.

K Approximation Feature

RISA-3D is able to approximate the recommended K values for a member based on the member's sway condition and end release configuration. The K-factor approximation is based on AISC 360-10, Table C-A-7.1. To access this feature you must be in the Members spreadsheet, on the tab which shows the unbraced lengths. Click the K-Factor icon on the Spreadsheets menu.



The following table gives the values the program calculates for various conditions.

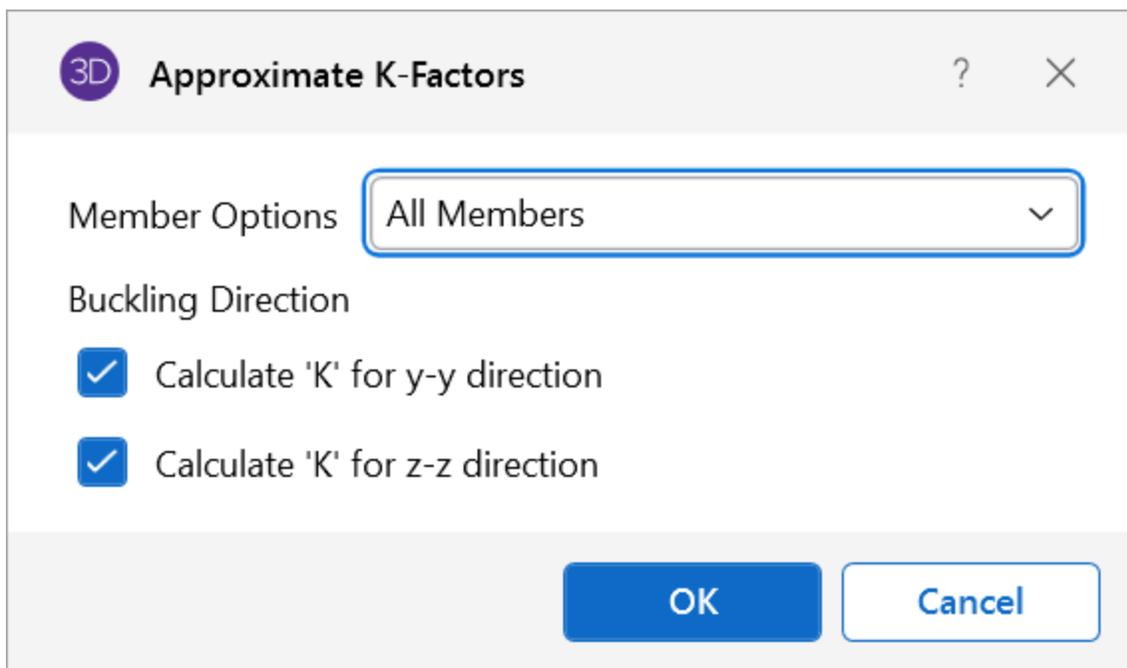
Table Case	End Conditions	Sidesway?	K-Value
(a)	Fixed-Fixed	No	.65
(b)	Fixed-Pinned	No	.80
(c)	Fixed-Fixed	Yes	1.2
(d)	Pinned-Pinned	No	1.0
(e)	Fixed-Free	Yes	2.1
(f)	Pinned-Fixed	Yes	2.0

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

Any configuration not described here is given the default value of '1.0'.

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

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



Limitation:

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

Sway Flags

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

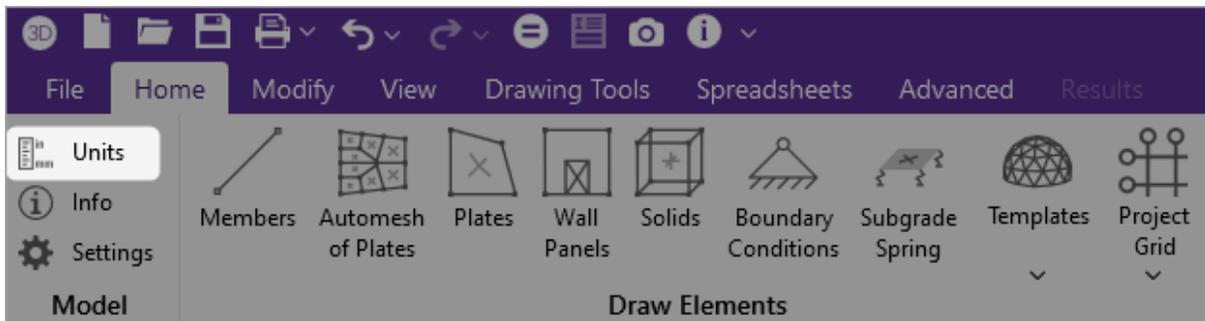
Units

You can work with imperial (kips, inches, etc.) or metric (KN, meters, etc.) units, or any combination of the two. The current units appropriate for each data item are shown at the tops of the data columns in the spreadsheets and with the plot of values in the model view.

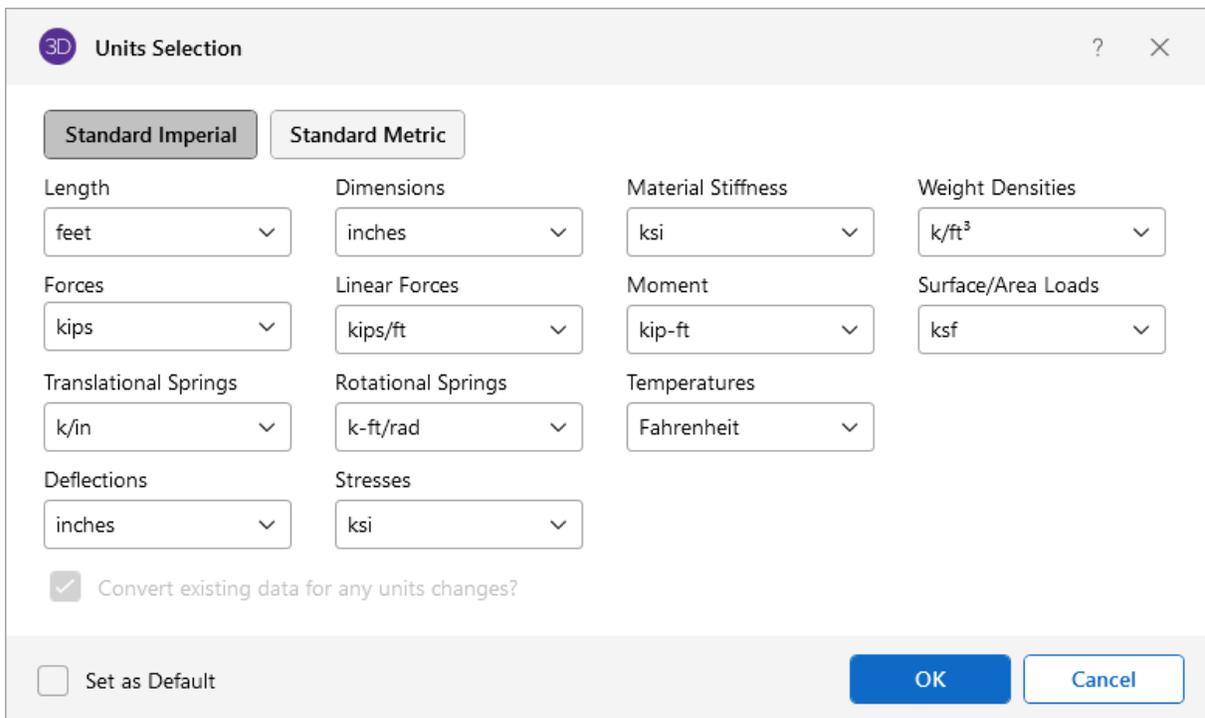
Change Units

To change units:

1. Click the **Units** icon in the 'Model' section of the Home ribbon.



The Units Selection window opens.



2. Specify the units you want for each item in the drop down lists.

Standard units systems are preset and may be specified by clicking the **Standard Imperial** and **Standard Metric** buttons.

- (Optional) Clear the check box for **Converting Existing Data** if you do not wish to convert values already entered (as shown in the following image).

The screenshot shows the 'Units Selection' dialog box with the following settings:

- Standard Imperial (selected)
- Length: feet
- Dimensions: inches
- Material Stiffness: ksi
- Weight Densities: k/ft³
- Forces: kips
- Linear Forces: kips/ft
- Moment: kip-ft
- Surface/Area Loads: ksf
- Translational Springs: k/in
- Rotational Springs: k-ft/rad
- Temperatures: Fahrenheit
- Deflections: inches
- Stresses: ksi

The checkbox 'Convert existing data for any units changes?' is unchecked and highlighted with a red box.

- (Optional) Check the **Set as Default** box to save your selections as the default.

You can save any of the units as the default setting so that when you start a new model the information is already there. (as shown in the following image).

- Click **OK** to save your change and close the window.

Standard Imperial Units

This is the units system currently prevalent in the United States. 'Feet' are used for location entries such as node coordinates and load locations, and 'Inches' are used for section property entries such as area and moment of inertia. Force and weight units are in 'kips', where 1 kip = 1000 pounds. Stress units are in 'ksi' (kips per square inch).

Standard Metric Units

This units system uses 'Meters' for location entries and 'Centimeters' for section property entries. Force units are in 'kN' (kiloNewtons), where 1 KN = 1000 Newtons. Stress units are in 'MegaPascals' (MPa), where a MegaPascal is 1,000,000 Newtons per square meter. Weight units are in 'Kilograms' and thermal units are in 'Degrees centigrade'.

Units Specifications

The following are the unit specifications and their applications:

Unit Specifications Measurements and Usage

Measurement	Usage
Lengths	Coordinates, Unbraced Lengths, Load Locations
Dimensions	Shape Properties, Plate Thickness, Member Offsets

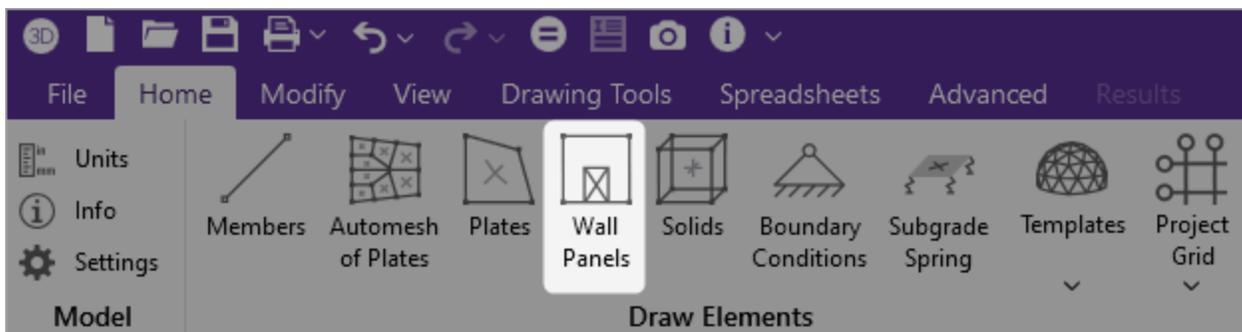
Measurement	Usage
Material Stiffness	E, G
Weight Densities	Material Density
Forces	Loads, Forces
Linear Forces	Distributed Loads
Moments	Loads, Forces
Surface / Area Loads	Plate/Shell Surface Loads, Area Loads
Translational Springs	X Y Z Boundary Conditions
Rotational Springs	X Rot, Y Rot, Z Rot Boundary Conditions
Temperatures	Thermal Coefficient, Temperatures
Deflections	Deflections, Displacements
Stresses	Fy, Allowable and Actual Stresses

Wall Panels

The wall panel element allows you to easily model walls for in plane and out of plane loads. Wall panel data may be viewed and edited in two ways:

- Graphically, in the [Wall Panel Editor](#)
- Texturally, in the [Wall Panels Spreadsheet](#).

Drawing Wall Panels



There are several graphic-editing features that make the creation and modification of models quite easy. Use the **Wall Panels** button to use these features in the model view. To create new wall panels, you can draw them using a drawing grid or draw "dot to dot" existing nodes. Once you have created these items you can use other graphic features to apply loads and set boundary conditions.

You can set many of the wall panel properties up front or you can modify these properties after you draw them. Modifying properties is discussed in the next sections. See the [Wall Panels](#) topic for information on wall panels and their properties.

The **Wall Panels** button lets you graphically draw wall panels in your model. Enter the appropriate wall panel parameters, and draw wall panels between existing nodes or on the drawing grid. Notice that the coordinates of the node or grid point that is closest to your cursor, are displayed next to your cursor. The new wall panels will be shown on screen and will be recorded in the **Wall Panels Spreadsheet**.

To actually draw a wall panel, you have two options:

- **Modifying your Drawing Grid according to how you wish to lay out your wall panels.**
 1. Use the **Create Wall Panels by Clicking on Grid Areas** option.
 2. Click in the grid areas formed by the intersecting grid lines.
As you click on an area, a wall panel is automatically created in that area.
- **Creating wall panels by drawing a diagonal that defines it.**

1. Click on the grid point or node that you want to be the start node for the diagonal.
2. Click on the end node for the diagonal.

The wall panel "stretches" like a rubber band as you draw from node to node.



Note:

- You must draw wall panels rectangular.
- Wall panels must be oriented vertically in your model.

Properties	
Wall Panel ^	
Thickness, in	18
Label	WP1
Wall Material	Concrete v
Material Set	Conc3000NW v ...
Design Rule	Typical v ...
Seismic Rule	None v ...
Additional Properties v	
Reinforcement v	
Advanced Properties v	

The parameters shown are the same parameters that you would enter on the [Wall Panels Spreadsheet](#).

To draw Wall Panels:

1. Open a model view, if you haven't done so already.
 - a. Open the **View** ribbon.
 - b. Click the **Open 3D View** icon, to open a new view.
2. If you are not drawing between existing nodes, create a drawing grid or define nodes on the **Node Coordinates** spreadsheet.
3. Click the **Wall Panels** icon.
4. Set the **Wall Panel** properties in the 'Properties' panel.
5. (Optional) In the **Properties** panel, set any other properties (Additional, Reinforcements and/or Advanced) by expanding each section and entering the properties you want to use.

6. Click **Apply** to start drawing wall panels by clicking on the nodes or grid points, using the left mouse button.

You must click on two points, or click in grid areas.

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

**Note:**

- To draw more wall panels with different properties, press CTRL-D to recall the **Wall Panel** action.
- You can also specify or edit wall panels in the [Wall Panels Spreadsheet](#).
- You can also view and edit wall panel properties by clicking on a wall panel.
- You can undo any mistakes by clicking the **Undo** icon.
- Drawing top to bottom or bottom to top determines the panel's local axes directions. See [Wall Panel Local Axis](#) for more information.

Modifying Wall Panels

There are two ways to modify wall panels. You can view and edit the member data in the **Wall Panel** spreadsheet or you can click a wall panel to view and edit its properties. You can also double click a wall panel to open the **Wall Panel Editor**.

A group of selected wall panels can be modified all at once by selecting multiple plates, then changing their properties in the **Properties** panel. See the [Graphic Selection](#) topic for more on selecting.

The parameters shown are the same as those used to define new wall panels.

Properties	
Wall Panel ^	
Thickness, in	18
Label	WP1
Wall Material	Concrete v
Material Set	Conc3000NW v ...
Design Rule	Typical v ...
Seismic Rule	None v ...
Additional Properties v	
Reinforcement v	
Advanced Properties v	

To modify Wall Panels:

1. Open a model view, if you haven't done so already.
 - a. Open **View** ribbon.
 - b. Click the **Open 3D View** icon, to open a new view.
2. You can choose to modify a single wall panel at a time or to an entire selection of wall panels.
 - To modify a single wall panel: Click on the wall panel and change the properties in the **Properties** panel.
 - To modify a selection of wall panels: Select the walls collectively by clicking and dragging over them, then change the properties in the **Properties** panel.



Note:

- To modify more wall panels with different parameters, press CTRL-D to recall the **Wall Panels** action.
- You can also modify wall panels in the [Wall Panels Spreadsheet](#).
- You can undo any mistakes by clicking the **Undo** button.
- The thickness option is only available if you are choosing a General or Concrete material. Wood and Masonry require you to change their thickness in the [Design Rules](#) spreadsheet.
- For existing models of version 9.1.1 or earlier, all wall panels will be brought in using **Custom** regions/openings. Here we provide an option that will reset all wall panels to base their design on the Wall Design Rule.

Wall Panel Spreadsheets

Another way of editing wall panels is through the **Wall Panel** spreadsheet. This spreadsheet is accessible through the Explorer panel, **Data Entry** section and includes data on two tabs: Primary and Advanced.

Wall Panel Data										
Primary										
	Label	A Node	B Node	C Node	D Node	Material Type	Material Set	Thickness [in]	Design Rule	Panel/Spacing
1	WP1	N1	N2	N3	N4	Concrete	Conc3000NW	18	Typical	N/A
2	WP2	N2	N5	N6	N3	Concrete	Conc3000NW	18	Typical	N/A
3	WP3	N4	N3	N7	N8	Concrete	Conc3000NW	18	Typical	N/A
4	WP4	N3	N6	N9	N7	Masonry	Concrete Matl	10	Typical	72
5	WP5	N8	N7	N10	N11	CFS	A653 SS Gr33	6 (stud)	Typical	0.018_#8@6/12_33 12
6	WP6	N7	N9	N12	N10	Wood	DF	5.5 (stud)	Typical	S1_3/8_8d@6 16

Wall Panel Advanced Data												
Advanced												
	Label	Seismic Rule	Design Method	SSAF	In-Plane Icr Factor	Out-Plane Icr Factor	K	Cm In-Plane	Cm Out-Plane	Stud Bracing	Sheathing Connect Dist [in]	Activation
1	WP1	None	N/A	N/A						N/A	N/A	
2	WP2	None	N/A	N/A						N/A	N/A	
3	WP3	None	N/A	N/A						N/A	N/A	
4	WP4	None	N/A	N/A				N/A	N/A	N/A	N/A	
5	WP5	N/A		N/A	N/A	N/A		N/A	N/A	All Steel	N/A	
6	WP6	N/A	Segmented	1	N/A	N/A		N/A	N/A	N/A	N/A	

The following data columns hold the primary data for the wall panels:

Wall Panel Labels

You can assign a unique label to any or all of the wall panels. You can then refer to a wall panel by its label. Each label must be unique, so if you try to enter the same label more than once, you will get an error message. You can relabel wall panels at any time, with the **Relabel Wall Panels** option on the Tools menu.

Wall Panel Nodes

The A, B, C, and D node entries are used to define the 4 corner nodes of a wall panel. The nodes must all lie on the same plane, be oriented parallel to the vertical axis and be entered in either a clockwise or counter-clockwise sequence. They can not be adjusted from the spreadsheet.

Wall Panel Material Type and Material Set

The material set label links the wall panel with the desired material defined on the [Material Spreadsheet](#).



Note: Currently wall panels can only be made up of concrete, masonry, wood or general materials.

Wall Panel Thickness

The thickness field on the **Wall Panels** Spreadsheet is the thickness of the element. This thickness is constant over the entire element. Note that the thickness for Masonry and Wood wall panels are set in the [Design Rules](#) spreadsheet. For concrete and general material walls the value is defined here.

Design Rule

This allows you to choose a specific design rule from the Design Rules spreadsheet. The design rule is where you can specify very detailed information for the wall.

Panel/Spacing

This shows the current panel for masonry and wood walls. Wood and masonry walls can require an iterative solution. This means that the panel properties used for a wall panel may change from solution to solution, so here is the place where that information is displayed. Note that the panel properties also show up in the output.

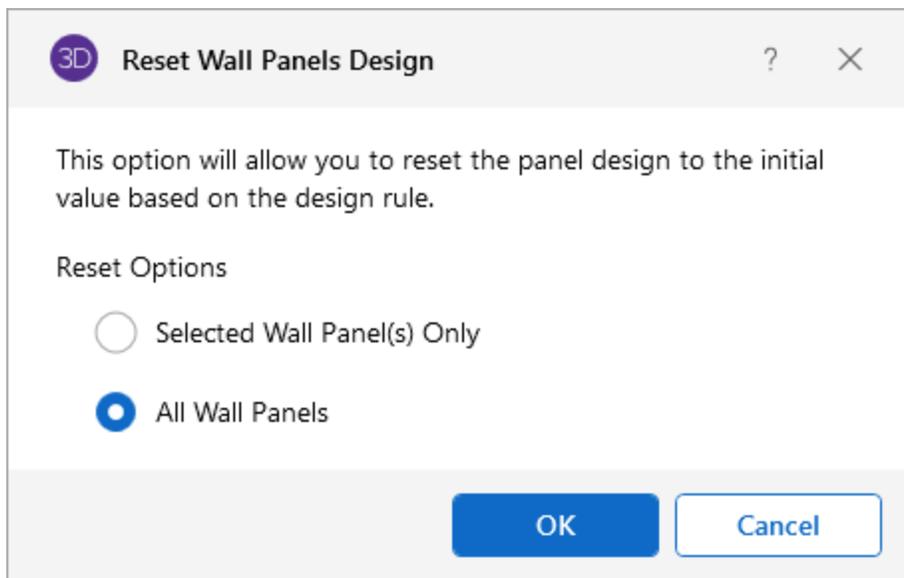
When you select a design rule for a wall panel, there may be a range of sheathing call-outs, stud spacings and bar/grout spacings. At the initial solution the program simply uses the first item in the list that meets the criteria of the design rule and then the optimization starts from there. This initial criteria is what will show up in the **Panel/Spacing** column. After you optimize your wall this panel criteria may update/change. If you want to reset all of these values to the original values, simply right click when in the **Wall Panels** spreadsheet and select **Reset Wall Panels Design**.

**Note:**

- For masonry walls we show the current bar/grout spacing.
- For wood walls we show the current sheathing call-out and the current stud spacing.
- For concrete walls this column is not applicable, as the stiffness of the wall is not affected by the program optimization.

Reset Wall Panels Design

The Wall Panels spreadsheet lists the panel parameters designed by the program after solution. From the Primary tab, you can reset the wall panels design by selecting the Reset Wall Panels Design from the right-click menu. This resets the panel design to the parameters, based on the Wall Design Rule.



Seismic Rule

This field defines a Seismic Design Rule and is applicable to concrete walls only. Please see the [Concrete Wall - Seismic](#) topic for more information.

Design Method

This is a column specific to wood wall panels and allows you to choose which design method you choose to work with: Segmented, Perforated or Force Transfer. See the [Wood Wall Panels](#) topic for more information. These design methods are not applicable for concrete, masonry, or general wall panels.



Note: Currently, wood wall panel design is only available for NDS design.

SSAF (Shear Stiffness Adjustment Factor)

This is a factor specific to wood wall panels that allows the user to manually adjust the shear stiffness of a particular wall panel. Because the program uses a finite element solution, the program does not automatically consider some contributions to the FEM deflections, such as nail slip. With this adjustment factor the user can match up the deflections from their hand calculations with the FEM node deflections at the top nodes in the wall.

Icr (In Plane and Out of Plane)

These values are considered for both masonry and concrete wall design and allow you to modify the stiffness of the wall for cracking considerations. This value will be multiplied by the I_{gross} of the wall.

Masonry Walls

By default (if left blank) the program will use a value of 0.50 for both In-plane and Out-plane I_{cr} Factor. This value comes from Section 3.1.5.2 of the ACI 530-11 code. If you have performed a cracked section analysis or want to override this default you can input it directly here.

Concrete Walls

By default (if left blank) the program will use a value of 0.70 for In-plane I_{cr} Factor and 0.35 for Out-plane I_{cr} Factor. These are the minimum factors for beams (Out-plane) and columns (In-plane) per ACI 318-14 Section 6.6.3.1.1 (ACI 318-11 Section 10.10.4.1). If you have performed a cracked section analysis or want to override this default you can input it directly here.

For service level analysis, the level of cracking will be significantly less. Therefore, the stiffness used in your analysis should be representative of the reduced loading and reduced cracking. Per the ACI 318-14 Section R6.6.3.2.2 (ACI 318-11 Section R10.10.4.1), the program will account for this increased stiffness by applying a factor of 1.43 to the cracked section properties for any load combination that has the “**Service Load**” flag checked on the **Design** tab of the **Load Combinations Spreadsheet**



Note:

- This factor will only be used if the **Use Cracked Sections** checkbox is checked in the Concrete tab of [Model Settings](#).
- This factor (or 1.43*I_{cr}) can not be greater than 1.0.

K Factor

This is the effective length factor and is available concrete, masonry and wood walls. If left blank this will be taken as 1.0.

**Note:**

- For masonry this factor affects ACI 530-11 equations 2-16, 2-17, 2-19, 2-21, 2-22, 3-11, 3-12, 3-18, 3-19, and Section 3.3.5.3.
- For wood this affects the C_p factor calculation (specifically F_{cE}) for wood compression capacity for both studs and chord design.

Cm In/Out

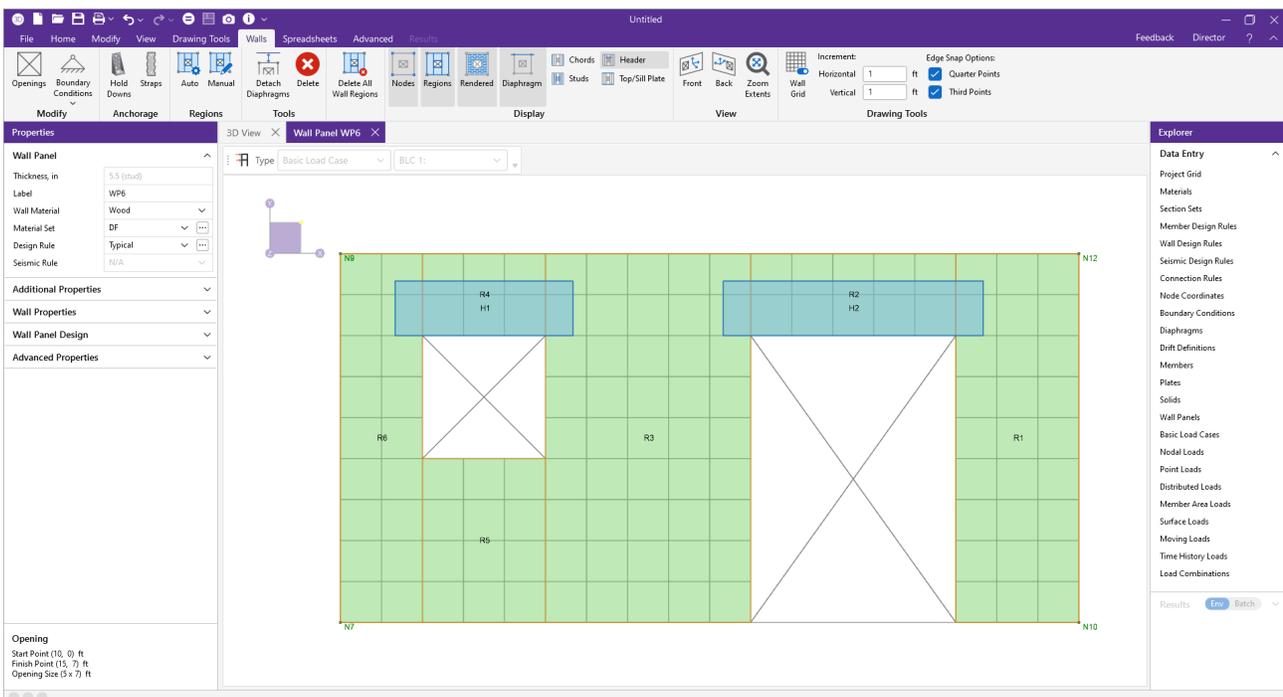
This input controls the C_m calculation for P-Little Delta calculations for concrete wall panels. If this is left blank the program will auto-calculate this value. See the **P-Little Delta** section of the [Concrete Wall - Design](#) topic for more information.

Activation

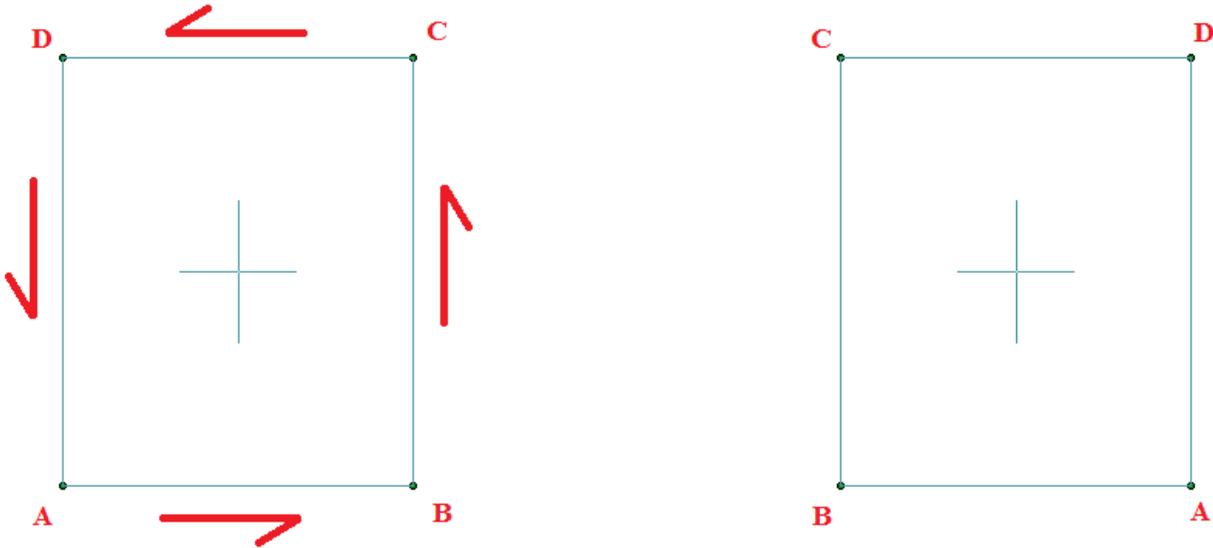
This column allows you define a wall panel as inactive or excluded. Please see the [Inactive and Excluded Elements](#) for more information.

Wall Panel Editor

The **Wall Panel Editor** lets you edit the detailed properties of a wall panel including openings, regions and boundary conditions. This window also provides design options and details for the specific panel, and is accessible by double-clicking on an existing wall panel.



If the wall panel is drawn upper right to lower left it will be drawn with the panel's local x-axis going from left to right. If drawn lower right to upper left the panel's x-axis will go from right to left and is presented here as though you are looking at the back side of the panel.



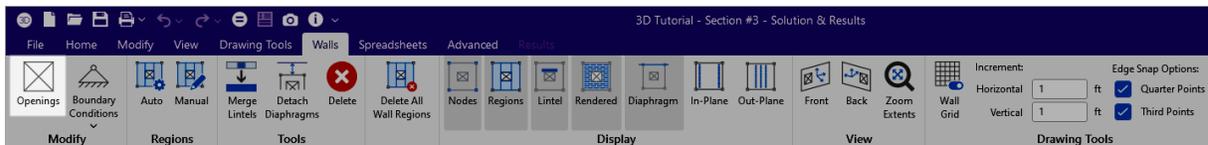
Note: There are many icons, dropdown lists and information shown depending on the type of wall panel you are working with. See the [Masonry Wall - Modeling](#), [Wood Wall - Design](#) and [Concrete Wall - Design](#) topics for more information.

Create Openings

In the Wall Panel Editor, you can add rectangular openings to the wall panel.

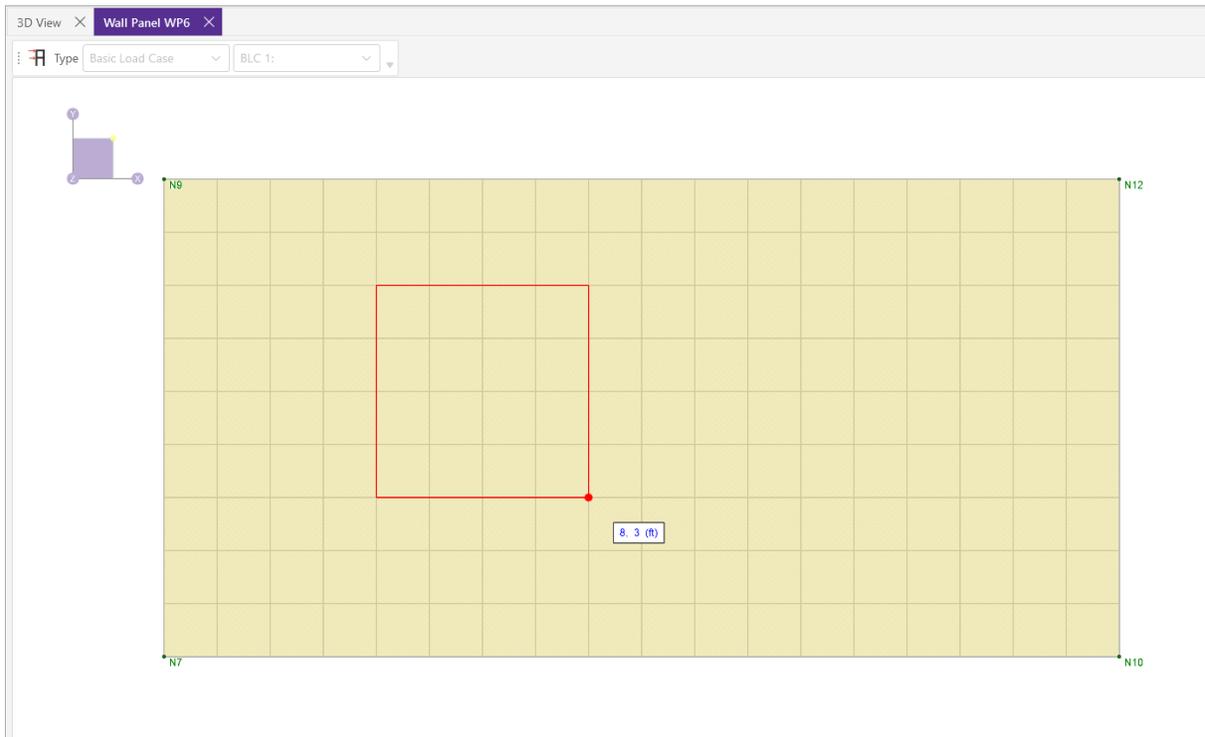
To draw an opening:

1. Click the **Openings** icon on the Walls ribbon.



2. Select two grid intersections which make up the two diagonal corners of your opening.

Notice that you can view your cursor coordinates next to your cursor. The opening dimension is also displayed based on the first click of the opening relative to the second click.



3. (Optional) Right-click your mouse, to exit this tool.

**Note:**

- Drawing an opening in a concrete wall will create a lintel above the opening. We will not design the lintel but we will give analysis results. See the **Concrete Wall - Design** topic on [lintels](#).
- Drawing an opening in a masonry wall will create a lintel above the opening. For more information on defining lintel geometry and design properties, see the **Masonry Wall - Design** topic on [lintels](#).
- Drawing an opening in wood wall will create a header above the opening. For more information on defining the header properties, see the **Wood Wall - Design** topic on [headers](#).
- Drawing an opening in a general wall panel, there is no header/lintel automatically created. The general wall panel is given as an option for analysis only.
- Openings can not overlap a region. Regions must be deleted before you draw an opening in an area. After the opening is created you can go back and redraw the regions.
- Openings must be drawn greater than 3 inches away from wall edges.

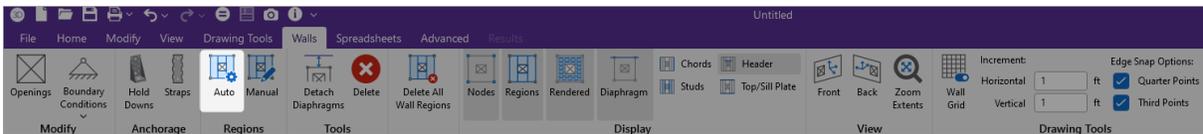
Creating Regions

Within the Wall Panel Editor, you can also create different rectangular regions within your wall panel. Regions are used to further define areas of your wall panel for use in analysis/design. If you do not specify a region in a wall panel without openings, then the entire wall panel is considered a region.

Automatically Draw a Region

To automatically draw regions:

1. Input your openings, if not already done.
2. Click the **Auto** icon in the 'Regions' section of the Walls ribbon



The program defines regions as we would expect a user to want them.



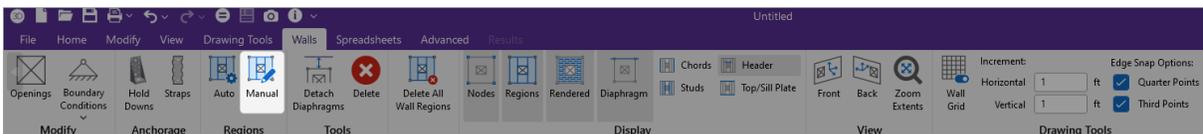
Note: If the regions defined are not located correctly by the generator, you can delete the generated regions with the **Delete** button and redraw them manually. See below for more information on this.

3. (Optional) Right-click your mouse, to exit this tool.

Manually Draw a Region

To manually draw a region:

1. Click the **Manual** icon in the 'Regions' section of the Walls ribbon.



2. Use your cursor to select two grid intersections which make up the diagonal corners of the region.

Similar to wall openings, local coordinates and region dimensions will be displayed next to the cursor in the Wall Panel Editor.

3. (Optional) Right-click your mouse, to exit this tool.

**Note:**

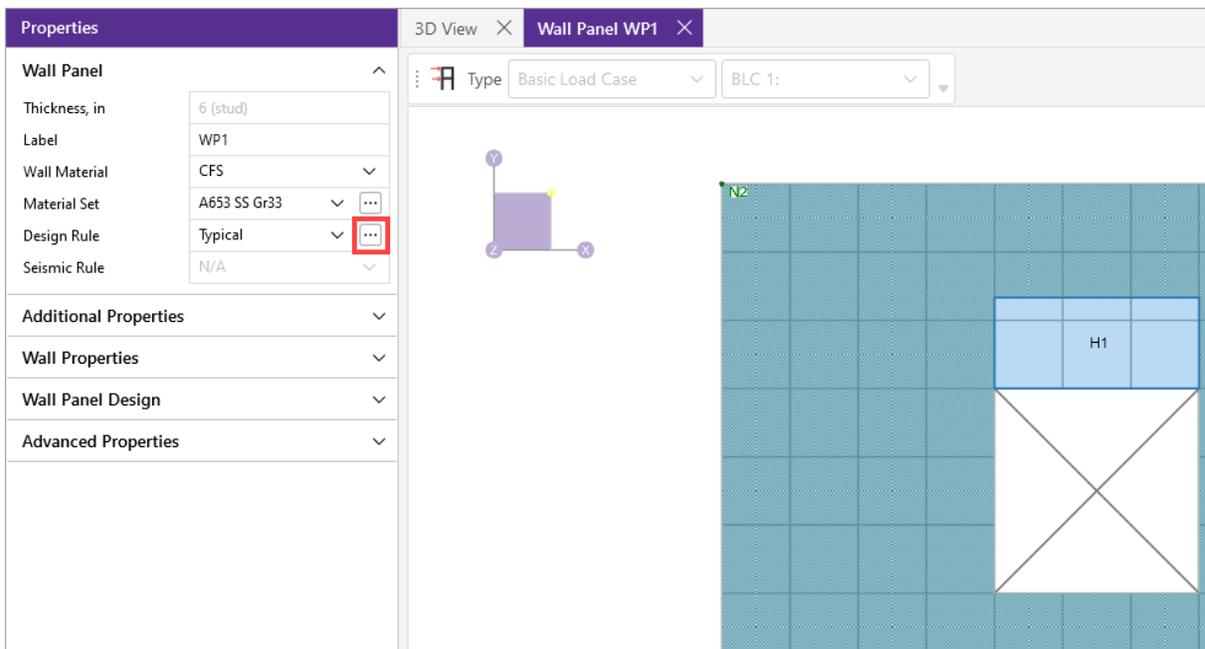
- For masonry wall panels, there is a region editor that allows you to define design properties for the region. Double-click inside the boundary of the drawn region to open this editor. See the **Masonry Wall - Design** topic for [region](#) information. Note that design and analysis results are displayed by region.
- For wood wall panels the design and analysis results are displayed by region.
- For concrete wall design, the program will automatically create regions at solution around openings. If there are floors/diaphragms that cut through your wall, then separate regions/designs will be created above and below the floors/diaphragms.
- For general wall panels we will not do any design for you. However, you can lay out your regions so that your analysis results will allow you to design your general wall panels much more easily.

Access Design Rule From Properties Panel

When using the Wall Panel Editor, you can view or edit an existing Design Rule or create a new design rule, by using the Properties panel.

To access the Wall Panel Design Rule options:

1. Go to the **Properties** panel.



2. Click on the **Design Rule**  ellipsis button.



Tip: You can also just double-click inside a wall panel in the Wall Panel Editor, to open the Review or Change Wall Design Rule.

The Review or Change Wall Design Rule window opens.



The following table explains the available Design Rule options.

Wall Design Rule - Options

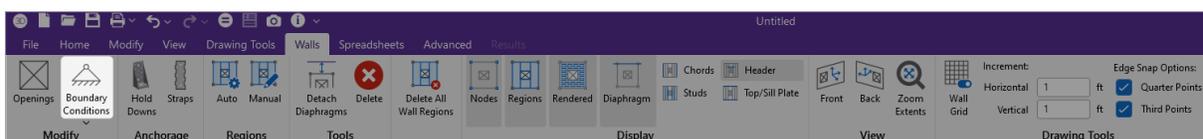
Option	Description
Create New	Create New lets you create a new Wall Design Rule to be applied to the wall.
View & Edit Existing	View & Edit Existing lets you view, as well as edit, the Wall Design Rule that is currently applied to the wall.

Boundary Conditions

Within the 'Wall Panel Editor', all boundary conditions are applied as continuous along a wall panel edge.

To set boundary conditions within the Wall Panel Editor:

1. Open the **Walls** ribbon, if not already open.



2. Click the **Boundary Conditions** icon in the 'Regions' section.
3. Select your boundary condition criteria, and click the wall panel edge to apply the boundary condition too.

You can also apply boundary conditions to your wall panel outside of the Wall Panel Editor as well, but this is the only place where you can define a continuous boundary condition.

For wood wall panels, hold downs and straps are used in the program as well. For more information on adding hold-downs and straps, see the [Wood Wall - Design](#) topic.

- (Optional) To exit out of this tool right-click your mouse.

Wall Grid

The drawing grid, which appears in the upper right corner of the Wall Panel Editor screen, gives the user options for drawing within the Wall Panel Editor window. The options include:

Snap Options allows you to provide snap points at the edges of the wall panel at quarter and third points.

Grid Increments allow you to set a drawing grid within the Wall Panel Editor separate from that in the main model view that you can snap to when drawing openings and regions. This field can work in two separate ways:

- **Single input increment:** If you put a single value here (0.5 ft for example), this increment will be used across the entire wall until the end of the wall is reached.
- **Multiple input increments:** If you have specific locations you wish to define, you can use this field to place exact grid points. Commas are used to delineate different grids and @ symbols can be used to apply the same increment multiple times (5, 10, 2@8, 3 for example). If you place multiple increments in and you are not yet to the end of the wall then the last increment defined will be used until the end of the wall is reached.

View Controls

In addition to the wall panel editing tools, the **Wall Panel Editor** window includes the view controls listed in the following table.

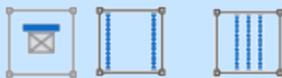
Wall View Controls

Icon	Description
 Delete	The Delete icon lets you delete openings, regions or boundary conditions from the wall panel.
 Rendered	The Rendered icon turns rendering of the current model view on or off, depending on the current setting.
 Nodes	The Nodes icon shows any nodes that fall in the plane of the wall panel, and allows you to snap to them when drawing regions or openings.

Icon	Description
 Wall Grid	The Wall Grid icon turns the display of the Drawing Grid on or off, depending on the current setting.
 Diaphragm	The Diaphragm icon turns the display of the Diaphragms on or off, depending on the current setting.
 Detach Diaphragms	The Detach Diaphragm icon lets you click on a diaphragm in the editor to detach it from the wall. The detached diaphragm then shows up in gray.
 Zoom Extents	The Zoom Extents icon redraws the wall panel to fit within the Wall Panel Editor window.
	The Loads button in the 3D View editing panel, turns the display of the wall panel loads on or off, depending on the current setting.
	The Print icon on the Quick Access toolbar, lets you print your current wall panel view.

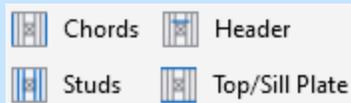
**Note:**

- There are also view controls specific to concrete and masonry. For more information see the [Concrete Wall - Design](#) and [Masonry Wall - Design](#) topic.



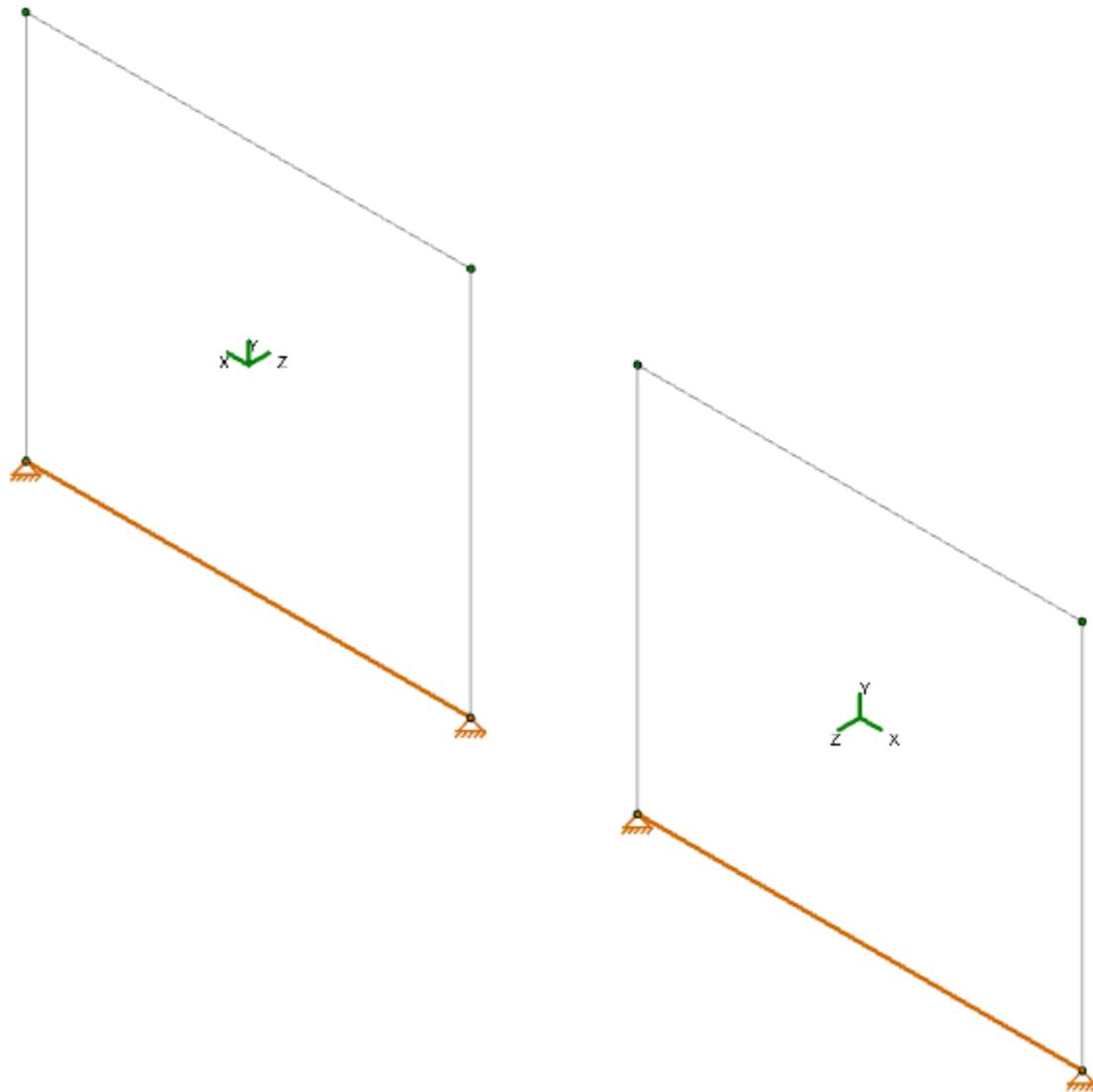
Lintel In-Plane Out-Plane

- There are also view controls specific to wood. For more information see the [Wood Wall - Design](#) topic.



Wall Panel Local Axes

When a wall panel is drawn it comes along with a set of local axes. If the wall is drawn top left to bottom right versus bottom left to top right it will change the direction of the local axes.



In most cases the local axes direction does not affect results. Here is a list where the local axis is important:

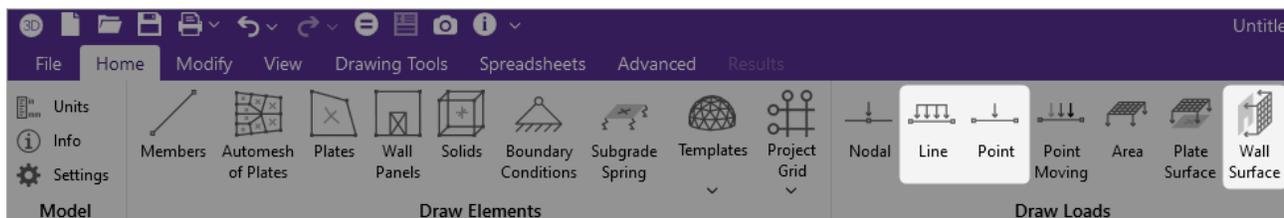
- **Applying Loads:** Wall panel surface loads can be applied to the local axes directions. Thus, the local axes direction will affect the sign of the applied load.
- **Viewing Plate Contours:** Wall plate contours can be viewed based on local axes, thus knowing the orientation is necessary to get meaningful information from these contours
- **Out-of-Plane Concrete Wall Design:** Concrete cover can be different for each face of a concrete wall. Which face you are defining is based on the +z and -z local axes.



Note: Check out the [Flip Axis](#) command to allow you to change these local axes.

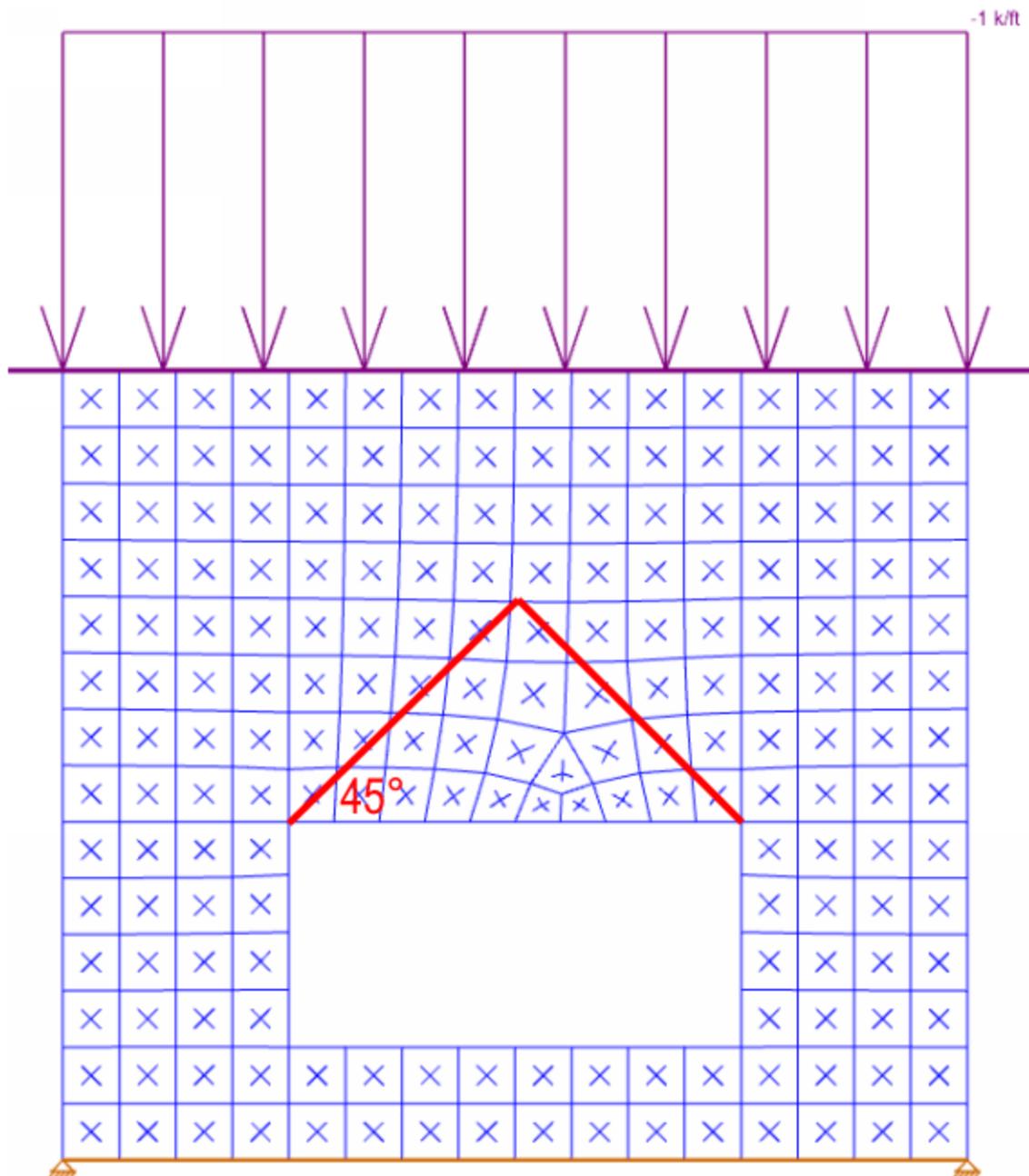
Loading

Wall panels can be loaded either directly or indirectly from supporting other elements (other walls, columns, beams, etc.). Line loads and distributed loads can be applied directly to wall panels, similar to how they are applied to members. These loads can be applied vertically or horizontally. Wall panel surface loads can also be applied directly to the wall.

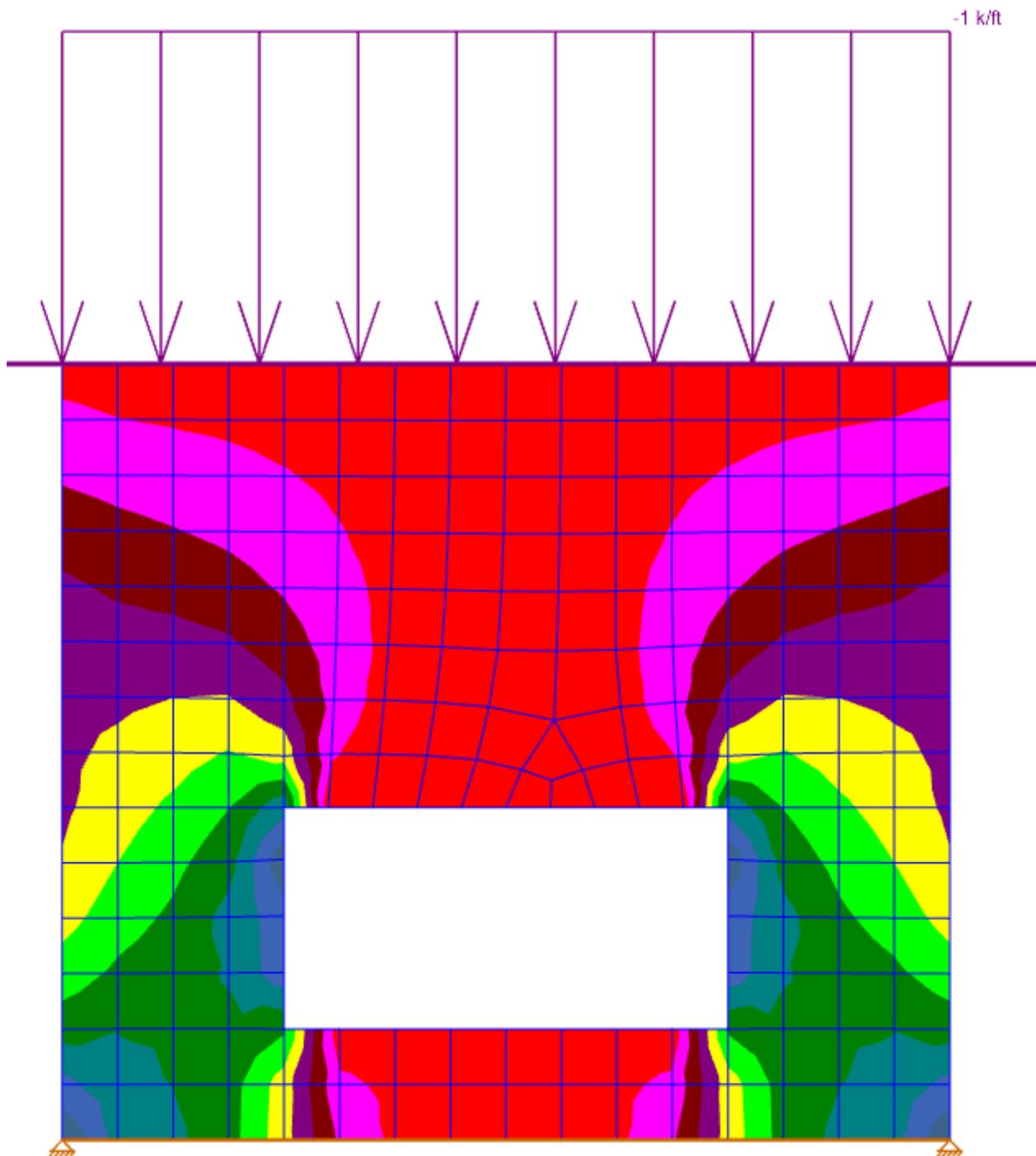


Load Attribution

In RISA-3D, the use of finite elements dictates how loads pass through wall panels. Load is attributed to the structure according to relative stiffnesses of elements. In concrete, wood and masonry design, many empirical equations are formed based on approximations or idealizations. Because of this, you may not get your loading in your wall panel elements (regions and lintels) to match hand calculations exactly. A prime example of this occurs with the 45 degree rule for lintels for masonry. According to theory, arching action occurs in lintels to the point that, if the top of your wall is a sufficient distance away, only the load in the triangular portion above your lintel would actually be taken into the lintel itself. Also, no load applied at the floor level would be felt by the lintel either. See the image below.



Within RISA-3D, this idealization will not hold true. The wall panel is a finite element mesh that attributes load according to the plate mesh FEM behavior. The load that is getting into the lintel is a true representation of how the wall is actually working. There is still arch action taking place as you can tell if you look at the vertical force contours in the wall panel.



In the image above, the red color is an area of very low axial force in the wall. Thus, you can see that, due to plate distribution of force, there is still arching action taking place. This arching action, however, will not be immune to additional loads added to the wall or the opening being located lower

in the wall (as is assumed with the idealized arching action in many texts). Thus, though your loads for lintel design may not be identical to what idealized methods might consider, this is a rational loading for the geometry and loading input on the wall panel.

Meshing the Wall Panels

At solution time, the wall panels will be automatically meshed into quadrilateral plate elements. Unlike the plate elements created directly by the user, the automatically generated plate elements are transient in the program and will not be saved in the input file.

The wall panel meshing is treated similar to analysis results. When the results of an analysis are deleted, the wall panel mesh is cleared to be re-built during the next solution. When a solution results file is saved, the meshed elements will be included in that file.

Mesh Size

The global mesh size for the wall panels can be input on the [Solution](#) tab of the Model Settings window. The smaller the mesh size, the more accurate the analysis will be. However, smaller mesh size also leads to longer solution time and more memory usage. The default mesh size is 12 inches in RISA-3D.

Localized small mesh sizes are used in the lintel locations for masonry walls, in order to achieve more accuracy for the lintel forces.

Graphical Display of the Wall Panel Mesh

By default, the plate elements associated with the wall panels are not visible to the user. The mesh can be turned on using the setting on the [Panels](#) tab of the Model Display Options window. The **Analysis Mesh** check box turns the display of the wall panel mesh on or off.

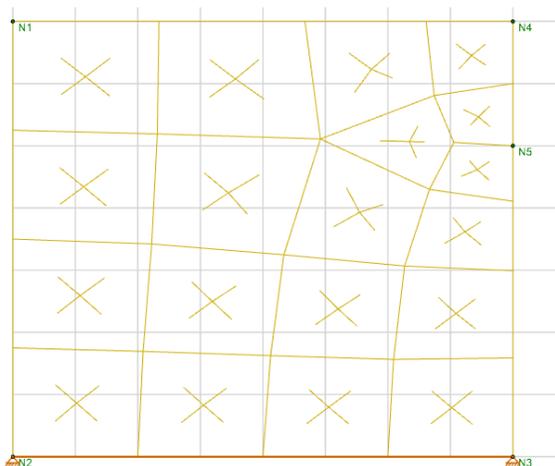


Note: The display of the mesh is only available when there are active analysis results.

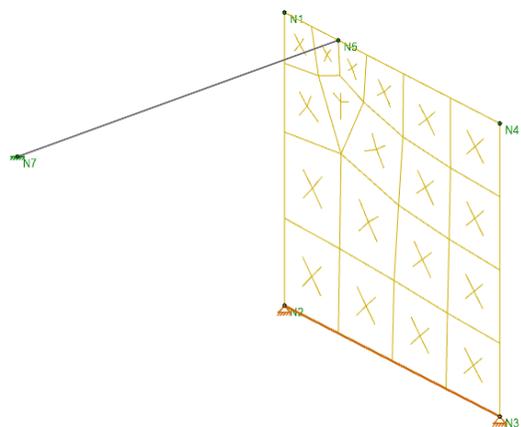
Point Constraints for the Panel Mesh

Point constraints are the locations within the wall panel that require connectivity to the meshed plate elements. The program automatically generates point constraints at the following locations:

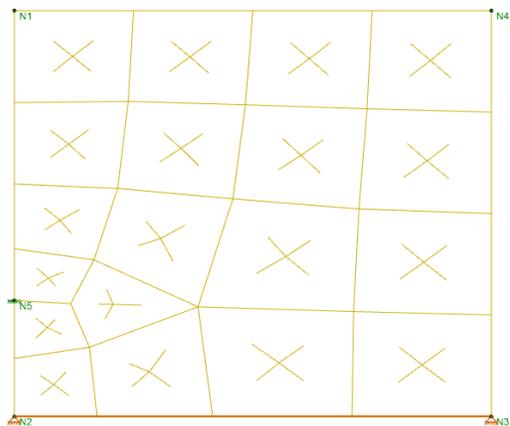
- Location of an existing node on the wall panel edges, region boundaries and opening boundaries



- Where beams intersects the wall panels (out-of-plane) on the wall panel edges, region boundaries and opening edges



- Location of an external boundary condition.



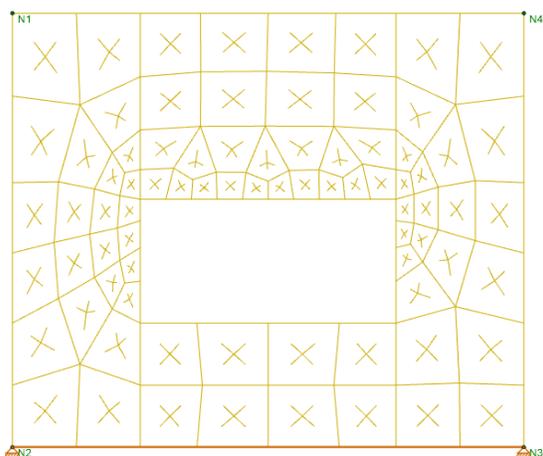


Note: Unattached nodes that are located on the wall panels can be considered as point constraints and prolong the meshing time. It is highly recommended that the user delete any unattached nodes before solving.

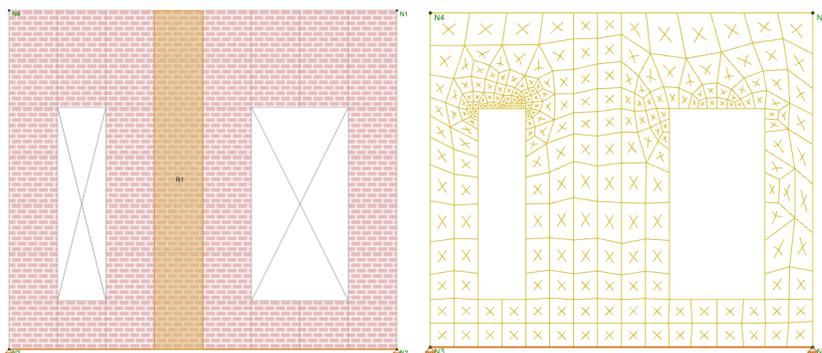
Line Constraints for the Panel Mesh

Line constraints are the locations within the wall panel that require continuous connectivity to plate edges rather than a single point. The program will automatically generate line constraints at the following locations:

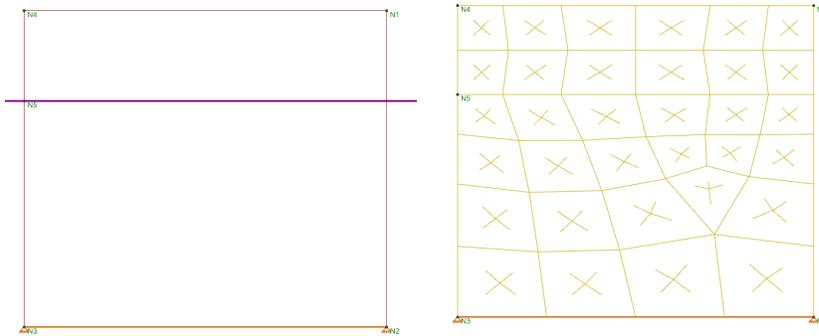
- Opening edges.



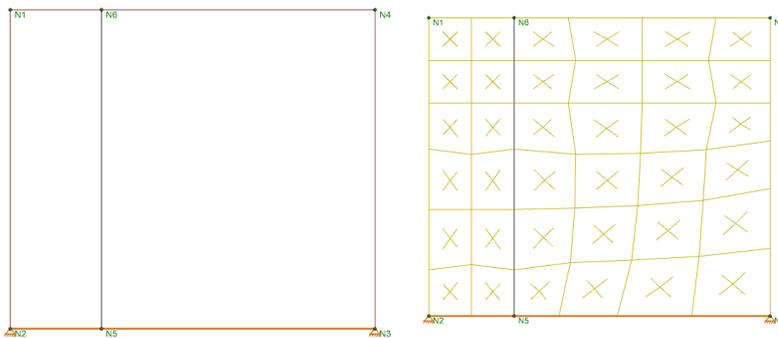
- The edge and vertical centerline of a defined region.



- Where a diaphragm intersects the wall panel.



- Where a beam or column element intersects within the plane of the wall panel.



Tips for Ensuring an Accurate Mesh

In order to generate an efficient mesh that gives accurate results, it is critical to place the line constraints and point constraints correctly. If line constraints or point constraints are very close to each other, the automeshing will be forced to generate small sized elements in order to satisfy the constraints. Therefore, a large number of plate elements will be generated and the solution will be slowed down significantly.

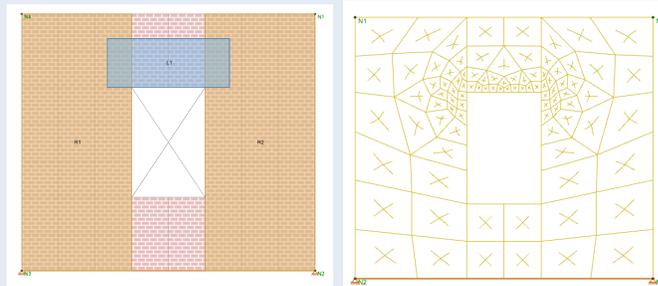
The following guidelines should be followed to ensure a quality mesh:

- Avoid generating very narrow regions and openings.
- Avoid small offsets between the external boundary conditions with the location of the region boundaries and wall boundaries.
- Avoid small offsets between opening edges with the region boundaries .
- When a wall panel is intersected by another wall panel, diaphragms, beams or plate elements, keep in mind that the intersection is a line/point constraint. Avoid the small offsets between intersections with the region boundaries or opening edges inside the wall panel.

Example #1: Region Boundaries

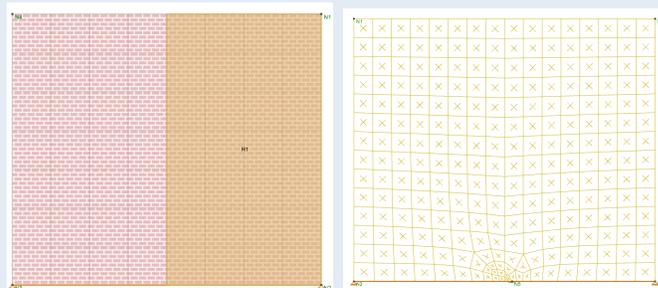
1. The left boundary region R1 is placed very close to the opening but not on the opening.
2. The left boundary of region R2 is placed very closed to the left edge of the wall but not exactly on the wall boundary.

In order to satisfy the line constraints required by the opening edge, region boundaries, and wall boundaries, the program is forced to generate very small meshes in the portion of the wall adjacent to these constraints.



Example #2: Poorly Located Boundary Conditions:

The left boundary of region R1 is at the vertical center line of a wall panel. At the same time, the user placed an external boundary condition at the bottom of the wall panel, which is slightly offset from the center line. In order to accommodate the line constraint of the region boundary and the point constraint of the external boundary condition, the automeshing is forced to generate a very small mesh adjacent to these constraints.



Merge Tolerance for Auto-Correction of Mesh

If the distance between the line constraints and point constraints are smaller than the merge tolerance specified on the Model Settings (which defaults to 0.12 inches) then the automeshing will automatically snap the constraints together during the meshing. This can eliminate some of the meshing issues that occur in the examples above.

Wall Panels - General Results

When the model is solved there are results spreadsheets specifically for Wall Panel Design and for Wall Forces. The Wall Panel Design spreadsheets are divided into different tabs: In plane, Out of Plane, Lintel, Wood Wall Axial, Wood Wall In-Plane, Wood Header and Wall Panel Seismic. Each tab provides code checks based on the relevant code, depending on the material type. These spreadsheets can be used as a summary of all of the panels in your model. To get detailed information about each panel, you can use the Wall Panel Detail Reports.

Concrete Wall Spreadsheet Results

- [In Plane Results Spreadsheet](#)
- [Out of Plane Results Spreadsheet](#)

Concrete Wall Detail Reports

- [Wall Summary Detail Report](#)
- [In Plane/Out of Plane Detail Report](#)

Masonry Wall Spreadsheet Results

- [In Plane Results Spreadsheet](#)
- [Out of Plane Results Spreadsheet](#)
- [Lintel Results Spreadsheet](#)

Masonry Detail Reports

- [In-Plane / Shear Wall Detail Report](#)
- [Out-of-Plane Detail Report](#)
- [Slender Wall Detail Report](#)
- [Lintel Detail Report](#)

Wood Wall Spreadsheet Results

- [Axial Results](#)
- [In Plane / Shear Results](#)
- [Header](#)

Wood Wall Detail Reports

- [In-Plane or Shear Wall Detail Report](#)

Wall Force Results

The Wall Force Results spreadsheet lists the axial force, shear and moments for each wall panel. The elevations where forces are reported are at the base of walls and at the locations of any diaphragms

that intersect the walls. The elevation is given in terms of global elevation.

	Wall Label	Elevation [ft]		Axial [k]	LC	x Shear [k]	LC	z Shear [k]	LC	x-x Moment [k-ft]	LC	z-z Moment [k-ft]	LC
1	WP3	0	max	70.581	2	34.104	11	-0.53	4	0.001	2	226.512	15
2		0	min	39.93	17	-32.676	15	-2.422	2	0	4	-465.755	11
3	WP9	0	max	56.961	21	4.988	21	1.907	19	0	18	-110.888	19
4		0	min	43.405	19	-0.245	19	-0.999	18	-0.001	19	-206.63	21
5	WP8A	0	max	31.697	18	-0.285	19	-0.082	19	-0.005	19	111.902	21
6		0	min	23.788	19	-7.273	21	-0.125	18	-0.011	18	72.071	19

	LC	Wall Label	Elevation [ft]	Axial [k]	x Shear [k]	z Shear [k]	x-x Moment [k-ft]	z-z Moment [k-ft]
1	1	WP3	0	51.539	-0.442	-0.62	0	-86.169
2	2	WP3	0	70.581	0.696	-2.422	0.001	-190.957
3	3	WP3	0	56.771	3.132	-1.138	0.001	-149.034
4	4	WP3	0	54.258	3.16	-0.53	0	-87.529
5	5	WP3	0	45.219	19.831	-0.593	0	-289.569
6	6	WP3	0	43.724	3.118	-0.568	0	-145.669
7	7	WP3	0	47.872	-9.396	-0.53	0	-8.63
8	8	WP3	0	42.209	15.642	-0.596	0	-268.171
9	9	WP3	0	46.491	-5.929	-0.544	0	-27.5
10	10	WP3	0	63.958	0.756	-1.088	0.001	-61.8
11	11	WP3	0	45.921	34.104	-1.214	0.001	-465.755
12	12	WP3	0	55.686	25.788	-1.163	0.001	-335.692
13	13	WP3	0	66.984	-24.297	-1.031	0.001	183.192
14	14	WP3	0	42.97	0.671	-1.164	0.001	-177.463
15	15	WP3	0	61.008	-32.676	-1.038	0.001	226.512
16	16	WP3	0	51.261	-24.36	-1.088	0.001	96.757

The following table provides descriptions for each of the columns in the results spreadsheet.

Column Header	Description
Wall Label	This column displays the wall panel label.
LC	This column reports the relevant Load Combination.
Elevation	This column report the elevation (in Global Coordinates) at which the forces are reported.
Axial	This column reports the axial force value in the wall. Compression force is reported as positive, Tension force as negative.

Column Header	Description
x-Shear	This column displays the shear force in the local x direction for the wall. This is the shear wall / strong direction of the wall.
z-Shear	This column displays the shear force in the local z direction for the wall. This is the cantilever / weak direction of the wall.
x-x Moment	This column displays the moment about the local x axis for the wall. This is the cantilever / weak direction of the wall.
z-z Moment	This column displays the moment about the local z axis for the wall. This is the shear wall / strong direction of the wall.

General Wall Detail Report

The general wall detail report shows the material type, height, length, and envelope forces for general walls.

Detail Report for - R1 of WP8A

Wall Panel Label WP8A Region R1 In-Plane

Detail Report: WP8A (In-Plane, Region R1)

GEOMETRY		MATERIALS	
Total Height (ft):	12	Wall Material:	gen_Conc3NW
Total Length (ft):	11		
Thickness (in):	10		

ENVELOPE DIAGRAMS

Min: 15.206 at 12 ft
Max: 67.475 at 0 ft

Min: -26.989 at 12 ft
Max: 26.097 at 0 ft

Min: -93.709 at 0 ft
Max: 217.833 at 0 ft

Print Add to Full Report Close

 **Note:** No detail report is generated for general walls with openings when there is no region defined.

The **Wall Panel Design** spreadsheet displays the calculated results for wall elements and may be accessed by selecting **Wall Panel Design** from the **Results** menu on the **Explorer Panel**. The spreadsheet has nine tabs: **Concrete In**, **Concrete Out**, **Masonry In**, **Masonry Out**, **Masonry Lintel**, **Wood Wall Axial**, **Wood Wall In-Plane**, **Wood Header** and **Wall Panel Seismic**.

The first two tabs as well as the last tab give results of **Concrete Wall** analysis. For more information on these tabs see [Concrete Wall Results](#) and [Concrete Wall - Seismic Results](#).

The next three tabs are results of **Masonry Wall** analysis. For more information on these tabs see [Masonry Wall Results](#).

The next three tabs are results of **Wood Wall** analysis. For more information on these tabs see [Wood Wall Results](#).

CFS Walls

Cold Formed Steel Walls - Design

The CFS wall panel element allows you to easily model, analyze and design CFS walls for in-plane loads. Here we will explain the specific inputs and design considerations. For general wall panel information, see the [Wall Panels](#) topic. For CFS wall results interpretation, see the [CFS Wall Results](#) topic.

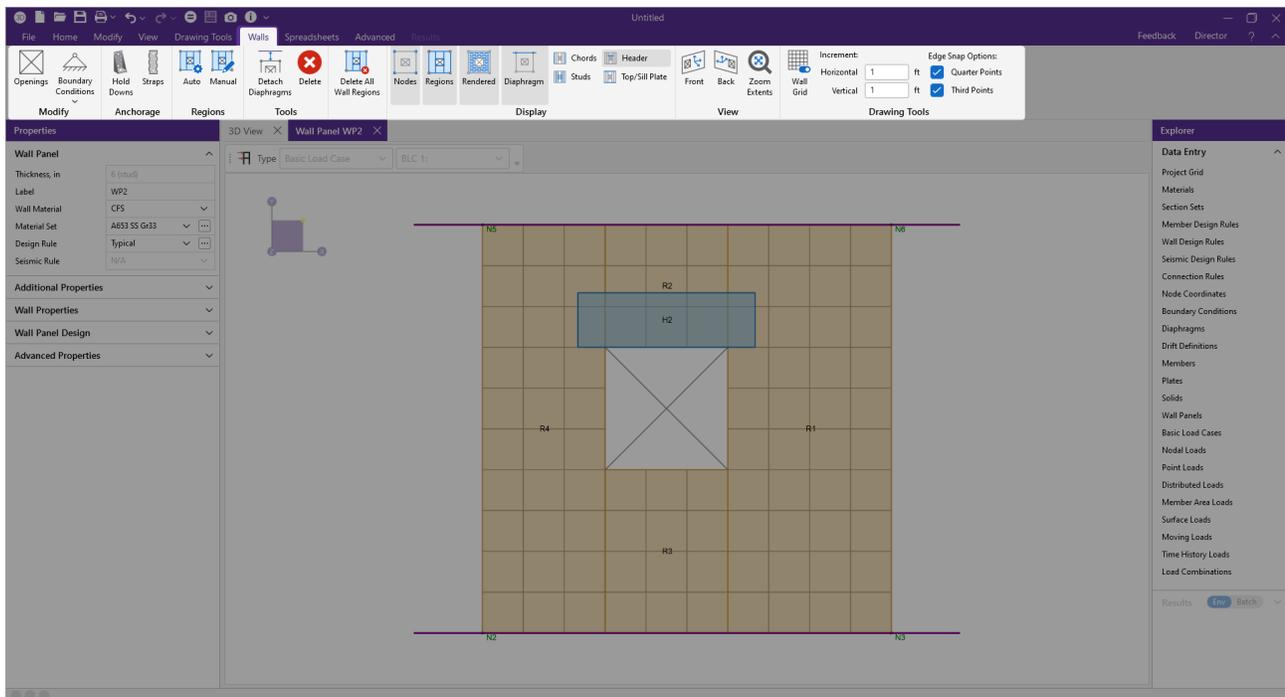
For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **CFS Walls**

Full code checking and design can be performed on the panel sheathing, studs, and chords based on the following codes:

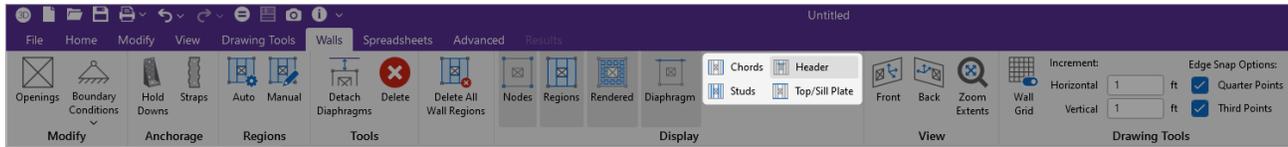
- The AISI S240-15 CFS Structural Framing
- AISI S400-15 w/S1-16 Seismic Design of CFS Structural Systems

CFS Wall Input

The **Walls** ribbon gives some specific information and options for modeling/analysis of CFS walls. It can be accessed by double clicking on a wall panel in the 3D view window.



CFS View Controls



Button/Icon	Name	Description
 Chords	Chords	Allows you to turn the display of wall panel region chords on and off.
 Studs	Studs	Allows you to turn the display of the studs on and off.
 Header	Header	Allows you to turn the display of the studs on and off.
 Top/Sill Plate	Top/Sill Plate	Allows you to turn the display of the top/sill plates on and off.

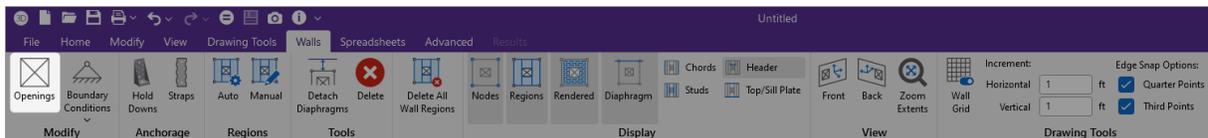
Design Rules

You must set up design rules for the stud/chord sizes, as well as make a database selection for shear panels. This is done in the **Wall Design Rules** spreadsheet in the **CFS Wall (Studs)** and **CFS Wall (Fasteners)** tabs. See the [CFS Wall - Design Rules](#) topic for more information.

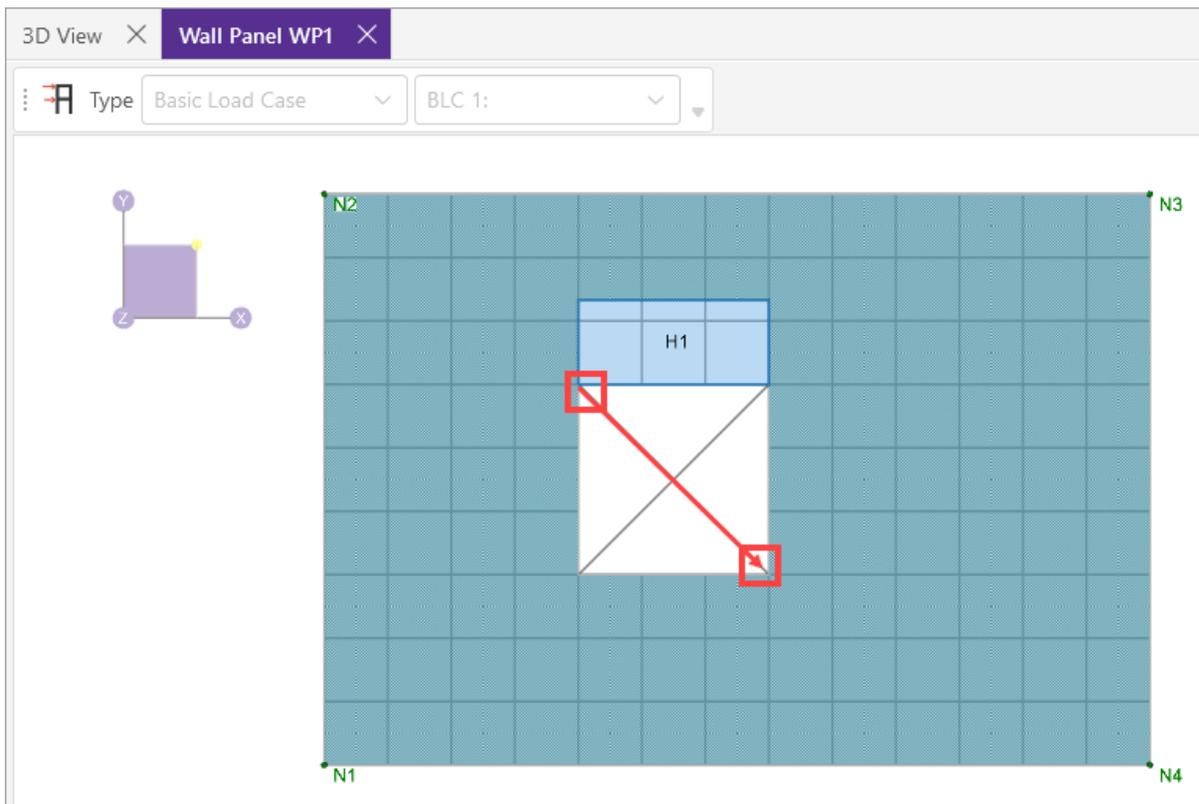
Create Openings in CFS Walls

In the **Walls** ribbon, you have the option of adding rectangular openings to CFS wall panels. To draw an opening:

1. Click the **Openings** icon.



2. Click on two nodes or grid intersections which make up the two diagonal corners of your opening.



Note: Currently header design is not available for CFS walls.

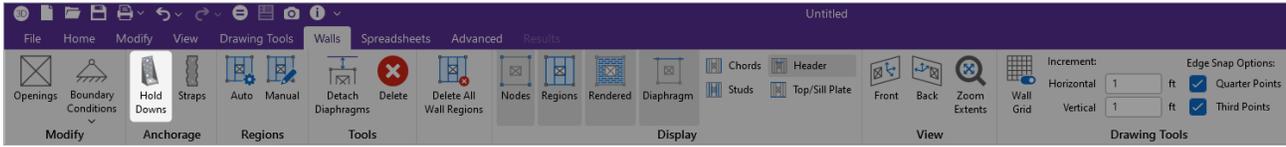
When an opening is drawn, a header beam is automatically created above the opening (as indicated by the H1 block in the image).

Hold-Downs and Straps

Hold-downs and straps are automatically added to your walls in their required locations. For a Type I - Segmented design, you must have hold-downs or straps at the bottom corners of each of your design segments. The program will not permit the drawing of hold-downs or straps at locations where they are not allowed.

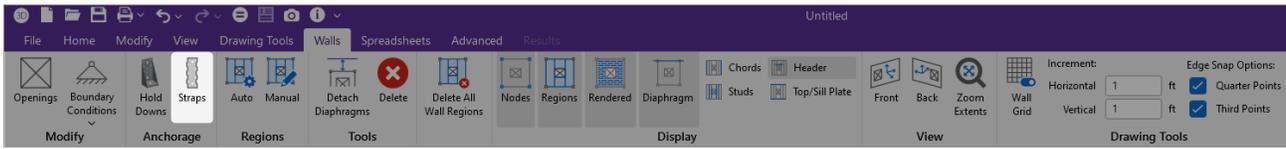
If there are custom locations that you want to add hold-downs or straps, you can do this from the **Walls** ribbon .

Hold downs can represent the anchorage of your wall to the foundation or the connection of shear wall chords between floors. To add hold-downs to the base of your wall, first select the **Hold Downs** icon in the 'Walls' ribbon.



Hold downs must be added after regions are created and can only be added at the corners of regions. Hold down requirements depend on the type of wall design you are performing.

Straps represent the connection of the current wall panel to a wall panel below. To add straps to the base of your wall, first select the **Straps** icon in the 'Walls' ribbon.



Straps also can only be added after regions are created and can only be added at the corners of regions. Strap requirements also follow the same logic as hold-downs as to where they must be defined as far as regions are concerned.



Note:

- Hold-downs and straps are not designed for CFS Walls. However, the required forces will still be reported. More information on this can be found in the CFS Wall Results topic.
- All boundary conditions for wall panels should be defined in the wall panel editor. Adding external boundary conditions can create problems.
- The locations of hold-downs and straps define where the program will calculate tension forces in your walls.
- By default, the program will automatically add hold-downs to the base of walls that have boundary conditions applied to them and will apply straps to walls that have walls below.
- All Straps and Hold-downs are placed at the base of walls only. So, if you would like to put a strap or hold-down between floors, you would apply it to the base of the upper wall.

General Requirements for Shear Walls

The design of CFS shear walls per the available design codes requires that many criteria are satisfied before a wall can be considered adequate. For RISA to work within this framework, we require that certain modeling practices be followed. Outlined below are many general wall modeling practices and limitations as well as specific requirements for the Type I - Segmented method that is available for CFS Walls in the program.



Note: RISA-3D does not currently support Type I with FTAO or Type II walls.

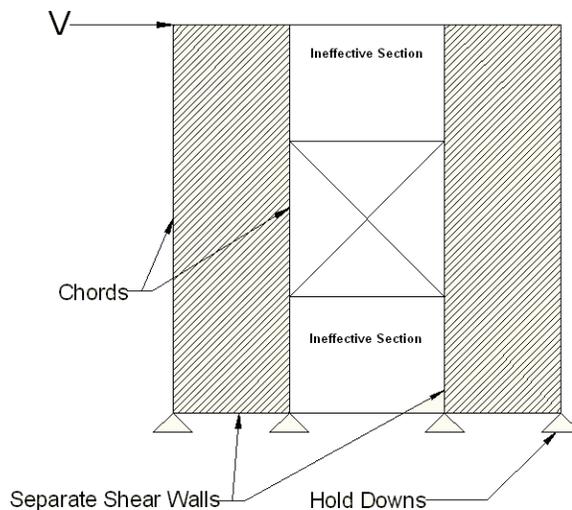
Type I - Segmented Method

The Type I - Segmented methodology is defined in Section B5.2 of the *AISI S240-15 North American Standard for Cold-Formed Steel Structural Framing*.

Where there is a wall panel with openings, the area above and below the openings is disregarded and the wall is designed as being made up of separate, smaller shear walls.

Like all wall panels, the segmented CFS wall is broken into a series of meshed plate elements to represent the overall wall. The portions of the segmented shear wall that are considered "ineffective" in resisting shear are modeled with a plate elements that have a significantly reduced shear stiffness so that they will not receive any significant moment or shear from the FEM analysis.

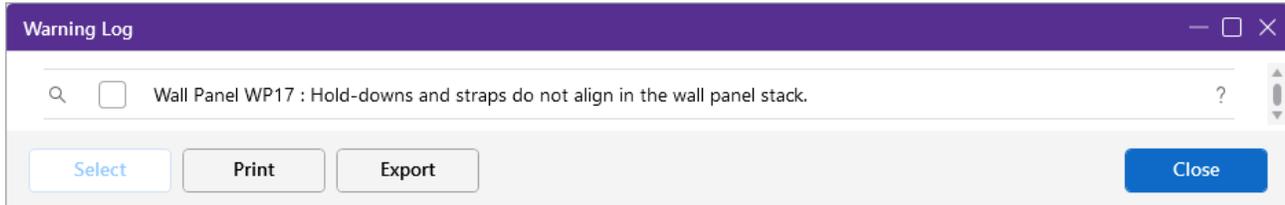
See the diagram below for more information:



In addition, the out of plane stiffness and in plane stiffnesses of the segmented CFS wall are modeled separately based on different assumed plate thicknesses. This is done to insure that the shear stiffness is based entirely on the properties of the sheathing and is not influenced by the out-of-plane stiffness of the wall studs.



Note: If you have several stacked Segmented wall panels with misaligned openings, you will receive the Warning Message shown below upon solution. This message means that RISA-3D has assumed that the strap force from the above wall panel will be spread out across the region directly under it. Therefore, you need to be aware of this assumption and detail the wall panel accordingly.



- A shear panel design will be chosen for the worst-case region in a segmented wall. That panel will then be used for all regions in that wall. The worst-case region is the one that has the highest Shear UC value.

Geometric Requirements and Limitations

The program will check the geometric requirements per AISI S240-15 B5.2.1.1 and AISI S400-E1.4 as required unless otherwise noted below. The sections noted below are assumed to be satisfied and are not checked.



Note:

- AISI S400-E1.4 parts (d) - (g) and (k) - (r)
- AISI S240 B5.2.2.3.2.1.1 parts (c) and (e)
- AISI S400 E2.3.1.1.1.1 parts (c) and (e)

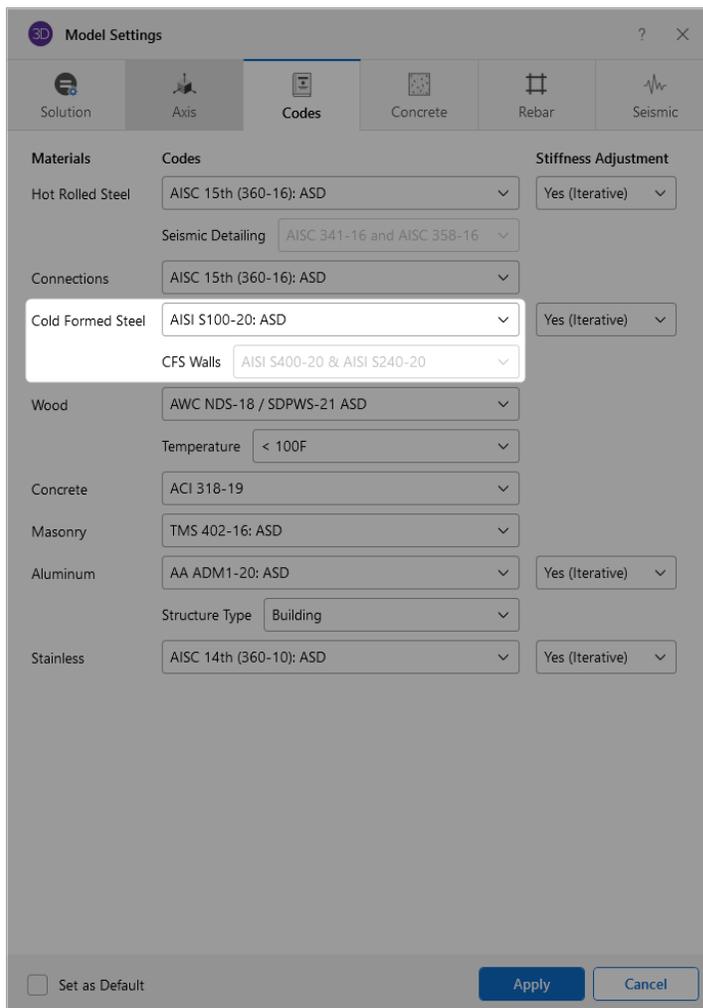
General Program Functionality and Limitations

Assigning Code and Design Methodology

CFS wall designs will be based on the *AISI S240-15 North American Standard for Cold-Formed Steel Structural Framing* and *S400-15 w/S1-16 North American Standard for Seismic Design of Cold-Formed Steel Structural Systems*. You can select these codes in the CFS Walls dropdown within the Codes section of the Model Settings. If you set this code to “None”, the program will not perform design for the CFS walls.

The design properties of CFS stud and chords within the wall are referenced from the AISI S100 specification referenced in the Cold Formed Steel dropdown.

In this manner, the design methodology (ASD or LRFD) for the entire wall panel (stud, chord and sheathing design) is determined by the design method selected in the Cold Formed Steel dropdown. For example, if the AISI S100-16: ASD code is selected as shown below, the CFS walls will use ASD factors in the stud, chord and sheathing designs.

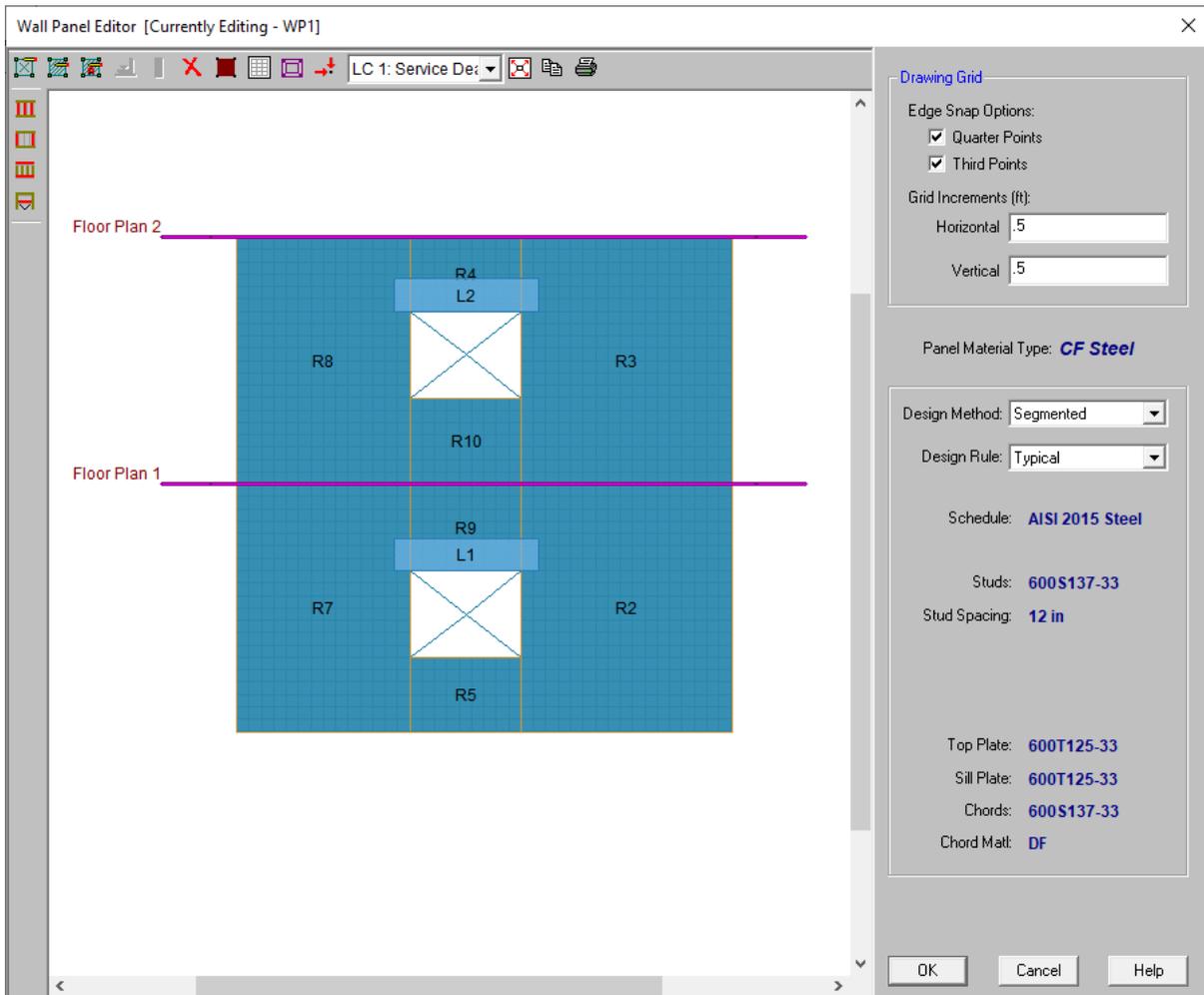


RISAFloor and RISA-3D Interaction

When using RISAFloor and RISA-3D in combination, the interface transitions nicely between the two programs. Here is a quick walk through of this interaction.

Input Interface

1. Model the entire building (gravity and lateral members) within RISAFloor.
Be sure to model all openings and regions for all of the wall panels in the model.



2. Add loading and solve the model.
3. Take the model into RISA-3D via the Director tool.
4. Once in RISA-3D you must add your hold downs and straps to your wall panels.

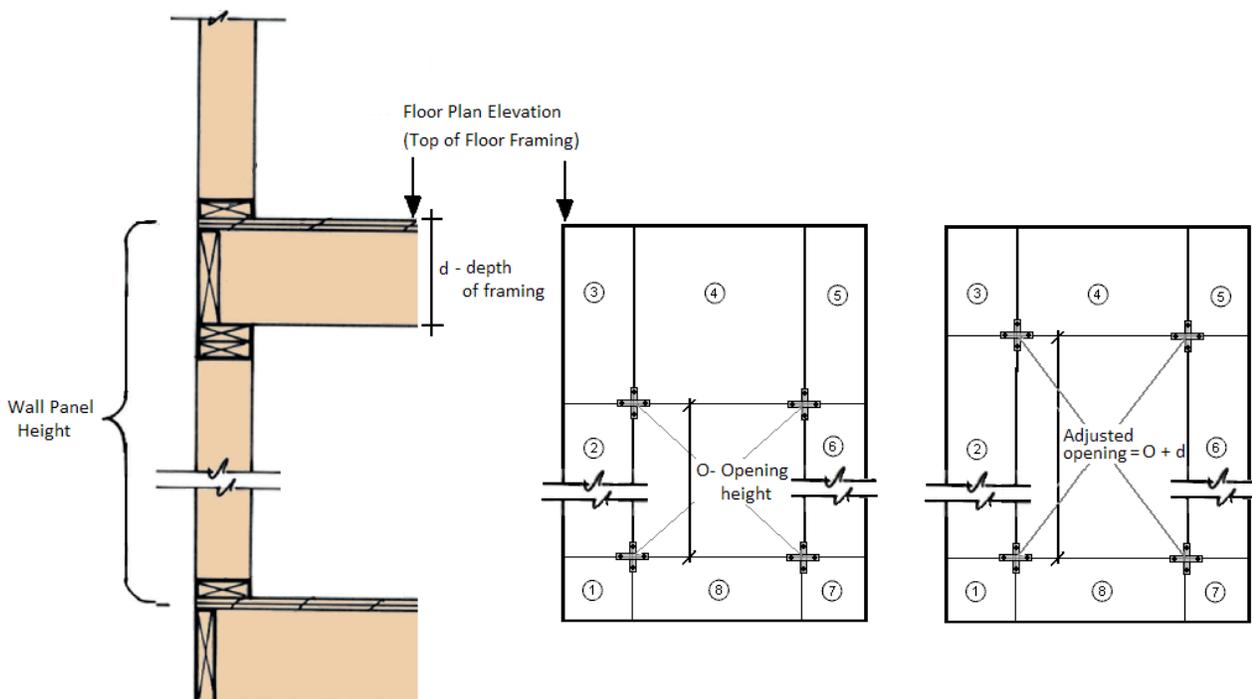
**Note:**

- You can not modify your openings or regions in RISA-3D. All region and opening modifications must be taken back to RISAFloor to be done.
- Hold downs and straps can not be added to wall panels in RISAFloor.
- Hold downs are required at the corners of all full height regions in the wall panel for the Segmented design method.
- The Design Rule and Design Method can be changed in either program at any time.

Modeling Tips- Platform Framing

The CFS wall height in RISA is measured from the bottom of the sill plate to the top of the floor framing as shown below. However, platform framing causes the wall height to be significantly shorter than a RISA model would represent.

This is especially problematic for the FTAO design method where the region above openings is assumed to be much larger than it is built. However, currently only the Type I - Segmented method is available and the portions above and below the opening are not used to transfer shear forces.



Limitations for Hold-downs / Straps (Including deformation)

- The program does not currently have a database for or optimize hold-downs/straps for CFS walls.
- The manufacturer listed or user defined hold-down deformation can be input in the Wall Design Rules spreadsheet for inclusion in the design deflection calculation. See the CFS Wall - Results topic for more information.

Automatic Boundary Conditions

- In RISA-3D, if no boundary conditions or hold downs are defined for CFS wall panels at the lowest level of the structure, the program will automatically create hold downs at the corners of the wall panel. If you do not want the hold downs to be automatically created, define a "free" boundary condition at the base of the wall panel in the wall panel editor.

Deflection

The program will report the design deflection for CFS shear walls that have wood or steel sheathing.

- There is currently no code check for drift or deflection for CFS shear wall panels.

Stud Design

Studs are only designed for load combinations which do not contain a wind or seismic load. The maximum axial load, determined as an envelope force from all of the "gravity" load combinations which have been solved, is determined for each region.

Studs are checked per AISI S240-15 B3.2: Wall Stud Design. The program currently only uses the All Steel Design methodology in which case bracing from the attached sheathing or structural bracing is ignored. Therefore the unbraced length for the buckling of the stud is taken as the full wall height. The top and bottom track thicknesses are considered negligible and are not deducted from the total height.



Note:

- The program currently does not optimize the stud size or spacing.
- The program will use an effective length factor, K , equal to 1.0 for the design of the studs.

Stud force demand and number of studs calculation:

The program first divides the maximum axial force in the region by the length of the region. Then it will multiply this distributed force by the stud spacing to get a maximum stud demand.

The program calculates the number of studs based on the following procedure:

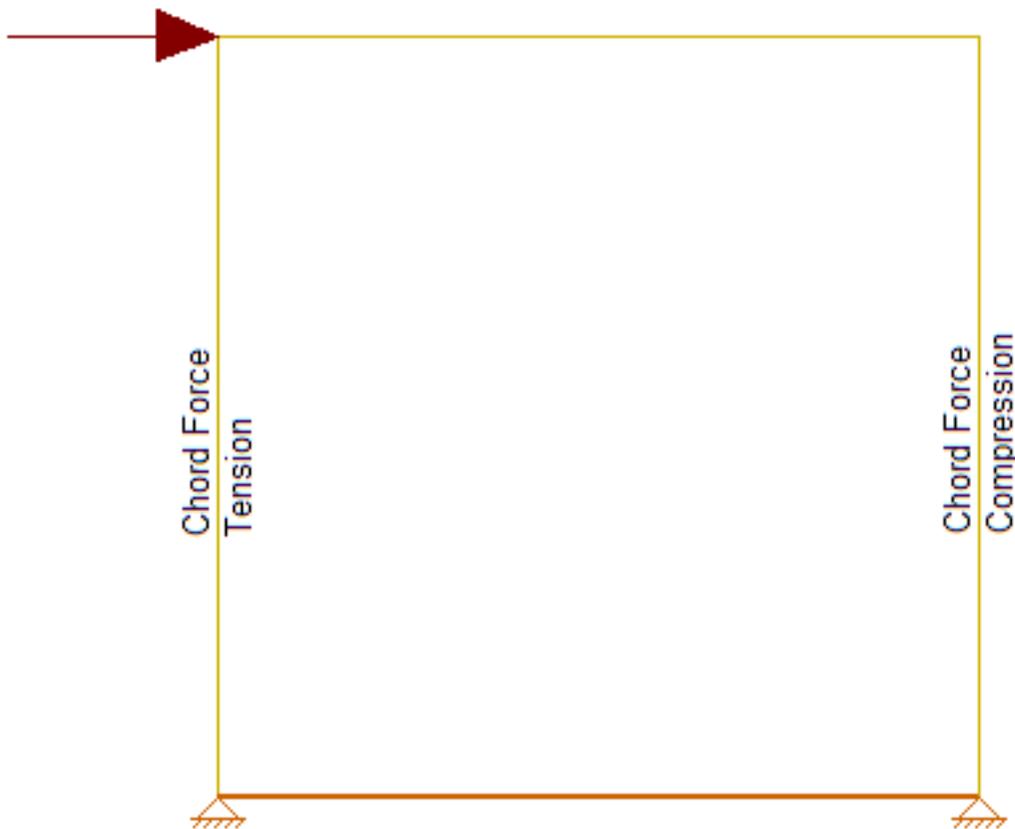
1. Determines the number of spaces between studs by dividing the region length by the stud spacing.
2. Rounds this number up to the nearest whole number.
3. Determines the number of studs by adding one to this number.

Openings

- Currently, the program does not perform header design. This will be available in an updated version.

Chord Design

The chord design is based on forces that are calculated differently for Compression versus Tension. The tension chord force is calculated including the dead load stabilizing moment. The compression chord force includes the only the tributary area of one stud spacing in the compression force. For Segmented design, the chord forces are found based on each region.



Chord forces are calculated per the following equations:

$$\text{Compression Force} = \frac{M}{L} + \frac{P}{n}$$

$$\text{Tension Force} = \frac{-M}{L} + \frac{P}{2}$$

Where:

- M = Moment at the base of the wall
- L = Length of the wall
- P = Axial force at the base of the wall
- n = Number of Studs = Length/Stud Spacing



Note:

- In RISA-3D, the edge of the CFS walls represent the centerline of the chord member. Therefore, the depth of the chord member will not affect the chord force calculation since the centerline of the chord member will always be assumed at the edge of the wall.
- The eccentricity checkbox in the Wall Design Rules spreadsheet will not affect the chord force calculation.
- RISA-3D models the shear walls using only the sheathing. The vertical resistance occurs only at the tension and compression chords. Thus, if two wall panels are stacked on top of each other, the load transfer will only happen at the chord locations. Therefore, the lateral analysis should agree very well with hand calculations. However, it also means that gravity load design may be more appropriate in RISAFloor.

Hold-Down Force

Hold-Down forces are calculated per the following equations:

Note that the hold-down force calculation depends upon whether the eccentricity checkbox has been checked, and the value entered in the eccentricity field.

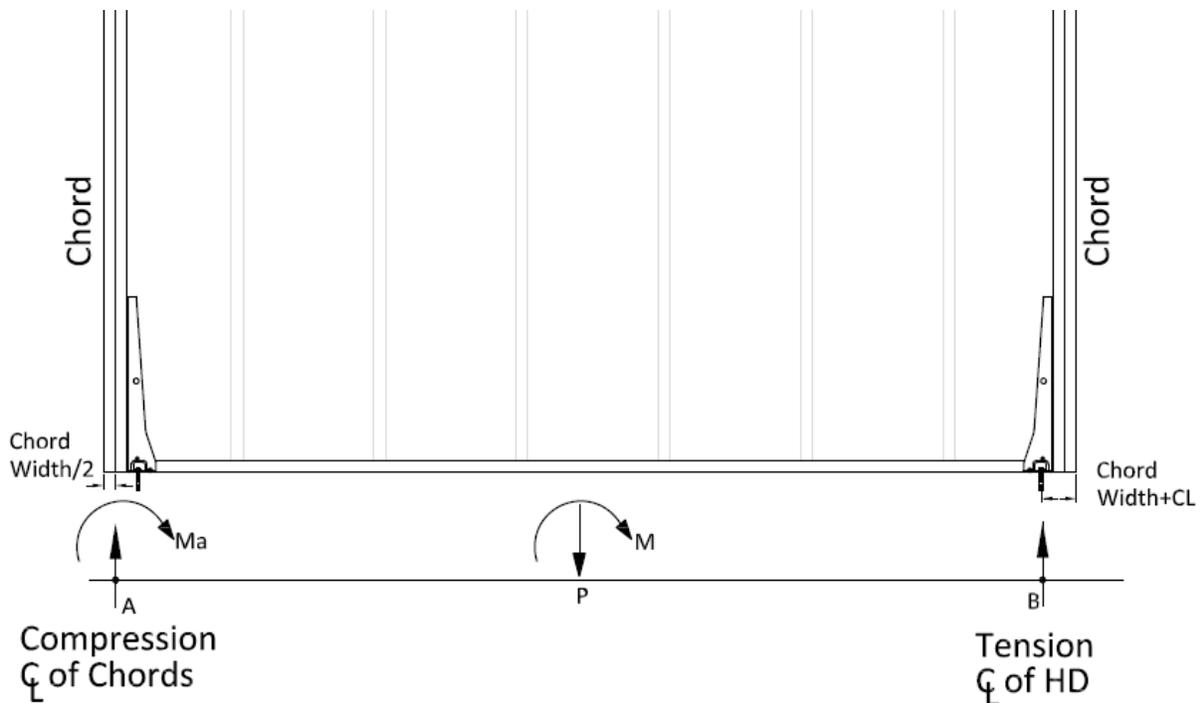
- When the "[Eccentricity](#)" check-box is unchecked in the CFS Wall Design Rule (Fasteners tab), the hold-down force is exactly equal to the chord force indicated above.

Additional CFS Wall Panel Parameters													
Unity Check	Concrete Wall (Rebar)	Concrete Wall (Misc)	Masonry Wall	Masonry In	Masonry Out	Masonry Lintel	Wood Wall (Studs)	Wood Wall (Fasteners)	CFS Wall (Studs)	CFS Wall (Fasteners)			
Label	Type of Wall	Sheathing Material	Schedule	Design Method	Min Panel Thick [in]	Max Panel Thick [in]	Fastener Min Edge Space	Fastener Max Edge Space	HD Chords	Eccentricity	Ecc Dist [in]	HD Deflection [in]	
1	12" [---]	Type I - Segmented	Steel Sheet sheathing	AISI 2020 Steel	Table	0.003	2	2-in.	6-in.	600S137-33	<input type="checkbox"/>	0	1
2	6"	Type I - Segmented	Steel Sheet sheathing	AISI 2020 Steel	Table	0.003	2	2-in.	6-in.	600S137-33	<input checked="" type="checkbox"/>	1.5	1

- When the "Eccentricity" check-box is checked in the CFS Wall Design Rule (Fasteners tab), an eccentricity distance should be indicated in the "Ecc Dist" column. This distance should be the distance between the hold-down and the centerline of the chord member.

Additional CFS Wall Panel Parameters													
Unity Check	Concrete Wall (Rebar)	Concrete Wall (Misc)	Masonry Wall	Masonry In	Masonry Out	Masonry Lintel	Wood Wall (Studs)	Wood Wall (Fasteners)	CFS Wall (Studs)	CFS Wall (Fasteners)			
Label	Type of Wall	Sheathing Material	Schedule	Design Method	Min Panel Thick [in]	Max Panel Thick [in]	Fastener Min Edge Space	Fastener Max Edge Space	HD Chords	Eccentricity	Ecc Dist [in]	HD Deflection [in]	
1	12" [---]	Type I - Segmented	Steel Sheet sheathing	AISI 2020 Steel	Table	0.003	2	2-in.	6-in.	600S137-33	<input type="checkbox"/>	0	1
2	6"	Type I - Segmented	Steel Sheet sheathing	AISI 2020 Steel	Table	0.003	2	2-in.	6-in.	600S137-33	<input checked="" type="checkbox"/>	1.5	1

The Hold-Down force is calculated by finding the Tension chord force. In order to accommodate an unsymmetrical vertical load on the wall, the program adopted a more accurate approach by calculating the moment at the compression side first and then finding the resulting reaction on the tension side.



The resulting equation to determine the hold-down force is as follows:

$$\text{Moment at Point } a = M_a = -M + \left(P * \left(\frac{L}{2} - \frac{b_c}{2} \right) \right)$$

$$\text{Reaction at Point } b = R_b = \frac{M_a}{L - 1.5 * b_c - CL}$$

Where:

- M_a = moment at point A(CL of compression chord)
- M = moment at base of the wall
- P = Axial load at the base of the wall
- L = length of the wall
- b_c = width of chord member
- R_b = reaction at point B = hold down force
- CL = Hold down eccentricity distance (per selected hold down)

Lateral Force Distribution

- The lateral force distribution between piers is based on the relative stiffness of the sheathing, not on the length of the shear wall. For example, if you have an 8 foot wall and a 4 foot wall, the 8 foot wall will take more than $8/(8+4)*100\%$ of the force. The moment of inertia in the 8 foot wall will allow for a larger proportion of load to go into that pier.

Shear Panel Material Capacities

Shear capacity of the wall will be dependent on the Sheathing Material and Design Method that was chosen in the Design Rules spreadsheet - CFS Wall (Fasteners) tab. When design values are pulled from design tables, the program will use the appropriate capacity from the table based on the height-to-width ratio of the wall, member thickness of the stud members and type of loading used.

Tabulated shear capacities are based on the *AISI S240-15 CFS Structural Framing* and *AISI S400-15 w/S1-16 Seismic Design of CFS Structural Systems* codes. If the governing lateral load is a seismic load (under the load category EL, ELX, ELY, ELZ, etc) then the capacity will be pulled from the AISI S400-15 code.



Note: Curtain wall considerations per AISI S240-15 are not considered in the program.

Wood Sheathing

Tabulated design values are referenced from AISI S240-15 Table B5.2.2.3-2 and AISI S400-15 Table E1.3-1.

Steel Sheathing

Tabulated design values are referenced from AISI S240-15 Table B5.2.2.3-1 and AISI S400-15 Table E2.3-1.

If the Effective Width Method is used as the design method, this will be referenced from AISI S400-15 Chapter E2.3.



Note: The program will not perform the end distance check in the effective width method. This is assumed to be satisfied.

Fiberboard

Tabulated design values are referenced from AISI S240-15 Table B5.2.2.3-4. AISI S400-15 does not apply for this material.

Gypsum

Tabulated design values are referenced from AISI S240-15 Table B5.2.2.3-3. AISI S400-15 does not apply for this material.

Stiffness Assumptions

Vertical Direction

RISA uses an orthotropic plate element to de-couple the vertical and shear stiffness of the CFS walls. The vertical stiffness will be based on the E value of the studs and chords as specified in the Materials spreadsheet and the thickness of the wall. The thickness is taken as

$$\textit{Thickness} = \frac{\textit{Total Wall Weight}}{\textit{Material Density} * \textit{Wall Length} * \textit{Wall Height}}$$

Where,

$$\textit{Total Wall Weight} = W_{\textit{studs}} + W_{\textit{chords}} + W_{\textit{top plate}} + W_{\textit{sill plate}} + W_{\textit{sheathing panel}}$$

In-Plane Shear

The in-plane shear stiffness will be based on the G_a value designated within the specified nailing schedule divided by the sheathing thickness. Both the G_a value and the sheathing thickness are defined in the specified sheathing schedule.

Out-of-Plane Shear

For the out of plane shear, RISA uses the same Young's Modulus (E) as used in the vertical direction and a thickness that is calculated from the out of plane moment of inertia. This becomes:

$$Thickness = \left(\frac{I_{chord} + I_{stud} + I_{panel}}{Wall\ Length} \right)^{1/3}$$

CFS Wall Self Weight

The program will calculate CFS wall self weight as a sum of all the weights of the components. The material density is used to calculate the self weight of the studs, chords, top plates, sill plate, and sheathing. These are all then summed together to give the self weight of the entire wall.

**Note:**

- For this calculation, stud height equals full wall height.
- The number of studs is calculated using the stud spacing specified in Design Rules.

CFS Wall Optimization

The program will optimize CFS walls based on the required demand forces. The program can optimize sheathing for in-plane shear design.

This optimization can substantially modify the stiffness of the wall. To properly adjust the stiffness requires an iterative solution that updates the stiffness of the model. This includes updating the strength properties of the wall as well as the stiffness. The program will handle this optimization/iteration automatically.

To update the stiffness portion of the wall, the program must re-solve your model with these updated stiffnesses as this will change the distribution of forces through the model. Thus, the program will start with its initial stiffness parameters and solve the model. It will then optimize the wall to meet strength criteria. Another solution will then be run with the new stiffnesses and the program will again optimize the wall to meet strength criteria. This procedure will continue to occur until all wall panel results match those of the previous solution.

The program will always present results in the output that coincide with the stiffness used in the final solution.

Panel Optimization

Cold Formed Wall Panel Schedule

Select Code: CFS Steel Sheeting

Available Panel Groups:

- AISI 2020 Steel
- AISI 2015 Steel

Current Selection Type: Use All Panels

	Label	Type Of Wall	Sheathing Material	Up 4 to 1?	Sheet Thickness	Fastener Spacing	Stud Block
1	0.018_#8@6/12_33	Type I - Segmented	Steel Sheet sheathing	No	0.018	6	No
2	0.027_#8@4/12_43	Type I - Segmented	Steel Sheet sheathing	No	0.027	4	No
3	0.027_#8@3/12_43	Type I - Segmented	Steel Sheet sheathing	No	0.027	3	No
4	0.027_#8@2/12_43	Type I - Segmented	Steel Sheet sheathing	No	0.027	2	No
5	0.027_#8@4/12_43_TALL	Type I - Segmented	Steel Sheet sheathing	Yes	0.027	4	No
6	0.027_#8@3/12_43_TALL	Type I - Segmented	Steel Sheet sheathing	Yes	0.027	3	No
7	0.027_#8@2/12_43_TALL	Type I - Segmented	Steel Sheet sheathing	Yes	0.027	2	No
8	0.027_#8@6/12_33	Type I - Segmented	Steel Sheet sheathing	No	0.027	6	No
9	0.027_#8@4/12_33	Type I - Segmented	Steel Sheet sheathing	No	0.027	4	No
10	0.027_#8@3/12_33	Type I - Segmented	Steel Sheet sheathing	No	0.027	3	No
11	0.027_#8@2/12_33	Type I - Segmented	Steel Sheet sheathing	No	0.027	2	No
12	0.027_#8@6/12_33_TALL	Type I - Segmented	Steel Sheet sheathing	Yes	0.027	6	No
13	0.027_#8@4/12_33_TALL	Type I - Segmented	Steel Sheet sheathing	Yes	0.027	4	No
14	0.027_#8@3/12_33_TALL	Type I - Segmented	Steel Sheet sheathing	Yes	0.027	3	No
15	0.027_#8@2/12_33_TALL	Type I - Segmented	Steel Sheet sheathing	Yes	0.027	2	No

OK Cancel

The procedure that RISA uses for design optimization is fundamentally based upon the assumption that there is a 'cost' to shear capacity, and therefore the ideal panel design would have as little shear capacity as possible to meet code requirements. Once the program has determined the shear demand on the wall it will choose the most economical panel configuration based on that which has a **Shear Capacity** closest to, but not exceeding the shear demand.

The program will also only pick shear panels that qualify based on the region height-to-width ratio and minimum stud thickness requirements.

The default shear panels in the program come straight from published tables in the AISI S240-15 and AISI S400-15.



Note:

- The shear forces listed in the XML spreadsheet are nominal shear strength values. The force capacity will be adjusted by the appropriate factor depending on if the ASD or LRFD design method is selected in the Cold Form Steel code dropdown.
- Panels with a label containing the characters "_W" together will be ignored during design optimization.

CFS Walls

Cold Formed Steel Walls - Design

The CFS wall panel element allows you to easily model, analyze and design CFS walls for in-plane loads. Here we will explain the specific inputs and design considerations. For general wall panel information, see the [Wall Panels](#) topic. For CFS wall results interpretation, see the [CFS Wall Results](#) topic.

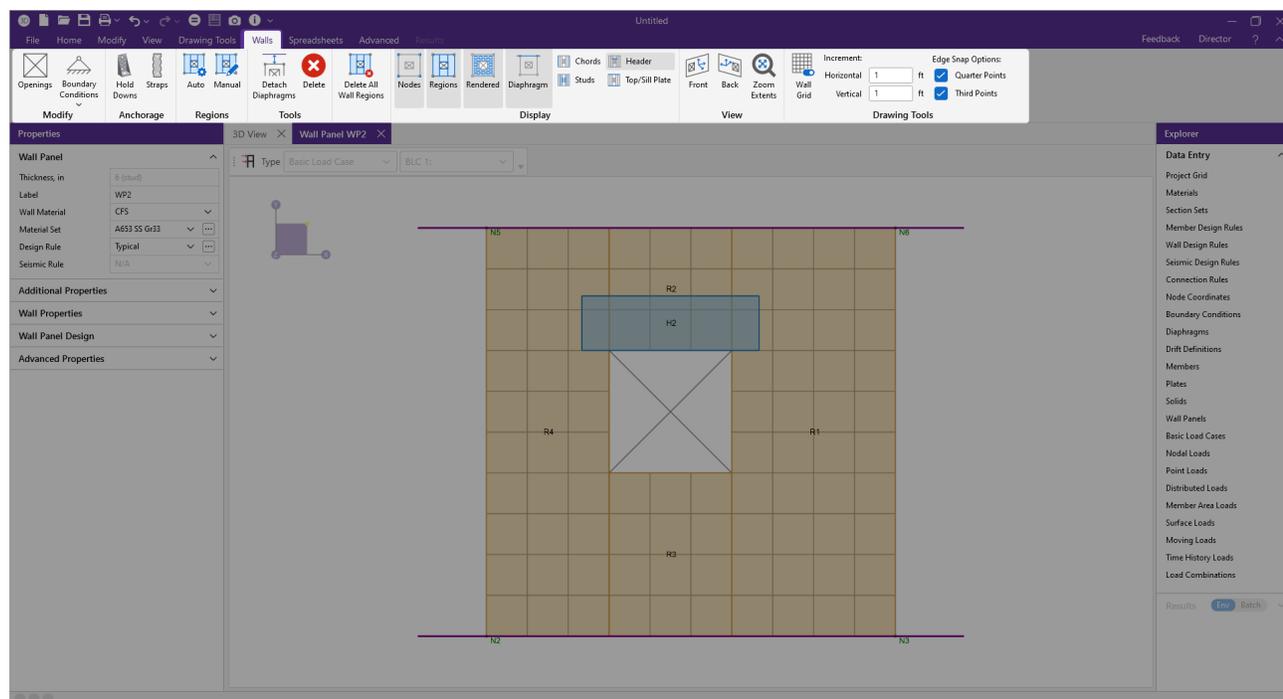
For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **CFS Walls**

Full code checking and design can be performed on the panel sheathing, studs, and chords based on the following codes:

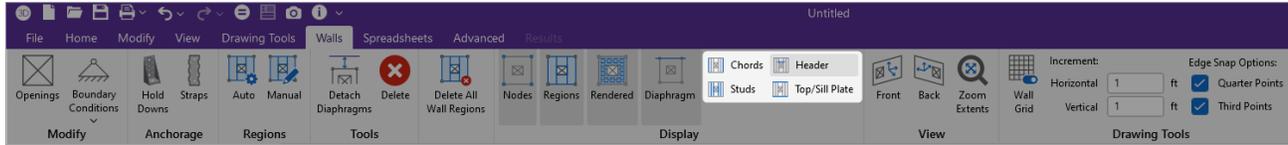
- The AISI S240-15 CFS Structural Framing
- AISI S400-15 w/S1-16 Seismic Design of CFS Structural Systems

CFS Wall Input

The **Walls** ribbon gives some specific information and options for modeling/analysis of CFS walls. It can be accessed by double clicking on a wall panel in the 3D view window.



CFS View Controls



Button/Icon	Name	Description
 Chords	Chords	Allows you to turn the display of wall panel region chords on and off.
 Studs	Studs	Allows you to turn the display of the studs on and off.
 Header	Header	Allows you to turn the display of the studs on and off.
 Top/Sill Plate	Top/Sill Plate	Allows you to turn the display of the top/sill plates on and off.

Design Rules

You must set up design rules for the stud/chord sizes, as well as make a database selection for shear panels. This is done in the **Wall Design Rules** spreadsheet in the **CFS Wall (Studs)** and **CFS Wall (Fasteners)** tabs. See the [CFS Wall - Design Rules](#) topic for more information.

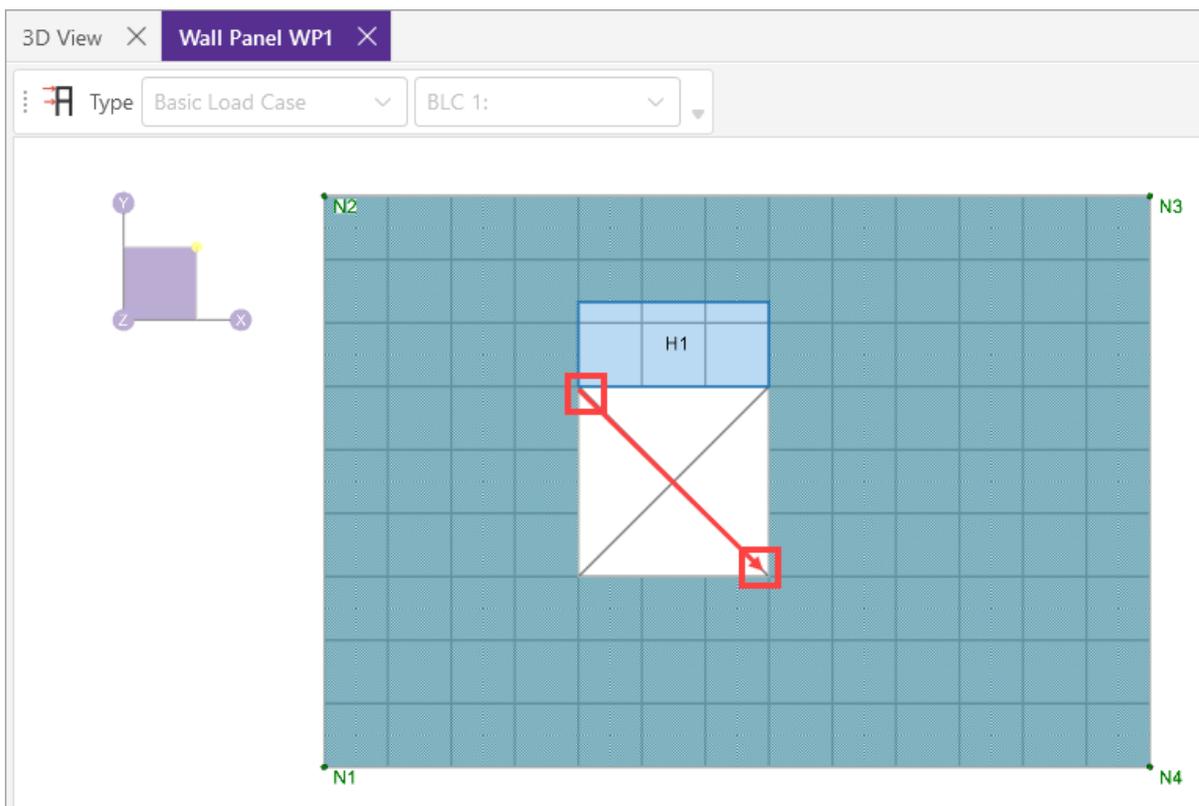
Create Openings in CFS Walls

In the **Walls** ribbon, you have the option of adding rectangular openings to CFS wall panels. To draw an opening:

1. Click the **Openings** icon.



2. Click on two nodes or grid intersections which make up the two diagonal corners of your opening.



Note: Currently header design is not available for CFS walls.

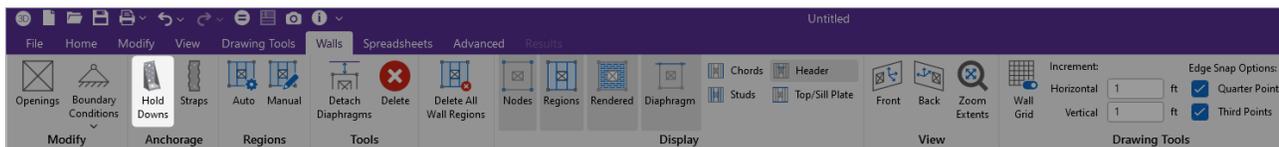
When an opening is drawn, a header beam is automatically created above the opening (as indicated by the H1 block in the image).

Hold-Downs and Straps

Hold-downs and straps are automatically added to your walls in their required locations. For a Type I - Segmented design, you must have hold-downs or straps at the bottom corners of each of your design segments. The program will not permit the drawing of hold-downs or straps at locations where they are not allowed.

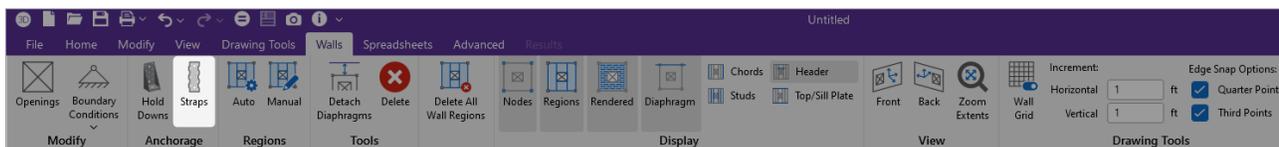
If there are custom locations that you want to add hold-downs or straps, you can do this from the **Walls** ribbon .

Hold downs can represent the anchorage of your wall to the foundation or the connection of shear wall chords between floors. To add hold-downs to the base of your wall, first select the **Hold Downs** icon in the 'Walls' ribbon.



Hold downs must be added after regions are created and can only be added at the corners of regions. Hold down requirements depend on the type of wall design you are performing.

Straps represent the connection of the current wall panel to a wall panel below. To add straps to the base of your wall, first select the **Straps** icon in the 'Walls' ribbon.



Straps also can only be added after regions are created and can only be added at the corners of regions. Strap requirements also follow the same logic as hold-downs as to where they must be defined as far as regions are concerned.



Note:

- Hold-downs and straps are not designed for CFS Walls. However, the required forces will still be reported. More information on this can be found in the CFS Wall Results topic.
- All boundary conditions for wall panels should be defined in the wall panel editor. Adding external boundary conditions can create problems.
- The locations of hold-downs and straps define where the program will calculate tension forces in your walls.
- By default, the program will automatically add hold-downs to the base of walls that have boundary conditions applied to them and will apply straps to walls that have walls below.
- All Straps and Hold-downs are placed at the base of walls only. So, if you would like to put a strap or hold-down between floors, you would apply it to the base of the upper wall.

General Requirements for Shear Walls

The design of CFS shear walls per the available design codes requires that many criteria are satisfied before a wall can be considered adequate. For RISA to work within this framework, we require that certain modeling practices be followed. Outlined below are many general wall modeling practices and limitations as well as specific requirements for the Type I - Segmented method that is available for CFS Walls in the program.



Note: RISA-3D does not currently support Type I with FTAO or Type II walls.

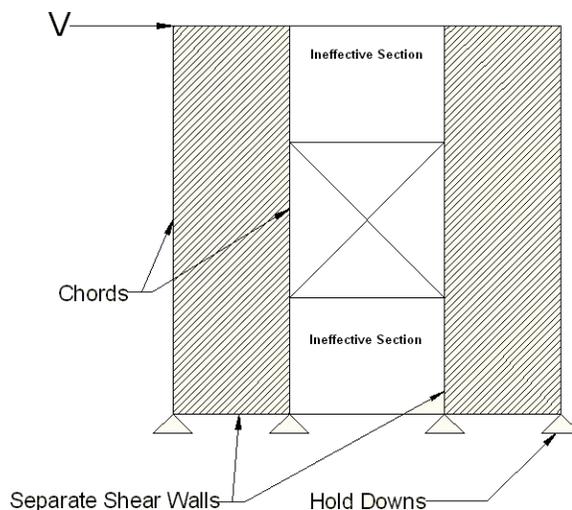
Type I - Segmented Method

The Type I - Segmented methodology is defined in Section B5.2 of the *AISI S240-15 North American Standard for Cold-Formed Steel Structural Framing*.

Where there is a wall panel with openings, the area above and below the openings is disregarded and the wall is designed as being made up of separate, smaller shear walls.

Like all wall panels, the segmented CFS wall is broken into a series of meshed plate elements to represent the overall wall. The portions of the segmented shear wall that are considered "ineffective" in resisting shear are modeled with a plate elements that have a significantly reduced shear stiffness so that they will not receive any significant moment or shear from the FEM analysis.

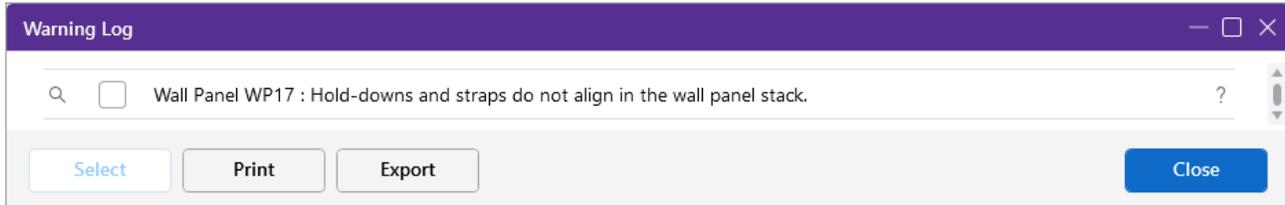
See the diagram below for more information:



In addition, the out of plane stiffness and in plane stiffnesses of the segmented CFS wall are modeled separately based on different assumed plate thicknesses. This is done to insure that the shear stiffness is based entirely on the properties of the sheathing and is not influenced by the out-of-plane stiffness of the wall studs.



Note: If you have several stacked Segmented wall panels with misaligned openings, you will receive the Warning Message shown below upon solution. This message means that RISA-3D has assumed that the strap force from the above wall panel will be spread out across the region directly under it. Therefore, you need to be aware of this assumption and detail the wall panel accordingly.



- A shear panel design will be chosen for the worst-case region in a segmented wall. That panel will then be used for all regions in that wall. The worst-case region is the one that has the highest Shear UC value.

Geometric Requirements and Limitations

The program will check the geometric requirements per AISI S240-15 B5.2.1.1 and AISI S400-E1.4 as required unless otherwise noted below. The sections noted below are assumed to be satisfied and are not checked.



Note:

- AISI S400-E1.4 parts (d) - (g) and (k) - (r)
- AISI S240 B5.2.2.3.2.1.1 parts (c) and (e)
- AISI S400 E2.3.1.1.1.1 parts (c) and (e)

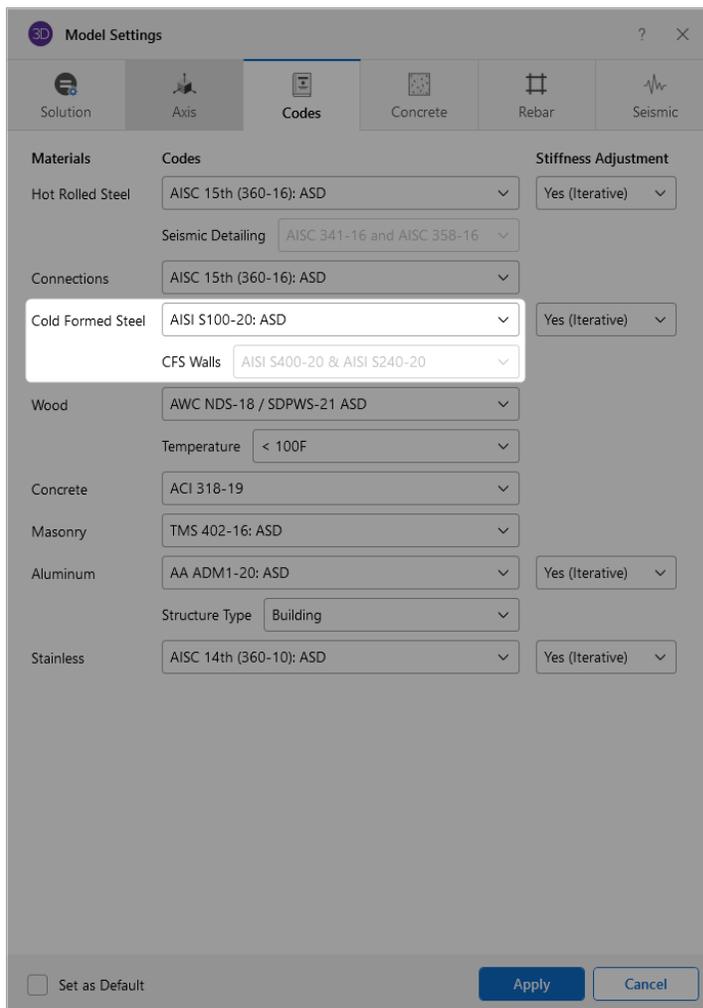
General Program Functionality and Limitations

Assigning Code and Design Methodology

CFS wall designs will be based on the *AISI S240-15 North American Standard for Cold-Formed Steel Structural Framing* and *S400-15 w/S1-16 North American Standard for Seismic Design of Cold-Formed Steel Structural Systems*. You can select these codes in the CFS Walls dropdown within the Codes section of the Model Settings. If you set this code to "None", the program will not perform design for the CFS walls.

The design properties of CFS stud and chords within the wall are referenced from the AISI S100 specification referenced in the Cold Formed Steel dropdown.

In this manner, the design methodology (ASD or LRFD) for the entire wall panel (stud, chord and sheathing design) is determined by the design method selected in the Cold Formed Steel dropdown. For example, if the AISI S100-16: ASD code is selected as shown below, the CFS walls will use ASD factors in the stud, chord and sheathing designs.

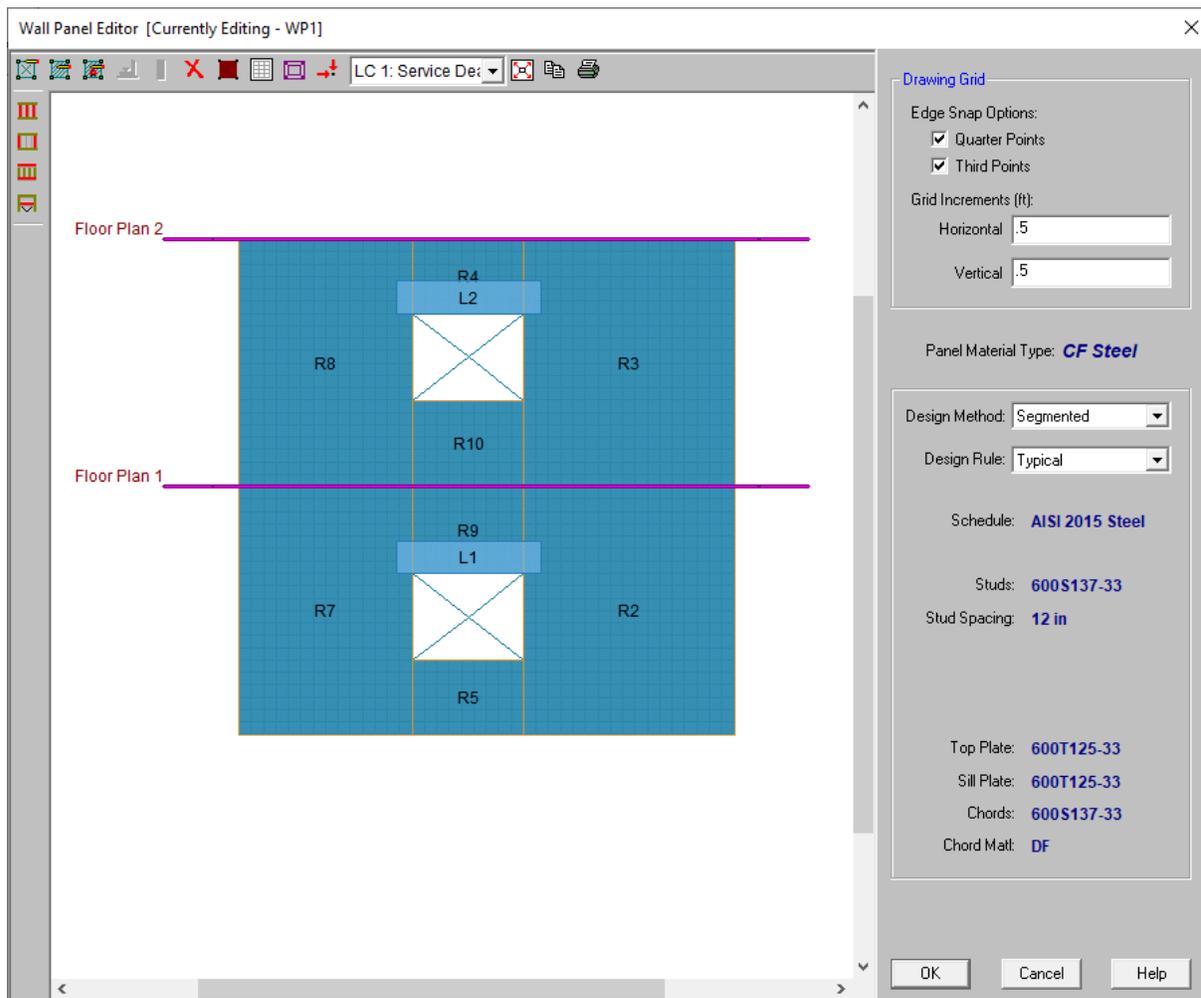


RISAFloor and RISA-3D Interaction

When using RISAFloor and RISA-3D in combination, the interface transitions nicely between the two programs. Here is a quick walk through of this interaction.

Input Interface

1. Model the entire building (gravity and lateral members) within RISAFloor.
Be sure to model all openings and regions for all of the wall panels in the model.



2. Add loading and solve the model.
3. Take the model into RISA-3D via the Director tool.
4. Once in RISA-3D you must add your hold downs and straps to your wall panels.

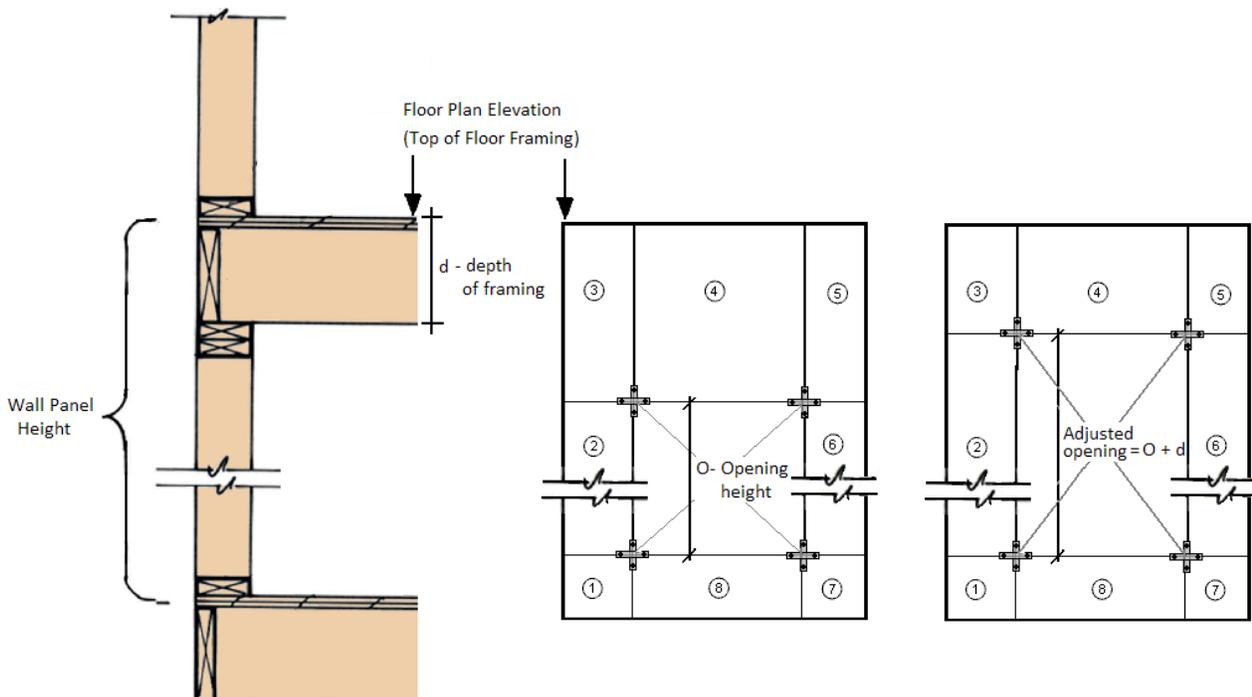
**Note:**

- You can not modify your openings or regions in RISA-3D. All region and opening modifications must be taken back to RISAFloor to be done.
- Hold downs and straps can not be added to wall panels in RISAFloor.
- Hold downs are required at the corners of all full height regions in the wall panel for the Segmented design method.
- The Design Rule and Design Method can be changed in either program at any time.

Modeling Tips- Platform Framing

The CFS wall height in RISA is measured from the bottom of the sill plate to the top of the floor framing as shown below. However, platform framing causes the wall height to be significantly shorter than a RISA model would represent.

This is especially problematic for the FTAO design method where the region above openings is assumed to be much larger than it is built. However, currently only the Type I - Segmented method is available and the portions above and below the opening are not used to transfer shear forces.



Limitations for Hold-downs / Straps (Including deformation)

- The program does not currently have a database for or optimize hold-downs/straps for CFS walls.
- The manufacturer listed or user defined hold-down deformation can be input in the Wall Design Rules spreadsheet for inclusion in the design deflection calculation. See the CFS Wall - Results topic for more information.

Automatic Boundary Conditions

- In RISA-3D, if no boundary conditions or hold downs are defined for CFS wall panels at the lowest level of the structure, the program will automatically create hold downs at the corners of the wall panel. If you do not want the hold downs to be automatically created, define a "free" boundary condition at the base of the wall panel in the wall panel editor.

Deflection

The program will report the design deflection for CFS shear walls that have wood or steel sheathing.

- There is currently no code check for drift or deflection for CFS shear wall panels.

Stud Design

Studs are only designed for load combinations which do not contain a wind or seismic load. The maximum axial load, determined as an envelope force from all of the "gravity" load combinations which have been solved, is determined for each region.

Studs are checked per AISI S240-15 B3.2: Wall Stud Design. The program currently only uses the All Steel Design methodology in which case bracing from the attached sheathing or structural bracing is ignored. Therefore the unbraced length for the buckling of the stud is taken as the full wall height. The top and bottom track thicknesses are considered negligible and are not deducted from the total height.



Note:

- The program currently does not optimize the stud size or spacing.
- The program will use an effective length factor, K , equal to 1.0 for the design of the studs.

Stud force demand and number of studs calculation:

The program first divides the maximum axial force in the region by the length of the region. Then it will multiply this distributed force by the stud spacing to get a maximum stud demand.

The program calculates the number of studs based on the following procedure:

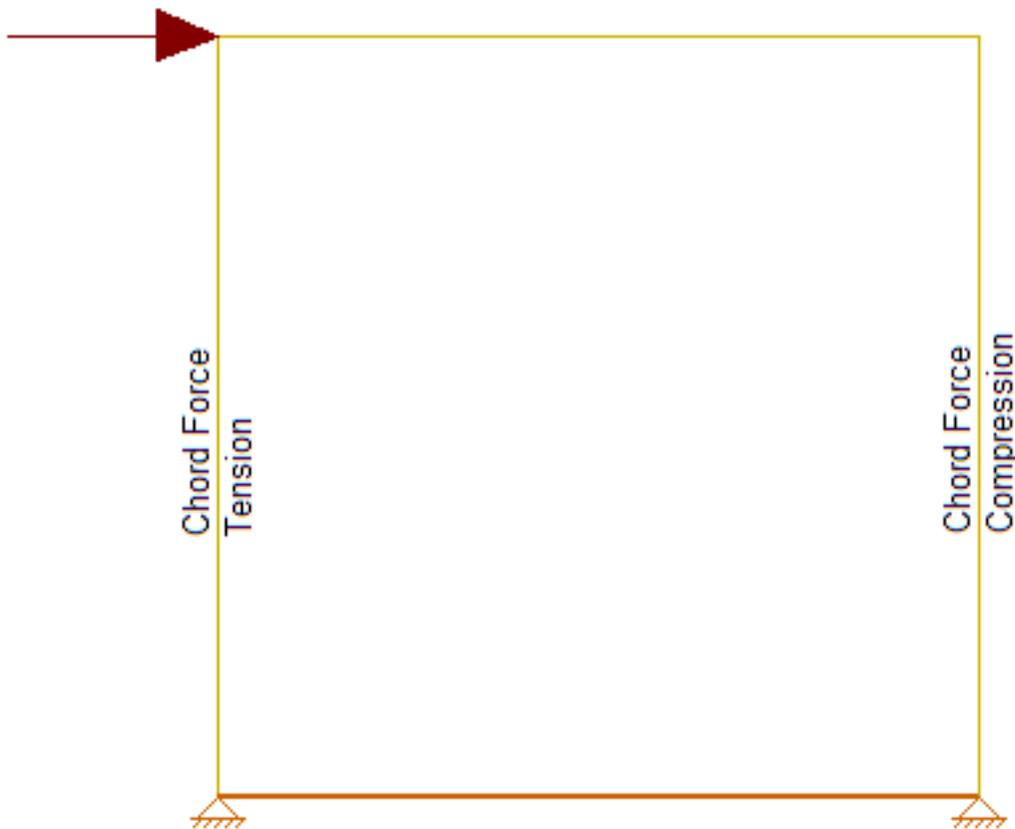
1. Determines the number of spaces between studs by dividing the region length by the stud spacing.
2. Rounds this number up to the nearest whole number.
3. Determines the number of studs by adding one to this number.

Openings

- Currently, the program does not perform header design. This will be available in an updated version.

Chord Design

The chord design is based on forces that are calculated differently for Compression versus Tension. The tension chord force is calculated including the dead load stabilizing moment. The compression chord force includes the only the tributary area of one stud spacing in the compression force. For Segmented design, the chord forces are found based on each region.



Chord forces are calculated per the following equations:

$$\text{Compression Force} = \frac{M}{L} + \frac{P}{n}$$

$$\text{Tension Force} = \frac{-M}{L} + \frac{P}{2}$$

Where:

- M = Moment at the base of the wall
- L = Length of the wall
- P = Axial force at the base of the wall
- n = Number of Studs = Length/Stud Spacing



Note:

- In RISA-3D, the edge of the CFS walls represent the centerline of the chord member. Therefore, the depth of the chord member will not affect the chord force calculation since the centerline of the chord member will always be assumed at the edge of the wall.
- The eccentricity checkbox in the Wall Design Rules spreadsheet will not affect the chord force calculation.
- RISA-3D models the shear walls using only the sheathing. The vertical resistance occurs only at the tension and compression chords. Thus, if two wall panels are stacked on top of each other, the load transfer will only happen at the chord locations. Therefore, the lateral analysis should agree very well with hand calculations. However, it also means that gravity load design may be more appropriate in RISAFloor.

Hold-Down Force

Hold-Down forces are calculated per the following equations:

Note that the hold-down force calculation depends upon whether the eccentricity checkbox has been checked, and the value entered in the eccentricity field.

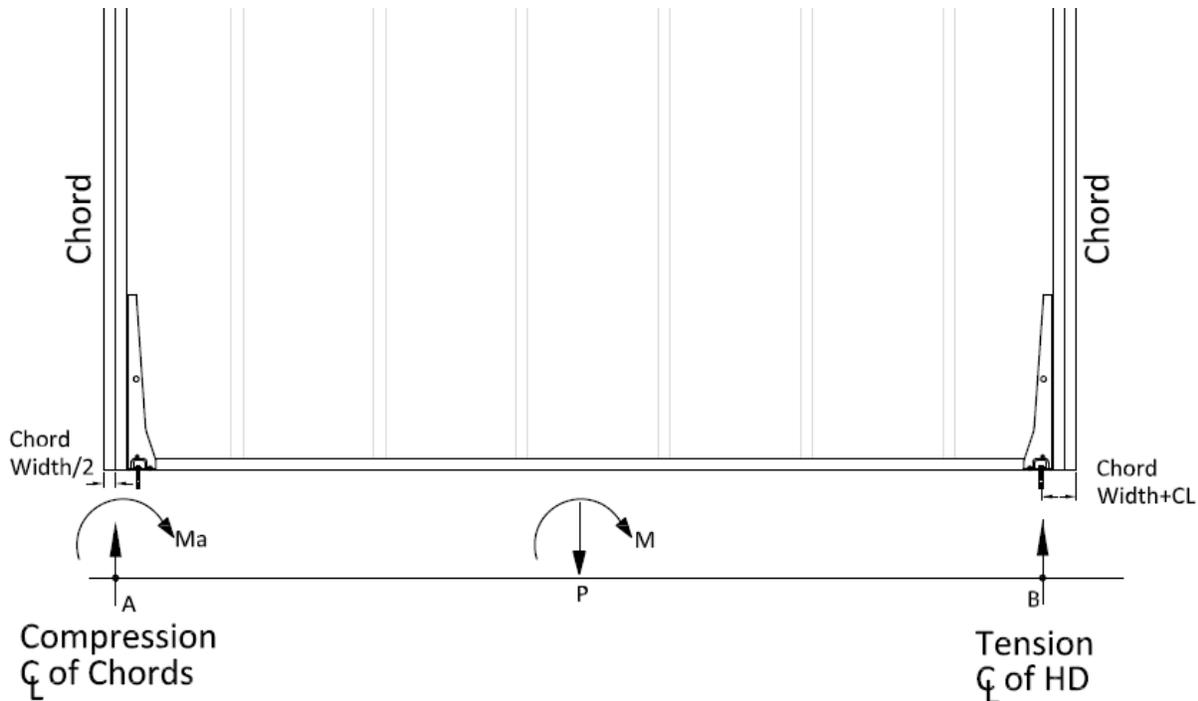
- When the "[Eccentricity](#)" check-box is unchecked in the CFS Wall Design Rule (Fasteners tab), the hold-down force is exactly equal to the chord force indicated above.

Additional CFS Wall Panel Parameters													
Unity Check	Concrete Wall (Rebar)	Concrete Wall (Misc)	Masonry Wall	Masonry In	Masonry Out	Masonry Lintel	Wood Wall (Studs)	Wood Wall (Fasteners)	CFS Wall (Studs)	CFS Wall (Fasteners)			
Label	Type of Wall	Sheathing Material	Schedule	Design Method	Min Panel Thick [in]	Max Panel Thick [in]	Fastener Min Edge Space	Fastener Max Edge Space	HD Chords	Eccentricity	Ecc Dist [in]	HD Deflection [in]	
1	12" [---]	Type I - Segmented	Steel Sheet sheathing	AISI 2020 Steel	Table	0.003	2	2-in.	6-in.	600S137-33	<input type="checkbox"/>	0	1
2	6"	Type I - Segmented	Steel Sheet sheathing	AISI 2020 Steel	Table	0.003	2	2-in.	6-in.	600S137-33	<input checked="" type="checkbox"/>	1.5	1

- When the "Eccentricity" check-box is checked in the CFS Wall Design Rule (Fasteners tab), an eccentricity distance should be indicated in the "Ecc Dist" column. This distance should be the distance between the hold-down and the centerline of the chord member.

Additional CFS Wall Panel Parameters													
Unity Check	Concrete Wall (Rebar)	Concrete Wall (Misc)	Masonry Wall	Masonry In	Masonry Out	Masonry Lintel	Wood Wall (Studs)	Wood Wall (Fasteners)	CFS Wall (Studs)	CFS Wall (Fasteners)			
Label	Type of Wall	Sheathing Material	Schedule	Design Method	Min Panel Thick [in]	Max Panel Thick [in]	Fastener Min Edge Space	Fastener Max Edge Space	HD Chords	Eccentricity	Ecc Dist [in]	HD Deflection [in]	
1	12" [---]	Type I - Segmented	Steel Sheet sheathing	AISI 2020 Steel	Table	0.003	2	2-in.	6-in.	600S137-33	<input type="checkbox"/>	0	1
2	6"	Type I - Segmented	Steel Sheet sheathing	AISI 2020 Steel	Table	0.003	2	2-in.	6-in.	600S137-33	<input checked="" type="checkbox"/>	1.5	1

The Hold-Down force is calculated by finding the Tension chord force. In order to accommodate an unsymmetrical vertical load on the wall, the program adopted a more accurate approach by calculating the moment at the compression side first and then finding the resulting reaction on the tension side.



The resulting equation to determine the hold-down force is as follows:

$$\text{Moment at Point } a = M_a = -M + \left(P * \left(\frac{L}{2} - \frac{b_c}{2} \right) \right)$$

$$\text{Reaction at Point } b = R_b = \frac{M_a}{L - 1.5 * b_c - CL}$$

Where:

- M_a = moment at point A(CL of compression chord)
- M = moment at base of the wall
- P = Axial load at the base of the wall
- L = length of the wall
- b_c = width of chord member
- R_b = reaction at point B = hold down force
- CL = Hold down eccentricity distance (per selected hold down)

Lateral Force Distribution

- The lateral force distribution between piers is based on the relative stiffness of the sheathing, not on the length of the shear wall. For example, if you have an 8 foot wall and a 4 foot wall, the 8 foot wall will take more than $8/(8+4)*100\%$ of the force. The moment of inertia in the 8 foot wall will allow for a larger proportion of load to go into that pier.

Shear Panel Material Capacities

Shear capacity of the wall will be dependent on the Sheathing Material and Design Method that was chosen in the Design Rules spreadsheet - CFS Wall (Fasteners) tab. When design values are pulled from design tables, the program will use the appropriate capacity from the table based on the height-to-width ratio of the wall, member thickness of the stud members and type of loading used.

Tabulated shear capacities are based on the *AISI S240-15 CFS Structural Framing* and *AISI S400-15 w/S1-16 Seismic Design of CFS Structural Systems* codes. If the governing lateral load is a seismic load (under the load category EL, ELX, ELY, ELZ, etc) then the capacity will be pulled from the AISI S400-15 code.



Note: Curtain wall considerations per AISI S240-15 are not considered in the program.

Wood Sheathing

Tabulated design values are referenced from AISI S240-15 Table B5.2.2.3-2 and AISI S400-15 Table E1.3-1.

Steel Sheathing

Tabulated design values are referenced from AISI S240-15 Table B5.2.2.3-1 and AISI S400-15 Table E2.3-1.

If the Effective Width Method is used as the design method, this will be referenced from AISI S400-15 Chapter E2.3.



Note: The program will not perform the end distance check in the effective width method. This is assumed to be satisfied.

Fiberboard

Tabulated design values are referenced from AISI S240-15 Table B5.2.2.3-4. AISI S400-15 does not apply for this material.

Gypsum

Tabulated design values are referenced from AISI S240-15 Table B5.2.2.3-3. AISI S400-15 does not apply for this material.

Stiffness Assumptions

Vertical Direction

RISA uses an orthotropic plate element to de-couple the vertical and shear stiffness of the CFS walls. The vertical stiffness will be based on the E value of the studs and chords as specified in the Materials spreadsheet and the thickness of the wall. The thickness is taken as

$$\textit{Thickness} = \frac{\textit{Total Wall Weight}}{\textit{Material Density} * \textit{Wall Length} * \textit{Wall Height}}$$

Where,

$$\textit{Total Wall Weight} = W_{\textit{studs}} + W_{\textit{chords}} + W_{\textit{top plate}} + W_{\textit{sill plate}} + W_{\textit{sheathing panel}}$$

In-Plane Shear

The in-plane shear stiffness will be based on the G_a value designated within the specified nailing schedule divided by the sheathing thickness. Both the G_a value and the sheathing thickness are defined in the specified sheathing schedule.

Out-of-Plane Shear

For the out of plane shear, RISA uses the same Young's Modulus (E) as used in the vertical direction and a thickness that is calculated from the out of plane moment of inertia. This becomes:

$$Thickness = \left(\frac{I_{chord} + I_{stud} + I_{panel}}{Wall\ Length} \right)^{1/3}$$

CFS Wall Self Weight

The program will calculate CFS wall self weight as a sum of all the weights of the components. The material density is used to calculate the self weight of the studs, chords, top plates, sill plate, and sheathing. These are all then summed together to give the self weight of the entire wall.



Note:

- For this calculation, stud height equals full wall height.
- The number of studs is calculated using the stud spacing specified in Design Rules.

CFS Wall Optimization

The program will optimize CFS walls based on the required demand forces. The program can optimize sheathing for in-plane shear design.

This optimization can substantially modify the stiffness of the wall. To properly adjust the stiffness requires an iterative solution that updates the stiffness of the model. This includes updating the strength properties of the wall as well as the stiffness. The program will handle this optimization/iteration automatically.

To update the stiffness portion of the wall, the program must re-solve your model with these updated stiffnesses as this will change the distribution of forces through the model. Thus, the program will start with it's initial stiffness parameters and solve the model. It will then optimize the wall to meet strength criteria. Another solution will then be run with the new stiffnesses and the program will again optimize the wall to meet strength criteria. This procedure will continue to occur until all wall panel results match those of the previous solution.

The program will always present results in the output that coincide with the stiffness used in the final solution.

Panel Optimization

3D Cold Formed Wall Panel Schedule

Select Code: CFS Steel Sheeting

Available Panel Groups:

- AISI 2020 Steel
- AISI 2015 Steel

Current Selection Type: Use All Panels

	Label	Type Of Wall	Sheathing Material	Up 4 to 1?	Sheet Thickness	Fastener Spacing	Stud Block
1	0.018_#8@6/12_33	Type I - Segmented	Steel Sheet sheathing	No	0.018	6	No
2	0.027_#8@4/12_43	Type I - Segmented	Steel Sheet sheathing	No	0.027	4	No
3	0.027_#8@3/12_43	Type I - Segmented	Steel Sheet sheathing	No	0.027	3	No
4	0.027_#8@2/12_43	Type I - Segmented	Steel Sheet sheathing	No	0.027	2	No
5	0.027_#8@4/12_43_TALL	Type I - Segmented	Steel Sheet sheathing	Yes	0.027	4	No
6	0.027_#8@3/12_43_TALL	Type I - Segmented	Steel Sheet sheathing	Yes	0.027	3	No
7	0.027_#8@2/12_43_TALL	Type I - Segmented	Steel Sheet sheathing	Yes	0.027	2	No
8	0.027_#8@6/12_33	Type I - Segmented	Steel Sheet sheathing	No	0.027	6	No
9	0.027_#8@4/12_33	Type I - Segmented	Steel Sheet sheathing	No	0.027	4	No
10	0.027_#8@3/12_33	Type I - Segmented	Steel Sheet sheathing	No	0.027	3	No
11	0.027_#8@2/12_33	Type I - Segmented	Steel Sheet sheathing	No	0.027	2	No
12	0.027_#8@6/12_33_TALL	Type I - Segmented	Steel Sheet sheathing	Yes	0.027	6	No
13	0.027_#8@4/12_33_TALL	Type I - Segmented	Steel Sheet sheathing	Yes	0.027	4	No
14	0.027_#8@3/12_33_TALL	Type I - Segmented	Steel Sheet sheathing	Yes	0.027	3	No
15	0.027_#8@2/12_33_TALL	Type I - Segmented	Steel Sheet sheathing	Yes	0.027	2	No

OK Cancel

The procedure that RISA uses for design optimization is fundamentally based upon the assumption that there is a 'cost' to shear capacity, and therefore the ideal panel design would have as little shear capacity as possible to meet code requirements. Once the program has determined the shear demand on the wall it will choose the most economical panel configuration based on that which has a **Shear Capacity** closest to, but not exceeding the shear demand.

The program will also only pick shear panels that qualify based on the region height-to-width ratio and minimum stud thickness requirements.

The default shear panels in the program come straight from published tables in the AISI S240-15 and AISI S400-15.



Note:

- The shear forces listed in the XML spreadsheet are nominal shear strength values. The force capacity will be adjusted by the appropriate factor depending on if the ASD or LRFD design method is selected in the Cold Form Steel code dropdown.
- Panels with a label containing the characters "_W" together will be ignored during design optimization.

Cold Formed Steel Walls - Design Rules

The CFS wall panel element allows you to easily model, analyze and design CFS walls for in plane loads. Here we will explain the CFS specific inputs and design considerations. For general wall panel

information, see the [Wall Panels](#) topic. For information on CFS wall design considerations, see the [CFS Wall - Design](#) topic. For CFS wall results interpretation, see the [CFS Wall Results](#) topic.

Unity Check

Wall Panel U.C. Parameters					
Masonry Lintel	Wood Wall (Studs)	Wood Wall (Fasteners)	CFS Wall (Studs)	CFS Wall (Fasteners)	
Unity Check	Concrete Wall (Rebar)	Concrete Wall (Misc)	Masonry Wall	Masonry In	Masonry Out
	Label	Max Bending Chk	Max Shear Chk		
1	8-in CFS Wall	1	1		
2	6-in CFS Wall	1	1		

Max Bending Chk

Wall Panel Unity Check Spreadsheet Columns

Column Header	Description
Max Bending Chk	The Max Bending Chk column defines the hold-down maximum code check value. Currently, hold-down design/optimization is not available for CFS walls. However, this will be made available in a future release.
Max Shear Chk	The Max Shear Chk column defines the shear panel selection maximum code check value. Typically this is always 1.0, but it allows the option of having the program pick out a shear panel layout that is not at maximum capacity. Putting a value of 0.8 chooses a panel thickness and nailing that is at 80% of capacity.

CFS Wall (Studs)

CFS Wall Panel Parameters						
Unity Check	Concrete Wall (Rebar)	Concrete Wall (Misc)	Masonry Wall	Masonry In	Masonry Out	
Masonry Lintel	Wood Wall (Studs)	Wood Wall (Fasteners)	CFS Wall (Studs)	CFS Wall (Fasteners)		
	Label	Top Plate	Sill Plate	Studs	Stud Space [in]	Header Size
1	8-in CFS Wall	600T125-33	600T125-33	600S137-33	12	2-600S137-43-BB
2	6-in CFS Wall	600T125-33	600T125-33	600S137-33	12	2-600S137-43-BB

CFS Wall Panel Parameters Spreadsheet Columns

Column Header	Description
Top Plate	The Top Plate column lets you specify the member to be used as a top plate for your wall. A top plate is a member that runs continuously along the top of the wall studs.
Sill Plate	The Sill Plate column lets you specify the member to be used as a sill plate for your wall. A sill plate is a member that runs continuously along the bottom of the wall studs.
Studs	The studs column lets you specify the member to be used for studs in your wall. Studs are vertical members in the wall, attached to the sill plate at the bottom and the top plate at the top. Note that you can use back-to-back or toe-to-toe options as well.
Stud Space	<p>The Stud Space column lets you specify the spacing between the studs within the wall.</p> <p>You may specify a minimum and maximum spacing of wall studs. The program can then optimize the stud spacing based on axial design only. For information on how the optimization works, see the Wood Wall - Design topic.</p> <ul style="list-style-type: none"> • If you specify the maximum and minimum stud spacing as the same value, then we will use that value exclusively. • Out of plane design is not performed for wood walls, so any optimization is based only on axial forces in the studs.
Header Size	The Header Size column lets you define the default header size for all openings. Note that the program does not provide a design for CFS headers at this time.

Design Rules - CFS Wall (Fasteners)

Unity Check	Concrete Wall (Rebar)	Concrete Wall (Misc)	Masonry Wall	Masonry In	Masonry Out	Masonry Lintel	Wood Wall (Studs)	Wood Wall (Fasteners)	CFS Wall (Studs)	CFS Wall (Fasteners)			
	Label	Type of Wall	Sheathing Material	Schedule	Design Method	Min Panel Thick [in]	Max Panel Thick [in]	Fastener Min Edge Space	Fastener Max Edge Space	HD Chords	Eccentricity	Ecc Dist [in]	HD Deflection [in]
1	8-in CFS Wall	Type I - Segmented	Steel Sheet sheathing	AISI 2020 Steel	Table	0.003	2	2-in.	6-in.	6005137-33	<input type="checkbox"/>	0	1
2	6-in CFS Wall	Type I - Segmented	Steel Sheet sheathing	AISI 2020 Steel	Table	0.003	2	2-in.	6-in.	6005137-33	<input checked="" type="checkbox"/>	1.5	1

Design Rules - CFS Walls (Fasteners) Spreadsheet Columns

Column Header	Description
Type of Wall	The Type of Wall column shows the design methodology of the CFS wall panel. Currently, CFS walls can only be designed using the Type I - Segmented method.
Sheathing Material	The Sheathing Material column lets you choose which sheathing material you would like to be used for lateral design optimization. The specific sheathing material that you choose automatically populates the corresponding sheathing database in the adjacent Schedule column.

Column Header	Description
Schedule	<p>The Schedule column lets you choose the sheathing database and Panel Group you would like to use for design optimization.</p> <p>By selecting the Select Individual Panel option, an individual panel type can be assigned.</p> <p>For information on how the optimization works, see the CFS Wall - Design topic. For more information on this schedule, as well as information on how to edit or create your own custom schedule, see Appendix G-CFS Design Databases.</p>
Design Method	<p>The Design Method column lets you choose the method in which the lateral capacity is determined by the program. If you choose the Table option, the program references tabulated shear capacities in the AISI S240 and AISI S400 manual. The Effective Strip Method is only available for the Steel Sheet sheathing material option.</p> <p>For more information on this see the CFS Wall - Design topic.</p>
Min Panel Thick	<p>The Min Panel Thick column lets you set minimums for the thickness of the sheathing to be designed.</p> <p>If the same value is set for both max and min, then that is the thickness used.</p>
Max Panel Thick	<p>The Max Panel Thick column lets you set maximums for the thickness of the sheathing to be designed.</p> <p>If the same value is set for both min and max, then that is the thickness used.</p>
Fastener Min Edge Space	<p>The Fastener Min Edge Space column lets you set minimums for the spacing of the nails that fasten the sheathing to the boundary members (top plate, sill plate, hold down chords).</p> <p>Note that a 12" spacing is assumed for all field nailing (nails fastening the sheathing to the internal studs).</p>
Fastener Max Edge Space	<p>The Fastener max Edge Space column lets you set maximums for the spacing of the nails that fasten the sheathing to the boundary members (top plate, sill plate, hold down chords).</p> <p>Note that a 12" spacing is assumed for all field nailing (nails fastening the sheathing to the internal studs).</p>
HD Chords	<p>The HD Chords columns lets you choose what member size you would like to use for the Hold Down Chords (Posts) at both ends of the wall panel.</p>
Eccentricity	<p>The Eccentricity checkbox lets you control whether the eccentricity distance of the hold-down (input into the Ecc Dist column) is used in the calculation of the hold-down force.</p> <p>For more information, see the Hold Down force calculation section in the</p>

Column Header	Description
	CFS Wall - Design topic.
Ecc Dist	The Ecc Dist column lets you input an eccentricity distance to be used in the calculation of the hold down force. This distance should be measured from the centerline of the HD chord to the actual hold-down anchorage.
HD Deflection	The HD Deflection column lets you input the manufacturer hold down deflection which is used in the reported shear wall design deflection calculation.

Cold Formed Steel Walls - Results

CFS Wall results are presented in the **Wall Panel Design** Spreadsheet and the detail reports.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **CFS Walls**.

CFS Wall Results Spreadsheets

CFS Wall Axial Results

CFS Wall Panel Axial Code Checks (AISI S100-20: ASD)											
Concrete In		Concrete Out		Concrete Seismic		Masonry In		Masonry Out		Masonry Lintel	
Masonry Seismic		Wood Wall Axial		Wood Wall In-Plane		Wood Header		CFS Wall Axial		CFS Wall In-Plane	
	Wall Panel	Region	Stud Size	Stud Spacing[in]	Axial Check	Gov LC	Chord Size	Chord Axial Check	Gov LC		
1	WP1	R1	600S137-33	12	0	N/A	600S137-33	NC	NC		
2		R4	600S137-33	12	0	N/A	600S137-33	NC	NC		
3	WP2	R1	600S137-33	12	0	N/A	600S137-33	NC	NC		
4		R4	600S137-33	12	0	N/A	600S137-33	NC	NC		
5		R7	600S137-33	12	0	N/A	600S137-33	NC	NC		
6	WP3	R1	600S137-33	12	0.013	1	600S137-33	0.169	1		

The following table provides descriptions of the columns in the spreadsheet.

CFS Walls Axial Results Spreadsheet

Column Header	Description
Wall Panel	The Wall Panel column lists the CFS wall panels that you have defined.
Region	The Region column lists the wall panel region that the results are based on.
Stud Size	The Stud Size column shows the size of the stud chosen for your wall

Column Header	Description
	based on the Design Rules you have defined.
Stud Spacing	The Stud spacing column shows the spacing between stud chosen for your wall, based on the Design Rules you have defined.
Axial Check	The Axial Check column displays the code check ratio between the member load and the member capacity.
Gov LC	The Gov LC column adjacent to the Axial Check column, shows the governing load combination for the design.
Chord Size	The Chord Size column shows the optimum chord size chosen for your wall, based on the Design Rules you have defined. Note that the chords are the vertical hold-down members/posts at the both ends of the wall.
Chord Axial Check	The Cord Axial Check column displays the code check ratio between the member load and the member capacity.
Gov LC	The Gov LC column adjacent to the Chord Axial Check column, shows the governing load combination for the design.



Note:

- When running the [Segmented](#) design method, the wall panel regions above and below the opening are not considered in design. Thus an NC (no calculation) is displayed.
- If there are some constraints that will not allow a wall to be designed, an NC (no calculation) will be displayed. Check the “Warning Log” within the program for more information on this.

CFS Wall In-Plane Results

Concrete In		Concrete Out		Concrete Seismic		Masonry In		Masonry Out		Masonry Lintel	
Masonry Seismic		Wood Wall Axial		Wood Wall In-Plane		Wood Header		CFS Wall Axial		CFS Wall In-Plane	
	Wall Panel	Shear Panel Label	Region	Shear Check	Shear Force[k/ft]	Gov LC	Hold-down Force[k]	Gov LC			
1	WP1	0.018_#8@6/12_...	R1	0	0	N/A	NC	NC			
2		0.018_#8@6/12_...	R4	NC	0	N/A	NC	NC			
3	WP2	0.018_#8@6/12_...	R1	0	0	N/A	NC	NC			
4		0.018_#8@6/12_...	R4	NC	0	N/A	NC	NC			
5		0.018_#8@6/12_...	R7	NC	0	N/A	NC	NC			
6	WP3	0.018_#8@6/12_...	R1	0.091	0.022	1	NC	NC			

The following table provides descriptions of the columns in the spreadsheet.

CFS Walls In-Plane Results Spreadsheet

Column Header	Description
Wall Panel	The Shear Panel Label shows the optimum shear panel arrangement chosen for your wall, based on the Design Rules you have defined.
Shear Panel Label	The Shear Panel Label column displays the name of the panel in the model.
Region	The Region gives the region for which the design values are being displayed.
Shear Check	The Shear Check value is a code check ratio between the panel shear load and the panel shear capacity.
Shear Force	The Shear Force column displays the value that governed the design.
Gov LC	The Gov LC column adjacent to the Shear Force column, displays the load combination that governed the design.
Hold-down Force	The Hold-Down Force show the maximum value of hold-down/strap that produced the highest strap force. This maximum force could occur on either end of the wall/region.
Gov LC	The Gov LC column adjacent to the Hold-down Force column, shows the load combination that produced the highest strap force.

**Note:**

- When running the [Segmented](#) design method, the wall panel regions above and below the opening are not considered in design. Thus an NC (no calculation) will be displayed. If running a combined RISAFloor and RISA-3D model use RISAFloor to get header results.
- If there are some constraints that will not allow a wall to be designed, an NC (no calculation) will be displayed. Check the “Warning Log” within the program for more information on this.
- If “default” is shown in the ‘Hold-Down Label’ column it means that a hold down is not required for this wall/region.

CFS Wall Self Weight

The program will calculate the self weight of a CFS wall based on the weights of the individual components. Using the material density, the self weight is calculated for the studs, chords, top plates, sill plates, and sheathing. These are all then summed for the total self weight of the wall.

CFS Wall Detail Reports

The detail report gives detailed information about the wall design. The detail reports are specifically molded to the type of design specified.

**Note:**

- Only the **Segmented** design method is considered for CFS walls at this time.
- Many of the values for design checks seen below are not performed in RISAFloor as it is strictly a gravity design program.

Accessing the Detail Reports and the Specific Windows

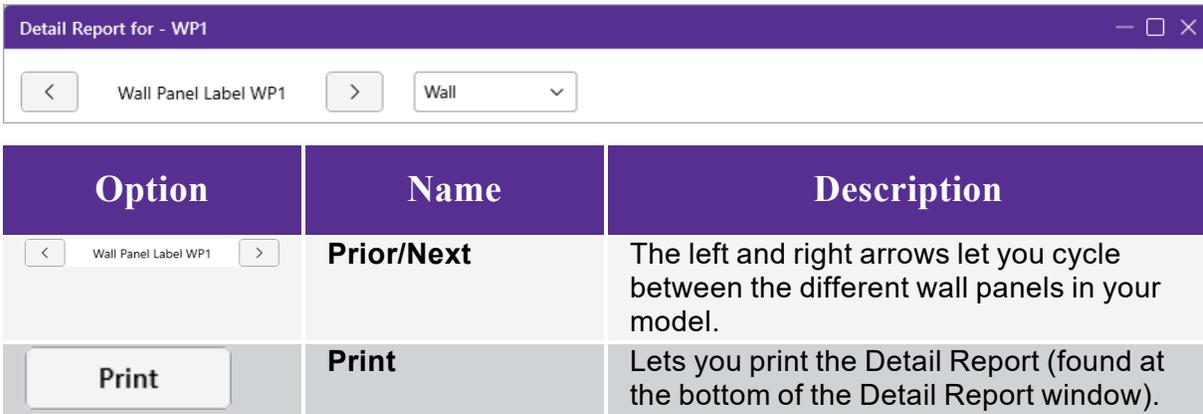
Once you have a solved model, the detail reports become available. They are accessible in two ways:

- If you have the **Wall Panel Design** spreadsheet open, you can right-click on a line in the spreadsheet and select **Detailed Report**.
- If you are in a graphic view of your model, there is a  button that will open up the detail report window.



Note: Detail report information is not available for an envelope solution.

Once the detail report window is open, you will see a dialog area at the top.



Option	Name	Description
 Wall Panel Label WP1 	Prior/Next	The left and right arrows let you cycle between the different wall panels in your model.
Print	Print	Lets you print the Detail Report (found at the bottom of the Detail Report window).

The Wall detail report gives an overall summary of your wall, complete with governing code checks and opening information.

Segmented Method

The **Segmented** design method uses each of the three detail report sections to give design information.

Wall Window

This window gives an overview of the wall, giving controlling region information and deflection information. Note that this window only gives information on the full-height segments in your wall, as this is the basis of the **Segmented** method.

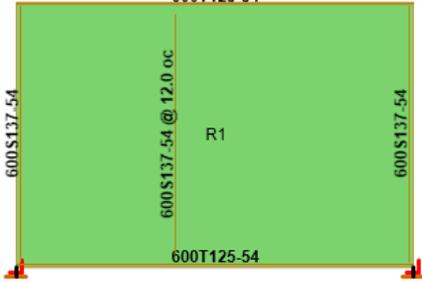
Input Echo

Detail Report for - WP1
— □ ×

<
Wall Panel Label WP1
>
Wall
▼

Detail Report: WP1

CFS Wall



GENERAL		GEOMETRY		MATERIALS	
Code:	AISI 20 Code	Total Height (ft):	10	Top PI	A653 SS Gr50/1 600T125-54
Design Method:	Segmented	Total Length (ft):	15	Sill	A653 SS Gr50/1 600T125-54
Wall Material:	A653 SS Gr50/1	Max H/W Ratio:	0.67	Wall Stud	A653 SS Gr50/1 600S137-54
Panel Schedule:	AISI 2020 Steel			Chord	A653 SS Gr50/1 600S137-54
Sheathing Material:	Steel Sheet sheathing				
Sel. Shear Panel:	0.033_#8@2/12_33_EQ				
Stud Design Method:	All Steel				
Shear Design Method:	Table				

ENVELOPED RESULTS

Controlling Shear Region	Shear Panel	Shear UC	Shear LC	Chord UC	Chord LC	Stud UC	Stud LC
R1	0.033_#8@2/12_33_EQ	0.958	6 (S)	4.367	3 (W)	N/C	N/C

REGION INFORMATION

Region Label	H/W Ratio	Shear UC	Shear LC	Hold-Down	Hold-Down UC	Hold-Down LC	Chord UC	Chord LC	Stud UC	Stud LC
R1	0.67	0.958	6 (S)	NC	NC	NC	4.367	3 (W)	N/C	N/C

DEFLECTION RESULTS

Maximum Region Deflection (in)	Deflection LC	Finite Element Deflection (in)	Shear Stiffness Adjustment Factor
0.136 (R1)	6 (S)	0.008	1

Print
 Add to Full Report
Close

This lists information about the wall, similar to the [Region](#) report and also gives an image of the wall. The image shows the location of hold-downs/straps, regions and headers.

Design Summary

Detail Report for - WP1 — □ ×

< Wall Panel Label WP1
> Wall ▾

ENVELOPED RESULTS

Controlling Shear Region	Shear Panel	Shear UC	Shear LC	Chord UC	Chord LC	Stud UC	Stud LC
R1	0.033_#8@2/12_33_EQ	0.958	6 (S)	4.336	3 (W)	0.695	11

REGION INFORMATION

Region Label	H/W Ratio	Shear UC	Shear LC	Hold-Down	Hold-Down UC	Hold-Down LC	Chord UC	Chord LC	Stud UC	Stud LC
R1	0.67	0.958	6 (S)	NC	NC	NC	4.336	3 (W)	0.695	11

DEFLECTION RESULTS

Maximum Region Deflection (in)	Deflection LC	Finite Element Deflection (in)	Shear Stiffness Adjustment Factor
0.136 (R1)	6 (S)	0.008	1

Print
 Add to Full Report
Close

Design Summary Sections

Section	Description
Envelope Results	The Enveloped Results gives the code checks for all of the controlling elements in the wall and their associated load combinations.
Region Information	The Region Information gives the tabulated results of all of the full-height regions in your wall.
Opening Information	The Opening Information simply states that header design cannot be completed with the Segmented method. The regions above and below the opening have their shear stiffnesses set to be zero and this causes the header forces to be invalid.
Deflection Results	The Deflection Results gives both the calculated AISI code deflection (Maximum Region Deflection) and the FE deflection for use as a means of comparison. Because the AISI equations are empirical and take into account many non-elastic considerations such as nail slip, these two values may not be the same.

Region Window

This window gives information for your wall on a region by region basis. Note that only full-height regions of the wall panel will have a region detail report. The **Segmented** method only considers these full height segments in the design of the wall.

The region detail report is split into four portions: [input echo](#), [diagrams](#), [design details](#), and [cross section detailing](#). Note that in RISAFloor the detail reports are less detailed because RISAFloor does not consider lateral forces which RISA-3D does.

Input Echo

Below is the input echo portion of the detail report.

Criteria table

Criteria	Description
Code	Gives the code used to design your wall panel.
Design Method	Gives the design method that was used.
Wall Material	Specifies the CFS type assigned to the entire wall.
Panel Schedule	Specifies the sheathing material used to optimize panel selection (set in Design Rules).

Materials table

Material	Description
Wall Studs	Specifies the CFS material type assigned to the wall studs.
Stud Size	Specifies the member size used for the wall studs.
Chord Material	Specifies the CFS material type assigned to the chords (vertical members at both ends of the wall).
Chord Size	Specifies the member size used for the chords (vertical members at both ends of the wall).
Top Plate & Sill	Specifies the CFS material type assigned to the top and sill plates.
Top Plate Size	Specifies the member size used for the top plate.
Sill Plate Size	Specifies the member size used for the sill plate.

Geometry table

Geometry	Description
Total Height	This is the height of the wall panel region.
Total Length	This is the length of the wall panel region.
Region H/W Ratio	This is the ratio of wall height to length, using the minimum wall height.
Cap. Adj (2w/h)	This is an aspect ratio reduction factor for the shear panel strength per AISI S240-15 B5.2.2.1. This factor applies only for wall panels that have an aspect ratio greater than 2 but less than or equal to 4. This factor is applied separately for each full-height region in your wall.

Geometry	Description
Stud Spacing	This is the optimized stud spacing based on your Design Rules.
K	This is the effective length, K Factor used for stud and chord compression design.
HD Eccentricity	This is the eccentricity of the hold-down connection. This is based on user input (set within the Design Rules).

Envelope Diagrams

These diagrams show the axial forces, in-plane shear, and in-plane moments of the wall, as well as the maximum and minimum forces and their locations.

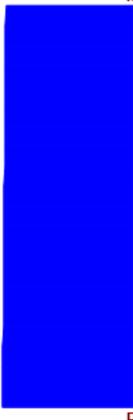
Detail Report for - R1 of WP1 — □ ×

< Wall Panel Label WP1
>
Region v R1 v
In-Plane v

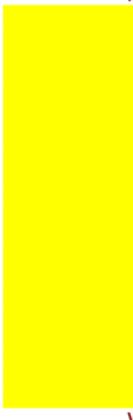
Detail Report: WP1 (Region R1)

CRITERIA	GEOMETRY	MATERIALS
Code: AISI 20 Code	Total Height (ft): 10	Top PI A653 SS Gr50/1 600T125-54
Design Method: Segmented	Total Length (ft): 15	Sill A653 SS Gr50/1 600T125-54
Wall Material: A653 SS Gr50/1	Region H/W: 0.67	Wall Stud A653 SS Gr50/1 600S137-54
Panel Schedule: AISI 2020 Steel	Aspect Ratio: 1.00	Chord A653 SS Gr50/1 600S137-54
Sheathing Material: Steel Sheet sheathing	HD Eccentricity (in): 0	

ENVELOPE DIAGRAMS



Min: 10.5 at 10 ft
Max: 10.783 at 0 ft



Max: 7.501 at 5 ft



Min: -75.002 at 0 ft
Max: -0.004 at 10 ft

Print
 Add to Full Report
Close



Note:

The diagrams are not actually enveloped.

- The **Axial** diagram shown is the diagram from the governing load combination for *Stud* design.
- The **Shear and Moment** diagram is the diagram from the governing load combination for *Shear Panel* design.

Design Details

This portion gives you the capacity and strength values at the section in the wall where the combined check is maximum, as well as the governing load combination. Much of this information is also reported in the **CFS Wall Panel Design** spreadsheets.

Detail Report for - R1 of WP1
— □ ×

< Wall Panel Label WP1 >
Region R1
In-Plane In-Plane

DESIGN SUMMARY

STUDS			
Required Cap (k):	0.719	Governing LC:	11
Provided Cap (k):	1.034	Stud Spacing (in):	12
Ratio:	0.695	Studs in Region:	16
		K:	1.00

CHORDS			
Max Comp Force (k):	5.578	Gov Comp LC:	3
Comp Capacity (k):	1.286	Max Tens Force (k):	1.534
Comp Ratio:	4.336	Tens Capacity (k):	15.389
		Tens Ratio:	0.1
		Gov Tens LC:	5

SHEAR PANEL			
Selected Shear Panel:	0.033_#8@2/12_33_EQ	Ratio:	0.958
Required Cap (k/ft):	0.5	Governing LC:	6 (Seismic)
Original Cap (k/ft):	1.305	Panel Thick (in):	0.033
Adjusted Cap (k/ft):	0.522	Fastener Spacing (in):	2
		Number of Sides:	One
		Screw Size:	#8

DEFLECTIONS			
Component 1 (in):	0.018	Component 3 (in):	0
Component 2 (in):	0.019	Component 4 (in):	0.1
		Tot Deflection (in):	0.136
		Governing LC:	6

Print
 Add to Full Report
Close

Design Details Sections

Section	Description
Studs	<p>The provided capacities of the Studs members are calculated using the standard provisions for tension/compression members. These members are assumed to be fully braced in the weak axis, and unbraced in the strong axis.</p>
Chords	<p>The provided capacities of the Chords members are calculated using the standard provisions for tension/compression members. These members are assumed to be fully braced in the weak axis, and unbraced in the strong axis.</p> <p>For more information on the chord force calculations, see the CFS Wall - Design topic.</p> <div data-bbox="492 758 1409 905" style="border: 1px solid black; padding: 5px; background-color: #e6f2ff;">  Note: For chord results, the tension/compression capacity is computed using the reduced cross-sectional properties caused by the hold-down bolt hole. </div>
Shear Panel	<p>The Selected Shear Panel section shows the call-out from the shear panel database. The information below it, is the information describing the call-out.</p> <p>The provided capacity of the shear panel is taken from the Shear Capacity column of the panel database. This is the adjusted capacity based on tables within the AISI S240-15 and S400-16. The Governing LC explicitly states if the controlling load combination was based on Wind or Seismic. The program does a unity check for all LCs that are being solved, finds the maximum value and reports that information.</p> <p>This section of the report echoes the database information for the selected shear panel. For more information on these properties refer to Appendix G-CFS Shear Wall Files.</p>

Section	Description
	 <p>Note:</p> <ul style="list-style-type: none"> The displayed "Shear Capacity" takes the given nominal shear capacity from the design code and multiplies or divides it by the appropriate code factor and aspect ratio factor when applicable. A shear panel design will be chosen for the worst-case region in a segmented wall. That panel will then be used for all regions in that wall. The worst-case region is the one that has the highest Shear UC value. Because wind and seismic loading allows for different design capacities, the highest shear in the wall may not be the governing shear (if that highest shear was due to wind).
Deflections	<p>The Deflections section shows the Detail Report based on an approximation from the design code.</p> <p>See the following Deflection section for more information.</p>

Deflection

DEFLECTIONS			
Component 1 (in):	0.018	Component 3 (in):	0
Component 2 (in):	0.019	Component 4 (in):	0.1
		Tot Deflection (in):	0.136
		Governing LC:	6

The deflection listed in the detail report is based on an approximation from the design code:

AISI Design Deflection

Per the AISI S240-15 North American Standard for Cold-Formed Steel Structural Framing Equation B5.2.5-1:

$$\delta = \frac{2 \cdot v \cdot h^3}{3 E_S \cdot A_C \cdot b} + \omega_1 \cdot \omega_2 \cdot \frac{v \cdot h}{\rho \cdot G \cdot t_{sheathing}} + \omega_1^{\frac{5}{4}} \cdot \omega_2 \cdot \omega_3 \cdot \omega_4 \cdot \left(\frac{v}{\beta}\right)^2 + \frac{h}{b} \cdot \delta_v$$

Refer to the AISI 240-15 standard page 52 for detail definitions for each term in the equation.

The first component of the above equation determines the Bending Component of the deflection.

The second component of the above equation determines the Shear Component of the deflection.

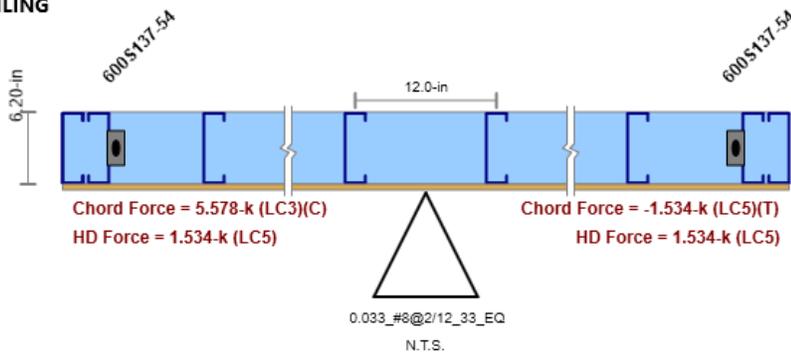
The third component of the above equation determines the Hold-Down Elongation, which causes additional deflection. The Δ_v term refers to vertical deformation of the hold-down. This is a user input within the Wall Design Rules.



Note: This is the theoretical deflection of the wall. This may differ from the deflection of the wall as performed by finite element analysis within RISA. Therefore, this deflection value may not coincide with the reported deflection value in the deflections spreadsheets.

Cross Section Detailing

CROSS SECTION DETAILING



The last section of the detail report consists of the wall detailing information. This information is provided as a visual confirmation of the wall design. The wall thickness, and stud spacing are shown as dimensions. The triangle shows sheathing on one side of the wall, with the abbreviated panel designation. The chord sizes/forces are shown at either end.



Note:

- The graphic will show hold-downs regardless if one is required or not.
- The chord members will always be shown as CS members face-to-face despite what is specified in the Design Rules. This is just a graphical representation of the chord member placement.

Concrete Wall Panel - Design

The concrete wall panel element allows you to easily model, analyze, and design concrete walls for in plane and out of plane loads for the codes defined below:

- The 2022, 2019, 2014, 2011, 2008 and 2005 Editions of ACI 318.
- The 2014 Edition of CSA A23.3.

**Note:**

- Unless otherwise specified, all code references below are to ACI 318-14.
- Walls designed in RISA meet all of the requirements for Ordinary Concrete Shear Walls.
- ACI 318-19 (22) re-approves previous ACI 318-19 without any technical changes, therefore all references in the help documentation and in the program continue to use ACI 318-19.

Here we will explain the concrete-specific inputs and design considerations. For general wall panel information, see the [Wall Panels](#) topic. For concrete wall design rule information, see the [Concrete Wall - Design Rules](#) topic. For concrete wall results interpretation, see the [Concrete Wall Results](#) topic.

Concrete Wall Panel - Design

The concrete wall panel element allows you to easily model, analyze, and design concrete walls for in plane and out of plane loads for the codes defined below:

- The 2022, 2019, 2014, 2011, 2008 and 2005 Editions of ACI 318.
- The 2014 Edition of CSA A23.3.

**Note:**

- Unless otherwise specified, all code references below are to ACI 318-14.
- Walls designed in RISA meet all of the requirements for Ordinary Concrete Shear Walls.
- ACI 318-19 (22) re-approves previous ACI 318-19 without any technical changes, therefore all references in the help documentation and in the program continue to use ACI 318-19.

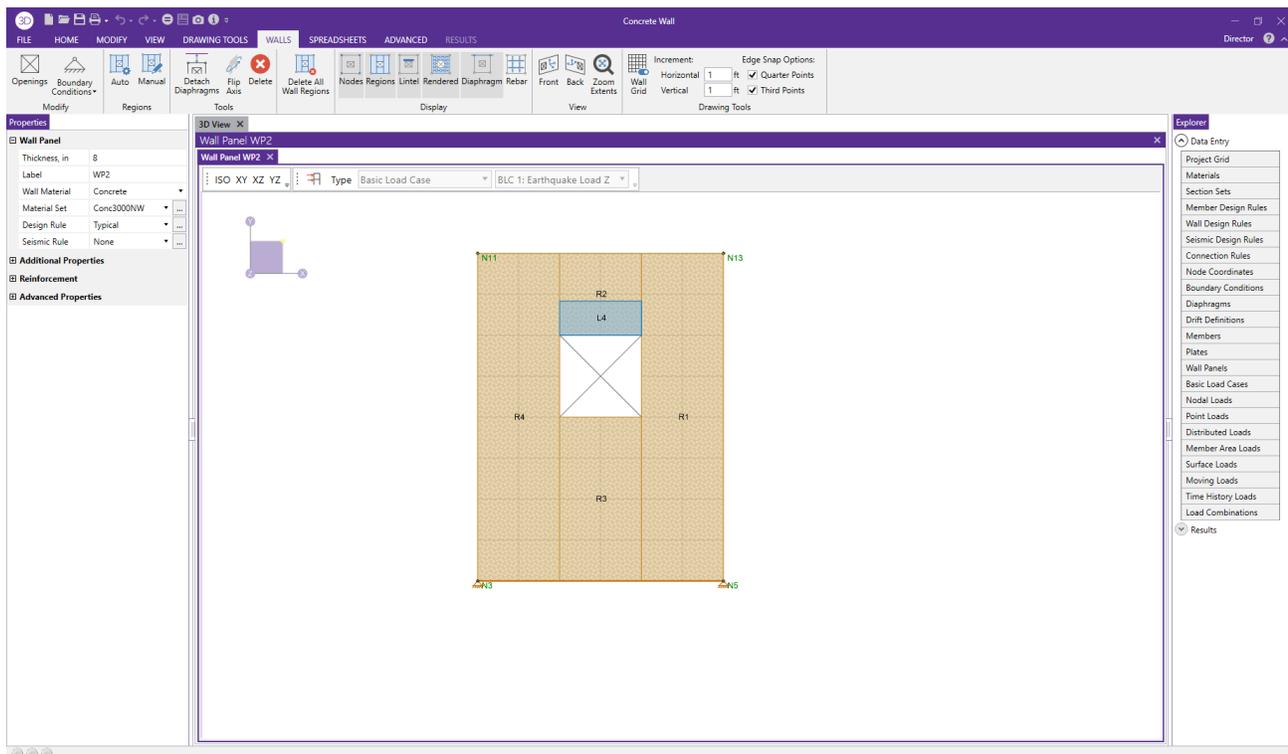
Here we will explain the concrete-specific inputs and design considerations. For general wall panel information, see the [Wall Panels](#) topic. For concrete wall design rule information, see the [Concrete Wall - Design Rules](#) topic. For concrete wall results interpretation, see the [Concrete Wall Results](#) topic.

Concrete Wall Input

To open the Wall Panel Editor:

- Double-click on the wall in the model.

A new window opens for the wall panel. The 'Walls' ribbon toolbar provides viewing options for Region and reinforcement display (after solution). You can also draw in openings, assign boundary conditions, and create regions. The properties of the wall panel can be modified in the **Properties** panel.



Concrete Wall Display Controls

Display Controls

Icon	Control	Description
 Nodes	Show Nodes	This options lets you turn the display of nodes on or off.
 Regions	Show Regions	This option lets you turn the display of regions on or off.

Icon	Control	Description
 Lintel	Show Lintel	This option lets you turn the display of lintels on or off.
 Rendered	Rendered	This option let you toggle the rendering of the wall panel on or off.
 Diaphragm	Show Diaphragm	This option lets you turn the display of diaphragms on or off.
 Rebar	Show Rebar	This option lets you turn the display of reinforcement on or off, after you have solved the model.

Concrete Wall View Controls

View Controls

Icon	Control	Description
 Front	Front	Lets you orient the wall panel so that you are looking at the front (positive X-axis to the right).
 Back	Back	Lets you orient the wall panel so that you are looking at the back (positive X-axis to the left).
 Zoom Extents	Zoom Extents	Lets you reposition the wall panel to fit in the Wall Panel view.

Concrete Wall Drawing Tools

The wall panel grid increment and snap points can be specified in the Drawing Tools section of the Walls tab.

The **Wall Grid** icon (shown below) allows you to modify your snap options, drawing grid in the wall panel editor.



Concrete Wall Regions

Concrete walls depend on Regions for results presentation. The program automatically creates these regions at solution time. If you have a wall panel with no diaphragms then a single region will be created over the entire wall and you will get a single reinforcement design for the entire wall. If there are diaphragms that pass through your wall panel and/or there are openings in the wall, then the wall will be broken up into multiple regions above and below the diaphragms and around the openings, giving a different reinforcement design for each region.

Within each region, the program will optimize the spacing of bars for strength, spacing and minimum reinforcement considerations of the wall.

From within the **Wall Panel Editor**, you have the option of creating rectangular regions within the concrete wall panel. Regions are used to define reinforcement in different parts of the wall. Each region will be assigned a uniform reinforcement, which may be different than the reinforcement in other parts of the same wall (unless you are using [Group Story](#) option from Wall Design Rules).

If no regions have been drawn on a wall then they will be automatically generated when a solution is performed. To automatically generate regions prior to running a solution, click the **Auto** icon (shown below).



Auto

To manually draw regions, select the **Manual** icon (shown below) and use your cursor to select two nodes or grid intersections which make up the diagonal corners of the region. To exit this tool right-click your mouse.



Manual

Code and Reinforcement Considerations

Reinforcement Design and Placement (ACI 318-14/19)

Reinforce Design

The program will design the reinforcement spacing for you. For this design reinforcement spacing, rho, and strength requirements are considered for design. If specific reinforcement is defined in the [Wall Design Rules](#) spreadsheet then it may be possible for the reinforcement design to not meet code requirements.

Sections 25.2 and Chapter 11 have provisions regarding min/max spacing, required reinforcement ratios, and proper proportioning of wall reinforcement. If your wall does not meet a code requirement the program will give a red warning message in the detail report.

Section 25.2 (General Reinforcement Requirements)

The minimum spacing requirements from Section 25.2.1 are considered for design.

Chapter 11 (Wall Reinforcement Detailing)

The maximum spacing requirements from section 11.7.2.1 and 11.7.3.1 are considered for design.



Note: The precast wall spacing requirements of Section 11.7.2.2 and 11.7.3.2 are not considered.

The minimum spacing requirements of Section 11.6.1 are also considered for design. Additionally, the thickness requirement from Section 11.7.2.3 as well as the proportioning and cover checks in that section are also considered.

The program will consider the reinforcement requirements of Section 11.6.2 if the V_u exceeds $0.5 \cdot \phi \cdot V_c$.

ACI 318-11 Reinforcement Design

The program will design the reinforcement spacing for you. For this design reinforcement spacing, ρ , and strength requirements are considered for design. If specific reinforcement is defined in the [Wall Design Rules](#) spreadsheet then it may be possible for the reinforcement design to not meet code requirements.

Sections 7.6, 14.3 and 11.9 all have provisions regarding min/max spacing, required reinforcement ratios, and proper proportioning of wall reinforcement. If your wall does not meet a code requirement the program will then give you a red warning message in the detail report.

Section 7.6 (General Reinforcement Requirements)

The minimum spacing requirements from 7.6.1 and the maximum spacing requirements from section 7.6.5 are considered for design.

Section 14.3 (Wall Reinforcement Requirements)

The minimum spacing requirements of Sections 14.3.2 and 14.3.3 and maximum spacing requirements of 14.3.5 are also considered for design. Additionally, the thickness requirement from Section 14.3.4 as well as the proportioning and cover checks of 14.3.4 are also considered.

Section 11.9.8 and 11.9.9 (Shear Reinforcement Requirements for Walls)

The program will consider the reinforcement requirements of Section 11.9.9 if the V_u exceeds $0.5 \cdot \phi \cdot V_c$ (per Section 11.9.8).

CSA A23.3-14 Reinforcement Design

The program will design the reinforcement spacing for you. For this design reinforcement spacing, ρ , and strength requirements are considered for design. If specific reinforcement is defined in the [Wall Design Rules](#) spreadsheet then it may be possible for the reinforcement design to not meet code requirements.

Clause 14.1.7.1 and 14.1.8 have provisions regarding wall geometry, min/max spacing, required reinforcement ratios, and proper proportioning of wall reinforcement. If your wall does not meet a code requirement the program will then give you a red warning message in the detail report.

Clause 14.1.7.1 (General Wall Geometry Requirements)

The thickness of the walls shall be not less than the smaller of $l_w / 25$ or $h_u / 25$, but not less than 150 mm.

Clause 14.1.8 (Wall Reinforcement Requirements)

The program considers the provisions for reinforcement diameter (14.1.8.2), number of layers (14.1.8.3), max spacing of reinforcement (14.1.8.4), and min reinforcement ratio (14.1.8.5 & 14.1.8.6).

Reinforcement Placement

The reinforcement is designed to meet spacing, rho, and strength requirements. This design may cause the reinforcement spacing design to not fit in the wall region at the exact spacing designed for. Therefore the program will add bars to the extreme ends of the wall region to take these remainders into account.

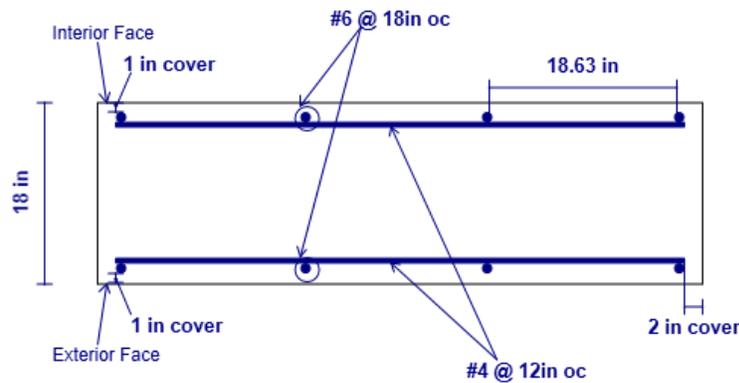
The reinforcement layout algorithm works as follows (picture looking down on a cross section of wall):

First the required spacing is calculated and the wall region length is divided by this spacing.

- a. If two bars cannot fit at this spacing, one bar (each face if specified) will be placed at each end of the wall (meeting cover req'ts).
- b. If only three bars fit at this spacing then one bar (each face if specified) will be placed at each end of the wall (meeting cover req'ts) and one bar in the center of the wall.
- c. If more than three bars are required, then reinforcement is filled in uniformly in the center of the wall and an equal remainder is left between the end bar and the second bar. The image below illustrates this (assume symmetry on both ends of the wall).



Note: This process is only required for very skinny walls. Almost all walls will fall into **Step c** above.

CROSS SECTION DETAILING**Axial Tension**

The axial tensile capacity for a wall assumes all reinforcement is fully developed. The capacity (with no bending interaction) equals:

$$\phi T_n := \phi \cdot n \cdot A_s \cdot f_y$$

where n = number of vertical bars in the wall.

Axial Compression

The axial compressive capacity (with no bending interaction) is taken from equation 10-2.

If CSA A23.3-14 code is selected, Clause 10.10.4 was used to calculate the axial compressive capacity (with no bending interaction).



Note: Slenderness is taken into account per section 6.6.4.5. See the [Second Order Effects](#) section below. CSA A23.3-14 uses the same procedure in considering slenderness effect.

Bending

Both in plane and out of plane capacity consider beam theory in design. For out of plane design (if no axial force) the capacity is simply defined as:

$$a := \frac{A_s \cdot f_y}{0.85 \cdot f_c \cdot b}$$

$$\phi M_n := \phi \cdot A_s \cdot f_y \cdot \left(d - \frac{a}{2} \right)$$



Note: For out of plane reinforcement design, the reinforcement on both faces is taken into account. Thus, the capacity equation above may have two parts to it in order to consider the extreme tension bar and also the bar nearest the compression face that may also be in tension

Minimum Required Moment

ACI 318-14 Section 6.6.4.5.4 (ACI 318-11 Section 10.10.6.5) requires a minimum required moment to be taken into account for each axis in the wall. Thus, if the value from ACI 318-14 Eq 6.6.4.5.4 (ACI 318-11 Eq 10-17) is greater than the calculated moment demand this value will be used. This is meant to account for a minimum eccentricity of the axial force in the wall. CSA A23.3-14 has similar requirements on minimum required moment as defined in 10.15.3.1.

The design moment may be factored up due to [P-Little Delta](#) effects (see below).

Shear

In-Plane

When ACI 318-19 is selected, V_c is calculated using Equation 11.5.4.3. For wall panels subjected to net axial tension, a_s is calculated using Equation 11.5.4.4.

When other ACI 318 editions are selected, V_c is taken into account using ACI 318-14 Detailed Equations from Table 11.5.4.6 (ACI 318-11 Equations 11-27 and 11-28), where:

- $d = 0.8 \cdot l_{wall}$
- N_u , M_u and V_u are taken at the location of maximum shear demand.

When CSA A23.3-14 are selected, V_c is taken into account using Equation 11.6. The factor β is calculated using Equation 11.11 where:

- The longitudinal strain ϵ_x is calculated using Equation 11.13.
- The equivalent crack spacing parameter s_{ze} is calculated using Equation 11.10.

V_s is taken from ACI 318-14 Equation 11.5.4.8 (ACI 318-11 Equation 11-29). Although V_s is only required if $V_u \geq 0.5 \cdot \phi \cdot V_c$, any minimum reinforcement requirement will be used as V_s and added to the shear capacity.

**Note:**

- The maximum V_n per ACI 318-14 Section 11.5.4.3 (ACI 318-11 Section 11.9.3) is also checked.
- In ACI 318-14 Section 11.6.2 (ACI 318-11 Section 11.9.9), the l_w term here is taken as the FULL length of the wall, not the length of the region. ACI 318-14 Section 11.5.4.6 (ACI 318-11 Section 11.9.6) take the l_w to be the length of the region.
- In ACI 318-14 Section 11.6.2 (ACI 318-11 Section 11.9.9) the h_w term here is taken as the height of the region.

When CSA A23.3-14 are selected, V_s is taken from Equation 11.7 where the angle of inclination θ is calculated using Equation 11.12.

Out-Of-Plane

When ACI 318-19 is selected, V_c is calculated using Equation (c) in Table 22.5.5.1. Note that ACI 318-19 code suggests p_w may be taken as the sum of the areas of longitudinal bars located more than two-thirds of the overall member depth away from the extreme compression fiber. Therefore, RISA calculates p_w as the sum of the areas of vertical bars on the tension face.

When other ACI 318 editions are selected, the equations from ACI 318-14 Sections 22.5.6.1 and 22.5.7.1 (ACI 318-11 Section 11.2.15 and 11.2.3) is used for V_c .

When CSA A23.3-14 are selected, V_c is calculated in the same way as In-Plane shear section.

If there are axial tension forces present it is possible for the V_c to go to zero. If axial tension is present the program will use shear friction of the vertical reinforcement in the wall, which will appear as $\phi \cdot V_{ns}$ in the output. $\phi \cdot V_{ns}$ is calculated as follows:

$$A_{tension} := A_b \cdot \left(\frac{P_u}{P_n} \right)$$

$$A_v := A_b - A_{tension}$$

$$\phi V_{ns} := \phi \cdot A_v \cdot f_y \cdot \mu$$

$$\phi V_{nsMAX} := \phi \cdot 2 \cdot \sqrt{f_c} \cdot b \cdot d$$

Thus, the program calculates the portion of the vertical reinforcement that is already engaged in tension and uses the remaining strength to be used for shear friction. The program also sets a maximum shear friction capacity equal to what the concrete wall with no axial force could take. A coefficient of friction, μ , of 1.0 is conservatively assumed.

When CSA A23.3-14 are selected, shear friction is ignored in the shear strength calculation.

SHEAR DETAILS					
UC Max:	0.022	phi*Vn (k):	177.088	Vs (k):	117.81
Location (ft):	4	Vnmax (k):	473.232	Gov LC:	6
Gov Vu (k):	3.867	Vc (k):	118.308		

The larger of ϕV_{nc} and ϕV_{ns} is used in the code check. If there is no axial tension in the wall then $\phi V_{ns} = 0$ and shear friction will not be considered. The intention is to only use shear friction when needed for a case where the wall is loaded in tension and can produce a very small ϕV_{nc} value.

Lambda

Lambda is considered differently for the 2005 and newer codes. For ACI 318-05 the program will always consider $\lambda = 0.75$ if the Density of concrete is ≤ 115 pcf and $\lambda = 1.0$ otherwise (ACI 318-05 Section 11.2.1). For the ACI 318-08 and newer codes, the program will use the Lambda value directly from the [Materials](#) spreadsheet.

Wall Capacity and Deflection Calculations

Deflections

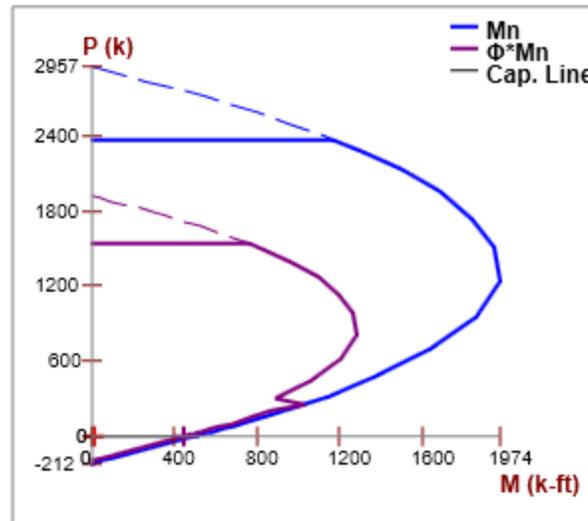
The deflection listed in the detail report is based on the finite element analysis of plate elements. This deflection agrees well with beam theory, thus can be calculated based on beam equations.



Note: If you are checking a hand calculation of the in-plane deflections, be sure to include the deflection due to shear and consider the cracked moment of inertia.

Interaction Diagrams

IN-PLANE WALL INTERACTION DIAGRAM



The program uses a concrete solver to create the interaction diagram and uses this diagram to calculate the capacity of a wall/wall region based on the demand axial force and moment. The program computes the code check based on making a straight line through the origin, the moment/axial force demand location, and where that line crosses the interaction diagram curve. For the out of plane report there is the possibility of different cover spacing at each face of the wall. Thus, the capacities for each face of the wall are reported.

Moment and Axial Force Thresholds

The program will ignore axial forces and moments that are below a certain threshold. If the moment or axial force is deemed to be inconsequential to the code check then the program will simply not include the interaction of that force. There are two thresholds that are considered:

- Axial Force Threshold: If $P_u < 0.01 \cdot f'_c \cdot A_g$ for that LC, then the axial force in the wall region will be ignored for code checks for that LC.
- Moment Threshold: If $M_u < 0.01 \cdot d \cdot P_u$ for that LC, then the moment in the wall region will be ignored for code checks for that LC.

These two thresholds allow the concrete solver to work much more efficiently while having little to no effect on code check values.

Second Order Effects

Per ACI 318-14 Section 6.7.1.1 (ACI 318-11 Section 10.10.4) an elastic second order analysis will satisfy code requirements. In RISA this means running a P-Delta analysis to consider secondary

moments induced due to the displacement of member ends and a P-Little Delta analysis to account for member curvature effects.

The ACI 318-14 requires that either Section 6.6.4, 6.7 or 6.8 are used for sway frames. The ACI 318-08/11 specifications require that either Section 10.10.3, 10.10.4 or 10.10.5 are used. The ACI 318-05 specification requires that you follow provisions of Section 10.13.

Because a P-Delta analysis (big and little) is a more robust analysis the program requires this analysis for all ACI specifications.

P-Delta

The secondary effects due to the displacements of member ends is taken into account with the inclusion of a P-Delta analysis.

To perform a P-Delta Analysis place a "Y" in the **P-Delta** column of the **Load Combinations** spreadsheet. For more information on this, see [P-Delta](#).

Element Curvature Effects (P-Little Delta)

The design moment (max of demand moment and minimum required moment) must be factored up per ACI 318-14 Section 6.6.4.5 (ACI 318-11 Section 10.10.6) if the wall is considered slender.

Slenderness requirements are given in ACI 318-14 Section 6.2.5 (ACI 318-11 Section 10.10.1). The program conservatively only considers ACI 318-14 Equation 6.2.5.a (ACI 318-11 Equation 10-6). Thus, if the KL/r (in either direction) exceeds 22, then the provisions of ACI 318-14 Section 6.6.4.5 (ACI 318-11 Section 10.10.6) are also considered. CSA A23.3-14 uses the same procedure in considering the P-Little Delta effect.

**Note:**

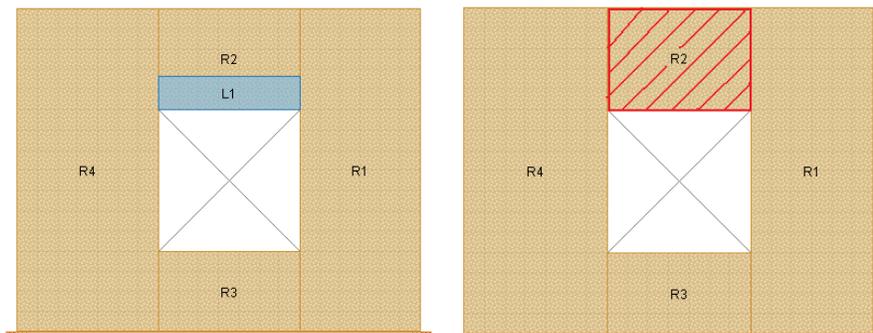
- The M_2 moment will be calculated separately at each section of the wall vertically, rather than using a single M_2 moment for the entire height of the wall.
- If $P_u > 0.75 \cdot P_c$ the program will fail the wall and not give a code check.
- The moment of inertia (I) in the P_c equation is taken conservatively as $0.25 \cdot I_g$ per ACI 318-14 Section R6.6.4.4.4 (ACI 318-11 Section R10.10.6.2).
- C_m is calculated per ACI 318-14 eq. 6.6.4.5.3a (ACI 318-11 eq. 10-16). User may override this and manually input C_m value in Walls spreadsheet - Advanced tab.
- L_u for the wall is either defined as the full height of wall or story (a story is broken up by diaphragms that cross the wall).
- P-Little Delta is only considered for full story height regions. Non full-height regions will give a note and not do P-Little Delta.
- If the controlling load combination experiences tension, the M_c results will be suppressed and show N/A because tension would actually reduce the amplified moment.
- For more information on this, see [P-Little Delta](#).

Concrete Lintel Considerations

The addition of openings into a wall in the wall panel editor will automatically create a lintel above the opening. It will be symbolized by a blue bar that has the name of the lintel inside of it. The program will produce axial, shear and moment diagrams for the lintel that can be viewed from the [Concrete Wall Detail Report](#) from the Lintel drop-down option. Here we will explain some of the different considerations.

How Lintels are Defined

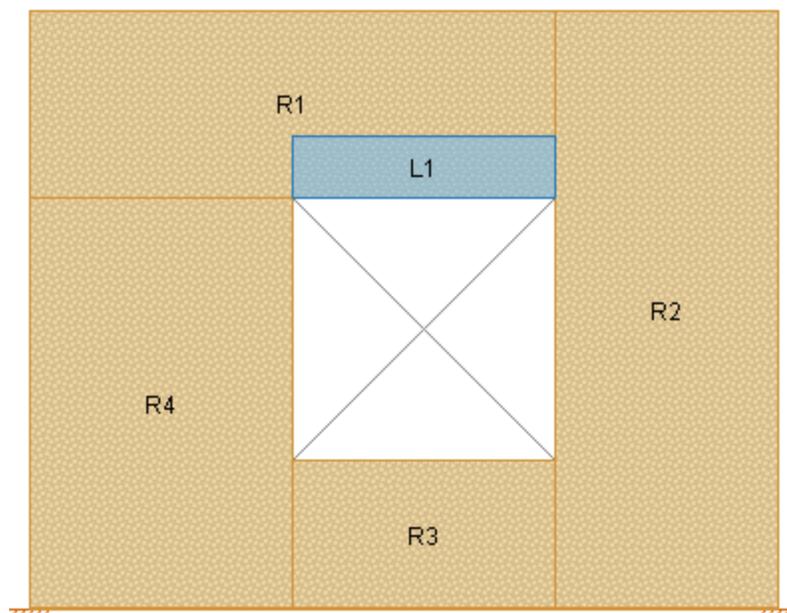
In RISA-3D the lintel is considered as the entire region directly above an opening (L1 below), even though the blue bar does not cover this entire area.



At solution the program will then perform a summation of forces over this entire region and presents the analysis results in the detail report. These results are presented as a "beam" analysis. The program will cut through the entire region vertically multiple times along the length of the region. At each cut the program will calculate the axial, shear and moments. The results from each of these cuts are then combined to form the force diagrams.



Note: Keep in mind that the program only reports results in the detail report if there is a region above the opening AND the width of the region matches exactly the width of the opening. Therefore, a region drawn off-center of the opening will not give lintel results.



Load Attribution

See the [Wall Panels](#) topic for more information.

Modeling Tips

Optimization and Enveloping

Optimization Procedure

The program will use the [Wall Design Rules](#) and start with the maximum spacing and check that configuration for strength, spacing, and minimum reinforcing requirements. If the max spacing works, then the design is done. If not then the program will reduce the spacing by the spacing increment and then do the same checks. This will occur until a bar spacing is reached to satisfy the code requirements.

Enveloping

RISAFloor/RISA-3D Design Enveloping

If you are using RISAFloor and RISA-3D in tandem to do gravity and lateral design, then the program envelopes the results of both. Thus, RISAFloor will never increase the spacing of bars larger than what was required in RISA-3D and vice versa. Also, when moving between programs, the reinforcement spacing that controls will be carried on into the other program.

For example, let's assume RISAFloor required vertical bars that are #6 @ 12" oc. We then use the [Director](#) to take the model to RISA-3D. In RISA-3D, the required vertical bars are #6 @ 8" oc. Now, if we take the model back to RISAFloor and solve again, the #6 @ 8" oc spacing is brought over to RISAFloor and code checks are now based on this spacing.

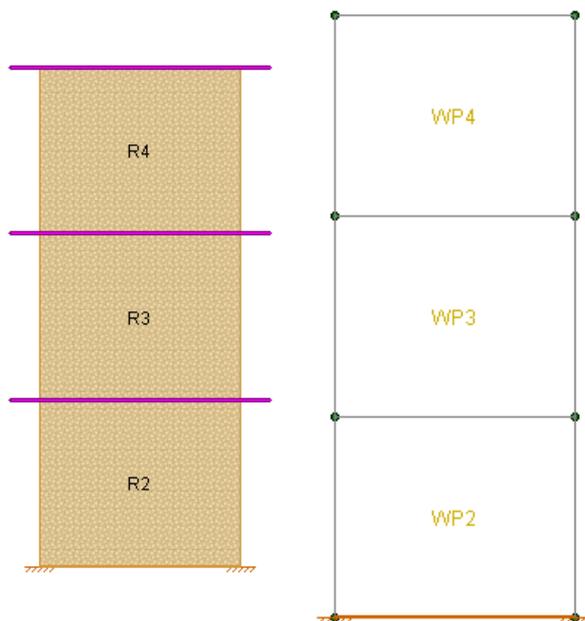


Note: When going back and forth between RISAFloor and RISA-3D, you must solve the model in each of the programs to capture this enveloping. If you do not re-solve, then you are then seeing the results from the previous time you solved in that program without updating for the enveloping.

Multi-Story Walls

For modeling of multi-story shear walls that have diaphragms intersecting, separate regions will be drawn above and below the diaphragms (see Figure 1). The design of each of these regions can be different. This, however, is only true of the spacing of reinforcement. The bar size must be equal for the full-height of the wall, because the [Design Rule](#) is for the entire wall and all regions.

If you want to change bar size over the height of the wall, simply create separate wall panels and stack them on top of one another. In this way you can define different [Design Rules](#) and thus different reinforcement bar sizes up and down the wall (see Figure 2).



For the stacked regions model where the intention is to drop off bars as you work your way up the wall, let's give an example. Let's say that at the base of the wall you have bars at a 4" o.c. spacing. At some point you want to drop off bars to create an 8" o.c. spacing.

All you need to do here is set your [Design Rules](#) such that the Min Vert Bar Spacing is set to 4", the Max Vert Bar Spacing is set to 8" and the Spacing Increment is set to 4".

Local Axes and Wall Cover Dimensions

The program is able to consider different reinforcement cover dimensions for each face of the wall. Because of this the orientation of the wall is important. In the program the **Exterior Face of the wall is oriented in the +z local axis direction**. The **Interior Face of the walls is oriented in the -z local axis direction**.



Note:

- In RISAFloor the wall local axis is irrelevant as we are only doing axial checks.
- If the cover on both faces is identical then the local axis orientation is irrelevant.

A wall drawn in a clock-wise fashion will have its local axis pointed in the positive direction. A wall drawn in a counter-clockwise fashion will have its local axis point in the negative direction.

If the wall local axis is facing the wrong direction then use the Local Axis Flip from the [Modify Walls dialog](#) to correct it.

When bringing a model from RISAFloor to RISA-3D then you want to think about how you are drawing in RISAFloor so that the local axes come in properly in RISA-3D. You will want to draw the walls in in RISAFloor in a counter-clockwise manner to get the local z axes to point outward.

Wall Panels Drawn in RISAFloor in Counter-Clockwise Fashion

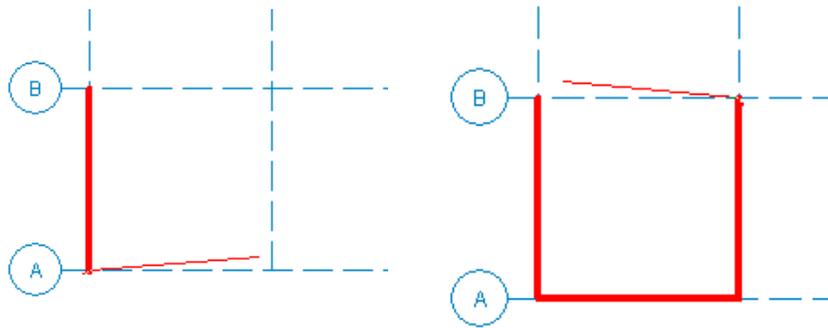
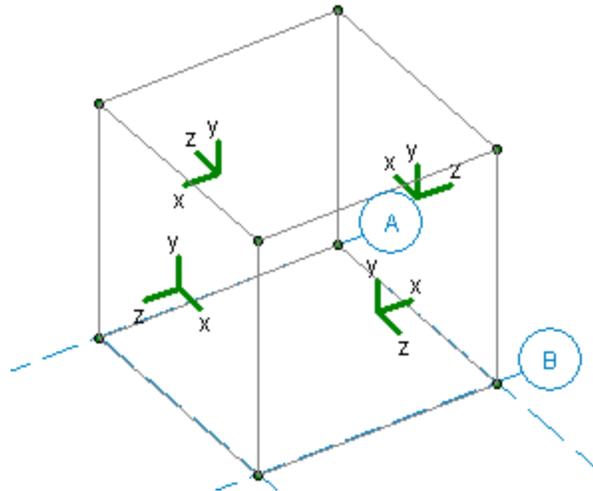


Image of Local Axes in RISA-3D



You can go to [Model Display Options - Panels](#) tab to view the local axes for the wall.

Limitations

- Horizontal reinforcement is designed for in plane shear forces, spacing, and minimum reinforcement checks. They are not used for bending design at this time.
- Walls with bars each face in the wall will require the same size bar and spacing for both faces.

- For sloped walls due to sloping floors, regions cannot be defined in the upper triangular area of the wall panel. Thus, the stiffness of the wall in this area is accurate, but you will get no design results for this portion of the wall.
- Concrete walls are considered completely separately in both the in plane and out of plane directions. Any interaction of the wall, reinforcement, etc., between in plane and out of plane behavior is not considered.
- Reinforcement development is not considered. All reinforcement is assumed to be fully developed.
- ACI 318-14 Chapter 18 (ACI 318-11 Chapter 21) and CSA A23.3-14 Clause 21 provisions for seismic design are currently not considered in the program.

Concrete Wall - Design Rules

The concrete wall panel element allows you to easily model, analyze, and design concrete walls for in plane and out of plane loads. Here we will explain how concrete design rules work. For general wall panel information, see the [Wall Panels](#) topic. For information on concrete wall design considerations, see the [Concrete Wall - Design](#) topic. For concrete wall results interpretation, see the [Concrete Wall Results](#) topic.

Unity Check

Wall Panel U.C. Parameters					
Masonry Lintel	Wood Wall (Studs)	Wood Wall (Fasteners)	CFS Wall (Studs)	CFS Wall (Fasteners)	
Unity Check	Concrete Wall (Rebar)	Concrete Wall (Misc)	Masonry Wall	Masonry In	Masonry Out
	Label	Max Bending Chk	Max Shear Chk		
1	Bearing Wall	1	1		
2	Shear Wall	1	1		

Setting a maximum Bending Check (Axial & Bending) or a maximum Shear Check controls the rebar which the program chooses for the wall design. A value of 0.9 denotes that the program may choose a rebar layout that is at 90% of capacity.



Note: The same unity check parameters are valid for masonry walls as well. However, these parameters are not considered in wood wall design. For wood walls these values are always assumed to equal 1.0.

Concrete Wall (Rebar) Rules

Concrete Wall Panel Rebar Parameters													
Masonry Lintel		Wood Wall (Studs)			Wood Wall (Fasteners)			CFS Wall (Studs)		CFS Wall (Fasteners)			
Unity Check		Concrete Wall (Rebar)			Concrete Wall (Misc)			Masonry Wall		Masonry In		Masonry Out	
	Label	Vert Bar Size	Max Vert Bar Space [in]	Min Vert Bar Space [in]	Vert Bar Inc [in]	Horz Bar Size	Max Horz Bar Space [in]	Min Horz Bar Space [in]	Horz Bar Inc [in]	Group Wall			
1	Bearing Wall	#6	18	4	2	#4	18	4	2	<input type="checkbox"/>			
2	Shear Wall	#6	18	4	2	#4	18	4	2	<input type="checkbox"/>			

Vert and Horz Bar Size

These are the vertical and horizontal bar sizes used for reinforcement of the wall.



Note: The bar size and spacing is assumed to be the same for each face of the wall. Currently reinforcement must be the same for both faces.

Max/Min Vert and Horz Bar Space

The program will design the reinforcement spacing based on these guidelines. If you want the reinforcement to be at an exact spacing, simply enter that spacing as both the min and max in order to force this spacing.

Horz and Vert Bar Increment

This is the spacing change increment that the program will use for design. If the maximum spacing does not work, the spacing will drop by this increment and be checked again. The program will work its way down until it reaches a spacing that meets all reinforcement requirements.

Group Wall

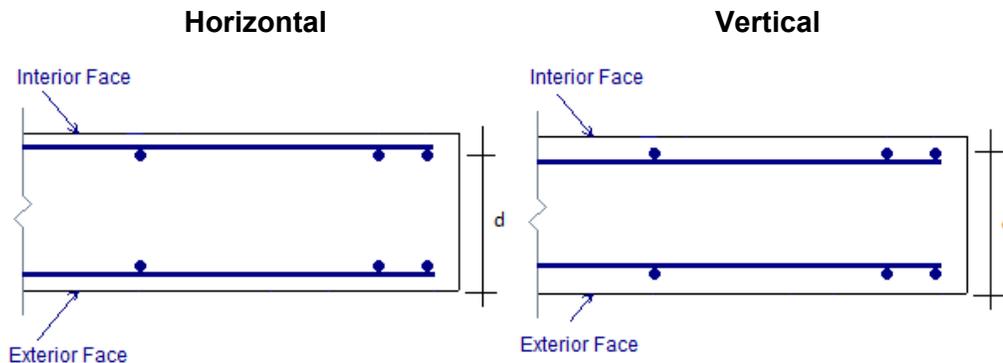
For walls that have multiple regions, this checkbox allows you to group the reinforcement for the regions in a wall. Thus, the worst case vertical and horizontal reinforcement spacing will be used for all regions in the wall.

Concrete Wall (Cover) Rules

Concrete Wall Panel Cover Parameters													
Masonry Lintel		Wood Wall (Studs)			Wood Wall (Fasteners)			CFS Wall (Studs)		CFS Wall (Fasteners)			
Unity Check		Concrete Wall (Rebar)			Concrete Wall (Misc)			Masonry Wall		Masonry In		Masonry Out	
	Label	Outer Bars	Location	Int Cover -z [in]	Ext Cover +z [in]	Edge Cover [in]	Transfer In	Transfer Out					
1	Bearing Wall	Vertical	Each Face	1	1	2	<input type="checkbox"/>	<input type="checkbox"/>					
2	Shear Wall	Vertical	Each Face	1	1	2	<input type="checkbox"/>	<input type="checkbox"/>					

Outer Bars

This defines whether the reinforcement mesh has the Horizontal or Vertical bars closest to the face of concrete. This will affect the "d" calculation for the wall. If the location is Centered then this defines which bar is nearest the outside face of concrete.



Location

Location allows you to locate reinforcement at each face of wall or centered. If the reinforcement is defined as centered then the program places the vertical bar directly at the center of the wall. The horizontal bar is then placed to one side or the other based on the "Outer Bars" designation.



Note: The ACI code requires two curtains of reinforcement if the wall is 10" thick or greater, thus the program will give a warning in the results if you configure your wall like this.

Int Cover (-z)

This is the clear cover distance from the interior face of wall to the outer reinforcement. The interior face of the wall is defined by the negative z local axis direction of the wall.

Ext Cover (+z)

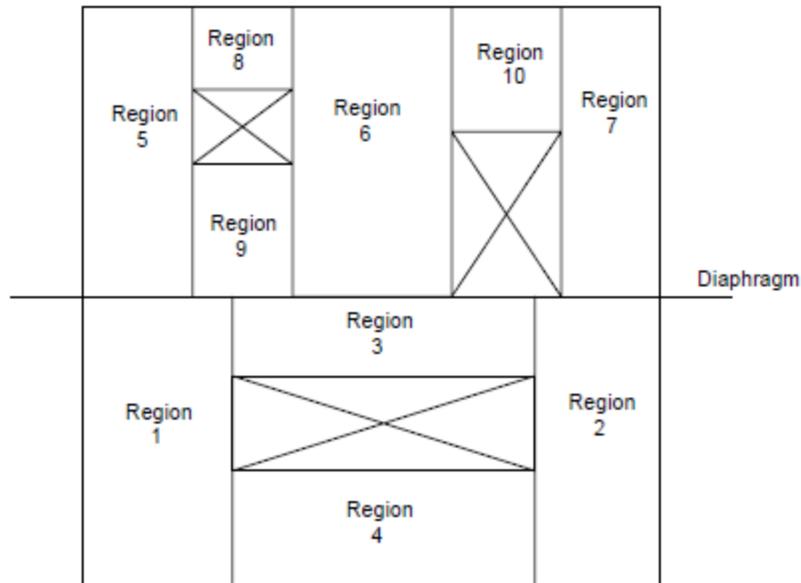
This is the clear cover distance from the exterior face of wall to the outer reinforcement. The exterior face of the wall is defined by the positive z local axis direction of the wall.

Edge Cover

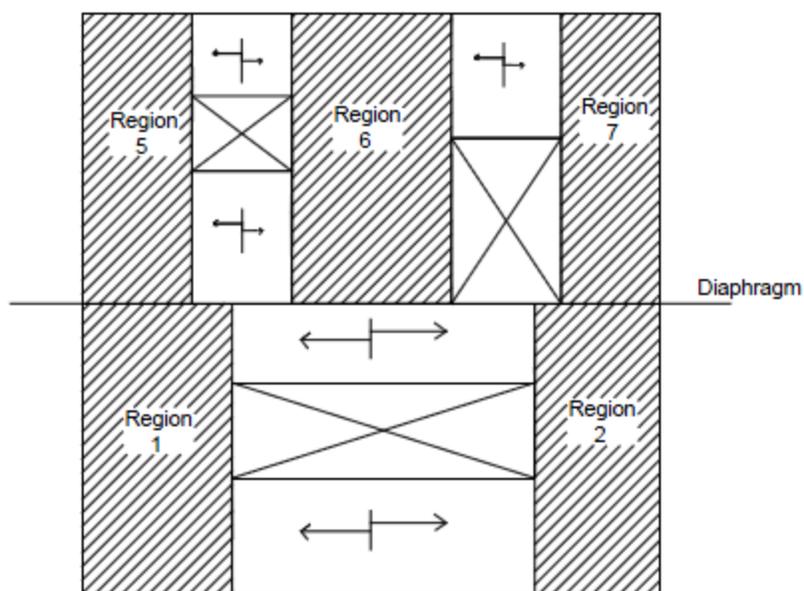
This is the "in plane" cover dimension for the outer edges of walls.

Transfer In and Transfer Out

These options allow you to transfer loads from regions above and below openings to adjacent full-height regions. Transfer in is for in plane loads and transfer out is for out of plane loads. Here is an image of a wall:



If either of the **Transfer** options are turned on for this wall, then any loading in that plane (in plane or out of plane) for regions above and below openings will have their load transferred into the adjacent regions.



A couple of things to keep in mind with the **Transfer** options:

- This is a design-level tool. That is, there is no stiffness change for the model. The program uses the stiffness of all regions for its stiffness. However, after solution, the forces that have accumulated in the regions above/below openings are moved into the adjacent regions. The adjacent region design will then include these forces.
- Results output will not give any information for these "transferred" regions. Only the regions adjacent to the openings will have results.

Concrete Wall Results

Concrete wall results are presented in the "Wall Panel Design" spreadsheet on the *Concrete* tabs, the "Concrete Reinforcing" spreadsheet on the *Concrete Wall* tab and the Detail Reports. Results are reported on a region by region basis.

Design Spreadsheet Results

The **Wall Panel Design** spreadsheet contains two tabs that involve concrete wall design: *Concrete In* and *Concrete Out*. Each tab gives code checks based on the chosen concrete code and can be used as a summary of all of the walls and wall regions in your model. To get detailed information about each region, you can see the [Wall Panel Detail Report](#).



Note:

- Concrete lintel results are not given in the output spreadsheets because the program analyzes them without designing. You must go to the detail report to see lintel analysis results.
- If a wall panel is set to Transfer forces, then regions above and below opening will not have results. See the [Concrete Wall - Design Rules](#) topic for more information.

In Plane

Wall Panel ACI 318-19: Concrete Code Checks (In Plane)											
Masonry Seismic		Wood Wall Axial		Wood Wall In-Plane		Wood Header		CFS Wall Axial		CFS Wall In-Plane	
Concrete In		Concrete Out		Concrete Seismic		Masonry In		Masonry Out		Masonry Lintel	
	Wall Panel	Region	Max UC	LC	Shear UC	LC	Pn*phi[k]	Mn*phi[k-ft]	Vn*phi[k]		
1	WP1	R1	0.017	6	0.018	7	NC	547.043	247.998		
2		R2	0.008	6	0.021	7	NC	253.181	153.601		
3		R3	0.014	6	0.019	6	NC	282.007	153.601		
4		R4	0.025	6	0.016	6	NC	451.617	177.088		
5	WP2	R3	0.012	7	0.013	7	NC	282.007	153.601		
6		R4	0.012	6	0.026	6	NC	445.653	177.088		
7		R2	0.007	6	0.019	6	NC	253.181	153.601		
8		R1	0.02	6	0.008	7	NC	547.043	247.998		

The *Concrete In* tab provides in plane code checks and capacities relevant to the in plane behavior of the wall.

ConcreteWall Spreadsheet - In Plane

Column	Description
Wall Panel	The Wall Panel column displays the concrete wall panels defined in the model.
Region	The Region column lists the concrete wall panel region that the results are based on.
Max UC	The Max UC column shows the maximum code check due to axial force plus in-plane bending.
LC	The LC column adjacent to the Max UC column, shows the load combination that produces each of the highest code check values.
Shear UC	The Shear UC column shows the in-plane shear code check. A value greater than 1.0 for any of these values would indicate failure.
LC	The LC column adjacent to the Shear UC column, shows the load combination that produces each of the highest code check values.
Pn*Phi	The Pn*Phi reports the axial capacity of the wall.
	<div style="border: 1px solid black; padding: 5px; background-color: #e6f2ff;">  Note: An NC means that the axial force in the wall is less than the threshold value, so the axial force is not considered. </div>
Mn*Phi	The Mn*Phi column shows the calculated in plane moment capacity for the region.

Column	Description
Vn*Phi	The Vn*Phi column shows the calculated in plane shear capacity for the region.

Out of Plane

Wall Panel ACI 318-19: Concrete Code Checks (Out Plane)											
Masonry Seismic		Wood Wall Axial		Wood Wall In-Plane		Wood Header		CFS Wall Axial		CFS Wall In-Plane	
Concrete In		Concrete Out		Concrete Seismic		Masonry In		Masonry Out		Masonry Lintel	
	Wall Panel	Region	Max UC	LC	Shear UC	LC	Pn*phi[k/ft]	Mn*phi[k-ft/ft]	Vn*phi[k/ft]		
1	WP1	R1	0.017 (Int)	11	0	14	NC	22.086	6.85		
2		R2	0.002 (Int)	2	0	14	306.213	NC	6.773		
3		R3	0 (Int)	14	0	14	306.213	NC	6.953		
4		R4	0.015 (Ext)	6	0	14	NC	26.205	7.261		
5	WP2	R3	0.007 (Int)	11	0	14	306.213	NC	6.954		
6		R4	0.014 (Ext)	11	0	14	NC	26.205	7.248		
7		R2	0.002 (Int)	11	0	14	306.213	NC	6.771		
8		R1	0.017 (Int)	11	0	14	NC	22.086	6.853		



Note: An NC means that the bending moment in the wall is less than the [threshold value](#), so the axial force is not considered.

The *Concrete Out* tab provides out of plane code checks and capacities relevant to the out of plane behavior of the wall. See the following table for description of the data in each column.

Concrete Wall Spreadsheet - Out of Plane

Column	Description
Wall Panel	The Wall Panel column displays the concrete wall panels defined in the model.
Region	The Region column lists the concrete wall panel region that the results are based on.
Max UC	The Max UC column shows the code check due to axial force plus out of plane bending.
LC	The LC column adjacent to the Max UC column, shows the load combination that produces each of the highest code check values.
Shear UC	The Shear UC column shows the out of plane shear code check. A value greater 1.0 for any of these values would indicate failure.

Column	Description
LC	The LC column adjacent to the Shear UC column, shows the load combination that produces each of the highest code check values.
Pn*Phi	The Pn*Phi reports the axial capacity of the wall. <div style="border: 1px solid black; padding: 5px; background-color: #e6f2ff;">  Note: An NC means that the axial force in the wall is less than the threshold value, so the axial force is not considered. </div>
Mn*Phi	The Mn*Phi reports the calculated out of plane moment capacity for the region. <div style="border: 1px solid black; padding: 5px; background-color: #e6f2ff;">  Note: An NC means that the bending moment in the wall is less than the threshold value, so the axial force is not considered. </div>
Vn*Phi	The Vn*Phi reports the calculated out of plane shear capacity for the region.

Reinforcing Spreadsheet Results

The *Concrete Wall* tab contains reinforcement results for each region in a concrete wall panel.

 **Note:** If a wall panel is set to Transfer forces, then regions above and below opening will not have results. See the [Concrete Wall - Design Rules](#) topic for more information.

Concrete Wall

Concrete Wall Reinforcement						
Beam Bending	Beam Shear	Column Bending	Column Shear	Concrete Wall	Masonry Wall	Masonry Lintel
	Wall	Region	Thickness[in]	Hor. Bar Size	Vert. Bar Size	Diag. Bar Size
1	WP1	R1	18	#4@12in oc (ef)	#6@18in oc (ef)	
2		R2	18	#4@18in oc (ef)	#6@18in oc (ef)	
3		R3	18	#4@18in oc (ef)	#6@18in oc (ef)	
4		R4	18	#4@12in oc (ef)	#6@18in oc (ef)	
5	WP2	R3	18	#4@18in oc (ef)	#6@18in oc (ef)	
6		R4	18	#4@12in oc (ef)	#6@18in oc (ef)	
7		R2	18	#4@18in oc (ef)	#6@18in oc (ef)	
8		R1	18	#4@12in oc (ef)	#6@18in oc (ef)	

The *Concrete Wall* tab displays the thickness, horizontal and vertical reinforcement sizes, and spacing for each region in the concrete wall.

Detail Reports

The detail reports show the overall geometry, analysis, and design for the individual regions/stories of the wall panel. The report also shows envelope diagrams for the forces and moments in the region.

Three basic types of detail reports are provided: Wall Summary, Region and Opening.

Accessing the Detail Reports and the Specific Windows

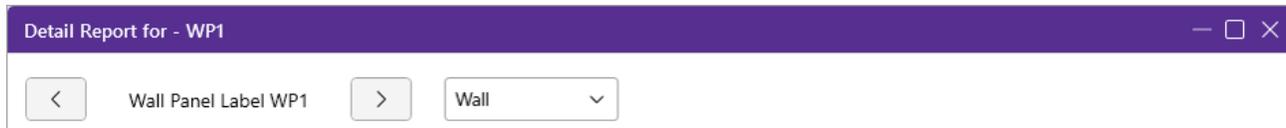
Once you have a solved model, the detail reports become available using either of the following methods:

- If you have the **Wall Panel Design** spreadsheet open, right-click on a wall panel row and choose **Detail Report** from the menu that appears.
- If you are in a graphic view of your model, click the  button on the Selection toolbar and then click on a wall panel.



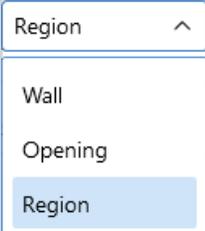
Note: If a wall panel is set to Transfer forces, then regions above and below opening do not have results. See the [Concrete Wall - Design Rules](#) topic for more information.

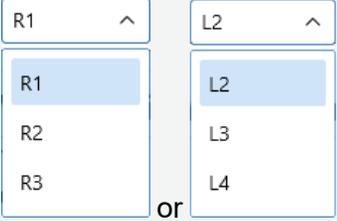
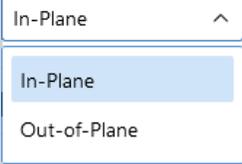
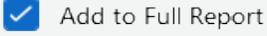
Once the detail report window is open, you will see a dialog area at the top.



The following table describes the options that control the display of the Detail Report.

Detail Report Control Options

Option	Name	Description
	Prior/Next	The left and right arrows let you scroll quickly between the different wall panels in your model.
	Wall Panel Part	The first drop-down list lets you choose between individual Region and Opening (Lintel) results and a summary of the entire Wall.

Option	Name	Description
	<p>Regions/Headers</p>	<p>The second drop-down list lets you select between different Regions or Openings within the individual wall panel. “R” represents regions, while “L” represents levels (headers).</p> <p>The is available only when ‘Opening’ or ‘Region’ is chosen in the Wall Panel Part drop-down.</p>
	<p>In/Out Plane</p>	<p>If you have selected a Region, then you have the option of whether to view the in plane or out of plane report.</p> <p>The is available only when ‘Region’ is chosen in the Wall Panel Part drop-down.</p>
<p>The following options can be found at the bottom of the Detail Report window.</p>		
	<p>Print</p>	<p>Lets you print the Detail Report.</p>
	<p>Add to Full Report</p>	<p>Lets you add the current detail report you are viewing to the printed report. View the Printing topic for more information.</p>

Wall Report

This report provides an overview of the wall, a summary of the controlling code checks and deflection information. This report also displays information about the wall, similar to the [Region Input Echo](#) and also gives an image of the wall. The image shows region locations, wall length and story dimensions, and the nodes that define the corners of the wall panel.

Detail Report for - WP1 — □ ×

< Wall Panel Label WP1 > Wall ▾

Detail Report: WP1

Concrete Wall

CRITERIA	GEOMETRY	MATERIALS
Code: ACI 318-19	Total Height (ft): 10	Material Set: Conc3000NW
Design Rule: Bearing Wall	Total Length (ft): 15	Concrete f'c (ksi): 3
Seismic Rule: None	Thickness (in): 18	Concrete E (ksi): 3156
Loc of r/f: Each Face	Int Cover (-z) (in): 1	Concrete G (ksi): 1372
Outer Bars: Vertical	Ext Cover (+z) (in): 1	Conc Density (k/ft ³): 0.145
Vert Bar Size: #6	Cover Open/Edge (in): 2	Lambda: 1
Horz Bar Size: #4	K: 1	Conc Str Blk: Rectangular
Transfer In?: No	Use Cracked?: Yes	Vert Bar Fy (ksi): 60
Transfer Out?: No	In lcr Factor: 0.7	Horz Bar Fy (ksi): 60
Group Wall?: No	Out lcr Factor: 0.35	Steel E (ksi): 29000

Print
 Add to Full Report
Close

Detail Report for - WP1

Wall Panel Label WP1 Wall

REGION RESULTS

Region	UC Max In Plane	LC	UC Shear In Plane	LC	Delta Max In Plane(in)	LC	UC Max Out Plane	LC	UC Shear Out Plane	LC	Delta Max Out Plane(in)	LC
R1	0.017	6	0.018	7	0.001	6	0.017	11	0	14	0	1
R2	0.008	6	0.021	7	0	6	0.002	2	0	14	0	1
R3	0.014	6	0.019	6	0	6	0	14	0	14	0	1
R4	0.025	6	0.016	6	0.001	7	0.015	6	0	14	0	1

REINFORCEMENT RESULTS

Region	Vertical Reinforcement	Horizontal Reinforcement	Diagonal Reinforcement
R1	#6@18in oc e.f.	#4@12in oc e.f.	N/A
R2	#6@18in oc e.f.	#4@18in oc e.f.	N/A
R3	#6@18in oc e.f.	#4@18in oc e.f.	N/A
R4	#6@18in oc e.f.	#4@12in oc e.f.	N/A

Print Add to Full Report Close

The **Region Results** section gives the tabulated results of all regions in the wall for in plane design axial/bending, shear and deflection for quick reference. You can view the individual region reports to get a more detailed explanation of these values.

The **Reinforcement Results** section gives the reinforcement results for each region in the wall.

Region Report - In Plane

This window gives information for your wall on a region by region basis. The Region detail report is split into five portions: [input echo](#), [diagrams and design](#), [wall section properties](#), [interaction diagrams](#) and [cross section detailing](#).



Note: In RISAFloor, the detail reports are less detailed because RISAFloor does not consider lateral forces.

Input Echo

Below is the input echo portion of the detail report.

Detail Report for - WP1 — □ ×

<
Wall Panel Label WP1
>
Wall

Detail Report: WP1

Concrete Wall

CRITERIA	GEOMETRY	MATERIALS
Code: ACI 318-19	Total Height (ft): 10	Material Set: Conc3000NW
Design Rule: Bearing Wall	Total Length (ft): 15	Concrete f'c (ksi): 3
Seismic Rule: None	Thickness (in): 18	Concrete E (ksi): 3156
Loc of r/f: Each Face	Int Cover (-z) (in): 1	Concrete G (ksi): 1372
Outer Bars: Vertical	Ext Cover (+z) (in): 1	Conc Density (k/ft ³): 0.145
Vert Bar Size: #6	Cover Open/Edge (in): 2	Lambda: 1
Horz Bar Size: #4	K: 1	Conc Str Blk: Rectangular
Transfer In?: No	Use Cracked?: Yes	Vert Bar Fy (ksi): 60
Transfer Out?: No	In lcr Factor: 0.7	Horz Bar Fy (ksi): 60
Group Wall?: No	Out lcr Factor: 0.35	Steel E (ksi): 29000

Print
 Add to Full Report
Close

Criteria	Description
Code	Gives the code used to design the wall.
Design Rule	Gives the design rule used to design reinforcement and cover for the wall.
Loc of r/f	States whether reinforcement is defined at each face of the wall or centered.
Outer Bars	States whether the outer bar in the wall is vertical or horizontal.
Bar Size	States the bar size for both horizontal and vertical reinforcement.
Bar Spacing	States the bar spacing for both horizontal and vertical reinforcement.

Criteria	Description
Group Wall?	States whether the regions in the wall are grouped or not. See the Concrete Wall - Design Rules topic for more information.

Materials	Description
Material Set	States the Material used for the design of the wall.
Concrete f'c	States the compressive strength of the concrete.
Concrete E	States the modulus of elasticity of the concrete.
Concrete G	States the shear modulus of the concrete.
Conc Density	States the unit density of concrete used for self-weight calculations.
Lambda	States the lightweight concrete factor for shear design.
Conc Str Blk	States whether a rectangular (Whitney's) or parabolic stress block was used.
Bar Fy	States the reinforcement strength for both vertical and horizontal bars
Steel E	States the modulus of elasticity for reinforcement.

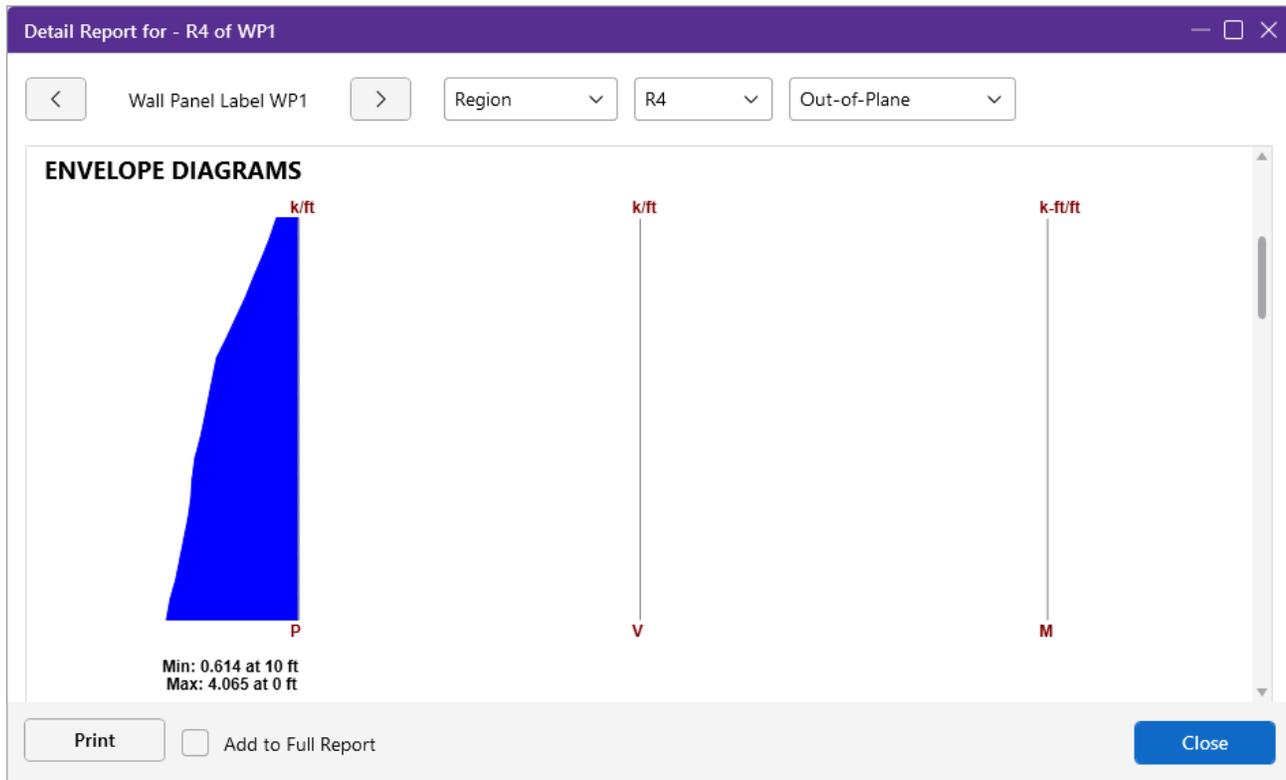
Geometry	Description
Wall Dimensions	States the height, length, and thickness of the wall panel region.
Cover Dimensions	States the interior, exterior, and edge reinf. cover dimensions.
K	States the effective length factor which is used in determining slenderness of the wall.
Use Cracked?	States whether a wall is considered to be cracked or not. Defines whether or not to use cracking in the determination of the moment of inertia.
Icr Factor	States the factor that I _{gross} is multiplied by to get the cracked moment of inertia. This defaults to 0.7 for in plane and 0.35 for out of plane.

Diagrams and Design

In Plane



Out of Plane



Envelope Diagrams

These diagrams show the axial forces, in-plane shear, and in-plane moments of the wall region, as well as the maximum and minimum forces and their locations. The results give an envelope solution of all load combinations .

Because the enveloped results displayed are always the maximum values and because axial and bending forces are checked per their combined effects, the forces in the envelope diagrams won't necessarily be the forces that the wall region is designed to. For example, if there is a high bending moment at the top of a wall region and a high axial force at the base of the wall region, the program will do a check *at each location up the wall region, considering the shear and moment at that location for THAT load combination*. Thus, the maximum axial force given at the location of maximum bending may NOT be the axial force for the LC that produced the maximum bending.

Code Check Summary

This portion of the report gives the capacity and strength values at the section in the wall region where the combined check is maximum, as well as the governing load combination. Much of this information is also reported in the **Concrete Wall Panel Design** spreadsheets.

Axial/Bending Details

The axial and bending capacity are based upon an interaction diagram for the wall region. See below for interaction diagram information. The program computes the code check based on making a straight line through the origin, the moment/axial force demand location and where that line crosses the interaction diagram curve.

For the out of plane report there is the possibility of different cover spacing at each face of the wall region. Therefore the program reports capacities for each face of the wall region. The capacity at the exterior face means that reinforcement at the exterior face of the wall is in tension and vice-versa for the interior face.



Note: The program considers walls to act completely separately in the out of plane direction from the in plane direction. Any out of plane/in plane interaction will need to be taken into account by hand.

AXIAL/BENDING DETAILS

UC Max:	0.025	phi*Pn:	NC	phi eff.:	0.9
Location (ft):	0	Gov Mu (k-ft):	-11.171	Gov LC:	6
Gov Pu (k):	0	phi*Mn (k-ft):	451.617		

Shear Details

The shear details section gives the shear demand required in the wall region. The in plane shear strength of the concrete and steel are listed separately, along with the code-prescribed maximum allowed shear. For out of plane shear design, the simplified equation will be used. For more information on shear capacity of concrete walls, see the [Concrete Wall - Design](#) topic.

SHEAR DETAILS

UC Max:	0.016	phi*Vn (k):	177.088	Vs (k):	117.81
Location (ft):	4	Vnmax (k):	473.232	Gov LC:	6
Gov Vu (k):	2.784	Vc (k):	118.308		

Deflections

The deflection listed in the detail report is based on the finite element analysis of plate elements. This deflection agrees well with beam theory, thus can be calculated based on beam equations.

**Note:**

- If the deflection ratio is larger than L/10000, then L/10000 will be reported.
- If you are checking a hand calculation for in plane deflections, be sure to include the deflection due to shear and consider the cracked moment of inertia.

DEFLECTION DETAILS

Delta max (in):	0.0006496	Location (ft):	10
Deflection Ratio:	H/10000	Gov LC:	7

Wall Section Properties**WALL SEGMENT SECTION PROPERTIES**

Total Length (ft):	5	Icracked (in ⁴):	2.268e+5	KL/r:	6.928
A (in ²):	1080	Cracked Mom, M _{cr} (k-ft):	369.713		
I _{gross} (in ⁴):	3.24e+5	r (in):	14.491		

This section reports the properties used to calculate the wall capacities. The reinforcement details (minimums and provided area) are reported.

Slender Wall Considerations (P-Little Delta)**SLENDER BENDING SPAN RESULTS**

KL/r in	Cm in	Lu in (ft)	Pc (k)	deltaNS	M _{act} (k-ft)	M ₂ min (k-ft)	Mc in (k-ft)
6.928	0	10	0	N/A	0	0	N/A

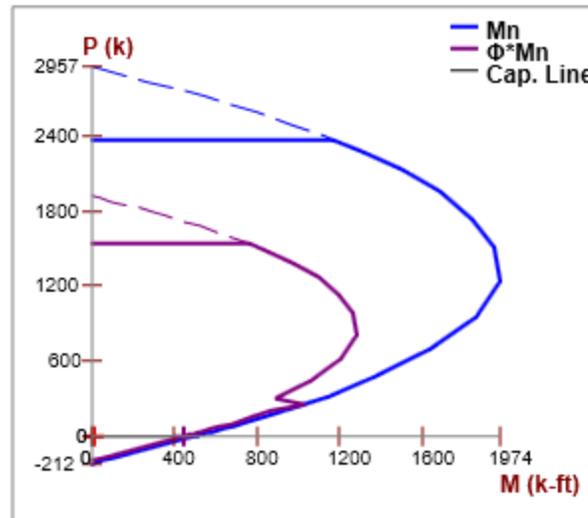
ACI 318-14 Section 6.6.4.5 (ACI 318-11 Section 10.10.6) considers moment magnification of non-sway frames. This is essentially the P-little delta effect in the form of an amplified moment due to the effects of element curvature, M_c . This moment replaces the actual demand moment in design checks. For more information on this see the [Concrete Wall - Design](#) topic.

**Note:**

- M_{act} is the actual demand moment in the wall from analysis. The program compares this to the M_{2min} moment, calculated per ACI 318-14 section 6.6.4.5.4 (ACI 318-11 section 10.10.6.5), and uses the maximum as the M_2 moment.
- Since the moment can be different positive or negative for out of plane bending, the program will provide an Interior and Exterior calculation in this case.
- If the controlling load combination experiences tension, the M_c results will be suppressed and show N/A because tension would actually reduce the amplified moment.

Interaction Diagrams

IN-PLANE WALL INTERACTION DIAGRAM

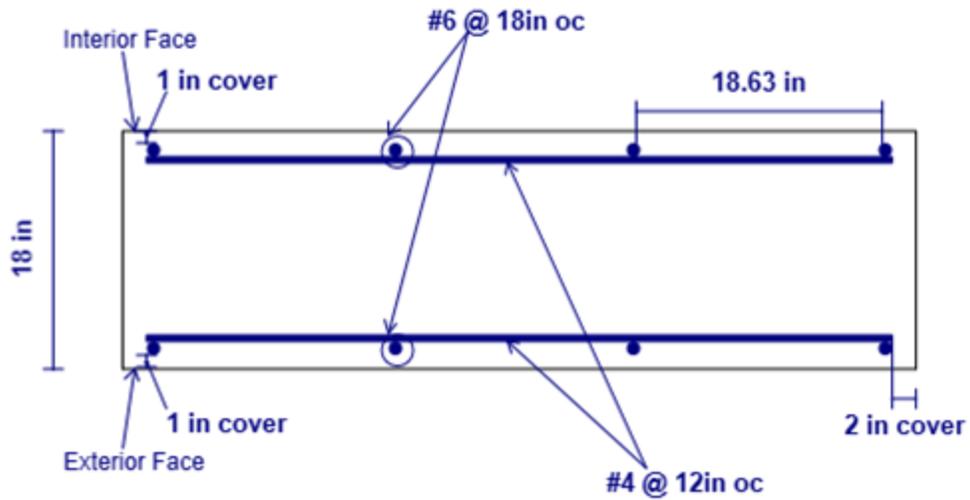


The program uses a concrete solver to create the interaction diagram and uses this diagram to calculate the capacity of each wall region based on the demand axial force and moment. The program then computes the code check based on making a straight line through the origin, the moment/axial force demand location and where that line crosses the interaction diagram curve.

Cross Section Detailing

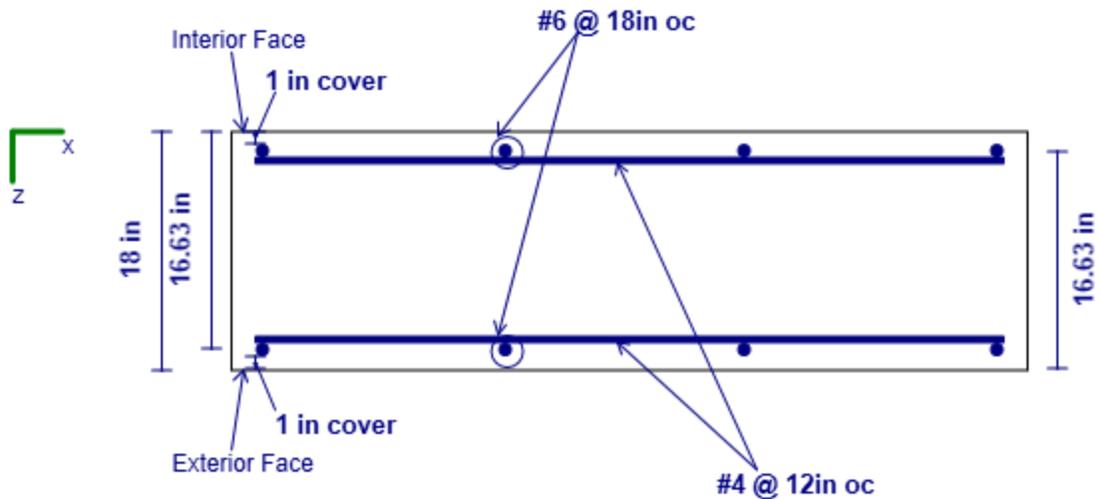
In Plane

CROSS SECTION DETAILING



Out of Plane

CROSS SECTION DETAILING



The last section of the detail report consists of a graphic cross-sectional view of the wall. This view gives cover dimensions, reinforcement size and spacing, and the wall thickness for placement verification.

Concrete Lintel Report

Lintels - Criteria / Materials / Geometry

The right section of the detail report echoes back the basic input parameters (Criteria, Materials, Geometry) entered by the user. An example is shown below:

Detail Report for - L1 of WP1
— □ ×

< Wall Panel Label WP1 >
Opening ▾
L1 ▾
In-Plane ▾

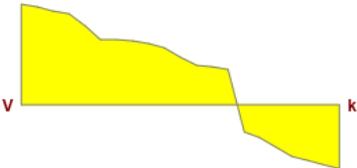
Detail Report: WP1 (In-Plane, Opening L1)

CRITERIA	GEOMETRY	MATERIALS
Code: ACI 318-19	Total Height (ft): 3	Material Set: Conc3000NW
Design Rule: Bearing Wall	Total Length (ft): 4	Concrete f'c (ksi): 3
Seismic Rule: None	Thickness (in): 18	Concrete E (ksi): 3156
Loc of r/f: Each Face	Int Cover (-z) (in): 1	Concrete G (ksi): 1372
Outer Bars: Vertical	Ext Cover (+z) (in): 1	Conc Density (k/ft ³): 0.145
Vert Bar Size: #6	Cover Open/Edge (in): 2	Lambda: 1
Horz Bar Size: #4		Conc Str Blk: Rectangular
Vert Bar Spac (in): 18		Vert Bar Fy (ksi): 60
Horz Bar Spac (in): 18		Horz Bar Fy (ksi): 60
Valid Region: R2		Steel E (ksi): 29000

ENVELOPE DIAGRAMS



Max: 0.708 at 0.6 ft
Min: -1.882 at 3 ft



Max: 4.574 at 0 ft
Min: -2.918 at 4 ft



Max: 2.905 at 1.8 ft
Min: -3.69 at 0 ft

Note: Lintel design is currently not available for concrete wall panel openings.

Print
 Add to Full Report
Close

Since the lintel is not being designed, much of this information is not used but is displayed for reference.

Information	Description
Valid Region	This is the region that the forces are being considered over. The program will sum forces of the region directly above the opening and the forces given in the detail report come from this portion of the wall.

Information	Description
Total Height	This is the height of the region above the opening that forces are being reported for.
Total Length	This is the width of the opening that the lintel is spanning.
Thickness	This is the thickness of the wall.
Other Information	All of the other information is general information for the overall wall.

Lintel Detail Reports - Diagrams

The left section of the detail report provides the enveloped axial, shear and moment diagrams over the length of the lintel. These can be viewed for both in and out of plane forces and are the "beam" forces for the lintel.

Concrete Wall - Seismic Design

Concrete walls can now be designed for seismic design provisions. Here the process is described.



Note: ACI 318-19 (22), ACI 318-19 and ACI 318-14 are the supported codes for this feature and all code references given in this topic are specific to these codes. ACI 318-19 (22) re-approves previous ACI 318-19 without any technical changes, therefore all references in the help documentation and in the program continue to use ACI 318-19.

Seismic Design Rules

Concrete Wall Seismic Design Rule			
Hot Rolled Frame		Concrete Walls	Masonry Walls
	Label	Wall Type	Diagonal Bar Size
1	Ordinary	Ordinary	N/A
2	Intermediate Precast	Intermediate Precast	N/A
3	Special	Special	#3

ConcreteWall Seismic Design Rule

Column	Description
Label	The label is the name for the Wall Seismic Design Rule that is called out for a wall panel.
Wall Type	These are the three wall types supported in RISA-3D.

Column	Description
Diagonal Bar Size	This specifies the diagonal bar size used for shear reinforcement.

The following are the three possible states for the Label and Wall Type columns.

- **Ordinary:** This option adds no extra checks to the wall on top of the regular code checks for a non-seismic consideration per section 18.2.1.6(b). The provisions required to be met are elements not considered in the RISA-3D analysis/design.
- **Intermediate Precast:** This option has provisions specific to wall piers per section 18.5. This Wall Type assumes Seismic Design category D, E or F.
- **Special:** This option has provisions specific to section 18.10.

Shear Force Amplification in ACI 318-19

ACI 318-19 Section 18.10.3.1 requires the special concrete wall to consider the design shear force amplification per Equation (18.10.3.1). RISA calculates the amplification factors Ω_v and ω_v automatically based on the code provisions. The governing Ω_v and ω_v values are provided in the wall panel/region Detail Report. These amplification factors are used to calculate the seismic design force V_e . Note that the combined amplification factors shall not exceed 3.0 per Equation (18.10.3.1).

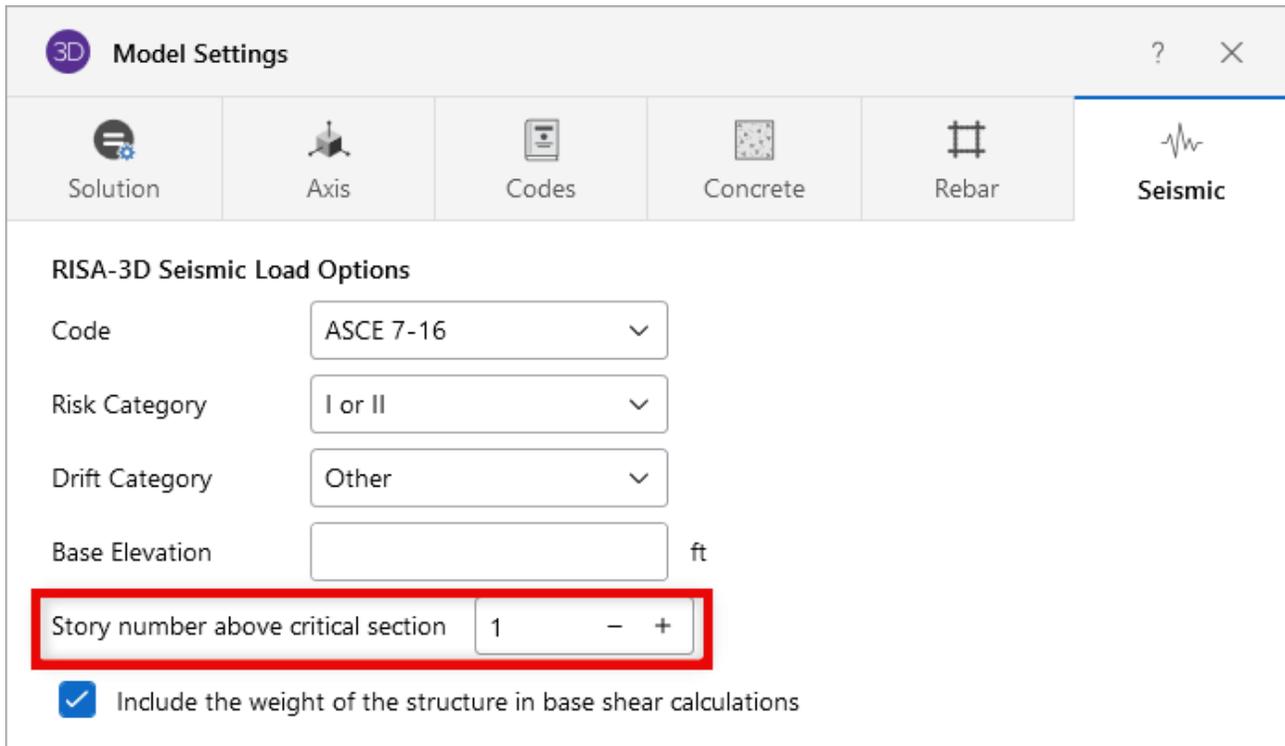
$$V_e = \Omega_v \omega_v V_u \leq 3 V_u \quad (18.10.3.1)$$

Ω_v is calculated per Table 18.10.3.1.2. RISA assumes the lowermost wall panel/region base to be the critical section. In the table, the h_{wcs} refers to the height of the entire structural wall, and l_w is the width of a wall panel/region. M_{pr} is the moment capacity calculated assume steel yield stress is $1.25 \cdot f_y$. The governing combination of M_{pr}/M_u will be used to calculate Ω_v factor. For the wall panel/region at critical section, M_{pr} and M_u values are reported in the Region result of the Detail Report.

Table 18.10.3.1.2—Overstrength factor Ω_v at critical section

Condition	Ω_v	
$h_{wcs}/l_w > 1.5$	Greater of	$M_{pr}/M_u^{[1]}$
		1.5 ^[2]
$h_{wcs}/l_w \leq 1.5$	1.0	

ω_v is calculated per Equation (18.10.3.1.3), the number of story above the critical section n_s is user-input parameter in Global Model Setting-Seismic Tab:



3D Model Settings ? X

Solution Axis Codes Concrete Rebar **Seismic**

RISA-3D Seismic Load Options

Code ASCE 7-16 ▾

Risk Category I or II ▾

Drift Category Other ▾

Base Elevation ft

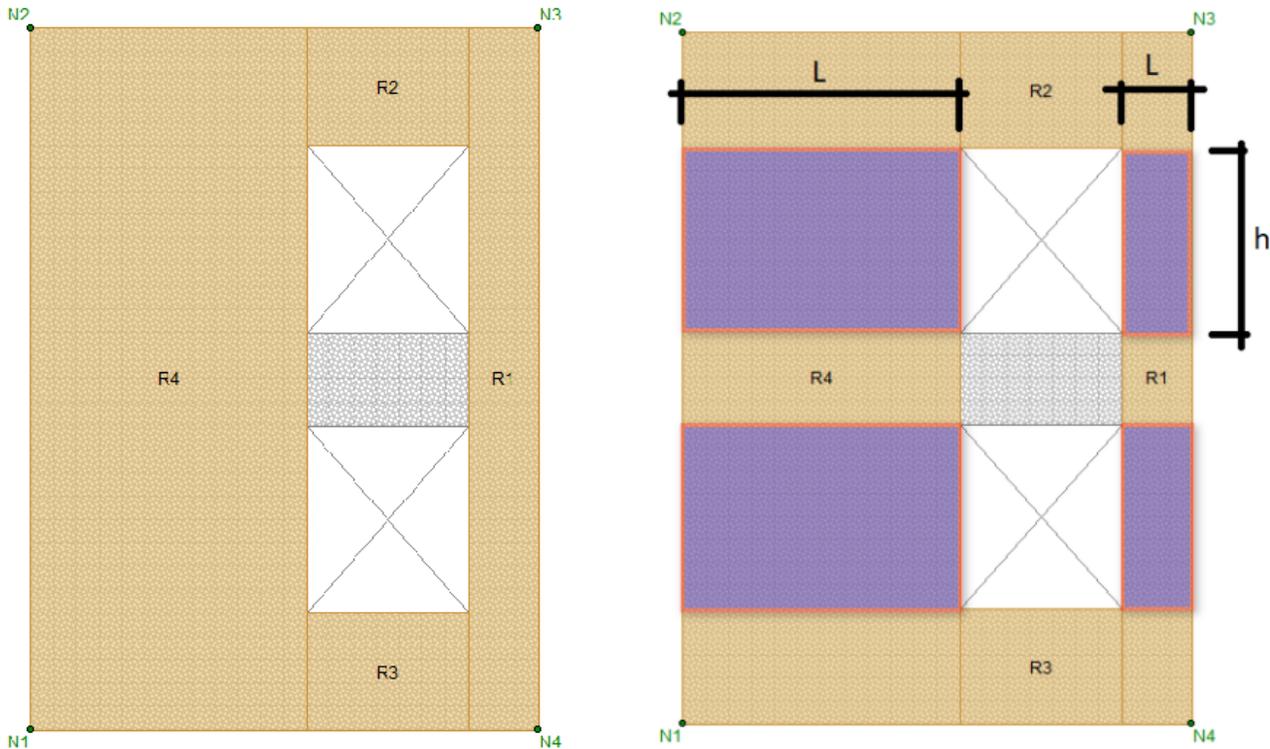
Story number above critical section 1 - +

Include the weight of the structure in base shear calculations

Pier Design

Pier design is governed by section 18.10.8. Section 18.14 is not considered. This design consideration is applicable to 'Intermediate Precast' and 'Special' shear wall types. To determine whether to perform pier checks we use the pier dimensions from Table R18.10.1. Piers must have vertical edges that meet at a wall edge or opening.

The wall below has four piers.



The program is checking the reinforcement in the piers based on Section 18.10.8.1, which references section 18.7.4, 18.7.5 and 18.7.6. If any of these cases gives a failure we will give red text in the detail report regarding this. Specifically, the program checks:

- Rho-min/rho-max per Section 18.7.4.1
- Defining closed hoops per Section 18.7.5.2.a
- Max spacing of vertical bars per Section 18.7.5.2.e
- Cross ties required per Section 18.7.5.2.f
- Max spacing of horizontal bars per Section 18.7.5.3
- Rho-min horizontal per Section 18.7.5.4.
- Wall pier shear checks per Section 18.7.6.1.1.
 - $V_e = 2*M_{pr}/H_p$. M_{pr} is calculated using $f_{ye} = 1.25*f_y$.
 - If $V_u < V_e$, use V_e . Otherwise, use V_u .
- If $(V_e / V_u) > 0.5$ and $P_u < A_g*f_c/20$ then V_c should be taken as zero per Section 18.7.6.2.1.

Coupling Beams

Coupling beam design is governed by section 18.10.7 and is applicable to the 'Special' shear wall type.

Specifically the program checks:

- The program uses a threshold of L/H of 4. If $L/H \geq 4$ there will be no design and you must manually design the coupling beam per Section 18.6. If $L/H < 4$ then the program will design the coupling beam using diagonal reinforcement.
- V_n per equation 18.10.7.4.
- Reinforcement checks per Section 18.10.7.4(d). Section 18.10.7.4(c) is not used because (d) is simpler.
 - Reinforcement must be "Each Face".
 - Outer bars must be "Horizontal".
 - Rho-min as the max of (i) & (ii) per Section 18.10.7.4(d).

Boundary Elements/Zones

Boundary zone checks are based only on section 18.10.6.3. The program performs a check of the maximum extreme fiber compression stress for the worst-case earthquake load combination. If this value exceeds $0.2 \cdot f_c$ then a boundary zone is required. The program will currently tell you if a boundary zone is required or not. It will not design for the provisions of the boundary zone.

Reinforcement Limits for Regions

These checks are for regions that are outside of pier locations.

Specifically the program checks:

- Rho-min Vertical and Horizontal per Section 18.10.2.1.
- Vertical and Horizontal max spacing per Section 18.10.2.1.
- Bar location based on H/L per Section 18.10.2.2.
- V_n -max per Section 18.10.4.1 and 18.10.4.4.
- Rho-min Vertical > Rho-min Horizontal based on H/L per Section 18.10.4.3.
- Rho-max Vertical per Section 18.10.6.5a.

Design Spreadsheet Results

Wall Panel Seismic Detailing

Results for Seismic Detailing checks are provided in the **Wall Panel Design** spreadsheet - **Concrete Seismic** tab. Results are provided per region, and the Seismic Detailing column provides a summary of the Seismic Detailing checks. For a more detailed output, check the "Seismic Detailing" portion of the Wall Panel Detail Report.

Concrete Wall Panel Seismic Code Checks (ACI 318-19)						
Masonry Seismic	Wood Wall Axial	Wood Wall In-Plane	Wood Header	CFS Wall Axial	CFS Wall In-Plane	
Concrete In	Concrete Out	Concrete Seismic		Masonry In	Masonry Out	Masonry Lintel
	Wall Panel	Coupling Beam/Wall Pier	Region/Pier	UC Shear In Plane	LC	BZ Req'd?
1	WP1	Pier	R1-P1	0.917	7	No
2		Coupling Beam	R2/L3	Fail. See Detail Report.		
3	WP2	Pier	R1-P1	1.032	7	No
4		Coupling Beam	R2/L4	Fail. See Detail Report.		

Concrete Wall Panel Seismic Code Checks

Column	Description
Wall Panel	The Wall Panel column displays the concrete seismic wall panels defined in the model.
Coupling Beam/Wall Pier	The Coupling Beam/Wall Pier column displays whether the output is for a coupling beam or a wall pier.
Region/Pier	The Region/Pier column displays the specific region and the associated pier or lintel.
UC Shear In Plane	There is one main shear code check for either piers (per Section 18.7.6.1.1) or coupling beams (per equation 18.10.7.4). If the load combination that controlled the design used the overstrength seismic forces (meaning it had an Ω factor in the load combination) then the governing load combination will be followed by an asterisk (*).
LC	The LC column displays the governing load combination.
BZ Req'd?	The BZ Req'd? column displays whether or not there are boundary elements/zones required. <ul style="list-style-type: none"> If No is displayed, it means there are no boundary elements/zones required. If Yes is displayed, it means boundary elements/zones are required.

Reinforcing Spreadsheet Results

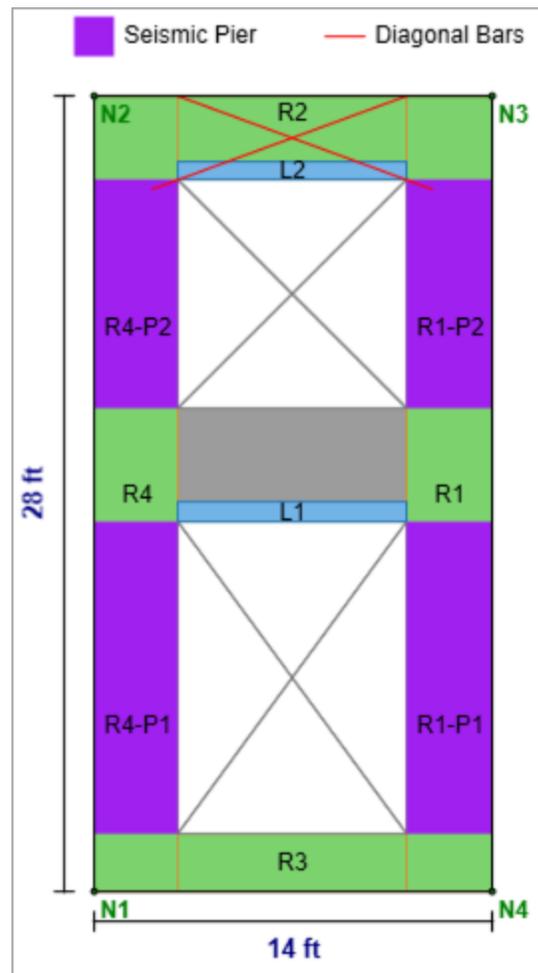
Wall Panel reinforcement is provided in the Concrete Reinforcing Spreadsheet - Concrete Wall tab. Results are provided per region. For a more detailed output, check the "Wall Reinforcement" portion of the Wall Panel Detail Report.

Concrete Wall Reinforcement						
Beam Bending	Beam Shear	Column Bending	Column Shear	Concrete Wall	Masonry Wall	Masonry Lintel
	Wall	Region	Thickness[in]	Hor. Bar Size	Vert. Bar Size	Diag. Bar Size
1	WP1	R1	18	#4@12in oc (ef)	#6@18in oc (ef)	
2		R2	18	#4@4in oc (ef)	#6@8in oc (ef)	Not Designed
3		R3	18	#4@12in oc (ef)	#6@18in oc (ef)	
4	WP2	R1	18	#4@12in oc (ef)	#6@18in oc (ef)	
5		R2	18	#4@4in oc (ef)	#6@8in oc (ef)	Not Designed
6		R3	18	#4@12in oc (ef)	#6@18in oc (ef)	

Detail Report Results

Wall Summary Report

In this report the program gives a color-coded view of the piers and coupling beams in the wall. In the tables the code checks and reinforcement is reported as well.



REGION RESULTS												
Region	UC Max In Plane	LC	UC Shear In Plane	LC	Delta Max In Plane(in)	LC	UC Max Out Plane	LC	UC Shear Out Plane	LC	Delta Max Out Plane(in)	LC
R1	0.111	13	0.11*	13	0.014	13	0.049	4	0	14	0	1
R2	0.009	14	0.04*	13	0.001	13	0	14	0	14	0	1
R3	0.017	13	0.041*	13	0	13	0	14	0	14	0	1
R4	0.119	13	0.093*	13	0.013	14	0.055	13	0	14	0	1
Pier			UC Shear In Plane						LC			
R1-P1			0.548						14			
R1-P2			0.754						13			
R4-P1			0.548						14			
R4-P2			0.754						13			
Coupling Beam			UC Shear In Plane						LC			
R2/L2			0.406						13			
REINFORCEMENT RESULTS												
Region	Vertical Reinforcement				Horizontal Reinforcement				Diagonal Reinforcement			
R1	#6@18in oc e.f.				#5@18in oc e.f.				N/A			
R2/L2	#6@8in oc e.f. Hoops				#5@2in oc e.f.				(4) #3 ea way			
R3	#6@18in oc e.f.				#5@18in oc e.f.				N/A			
R4	#6@18in oc e.f.				#5@18in oc e.f.				N/A			
Pier			Vertical Reinforcement						Horizontal Reinforcement			
R1-P1			#6@5.333in oc e.f.						#5@4in oc e.f. Hoops			
R1-P2			#6@5.333in oc e.f.						#5@4in oc e.f. Hoops			
R4-P1			#6@5.333in oc e.f.						#5@4in oc e.f. Hoops			
R4-P2			#6@5.333in oc e.f.						#5@4in oc e.f. Hoops			

Shear Details - Seismic Tab in ACI 318 - 19

When ACI 318-19 code is selected and special concrete shear walls are used in seismic analysis, a new Shear Details - Seismic tab is added next to the regular Shear Details tab. In this tab, the governing Ω_v value as well as M_{pr} , μ_u values are presented. The amplified shear demand will be used to calculate the new UC values.

Detail Report for - R1 of WP1
— □ ×

<
Wall Panel Label WP1
>
Region
R1
In-Plane

ENVELOPE DIAGRAMS

P

Min: 1.669 at 28 ft
Max: 34.008 at 2.8 ft

V

Min: -3.975 at 15.4 ft
Max: 2.191 at 7 ft

M

Min: -14.797 at 2.8 ft
Max: 7 at 12.6 ft

ACI 318-19 Code Check

AXIAL/BENDING DETAILS					
UC Max:	0.111	phi*Pn:	NC	phi eff.:	0.9
Location (ft):	0	Gov Mu (k-ft):	-14.797	Gov LC:	13
Gov Pu (k):	0	phi*Mn (k-ft):	132.92		

SHEAR DETAILS					
UC Max:	0.037	phi*Vn (k):	108.462	Vs (k):	73.631
Location (ft):	15.4	Vnmax (k):	283.939	Gov LC:	13
Gov Vu (k):	-3.975	Vc (k):	70.985		

SHEAR DETAILS - SEISMIC					
UC Max:	0.11	n:	1	Vnmax (k):	283.939
Location (ft):	15.4	Omega v:	160.137	Vc (k):	70.985
Gov Vu (k):	-3.975	omega v:	1.1	Vs (k):	73.631
Mpr (k-ft):	2877.516	Ve (k):	-11.924	Gov LC:	13
Mu seismic (k-ft):	-17.969	phi*Vn (k):	108.462		

Print
 Add to Full Report
Close

Please note that for each wall panel group (a group of seismic concrete shear walls stacked on each other to resist seismic load), the governing Ω_v value is used for all panels in this group. There may be multiple wall panels/regions at the critical location which produces different Ω_v values, the program only reports the governing Ω_v , Mpr, and Mu values in the detailed reports of each wall panel/region.

Wall Summary Report

In this report the program gives a color-coded view of the piers and coupling beams in the wall. In the tables the code checks and reinforcement is reported as well.

Pier Report

In the pier report the program performs a shear code check and gives results for the reinforcement limits required.

Detail Report for - R1-P1 of WP1
— □ ×

<
Wall Panel Label WP1
>

Pier ▾

R1-P1 ▾

In-Plane ▾

Detail Report: WP1 (Pier R1-P1)

CRITERIA	GEOMETRY	MATERIALS
Code: ACI 318-19	Total Height (ft): 11	Material Set: Conc3000NW
Design Rule: Typical	Total Length (ft): 3	Concrete f'c (ksi): 3
Seismic Rule: Special	Thickness (in): 18	Concrete E (ksi): 3156
Loc of r/f: Each Face	Int Cover (-z) (in): 1	Concrete G (ksi): 1372
Outer Bars: Horizontal	Ext Cover (+z) (in): 1	Conc Density (k/ft ³): 0.145
Vert Bar Size: #6	Cover Open/Edge (in): 2	Lambda: 1
Horz Bar Size: #5 (closed hoops)	K: 1	Conc Str Blk: Rectangular
Vert Bar Spac (in): 5.333	Use Cracked?: Yes	Vert Bar Fy (ksi): 60
Horz Bar Spac (in): 4	Icr Factor: 0.7	Horz Bar Fy (ksi): 60
Group Wall?: No		Steel E (ksi): 29000

ACI 318-19 Seismic Check

SHEAR DETAILS			
UC Max: 0.548	Vnmax (k): 153.82	rho-min: 0.004	
Gov Vu (k): 34.058	Gov LC: 14	hx (in): 30	
phi*Vn (k): 62.126	Ve (k): 34.058	Ach (in ²): 512	
Vc (k): 0	Mpr (k-ft): 187.321	Pu (k): 13.965	
Vs (k): 82.835	smax(horz) (in): 4		

BOUNDARY ELEMENTS

Comp Stress Max (ksi): 0.034	0.2 * f'c (ksi): 0.6
------------------------------	----------------------

Boundary Elements Not Required.

Print
 Add to Full Report

Close

Coupling Beam Report

In the coupling beam report the program defines the shear check based on the diagonal bars and gives overall wall reinforcement checks.

Detail Report for - L2 of WP1
— □ ×

< Wall Panel Label WP1 >
Opening ▾
L2 ▾
In-Plane ▾

Detail Report: WP1 (In-Plane, Opening L2)

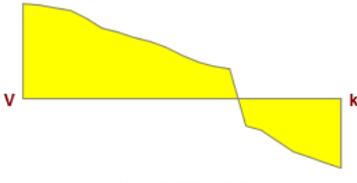
CRITERIA	GEOMETRY	MATERIALS
Code: ACI 318-19	Total Height (ft): 3	Material Set: Conc3000NW
Design Rule: Typical	Total Length (ft): 8	Concrete f'c (ksi): 3
Seismic Rule: Special	Thickness (in): 18	Concrete E (ksi): 3156
Loc of r/f: Each Face	Int Cover (-z) (in): 1	Concrete G (ksi): 1372
Outer Bars: Horizontal	Ext Cover (+z) (in): 1	Conc Density (k/ft ³): 0.145
Vert Bar Size: #6	Cover Open/Edge (in): 2	Lambda: 1
Horz Bar Size: #5		Conc Str Blk: Rectangular
Vert Bar Spac (in): 8		Vert Bar Fy (ksi): 60
Horz Bar Spac (in): 2		Horz Bar Fy (ksi): 60
Valid Region: R2		Steel E (ksi): 29000

ENVELOPE DIAGRAMS



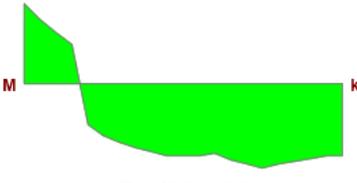
V

Max: 3.421 at 0 ft
Min: 1.263 at 4.4 ft



V

Max: 6.382 at 0 ft
Min: -4.694 at 8 ft



M

Max: 10.17 at 6 ft
Min: -9.431 at 0 ft

Note: Lintel design is currently not available for concrete wall panel openings.

ACI 318-19 Seismic Check

COUPLING BEAM RESULTS			
Region:	R2	Vn (k):	18.615
UC Shear:	0.406	rho-min (H):	0.036
Vu (k):	7.555	smax (H) (in):	2.25
alpha (deg):	20.556	Diag. Bars:	4 - #3's Each Way
		Gov LC:	13
		smax (V) (in):	8
		Ach (in ²):	512

Print
 Add to Full Report
Close

Masonry Wall - Modeling

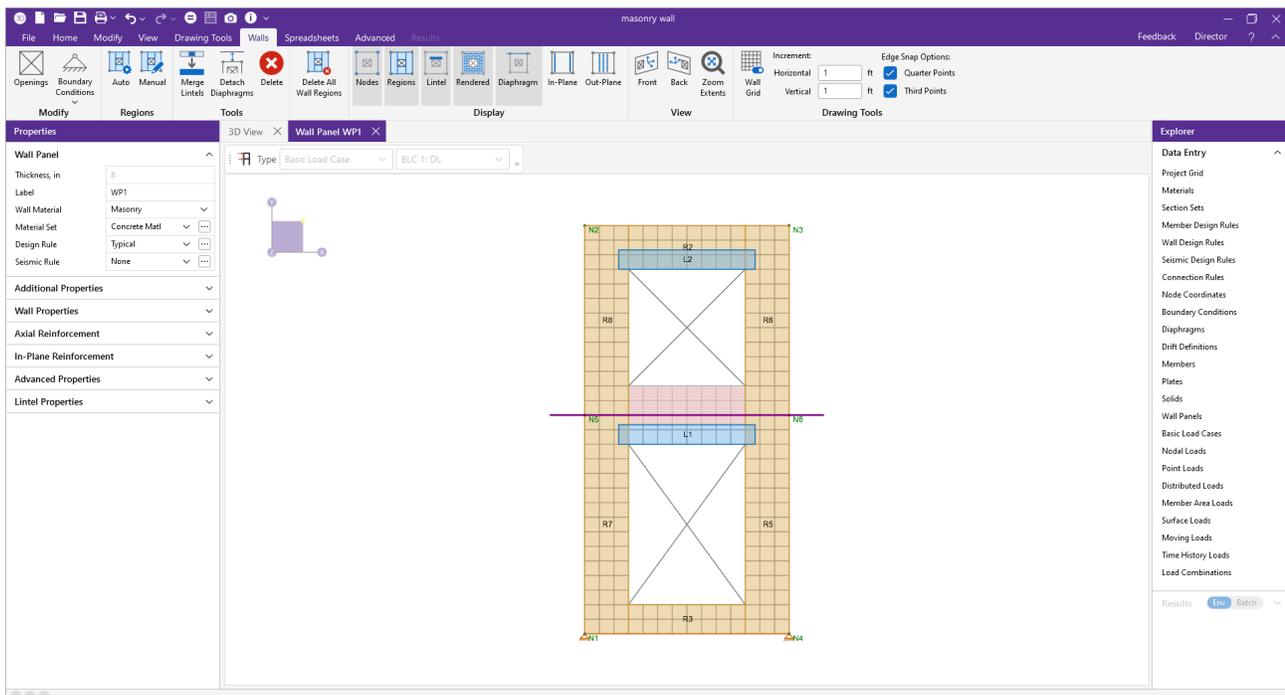
The masonry wall panel element allows you to easily model, analyze and design masonry walls for in plane and out of plane loads. Here we explain the masonry specific inputs and procedures for modeling. For general wall panel information, see the [Wall Panels](#) topic. For information on masonry design rules, see the [Masonry Wall - Design Rules](#) (this is where you can define block thickness and self-weight). For masonry calculation considerations and code references, see the [Masonry Wall - Design](#) topic. For masonry wall results interpretation, see the [Masonry Wall Results](#) topic.



Note: All code references in this topic refer to the **TMS 402-16** specification unless noted otherwise.

Masonry Wall Input

The **Wall Panel Editor** gives some specific information and options for modeling/analysis of masonry walls.



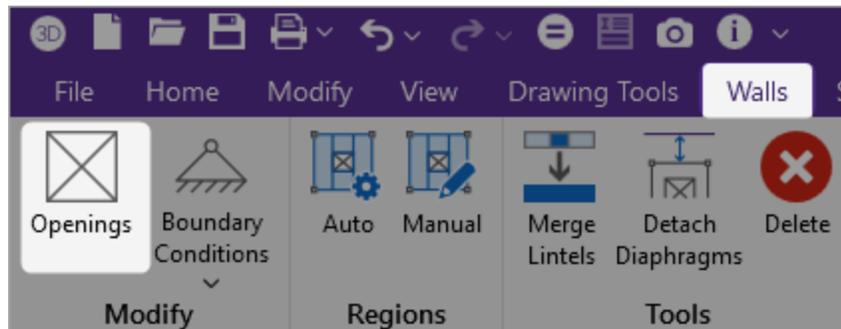
Masonry View Controls

Masonry Wall Panels have the following view controls:

View Controls

Icon	Control	Description
 Regions	Regions	The Regions icon lets you turn the display of regions on or off.
 Lintel	Lintel	The Lintel icon lets you turn the display of lintels on or off.
 Out-Plane	Out-Plane	The Out-Plane icon lets you turn the display of out of plane reinforcement on or off.
 In-Plane	In-Plane	The In-Plane icon lets you turn the display of in-plane reinforcement on or off, after you have solved the model.

Openings / Lintels



In the **Wall Panel Editor**, you have the option of adding rectangular openings to masonry wall panels. To draw an opening, select the **Openings** icon and then select two nodes or grid intersections which make up the two diagonal corners of your opening. When an opening is drawn a lintel is automatically created above the opening. To view or edit the properties of a masonry lintel, double-click inside the boundary of the drawn opening. This brings up the Editing Properties window for that particular lintel.

Masonry Wall Definition Editor
? X

Wall Definition Typical v

Design Rule Label Typical

General/Rebar
Lintel

	Minimum		Maximum	
Bars/Layer	1	to	3	CL Rebar to Lintel Bottom 3.5 in
No. of Rebar Layers	1		v	Bearing Distance 8 in
Bar Size	#5		v	Depth 16 in
Stirrup Size	#4		v	

OK
Cancel

This window shows the design options set in the [Wall Design Rules - Lintel](#) spreadsheet to design/analyze your lintel. If you have multiple lintels in a wall and want a specific design that differs from the other lintels, you can double click the lintel then select "**Convert to Custom...**". When using the custom option, the program uses all of the information set in the Lintel Editor and disregards any information given by the design rule.

The following table includes the different input options available for designing/analyzing lintels.

Lintels Input Options

Control	Description
Depth	Depth is the depth of the lintel.
Bearing Distance	Bearing Distance is the bearing length at either end of the lintel. This is used to calculate the effective length of the lintel.
Bar Size	Bar Size option is the reinforcement size you wish to use for the main reinforcing in the lintel.

Control	Description
Bars Per Layer	Bar Per Layer is the number of bars you wish to have in a given layer of reinforcement. There is also an option to have this value optimized based on geometry of the section, and also the number of layers that you have defined.
Number of Rebar Layers	Number of Rebar Layers lets you have multiple layers of reinforcement in the lintel.
c/c Spacing of Layers	c/c Spacing of Layers is the distance between layers (if there is more than one).
CL Rebar to Lintel Bot	CL Rebar to Lintel Bot is a value used to calculate the “d” value for the lintel.
Stirrup Size	Stirrup Size is the size of stirrup to be added to the lintel if required.



Note: When inputting bar sizes for your lintels, the program will not allow you to place reinforcement that will not actually fit into the lintel because of width constraints. We use the actual dimensions of the block, the face-shell thickness for the given block chosen and use a 1/2" clear cover between reinforcement and the block per section 6.1.3.5.

Regions

Within the Wall Panel Editor, you have the option of creating rectangular regions within the masonry wall panel. Regions are used to define reinforcement in different parts of the wall. Each region will be assigned a uniform reinforcement, which may be different than the reinforcement in other parts of the same wall.

If no regions have been drawn on a wall, then they are automatically generated when a solution is performed.

To automatically generate regions prior to running a solution, click the **Automatic Wall Regions** icon.

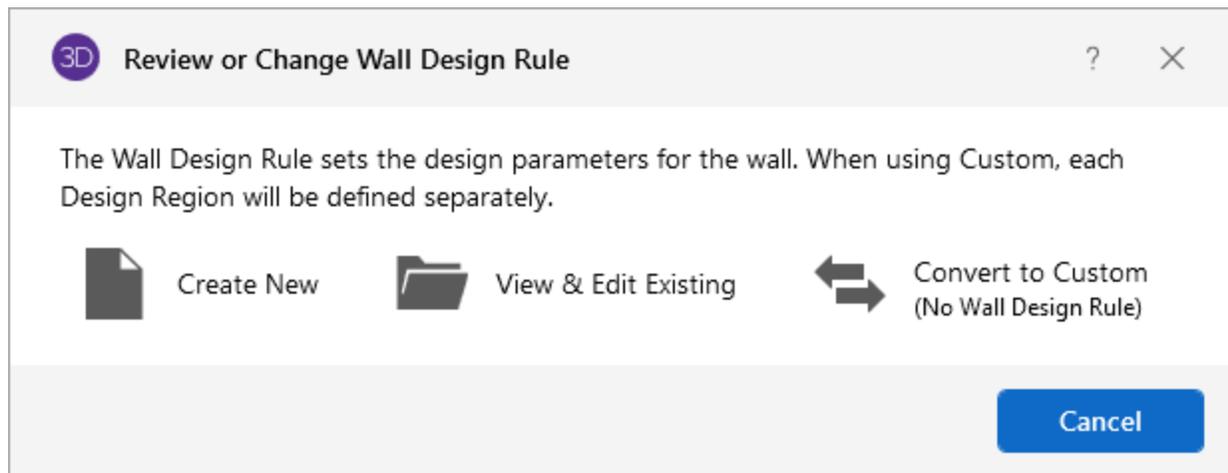


To manually draw regions, select the **Manual Wall Regions** button and use your cursor to select two nodes or grid intersections which make up the diagonal corners of the region. To exit this tool right-



click your mouse. **Manual**

To view or edit the properties of a masonry region, double-click inside the boundary of the drawn region. This brings up the **Review or Change Wall Design Rule** window for that particular region.



You can choose to edit/view the existing wall design rule, or create a new wall design rule. The information is populated from the Wall Design Rules spreadsheet for masonry. For more information see the [Masonry Wall - Design Rules](#) spreadsheet on setting this up.

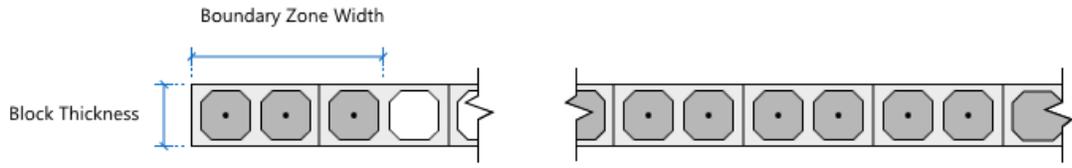
3D Masonry Wall Definition Editor
? X

Wall Definition

Design Rule Label

General/Rebar

Lintel



In-Plane Reinforcement

Boundary Zone Width to in

Bars/Cell

Block Thickness

Vertical Bar Size

Horizontal Bar Size

Axial Reinforcement

Block Grouting

Wall Type

Wall Area Method

Bar/Grout Spacing to

Location

Vertical Bar Size

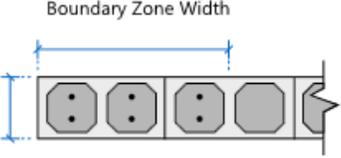
For masonry wall panel regions, you can also customize regions which makes it different than the Wall Design Rule. To use a custom design region, select **Convert to Custom Design Region**. When using the custom option, the program uses all of the information set in the Region Editor and disregards any information given by the design rule for that region.

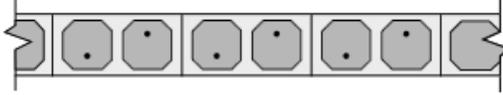
3D Custom Masonry Wall Region Definition Editor ? X

Region

General/Rebar

Boundary Zone Width





In-Plane Reinforcement

Boundary Zone Width in

Bars/Cell ▼

Block Thickness ▼

Vertical Bar Size ▼

Horizontal Bar Size ▼

1.5x Shear Increase

Axial Reinforcement

Block Grouting ▼

Wall Type ▼

Wall Area Method ▼

Bar/Grout Spacing in ▼

Location ▼

Vertical Bar Size ▼

Note: For models created with version 9.1.1 or earlier ALL regions will come in set to Custom, bringing over the information exactly as it comes from the existing model. For all newly created models in version 10 or later the regions will default to the wall design rule.

Within this dialog you can specify the properties which will be used for the design of the region. The program can optimize the bar spacing and the boundary zone width based on code checks. The block size, reinforcing strength and the method of self-weight calculation are defined in the Design Rules under the [Masonry Wall](#) tab.

Page 1284 of 1518



Note: The program designs regions separately for out-of-plane and in-plane forces, thus the Region Editor is divided into different parts.

Here we will walk through the different input options available for designing/analyzing regions:

Transfer - This option lets you choose whether or not you want this region to transfer Out of Plane and In Plane Loads. If you check these transfer options, the program dumps those loads above and below an opening into the adjacent regions for design.



Note:

- These transfer options are only available when you have defined a region above or below an opening.
- If you do not choose to check these transfer options you will see design results for regions above and below openings.

Axial - Lets you to define properties of a region based on axial forces.

- **Block Grouting** - Lets you to define how you want your wall grouted. If you choose "Partially Grouted" the program will optimize the grout spacing with the Bar/Grout Spacing
- **Reinforced** - Gives the option of designing the wall as reinforced or unreinforced.
- **Bar/Grout Spacing** - This Lets you to define the bar/grout spacing. If you have the "Optimize" box checked, the program will optimize the reinforcement spacing based on code checks.
- **Vert Bar Size** - This is the main vertical bar size that will be used in design.
- **Location** - This defines how you want to lay out the reinforcement in your region. Each Face puts reinforcement on both faces of a given cell. Staggered alternates the bars to either face of wall region.



Note: When using the staggered option, you are selecting to space the bars at each face at double the bar/grout spacing defined above. For example, if you have a staggered spacing at 24" oc you have a bar on the outside face at 48" oc and a bar on the inside face at 48" oc. These bars are staggered, thus you have grout filled cores at 24" oc.

In-Plane Reinforcement - Lets you to define properties of a region based on in-plane forces.

- **Vertical Bar Size** - Lets you to define vertical bar size for the boundary zones.
- **Bars Per Cell** - Lets you to define one or two bars per cell.
- **Boundary Zone Width** - You must define a boundary zone width, but RISA optimizes the width if the "Optimize" box is checked.



Note: If you have the optimize checkbox selected, the program optimizes the boundary zone width based on code checks.

- **Horizontal Bar Size** - Lets you define horizontal bar size to be used if horizontal reinforcing is required.
- **1.5x Shear Increase** - This option may be required in high seismic zones. The option is available per section 7.3.2.6.1.2.

Merge Lintels

When an opening is drawn in a masonry wall panel, you will notice that a lintel beam region is automatically created above the opening. If you have multiple openings, you may want to merge the individual lintels into one. To do this, click the **Merge Lintels** icon.



If you have two lintels you want to merge, then click within each of the openings to merge them into one. If you have multiple lintels that you want to merge, click inside of the two openings that define the ends of the merged lintel that you want. You will see that this merges your multiple openings into one. To exit out of this tool right-click your mouse.

When merging lintels, the top edges of the lintels have to be identical. If, once you have merged lintels, you delete one of the openings the entire lintel will be deleted. At that point, you have to delete any openings left in the wall that don't have a lintel over them.

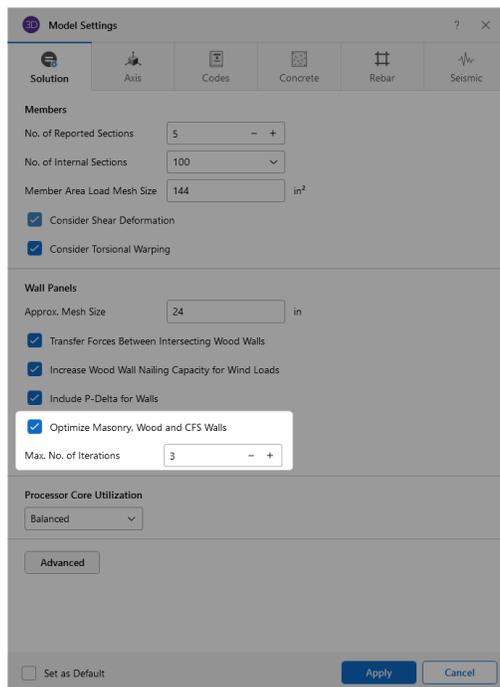
Optimization / Limitations

Optimization

The program optimizes masonry walls and lintels based on the required demand forces. The program can optimize:

- Vertical bar/grout spacing for out of plane design.
- Boundary zone widths for in plane flexural design.
- Horizontal bar spacing for in plane shear design.
- Reinforcement bars for lintel flexural design.

Of these optimizations, the only one that substantially modifies the stiffness of the wall is the vertical bar/grout spacing. To properly adjust the stiffness requires an iterative solution that updates the stiffness of the model. This includes updating the strength properties of the wall as well as the stiffness. This optimization/iteration can be done automatically (by checking the box) or can be done manually (by unchecking the box) in the [Model Settings - Solution](#) tab.



To update the stiffness portion of the wall, the program must re-solve your model with these updated stiffnesses as this will change the distribution of forces through the model. By checking the box you are telling the program to re-solve automatically. Thus, the program will start with its initial stiffness parameters and solve the model. It will then optimize the wall to meet strength criteria. Another solution will then be run with the new stiffnesses and the program will again optimize the wall to meet strength criteria. This procedure will continue to occur until you reach the number of **Iterations** set or until all wall panel results match those of the previous solution.

By unchecking the box the program will only run the solution once and the results will be based on the original configuration. You can then manually optimize your walls using the [Suggested Design](#) spreadsheet.

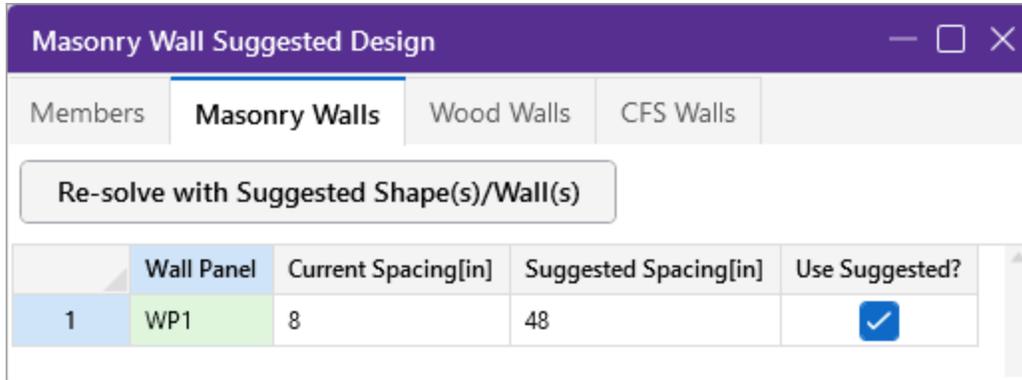
After the solution is run (with or without optimization) the design results are based on the stiffness used in the last iteration (by unchecking the box a single iteration is run). The program will then compare the design of the last iteration with the stiffness used in that last iteration. If the two are the same, the results shown are the final results. If the two are not the same, the program will then provide these two different results in the **Suggested Design** spreadsheet.

The program will always present results in the output that coincide with the stiffness used in the final solution.



Note: The updating of the stiffness for the model is only required for the vertical bar/grout. Thus, boundary zone widths, horizontal shear reinforcement and lintel reinforcement are optimized automatically.

Suggested Design



	Wall Panel	Current Spacing[in]	Suggested Spacing[in]	Use Suggested?
1	WP1	8	48	<input checked="" type="checkbox"/>

In the **Suggested Design** spreadsheet you will get a list of wall panels in your model that are not yet fully optimized, showing the vertical bar and/or grout spacing of the last iteration (Current Spacing) and the program optimized spacing (Suggested Spacing). From here you have the ability to **Use Suggested?** which means that you want to re-run the solution with the Suggested Spacing. You can choose this for each wall panel individually. Once you have these checkboxes checked appropriately, click the icon.



After this, the stiffness matrix is re-formulated and may cause some redistribution of loads through the model. Because of this, the Suggested Spacing may also update and you may need to **Use Suggested?** multiple times to converge on a solution.



Note:

- If the wall does not show up in the Suggested Design spreadsheet, then the current wall panel settings used are the optimal ones.
- For more information on wood wall optimization see the [Wood Wall - Design](#) topic.
- For more information on member optimization see the [Design Optimization](#) topic.
- Concrete walls do not show up here because the reinforcement optimization does not affect the stiffness of the wall.

Lintels

For masonry lintels you must input the dimensions, bar size and number of layers of bars for the lintel, but are given the option of optimizing how many bars are in a given layer. If you provide a max/min number of bars in the [Wall Design Rules - Masonry Lintel](#) tab then the program will optimize the number of bars in a layer. Because this is just a change in reinforcement, this is an automatic optimization that does not require an iterative solution.

Lintel Reinforcement Placement Check

The program will do a check to see that the reinforcement in the lintel will fit properly. The available width for reinforcing in the lintel is based on the width of block, thickness of grout between the block and the reinforcement, and it assumes there will be a double-leg stirrup.

$$W_{available} := W_{Block} - 2 \cdot t_{faceshell} - 2 \cdot t_{grout} - 2 \cdot d_{bstirrup}$$

where

t_{grout} = 0.5 in (per Section 6.1.3.5 of ACI 530-13; the program assumes coarse grout)

$dbstirrup$ = diameter of stirrup (from the [Wall Design Rules - Masonry Lintel](#) tab)

Note that we use a minimum bar spacing equal to the minimum of either d_b or 1".

Regions

For masonry regions, there are two options for optimizing. For out of plane design, the program will optimize the spacing of bars for strength (not deflection) considerations of the wall. For in-plane design, the boundary zone width will be optimized for strength as well. The spacing of reinforcement/grouting affects the overall stiffness of the wall thus you must iterate your solution to update this spacing. The boundary zone design affects the overall stiffness very little, thus this is an automatic optimization that does not require an iterative solution. If you provide a min/max boundary zone width in the [Wall Design Rules - Masonry In](#) tab, then the program will optimize the boundary zone width.

Limitations

- Openings and regions must be input only in RISAFloor if you are using RISA-3D and RISAFloor in tandem.
- For sloped walls due to sloping floors, openings and regions can not be defined in the upper triangular area of the wall panel. These openings are not supported at this time. This will be addressed in a future version.
- For areas of masonry wall panels that are not defined as a region, the stiffness of the wall is assumed to be that of an ungrouted masonry wall and the weight of the wall is assumed to be that of a fully grouted wall.
- Masonry has specific requirements regarding the spreading out of concentrated loads at certain angles and for certain depths (Section 5.1.3). These provisions are not considered in RISA, as the finite element mesh stiffness is what determines load path. See the [Wall Panels - Load Attribution](#) topic for more information.

Masonry Wall - Modeling

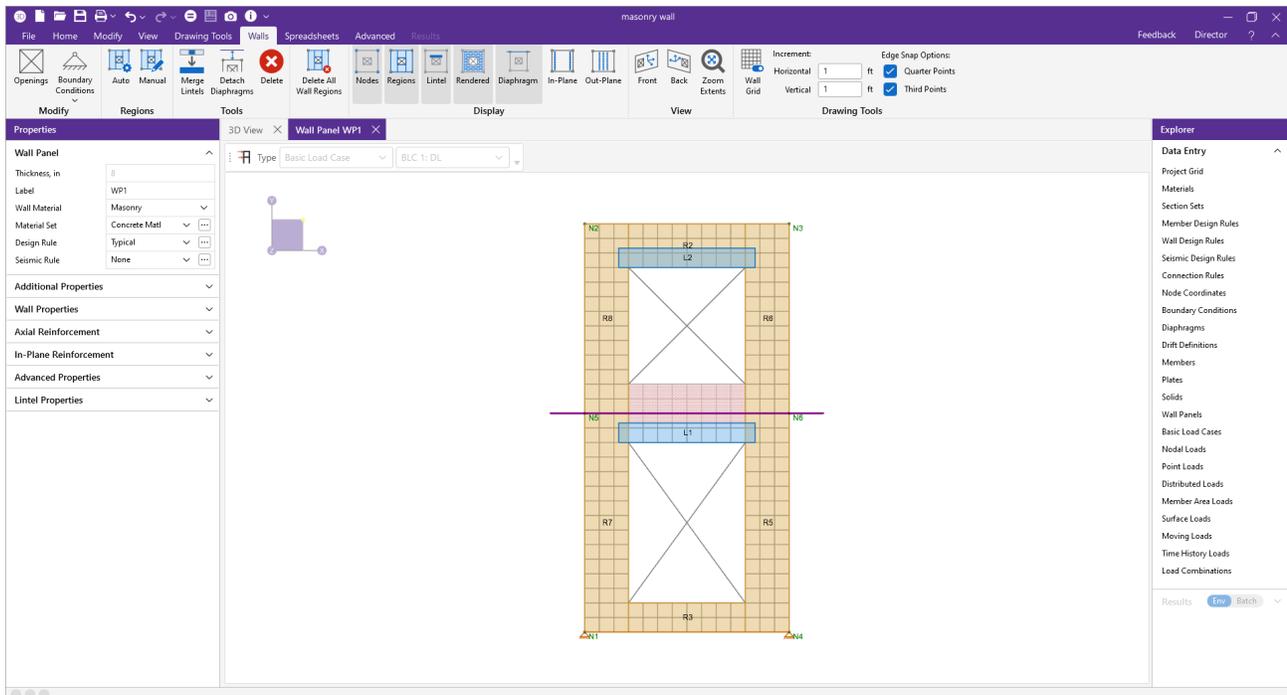
The masonry wall panel element allows you to easily model, analyze and design masonry walls for in plane and out of plane loads. Here we explain the masonry specific inputs and procedures for modeling. For general wall panel information, see the [Wall Panels](#) topic. For information on masonry design rules, see the [Masonry Wall - Design Rules](#) (this is where you can define block thickness and self-weight). For masonry calculation considerations and code references, see the [Masonry Wall - Design](#) topic. For masonry wall results interpretation, see the [Masonry Wall Results](#) topic.



Note: All code references in this topic refer to the **TMS 402-16** specification unless noted otherwise.

Masonry Wall Input

The **Wall Panel Editor** gives some specific information and options for modeling/analysis of masonry walls.



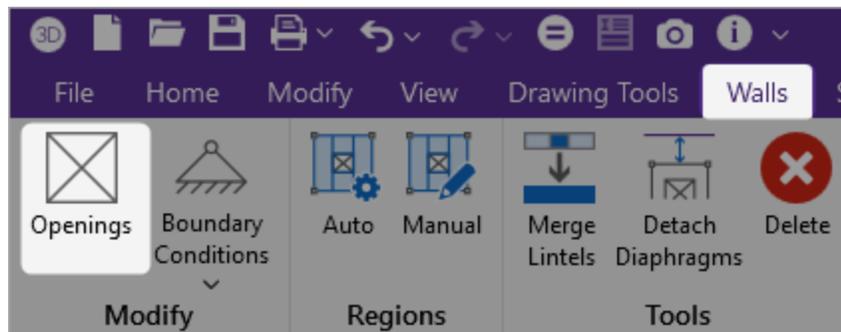
Masonry View Controls

Masonry Wall Panels have the following view controls:

View Controls

Icon	Control	Description
 Regions	Regions	The Regions icon lets you turn the display of regions on or off.
 Lintel	Lintel	The Lintel icon lets you turn the display of lintels on or off.
 Out-Plane	Out-Plane	The Out-Plane icon lets you turn the display of out of plane reinforcement on or off.
 In-Plane	In-Plane	The In-Plane icon lets you turn the display of in-plane reinforcement on or off, after you have solved the model.

Openings / Lintels



In the **Wall Panel Editor**, you have the option of adding rectangular openings to masonry wall panels. To draw an opening, select the **Openings** icon and then select two nodes or grid intersections which make up the two diagonal corners of your opening. When an opening is drawn a lintel is automatically created above the opening. To view or edit the properties of a masonry lintel, double-click inside the boundary of the drawn opening. This brings up the Editing Properties window for that particular lintel.

Masonry Wall Definition Editor
? X

Wall Definition

Design Rule Label

General/Rebar

Lintel

	Minimum		Maximum		
Bars/Layer	<input type="text" value="1"/>	to	<input type="text" value="3"/>	CL Rebar to Lintel Bottom	<input type="text" value="3.5"/> in
No. of Rebar Layers	<input type="text" value="1"/>		▼	Bearing Distance	<input type="text" value="8"/> in
Bar Size	<input type="text" value="#5"/>		▼	Depth	<input type="text" value="16"/> in
Stirrup Size	<input type="text" value="#4"/>		▼		

OK

Cancel

This window shows the design options set in the [Wall Design Rules - Lintel](#) spreadsheet to design/analyze your lintel. If you have multiple lintels in a wall and want a specific design that differs from the other lintels, you can double click the lintel then select "**Convert to Custom...**". When using the custom option, the program uses all of the information set in the Lintel Editor and disregards any information given by the design rule.

The following table includes the different input options available for designing/analyzing lintels.

Lintels Input Options

Control	Description
Depth	Depth is the depth of the lintel.
Bearing Distance	Bearing Distance is the bearing length at either end of the lintel. This is used to calculate the effective length of the lintel.
Bar Size	Bar Size option is the reinforcement size you wish to use for the main reinforcing in the lintel.

Control	Description
Bars Per Layer	Bar Per Layer is the number of bars you wish to have in a given layer of reinforcement. There is also an option to have this value optimized based on geometry of the section, and also the number of layers that you have defined.
Number of Rebar Layers	Number of Rebar Layers lets you have multiple layers of reinforcement in the lintel.
c/c Spacing of Layers	c/c Spacing of Layers is the distance between layers (if there is more than one).
CL Rebar to Lintel Bot	CL Rebar to Lintel Bot is a value used to calculate the “d” value for the lintel.
Stirrup Size	Stirrup Size is the size of stirrup to be added to the lintel if required.



Note: When inputting bar sizes for your lintels, the program will not allow you to place reinforcement that will not actually fit into the lintel because of width constraints. We use the actual dimensions of the block, the face-shell thickness for the given block chosen and use a 1/2" clear cover between reinforcement and the block per section 6.1.3.5.

Regions

Within the Wall Panel Editor, you have the option of creating rectangular regions within the masonry wall panel. Regions are used to define reinforcement in different parts of the wall. Each region will be assigned a uniform reinforcement, which may be different than the reinforcement in other parts of the same wall.

If no regions have been drawn on a wall, then they are automatically generated when a solution is performed.

To automatically generate regions prior to running a solution, click the **Automatic Wall Regions** icon.



Auto

To manually draw regions, select the **Manual Wall Regions** button and use your cursor to select two nodes or grid intersections which make up the diagonal corners of the region. To exit this tool right-



click your mouse. **Manual**

To view or edit the properties of a masonry region, double-click inside the boundary of the drawn region. This brings up the **Review or Change Wall Design Rule** window for that particular region.



You can choose to edit/view the existing wall design rule, or create a new wall design rule. The information is populated from the Wall Design Rules spreadsheet for masonry. For more information see the [Masonry Wall - Design Rules](#) spreadsheet on setting this up.

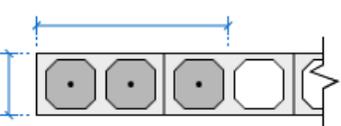
3D Masonry Wall Definition Editor
? X

Wall Definition Typical ▼

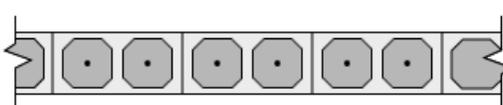
Design Rule Label R8

General/Rebar Lintel

Boundary Zone Width



Block Thickness



In-Plane Reinforcement

Boundary Zone Width 8 to 40 in

Bars/Cell 1 ▼

Block Thickness 8" ▼

Vertical Bar Size #5 ▼

Horizontal Bar Size #5 ▼

Axial Reinforcement

Block Grouting Partially Grouted ▼

Wall Type Reinforced ▼

Wall Area Method NCMA ▼

Bar/Grout Spacing 8" ▼ to 48" ▼

Location Center ▼

Vertical Bar Size #5 ▼

OK
Cancel

For masonry wall panel regions, you can also customize regions which makes it different than the Wall Design Rule. To use a custom design region, select **Convert to Custom Design Region**. When using the custom option, the program uses all of the information set in the Region Editor and disregards any information given by the design rule for that region.

3D Custom Masonry Wall Region Definition Editor ? X

Region

General/Rebar

Boundary Zone Width

Block Thickness

In-Plane Reinforcement

Axial Reinforcement

Boundary Zone Width <input style="width: 60px;" type="text" value="8"/> in	Block Grouting <input style="width: 100px;" type="text" value="Partially Grouted"/>
Bars/Cell <input style="width: 60px;" type="text" value="1"/>	Wall Type <input style="width: 100px;" type="text" value="Reinforced"/>
Block Thickness <input style="width: 60px;" type="text"/>	Wall Area Method <input style="width: 100px;" type="text" value="NCMA"/>
Vertical Bar Size <input style="width: 60px;" type="text" value="#5"/>	Bar/Grout Spacing <input style="width: 60px;" type="text" value="48"/> in
Horizontal Bar Size <input style="width: 60px;" type="text" value="#5"/>	Location <input style="width: 100px;" type="text" value="Center"/>
<input type="checkbox"/> 1.5x Shear Increase	Vertical Bar Size <input style="width: 60px;" type="text" value="#5"/>

Note: For models created with version 9.1.1 or earlier ALL regions will come in set to Custom, bringing over the information exactly as it comes from the existing model. For all newly created models in version 10 or later the regions will default to the wall design rule.

Within this dialog you can specify the properties which will be used for the design of the region. The program can optimize the bar spacing and the boundary zone width based on code checks. The block size, reinforcing strength and the method of self-weight calculation are defined in the Design Rules under the [Masonry Wall](#) tab.



Note: The program designs regions separately for out-of-plane and in-plane forces, thus the Region Editor is divided into different parts.

Here we will walk through the different input options available for designing/analyzing regions:

Transfer - This option lets you choose whether or not you want this region to transfer Out of Plane and In Plane Loads. If you check these transfer options, the program dumps those loads above and below an opening into the adjacent regions for design.



Note:

- These transfer options are only available when you have defined a region above or below an opening.
- If you do not choose to check these transfer options you will see design results for regions above and below openings.

Axial - Lets you to define properties of a region based on axial forces.

- **Block Grouting** - Lets you to define how you want your wall grouted. If you choose "Partially Grouted" the program will optimize the grout spacing with the Bar/Grout Spacing
- **Reinforced** - Gives the option of designing the wall as reinforced or unreinforced.
- **Bar/Grout Spacing** - This Lets you to define the bar/grout spacing. If you have the "Optimize" box checked, the program will optimize the reinforcement spacing based on code checks.
- **Vert Bar Size** - This is the main vertical bar size that will be used in design.
- **Location** - This defines how you want to lay out the reinforcement in your region. Each Face puts reinforcement on both faces of a given cell. Staggered alternates the bars to either face of wall region.



Note: When using the staggered option, you are selecting to space the bars at each face at double the bar/grout spacing defined above. For example, if you have a staggered spacing at 24" oc you have a bar on the outside face at 48" oc and a bar on the inside face at 48" oc. These bars are staggered, thus you have grout filled cores at 24" oc.

In-Plane Reinforcement - Lets you to define properties of a region based on in-plane forces.

- **Vertical Bar Size** - Lets you to define vertical bar size for the boundary zones.
- **Bars Per Cell** - Lets you to define one or two bars per cell.
- **Boundary Zone Width** - You must define a boundary zone width, but RISA optimizes the width if the "Optimize" box is checked.



Note: If you have the optimize checkbox selected, the program optimizes the boundary zone width based on code checks.

- **Horizontal Bar Size** - Lets you define horizontal bar size to be used if horizontal reinforcing is required.
- **1.5x Shear Increase** - This option may be required in high seismic zones. The option is available per section 7.3.2.6.1.2.

Merge Lintels

When an opening is drawn in a masonry wall panel, you will notice that a lintel beam region is automatically created above the opening. If you have multiple openings, you may want to merge the individual lintels into one. To do this, click the **Merge Lintels** icon.



If you have two lintels you want to merge, then click within each of the openings to merge them into one. If you have multiple lintels that you want to merge, click inside of the two openings that define the ends of the merged lintel that you want. You will see that this merges your multiple openings into one. To exit out of this tool right-click your mouse.

When merging lintels, the top edges of the lintels have to be identical. If, once you have merged lintels, you delete one of the openings the entire lintel will be deleted. At that point, you have to delete any openings left in the wall that don't have a lintel over them.

Optimization / Limitations

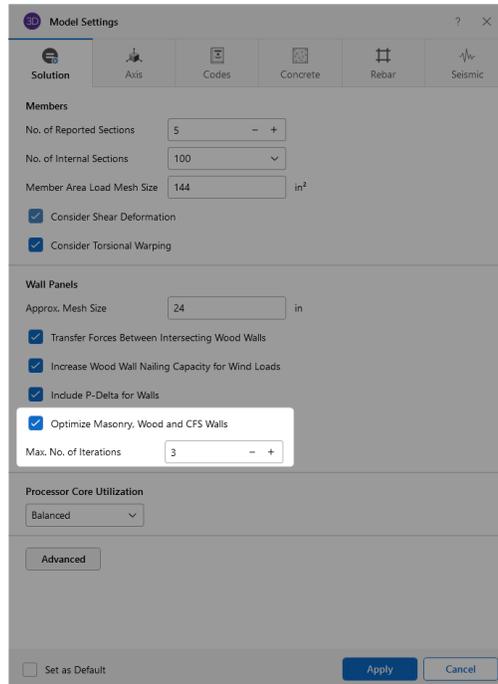
Optimization

The program optimizes masonry walls and lintels based on the required demand forces. The program can optimize:

- Vertical bar/grout spacing for out of plane design.
- Boundary zone widths for in plane flexural design.
- Horizontal bar spacing for in plane shear design.
- Reinforcement bars for lintel flexural design.

Of these optimizations, the only one that substantially modifies the stiffness of the wall is the vertical bar/grout spacing. To properly adjust the stiffness requires an iterative solution that updates the stiffness of the model. This includes updating the strength properties of the wall as well as the

stiffness. This optimization/iteration can be done automatically (by checking the box) or can be done manually (by unchecking the box) in the [Model Settings - Solution](#) tab.



To update the stiffness portion of the wall, the program must re-solve your model with these updated stiffnesses as this will change the distribution of forces through the model. By checking the box you are telling the program to re-solve automatically. Thus, the program will start with its initial stiffness parameters and solve the model. It will then optimize the wall to meet strength criteria. Another solution will then be run with the new stiffnesses and the program will again optimize the wall to meet strength criteria. This procedure will continue to occur until you reach the number of **Iterations** set or until all wall panel results match those of the previous solution.

By unchecking the box the program will only run the solution once and the results will be based on the original configuration. You can then manually optimize your walls using the [Suggested Design](#) spreadsheet.

After the solution is run (with or without optimization) the design results are based on the stiffness used in the last iteration (by unchecking the box a single iteration is run). The program will then compare the design of the last iteration with the stiffness used in that last iteration. If the two are the same, the results shown are the final results. If the two are not the same, the program will then provide these two different results in the **Suggested Design** spreadsheet.

The program will always present results in the output that coincide with the stiffness used in the final solution.



Note: The updating of the stiffness for the model is only required for the vertical bar/grout. Thus, boundary zone widths, horizontal shear reinforcement and lintel reinforcement are optimized automatically.

Suggested Design

	Wall Panel	Current Spacing[in]	Suggested Spacing[in]	Use Suggested?
1	WP1	8	48	<input checked="" type="checkbox"/>

In the **Suggested Design** spreadsheet you will get a list of wall panels in your model that are not yet fully optimized, showing the vertical bar and/or grout spacing of the last iteration (Current Spacing) and the program optimized spacing (Suggested Spacing). From here you have the ability to **Use Suggested?** which means that you want to re-run the solution with the Suggested Spacing. You can choose this for each wall panel individually. Once you have these checkboxes checked appropriately, click the icon.



After this, the stiffness matrix is re-formulated and may cause some redistribution of loads through the model. Because of this, the Suggested Spacing may also update and you may need to **Use Suggested?** multiple times to converge on a solution.



Note:

- If the wall does not show up in the Suggested Design spreadsheet, then the current wall panel settings used are the optimal ones.
- For more information on wood wall optimization see the [Wood Wall - Design](#) topic.
- For more information on member optimization see the [Design Optimization](#) topic.
- Concrete walls do not show up here because the reinforcement optimization does not affect the stiffness of the wall.

Lintels

For masonry lintels you must input the dimensions, bar size and number of layers of bars for the lintel, but are given the option of optimizing how many bars are in a given layer. If you provide a max/min number of bars in the [Wall Design Rules - Masonry Lintel](#) tab then the program will optimize the number of bars in a layer. Because this is just a change in reinforcement, this is an automatic optimization that does not require an iterative solution.

Lintel Reinforcement Placement Check

The program will do a check to see that the reinforcement in the lintel will fit properly. The available width for reinforcing in the lintel is based on the width of block, thickness of grout between the block and the reinforcement, and it assumes there will be a double-leg stirrup.

$$W_{available} := W_{Block} - 2 \cdot t_{faceshell} - 2 \cdot t_{grout} - 2 \cdot d_{bstirrup}$$

where

t_{grout} = 0.5 in (per Section 6.1.3.5 of ACI 530-13; the program assumes coarse grout)

$d_{bstirrup}$ = diameter of stirrup (from the [Wall Design Rules - Masonry Lintel](#) tab)

Note that we use a minimum bar spacing equal to the minimum of either d_b or 1".

Regions

For masonry regions, there are two options for optimizing. For out of plane design, the program will optimize the spacing of bars for strength (not deflection) considerations of the wall. For in-plane design, the boundary zone width will be optimized for strength as well. The spacing of reinforcement/grouting affects the overall stiffness of the wall thus you must iterate your solution to update this spacing. The boundary zone design affects the overall stiffness very little, thus this is an automatic optimization that does not require an iterative solution. If you provide a min/max boundary zone width in the [Wall Design Rules - Masonry In](#) tab, then the program will optimize the boundary zone width.

Limitations

- Openings and regions must be input only in RISAFloor if you are using RISA-3D and RISAFloor in tandem.
- For sloped walls due to sloping floors, openings and regions can not be defined in the upper triangular area of the wall panel. These openings are not supported at this time. This will be addressed in a future version.
- For areas of masonry wall panels that are not defined as a region, the stiffness of the wall is assumed to be that of an ungrouted masonry wall and the weight of the wall is assumed to be that of a fully grouted wall.

- Masonry has specific requirements regarding the spreading out of concentrated loads at certain angles and for certain depths (Section 5.1.3). These provisions are not considered in RISA, as the finite element mesh stiffness is what determines load path. See the [Wall Panels - Load Attribution](#) topic for more information.

Masonry Wall - Design

The masonry wall panel element allows you to easily model, analyze and design masonry walls for in plane and out of plane loads for these specific codes:

- The 2016 TMS 402 ASD and 2013, 2011, 2008, 2005, 2002 and 1999 Editions of ACI 530 ASD
- The 2016 TMS 402 Strength and 2013, 2011, 2008, 2005, 2002 and 1999 Editions of ACI 530 Strength
- The 1997 edition of the UBC for ASD and Strength

Here we will explain the calculation concepts and code references used in the program. For general wall panel information, see the [Wall Panels](#) topic. For information on masonry design rules, see the [Masonry Wall - Design Rules](#) (this is where you can define block thickness and self-weight). For masonry wall modeling procedures, see the [Masonry Wall - Modeling](#) topic. For masonry wall results interpretation, see the Masonry Wall Results topic.



Note:

- All code references in this topic refer to the **TMS 402-16** specification unless noted otherwise.
- References will be made to RMEH and NCMA. For more information on these designations, see the [Masonry Wall - Design Rules](#) topic.
- Walls designed in RISA meet all of the requirements for Ordinary Masonry Shear Walls except for the Minimum Reinforcement requirements of TMS 402-16 Section 7.3.2.3.1. That provision should be checked by hand outside of RISA.

Shear Design - In Plane ASD

In Plane Shear Stress, f_v

This stress is calculated from Equation 8-24:

$$f_v := \frac{V}{A_{nv}}$$

where:

- **V** = Total in plane shear in the wall region
- **Anv** = net shear area

In RISA **Anv** is defined as follows:

$$A_{nvRMEH} := Eq_{solidthickness} \cdot d_{inplane}$$

$$A_{nvNCMA} := \frac{A_{nvNCMAperfoot}}{\left(\frac{12 \text{ in}}{1 \text{ ft}}\right)} \cdot d_{inplane}$$

where:

- **d_{inplane}** = distance from compression face to centroid of boundary zone
- **Eq_{solidthickness}** = value from RMEH textbook which represents the "average" thickness of wall considering block voids.



Note: If the **1.5x Shear Inc** checkbox is checked in the [Masonry Wall - Design Rules](#), then f_v is factored up by a 1.5 factor. This option is available per section 7.3.2.6.1.2.

In Plane Shear Capacity, Fv

The program calculates the capacity, Fv, from Equation 8-22:

$$F_v := (F_{vm} + F_{vs}) \cdot \gamma_g$$

where:

$$F_{vm} := \frac{1}{2} \cdot \left(\left(4 - 1.75 \cdot \left(\frac{M}{V \cdot d} \right) \right) \cdot \sqrt{f_m} \right) + 0.25 \frac{P}{A_n}$$

$$F_{vs} := 0.5 \cdot \left(\frac{A_v \cdot F_s \cdot d}{A_n \cdot s} \right)$$

$\gamma_g = 0.75$ for partially grouted walls and 1.0 otherwise

The program also checks to verify we do not exceed the **Fv max** value from Equations 8-23 and 8-24 (or interpolation between them) that is reported in the detail report.

Fvs is only required if $f_v > F_{vm}$. If $f_v < F_{vm}$, then the program will not add any shear reinforcement and **Fvs** = 0. If **Fvs** is required, then the program will back calculate a spacing, **s**, that will satisfy the steel shear capacity required. This shear spacing is reported in the detail report.

When shear reinforcement is required the program will also meet the d/2 or 48" spacing required by 8.3.5.2.1. If **Fv max** must be exceeded to pass the code check, the program will use **Fv max** as the capacity and state "Over Allowable" for the Shear Steel Spacing.



Note:

- The program does not do explicit seismic design, thus Equation 8-25 is not used.
- The M in the F_{vm} equation above is taken as the maximum moment in the region instead of the moment at the location where the shear is maximum.

Shear Design - Out of Plane ASD

Out of Plane Shear Stress, f_v

This stress is calculated from Equation 8-21:

$$f_v := \frac{V}{A_{nv}}$$

V = Shear out of plane, per foot

$$A_{nvRMEH} := E q_{solidthickness} \cdot 1 \cdot ft$$

$$A_{nvNCMA} := A_{nvNCMAperfoot}$$

Out of Plane Shear Capacity, F_v

The program calculates the capacity, F_v , from Equation 8-25 shown above, except that only the F_{vm} term is considered. There is no way to add shear reinforcing steel.

The program also checks to verify we do not exceed the **Fv max** value from Equations 8-26 and 8-27 (or interpolation between them) that is reported in the detail report.

Axial Design - ASD

In and Out of Plane Axial Stress, f_a

The axial stress in a wall due to axial forces, f_a , is calculated as:

$$f_a := \frac{P}{A_n}$$

P = Total axial force in wall at the reported location

$$A_{nRMEH} := E q_{solidthickness} \cdot L_{wall}$$

$$A_{nNCMA} := A_{nNCMAperfoot} \cdot L_{wall}$$

This is applicable for all out of plane and in-plane masonry calculations. When $f_a < f_b$ then the masonry has cracked and then a cracked section analysis is performed. See the [Bending Design - ASD](#) section below for more information on this procedure.



Note: In and out of plane stress, f_a will be the same. However, the location where these forces are reported may be different. In plane forces will generally be reported at the base of a wall. Out of plane forces may be reported at the base (cantilevered walls) or at mid-height (simply supported walls).



Note: The program does not design masonry for net tension forces. f_a will equal 0 ksi in these cases.

In and Out of Plane Axial Capacity, F_a

The calculation of F_a is per either Equation 8-13 or 8-14, depending on the h/r ratio. These equations match Equations 8-18 and 8-19 if you assume $A_{st} = 0$. RISA conservatively uses only the masonry in calculating the compression capacity. The equations are as follows:

if $h/r \leq 99$

$$F_a := \frac{1}{4} \cdot f_m \cdot \left(1 - \left(\frac{h}{140 \cdot r} \right)^2 \right)$$

if $h/r > 99$

$$F_a := \frac{1}{4} \cdot f_m \cdot \left(\frac{70 \cdot r}{h} \right)^2$$

where:

- **r** is taken from the UBC-97 Table 21-H-1 (concrete masonry units) and Table 21-H-2 (clay masonry units).
- **h** = effective height of wall region = $K \cdot h_{\text{actual}}$



Note: The program does not design masonry for net tension forces. If there is net tension, F_a will still equal the compressive capacity of masonry.

Bending Design - ASD

In Plane Stresses f_b and f_s

These calculations are different depending on a cracked section or uncracked section. The wall is considered uncracked if f_a (compression) $> f_b$. The wall is considered cracked if f_a (compression) $< f_b$.

For uncracked masonry, the calculations are as follows:

$$f_b := \frac{M}{S_x}$$

where:

$$S_{xRMEH} := \frac{E q_{\text{solidthickness}} \cdot d^2}{6} \quad S_{xNCMA} := \frac{A_{nv} \cdot L_{\text{wall}} \cdot d}{6}$$

$$f_s := 0$$

where:

- **d** = Total length of wall

Because the masonry is uncracked, no stress can develop in the steel.

For **cracked masonry**, the program performs an iterative analysis to determine the section properties of the cracked wall.

The maximum masonry stress (f_m) is obtained by solving the moment equilibrium equation as a quadratic equation of kd . This equation comes from vertical equilibrium ($C - P - T = 0$).

Here is a representative wall with axial force and moment:

$$f_m := f_a + f_b$$

$$C := \frac{1}{2} \cdot f_m \cdot t \cdot k \cdot d$$

$$T := C - P$$

A good reference for this iterative procedure is Section 7.10.1 of **Design of Reinforced Masonry Structures**, copyright 2001 by Narendra Taly and published by McGraw-Hill.

From this analysis f_m is calculated. This must then be broken down into f_a and f_b for the detail report. Because Section 8.3.4.2.2 states:

$$f_a + f_b \leq 0.45 \cdot f_m$$

the proportion is simplified. RISA will simply take $f_a = P/A_n$ for f_a and the remaining value from f_m is used as f_b .

From the tension force in the wall, T , the program calculates the required number of boundary zone bars and places them in the wall. The reported code check is then based on:

$$f_s := \frac{T}{A_s}$$

$$Steel_{Unity} := \frac{f_s}{F_s}$$

Out of Plane Stresses f_b and f_s

The masonry bending stresses are referenced in UBC Section 2107.2.15 and are as follows:

$$f_b := \frac{2 \cdot M}{j \cdot k \cdot b \cdot d^2} \quad f_s := \frac{M}{A_s \cdot j \cdot d}$$

where:

$$\rho := \frac{A_s}{A_n} \quad n := \frac{E_s}{E_m}$$

$$k := \sqrt{2 \cdot \rho \cdot n + (\rho \cdot n)^2} - \rho \cdot n$$

$$\rho := \frac{A_s}{A_n} \quad n := \frac{E_s}{E_m}$$

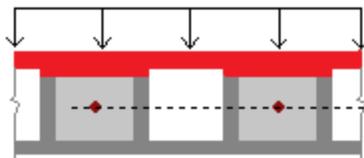
$$j := 1 - \frac{k}{3}$$

b = spacing of reinforcement

M = moment over a width b

However, if you are using a partially grouted wall where the neutral axis passes through the webs of your masonry, then RISA will do a T-section analysis to define the section properties. We use a similar analysis as if you were doing a t-beam analysis for a concrete tee section. For more information on this, see "Design of Reinforced Masonry Structures" by Narendra Taly, copyright 2001, example 6.3, P6.61.

T-SECTION ANALYSIS
kd : 1.638 in
Flange thickness: 1.375 in
Web width : 8.5 in



The area in red is shown as the compression block in the image above.

In and Out of Plane Masonry Bending Capacity, Fb

For unreinforced masonry, the Equation 8-15 is:

$$F_b := \frac{1}{3} \cdot f_m$$

For reinforced masonry, Section 8.3.4.2.2 states:

$$f_a + f_b \leq 0.45 \cdot f_m$$

Because of this provision, RISA defines:

$$F_b := 0.45 \cdot f_m$$

In and Out of Plane Steel Capacity, F_s

Section 8.3.3.1 defines the allowable steel stress, F_s .

In Plane Design - Strength

In Plane Axial Strength, P_n

This is calculated from Equations 9-15 and 9-16:

if $h/r \leq 99$:

$$P_n := 0.8 \cdot \left(0.8 \cdot A_{n1} \cdot f_m \cdot \left(1 - \left(\frac{h}{140 \cdot r} \right)^2 \right) \right)$$

if $h/r > 99$:

$$P_n := 0.8 \cdot \left(0.8 \cdot A_n \cdot f_m \cdot \left(\frac{70 \cdot r}{h} \right)^2 \right)$$



Note:

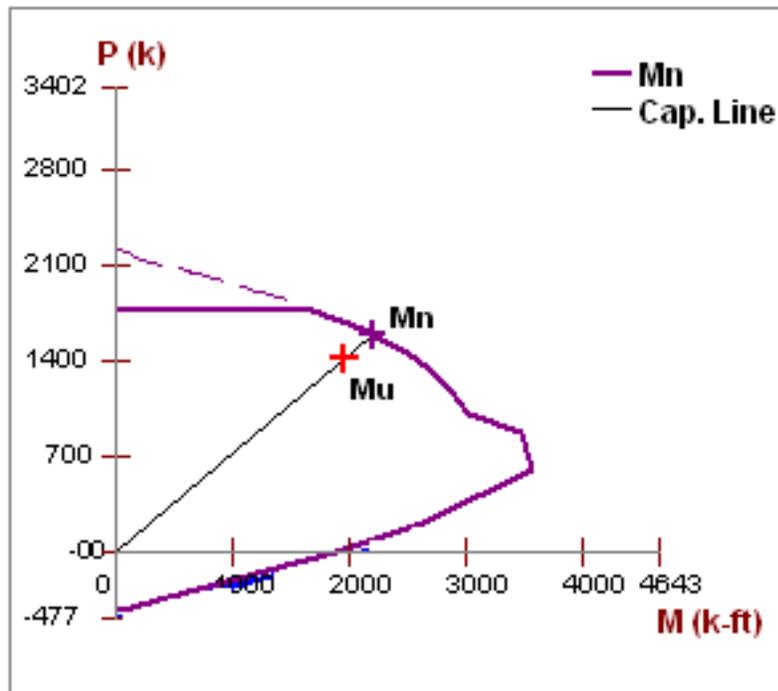
- Keep in mind that A_n is calculated differently for NCMA vs RMEH. See the [Masonry Wall - Design Rules](#) topic for more information.
- h = effective height of wall region = $K \cdot h_{\text{actual}}$

In Plane Moment Strength, M_n

The M_n value is based on the axial - moment interaction diagram for the wall based on strain compatibility at different levels of axial force. Once the interaction diagram is created the M_u is plotted and the code check is then calculated.



Note: In plane moment strength is solely accounted for by boundary zone steel. We do not use every bar in the wall and calculate the steel stress per bar. The "d" used for in plane design is from the extreme compression fiber to the centroid of reinforcement in the boundary zone. For a reference on this, see "Masonry Structures Behavior and Design" by Drysdale, Hamid, and Baker, copyright 1999.



For a wall with no axial force, the M_n can be calculated from the equation:

$$M_n := A_s \cdot f_y \cdot \left(d - \frac{a}{2} \right)$$



Note: The maximum usable masonry strain at the extreme compression fiber, ϵ_{mu} , is assumed to be 0.0025. For assumptions of the extreme compression fiber for Seismic Detailing checks of Masonry Walls, refer to the [Maximum Area of Flexural Tensile Reinforcement \(Max. Flexural Rho Check - Strength\)](#) section.

In Plane Shear Strength, V_n

The program calculates the capacity, V_n , from Equation 9-17:

$$V_n := (V_{nm} + V_{ns}) \cdot \gamma_g$$

where:

$$V_{nm} := \left(4 - 1.75 \cdot \left(\frac{M_u}{V_u \cdot d} \right) \right) \cdot A_{nv} \cdot \sqrt{f_m} + 0.25 \cdot P_u$$

$$V_{ns} := 0.5 \cdot \left(\frac{A_v}{s} \right) \cdot f_y \cdot d_v$$

$\gamma_g = 0.75$ for partially grouted walls and 1.0 otherwise

The program also checks to verify the **Vn max** value from Equations 9-18 and 9-19 (or interpolation between them) that is reported in the detail report is not exceeded.

Vns is only required if **Vu > Phi*Vnm**. If **Vu < Phi*Vnm**, then the program will not add any shear reinforcement and **Vns = 0**. If Vns is required, then the program will back calculate a spacing, **s**, that will both satisfy the steel shear capacity required and satisfy the maximum reinforcement spacing. This shear spacing is reported in the detail report.

If **Vn max** must be exceeded to pass the code check, the program will use **Vn max** as the capacity (so the wall fails in shear) and state "Over Allowable" for the Shear Steel Spacing.



Note: The M in the V_{nm} equation above is taken as the maximum moment in the region instead of the moment at the location where the shear is maximum. This is generally a conservative assumption.

Out of Plane Design - Strength

Out of Plane Axial Strength, Pn at max Mom

Here there are two different possible calculations.

If there is out-of-plane moment on the walls this is calculated from Section 9.3.5.4.2 as follows:

- If $h/t < 30$, **Pn at max Mom** = $0.20 \cdot f_m \cdot A_n$
- If $h/t > 30$, **Pn at max Mom** = $0.05 \cdot f_m \cdot A_n$

If there is no out-of-plane moment this is calculated from Equations 9-15 and 9-16:

if $h/r \leq 99$:

$$P_n := 0.8 \cdot \left(0.8 \cdot A_{n1} \cdot f_m \cdot \left(1 - \left(\frac{h}{140 \cdot r} \right)^2 \right) \right)$$

if $h/r > 99$:

$$P_n := 0.8 \cdot \left(0.8 \cdot A_n \cdot f_m \cdot \left(\frac{70 \cdot r}{h} \right)^2 \right)$$



Note:

- t from the above equations is taken as the Block Nominal Thickness.
- h = effective height of wall region = $K \cdot h_{\text{actual}}$

Out of Plane Moment Strength, M_n

This calculation comes from the Commentary of Section 9.3.5.2 as follows:

$$M_n := \left(\frac{P_u}{\phi} + A_s \cdot f_y \right) \cdot \left(\frac{t_{sp} - a}{2} \right) + A_s \cdot f_y \cdot \left(d - \frac{t_{sp}}{2} \right)$$

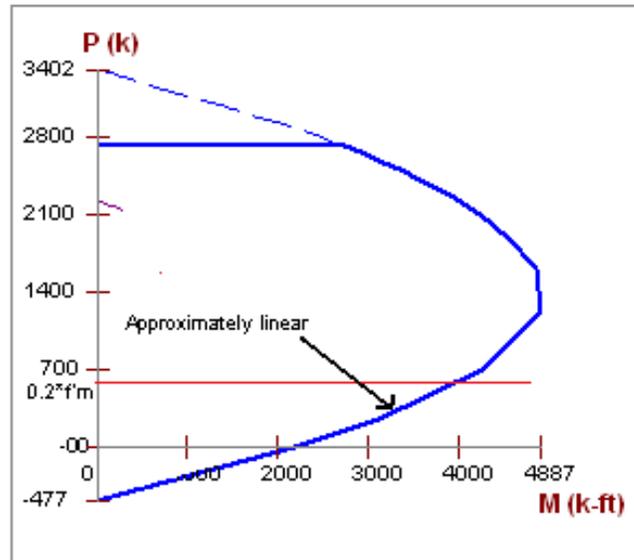
where:

$$a := \frac{A_s \cdot f_y + \frac{P_u}{\phi}}{0.80 \cdot f_m \cdot b}$$

b = reinforcement spacing

t_{sp} = actual block thickness

The program is actually using an interaction diagram for out of plane bending as well. However, the code places a limit on axial force from Section 9.3.5.4.2. This limit essentially means that only the lower portion of the interaction diagram will be used. In this lower portion of the interaction diagram, the bending capacity changes in a linear fashion with respect to axial force.



Thus, the equation above is nearly identical to the value that the program's interaction diagram will produce.



Note:

- In the case of high axial tension, it is possible that moment capacity will near zero.
- In partially grouted walls an effective width is calculated for b from Section 5.1.2

Out of Plane (Slender) Moment Demand, M_u

The out of plane demand moment, M_u , comes from the finite element analysis only. Equation 9-23 gives a formula for calculating the midheight moment due to out of plane loading, axial eccentric loading, and P-Little Delta effects. In RISA the first two portions can be captured by modeling the loads accordingly. Eccentric axial loading would need to be modeled with both a vertical load and a moment. The [P-Delta](#) analysis that RISA uses during the Finite Element solution does not account for P-Delta effects on plate elements or wall panels themselves. Instead the program captures P-Little Delta effects from the iterative procedure shown in the **Moment and Deflection Amplification** section below. This is very similar to that provided in the UBC and MSJC slender wall design provisions.

Out of Plane (Slender) Deflection, δ_s

Out of plane deflections are calculated based on the Wall's Finite Element Analysis using a modified thickness, t_{mod} , for the wall plates. The modified thickness is calculated using the wall panel's out of plate cracked moment of inertia, I_{cr} , which can be input in the Advanced tab of the Wall Panels spreadsheet.

$$t_{mod} := t_{original} \cdot \sqrt[3]{I_{cr}}$$



Note: The original thickness of the wall panel is the actual thickness of the wall panel, not to be confused with the nominal thickness.

RISA-3D then uses Finite Element Analysis using the modified thickness of the plates to calculate the out of plane deflection of the wall.

Out of plane deflections are calculated very similarly to Section 9.3.5.4.2 equations 9-25 or 9-26. Note that because of P-Little Delta effects, moment and deflection calculations are iterative. RISA uses the iterative procedure as follows:

Moment and Deflection Amplification

1. The initial moment diagram is calculated at 20 segments along the height of the wall using results taken directly from the FEM solution.
2. The program then uses this moment and compares it to M_{cr} to see what proportion of moment is under M_{cr} and over M_{cr} . From this an equivalent moment of inertia (I_{equiv}) is calculated to approximate cracking, similar to Section 9.3.5.4.2.
3. Using double integration and accounting for cracking the program calculates first order deflections.
4. Using these first order deflections and the axial force in the wall the program adds in additional P-little delta moments.
5. The program adds in the P-Delta moments based on these original elastic moments and deflections.
6. New deflections and moments are calculated and compared to the results from the previous iteration. The program iterates the procedure until the results have converged within 0.5% of the previous iteration, or until it has iterated 100 times.

**Note:**

- This procedure is intended only to determine the localized P-Little Delta amplification of Moments and Deflections within the wall itself. This procedure does NOT amplify the shear forces, nor does it contribute to the "leaning wall" effect by pushing on the rest of the structure.
- Deflection calculations are only given for **Service** load combinations.
- The program calculates I_{equiv} based on a cracked section with a modified thickness that depends on the cracked factor I_{cr} :MSJC equations that were specifically geared toward simply-supported and cantilever cases.
- I_{equiv} is taken from the maximum moment in a wall region and used to update the stiffness of the entire height of the wall.
- If the program iterates through this procedure 100 times it means that the wall is either too slender or too highly loaded. If this occurs then the message "Iterative P-Little Delta procedure did not converge. Wall is failing in bending." is reported.

Unreinforced Masonry Design - ASD

Out of Plane Bending and Axial Design - ASD

For out of plane design for axial and bending the program performs 3 checks: a compression check, a tension check, and a slenderness check.

Compression Check (per Equation 8-11)

$$Unity := \frac{f_a}{F_a} + \frac{f_b}{F_b}$$

where:

- f_a and F_a are computed the same way as for [reinforced masonry](#).
- $f_b = M/S$. $S = S_x$ value from NCMA TEK 141B (regardless of Wall Area Method designation).
- $F_b = 1/3 * f'm$ per Equation 8-15

Tension Check (per Section 8.2.4.2)

$$Unity := \frac{(f_b - f_a)}{F_{bt}}$$

where:

- **Fbt** = Allowable flexural tensile stress per Table 8.2.4.2

Slenderness Check (per Equation 8-12)

$$Unity := \frac{P}{0.25 \cdot P_{\epsilon}}$$

where:

$$P_{\epsilon} := \frac{\pi^2 \cdot E_m \cdot I}{h^2} \cdot \left(1 - 0.577 \frac{e}{r}\right)^3$$

- Per Equation 8-16.
- **I** comes directly from NCMA TEK 141B for both Wall Area Methods, NCMA and RMEH.
- **e** is always assumed equal to 0.
- **h** = effective height of wall region = $K \cdot h_{\text{actual}}$



Note: If using an **Unreinforced** wall the program will ALWAYS use an ungrouted wall for design, regardless of what is set in the masonry wall design rules.

Out of Plane Shear Design - ASD

This stress is calculated from Equation 8-21:

$$f_V := \frac{V}{A_{nv}}$$

V = Shear out of plane, per foot

$$A_{nvRMEH} := E q_{\text{solidthickness}} \cdot 1 \cdot \text{ft}$$

$$A_{nvNCMA} := A_{nvNCMA \text{ per foot}}$$



Note: We do not use Equation 8-17 to calculate shear stress.

The program calculates the capacity, F_v , from Equation 8-22 shown above, except that only the F_{vm} term is considered. There is no way to add shear reinforcing steel.

The program also checks to verify we do not exceed the **Fv max** value from Equations 8-23 and 8-24 (or interpolation between them) that is reported in the detail report.

In Plane Bending and Axial Design - ASD

For unreinforced masonry we use the same design provisions from reinforced masonry. Thus, we still use reinforcement for both the boundary zones and shear reinforcement (if needed). See the reinforced sections for more information on this design.



Note: Because this type of design is very uncommon it is not currently available.

In Plane Shear Design - ASD

The in plane shear stress, f_v , is calculated as follows:

$$f_v := \frac{V}{A_{nv}}$$

The in-plane shear stress capacity, F_v , is calculated from Section 8.2.6.2. Because we do not know whether the wall is in running bond or not we omit options (c), (d), (e) and (f).

Unreinforced Masonry Design - Strength

Out of Plane Bending and Axial Design - Strength

For unreinforced masonry we perform the axial check per Section 9.3.5. For bending the program does not give results.



Note: Because this type of design is very uncommon it is not currently available.

In Plane Bending, Axial, and Shear Design - Strength

For unreinforced masonry we use the same design provisions from reinforced masonry. Thus, we still use reinforcement for both the boundary zones and shear reinforcement (if needed). See the reinforced sections for more information on this design.



Note: Because this type of design is very uncommon it is not currently available.

Masonry Lintel Design - ASD

Here we will talk about specific calculations regarding lintel design for allowable stress design.



Note: There is no lintel design for unreinforced lintels. Regardless of whether you design the wall as reinforced or unreinforced, all lintel design will use reinforcement. Because unreinforced lintels are very uncommon it is not currently available.

Shear Stress, f_v

This stress is calculated from Equation 8-21:

$$f_v := \frac{V}{b \cdot d}$$

where:

- d = distance from extreme compression fiber in top of lintel to the centroid of reinforcement.

Shear Capacity, F_v

The program calculates the capacity, F_v , from Equation 8-25, which is shown in the previous [In Plane Shear Capacity \$F_v\$](#) section.

The program also checks to verify we do not exceed the **F_v max** value from Equations 8-23 and 8-24 (or interpolation between them) that is reported in the detail report.

F_v s is only required if $f_v > F_{vm}$. If $f_v < F_{vm}$, then the program will not add any shear reinforcement and **F_v s = 0**. If F_v s is required, then the program will back calculate a spacing, s , that will satisfy the steel shear capacity required. This shear spacing is reported in the detail report.

When shear reinforcement is required the program will also meet the $d/2$ or 48" spacing required by 8.3.5.2.1. If **F_v max** must be exceeded to pass the code check, the program will use **F_v max** as the capacity and state "Over Allowable" for the Shear Steel Spacing.



Note:

- The program does not do explicit seismic design, thus Equation 8-25 is not used.
- The M in the F_{vm} equation above is taken as the maximum moment in the region instead of the moment at the location where the shear is maximum. This is generally a conservative assumption.
- The detail report shows a stirrup with two legs. However, the program only considers one leg in the A_v calculation.

Bond Stress, u , and Bond Capacity U

The bond stress check is one that has been grandfathered in from the UBC-1997 specification section 2107.2.16 and is defined previously in the [In Plane Bond Stress \$u\$](#) section. Σo is the

summation of the circumference of all vertical bars.

The bond stress capacity was defined previously in the [Bond Stress Capacity U](#) section.

Stresses fb and fs

The masonry bending stresses were defined previously in the [Out of Plane Stresses fb and fs](#) section.

Bending Capacity, Fm (Fb) and Steel Capacity, Fs

Section 8.3.4.2.2 states:

$$F_b := 0.45 \cdot f_m$$

Section 8.3.3.1 defines the allowable steel stress, Fs.

Masonry Lintel Design - Strength

Here we will talk about specific calculations regarding lintel design for strength design.



Note: There is no lintel design for unreinforced lintels. Regardless of whether you design the wall as reinforced or unreinforced, all lintel design will use reinforcement. Because unreinforced lintels are very uncommon it is not currently available.

Shear Strength, Vn

The program calculates the capacity, **Vn, total** from Equation 9-17, which was shown previously in the [In Plane Shear Strength Vn](#) section.

The program also checks to verify the **Vn max** value from Equations 9-18 and 9-19 (or interpolation between them) that is reported in the detail report is not exceeded.

Vn,steel is only required if **Vu > Phi*Vn,masonry**. If **Vu < Phi*Vn,masonry**, then the program will not add any shear reinforcement and **Vn,steel = 0**. If **Vn,steel** is required, then the program will back calculate a spacing, **s**, that will satisfy the steel shear capacity required. This shear spacing is reported in the detail report.

If **Vn max** must be exceeded to pass the code check, the program will use **Vn max** as the capacity (so the wall fails in shear) and state "Over Allowable" for the Shear Steel Spacing.



Note: The M in the V_{nm} equation above is taken as the maximum moment in the region instead of the moment at the location where the shear is maximum. This is generally a conservative assumption.

Moment Strength, Mn

Lintel design in RISA does not account for axial force, thus Mn is calculated simply from:

$$M_n := A_s \cdot f_y \cdot \left(d - \frac{a}{2} \right)$$

Here are the calculations of other values reported in the detail report:

$a := \frac{A_s \cdot f_y}{0.8 \cdot f_m \cdot b}$	Depth of compression block
$c := \frac{a}{0.8}$	Neutral Axis Location
$C := a \cdot b \cdot 0.8 \cdot f_m$	Total compression in the compression block
$M_{nmasonry} := C \cdot \left(c - \frac{a}{2} \right)$	Moment capacity from NA to compression block
$M_{nsteel} := A_s \cdot f_y \cdot (d - c)$	Moment capacity from NA to tension steel

Masonry Wall - Design Rules

The masonry wall panel element allows you to easily model, analyze and design masonry walls for in plane and out of plane loads. In this section, we will explain the masonry specific inputs and design considerations. For general wall panel information, see the [Wall Panels](#) topic. For masonry wall modeling procedures, see the [Masonry Wall - Modeling](#) topic. For masonry calculation considerations and code references, see the [Masonry Wall - Design](#) topic. For masonry wall results interpretation, see the [Masonry Wall Results](#) topic.



Note: All code references in this topic refer to the **TMS 402-16** specification unless noted otherwise.

Unity Check

Wall Panel U.C. Parameters					
Masonry Lintel	Wood Wall (Studs)	Wood Wall (Fasteners)	CFS Wall (Studs)	CFS Wall (Fasteners)	
Unity Check	Concrete Wall (Rebar)	Concrete Wall (Misc)	Masonry Wall	Masonry In	Masonry Out
	Label	Max Bending Chk	Max Shear Chk		
1	Typical	1	1		
2	Concrete Wall	1	1		
3	Masonry Wall	0.9	1		
4	Wood Wall	0.9	0.8		

Setting a maximum Bending Check (Axial & Bending) or a maximum Shear Check controls the rebar which the program chooses for the wall design. A value of 1.0 denotes that the program may choose a rebar layout that is at 100% of capacity.



Note: For the **Max Shear Check** for in plane design, the program will only optimize the spacing of reinforcement. It will not optimize the grout spacing. If you want to adjust grout spacing, review the **Masonry Out** tab.

Masonry Wall General

Masonry Wall Panel Parameters					
Masonry Lintel	Wood Wall (Studs)	Wood Wall (Fasteners)	CFS Wall (Studs)	CFS Wall (Fasteners)	
Unity Check	Concrete Wall (Rebar)	Concrete Wall (Misc)	Masonry Wall	Masonry In	Masonry Out
	Label	Block Nom Width	Block Grouting	Reinforced	Wall Area Method
1	6" Masonry Wall	6"	UngROUTed	<input type="checkbox"/>	RMEH
2	8" Masonry Wall	8"	Fully Grouted	<input checked="" type="checkbox"/>	NCMA
3	10" Masonry Wall	10"	Partially Grouted	<input checked="" type="checkbox"/>	NCMA ▼

Masonry Wall Parameters

Parameter	Description
Block Nom Width	Block Nominal Width is the nominal thickness of masonry walls. The program will subtract 3/8" from this value to get the actual thickness. This value is used, along with the value of grout / bar spacing, to determine the effective thickness of the wall. The effective thickness is based on table B3 of the Reinforced Masonry Engineering Handbook,

Parameter	Description
	by Amrhein, Copyright 1998.
Block Grouting	Block Grouting defines how the wall is grouted. If "Partially Grouted" is chosen, then the spacing of grout will be based on the bar spacing defined on the Masonry Out tab.
Reinforced	Reinforced defines whether the wall is reinforced or not.
Wall Area Method	<p>Wall Area Method defines where the wall area is taken from. The NCMA option pulls the "A_n" value from the NCMA TEK 141B document. The RMEH option pulls the "Equivalent Solid Thickness" value from Table B-3a and B-3b from the <i>Reinforced Masonry Engineering Handbook</i>, James Amrhein, 5th edition copyright 1998.</p> <p>In doing research on these two methods of calculating the area for a masonry wall, the two methods produce very different results. The NCMA values assume face-shell mortar bedding and web bedding around grout-filled cells. The RMEH values assume full mortar bedding (both face-shells and all webs). The Amrhein values also appear to average in the area of horizontal bond beams as well. This would make the area conservative for a self-weight calculation, but unconservative for stress calculations. With these considerations in mind we are defaulting the behavior to use NCMA by default.</p> <p>See the Masonry Wall - Design topic for the specific calculations regarding NCMA or RMEH.</p> <div data-bbox="492 1161 1409 1631" style="border: 1px solid black; padding: 10px; background-color: #e6f2ff;"> <p> Note:</p> <ul style="list-style-type: none"> • In version RISA-3D V10.0.1 and older the program used both methods in different circumstances. Now the program is consistent with this usage. • If you open an existing model from RISA-3D V10.0.1 and older the program will default to the RMEH option. • If you start a model from scratch the program will default to the NCMA option. • For self-weight the program will ALWAYS use RMEH, as the presence of bond beams in the wall is realistic and this is a conservative value. </div>

Masonry Wall In Plane Design

Masonry Wall Panel In Plane Parameters							
Masonry Lintel	Wood Wall (Studs)	Wood Wall (Fasteners)	CFS Wall (Studs)	CFS Wall (Fasteners)			
Unity Check	Concrete Wall (Rebar)	Concrete Wall (Misc)	Masonry Wall	Masonry In	Masonry Out		
	Label	Vert Bar Size	Bars Per Cell	Min Bound Zone Width [in]	Max Bound Zone Width [in]	Horz Bar Size	Transfer Load
1	6" Masonry Wall	#5	1	8	40	#5	<input type="checkbox"/>
2	8" Masonry Wall	#6	1	8	40	#6	<input type="checkbox"/>
3	10" Masonry Wall	#5	1	8	40	#5	<input type="checkbox"/>

Masonry Wall In-Plane Parameters

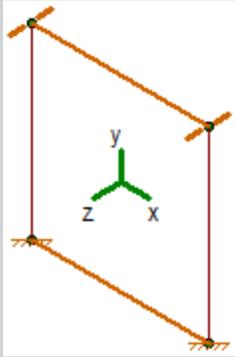
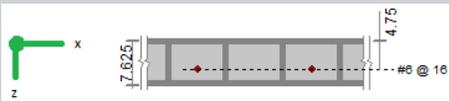
Parameter	Description
Vert Bar Size	The Vertical Bar Size column lets you define the vertical bar size for the boundary zones.
Bars Per Cell	The Bars Per Cell columns lets you define one or two bars per cell in the boundary zones.
Min/Max Boundary Zone	The Min/Max Boundary Zone column lets you give maximum and minimum boundary zone widths. The program then designs the width based on 8" increments (1/2 of a block length).
Horz Bar size	The Horizontal Bar Size column lets you define horizontal bar size to be used if horizontal reinforcing is required.
1.5x Shear Inc	The 1.5x Shear Inc checkbox has been moved to the Seismic Design Rules spreadsheet in the Masonry Walls tab. Please see Masonry Wall - Seismic Design .
Transfer Load	The Transfer Load option transfers in-plane loads from regions above and below openings into the regions adjacent to the openings. Note that the design of regions above and below openings are omitted.

Masonry Wall Out of Plane Design

Masonry Wall Panel Out of Plane Parameters									
Masonry Lintel	Wood Wall (Studs)	Wood Wall (Fasteners)	CFS Wall (Studs)	CFS Wall (Fasteners)					
Unity Check	Concrete Wall (Rebar)	Concrete Wall (Misc)	Masonry Wall	Masonry In	Masonry Out				
	Label	Bar Size	Bar Space Min	Bar Space Max	Bar Placement	Cover [in]	Mortar Type	Cement Type	Transfer Load
1	6" Masonry Wall	#5	8"	48"	Center	Min	Type M or S	Portland, Lime/Mortar	<input type="checkbox"/>
2	8" Masonry Wall	#5	8"	48"	Center	Min	Type M or S	Portland, Lime/Mortar	<input type="checkbox"/>
3	10" Masonry Wall	#5	8"	48"	Center	Min	Type M or S	Portland, Lime/Mortar	<input type="checkbox"/>

Masonry-Wall-Out-of-Plane Parameters

Parameter	Description
Bar Size	Bar Size is the main vertical bar size to be used for axial/out of plane design.
Bar Space Min/Max	Bar Space Min/Max lets you give a maximum and minimum bar/grout spacing. If you give a range between the max and min, then the program optimizes the reinforcement spacing according to strength requirements (code checks).
Bar Placement	<p>Bar Placement defines how the reinforcement is placed in the wall region. Possible placement is:</p> <ul style="list-style-type: none"> • Center - puts a single bar centered in a given cell. • Non-Center puts a single bar non-centered in a given cell. • Each Face puts reinforcement on both faces of a given cell. • Staggered alternates the bars on either face along the length of the region. <div style="border: 1px solid black; padding: 5px; margin-top: 10px;">  Note: When using the staggered option, you are selecting to space the bars at each face at double the bar/grout spacing defined above. For example, if you have a staggered spacing at 24" oc you have a bar on the outside face at 48" oc and a bar on the inside face at 48" oc. These bars are staggered, thus you have grout filled cores at 24" oc. </div>
Cover	<p>Allows you to specify the distance from the exterior of the masonry block to the extreme fiber of the reinforcement. The default cover value is defined as Min. The Min input will use whichever governs between these two sections of the specification:</p> <ul style="list-style-type: none"> • Section 6.1.3.5 states a minimum distance between the inside of the face shell and the reinforcement . RISA is conservative and always assumes this to be 1/2". Thus, the cover calculated in this fashion = $t_{\text{faceshell}} + 1/2"$ • Section 6.1.4 requires a cover dimension = 1-1/2" or 2" depending on bar size and whether the masonry is exposed to weather. In the program this is always assumed to be 1-1/2". <p>Thus, the Min term will create a cover equal to the maximum of $t_{\text{faceshell}} + 1/2"$ OR 1-1/2".</p> <p>For any other cover you wish to impose simply overwrite the Min value with your value. If using the Non-Center option then the cover will always be from the +z local axis face of the wall.</p> <p>For example, let's assume the wall shown below is using the "Non-Centered" reinforcement and the cover is set to 2.5". The image below</p>

Parameter	Description
	<p>shows exactly where this bar is then located in the wall ($d = 7.625" - 2.5" - 3/8" = 4.75"$).</p> <div style="display: flex; justify-content: space-around; align-items: center;">   </div> <p>The actual "d" used in design will depend on the governing direction of loading. If load is applied in both out-of-plane directions to the wall then it is possible that a lower level of loading can produce a higher code check because the "d" is smaller in one direction than the other.</p> <p>If a non-sensical value is defined (i.e. one where the bar does not fall inside of the block core then the program will give an error in the results and place the reinforcement just inside the faceshell of the block.</p> <div style="border: 1px solid black; padding: 5px; margin-top: 10px;"> Note: For the Center bar placement this field is ignored. </div>
Mortar/Cement Types	Mortar/Cement Types lets you specify the type of mortar/cement in the wall. This affects the modulus of rupture (flexural tensile stresses) from Tables 8.2.4.2 and 9.1.9.2.
Transfer Load	Transfer Load lets you transfer out of plane loads from regions above and below openings into the regions adjacent to the openings. Note that the design of regions above and below openings will be omitted.

Masonry Wall Lintel Design

Masonry Wall Panel Lintel Parameters											
Unity Check	Concrete Wall (Rebar)			Concrete Wall (Misc)			Masonry Wall	Masonry In		Masonry Out	
Masonry Lintel		Wood Wall (Studs)			Wood Wall (Fasteners)			CFS Wall (Studs)		CFS Wall (Fasteners)	
	Label	Depth [in]	Bear Length [in]	Bar Size	Min # Bars Per Layer	Max # Bars Per Layer	Num of Layers	c/c Sp of Layers [in]	Dist To Bot [in]	Stirrup Size	Analysis Method
1	6" Masonry...	16	8	#5	1	3	1	N/A	3.5	#4	Simply Supported
2	8" Masonry Wall	16	8	#5	1	3	1	N/A	3.5	#4	Simply Supported
3	10" Masonry Wall	16	8	#5	1	3	1	N/A	3.5	#4	Simply Supported

Masonry Wall Panel Lintel Parameters

Parameter	Description
Depth	Depth is the total depth of your lintel.
Bar Length	Bar Length is the bearing length at either end of the lintel. This is used to calculate the effective length of the lintel.
Bar Size	Bar Size is the reinforcement size for your main reinforcing in the lintel.
Min/Max # Bars Per Layer	Min/Max # Bars Per Layer is the number of bars you wish to have in a given layer of reinforcement. If you give a range between the max and min, then the program will optimize the reinforcement spacing based on geometry of the section and also the number of layers that you have defined.
Num of Layers	Num of Layers is an option if you require multiple layers of reinforcement in the lintel.
c/c Sp of Layers	c/c Spacing of Layers is the distance between layers (if there is more than one).
Dist to Bot	Distance to Bottom defines the distance from the centerline of the lowest-most bar to the bottom fiber of the lintel.
Stirrup Size	Stirrup Size is the size of stirrup that will be added to the lintel if required.
Analysis Method	Analysis Method is the is the lintel analysis method that determines the design of the masonry lintel. The Simply Supported analysis method produces lintel reinforcement design results, which can be seen in the masonry lintel detail report. The FEA analysis method only displays shear and moment diagrams in the detail report for the masonry lintel based on the results of finite element analysis.

Masonry Wall Results

Masonry wall results are presented in the wall results spreadsheets and the detail reports. Results are reported on a region by region basis. In addition, the code checks for lintels spanning openings are reported separately.

For general wall panel information, see the [Wall Panels](#) topic. For information on masonry design rules, see the [Masonry Wall - Design Rules](#). For masonry wall modeling procedures, see the [Masonry Wall - Modeling](#) topic. For masonry calculation considerations and code references, see the [Masonry Wall - Design](#) topic.



Note: All code references in this topic refer to the **TMS 402-16** specification unless noted otherwise.

Spreadsheet Results

The information on this spreadsheet is present on three tabs that involve Masonry design: In plane, Out of Plane and Lintel. Each tab gives code checks based on the chosen masonry code and can be used as a summary of all of the panels and panel regions in your model. To get detailed information about each region, you can see the Wall Panel Detail Report.

In Plane

Wall Panel TMS 402-16: ASD Masonry Code Checks for Wall Regions (In Plane)										
Masonry Seismic		Wood Wall Axial		Wood Wall In-Plane		Wood Header		CFS Wall Axial		CFS Wall In-Plane
Concrete In		Concrete Out		Concrete Seismic		Masonry In		Masonry Out		Masonry Lintel
	Wall Panel	Region	Design Rule	Combined UC	LC	Shear UC	LC	Fa[ksi]	Fb[ksi]	Fv[ksi]
1	WP1	R2	6" Masonry Wall	0.262	14	1.714	14	0.369	0.5	0.058
2		R3	6" Masonry Wall	0.222	14	1.238	14	0.372	0.5	0.058
3		R5	6" Masonry Wall	1.272	14	2.54	14	0.232	0.5	0.058
4		R6	6" Masonry Wall	0.79	14	2.366	14	0.267	0.5	0.058
5		R7	6" Masonry Wall	1.778	14	2.638	14	0.232	0.5	0.058
6		R8	6" Masonry Wall	0.957	14	2.253	14	0.267	0.5	0.058

The In Plane results spreadsheet is intended to provide the code checks relevant to shear wall behavior for the wall.

Masonry Wall Spreadsheet - In Plane

Column	Description
Wall Panel	The Wall Panel column displays the masonry wall panels defined in the model.
Region	The Region column lists the masonry wall panel region that the results are based on.
Design Rule	The Design Rule column displays the rule governing the design.
Combined UC	For ASD codes, the Combined UC column displays the code check due to axial force plus in-plane bending.
LC	The LC column adjacent to the Combined UC column, shows the individual code check for Combined UC. A value greater 1.0 for either of these values would indicate failure.
Shear UC	The Shear UC column shows the in-plane shear code check. A value greater than 1.0 for any of these values would indicate failure.
LC	The LC column adjacent to the Shear UC column, shows the load combination that produces each of the highest code check values.
Fa (or Pn*Phi)	The Fa or Pn*Phi column displays the allowable axial stress or axial capacity.
Fb (or Mn*Phi)	The Fb or Mn*Phi column displays the calculated allowable bending or moment capacity for the region.
Fv (or Vn*Phi)	The Fv or Vn*Phi column displays the calculated allowable shear stress or Shear Capacity for the region.

Out of Plane

Wall Panel TMS 402-16: ASD Masonry Code Checks for Wall Regions (Out of Plane)										
Masonry Seismic		Wood Wall Axial		Wood Wall In-Plane		Wood Header		CFS Wall Axial		CFS Wall In-Plane
Concrete In		Concrete Out		Concrete Seismic		Masonry In		Masonry Out		Masonry Lintel
	Wall Panel	Region	Design Rule	Combined UC	LC	Shear UC	LC	Fa[ksi]	Fb[ksi]	Fv[ksi]
1	WP1	R2	6" Masonry Wall	0.021	14	0	N/A	0.369	0.5	0.039
2		R3	6" Masonry Wall	0.022	4	0	N/A	0.372	0.5	0.039
3		R5	6" Masonry Wall	0.249	4	0	N/A	0.232	0.5	0.039
4		R6	6" Masonry Wall	0.138	4	0	N/A	0.267	0.5	0.039
5		R7	6" Masonry Wall	1.329	14	0	N/A	0.232	0.5	0.039
6		R8	6" Masonry Wall	0.519	14	0	N/A	0.267	0.5	0.039

The Out-of-Plane results spreadsheet is intended to provide the code checks relevant to out of plane bending for the wall.

Masonry Wall Spreadsheet - Out of Plane

Column	Description
Wall Panel	The Wall Panel column displays the masonry wall panels defined in the model.
Region	The Region column lists the masonry wall panel region that the results are based on.
Design Rule	The Design Rule column displays the rule governing the design.
Combined UC	For ASD codes, the Combined UC column displays the code check due to axial force plus out of plane bending. A value greater than 1.0 would indicate failure. Note that this value represents the maximum of the different combined checks shown in the detail report $((fa+fb)/Fb, fa/Fa,$ and fs/Fs).
LC	The LC column adjacent to the Combined UC column, shows the individual code check for Combined UC. A value greater 1.0 for either of these values would indicate failure.
Shear UC	The Shear UC column shows the in-plane shear code check. A value greater than 1.0 for any of these values would indicate failure.
LC	The LC column adjacent to the Shear UC column, shows the load combination that produces each of the highest code check values.
Fa (or Pn*Phi)	The Fa or Pn*Phi column displays the allowable axial stress or axial capacity.
Fb (or Mn*Phi)	The Fb or Mn*Phi column displays the calculated allowable bending or moment capacity for the region.

Column	Description
Fv (or Vn*Phi)	The Fv or Vn*Phi column displays the calculated allowable shear stress for the region.

For slender wall design additional checks and analyses are required. These are reported in the region's detail report.

Masonry Lintels

Masonry Seismic		Wood Wall Axial		Wood Wall In-Plane		Wood Header		CFS Wall Axial		CFS Wall In-Plane	
Concrete In		Concrete Out		Concrete Seismic		Masonry In		Masonry Out		Masonry Lintel	
	Wall Panel	Lintel	Design Rule	Flexure UC	LC	Shear UC	LC	Fvm[ksi]	Fvs[ksi]	Fm[ksi]	Fs[ksi]
1	WP1	L1	6" Masonry Wall	0.435	14	0.739	14	0.077	0	0.5	32
2		L2	6" Masonry Wall	0.801	14	0.765	14	0.077	0.089	0.5	32

The Masonry Lintel results spreadsheet provides the results for the masonry lintels that span over user defined openings in the wall. They can also be viewed by looking at the detail report associated with each opening.

Masonry Wall Spreadsheet - Lintel

Column	Description
Wall Panel	The Wall Panel column displays the masonry wall panels defined in the model.
Region	The Region column lists the masonry wall panel region that the results are based on.
Design Rule	The Design Rule column displays the rule governing the design.
Flexure UC	The Flexure UC column displays the the code check due pure flexure of the Lintel. Axial force is not considered in this code check at all.
LC	The LC column adjacent to the Flexure UC column, shows the load combination that produces each of the highest code check values.
Shear UC	The Shear UC column displays the code check for shear. A value greater 1.0 for either of these values would indicate failure.
LC	The LC column adjacent to the Shear UC column, shows the load combination that produces each of the highest code check values.
Fvm (or Vn*Phi)	The Fvm or Vn*Phi column displays the allowable shear capacity.
Fvs (or Vn*Phi)	The Fvs or Vn*Phi column displays the allowable shear stress.
Fm (or Mn*Phi)	The Fm or Pn*Phi column displays the calculated allowable moment capacity for the region.
Fs (or Mn*Phi)	The Fs or Mn*Phi column displays the calculated allowable bending stress

Column	Description
	for the region.

Concrete Reinforcing Spreadsheet Results

The last two tabs of this spreadsheet contain results for Masonry Wall reinforcement.

Masonry Wall

In Plane Masonry - Axial Details

Masonry Wall Reinforcement					
Beam Bending		Beam Shear		Column Bending	
Concrete Wall		Masonry Wall		Masonry Lintel	
	Wall	Region	Hor. Bar Size	Vert. Bar Size	Boundary Reinf.
1	WP1	R2	Not Reqd.	#5@48" oc (ctr)	4-#5
2		R3	Not Reqd.	#5@48" oc (ctr)	4-#5
3		R5	Not Reqd.	#5@48" oc (ctr)	2-#5
4		R6	Not Reqd.	#5@48" oc (ctr)	2-#5
5		R7	Not Reqd.	#5@48" oc (ctr)	2-#5
6		R8	Not Reqd.	#5@48" oc (ctr)	2-#5

The Masonry Wall spreadsheet displays analysis results for the reinforcement of each region defined in your masonry wall.

Lintel Reinforcing

Masonry Lintel Reinforcement				
Beam Bending		Beam Shear		Column Bending
Column Shear		Concrete Wall	Masonry Wall	Masonry Lintel
	Wall	Lintel	Flex. Steel	Stirrup
1	WP1	L1	1-#5	Not Req'd.
2		L2	1-#5	#4-6 oc

The Masonry Lintel spreadsheet displays analysis results for the reinforcement of each lintel defined in your masonry wall.

Masonry Wall Detail Report

The detail reports show the overall geometry, analysis and design for the individual regions of the wall panel. The report also shows envelope diagrams for the forces and moments in the region.

Three basic types of detail reports are provided: Wall Summary, Region and Opening.

Accessing the Detail Reports and the Specific Windows

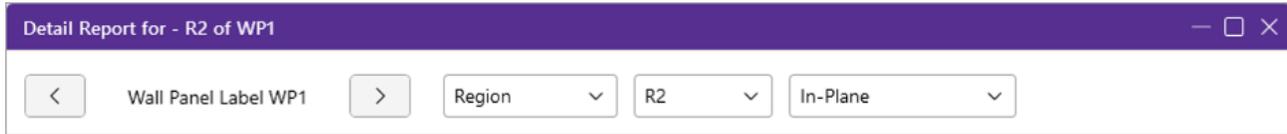
Once you have a solved model, the detail reports become available. They are accessible in two ways:

- If you have the **Wall Panel Design** spreadsheet open, there is a button at the top of the screen: [Detail Report for Current Item](#). This opens the detail report window.
- If you are in a graphic view of your model, there is a  button on the Selection toolbar. Clicking this button and clicking on a wall panel will open up the detail report window for that wall panel.



Note: Note: If a wall panel is set to Transfer forces, then regions above and below opening will not have results. See the [Concrete Wall - Design Rules](#) topic for more information.

Once the detail report window is open, you see an area at the top.



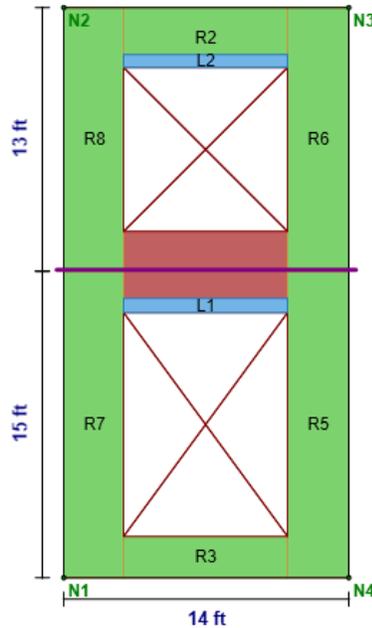
The following table describes the options that control the display of the Detail Report.

Detail Report Control Options

Option	Name	Description
	Prior/Next	The left and right arrows let you scroll quickly between the different wall panels in your model.
	Wall Panel Part	The first drop-down list lets you choose between individual Region and Opening (Lintel) results and a summary of the entire Wall.
	Regions/Headers	<p>The second drop-down list lets you select between different Regions or Openings within the individual wall panel. “R” represents regions, while “L” represents levels (headers).</p> <p>The is available only when ‘Opening’ or ‘Region’ is chosen in the Wall Panel Part drop-down.</p>
	In/Out Plane	<p>If you have selected a Region, then you have the option of whether to view the in plane or out of plane report.</p> <p>The is available only when ‘Region’ is chosen in the Wall Panel Part drop-down.</p>
<p>The following options can be found at the bottom of the Detail Report window.</p>		
	Print	Lets you print the Detail Report.
	Add to Full Report	Lets you add the current detail report you are viewing to the printed report. View the Printing topic for more information.

Detail Report - Wall Summary

This report gives an overview of the wall, a summary of the controlling code checks and deflection information. This report also displays information about the wall, similar to the [Region Input Echo](#) and also gives an image of the wall. The image shows region locations, wall length and story dimensions, and the nodes that define the corners of the wall panel.



CRITERIA		GEOMETRY		MATERIALS	
Code:	TMS 402-16: ASD	Total Height (ft):	28	Material Type:	Masonry
Special Insp:	No	Total Length (ft):	14	Material Set:	Concrete Matl
Design Rule:	6" Masonry Wall	Total Lintel Depth (in):	16	Masonry f'm (ksi):	1.5
Seismic Design Rule:	None	Block Nom Width:	6"	Masonry Em (ksi):	1350
Wall Area:	RMEH	Block Grouting:	UngROUTED	Steel fy (ksi):	60
Transfer In?:	No	1.5 Shear Factor:	No	Steel E (ksi):	29000
Transfer Out?:	No			Block Material:	Conc 115 pcf
K:	1			Grout Weight:	140 pcf
Use Cracked?:	Yes			Mortar Type:	Type M or S
In Icr Factor:	0.5			Cement Type:	Portland, Lime/Mortar
Out Icr Factor:	0.5				
Custom Regions:	No				

The **Region Results** section gives the tabulated results of all regions in the wall for in plane design axial/bending, shear and deflection for quick reference. You can view the individual region reports to get a more detailed explanation of these values.

The **Reinforcement Results** section gives the reinforcement results for each region in the wall.

The **Lintel Reinforcement Results** section gives the label and reinforcement for any lintels in the wall.

Detail Report - In Plane

In Plane-Masonry - Input Echo

The top section of the detail report echoes back the input information used in the design of the wall region or lintel. This information is summarized below:

CRITERIA		GEOMETRY		MATERIALS	
Code:	TMS 402-16: ASD	Total Height (ft):	28	Material Type:	Masonry
Special Insp:	No	Total Length (ft):	14	Material Set:	Concrete Matl
Design Rule:	6" Masonry Wall	Total Lintel Depth (in):	16	Masonry f'm (ksi):	1.5
Seismic Design Rule:	None	Block Nom Width:	6"	Masonry Em (ksi):	1350
Wall Area:	RMEH	Block Grouting:	UngROUTED	Steel fy (ksi):	60
Transfer In?:	No	1.5 Shear Factor:	No	Steel E (ksi):	29000
Transfer Out?:	No			Block Material:	Conc 115 pcf
K:	1			Grout Weight:	140 pcf
Use Cracked?:	Yes			Mortar Type:	Type M or S
In lcr Factor:	0.5			Cement Type:	Portland, Lime/Mortar
Out lcr Factor:	0.5				
Custom Regions:	No				

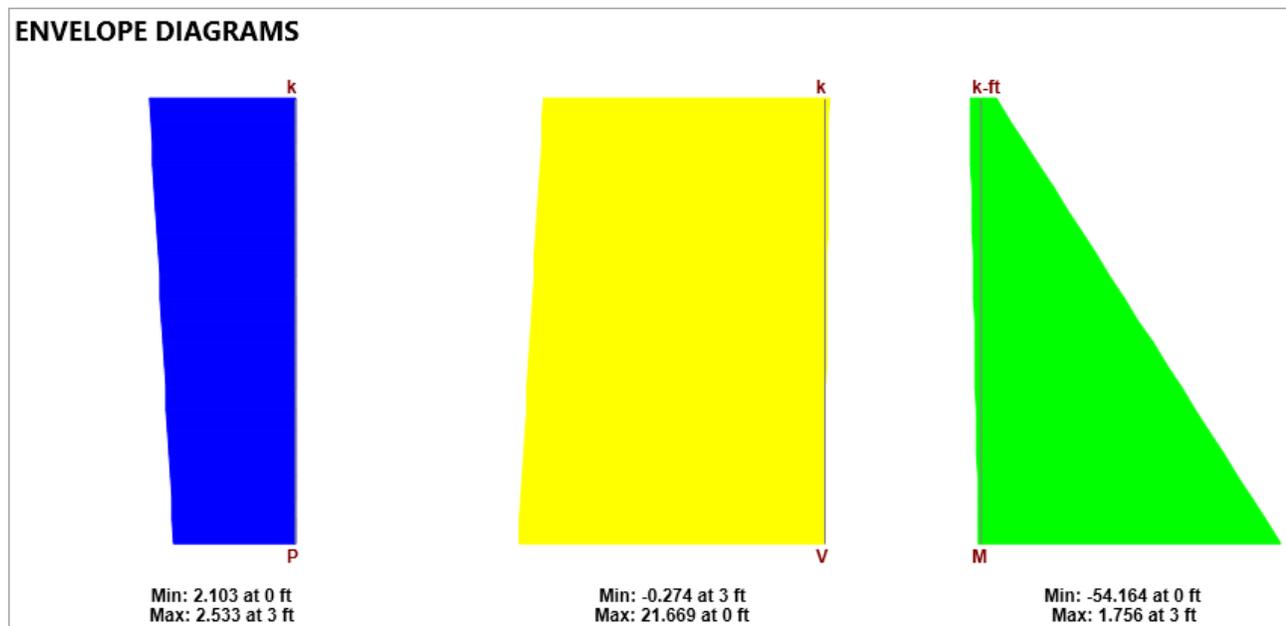
Detail Report - Criteria Column

Criteria	Description
Code	Gives the code used to design your wall panel.
Special Inspection	Indicates whether special inspection is required for your wall. Special inspection is normally required for all walls designed to the MSJC / IBC codes. The UBC 1997 is the one code that allows the user to decide whether special inspection is required.
Wall Area	See the Wall Design Rules - Masonry Wall spreadsheet
Horizontal Bar Size	Indicates the bar size to be used to resist shear forces.
Vertical Bar Size	Indicates the bar size to be used to resist the tensile stress due to moment.
Number of Tension Bars	Indicates the number of vertical bars present in the boundary region of the wall.
Effective depth	This gives you the distance from the compression face of the wall to the centroid of tension reinforcement.

The **Geometry** column gives the basic geometry of the wall. Block grouting and the grout spacing are specified in the Wall Panel Editor. The block nominal width is input under the [Wall Design Rules - Masonry Wall](#) tab.

The **Materials** column can be mostly modified under the Materials button on the Data Entry toolbar.

The **Envelope Diagrams** section of the detail report displays the envelope axial shear and moment diagrams as well as a summary of the code checks performed on the shear wall.



The **Combined Check Summary** gives maximum overall code checks considering the effects of both bending and axial stresses. This section is not reported for MSJC strength design.

The **Axial Summary** and **Bending Summary** give the values used in the axial and flexural code checks at the location which controls the combined check $(f_a + f_b)/F_b$ for ASD design.

For ASD design, the maximum bending stress in the flexural reinforcement is reported as **fs**, and the allowable steel bending stress as **Fs**. These values are given for the Load Combination and section that produce the maximum code check (f_s/F_s) .

The **Shear Summary** reports the maximum shear demand on the beam as **fv** or **Vu**. The allowable shear stresses are then reported as **Fvm** and **Fvs**, where **Fvm** is the allowable of the masonry alone and **Fvs** is the allowable considering the effects of the shear reinforcement.

AXIAL DETAILS			
fa/Fa:	0.021	Axial (k):	2.533
fa (ksi):	0.008	Location (ft):	3
Fa (ksi):	0.369	Load Comb:	14
		Rad gyration r (in):	2.08
		h'/r:	17.308
		Red Factor R:	0.985

Detail Report - Axial Details

Axial Details	Description
Maximum Axial Force	The maximum axial force in the wall.
Location	The location along the height of the wall which produces the maximum axial force.
Load Combination	The load combination that produced the maximum axial force.
Radius of Gyration (r)	The out of plane radius of gyration for the wall.
h/r	Slenderness ratio of the wall
Reduction Factor R	This is the slenderness reduction portion of equations 8-13, 8-14, 9-11 and 9-12.

In Plane Masonry - Bending Details

Working Stress Design (ASD)

BENDING DETAILS					
fa/Fa + fb/Fb:	0.262	Moment (k-ft):	54.164	k*d (in):	0
fb (ksi):	0.124	Location (ft):	0	j:	1
Fb (ksi):	0.5	Load Comb:	14		
fs/Fs:	0	Sect Mod S (in ³):	5222		
fs (ksi):	0	Tension St Asv (in ²):	0		
Fs (ksi):	32	Per of steel p:	0		

Ultimate Strength Design

BENDING SUMMARY					
Mu/(Mn*Φ):	0.298	Φ:	0.9	a (in):	2.587
Mu (k-ft):	18.717	Location (in):	0	c (in):	3.234
Mn (k-ft):	69.76	Load Comb:	20	d (in):	50

Detail Report - Bending Details

Bending Details	Description
Maximum Moment	The maximum moment in the wall.
Location	The height of the wall where the maximum bending moment is located.
Load Combination	The load combination that produced the maximum bending moment.
Section Modulus (S)	The uncracked section modulus. This is based on the effective thickness and the length of the wall.

Bending Details	Description
Tension Steel A_{sv}	The area of tension steel in this region of the wall panel.
Percentage of Steel (p)	The reinforcement ratio in this region of the wall panel.
$k*d$	The length of the compression block.
j	The ratio of the distance between the centroid of the compressive and tensile forces (j).
a	Depth of compression block.
c	Depth of Neutral Axis.
d	Depth of section from compression fiber to centroid of tensile reinforcement.

In Plane Masonry - Cracked Section Details

```

CRACKED SECT ANALYSIS
fm = fa + fb      : .144  ksi
C                : 28.493 k
T                : .57   k

```

See the [Masonry Wall - Design](#) topic for more information.

In Plane Masonry - Shear Details

```

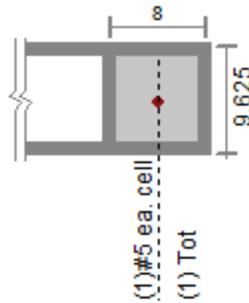
SHEAR DETAILS
Max Shear       : .085  k
Location        : 0    ft
Load Comb       : 1

Corresponding M : .2473 k-ft
M / (V*d)       : 1.0916
Shear St Area   : Not Req.
Shear Spacing   : N/A
Peri of bars    : N/A

```

Detail Report - Shear Details

Shear Details	Description
Moment	The moment corresponding to the maximum shear check is reported along with the M/Vd ratio.
Shear Bar Spacing	The spacing of shear steel in the region of the wall panel.
Perimeter of Bars	Used in the bond stress checks for ASD design.



No horizontal reinforcement required

NOTE: All units are in "in."

This section of the detail report is meant to provide a visual confirmation to the user of the boundary zone width and reinforcement.

Detail Report - Out of Plane

Out of Plane Walls - Input Echo

The top section of the detail report echoes back the input information used in the design of the wall region or lintel. This information is summarized below:

CRITERIA		GEOMETRY		MATERIALS	
Code:	TMS 402-16: ASD	Total Height (ft):	3	Masonry f'm (ksi):	1.5
Special Insp:	No	Eq Sld Thickness:	3.4"	Masonry Em (ksi):	1350
Wall Area:	RMEH	Block Grouting:	UngROUTed	Steel fy (ksi):	60
Type of Design:	ASD	Grout/Bar Spacing:	N/A	Steel E (ksi):	29000
Reinforced:	No			Block Material:	Conc 115 pcf
Vertical Bar Size:	N/A			Grout Weight:	140 pcf
				Wall Dead Wt (ksf):	0.032
				Mortar Type:	Type M or S
				Cement Type:	Portland, Lime/Mortar

The information in the Criteria section is mostly described in detail in the section on [In Plane Detail Report](#). The End Face Dist is the distance from the edge of wall to centroid of vertical bar.

The **Materials** column can be mostly modified under the Materials button on the Data Entry toolbar.

The **Geometry** column gives the basic geometry of the wall. Block grouting and the grout spacing are specified in the Wall Panel Editor. The block nominal width is input under the Design Rules>Masonry wall tab.

The **Eq Solid Thickness** comes from the "Wall Area Method" in the [Wall Design Rules - Masonry Wall](#) tab.

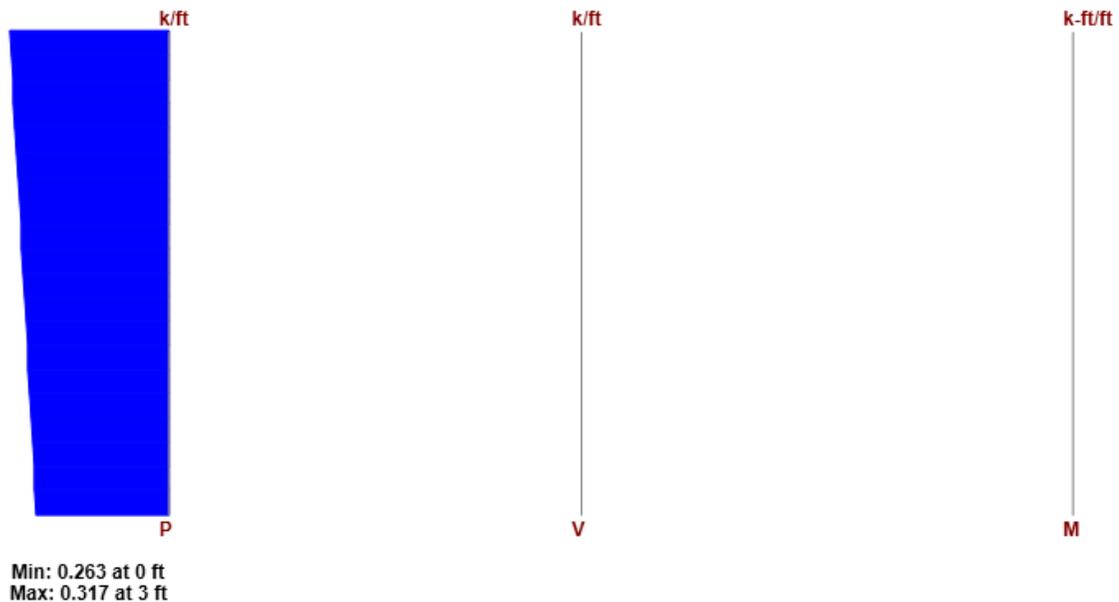
Out of Plane Walls - Force Diagrams and Code Summary (ASD)

The next section of the detail report displays the envelope axial shear and moment diagrams as well as a summary of the code checks performed on the transverse wall.



Note: The forces given in the detail report are given on a per foot basis. Thus these forces are averaged over the width of a given region. For specialty loading conditions, you may need to create much smaller regions to get more accurate local load conditions.

ENVELOPE DIAGRAM



The **Combined Check Summary** gives maximum overall code checks considering the effects of both bending and axial stresses. This section is not reported for MSJC strength design.

The **Axial Summary** and **Bending Summary** give the values used in the axial and flexural code checks at the location which controls the combined check $(f_a/F_a) + (f_b/F_b)$.

For ASD design, the maximum bending stress in the flexural reinforcement is reported as **fs**, and the allowable steel bending stress as **Fs**. These values are given for the Load Combination and section that produce the maximum code check (f_s/F_s) .

The **Shear Summary** reports the maximum shear demand on the beam as **fv** or **Vu**. The allowable shear stresses are then reported as **Fv**.

AXIAL DETAILS					
fa/Fa:	0.021	Max Axial (k/ft):	0.317	Rad gyration r (in):	2.08
fa (ksi):	0.008	Location (ft):	3	h'/r:	17.308
Fa (ksi):	0.369	Load Comb:	14		
BENDING DETAILS					
fa/Fa + fb/Fb:	0.021	M (k-ft/ft):	0	An/ft (in ²):	40.8
fb (ksi):	0	Location (ft):	3	Ix/ft (in ⁴):	139.3
Fb (ksi):	0.5	Load Comb:	14	Sx/ft (in ³):	46.3
(fb - fa)/Fbt:	0				
P / (0.25*Pe):	0.0008844				
Fbt (ksi):	0.033				
SHEAR DETAILS					
fv/Fv (ksi):	0	V (k/ft):	0	Corresponding M (k-ft/ft):	0
fv (ksi):	0	Location (ft):	0	Corresponding P (k/ft):	0
Fv (ksi):	0.039	Load Comb:	N/A	M / (V*d):	0
				Gammag:	1

Detail Report - Design Details

Design Details	Description
Radius of Gyration (r)	The out of plane radius of gyration for the wall.
h/r	Slenderness ratio of the wall.
k	This value multiplied by d gives the depth to the neutral axis.
d	The depth between the extreme face of masonry in compression and the center of the tension reinf.
j	This value multiplied by d gives the moment arm between the compression and tension resultants.
Width for Shear	This is the value used to calculate the shear capacity. For fully-grouted walls, this is simply the center to center spacing of vertical reinforcement. For partially-grouted walls, this value is conservatively taken as the width of the grouted cell plus the thickness of the web and end wall on either side. An example of this can be seen in Example 6.3 (p6.65) of "Design of Reinforced Masonry Structures" by Taly, copyright 2001.
Corresponding M & P	These are the moments and axial forces used to calculate the shear capacity. These are both conservatively taken as the maximum M & P in the entire region, rather than the M & P at the location of maximum shear.
M/(V*d)	This is the M/Vd ratio used to calculate the shear capacity.

T-Section Analysis

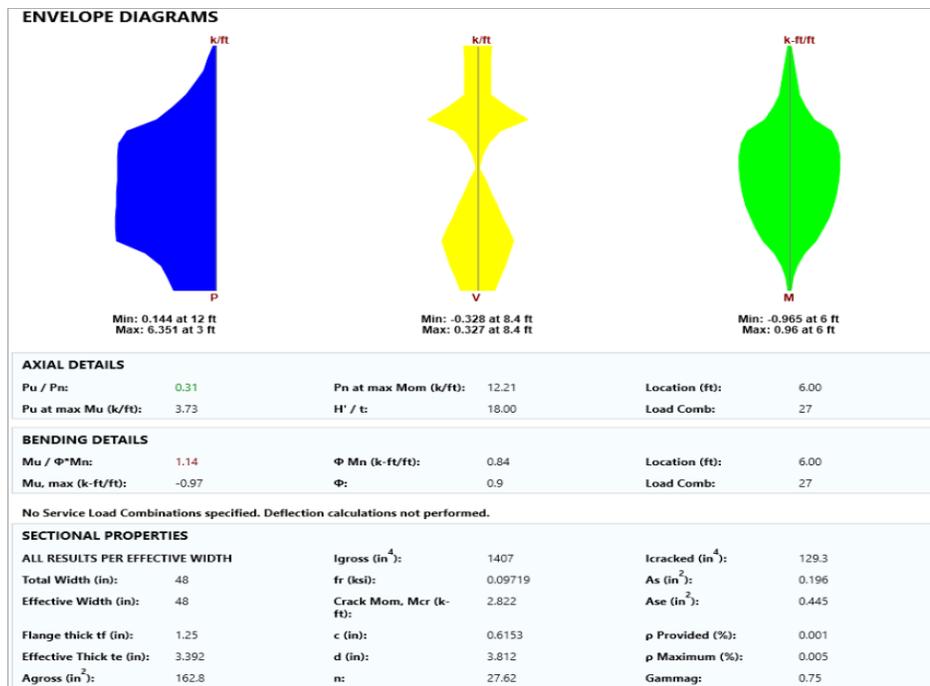
See the [Masonry Wall - Design](#) topic for information on a T-section analysis.

T-SECTION ANALYSIS
 kd : **1.609** in
 Flange thickness: **1.375** in
 Web width : **8.5** in

Detail Report - Out of Plane Slender

Out of Plane Walls - Force Diagrams and Code Summary (Strength)

The next section of the detail report displays the envelope axial shear and moment diagrams as well as a summary of the code checks performed on the out-of-plane loaded slender wall.



The **Axial Details** provide the axial code check for pure compression forces. The **Pn at max Mom** value reported is the maximum allowable axial force based on Section 9.3.5.4.2

The **Bending Details** are explained below:

- **Mu / ϕ *Mn** represents the ratio of applied moment to moment capacity of the wall
- **Mu, max** is the out-of-plane bending moment at the controlling location of the wall.
- **ϕ Mn** is the out-of-plane moment capacity of the wall at the controlling location. It is calculated per Section 9.3.5.2.

- **phi** is the strength reduction factor specified in Section 9.1.4.
- **Location** is the elevation of the wall which resulted in the highest ($M_u / \phi * M_n$) ratio. It is the location at which M_u and P_u are used to calculate ϕM_n .
- **Load Combination** is the load combination which resulted in an M_u and P_u which yielded the highest ($M_u / \phi * M_n$) ratio.

**Note:**

If there is an over-reinforced section this can cause a failure in the masonry wall solver. If this occurs the program will produce an **Over-reinforced** message. To reconcile this you will need to either increase the thickness of masonry block, decrease the grout spacing or decrease the bar size or number of bars in the wall.

BENDING DETAILS

Mu / $\Phi * M_n$:	Over-reinforced	ΦM_n (k-ft/ft):	0.00	Location (ft):	1.40
Mu, max (k-ft/ft):	0.11	Φ:	0.9	Load Comb:	21

Deflection Details

Dmax/Dlimit is a code check result. Thus, if the value is larger than one, your wall fails deflection criteria. **Dmax (Δ_{max})** is the calculated deflecting using the iterative slender wall design procedure detailed in the [Masonry Wall - Design](#) topic. The **Dlimit (Deflection Limit)** is the allowable deflection from TMS 402-13 Section 9.3.5.5.

**Note:**

- Both the **Service** and the **Masonry** flags must be checked on the Design tab of the Load Combinations for the deflections to be checked for that load combination.
- The forces in the region are given on a per foot basis and are an average of the forces over the width of the region.

Sectional Properties (Strength)

SECTIONAL PROPERTIES

ALL RESULTS PER EFFECTIVE WIDTH		I_{gross} (in⁴):	436.5	I_{cracked} (in⁴):	83.31
Total Width (in):	48	f_r (ksi):	0.09719	A_s (in²):	0.785
Effective Width (in):	36	Crack Mom, M_{cr} (k-ft):	1.198	A_{se} (in²):	0.842
Flange thick t_f (in):	1	c (in):	1.599	ρ Provided (%):	0.008
Effective Thick t_e (in):	2.625	d (in):	2.812	ρ Maximum (%):	0.005
A_{gross} (in²):	94.5	n:	27.62	Gammag:	0.75

The center to center distance between reinforcing **Total Width** and the **Eff Width** (which accounts for partially grouted walls) are both reported in this section. The **Flange Thick t_f** refers to the

thickness of the face shell whereas the **Effective Thickness** refers to the overall effective depth of the block.

The gross area and moment of inertia are reported along with the modulus of rupture, cracking moment, neutral axis, modular ratio between steel and masonry, and the cracked moment of inertia.

The area of steel **As** and the effective area of steel **As_e** are reported, with the effective area based upon $As + Pu/fy$.

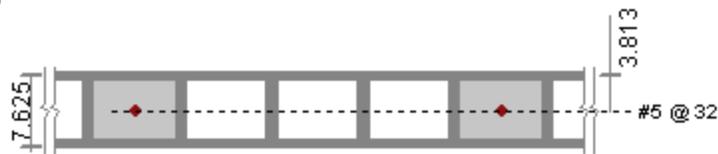
The gross steel ratio (rho gross) is calculated as the area of steel divided by the total area of the section.

The **Rho Provided (%)** is calculated as the area of steel divided by the total width times the effective depth (As/bd). This is limited to a **Rho Maximum (%)** value of $0.5 * \rho$ balanced per the UBC-97 Section 2108.2.3.7. This provision was implemented in future codes as well.

The Sectional Properties gives a number of the properties used during the iterative slender wall design procedure. All of these properties are reported based on a section of wall equal to the center to center spacing of the reinforcement.

Out of Plane Masonry - Cross Section Detailing

CROSS SECTION DETAILING



NOTE: All units are in "in."

This section of the detail report is meant to provide a visual confirmation to the user of the spacing and location of reinforcement.

Detail Report - Lintels

Lintels - Criteria / Materials / Geometry

The first section of the detail report echoes back the basic input parameters (Criteria, Materials, Geometry) entered by the user. An example is shown below:

The geometry portion and the dead weight of the wall warrant further explanation.

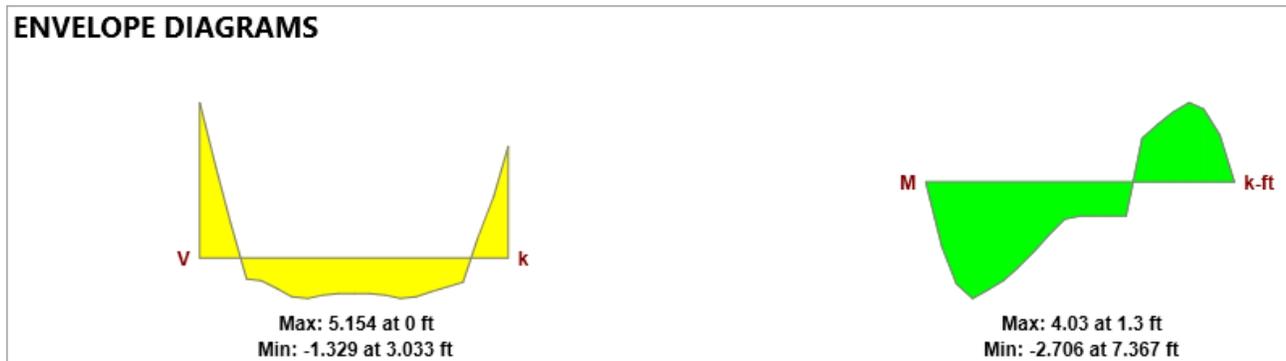
Detail Report - Lintels

Attribute	Description
Dist to Top of Wall	This is the distance from the top of the lintel to the top of the wall and is used in the calculation of the arching action of the lintel loads.

Attribute	Description
Actual Length	This is the total width of the opening.
Bearing Length	This is the bearing length of the lintel on either side of the opening.
Effective Length	The Center to Center distance between lintel supports. Assumed to be length of the opening plus half of the bearing area on each side.
Effective Width	The thickness of the block used to define the lintel.
Effective depth	Equivalent to the "d" distance, the distance from the effective compression face to the centerline of the tension reinforcing.
Total Depth	This is the total depth of the fully grouted portion of the lintel.
Beam Dead Weight	This is based on the user defined density for the Lintel / Opening.
Wall Dead Weight	This is based on the wall self weight or the self weight of the region immediately above the opening.

Lintel Detail Reports - Diagrams and Code Check Summary

The next section of the detail report provides the envelope shear and moment diagrams as well as a summary of the code checks for the Lintel.



Lintel Detail Reports - Design Details

The next section of the detail report gives further details for the design checks performed on the lintel.

DESIGN DETAILS

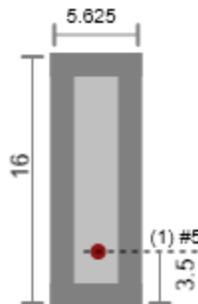
BENDING DETAILS					
fs/Fs:	0.348	Fm (ksi):	0.5	Ms (k-ft):	11.566
fs (ksi):	11.151	M (k-ft):	4.03	Steel Area As (in ²):	0.307
Fs (ksi):	32	Location (ft):	1.3	Steel Ratio ρ:	0.004
fm/Fm:	0.435	Load Comb:	14	k:	0.349
fm (ksi):	0.218	Mm (k-ft):	9.258	j:	0.884

SHEAR DETAILS					
f _v /F _v :	0.739	F _v Max (ksi):	0.116	M / (V*d):	6.25e-8
f _v (ksi):	0.057	V (k):	5.154	Tie Spacing:	Not Required
F _v (ksi):	0.077	Location (ft):	0		
F _{vm} (ksi):	0.077	Load Comb:	14		

This **Bending Details** show the magnitude, location and load combination that correspond to the maximum moment used in the design as well as the area of flexural steel required (A_s) and the steel ratio (rho).

The **Compressive Stress Block Details** show the parameters used in calculating the strength of the lintel.

The **Shear Details** show the magnitude, location and load combination that correspond to the maximum shear used in the design, the M/(V*d) ratio for shear design, and the required spacing of ties (if applicable).

Lintel Detail Reports - Cross Section Detailing

NOTE: All units are in "in."

This section of the detail report is meant to provide a visual confirmation to the user of the basic geometry and reinforcement provided in the lintel.

Masonry Wall - Seismic Design

Masonry walls can now be designed for seismic design provisions. Here the process is described.

**Note:**

- Currently seismic design in masonry walls is only supported under the ACI 530-13 and TMS 402-16 codes. All code references given in this topic are specific to these codes.
- Seismic design will be performed when the following input are met:
 - Supported codes (ACI 530/TMS 402 2013 or 2016)
 - Seismic design rules are assigned properly
 - Seismic load combinations are used in solution

Seismic Design Rules

For seismic design of masonry walls to be performed, the masonry wall panel must be assigned a Seismic Design Rule. Three default Seismic Design Rules are provided in the Seismic Design Rules spreadsheet - Masonry Walls tab as a starting point. Users are responsible to edit/create seismic design rules that fit specific design scenarios.

Masonry Wall Seismic Design Rule				
Hot Rolled Frame		Concrete Walls		Masonry Walls
	Label	Wall Types	Special Boundary Elements	1.5x Shear ASD
1	Ordinary	Ordinary	<input type="checkbox"/>	<input type="checkbox"/>
2	Intermediate	Intermediate	<input type="checkbox"/>	<input type="checkbox"/>
3	Special	Special	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>

Masonry Wall Seismic Design Rule

Column	Description
Label	The label is a user defined text string that is used as a unique identifier for each of the seismic design rules defined for the structure. The program comes pre-loaded with a number of generic seismic design rules that you can edit to fit their design projects.
Wall Type	These are the three masonry wall types supported for masonry wall seismic design in RISA-3D.
Special Boundary Elements	This setting indicates that the wall will be designed with Special Boundary Elements per Section 9.3.3.6. The program will not check for the maximum area of vertical reinforcement requirements for walls with this checkbox turned on in the seismic design rules. Refer to the Design Considerations section for additional information on checks provided for walls designed with Special Boundary Elements.

Column	Description
	<ul style="list-style-type: none"> This setting is turned on by default for Special wall types. But you can edit the seismic design rules to meet your design needs.
1.5x Shear ASD	<p>This setting applies a 1.5x factor to the shear demand of the walls with the seismic design rule applied in models solved with either TMS 402-16: ASD or ACI 530-13: ASD codes.</p> <ul style="list-style-type: none"> This setting is turned on by default for Special wall types columns.

The following are the three possible states for the Label and Wall Type columns.

- **Ordinary:** This wall type will perform design checks based on the minimum reinforcement requirements of *Ordinary Reinforced Masonry Shear Walls* per section 7.3.2.4.
- **Intermediate:** This wall type will perform design checks for *Minimum Reinforcement Requirements* of Chapter 7, in addition to additional requirements of *Intermediate Reinforced Masonry Shear Walls* per section 7.3.2.5.
- **Special:** This wall type will perform design checks for *Minimum Reinforcement Requirements* of Chapter 7, in addition to additional requirements of *Special Reinforced Masonry Shear Walls* per section 7.3.2.6. These requirements include:
 - Additional requirements are checked for *Special Reinforced Masonry Walls* per section 8.3 (ASD Solution) and 9.3 (Strength Solution) in RISA-3D.

Design Considerations

All Wall Panels with Seismic Design Rules applied will be checked for the following Minimum Reinforcement Requirements.

Minimum Reinforcement Requirements

All masonry design walls assigned a seismic design rule will be checked for minimum reinforcement requirements per ACI 530/TMS 402 section 7.3.2. including:

- Vertical Reinforcement
 - At Wall Ends (for wall regions at the ends of the wall panel)
 - Minimum Bar Size
 - Maximum Bar Spacing Around Openings (for applicable Wall Panel Regions)
- Horizontal Reinforcement
 - At Top of Wall Region
 - Minimum Bar Size
 - Maximum Bar Spacing Around Openings (for applicable Wall Panel Regions)

Additionally, all wall types are checked for Mortar type requirements per ACI 530/TMS 402 section 7.4.4.2.2.

Special Reinforced Masonry Walls - Additional Requirements

Additional checks are provided for wall types assigned a “Special” seismic design rule per section 7.3.2.6. These additional checks include:

- Maximum vertical spacing of vertical reinforcement
 - Shall be the smallest of one-third of the length of the wall region, one-third the height of the wall region and 48 inches.
- Maximum spacing of horizontal reinforcement
 - Shall be the smallest of one-third of the length of the wall region, one-third the height of the wall region and 48 inches.
- Minimum cross-sectional area of vertical & horizontal reinforcement
 - The sum of vertical and horizontal cross-sectional area shall be at least $0.002A_g$ (vert), where A_g (vert) is the cross-sectional area of the wall panel region (on a horizontal plane)
 - Minimum vertical & horizontal cross-sectional area shall be $0.0007A_g$, where A_g is the gross cross-sectional area perpendicular to reinforcement being considered



Note: All seismic checks performed in the program are listed in the seismic detailing section per regions in the detailed report. Users can refer to section 7.3.2.6 for additional requirements not checked by the program.

ASD - Additional Requirements

The ACI 530/TMS 402 code has additional seismic detailing requirements for masonry shear wall panels, some of which are considered in RISA-3D. The seismic detailing requirements in this section are only considered if the model is solved using one of the supported Service (ASD) masonry codes.

Max. Flexural Rho Check

Masonry Wall Panels with a Seismic Design Rule of “Special”, a shear span ratio (M/Vd_v) ≥ 1 , and an axial load $\geq 0.05f'_m A_n$, shall be checked for a maximum reinforcement ratio equal to ρ_{max} , calculated based on ACI 530/TMS 402 Equation 8-20:

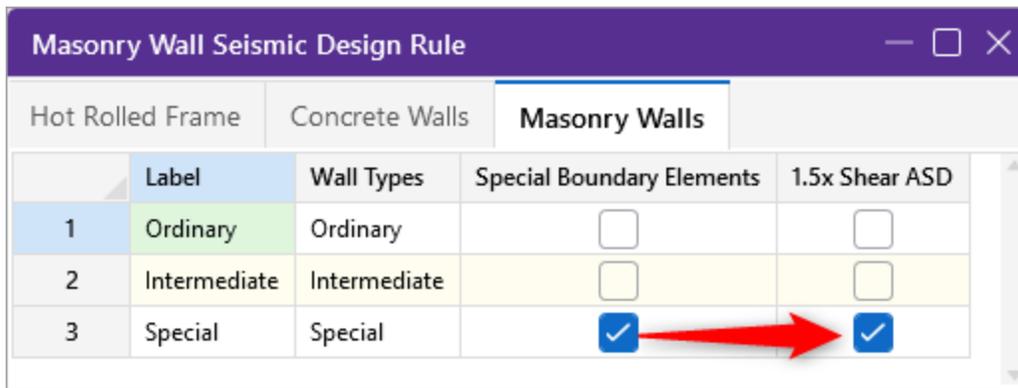
$$\rho_{max} = \frac{n f'_m}{2 f_y \left(n + \frac{f_y}{f'_m} \right)}$$

Where:

- n is the modular ratio of steel reinforcement
- f'_m is the compressive strength of masonry
- f_y is the yield strength of steel reinforcement

ASD Wall Shear Stress Increase Check

Masonry Wall Panels with a Seismic Design Rule of “Special” will be checked if a ASD shear stress increase of 150% (1.5x factor) is applied to the in-plane shear demand as required in Section 7.3.2.6.1.2. The program automatically handles this requirement if the “1.5x Shear ASD” setting is turned on in the Seismic Design Rules - Masonry Walls tab, and will apply a 1.5 factor to the shear demand in the wall region.



Masonry Wall Seismic Design Rule				
Hot Rolled Frame		Concrete Walls	Masonry Walls	
	Label	Wall Types	Special Boundary Elements	1.5x Shear ASD
1	Ordinary	Ordinary	<input type="checkbox"/>	<input type="checkbox"/>
2	Intermediate	Intermediate	<input type="checkbox"/>	<input type="checkbox"/>
3	Special	Special	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>

LRFD - Additional Requirements

The ACI 530/TMS 402 code has additional seismic detailing requirements for masonry shear wall panels, some of which are considered in RISA-3D. The seismic detailing requirements in this section are only considered if the model is solved using one of the supported Strength (LRFD) masonry codes.

Special Boundary Elements Requirements

For Masonry Wall Panels with a Seismic Design Rule of “Special” and walls “Special Boundary Elements” setting turned on in the Seismic Design Rule, the program will determine if Special Boundary Elements are required per Section 9.3.6.6.1. Two conditions are considered to determine if Special Boundary Elements are required:

$$P_u \leq 0.10 A_g f'_m \quad \text{and} \quad \frac{M}{V d_v} \leq 1.0$$

or

$$P_u \leq 0.10 A_g f'_m \quad \text{and} \quad V_u \leq 3 A_{nv} \cdot \sqrt{f'_m} \quad \text{and} \quad \frac{M}{V d_v} \leq 3.0$$

If any of the conditions above hold true, then Special Boundary Elements are not required per Section 9.3.6.6.1. If both of the conditions above do NOT hold true, then Special Boundary Elements are required. The program does not design the walls with the Special Boundary Elements. Users shall proceed to the requirements of Section 9.3.6.6 for further design requirements if special boundary elements are required.

**Note:**

- The requirements of Section 9.3.6.6.1 are checked per wall panel regions, not the entire wall panel.
- Wall is assumed to be rectangular in shape and geometrically symmetrical.
- For wall panels designed using Special Boundary Elements, no checks for maximum vertical reinforcement are required per ACI 530/TMS 402. Refer to TMS 402/602-16 and ACI 530-13 for additional information on ductility requirements.

Design Seismic Shear Capacity Limits

For Masonry Wall Panels with a Seismic Design Rule of “Special”, the program checks if the Design Shear Strength, ϕV_n , exceeds a shear value (V_u) that corresponds to 1.25 times the nominal flexural strength of the wall, calculated as follows:

$$V_u = 1.25 \left(\frac{M_n}{M_u} \right) V$$

Where:

- M_n is the in-plane nominal flexural strength of the wall panel region
- M_u is the in-plane moment (flexure) demand in the wall panel region
- V is the actual shear demand in the wall panel region

Additionally, the Design Shear Strength must not exceed 250% of the shear demand in the wall panel region:

$$V_u = 2.5 V$$

If any of the minimum and maximum limits to the Design Shear Strength is not met, a “Fail” will be indicated in this check in the Seismic Detailing section of the Wall Panel Detail Report for the wall panel region.

Maximum Area of Flexural Tensile Reinforcement (max. Flexural Rho Check - Strength)

For walls not designed with Special Boundary Elements (without the “Special Boundary Elements” and designed using a Strength code, all wall panel regions will be checked against maximum flexural reinforcement requirements per Section 9.3.3.2. Section 9.3.3.2 stipulates minimum levels of strain at the extreme tension fiber, which the program handles individually for each wall panel region, with a minimum tension reinforcement strain factor, “ α ”, for each masonry seismic wall type.

The tension reinforcement strain factor is dependent on the wall type, seismic response modification R factor, and the shear span ratio (M/Vd_v) ratio, as shown in the table below:

Minimum Tension Reinforcement Strain Factor, α , at Strength Level for Masonry Seismic Shear Walls

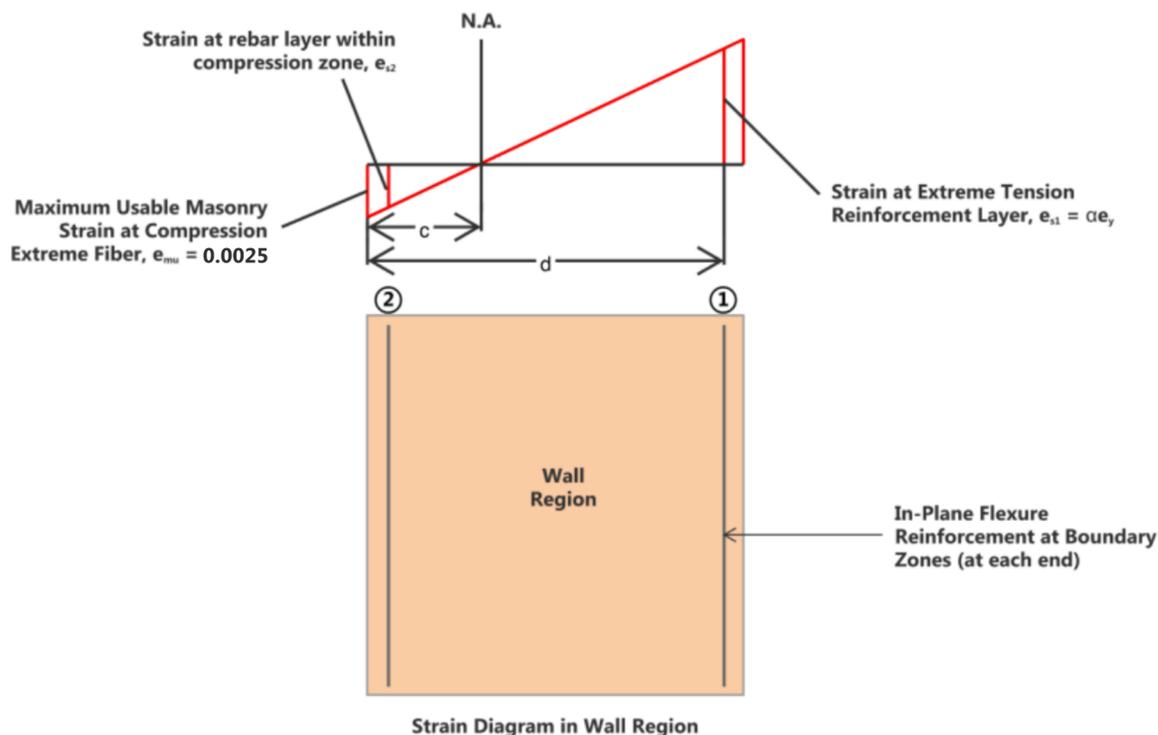
Masonry Seismic Wall Type	R	$M_v / V_v d_v$	α
Special	5	≥ 1.0	4.0
Intermediate	3.5	≥ 1.0	3.0
All Others	ALL	≥ 1.0	1.5
	≥ 1.5	< 1.0	1.5
	< 1.5	< 1.0	NONE

The strain ratio for the extreme rebar layer, calculated as e_s/e_y , where e_s is the strain in the steel at the extreme tension rebar layer and e_y is the yield strain of reinforcement.

The procedure for determining compliance is as follows:

- Wall Panel Region in-plane reinforcement design is done based on the flexure (moment) demand in the wall panel region.
- Shear span ratio (M/Vd_v), depth to neutral axis c and seismic response modification R factor are reported.
- Required minimum tension reinforcement strain factor is obtained per the table above.
- Strain ratio, calculated as e_s/e_y , where e_s is the strain in the steel at the extreme tension rebar layer and e_y is the yield strain of reinforcement, is compared to α .
- If $e_s/e_y \geq \alpha$, then reinforcement in the wall panel region is in compliance with Section 9.3.3.2.
- If $e_s/e_y < \alpha$, then the reinforcement in the wall panel region is NOT in compliance with Section 9.3.3.2.

- The strain in the steel at the extreme tension rebar layer is calculated based on the depth to neutral axis c , and the effective depth of the wall region, d , as shown in the diagram below:



The calculation of the strain at the extreme tension reinforcement layer is as follows:

$$e_{s1} = \frac{(d - c)}{c} \cdot e_{mu}$$

**Note:**

- Out of plane reinforcement is not considered in meeting ductility requirements for Section 9.3.3.2.
- The effects of compression reinforcement is considered in the in-plane flexural capacity of the wall panel regions when calculating the location of the neutral axis.
- The maximum usable masonry strain at the extreme compression fiber, ϵ_{mu} , is assumed to be 0.0025 for the calculation of the strain at the extreme tension steel for Seismic Detailing checks. For assumptions of the extreme compression fiber for non-seismic in-plane checks, refer to the [In Plane Design - Strength](#) section.
- Section 9.3.3.2.1 requires axial forces to be taken from the loading combination given by $D + 0.75L + 0.525QE$. The program has no way of determining if this load combination is present in the model, or if it governs the design of the wall panel. Therefore, it is recommended that users include this load combination in the set of solved load combinations, and determine if ductility requirements are met for this specific load combination.

Assumptions and Limitations

- All walls are assumed to be laid in a running bond.
- No checks are provided for seismic detailing requirements at roof and floor diaphragms, at wall corners, for dowels or through control joints. Additional requirements may apply for seismic design categories D, E and F. Refer to TMS 402-16 or ACI 530-13 Chapter 7 for additional requirements.

Masonry Wall - Seismic Design Results

Results for Seismic Detailing of Masonry Walls are provided via Spreadsheets and wall panel Detail Report.

Design Spreadsheet Results

Wall Panel Seismic Detailing

Results for Seismic Detailing checks are provided in the *Wall Panel Design* spreadsheet - **Masonry Seismic** tab. Results are provided per region, and the Seismic Detailing column provides a summary of the Seismic Detailing checks. For a more detailed output, check the "Seismic Detailing" portion of the Wall Panel Detail Report.

Masonry Wall Panel Seismic Code Checks (TMS 402-16: ASD)							
Concrete In		Concrete Out		Concrete Seismic		Masonry In	
Masonry Seismic		Wood Wall Axial		Wood Wall In-Plane		Wood Header	
						CFS Wall Axial	
						CFS Wall In-Plane	
	Wall Panel	Seismic Design Rule	Region	UC Shear In Plane	LC	Seismic Detailing	BZ Req'd?
1	WP1	Special	R2	0.395	14	Fail. See Detail Report.	No
2			R3	0.288	14	Fail. See Detail Report.	No
3			R5	0.544	13	Fail. See Detail Report.	No
4			R6	0.562	13	Fail. See Detail Report.	No
5			R7	0.616	13	Fail. See Detail Report.	No
6			R8	0.475	13	Fail. See Detail Report.	No

Masonry Wall - Seismic Detailing

Column	Description
Wall Panel	The Wall Panel column displays the masonry seismic wall panels defined in the model.
Seismic Design Rule	The Seismic Design Rule column displays the rule governing the design.
Region	The Region column lists the masonry seismic wall panel region that the results are based on.
UC Shear In Plane	The Shear UC In Plane column shows the in plane shear code check. A value greater than 1.0 for any of these values would indicate failure.
LC	The LC column adjacent to the UC Shear In Plane column, shows the load combination that produces each of the highest code check values.
Seismic Detailing	The Seismic Detailing column displays whether or not the masonry seismic wall panel passed or failed the analysis.
BZ Req'd?	<p>The BZ Req'd? column displays whether or not there are boundary elements/zones required.</p> <ul style="list-style-type: none"> • If No is displayed, it means there are no boundary elements/zones required. • If Yes is displayed, it means boundary elements/zones are required.

Reinforcing Spreadsheet Results

Wall Panel reinforcement is provided in the Concrete Reinforcing Spreadsheet - Masonry Wall tab. Results are provided per region. For a more detailed output, check the "Seismic Detailing" portion of the Wall Panel Detail Report.

Masonry Wall Reinforcement						
Beam Bending		Beam Shear		Column Bending		Column Shear
Concrete Wall			Masonry Wall		Masonry Lintel	
	Wall	Region	Hor. Bar Size	Vert. Bar Size	Boundary Reinf.	
1	WP1	R2	#5@12 oc	#5@48" oc (ctr)	1-#5	
2		R3	#5@8 oc	#5@48" oc (ctr)	1-#5	
3		R5	#5@12 oc	#5@48" oc (ctr)	1-#5	
4		R6	#5@12 oc	#5@48" oc (ctr)	1-#5	
5		R7	#5@12 oc	#5@48" oc (ctr)	1-#5	
6		R8	#5@12 oc	#5@48" oc (ctr)	1-#5	

Detailed Reports

Seismic Detailing Checks - Wall Panel Region

Seismic Detailing checks are provided for each of the wall panel regions at the top of the report, when Region is chosen in the first drop down. Each Seismic Detailing section has a header "Region RX (Seismic Detailing)", where "R" indicates a "Region" and "X" the region number.

Detail Report for - R1 of WP21

< Wall Panel Label WP21 >
Region v
R1 v
Seismic v

Detail Report | JSON Detail Report | Log Viewer.

SEISMIC DETAILING REQUIREMENTS (SPECIAL SHEAR WALL - ASD)

AT OPENINGS:

Min. Vert. Bar at Wall Openings (Section 7.3.2.3.1):	1-#4 within 16 in each side of openings	N/A
Min. Hori. Bar at Wall Openings (Section 7.3.2.3.1):	1-#4 at top and bottom of openings	N/A

AT WALL EDGE:

Min. Vert. Bar at Wall Edge (Section 7.3.2.3.1):	1-#4 within 8 in of wall ends	PASS
--	-------------------------------	------

AT TOP OF WALL:

Min. Hori. Bar at Top of Wall (Section 7.3.2.3.1):	1-#4 within 16 in of top of wall	PASS
--	----------------------------------	------

HORIZONTAL REINFORCEMENT:

Min. Hori. Bar Size (Section 7.3.2.3.1):	#4 (0.196 sq. in)	PASS
Max Hori. Bar Spacing (Section 7.3.2.3.1):	16.6667 in	PASS
	Max Spacing = min(1/3L, 1/3H, 48in) = 16.667 in	
	*Masonry assumed to be laid in running bond.	
	*Horizontal bars shall be embedded in grout.	
Min. Hori. Reinf. Area (Section 7.3.2.6c):	Ahor >= 0.0007Ag(horz)	PASS
	Ahor = 2.061 in ²	
	0.0007Ag(horz) = 0.598 in ²	
	*Masonry assumed to be laid in running bond.	

VERTICAL REINFORCEMENT:

Min. Vert. Bar Size (Section 7.3.2.3.1):	#4 (0.196 sq. in)	PASS
Max Vert. Bar Spacing (Section 7.3.2.5):	16.6667 in	PASS
	Max Spacing = min(1/3L, 1/3H, 48in) = 16.667 in	
	*Masonry assumed to be laid in running bond	
Min. Vert. Reinf. Area (Section 7.3.2.6c):	Aver >= larger of (0.0007Ag(vert), 1/3Ahor)	PASS
	Aver = 1.266 in ²	
	0.0007Ag(vert) = 0.267 in ²	

Print
Export to Xps
 Add to Full Report
Close

The Seismic Detailing checks depend on which design philosophy is being performed (e.g. ASD vs Strength) and the Wall Panel’s Seismic Design Rule. See below an example of the checks provided for a Strength design of a Masonry Wall Panel with a “Special” Seismic Design Rule designed with Special Boundary Elements:

Legend:

- A “PASS” indicates that the seismic detailing check is in compliance with the referenced section.
- A “FAIL” indicates that the seismic detailing check is NOT in compliance with the referenced section.
- A “N/A” indicates that the seismic detailing check is NOT APPLICABLE for the Wall Panel Region.

Wood Wall - Design

The wood wall panel element allows you to easily model, analyze and design wood walls for in-plane loads. Here we will explain the wood specific inputs and design considerations. For general wall panel information, see the [Wall Panels](#) topic. For wood wall results interpretation, see the [Wood Wall Results](#) topic.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Wood Walls**.

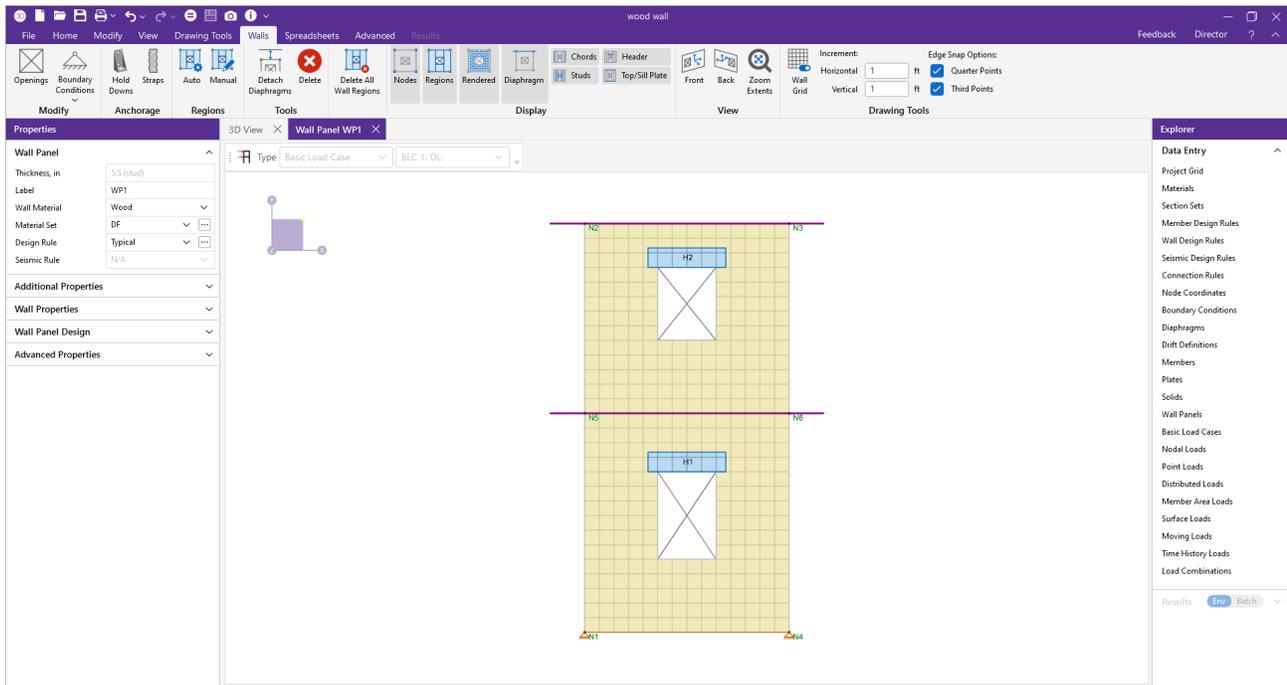
Full code checking and design can be performed on the panel sheathing, studs, chords, headers and hold-downs based on the following codes:

- The 2018 edition of the NDS. Lateral wall design can be performed using either 2015 or 2021 SDPWS when using 2018 NDS.
- The 2015 edition of the NDS
- The 2012 edition of the NDS
- The 2005/08 edition of the NDS (National Design Specification)
- The 2014 edition of the CSA O86
- The 2009 edition of the CSA O86

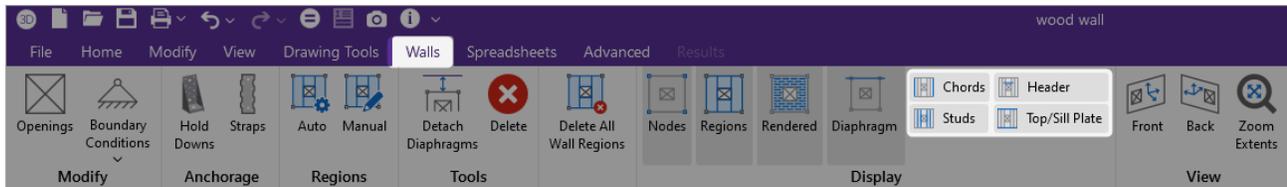
Wood Wall Input

The **Walls** ribbon provides specific information and options for the modeling/analysis of wood walls.

To open the **Walls** ribbon, double-click on a wall panel within the 3D view window. This action also opens the wall panel in the Wall Panel Editor.



Wood View Controls



The following table provides descriptions for the wood wall panel view controls.

Wood Wall Panel View Controls

Control	Description
 Chords	The Chords icon lets you toggle the display of the wall panel region chords on and off.
 Header	The Header icon lets you toggle the display of the headers on and off.
 Studs	The Studs icon lets you toggle the display of the wall studs on and off.
 Top/Sill Plate	The Top/Sill Plate icon lets you toggle the display of the top/sill plates on and off.

Design Rules

You must set up design rules for the stud/chord sizes, as well as make database selection for shear panels and hold-downs. This is done in the **Wall Design Rules** spreadsheet in the **Wood Wall**

(**Studs**) and **Wood Wall (Fasteners)** tabs. See the [Wood Wall - Design Rules](#) topic for more information.

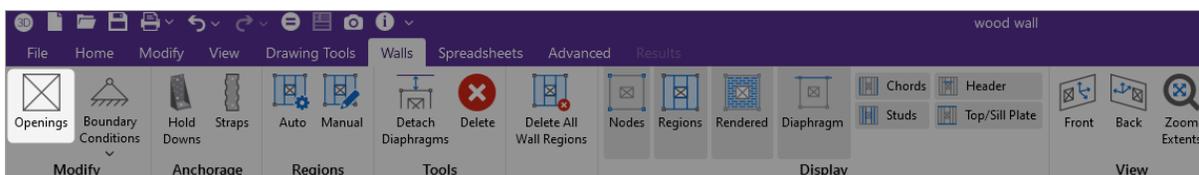
Create Openings in Wood Walls with Headers

Within the **Walls** ribbon, you have the option of adding rectangular openings to wood wall panels.

Draw an Opening

To draw an opening:

1. Click the **Openings** icon.



2. Click on the two nodes or grid intersections which make up the two diagonal corners of your opening.

When an opening is drawn, a header beam is automatically created above the opening.

View/Edit Header Beam Properties

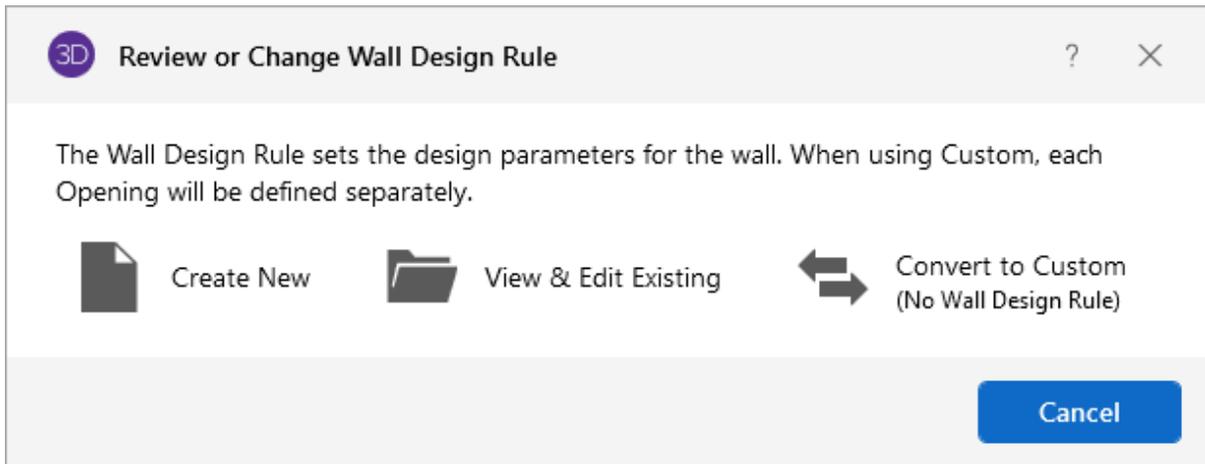
To view or edit the properties of a header beam:

1. Open the [Wall Design Rules](#) spreadsheet.

In the case where you may have different opening headers in the same wall, you can define the different headers by following the next step.

2. Double-click inside the boundary of the drawn opening, or click the **Design Rule**  ellipsis button in the 'Properties' panel.

This opens the **Wood Wall Definition Editor** window for that particular opening, which then lets you modify the header properties.



The following options appear:

Review or Change Wall Design Rule

Option	Description
Create New	Create New lets you create a new Wall Design Rule to be applied to the wall.
View & Edit Existing	View Edit Existing lets you view as well as make edits to the Wall Design Rule that is currently applied to the wall.
Convert to Custom	The Convert to Custom (No Wall Design Rule) option is only available when each opening is defined separately. This option lets you define a header size and material that is different from the Wall Design Rule. This option only applies to Masonry Walls.



Note: You set a custom header only if you have multiple headers in the same wall that are different sizes.

- Click on the **View & Edit Existing** or **Convert to Custom** (if making custom modifications) to open the related window.

Refer to [Wall Design Rules](#) for more information on the available design rule options.

Hold-Downs and Straps

Hold-downs and straps are automatically added to your walls in their required locations. For a “Segmented” design, you must have hold-downs or straps at the bottom corners of each of your design segments. For “Perforated” and “Force Transfer Around Openings (FTAO)”, hold-downs straps are only allowed at the two far corners of the wall panel. The program will not permit the drawing of hold-downs or straps at locations where they are not allowed.

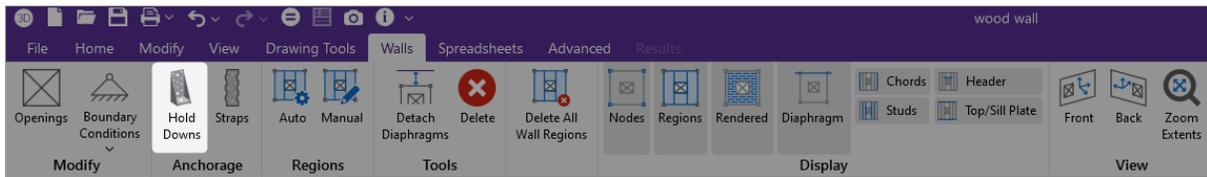
If there are custom locations that you want to add hold-downs or straps, you can do this from the **Walls** ribbon.

Hold downs

'Hold downs' can represent the anchorage of your wall to the foundation or the connection of shear wall chords between floors.

To add hold-downs to the base of your wall:

- Click the **Hold Downs** icon on the **Walls** ribbon.



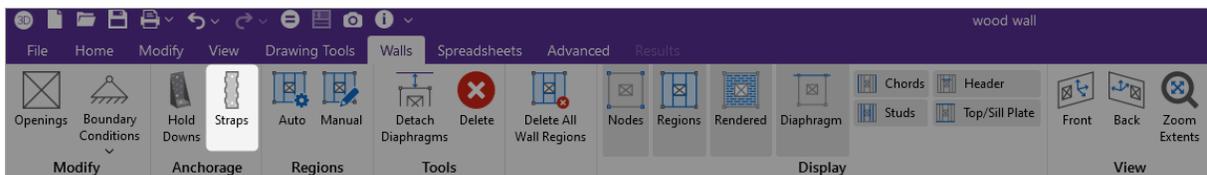
Hold downs must be added after regions are created and can only be added at the corners of regions. Hold down requirements depend on the type of wall design you are performing.

Straps

'Straps' represent the connection of the current wall panel to a wall panel below.

To add straps to the base of your wall:

- Click the **Straps** icon on the **Walls** ribbon.



Straps also can only be added after regions are created and can only added at the corners of regions. Strap requirements also follow the same logic as hold downs as to where they must be defined as far as regions are concerned.

**Note:**

- All boundary conditions for wall panels should be defined in the wall panel editor. Adding external boundary conditions can create problems.
- The locations of hold-downs and straps define where the program will calculate tension forces in your walls.
- If you have applied your hold-downs or straps for the Segmented design with openings in your wall, then they will be required at the interior of the wall panel. However, running Perforated or FTAO does not require hold-downs or straps at the interior of the wall panel. Thus, if you toggle between Segmented to Perforated or FTAO, then the hold-downs or straps you drew will be removed at the interior of the walls. If you switch back to Segmented, the interior hold-downs or straps will come back again.
- By default, the program will automatically add hold-downs to the base of walls that have boundary conditions applied to them and will apply straps to walls that have walls below.
- All Straps and Hold-downs are placed at the base of walls only. So, if you would like to put a strap or hold-down between floors, you would apply it to the base of the upper wall.

The output for straps and hold-downs will show up on the detail report for the wall panel. More information on this can be found in the [Wood Wall Results](#) topic.

General Requirements for Shear Walls

The design of wood shear walls per the available design codes requires that many criteria are satisfied before a wall can be considered adequate. For RISA to work within this framework, we require that certain modeling practices be followed. Outlined below are many general wall modeling practices and limitations. Also included are specific requirements for each of the three design procedures for wood wall design with openings: Segmented, Force Transfer Around Openings, and Perforated.

The three different types of shear walls are defined in Section 4.3 of the NDS *Special Design Provisions for Wind and Seismic*.



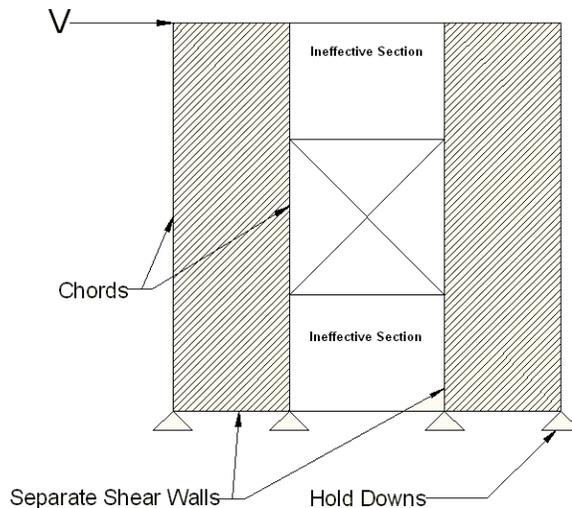
Note: The Canadian CSA O86 design code only supports the Segmented method, as described below.

Segmented Method

Where there is a wall panel with openings, the area above and below the openings is disregarded and the wall is designed as being made up of separate, smaller shear walls.

Like all wall panels, the segmented wood wall is broken into a series of meshed plate elements to represent the overall wall. The portions of the segmented shear wall that are considered "ineffective" in resisting shear are modeled with a plate elements that have a significantly reduced shear stiffness so that they will not receive any significant moment or shear from the FEM analysis.

See the diagram below for more information:



In addition, the out of plane stiffness and in plane stiffnesses of the segmented wood wall are modeled separately based on different assumed plate thicknesses. This is done to insure that the shear stiffness is based entirely on the properties of the sheathing and is not influenced by the out-of-plane stiffness of the wall studs.



Note:

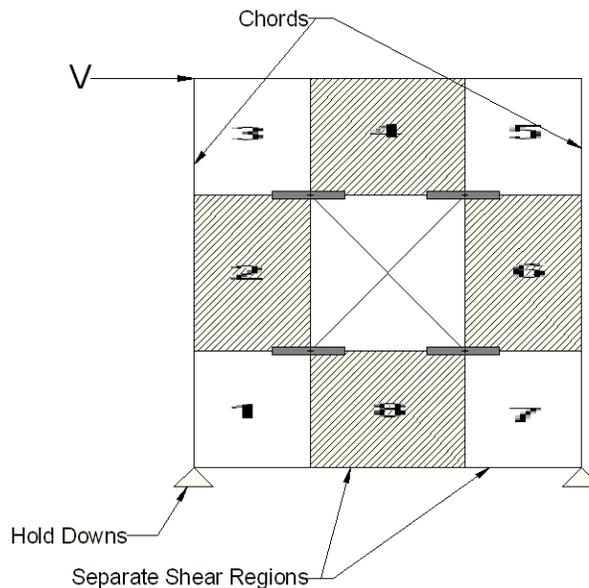
- If you have several stacked Segmented wall panels with misaligned openings, you will receive the Warning Message shown below upon solution. This message means that RISA-3D has assumed that the strap force from the above wall panel will be spread out across the region directly under it. Therefore, you need to be aware of this assumption and detail the wall panel accordingly.



- A shear panel design will be chosen for the worst-case region in a segmented wall. That panel will then be used for all regions in that wall. The worst-case region is the one that has the highest Shear UC value.

Force Transfer Around Openings Method

This method is based on a rational analysis of the wall generally referred to as the "Diekmann Method" This method is documented in detail in *Design of Wood Structures ASD/LRFD (6th edition)* by Breyer, Fridley, Cobeen, and Pollock. The assumption being that straps and blocking can be added at the corners of the openings to transfer the sheathing forces across these nodes. This method essentially allows you to use the entire area of the wall (minus the opening) to resist the shear in the wall.



The basic assumptions made in the shear wall analysis are the following:

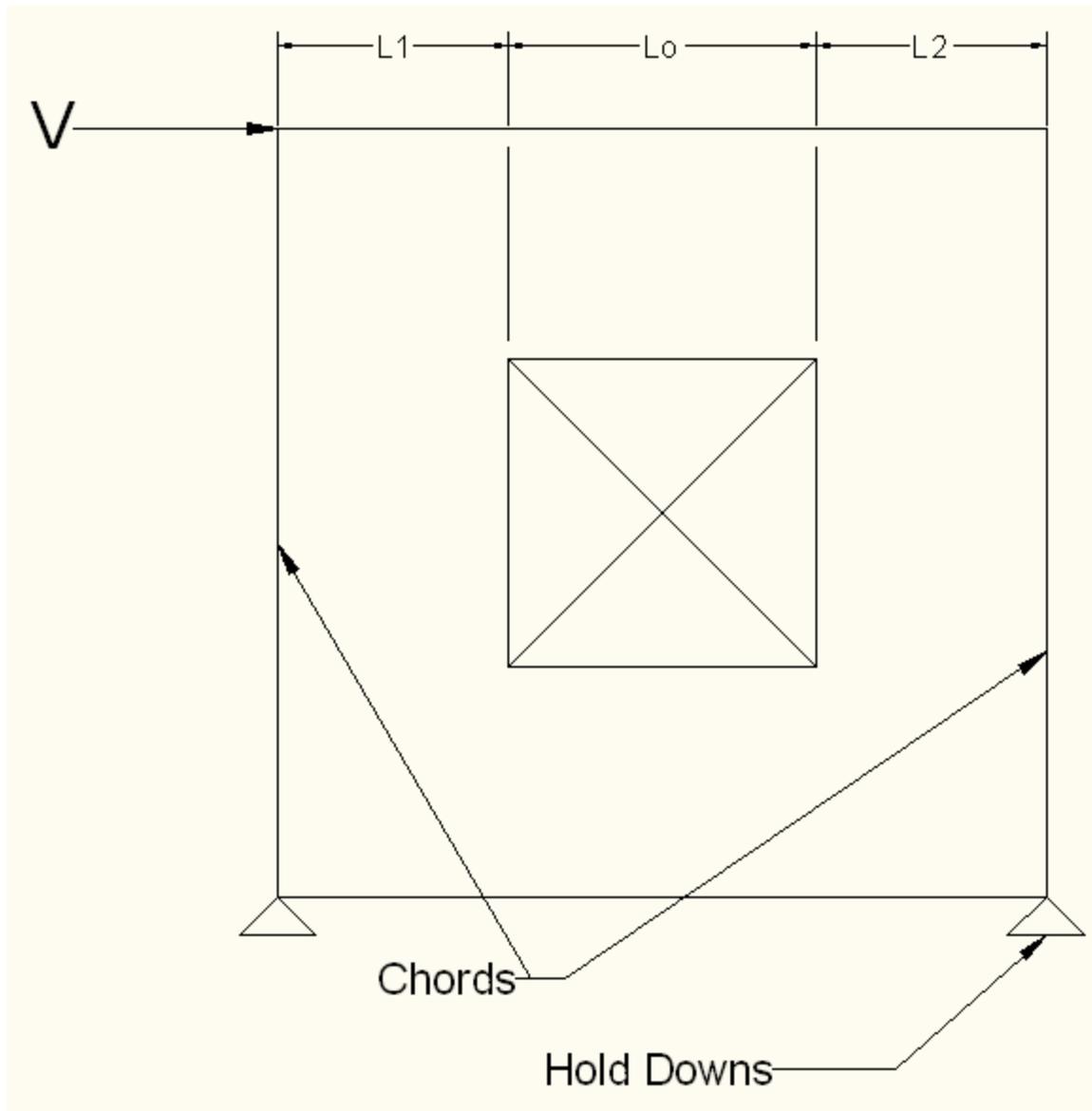
- The sheathing resists the shear forces. The average shear force in each block of the wall (numbered 1~8 as shown in the image above) is used as the controlling shear force in that location. The maximum shear in each of these locations will control the design of the wall. The program uses an area weighted average of the F_{xy} plate forces to determine the average shear for each block.
- The moment at the edge of each block that is above or below an opening is assumed to be transmitted across the opening interface by horizontal tension straps or compression blocks as shown in the image above. The required force is reported to the user, but the design and length of these elements is left to the engineer.
- The moment at the edge of each block that is to the right or left of an opening is assumed to be transmitted across the opening by tension straps or compression blocks. Since it is likely that the sheathing and king studs will be capable of transmitting these forces, these elements are not shown in the image above. However, these forces are reported so that the design of the studs and sheathing in these regions can be checked by the engineer to consider these effects.

**Note:**

- The program is limited in the automatic generation of regions for walls with multiple openings that are not aligned. Therefore, it is recommended that complex walls with multiple openings be simplified based on engineering judgment to facilitate easier detailing of the force transfer around these openings.
- This design method is not available for Canadian CSA O86 design. If you select this design method, the program will automatically change it back to Segmented.

Perforated Method

This method for design of wood shear walls with openings may end up being the most cost effective. It only requires hold downs at the corners of the wall, yet it does not require straps or blocking around the openings. A perforated shear wall design approach is, however, subject to a number of code constraints about when it can be used.



The basic design procedure for perforated walls is to essentially ignore the portions of the wall that do not have full height sheathing and treat the wall instead as a significantly shorter wall. This amplifies the chord and hold down design forces significantly while at the same time increasing the design unit shear as shown in the equations below:

$$T := \frac{V \cdot h}{C_o \cdot \Sigma L_i} \quad u_{\max} := \frac{V}{C_o \cdot \Sigma L_i}$$

Where:

$$\Sigma L_i = L1 + L2$$

**Note:**

- The perforated method of design also has many caveats that are given in Section 4.3.5.3 of the 2015 *Special Design Provisions for Wind & Seismic* / 4.3.5.6. of the 2021 *Special Design Provisions for Wind & Seismic*. The program will not allow the design of wall panels that do not follow these provisions.
- For multi-story perforated shear walls, the amplified hold down forces required per the code become difficult to interpret for the lower walls. Therefore, when RISA calculates the hold down forces for the lower wall it assumes that the reduction coefficients for the upper wall are identical to the values for the lower wall. This will result in conservative hold down forces when the lower wall has more openings than the upper wall. But, it may be un-conservative for situations where this is not the case. This assumption does NOT affect the shear design of either wall, nor does it affect the strap force calculations in the upper wall.
- This design method is not available for Canadian CSA O86 design. If you select this design method, the program will automatically change it back to Segmented.

Shear Capacity Adjustment Factor, Co:

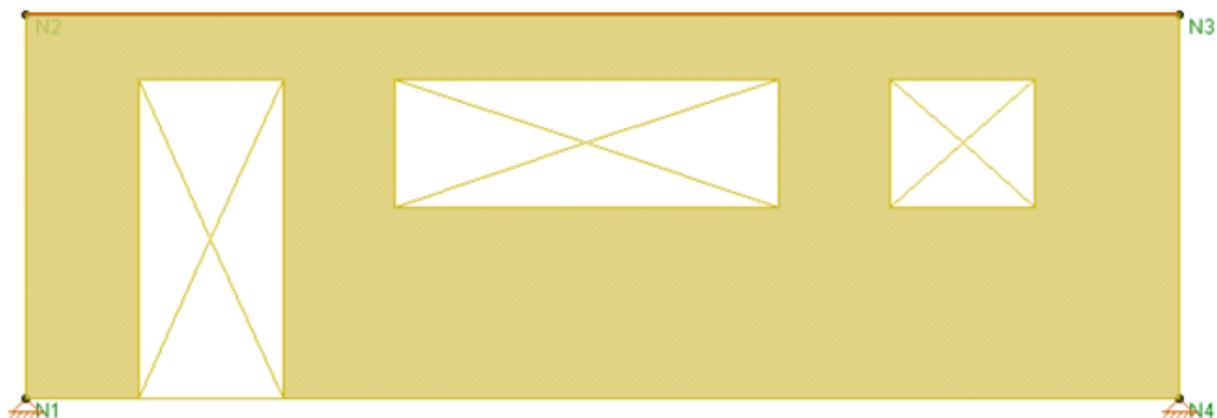
The NDS *Special Design Provisions for Wind & Seismic* lists Effective Shear Capacity Ratio (Co) values that are used in calculating the nominal shear capacity of perforated shear walls. Because the tabular values are limited to wall heights of 8' & 10', RISA instead uses equation (4.3-5) from the 2015 *NDS SDPWS* (as shown below) or equation (4.3-6) from the 2021 *NDS SDPWS* to calculate the Co factor for any height wall.

$$C_o = \left(\frac{r}{3 - (2 * r)} \right) * \frac{L_{tot}}{\Sigma L_i}$$

Where:

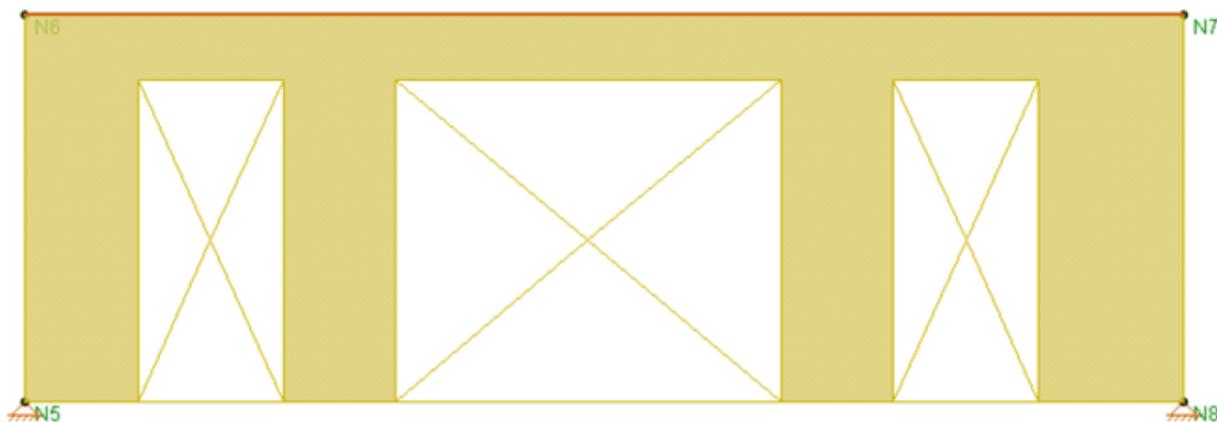
$$r = \frac{1}{1 + \left(\frac{A_o}{h * \Sigma L_i} \right)}$$

When using these equations, RISA takes Ao as the true area of the openings. However, Table 4.3.3.5 of the 2015 *NDS SDPWS* (Table 4.3.3.4 from older versions of the NDS SDPWS) references Co values based on all opening heights equal to the maximum opening height. Therefore if you want the program to calculate Co equal to that in Table 4.3.3.5 of the NDS, you must draw all openings as equal to the maximum height. Please see the image below for reference.



$C_o = 0.84$

VS.



$C_o = 0.53$

(Full height openings assumption- matches NDS Table 4.3.3.4)



Note: When the assumption is made that all opening heights are equal to the maximum opening height then the equation produces values of C_o that are within 1% for all the values shown in the NDS table.

C_o Limitation for Stacked Wall Panels

For wall panels that are stacked on one another, each wall panel has its own C_o value. These C_o values will be different if the openings in each wall are different. This is the source of a limitation in the program.

The program determines the design forces in each wall panel separately using the finite element solution. We then take those design forces and factor them for the C_o values. The problem is that the design forces for the bottom wall are affected by the C_o value for the upper wall. Thus, the forces that come down on a lower wall from an upper wall have been factored for the C_o from the upper wall. RISA-3D does not consider this. The program takes the forces from the finite element solution and uses the C_o value only for the wall in question.

Thus, the highest level in a stacked wall configuration will always use C_o in a correct manner. The lower wall(s), however, will be conservative if they have more openings than the upper wall(s) and can be unconservative if the upper wall(s) have more openings than the lower wall(s).

This limitation must be considered when designing Perforated walls that are stacked.

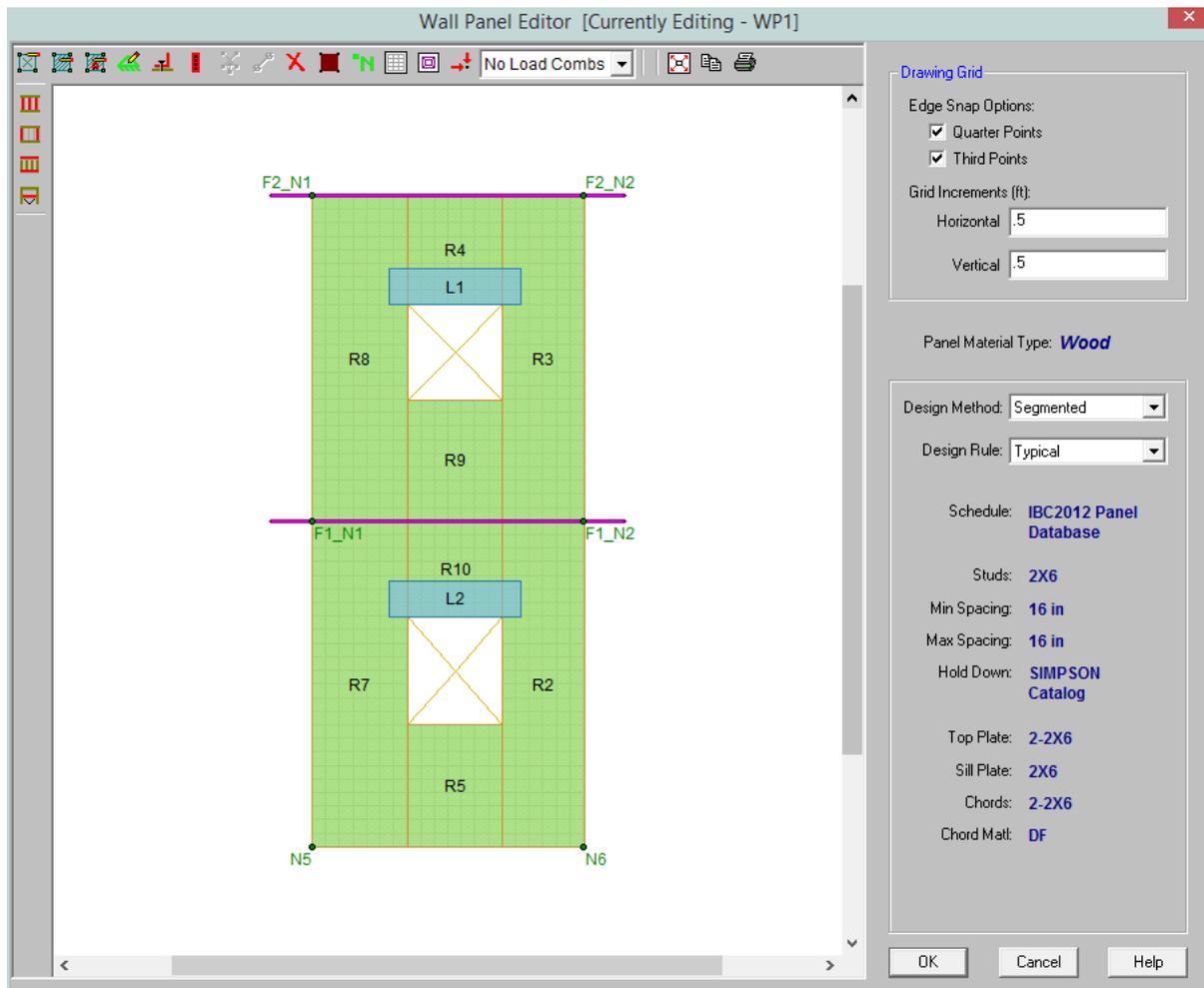
General Program Functionality and Limitations

RISAFloor and RISA-3D Interaction

When using RISAFloor and RISA-3D in combination, the interface transitions nicely between the two programs. Here is a quick walk through of this interaction.

Input Interface

1. Model the entire building (gravity and lateral members) within RISAFloor. Be sure to model all openings and regions for all of the wall panels in the model.



Note: You can not modify your openings or regions in RISA-3D. All region and opening modifications must be taken back to RISAFloor to be done.

2. Add loading and solve the model.
3. Click on the **Director** link to take the model into RISA-3D.
4. Once in RISA-3D, add your hold downs and straps to your wall panels.

**Note:**

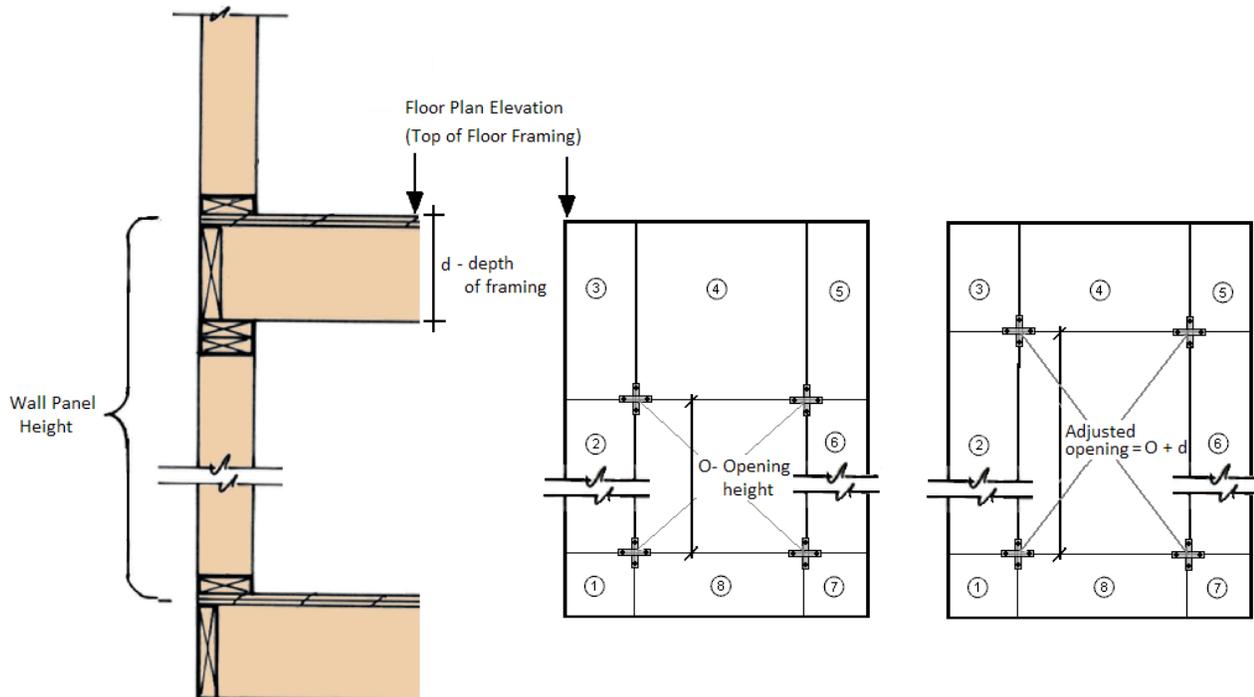
- Hold downs and straps can not be added to wall panels in RISAFloor.
- Hold downs are only allowed to be added to the lower corners of the wall panel for the Perforated or Force Transfer Around Openings design methods.
- Hold downs are required at the corners of all full height regions in the wall panel for the Segmented design method.
- The Design Rule, Design Method and SSAF can be changed in either program at any time.

Modeling Tips- Platform Framing

The wood wall height in RISA is measured from the bottom of the sill plate to the top of the floor framing as shown below. However, platform framing causes the wall height to be significantly shorter than a RISA model would represent.

The design method FTAO is the only method that will cause significant problems because the region above openings is assumed to be much larger than it is built. In order to adjust for this framing depth, you can adjust your opening height to include the depth of the floor framing. This will reduce the portion of the wall above the opening thus reducing the amount of area to transfer shear forces.

In Segmented and Perforated design methods, the portion above and below the opening are not used to transfer shear forces.



Limitations for Hold-downs / Straps (Including deformation)

- The program does not currently have a database for continuous tie rod hold-downs. However, the design results that are presented for each floor are intended to provide the type of information necessary for the design of these types of hold downs.
- The d_a values (deflection at peak load) from our hold-down database are based entirely on the manufacturers listed values. These do not include any allowance for shrinkage (which is often as much as 1/4" or more per floor), or crushing of the sill plate.
- There is not currently a d_a value used in the calculation for strap deformation for upper floor levels.

Automatic Boundary Conditions

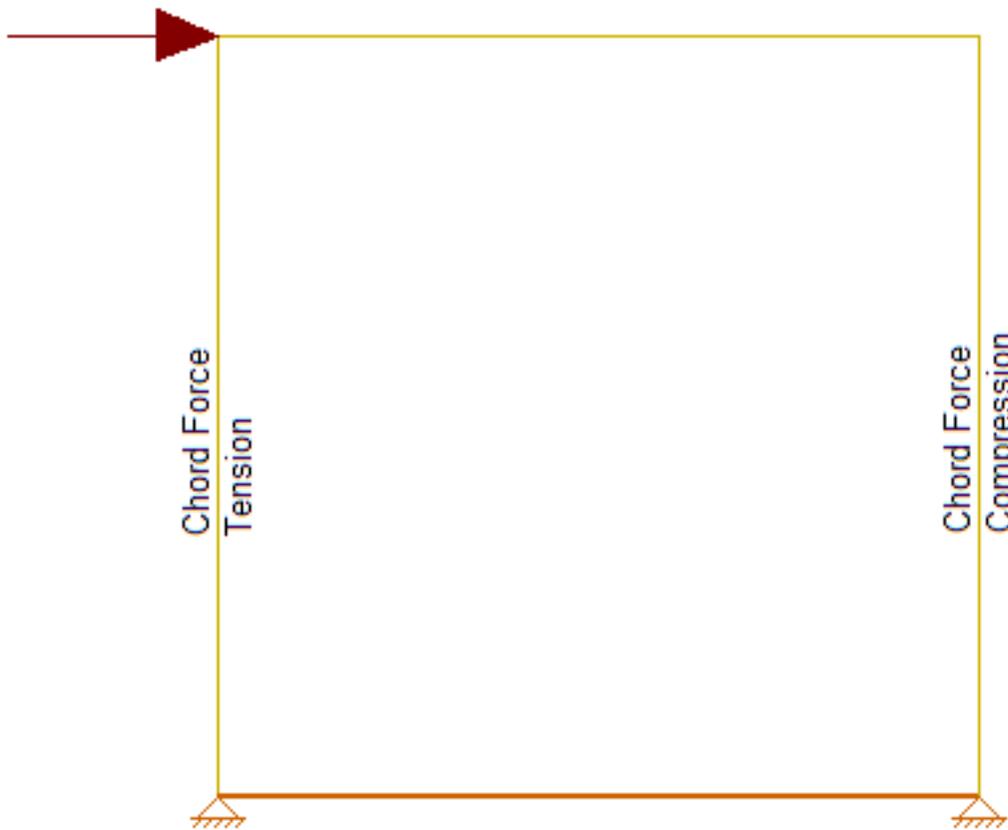
- In RISA-3D, if no boundary conditions or hold downs are defined for wood wall panels at the lowest level of the structure, the program will automatically create hold downs at the corners of the wall panel. If you do not want the hold downs to be automatically created, define a "free" boundary condition at the base of the wall panel in the wall panel editor.

Deflection

- There is currently no code check for drift or deflection for shear wall panels.

Chord Design

The chord design is based on forces that are calculated differently for Compression versus Tension. The tension chord force is calculated including the dead load stabilizing moment as per Section 4.3.6 of the NDS 2015 *Special Design Provisions of Wind & Seismic*. The compression chord force includes the only the tributary area of one stud spacing in the compression force. For Segmented design, the chord forces are found based on each region, and in FTAO and Perforated design methods the chord forces are determined for the entire wall.



Chord forces are calculated per the following equations:

- When the "[Eccentricity](#)" check-box is selected in the Wood Wall Design Rule (Fasteners tab):

Additional Wood Wall Panel Parameters												
Unity Check	Concrete Wall (Rebar)	Concrete Wall (Misc)	Masonry Wall	Masonry In	Masonry Out	Masonry Lintel	Wood Wall (Studs)	Wood Wall (Fasteners)	CFS Wall (Studs)	CFS Wall (Fasteners)		
Label	Schedule	Min Panel Thick [in]	Max Panel Thick [in]	Double Sided Panel?	Max. Nail Spacing	Min. Nail Spacing	HD Chords	HD Chord Matl	Hold Down	Chord Strap	Eccentricity	
1	Typical	AWC 2021 OSB	0.375	0.75	Optimum	6-in.	2-in.	2-2X8	Same as Wall	SIMPSON HoldDowns	SIMPSON Chord Straps	<input checked="" type="checkbox"/>

$$\text{Compression Force} = \frac{M}{L - b_c} + \frac{P}{n}$$

$$\text{Tension Force} = \frac{-M}{L - b_c} + \frac{P}{2}$$

- When the "[Eccentricity](#)" check-box is NOT selected in the Wood Wall Design Rule (Fasteners tab):

Additional Wood Wall Panel Parameters												
Unity Check	Concrete Wall (Rebar)	Concrete Wall (Misc)	Masonry Wall	Masonry In	Masonry Out	Masonry Lintel	Wood Wall (Studs)	Wood Wall (Fasteners)	CFS Wall (Studs)	CFS Wall (Fasteners)		
Label	Schedule	Min Panel Thick [in]	Max Panel Thick [in]	Double Sided Panel?	Max. Nail Spacing	Min. Nail Spacing	HD Chords	HD Chord Matl	Hold Down	Chord Strap	Eccentricity	
1	Typ...	AWC 2021 OSB	0.375	0.75	Optimum	6-in.	2-in.	2-2x8	Same as Wall	SIMPSON HoldDowns	SIMPSON Chord Straps	<input type="checkbox"/>

$$\text{Compression Force} = \frac{M}{L} + \frac{P}{n}$$

$$\text{Tension Force} = \frac{-M}{L} + \frac{P}{2}$$

Where:

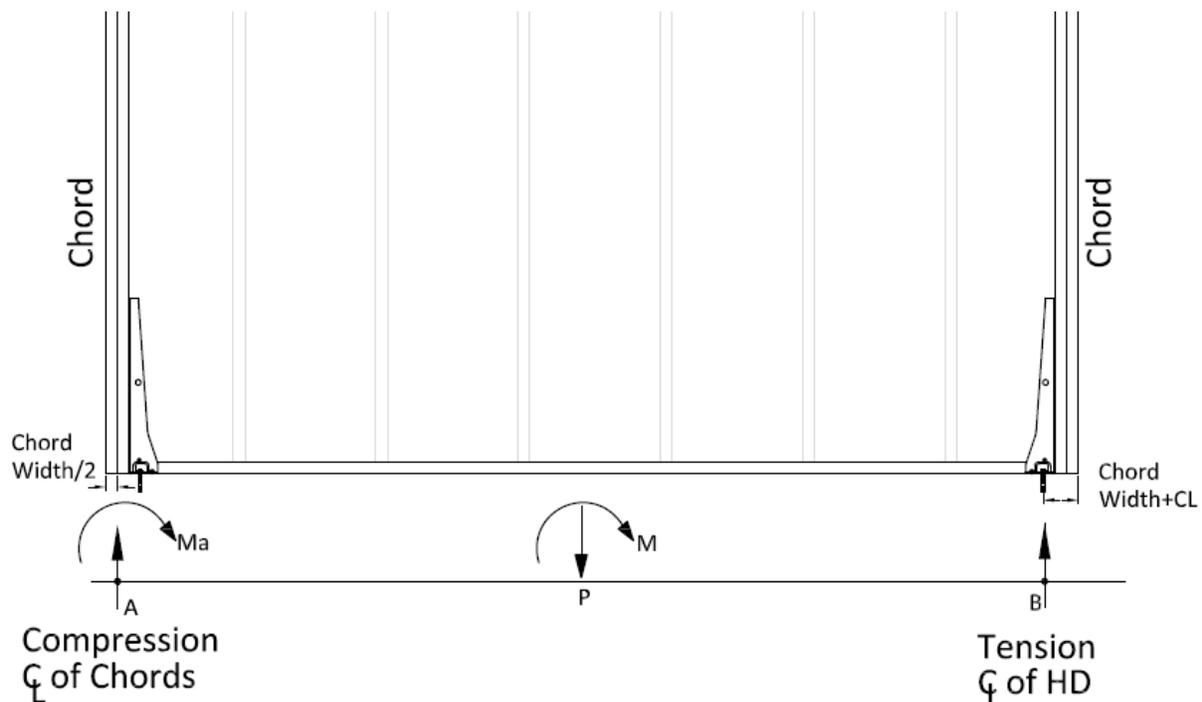
- M = Moment at the base of the wall
- L = Length of the wall
- b_c = width of chord member
- P = Axial force at the base of the wall
- n = Number of Studs = Length/Stud Spacing

**Note:**

- For Perforated Design, the term $M/(\text{Length}-\text{ChordWidth})$ found in the above equation is replaced with $V_h/(C_o \sum L_i - \text{ChordWidth})$ as indicated in the Equation 4.3-8 of the NDS 2015 *Special Design Provisions for Wind & Seismic*.
- RISA-3D models the shear walls using only the sheathing. The vertical resistance occurs only at the tension and compression chords. Thus, if two wall panels are stacked on top of each other, the load transfer will only happen at the chord locations. Therefore, the lateral analysis should agree very well with hand calculations. However, it also means that gravity load design may be more appropriate in RISAFloor.

Hold-Down Force

The Hold-Down force is calculated by finding the Tension chord force. In order to accommodate an unsymmetrical vertical load on the wall, the program adopted a more accurate approach by calculating the moment at the compression side first and then finding the resulting reaction on the tension side.



Hold-Down forces are calculated per the following equations:

- When the "[Eccentricity](#)" check-box is selected in the Wood Wall Design Rule (Fasteners tab):

Additional Wood Wall Panel Parameters												
Unity Check	Concrete Wall (Rebar)	Concrete Wall (Misc)	Masonry Wall	Masonry In	Masonry Out	Masonry Lintel	Wood Wall (Studs)	Wood Wall (Fasteners)	CFS Wall (Studs)	CFS Wall (Fasteners)		
Label	Schedule	Min Panel Thick [in]	Max Panel Thick [in]	Double Sided Panel?	Max. Nail Spacing	Min. Nail Spacing	HD Chords	HD Chord Matl	Hold Down	Chord Strap	Eccentricity	
1	Typical	AWC 2021 OSB	0.375	0.75	Optimum	6-in.	2-in.	2-2X8	Same as Wall	SIMPSON HoldDowns	SIMPSON Chord Straps	<input checked="" type="checkbox"/>

$$\text{Moment at Point } a = M_a = -M + \left(P * \left(\frac{L}{2} - \frac{b_c}{2} \right) \right)$$

$$\text{Reaction at Point } b = R_b = \frac{M_a}{L - 1.5 * b_c - CL}$$

- When the "Eccentricity" check-box is NOT selected in the Wood Wall Design Rule (Fasteners tab):

Additional Wood Wall Panel Parameters												
Unity Check	Concrete Wall (Rebar)	Concrete Wall (Misc)	Masonry Wall	Masonry In	Masonry Out	Masonry Lintel	Wood Wall (Studs)	Wood Wall (Fasteners)	CFS Wall (Studs)	CFS Wall (Fasteners)		
Label	Schedule	Min Panel Thick [in]	Max Panel Thick [in]	Double Sided Panel?	Max. Nail Spacing	Min. Nail Spacing	HD Chords	HD Chord Matl	Hold Down	Chord Strap	Eccentricity	
1	Typ...	AWC 2021 OSB	0.375	0.75	Optimum	6-in.	2-in.	2-2X8	Same as Wall	SIMPSON HoldDowns	SIMPSON Chord Straps	<input type="checkbox"/>

$$\text{Moment at Point } a = M_a = -M + \left(\frac{P * L}{2} \right)$$

$$\text{Reaction at Point } b = R_b = \frac{M_a}{L - CL}$$

Where:

- M_a = moment at point A(CL of compression chord)
- M = moment at base of the wall
- P = Axial load at the base of the wall
- L = length of the wall
- b_c = width of chord member
- R_b = reaction at point B = hold down force
- CL = Hold down eccentricity distance (per selected hold down)

Stud Design

Studs are only designed for load combinations which do not contain a wind or seismic load. The maximum axial load, determined as an envelope force from all of the "gravity" load combinations which have been solved, is determined for each region.

The compression capacity for the specified stud size is calculated with the assumption that the stud is fully braced against buckling about its minor axis (within the plane of the wall). This is because the blocking and sheathing are assumed to provide this bracing. The unbraced length for major axis buckling is taken as the wall height, minus the thickness of the top and sill plates.

The program divides the region axial force by the number of studs that would be present in that region for a given stud spacing. An optimal stud spacing is then selected based on the stud capacity, and the parameters defined in the [Design Rules](#).

Force Distribution

- The lateral force distribution between piers is based on the relative stiffness of the sheathing, not on the length of the shear wall. For example, if you have an 8 foot wall and a 4 foot wall, the 8 foot wall will take more than $8/(8+4)*100\%$ of the force. The moment of inertia in the 8 foot wall will allow for a larger proportion of load to go into that pier.

Shear Capacity Adjustment Factors

40% Increase Factor Wind Load Cases

The allowable shear stress values tabulated in [Appendix F](#) are intended to be the allowable shear for seismic loads. The [solution](#) tab of the Model Settings has a check-box which will automate the 40% shear capacity increase for any load combinations that include Wind Loads. This is called "**Wind ASIF**" in the detailed report output. When the code is set to 2021 SDPWS LRFD, the code allows for a 60% increase in shear capacity for wind loads.

Allowable Stress Reductions for Slender Wall Segments

The program will modify the allowable stress of the wall based on the $2b/h$ adjustment factor per NDS SDPWS section 4.3.3.4 exception 1. This adjustment factor is only applied to the wall for load combinations which include seismic loads. This adjustment factor only affects walls with an aspect ratio between 2.0 and 3.5. This is called "**Cap.Adj. (2w/h)**" in the detailed report output.

For Segmented and FTAO wall panels designed per the NDS 2015, the program also checks the Aspect Ratio factor per section 4.3.4.2. This factor will reduce the sheathing capacity for lateral (Wind or Seismic) load combinations on walls who have a design region whose aspect ratio is greater than 2.0. This is called "**Aspect Ratio**" in the detailed report output.

The final governing factor (minimum of **Cap. Adj. (2w/h)** and **Aspect Ratio**) is reported as the **Gov. H/W Factor**. This is then multiplied by the sheathing capacity to give the final **Adj. Shear Cap.**

Detail Report: WP1 (Region R3)

CRITERIA		GEOMETRY		MATERIALS	
Code:	AWC NDS-18 / SDPWS-21:ASD	Total Height (ft):	15	Wall Studs:	DF
Wall Material:	DF	Total Length (ft):	5	Stud Size:	2X6
Panel Schedule:	AWC 2021 OSB	Region H/W:	3.00	Chord Material:	DF
Optimize HD:	Yes	Cap. Adj. (2w/h):	1.00	Chord Size:	2-2X8
HD Manufacturer:	SIMPSON	Aspect Ratio:	0.88	Top Pl. & Sill:	DF
		Gov. H/W Factor:	1.00	Top Pl. Size:	2-2X8
				Sill Pl. Size:	2X6
				Specific Gravity:	0.5

Specific Gravity Adjustment Factor

The NDS defines an adjustment factor associated with using stud material that is less dense than Douglas-Fir-Larch or Southern Pine. The program automatically accounts for this factor in the design of Wood Shear Walls.

Unblocked Shear Wall Adjustment Factor

Section 4.3.3.2 of the 2015 NDS *Special Design Provisions for Wind and Seismic* is ignored in RISA-3D. The program will always assume that the sheathing panel is blocked for both NDS and CSA design.

Hold Down Adjustment Factor (Applicable to only 2021 SDPWS)

Per footnote 10 of Table 4.3A of the 2021 SDPWS, the nominal unit shear capacity for shear wall shall be multiplied by 0.92 if shear walls are using 10d nails and where hold-downs are attached to the inside face of the end post. The program assumes that all Wood Walls have hold-downs anchored to the inside face of the end post.

Uplift Limitation

Section 4.4 of the 2015 NDS *Special Design Provisions for Wind and Seismic* is ignored in RISA-3D. The program currently does not design wall panels for uplift forces.

Stiffness Adjustment Factors

Shear Stiffness Adjustment Factor (to change FEM results)

This stiffness adjustment factor is set on the Wall Panels spreadsheet and is intended as a way to force the FEM stiffness of the wall to more closely resemble the stiffness from the APA / NDS three term deflection equation.

This adjustment affects the stiffness of the entire wall. Therefore, for segmented walls the engineer may be forced to model the piers separately if they need to adjust the pier stiffnesses independently. In a future release, this may become an automated factor.

Green Lumber

The [Wood Wall \(Studs\)](#) tab of the Design Rules spreadsheet contains a Green Lumber check-box to account for the 50% reduction in the G_a value defined in the NDS footnote.

Stiffness Assumptions

Vertical Direction

RISA uses an orthotropic plate element to de-couple the vertical and shear stiffness of the wood walls. The vertical stiffness will be based on the E value of the studs and chords as specified in the Materials spreadsheet and the thickness of the wall. The thickness is taken as

$$\textit{Thickness} = \frac{\textit{Total Wall Weight}}{\textit{Material Density} * \textit{Wall Length} * \textit{Wall Height}}$$

Where:

$$\textit{Total Wall Weight} = W_{\textit{studs}} + W_{\textit{chords}} + W_{\textit{top plate}} + W_{\textit{sill plate}} + W_{\textit{sheathing panel}}$$

In-Plane Shear

The in-plane shear stiffness will be based on the G_a value designated within the specified nailing schedule divided by the sheathing thickness. Both the G_a value and the sheathing thickness are defined in the specified sheathing schedule.

Out-of-Plane Shear

For the out of plane shear, RISA uses the same Young's Modulus (E) as used in the vertical direction and a thickness that is calculated from the out of plane moment of inertia. This becomes:

$$\textit{Thickness} = \left(\frac{I_{\textit{chord}} + I_{\textit{stud}} + I_{\textit{panel}}}{\textit{Wall Length}} \right)^{1/3}$$

Wood Wall Self Weight

The program will calculate wood wall self weight as a sum of all the weights of the components. The material density is used to calculate the self weight of the studs, chords, top plates, sill plate, and sheathing. These are all then summed together to give the self weight of the entire wall.

**Note:**

- For this calculation, stud height equals wall height minus the thickness of the sill plate and the top plate.
- The number of studs is calculated using the stud spacing specified in Design Rules.

Wood Wall Optimization

The program will optimize wood walls based on the required demand forces. The program can optimize:

- Sheathing for in-plane shear design.
- Stud spacing for axial design.
- Hold-downs for in plane design.

Of these optimizations the two that substantially modifies the stiffness of the wall are the sheathing and the stud spacing. To properly adjust the stiffness requires an iterative solution that updates the stiffness of the model. This includes updating the strength properties of the wall as well as the stiffness. This optimization/iteration can be done automatically by checking **Optimize Masonry and Wood Walls** in the [Model Settings - Solution](#) tab or can be done manually by unchecking this option.

The screenshot shows the 'Model Settings' dialog box with the 'Solution' tab selected. The 'Members' section includes 'No. of Reported Sections' (5), 'No. of Internal Sections' (100), and 'Member Area Load Mesh Size' (144 in²). The 'Wall Panels' section includes 'Approx. Mesh Size' (24 in) and several checked options: 'Transfer Forces Between Intersecting Wood Walls', 'Increase Wood Wall Nailing Capacity for Wind Loads', 'Include P-Delta for Walls', and 'Optimize Masonry, Wood and CFS Walls'. The 'Max. No. of Iterations' is set to 3. The 'Processor Core Utilization' is set to 'Balanced'. The 'Advanced' button is visible, and the 'Apply' and 'Cancel' buttons are at the bottom right.

To update the stiffness portion of the wall, the program must re-solve your model with these updated stiffnesses as this will change the distribution of forces through the model. By checking **Optimize Masonry and Wood Walls** you are telling the program to re-solve automatically. Thus, the program will start with its initial stiffness parameters and solve the model. It will then optimize the wall to meet strength criteria. Another solution will then be run with the new stiffnesses and the program will again optimize the wall to meet strength criteria. This procedure will continue to occur until you reach the **Maximum Number of Iterations** set or until all wall panel results match those of the previous solution.

By unchecking **Optimize Masonry and Wood Walls** the program will only run the solution once and the results will be based on the original configuration. You can then manually optimize your walls using the [Suggested Design](#) spreadsheet.

After the solution is run (with or without optimization) the design results are based on the stiffness used in the last iteration (by unchecking the **Optimize Masonry and Wood Walls** option a single iteration is run). The program will then compare the design of the last iteration with the stiffness used in that last iteration. If the two are the same the results shown are the final results. If the two are not the same the program will then provide these two different results in the **Suggested Design** spreadsheet.

The program will always present results in the output that coincide with the stiffness used in the final solution.



Note: The updating of the stiffness for the model is only required for the sheathing and stud spacing optimization. Thus, hold-down call-outs are optimized automatically.

Suggested Design

Wood Wall Suggested Design

Members | Masonry Walls | **Wood Walls** | CFS Walls

Re-solve with Suggested Shape(s)/Wall(s)

	Wall Panel	Current Panel	Suggested Panel	Current Stud Space[in]	Suggested Stud Space[in]	Use Panel?	Use Stud Space?
1	WP1	S1_3/8_8d@6	RS_3/8_6d@6			<input checked="" type="checkbox"/>	<input type="checkbox"/>

File Home Modify View Drawing Tools Spreadsheets Advanced Results

Envelope LC Dynamic Sort by Combination Sort by Item Filter Results Node Wall TH Trace Export TH Trace Contour Diagram Clear Results Warning Log Suggested Design Detailed Report

Results Force Summation Time History Contour Formatting

In the **Suggested Design** spreadsheet you will get a list of wall panels in your model that are not yet fully optimized, showing the panel and stud spacing of the last iteration and the program optimized values. From here you have the ability to **Use Panel?** or **Use Stud Space?** which means that you want to re-run the solution with the suggested design. You can choose this for each suggestion for each wall panel individually. Once you have these checkboxes checked appropriately click the **Suggested Design** icon in the Results tab to re-solve the model with the suggested design. After this the stiffness matrix is re-formulated and may cause some redistribution of loads through the model. Because of this the suggested design may also update and you may need to **Use Panel?** and/or **Use Stud Space?** multiple times to converge on a solution.



Note:

- If the wall does not show up in the Wood Wall Suggested Design spreadsheet then the current wall panel settings used are the optimal ones.
- If either the stud spacing or the panel fields are blank that means that the current selections are the optimal ones.
- For more information on wood wall optimization see the [Wood Wall - Design](#) topic.
- For more information on member optimization see the [Design Optimization](#) topic.
- Concrete walls do not show up here because the reinforcement optimization does not affect the stiffness of the wall.

Panel Optimization

Wood Wall Panel Schedule

Select Code: AWC 2021 OSB

Available Panel Groups:

- AWC 2021 OSB
- AWC 2021 OSB 0.3125
- AWC 2021 OSB 0.375
- AWC 2021 OSB 0.4375
- AWC 2021 OSB 0.469 (8d)
- AWC 2021 OSB 0.469 (10d)

Current Selection Type: Use All Panels

	Label	Grade	Min Thick [in]	Min Penetration	Over Gypsum	One/Two Sided	Nail Size	Staple Size	Nail
1	S1_5/16_6d@6	St-I	0.312	1.25	No	1	6.000000	n/a	6
2	S1_(2)5/16_6d@6	St-I	0.312	1.25	No	2	6.000000	n/a	6
3	S1_3/8_8d@6	St-I	0.375	1.375	No	1	8.000000	n/a	6
4	S1_(2)3/8_8d@6	St-I	0.375	1.375	No	2	8.000000	n/a	6
5	S1_7/16_8d@6	St-I	0.438	1.375	No	1	8.000000	n/a	6
6	S1_(2)7/16_8d@6	St-I	0.438	1.375	No	2	8.000000	n/a	6
7	S1_15/32_8d@6	St-I	0.469	1.375	No	1	8.000000	n/a	6
8	S1_(2)15/32_8d@6	St-I	0.469	1.375	No	2	8.000000	n/a	6
9	S1_15/32_10d@6	St-I	0.469	1.5	No	1	10.000000	n/a	6
10	S1_(2)15/32_10d@6	St-I	0.469	1.5	No	2	10.000000	n/a	6
11	S1_5/16_6d@4	St-I	0.312	1.25	No	1	6.000000	n/a	4
12	S1_(2)5/16_6d@4	St-I	0.312	1.25	No	2	6.000000	n/a	4
13	S1_3/8_8d@4	St-I	0.375	1.375	No	1	8.000000	n/a	4
14	S1_(2)3/8_8d@4	St-I	0.375	1.375	No	2	8.000000	n/a	4
15	S1_7/16_8d@4	St-I	0.438	1.375	No	1	8.000000	n/a	4

OK Cancel

The procedure that RISA uses for design optimization is fundamentally based upon the assumption that there is a 'cost' to shear capacity, and therefore the ideal panel design would have as little shear capacity as possible to meet code requirements. Once the program has determined the shear demand on the wall it will choose the most economical panel configuration based on that which has a **Shear Capacity** closest to, but not exceeding the shear demand.

The default shear panels in the program come straight from published tables in the IBC, UBC, and CSA O86 design manuals. "St-I" refers to Structural I panels and "RS" refers to Rated Sheathing panels.



Note:

- The shear force listed in the XML spreadsheet is tabulated for the seismic values. These values will automatically be multiplied by 1.4 for wind forces if the [Wind ASIF](#) function is enabled. These values will be multiplied by 1.6 for wind forces when the code is set to 2021 SDPWS LRFD.
- Panels with a label containing the characters "_W" together will be ignored during design optimization.
- "Rated Sheathing" is a shortened form of "Span Rated Sheathing" or APA Rated Sheathing. This terminology comes directly from the Department of Commerce standards DOC PS1 and PS2.

Hold Down Optimization

Hold Down Schedule
? ×

Select Manufacturer Simpson HoldDowns ▾

Available HD Series

SIMPSON HoldDowns

DTT_DF-SP

DTT_SPF-HF

HDU_DF-SP

HDU_SPF-HF

HDB_DF-SP

HDB_SPF-HF

LTT_DF-SP

LTT_SPF-HF

HTT_DF-SP

HTT_SPF-HF

Current Selection Type Optimize From Series ▾

#	Label	Manufacturer	Min Chord Thick [in]	Reqd Chord Species	Bolt Size	Defl at Peak Load [in]
1	DTT1Z-SD9x1.5_DF-SP	Simpson HoldDowns	1.5	DF	N/A	0.17
2	DTT1Z-(6)0.148x1.5_DF-SP	Simpson HoldDowns	1.5	DF	N/A	0.167
3	DTT2Z-SDS1.5_1.5_DF-SP	Simpson HoldDowns	1.5	DF	N/A	0.105
4	DTT2Z-SDS1.5_3_DF-SP	Simpson HoldDowns	3	DF	N/A	0.128
5	DTT2Z-SDS2.5_3_DF-SP	Simpson HoldDowns	3	DF	N/A	0.128
6	DTT1Z-SD9x1.5_SPF-HF	Simpson HoldDowns	1.5	HF	N/A	0.17
7	DTT1Z-(6)0.148x1.5_SPF-HF	Simpson HoldDowns	1.5	HF	N/A	0.167
8	DTT1Z-(8)0.148x1.5_SPF-HF	Simpson HoldDowns	1.5	HF	N/A	0.167
9	DTT2Z-SDS1.5_1.5_SPF-HF	Simpson HoldDowns	1.5	HF	N/A	0.105
10	DTT2Z-SDS1.5_3_SPF-HF	Simpson HoldDowns	3	HF	N/A	0.128
11	DTT2Z-SDS2.5_3_SPF-HF	Simpson HoldDowns	3	HF	N/A	0.128
12	HDU2-SDS2.5_DF-SP	Simpson HoldDowns	3	DF	N/A	0.088
13	HDU4-SDS2.5_DF-SP	Simpson HoldDowns	3	DF	N/A	0.114
14	HDU5-SDS2.5_DF-SP	Simpson HoldDowns	3	DF	N/A	0.115
15	HDU8-SDS2.5_3_DF-SP	Simpson HoldDowns	3	DF	N/A	0.11

OK
Cancel

The procedure that RISA uses for hold-down optimization is fundamentally based upon the assumption that there is a 'cost' to allowable tension in a product, and therefore the ideal hold-down would have as little tensile capacity as possible to meet code requirements. Once the program has determined the tensile force required to hold-down the wall it will choose the most economical hold-down product based on that which most closely matches (but does not exceed) the tension demand. The program looks to the **Allowable Tension** field of the hold-down schedule to choose the design.

Strap Optimization

Strap Schedule

Select Manufacturer: Simpson Chord Straps

Available HD Series:

- SIMPSON Chord Straps
- CMST_DF-SP
- CMST_SPF-HF
- CS_DF-SP
- CS_SPF-HF
- CSHP_DF-SP
- CSHP_SPF-HF
- MST_DF-SP
- MST_SPF-HF
- HST_DF-SP
- HST_SPF-HF

Current Selection Type: Optimize From Series

	Label	Manufacturer	Clear Span [in]	End Length [in]	Req'd Chord Species	Fastener Type	Number
1	MSTA49_18_DF/SP_Nail	Simpson Chord Straps	18	n/a	DF/SP	Nail	26
2	MSTA49_16_DF/SP_Nail	Simpson Chord Straps	16	n/a	DF/SP	Nail	26
3	MSTC28_18_DF/SP_Nail	Simpson Chord Straps	18	n/a	DF/SP	Nail	12
4	MSTC28_16_DF/SP_Nail	Simpson Chord Straps	16	n/a	DF/SP	Nail	16
5	MSTC40_24_DF/SP_Nail	Simpson Chord Straps	24	n/a	DF/SP	Nail	20
6	MSTC40_18_DF/SP_Nail	Simpson Chord Straps	18	n/a	DF/SP	Nail	28
7	MSTC40_16_DF/SP_Nail	Simpson Chord Straps	16	n/a	DF/SP	Nail	32
8	MSTC52_24_DF/SP_Nail	Simpson Chord Straps	24	n/a	DF/SP	Nail	36
9	MSTC52_18_DF/SP_Nail	Simpson Chord Straps	18	n/a	DF/SP	Nail	44
10	MSTC52_16_DF/SP_Nail	Simpson Chord Straps	16	n/a	DF/SP	Nail	48
11	MSTC66_30_DF/SP_Nail	Simpson Chord Straps	30	n/a	DF/SP	Nail	48
12	MSTC66_24_DF/SP_Nail	Simpson Chord Straps	24	n/a	DF/SP	Nail	54
13	MSTC66_18_DF/SP_Nail	Simpson Chord Straps	18	n/a	DF/SP	Nail	64
14	MSTC66_16_DF/SP_Nail	Simpson Chord Straps	16	n/a	DF/SP	Nail	68
15	MSTC78_30_DF/SP_Nail	Simpson Chord Straps	30	n/a	DF/SP	Nail	64

Buttons: OK, Cancel

The procedure that RISA uses for strap optimization is fundamentally based upon the assumption that there is a 'cost' to allowable tension in a product, and therefore the ideal strap would have as little tensile capacity as possible to meet code requirements. Once the program has determined the tensile force required to strap down the wall it will choose the most economical strap product based on that which most closely matches (but does not exceed) the tension demand. The program looks to the **Allowable Tension** field of the strap schedule to choose the design.

Optimization Options

For users who are new to wood wall design within RISA, the best procedure is to utilize the full databases, and to limit the potential designs by utilizing the [design rules spreadsheets](#). This results in a design based on the maximum number of options, which is often the most efficient design.

For experienced users who have more specific limitations in terms of the designs they would like to see, user-defined **Groups** (or families) are the solution. For example, an engineer who prefers to use only one sheathing thickness, or one nail type can create a custom Group that contains only the arrangements they want. For more information on creating these custom groups see [Appendix F- Wood Shear Wall Files](#).

For more information on Wood Walls see [Wood Wall Results](#).

Wood Wall - Design

The wood wall panel element allows you to easily model, analyze and design wood walls for in-plane loads. Here we will explain the wood specific inputs and design considerations. For general wall

panel information, see the [Wall Panels](#) topic. For wood wall results interpretation, see the [Wood Wall Results](#) topic.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Wood Walls**.

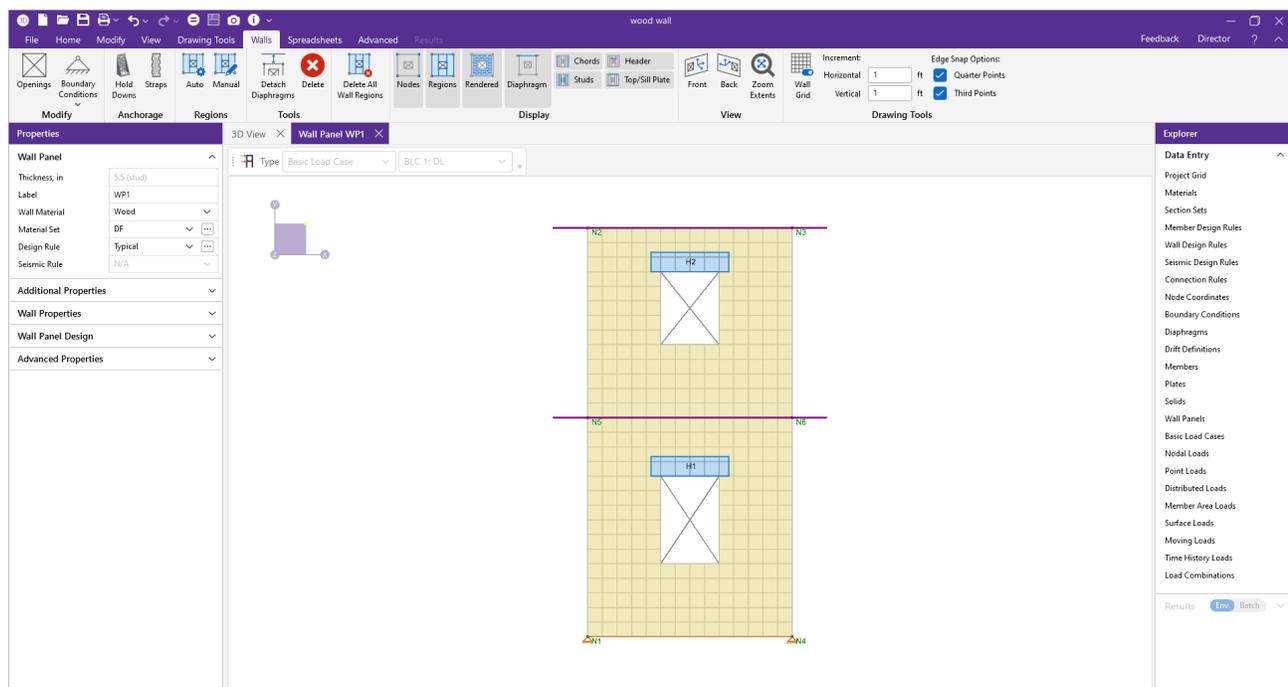
Full code checking and design can be performed on the panel sheathing, studs, chords, headers and hold-downs based on the following codes:

- The 2018 edition of the NDS. Lateral wall design can be performed using either 2015 or 2021 SDPWS when using 2018 NDS.
- The 2015 edition of the NDS
- The 2012 edition of the NDS
- The 2005/08 edition of the NDS (National Design Specification)
- The 2014 edition of the CSA O86
- The 2009 edition of the CSA O86

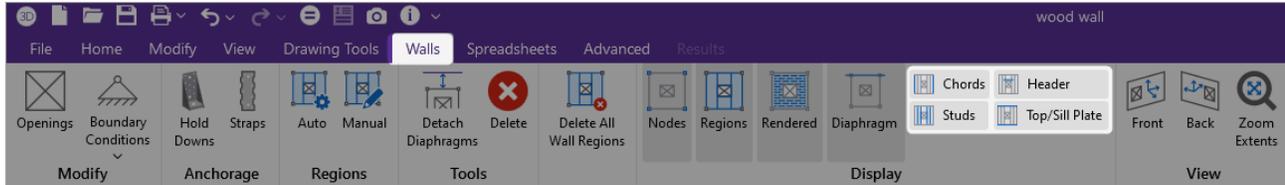
Wood Wall Input

The **Walls** ribbon provides specific information and options for the modeling/analysis of wood walls.

To open the **Walls** ribbon, double-click on a wall panel within the 3D view window. This action also opens the wall panel in the Wall Panel Editor.



Wood View Controls



The following table provides descriptions for the wood wall panel view controls.

Wood Wall Panel View Controls

Control	Description
 Chords	The Chords icon lets you toggle the display of the wall panel region chords on and off.
 Header	The Header icon lets you toggle the display of the headers on and off.
 Studs	The Studs icon lets you toggle the display of the wall studs on and off.
 Top/Sill Plate	The Top/Sill Plate icon lets you toggle the display of the top/sill plates on and off.

Design Rules

You must set up design rules for the stud/chord sizes, as well as make database selection for shear panels and hold-downs. This is done in the **Wall Design Rules** spreadsheet in the **Wood Wall (Studs)** and **Wood Wall (Fasteners)** tabs. See the [Wood Wall - Design Rules](#) topic for more information.

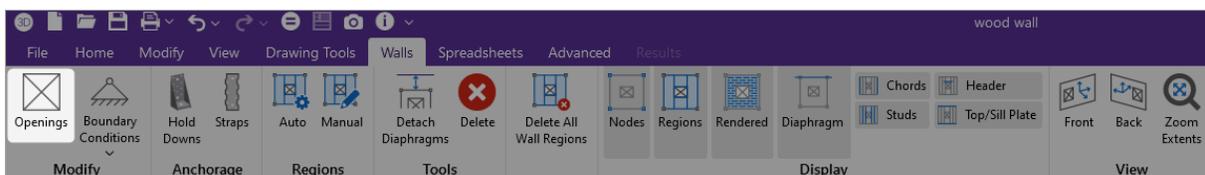
Create Openings in Wood Walls with Headers

Within the **Walls** ribbon, you have the option of adding rectangular openings to wood wall panels.

Draw an Opening

To draw an opening:

1. Click the **Openings** icon.



- Click on the two nodes or grid intersections which make up the two diagonal corners of your opening.

When an opening is drawn, a header beam is automatically created above the opening.

View/Edit Header Beam Properties

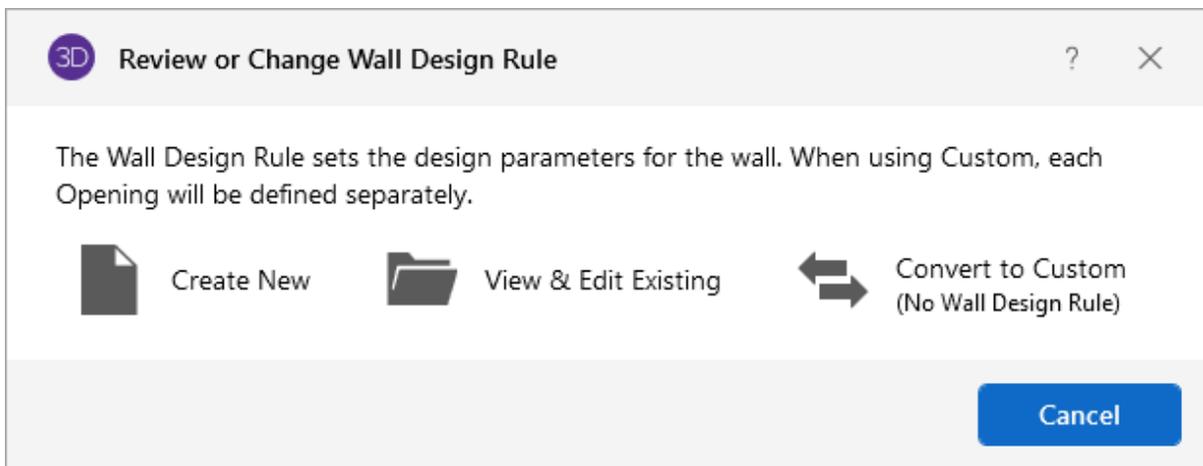
To view or edit the properties of a header beam:

- Open the [Wall Design Rules](#) spreadsheet.

In the case where you may have different opening headers in the same wall, you can define the different headers by following the next step.

- Double-click inside the boundary of the drawn opening, or click the **Design Rule**  button in the 'Properties' panel.

This opens the **Wood Wall Definition Editor** window for that particular opening, which then lets you modify the header properties.



The following options appear:

Review or Change Wall Design Rule

Option	Description
Create New	Create New lets you create a new Wall Design Rule to be applied to the wall.
View & Edit Existing	View Edit Existing lets you view as well as make edits to the Wall Design Rule that is currently applied to the wall.
Convert to Custom	The Convert to Custom (No Wall Design Rule) option is only available when each opening is defined separately. This option lets you define a header size and material that is different from the Wall Design Rule. This option only applies to Masonry Walls.

Option	Description
	 Note: You set a custom header only if you have multiple headers in the same wall that are different sizes.

- Click on the **View & Edit Existing** or **Convert to Custom** (if making custom modifications) to open the related window.

Refer to [Wall Design Rules](#) for more information on the available design rule options.

Hold-Downs and Straps

Hold-downs and straps are automatically added to your walls in their required locations. For a “Segmented” design, you must have hold-downs or straps at the bottom corners of each of your design segments. For “Perforated” and “Force Transfer Around Openings (FTAO)”, hold-downs straps are only allowed at the two far corners of the wall panel. The program will not permit the drawing of hold-downs or straps at locations where they are not allowed.

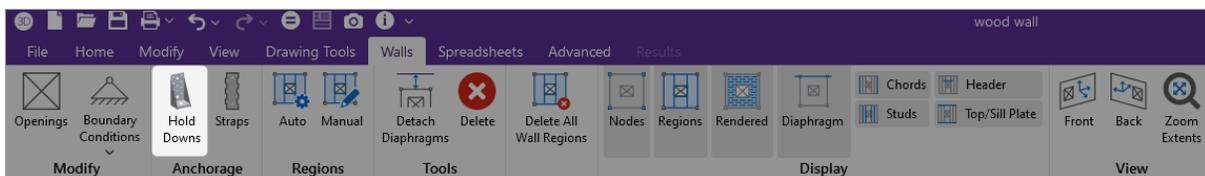
If there are custom locations that you want to add hold-downs or straps, you can do this from the **Walls** ribbon.

Hold downs

‘Hold downs’ can represent the anchorage of your wall to the foundation or the connection of shear wall chords between floors.

To add hold-downs to the base of your wall:

- Click the **Hold Downs** icon on the **Walls** ribbon.



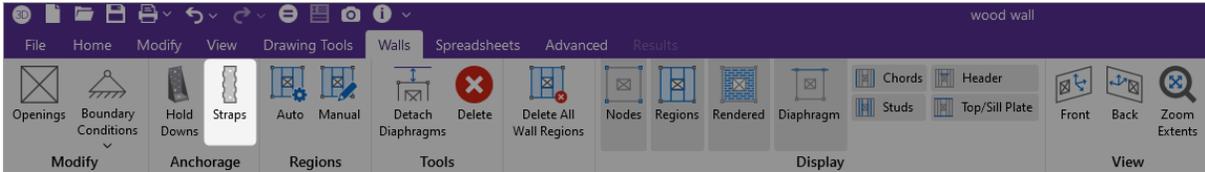
Hold downs must be added after regions are created and can only be added at the corners of regions. Hold down requirements depend on the type of wall design you are performing.

Straps

‘Straps’ represent the connection of the current wall panel to a wall panel below.

To add straps to the base of your wall:

- Click the **Straps** icon on the **Walls** ribbon.



Straps also can only be added after regions are created and can only be added at the corners of regions. Strap requirements also follow the same logic as hold downs as to where they must be defined as far as regions are concerned.



Note:

- All boundary conditions for wall panels should be defined in the wall panel editor. Adding external boundary conditions can create problems.
- The locations of hold-downs and straps define where the program will calculate tension forces in your walls.
- If you have applied your hold-downs or straps for the Segmented design with openings in your wall, then they will be required at the interior of the wall panel. However, running Perforated or FTAO does not require hold-downs or straps at the interior of the wall panel. Thus, if you toggle between Segmented to Perforated or FTAO, then the hold-downs or straps you drew will be removed at the interior of the walls. If you switch back to Segmented, the interior hold-downs or straps will come back again.
- By default, the program will automatically add hold-downs to the base of walls that have boundary conditions applied to them and will apply straps to walls that have walls below.
- All Straps and Hold-downs are placed at the base of walls only. So, if you would like to put a strap or hold-down between floors, you would apply it to the base of the upper wall.

The output for straps and hold-downs will show up on the detail report for the wall panel. More information on this can be found in the [Wood Wall Results](#) topic.

General Requirements for Shear Walls

The design of wood shear walls per the available design codes requires that many criteria are satisfied before a wall can be considered adequate. For RISA to work within this framework, we require that certain modeling practices be followed. Outlined below are many general wall modeling practices and limitations. Also included are specific requirements for each of the three design procedures for wood wall design with openings: Segmented, Force Transfer Around Openings, and Perforated.

The three different types of shear walls are defined in Section 4.3 of the NDS *Special Design Provisions for Wind and Seismic*.



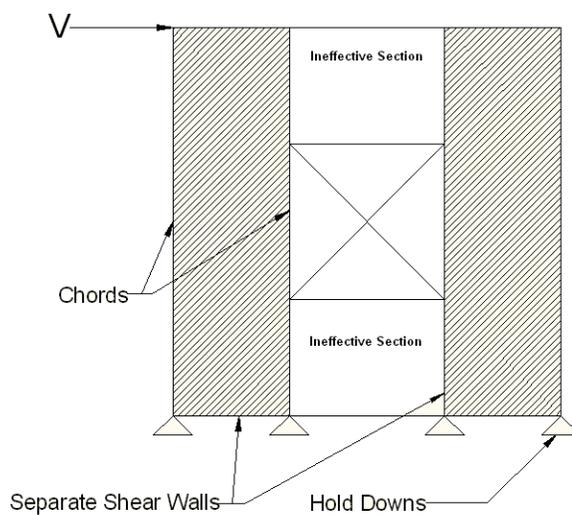
Note: The Canadian CSA O86 design code only supports the Segmented method, as described below.

Segmented Method

Where there is a wall panel with openings, the area above and below the openings is disregarded and the wall is designed as being made up of separate, smaller shear walls.

Like all wall panels, the segmented wood wall is broken into a series of meshed plate elements to represent the overall wall. The portions of the segmented shear wall that are considered "ineffective" in resisting shear are modeled with a plate elements that have a significantly reduced shear stiffness so that they will not receive any significant moment or shear from the FEM analysis.

See the diagram below for more information:



In addition, the out of plane stiffness and in plane stiffnesses of the segmented wood wall are modeled separately based on different assumed plate thicknesses. This is done to insure that the shear stiffness is based entirely on the properties of the sheathing and is not influenced by the out-of-plane stiffness of the wall studs.

**Note:**

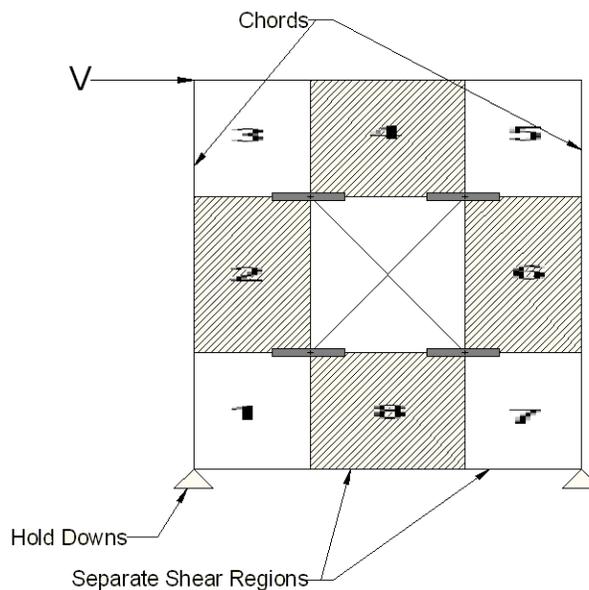
- If you have several stacked Segmented wall panels with misaligned openings, you will receive the Warning Message shown below upon solution. This message means that RISA-3D has assumed that the strap force from the above wall panel will be spread out across the region directly under it. Therefore, you need to be aware of this assumption and detail the wall panel accordingly.



- A shear panel design will be chosen for the worst-case region in a segmented wall. That panel will then be used for all regions in that wall. The worst-case region is the one that has the highest Shear UC value.

Force Transfer Around Openings Method

This method is based on a rational analysis of the wall generally referred to as the "Diekmann Method" This method is documented in detail in *Design of Wood Structures ASD/LRFD (6th edition)* by Breyer, Fridley, Cobeen, and Pollock. The assumption being that straps and blocking can added at the corners of the openings to transfer the sheathing forces across these nodes. This method essentially allows you to use the entire area of the wall (minus the opening) to resist the shear in the wall.



The basic assumptions made in the shear wall analysis are the following:

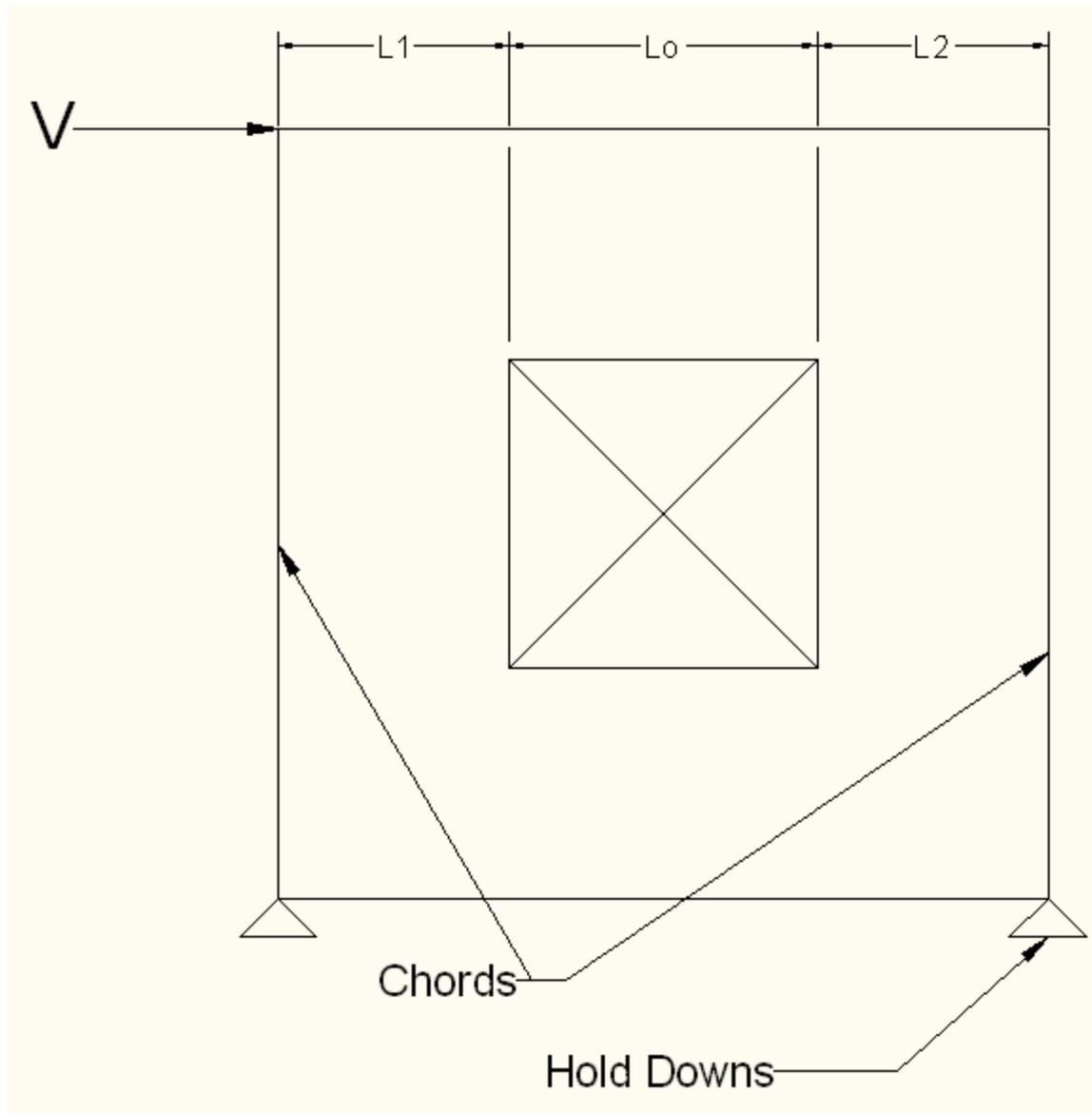
- The sheathing resists the shear forces. The average shear force in each block of the wall (numbered 1~8 as shown in the image above) is used as the controlling shear force in that location. The maximum shear in each of these locations will control the design of the wall. The program uses an area weighted average of the Fxy plate forces to determine the average shear for each block.
- The moment at the edge of each block that is above or below an opening is assumed to be transmitted across the opening interface by horizontal tension straps or compression blocks as shown in the image above. The required force is reported to the user, but the design and length of these elements is left to the engineer.
- The moment at the edge of each block that is to the right or left of an opening is assumed to be transmitted across the opening by tension straps or compression blocks. Since it is likely that the sheathing and king studs will be capable of transmitting these forces, these elements are not shown in the image above. However, these forces are reported so that the design of the studs and sheathing in these regions can be checked by the engineer to consider these effects.

**Note:**

- The program is limited in the automatic generation of regions for walls with multiple openings that are not aligned. Therefore, it is recommended that complex walls with multiple openings be simplified based on engineering judgment to facilitate easier detailing of the force transfer around these openings.
- This design method is not available for Canadian CSA O86 design. If you select this design method, the program will automatically change it back to Segmented.

Perforated Method

This method for design of wood shear walls with openings may end up being the most cost effective. It only requires hold downs at the corners of the wall, yet it does not require straps or blocking around the openings. A perforated shear wall design approach is, however, subject to a number of code constraints about when it can be used.



The basic design procedure for perforated walls is to essentially ignore the portions of the wall that do not have full height sheathing and treat the wall instead as a significantly shorter wall. This amplifies the chord and hold down design forces significantly while at the same time increasing the design unit shear as shown in the equations below:

$$T := \frac{V \cdot h}{C_o \cdot \Sigma L_i} \quad u_{\max} := \frac{V}{C_o \cdot \Sigma L_i}$$

Where:

$$\Sigma L_i = L1 + L2$$

**Note:**

- The perforated method of design also has many caveats that are given in Section 4.3.5.3 of the 2015 *Special Design Provisions for Wind & Seismic* / 4.3.5.6. of the 2021 *Special Design Provisions for Wind & Seismic*. The program will not allow the design of wall panels that do not follow these provisions.
- For multi-story perforated shear walls, the amplified hold down forces required per the code become difficult to interpret for the lower walls. Therefore, when RISA calculates the hold down forces for the lower wall it assumes that the reduction coefficients for the upper wall are identical to the values for the lower wall. This will result in conservative hold down forces when the lower wall has more openings than the upper wall. But, it may be un-conservative for situations where this is not the case. This assumption does NOT affect the shear design of either wall, nor does it affect the strap force calculations in the upper wall.
- This design method is not available for Canadian CSA O86 design. If you select this design method, the program will automatically change it back to Segmented.

Shear Capacity Adjustment Factor, Co:

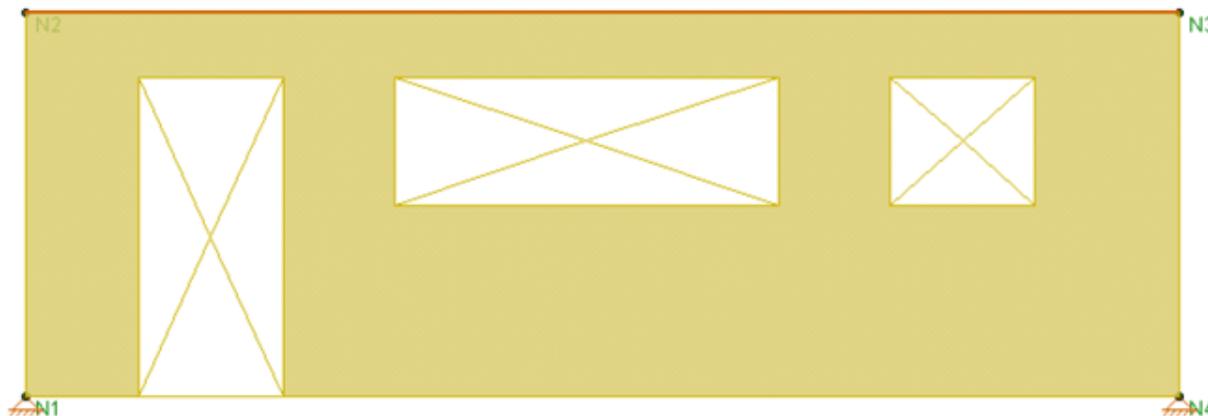
The NDS *Special Design Provisions for Wind & Seismic* lists Effective Shear Capacity Ratio (Co) values that are used in calculating the nominal shear capacity of perforated shear walls. Because the tabular values are limited to wall heights of 8' & 10', RISA instead uses equation (4.3-5) from the 2015 *NDS SDPWS* (as shown below) or equation (4.3-6) from the 2021 *NDS SDPWS* to calculate the Co factor for any height wall.

$$C_o = \left(\frac{r}{3 - (2 * r)} \right) * \frac{L_{tot}}{\Sigma L_i}$$

Where:

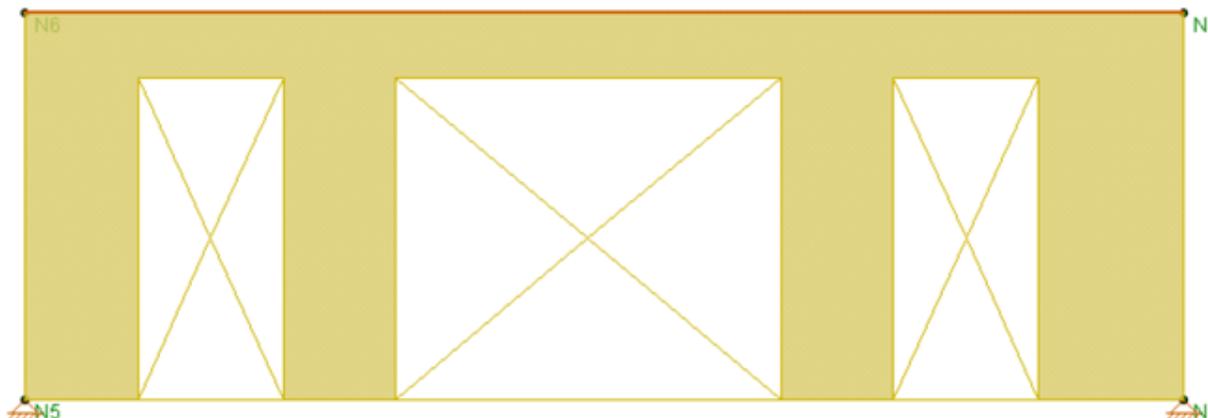
$$r = \frac{1}{1 + \left(\frac{A_o}{h * \Sigma L_i} \right)}$$

When using these equations, RISA takes Ao as the true area of the openings. However, Table 4.3.3.5 of the 2015 *NDS SDPWS* (Table 4.3.3.4 from older versions of the *NDS SDPWS*) references Co values based on all opening heights equal to the maximum opening height. Therefore if you want the program to calculate Co equal to that in Table 4.3.3.5 of the *NDS*, you must draw all openings as equal to the maximum height. Please see the image below for reference.



$C_o = 0.84$

VS.



$C_o = 0.53$

(Full height openings assumption- matches NDS Table 4.3.3.4)



Note: When the assumption is made that all opening heights are equal to the maximum opening height then the equation produces values of C_o that are within 1% for all the values shown in the NDS table.

C_o Limitation for Stacked Wall Panels

For wall panels that are stacked on one another, each wall panel has its own C_o value. These C_o values will be different if the openings in each wall are different. This is the source of a limitation in the program.

The program determines the design forces in each wall panel separately using the finite element solution. We then take those design forces and factor them for the C_o values. The problem is that the design forces for the bottom wall are affected by the C_o value for the upper wall. Thus, the forces that come down on a lower wall from an upper wall have been factored for the C_o from the upper wall. RISA-3D does not consider this. The program takes the forces from the finite element solution and uses the C_o value only for the wall in question.

Thus, the highest level in a stacked wall configuration will always use C_o in a correct manner. The lower wall(s), however, will be conservative if they have more openings than the upper wall(s) and can be unconservative if the upper wall(s) have more openings than the lower wall(s).

This limitation must be considered when designing Perforated walls that are stacked.

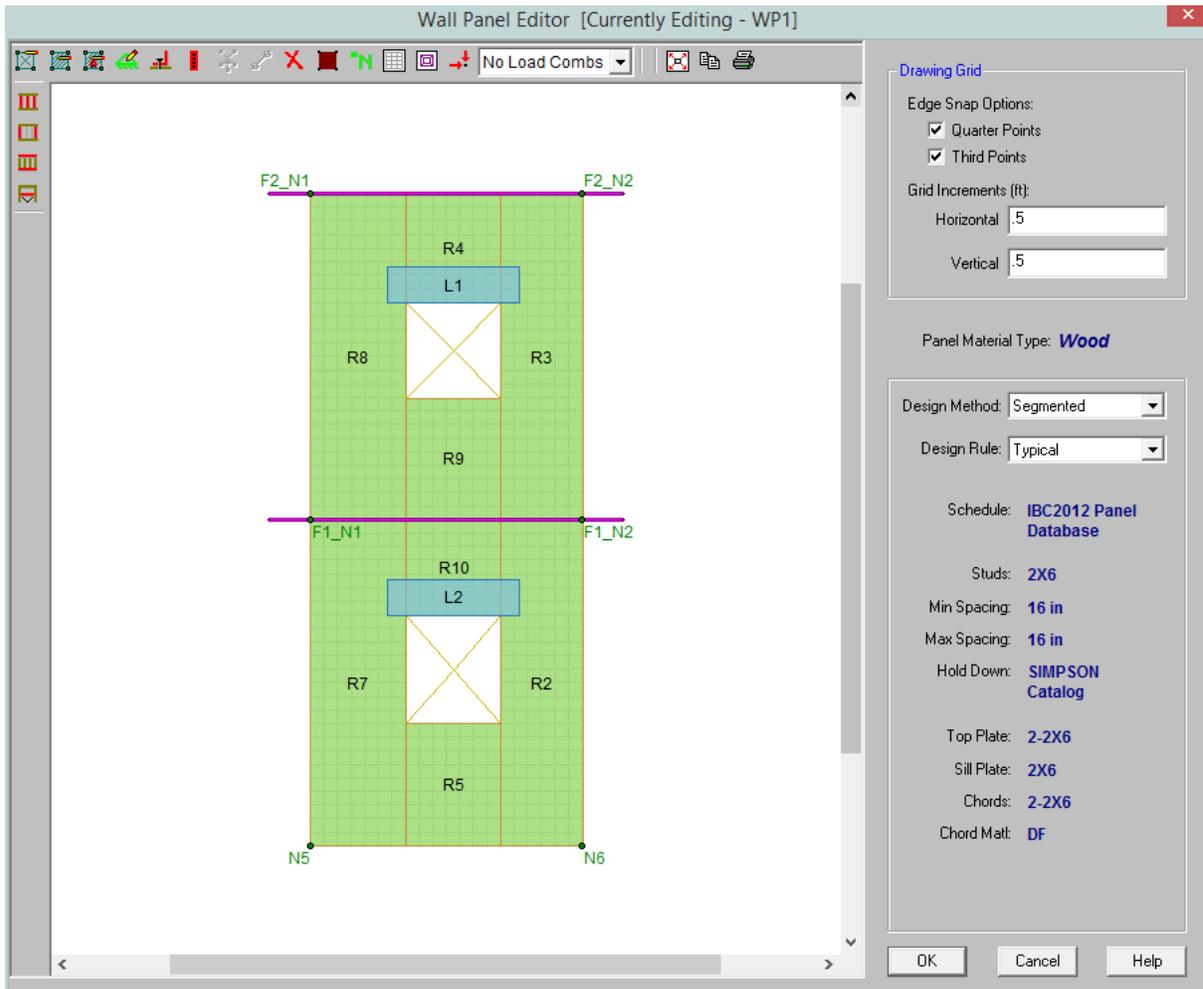
General Program Functionality and Limitations

RISAFloor and RISA-3D Interaction

When using RISAFloor and RISA-3D in combination, the interface transitions nicely between the two programs. Here is a quick walk through of this interaction.

Input Interface

1. Model the entire building (gravity and lateral members) within RISAFloor. Be sure to model all openings and regions for all of the wall panels in the model.



Note: You can not modify your openings or regions in RISA-3D. All region and opening modifications must be taken back to RISAFloor to be done.

2. Add loading and solve the model.
3. Click on the **Director** link to take the model into RISA-3D.
4. Once in RISA-3D, add your hold downs and straps to your wall panels.

**Note:**

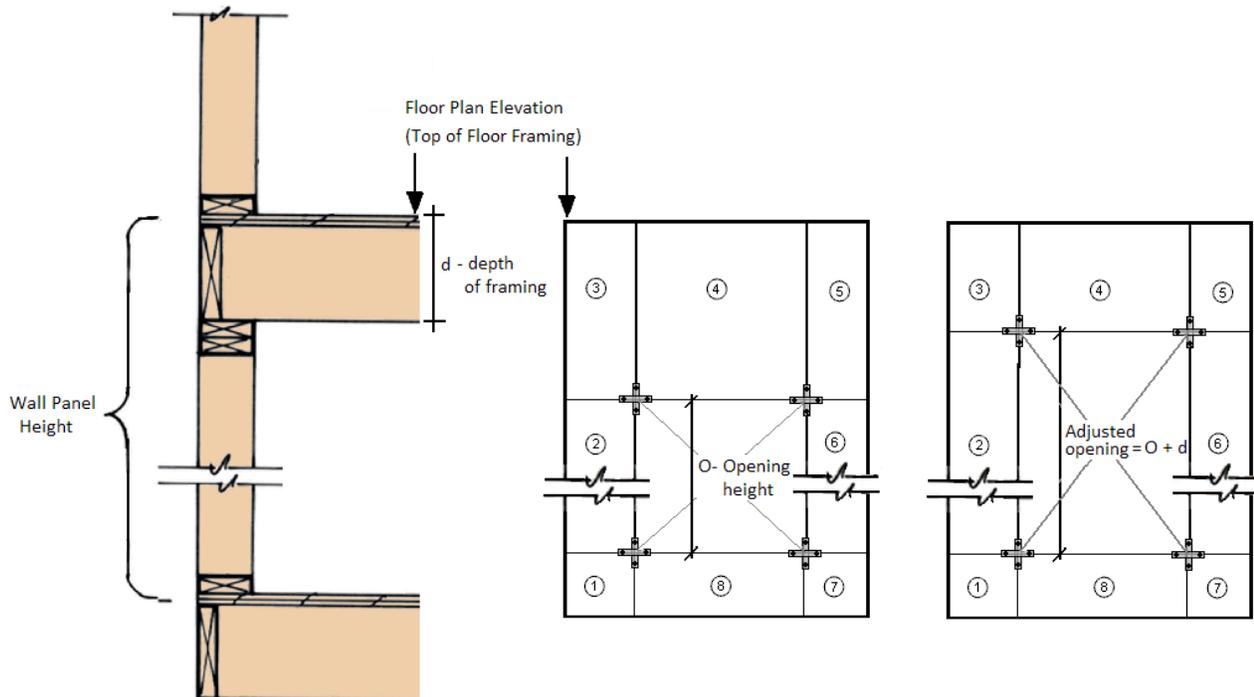
- Hold downs and straps can not be added to wall panels in RISAFloor.
- Hold downs are only allowed to be added to the lower corners of the wall panel for the Perforated or Force Transfer Around Openings design methods.
- Hold downs are required at the corners of all full height regions in the wall panel for the Segmented design method.
- The Design Rule, Design Method and SSAF can be changed in either program at any time.

Modeling Tips- Platform Framing

The wood wall height in RISA is measured from the bottom of the sill plate to the top of the floor framing as shown below. However, platform framing causes the wall height to be significantly shorter than a RISA model would represent.

The design method FTAO is the only method that will cause significant problems because the region above openings is assumed to be much larger than it is built. In order to adjust for this framing depth, you can adjust your opening height to include the depth of the floor framing. This will reduce the portion of the wall above the opening thus reducing the amount of area to transfer shear forces.

In Segmented and Perforated design methods, the portion above and below the opening are not used to transfer shear forces.



Limitations for Hold-downs / Straps (Including deformation)

- The program does not currently have a database for continuous tie rod hold-downs. However, the design results that are presented for each floor are intended to provide the type of information necessary for the design of these types of hold downs.
- The d_a values (deflection at peak load) from our hold-down database are based entirely on the manufacturers listed values. These do not include any allowance for shrinkage (which is often as much as 1/4" or more per floor), or crushing of the sill plate.
- There is not currently a d_a value used in the calculation for strap deformation for upper floor levels.

Automatic Boundary Conditions

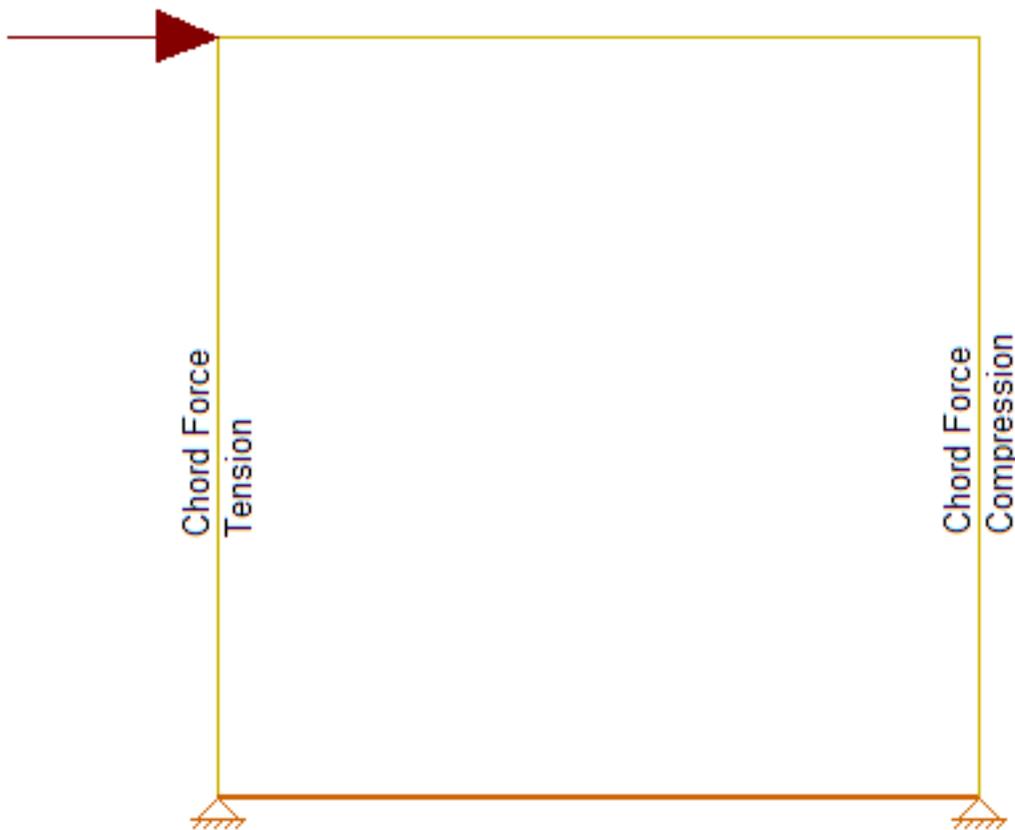
- In RISA-3D, if no boundary conditions or hold downs are defined for wood wall panels at the lowest level of the structure, the program will automatically create hold downs at the corners of the wall panel. If you do not want the hold downs to be automatically created, define a "free" boundary condition at the base of the wall panel in the wall panel editor.

Deflection

- There is currently no code check for drift or deflection for shear wall panels.

Chord Design

The chord design is based on forces that are calculated differently for Compression versus Tension. The tension chord force is calculated including the dead load stabilizing moment as per Section 4.3.6 of the NDS 2015 *Special Design Provisions of Wind & Seismic*. The compression chord force includes the only the tributary area of one stud spacing in the compression force. For Segmented design, the chord forces are found based on each region, and in FTAO and Perforated design methods the chord forces are determined for the entire wall.



Chord forces are calculated per the following equations:

- When the "[Eccentricity](#)" check-box is selected in the Wood Wall Design Rule (Fasteners tab):

Additional Wood Wall Panel Parameters												
Unity Check	Concrete Wall (Rebar)	Concrete Wall (Misc)	Masonry Wall	Masonry In	Masonry Out	Masonry Lintel	Wood Wall (Studs)	Wood Wall (Fasteners)	CFS Wall (Studs)	CFS Wall (Fasteners)		
Label	Schedule	Min Panel Thick [in]	Max Panel Thick [in]	Double Sided Panel?	Max. Nail Spacing	Min. Nail Spacing	HD Chords	HD Chord Matl	Hold Down	Chord Strap	Eccentricity	
1	Typical	AWC 2021 OSB	0.375	0.75	Optimum	6-in.	2-in.	2-2X8	Same as Wall	SIMPSON HoldDowns	SIMPSON Chord Straps	<input checked="" type="checkbox"/>

$$\text{Compression Force} = \frac{M}{L - b_c} + \frac{P}{n}$$

$$\text{Tension Force} = \frac{-M}{L - b_c} + \frac{P}{2}$$

- When the "Eccentricity" check-box is NOT selected in the Wood Wall Design Rule (Fasteners tab):

Additional Wood Wall Panel Parameters												
Unity Check	Concrete Wall (Rebar)	Concrete Wall (Misc)	Masonry Wall	Masonry In	Masonry Out	Masonry Lintel	Wood Wall (Studs)	Wood Wall (Fasteners)	CFS Wall (Studs)	CFS Wall (Fasteners)		
Label	Schedule	Min Panel Thick [in]	Max Panel Thick [in]	Double Sided Panel?	Max. Nail Spacing	Min. Nail Spacing	HD Chords	HD Chord Matl	Hold Down	Chord Strap	Eccentricity	
1	Typ...	AWC 2021 OSB	0.375	0.75	Optimum	6-in.	2-in.	2-2X8	Same as Wall	SIMPSON HoldDowns	SIMPSON Chord Straps	<input type="checkbox"/>

$$\text{Compression Force} = \frac{M}{L} + \frac{P}{n}$$

$$\text{Tension Force} = \frac{-M}{L} + \frac{P}{2}$$

Where:

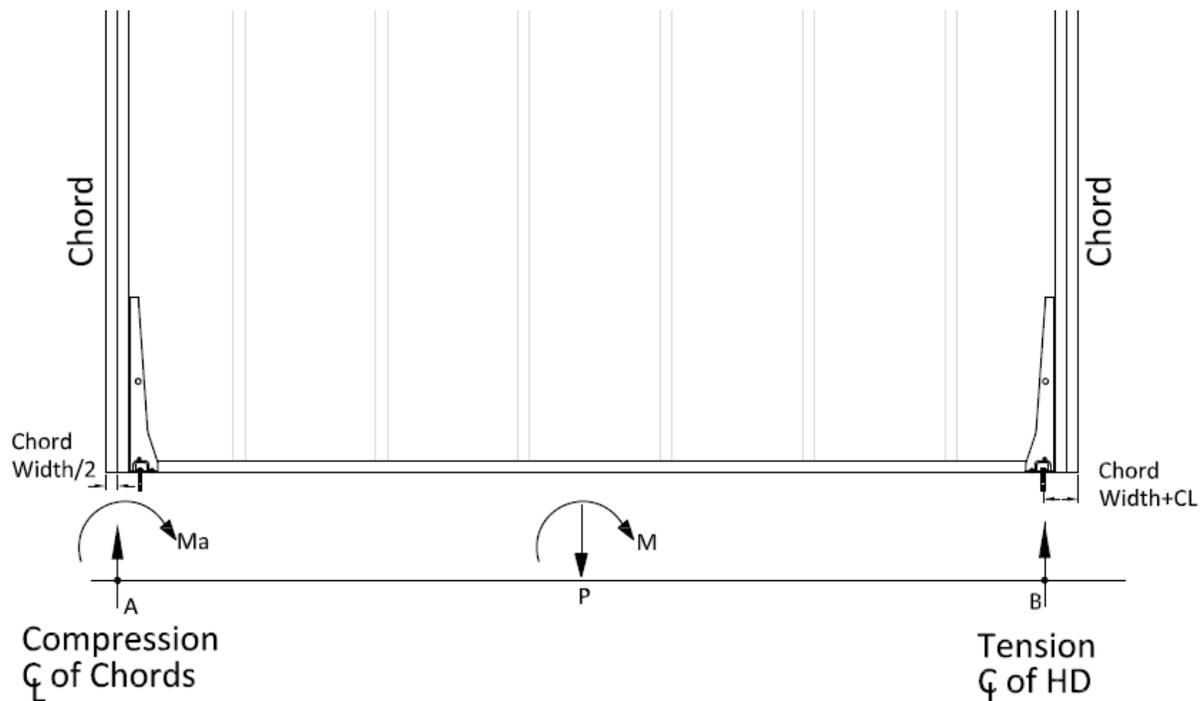
- M = Moment at the base of the wall
- L = Length of the wall
- b_c = width of chord member
- P = Axial force at the base of the wall
- n = Number of Studs = Length/Stud Spacing

**Note:**

- For Perforated Design, the term $M/(\text{Length}-\text{ChordWidth})$ found in the above equation is replaced with $Vh/(C_o\Sigma L_i-\text{ChordWidth})$ as indicated in the Equation 4.3-8 of the NDS 2015 *Special Design Provisions for Wind & Seismic*.
- RISA-3D models the shear walls using only the sheathing. The vertical resistance occurs only at the tension and compression chords. Thus, if two wall panels are stacked on top of each other, the load transfer will only happen at the chord locations. Therefore, the lateral analysis should agree very well with hand calculations. However, it also means that gravity load design may be more appropriate in RISAFloor.

Hold-Down Force

The Hold-Down force is calculated by finding the Tension chord force. In order to accommodate an unsymmetrical vertical load on the wall, the program adopted a more accurate approach by calculating the moment at the compression side first and then finding the resulting reaction on the tension side.



Hold-Down forces are calculated per the following equations:

- When the ["Eccentricity" check-box](#) is selected in the Wood Wall Design Rule (Fasteners tab):

Additional Wood Wall Panel Parameters												
Unity Check	Concrete Wall (Rebar)	Concrete Wall (Misc)	Masonry Wall	Masonry In	Masonry Out	Masonry Lintel	Wood Wall (Studs)	Wood Wall (Fasteners)	CFS Wall (Studs)	CFS Wall (Fasteners)		
Label	Schedule	Min Panel Thick [in]	Max Panel Thick [in]	Double Sided Panel?	Max. Nail Spacing	Min. Nail Spacing	HD Chords	HD Chord Matl	Hold Down	Chord Strap	Eccentricity	
1	Typical	AWC 2021 OSB	0.375	0.75	Optimum	6-in.	2-in.	2-2X8	Same as Wall	SIMPSON HoldDowns	SIMPSON Chord Straps	<input checked="" type="checkbox"/>

$$\text{Moment at Point } a = M_a = -M + \left(P * \left(\frac{L}{2} - \frac{b_c}{2} \right) \right)$$

$$\text{Reaction at Point } b = R_b = \frac{M_a}{L - 1.5 * b_c - CL}$$

- When the "Eccentricity" check-box is NOT selected in the Wood Wall Design Rule (Fasteners tab):

Additional Wood Wall Panel Parameters												
Unity Check	Concrete Wall (Rebar)	Concrete Wall (Misc)	Masonry Wall	Masonry In	Masonry Out	Masonry Lintel	Wood Wall (Studs)	Wood Wall (Fasteners)	CFS Wall (Studs)	CFS Wall (Fasteners)		
Label	Schedule	Min Panel Thick [in]	Max Panel Thick [in]	Double Sided Panel?	Max. Nail Spacing	Min. Nail Spacing	HD Chords	HD Chord Matl	Hold Down	Chord Strap	Eccentricity	
1	Typ...	AWC 2021 OSB	0.375	0.75	Optimum	6-in.	2-in.	2-2X8	Same as Wall	SIMPSON HoldDowns	SIMPSON Chord Straps	<input type="checkbox"/>

$$\text{Moment at Point } a = M_a = -M + \left(\frac{P * L}{2} \right)$$

$$\text{Reaction at Point } b = R_b = \frac{M_a}{L - CL}$$

Where:

- M_a = moment at point A(CL of compression chord)
- M = moment at base of the wall
- P = Axial load at the base of the wall
- L = length of the wall
- b_c = width of chord member
- R_b = reaction at point B = hold down force
- CL = Hold down eccentricity distance (per selected hold down)

Stud Design

Studs are only designed for load combinations which do not contain a wind or seismic load. The maximum axial load, determined as an envelope force from all of the "gravity" load combinations which have been solved, is determined for each region.

The compression capacity for the specified stud size is calculated with the assumption that the stud is fully braced against buckling about its minor axis (within the plane of the wall). This is because the blocking and sheathing are assumed to provide this bracing. The unbraced length for major axis buckling is taken as the wall height, minus the thickness of the top and sill plates.

The program divides the region axial force by the number of studs that would be present in that region for a given stud spacing. An optimal stud spacing is then selected based on the stud capacity, and the parameters defined in the [Design Rules](#).

Force Distribution

- The lateral force distribution between piers is based on the relative stiffness of the sheathing, not on the length of the shear wall. For example, if you have an 8 foot wall and a 4 foot wall, the 8 foot wall will take more than $8/(8+4)*100\%$ of the force. The moment of inertia in the 8 foot wall will allow for a larger proportion of load to go into that pier.

Shear Capacity Adjustment Factors

40% Increase Factor Wind Load Cases

The allowable shear stress values tabulated in [Appendix F](#) are intended to be the allowable shear for seismic loads. The [solution](#) tab of the Model Settings has a check-box which will automate the 40% shear capacity increase for any load combinations that include Wind Loads. This is called "**Wind ASIF**" in the detailed report output. When the code is set to 2021 SDPWS LRFD, the code allows for a 60% increase in shear capacity for wind loads.

Allowable Stress Reductions for Slender Wall Segments

The program will modify the allowable stress of the wall based on the $2b/h$ adjustment factor per NDS SDPWS section 4.3.3.4 exception 1. This adjustment factor is only applied to the wall for load combinations which include seismic loads. This adjustment factor only affects walls with an aspect ratio between 2.0 and 3.5. This is called "**Cap.Adj. (2w/h)**" in the detailed report output.

For Segmented and FTAO wall panels designed per the NDS 2015, the program also checks the Aspect Ratio factor per section 4.3.4.2. This factor will reduce the sheathing capacity for lateral (Wind or Seismic) load combinations on walls who have a design region whose aspect ratio is greater than 2.0. This is called "**Aspect Ratio**" in the detailed report output.

The final governing factor (minimum of **Cap. Adj. (2w/h)** and **Aspect Ratio**) is reported as the **Gov. H/W Factor**. This is then multiplied by the sheathing capacity to give the final **Adj. Shear Cap.**

Detail Report: WP1 (Region R3)

CRITERIA		GEOMETRY		MATERIALS	
Code:	AWC NDS-18 / SDPWS-21:ASD	Total Height (ft):	15	Wall Studs:	DF
Wall Material:	DF	Total Length (ft):	5	Stud Size:	2X6
Panel Schedule:	AWC 2021 OSB	Region H/W:	3.00	Chord Material:	DF
Optimize HD:	Yes	Cap. Adj. (2w/h):	1.00	Chord Size:	2-2X8
HD Manufacturer:	SIMPSON	Aspect Ratio:	0.88	Top Pl. & Sill:	DF
		Gov. H/W Factor:	1.00	Top Pl. Size:	2-2X8
				Sill Pl. Size:	2X6
				Specific Gravity:	0.5

Specific Gravity Adjustment Factor

The NDS defines an adjustment factor associated with using stud material that is less dense than Douglas-Fir-Larch or Southern Pine. The program automatically accounts for this factor in the design of Wood Shear Walls.

Unblocked Shear Wall Adjustment Factor

Section 4.3.3.2 of the 2015 NDS *Special Design Provisions for Wind and Seismic* is ignored in RISA-3D. The program will always assume that the sheathing panel is blocked for both NDS and CSA design.

Hold Down Adjustment Factor (Applicable to only 2021 SDPWS)

Per footnote 10 of Table 4.3A of the 2021 SDPWS, the nominal unit shear capacity for shear wall shall be multiplied by 0.92 if shear walls are using 10d nails and where hold-downs are attached to the inside face of the end post. The program assumes that all Wood Walls have hold-downs anchored to the inside face of the end post.

Uplift Limitation

Section 4.4 of the 2015 NDS *Special Design Provisions for Wind and Seismic* is ignored in RISA-3D. The program currently does not design wall panels for uplift forces.

Stiffness Adjustment Factors

Shear Stiffness Adjustment Factor (to change FEM results)

This stiffness adjustment factor is set on the Wall Panels spreadsheet and is intended as a way to force the FEM stiffness of the wall to more closely resemble the stiffness from the APA / NDS three term deflection equation.

This adjustment affects the stiffness of the entire wall. Therefore, for segmented walls the engineer may be forced to model the piers separately if they need to adjust the pier stiffnesses independently. In a future release, this may become an automated factor.

Green Lumber

The [Wood Wall \(Studs\)](#) tab of the Design Rules spreadsheet contains a Green Lumber check-box to account for the 50% reduction in the G_a value defined in the NDS footnote.

Stiffness Assumptions

Vertical Direction

RISA uses an orthotropic plate element to de-couple the vertical and shear stiffness of the wood walls. The vertical stiffness will be based on the E value of the studs and chords as specified in the Materials spreadsheet and the thickness of the wall. The thickness is taken as

$$Thickness = \frac{Total\ Wall\ Weight}{Material\ Density * Wall\ Length * Wall\ Height}$$

Where:

$$Total\ Wall\ Weight = W_{studs} + W_{chords} + W_{top\ plate} + W_{sill\ plate} + W_{sheathing\ panel}$$

In-Plane Shear

The in-plane shear stiffness will be based on the G_a value designated within the specified nailing schedule divided by the sheathing thickness. Both the G_a value and the sheathing thickness are defined in the specified sheathing schedule.

Out-of-Plane Shear

For the out of plane shear, RISA uses the same Young's Modulus (E) as used in the vertical direction and a thickness that is calculated from the out of plane moment of inertia. This becomes:

$$Thickness = \left(\frac{I_{chord} + I_{stud} + I_{panel}}{Wall\ Length} \right)^{1/3}$$

Wood Wall Self Weight

The program will calculate wood wall self weight as a sum of all the weights of the components. The material density is used to calculate the self weight of the studs, chords, top plates, sill plate, and sheathing. These are all then summed together to give the self weight of the entire wall.

**Note:**

- For this calculation, stud height equals wall height minus the thickness of the sill plate and the top plate.
- The number of studs is calculated using the stud spacing specified in Design Rules.

Wood Wall Optimization

The program will optimize wood walls based on the required demand forces. The program can optimize:

- Sheathing for in-plane shear design.
- Stud spacing for axial design.
- Hold-downs for in plane design.

Of these optimizations the two that substantially modifies the stiffness of the wall are the sheathing and the stud spacing. To properly adjust the stiffness requires an iterative solution that updates the stiffness of the model. This includes updating the strength properties of the wall as well as the stiffness. This optimization/iteration can be done automatically by checking **Optimize Masonry and Wood Walls** in the [Model Settings - Solution](#) tab or can be done manually by unchecking this option.

The screenshot shows the 'Model Settings' dialog box with the 'Solution' tab selected. The 'Members' section includes 'No. of Reported Sections' (5), 'No. of Internal Sections' (100), and 'Member Area Load Mesh Size' (144 in²). The 'Wall Panels' section includes 'Approx. Mesh Size' (24 in) and several checked options: 'Transfer Forces Between Intersecting Wood Walls', 'Increase Wood Wall Nailing Capacity for Wind Loads', 'Include P-Delta for Walls', and 'Optimize Masonry, Wood and CFS Walls'. The 'Max. No. of Iterations' is set to 3. The 'Processor Core Utilization' is set to 'Balanced'. The 'Advanced' button is visible at the bottom, along with 'Set as Default', 'Apply', and 'Cancel' buttons.

To update the stiffness portion of the wall, the program must re-solve your model with these updated stiffnesses as this will change the distribution of forces through the model. By checking **Optimize Masonry and Wood Walls** you are telling the program to re-solve automatically. Thus, the program will start with it's initial stiffness parameters and solve the model. It will then optimize the wall to meet strength criteria. Another solution will then be run with the new stiffnesses and the program will again optimize the wall to meet strength criteria. This procedure will continue to occur until you reach the **Maximum Number of Iterations** set or until all wall panel results match those of the previous solution.

By unchecking **Optimize Masonry and Wood Walls** the program will only run the solution once and the results will be based on the original configuration. You can then manually optimize your walls using the [Suggested Design](#) spreadsheet.

After the solution is run (with or without optimization) the design results are based on the stiffness used in the last iteration (by unchecking the **Optimize Masonry and Wood Walls** option a single iteration is run). The program will then compare the design of the last iteration with the stiffness used in that last iteration. If the two are the same the results shown are the final results. If the two are not the same the program will then provide these two different results in the **Suggested Design** spreadsheet.

The program will always present results in the output that coincide with the stiffness used in the final solution.



Note: The updating of the stiffness for the model is only required for the sheathing and stud spacing optimization. Thus, hold-down call-outs are optimized automatically.

Suggested Design

Wood Wall Suggested Design

Members | Masonry Walls | **Wood Walls** | CFS Walls

Re-solve with Suggested Shape(s)/Wall(s)

	Wall Panel	Current Panel	Suggested Panel	Current Stud Space[in]	Suggested Stud Space[in]	Use Panel?	Use Stud Space?
1	WP1	S1_3/8_8d@6	RS_3/8_6d@6			<input checked="" type="checkbox"/>	<input type="checkbox"/>

wood wall

File Home Modify View Drawing Tools Spreadsheets Advanced Results

Envelope LC Dynamic Sort by Combination Sort by Item Filter Results Node Wall TH Trace Export TH Trace Contour Diagram Clear Results Warning Log Suggested Design Detailed Report

Results Force Summation Time History Contour Formatting

In the **Suggested Design** spreadsheet you will get a list of wall panels in your model that are not yet fully optimized, showing the panel and stud spacing of the last iteration and the program optimized values. From here you have the ability to **Use Panel?** or **Use Stud Space?** which means that you want to re-run the solution with the suggested design. You can choose this for each suggestion for each wall panel individually. Once you have these checkboxes checked appropriately click the **Suggested Design** icon in the Results tab to re-solve the model with the suggested design. After this the stiffness matrix is re-formulated and may cause some redistribution of loads through the model. Because of this the suggested design may also update and you may need to **Use Panel?** and/or **Use Stud Space?** multiple times to converge on a solution.



Note:

- If the wall does not show up in the Wood Wall Suggested Design spreadsheet then the current wall panel settings used are the optimal ones.
- If either the stud spacing or the panel fields are blank that means that the current selections are the optimal ones.
- For more information on wood wall optimization see the [Wood Wall - Design](#) topic.
- For more information on member optimization see the [Design Optimization](#) topic.
- Concrete walls do not show up here because the reinforcement optimization does not affect the stiffness of the wall.

Panel Optimization

Wood Wall Panel Schedule

Select Code: AWC 2021 OSB

Available Panel Groups:

- AWC 2021 OSB
- AWC 2021 OSB 0.3125
- AWC 2021 OSB 0.375
- AWC 2021 OSB 0.4375
- AWC 2021 OSB 0.469 (8d)
- AWC 2021 OSB 0.469 (10d)

Current Selection Type: Use All Panels

	Label	Grade	Min Thick [in]	Min Penetration	Over Gypsum	One/Two Sided	Nail Size	Staple Size	Nail
1	S1_5/16_6d@6	St-I	0.312	1.25	No	1	6.000000	n/a	6
2	S1_(2)5/16_6d@6	St-I	0.312	1.25	No	2	6.000000	n/a	6
3	S1_3/8_8d@6	St-I	0.375	1.375	No	1	8.000000	n/a	6
4	S1_(2)3/8_8d@6	St-I	0.375	1.375	No	2	8.000000	n/a	6
5	S1_7/16_8d@6	St-I	0.438	1.375	No	1	8.000000	n/a	6
6	S1_(2)7/16_8d@6	St-I	0.438	1.375	No	2	8.000000	n/a	6
7	S1_15/32_8d@6	St-I	0.469	1.375	No	1	8.000000	n/a	6
8	S1_(2)15/32_8d@6	St-I	0.469	1.375	No	2	8.000000	n/a	6
9	S1_15/32_10d@6	St-I	0.469	1.5	No	1	10.000000	n/a	6
10	S1_(2)15/32_10d@6	St-I	0.469	1.5	No	2	10.000000	n/a	6
11	S1_5/16_6d@4	St-I	0.312	1.25	No	1	6.000000	n/a	4
12	S1_(2)5/16_6d@4	St-I	0.312	1.25	No	2	6.000000	n/a	4
13	S1_3/8_8d@4	St-I	0.375	1.375	No	1	8.000000	n/a	4
14	S1_(2)3/8_8d@4	St-I	0.375	1.375	No	2	8.000000	n/a	4
15	S1_7/16_8d@4	St-I	0.438	1.375	No	1	8.000000	n/a	4

OK Cancel

The procedure that RISA uses for design optimization is fundamentally based upon the assumption that there is a 'cost' to shear capacity, and therefore the ideal panel design would have as little shear capacity as possible to meet code requirements. Once the program has determined the shear demand on the wall it will choose the most economical panel configuration based on that which has a **Shear Capacity** closest to, but not exceeding the shear demand.

The default shear panels in the program come straight from published tables in the IBC, UBC, and CSA O86 design manuals. "St-I" refers to Structural I panels and "RS" refers to Rated Sheathing panels.



Note:

- The shear force listed in the XML spreadsheet is tabulated for the seismic values. These values will automatically be multiplied by 1.4 for wind forces if the [Wind ASIF](#) function is enabled. These values will be multiplied by 1.6 for wind forces when the code is set to 2021 SDPWS LRFD.
- Panels with a label containing the characters "_W" together will be ignored during design optimization.
- "Rated Sheathing" is a shortened form of "Span Rated Sheathing" or APA Rated Sheathing. This terminology comes directly from the Department of Commerce standards DOC PS1 and PS2.

Hold Down Optimization

3D Hold Down Schedule
?
×

Select Manufacturer: Simpson HoldDowns

Available HD Series

- SIMPSON HoldDowns
- DTT_DF-SP
- DTT_SPF-HF
- HDU_DF-SP
- HDU_SPF-HF
- HDB_DF-SP
- HDB_SPF-HF
- LTT_DF-SP
- LTT_SPF-HF
- HTT_DF-SP
- HTT_SPF-HF

Current Selection Type: Optimize From Series

	Label	Manufacturer	Min Chord Thick [in]	Reqd Chord Species	Bolt Size	Defl at Peak Load [in]
1	DTT1Z-SD9x1.5_DF-SP	Simpson HoldDowns	1.5	DF	N/A	0.17
2	DTT1Z-(6)0.148x1.5_DF-SP	Simpson HoldDowns	1.5	DF	N/A	0.167
3	DTT2Z-SDS1.5_1.5_DF-SP	Simpson HoldDowns	1.5	DF	N/A	0.105
4	DTT2Z-SDS1.5_3_DF-SP	Simpson HoldDowns	3	DF	N/A	0.128
5	DTT2Z-SDS2.5_3_DF-SP	Simpson HoldDowns	3	DF	N/A	0.128
6	DTT1Z-SD9x1.5_SPF-HF	Simpson HoldDowns	1.5	HF	N/A	0.17
7	DTT1Z-(6)0.148x1.5_SPF-HF	Simpson HoldDowns	1.5	HF	N/A	0.167
8	DTT1Z-(8)0.148x1.5_SPF-HF	Simpson HoldDowns	1.5	HF	N/A	0.167
9	DTT2Z-SDS1.5_1.5_SPF-HF	Simpson HoldDowns	1.5	HF	N/A	0.105
10	DTT2Z-SDS1.5_3_SPF-HF	Simpson HoldDowns	3	HF	N/A	0.128
11	DTT2Z-SDS2.5_3_SPF-HF	Simpson HoldDowns	3	HF	N/A	0.128
12	HDU2-SDS2.5_DF-SP	Simpson HoldDowns	3	DF	N/A	0.088
13	HDU4-SDS2.5_DF-SP	Simpson HoldDowns	3	DF	N/A	0.114
14	HDU5-SDS2.5_DF-SP	Simpson HoldDowns	3	DF	N/A	0.115
15	HDU8-SDS2.5_3_DF-SP	Simpson HoldDowns	3	DF	N/A	0.11

OK
Cancel

The procedure that RISA uses for hold-down optimization is fundamentally based upon the assumption that there is a 'cost' to allowable tension in a product, and therefore the ideal hold-down would have as little tensile capacity as possible to meet code requirements. Once the program has determined the tensile force required to hold-down the wall it will choose the most economical hold-down product based on that which most closely matches (but does not exceed) the tension demand. The program looks to the **Allowable Tension** field of the hold-down schedule to choose the design.

Strap Optimization

3D Strap Schedule
?
×

Select Manufacturer Simpson Chord Straps

Available HD Series

SIMPSON Chord Straps

CMST_DF-SP

CMST_SPF-HF

CS_DF-SP

CS_SPF-HF

CSHP_DF-SP

CSHP_SPF-HF

MST_DF-SP

MST_SPF-HF

HST_DF-SP

HST_SPF-HF

Current Selection Type Optimize From Series

	Label	Manufacturer	Clear Span [in]	End Length [in]	Reqd Chord Species	Fastener Type	Number
1	MSTA49_18_DF/SP_Nail	Simpson Chord Straps	18	n/a	DF/SP	Nail	26
2	MSTA49_16_DF/SP_Nail	Simpson Chord Straps	16	n/a	DF/SP	Nail	26
3	MSTC28_18_DF/SP_Nail	Simpson Chord Straps	18	n/a	DF/SP	Nail	12
4	MSTC28_16_DF/SP_Nail	Simpson Chord Straps	16	n/a	DF/SP	Nail	16
5	MSTC40_24_DF/SP_Nail	Simpson Chord Straps	24	n/a	DF/SP	Nail	20
6	MSTC40_18_DF/SP_Nail	Simpson Chord Straps	18	n/a	DF/SP	Nail	28
7	MSTC40_16_DF/SP_Nail	Simpson Chord Straps	16	n/a	DF/SP	Nail	32
8	MSTC52_24_DF/SP_Nail	Simpson Chord Straps	24	n/a	DF/SP	Nail	36
9	MSTC52_18_DF/SP_Nail	Simpson Chord Straps	18	n/a	DF/SP	Nail	44
10	MSTC52_16_DF/SP_Nail	Simpson Chord Straps	16	n/a	DF/SP	Nail	48
11	MSTC66_30_DF/SP_Nail	Simpson Chord Straps	30	n/a	DF/SP	Nail	48
12	MSTC66_24_DF/SP_Nail	Simpson Chord Straps	24	n/a	DF/SP	Nail	54
13	MSTC66_18_DF/SP_Nail	Simpson Chord Straps	18	n/a	DF/SP	Nail	64
14	MSTC66_16_DF/SP_Nail	Simpson Chord Straps	16	n/a	DF/SP	Nail	68
15	MSTC78_30_DF/SP_Nail	Simpson Chord Straps	30	n/a	DF/SP	Nail	64

OK
Cancel

The procedure that RISA uses for strap optimization is fundamentally based upon the assumption that there is a 'cost' to allowable tension in a product, and therefore the ideal strap would have as little tensile capacity as possible to meet code requirements. Once the program has determined the tensile force required to strap down the wall it will choose the most economical strap product based on that which most closely matches (but does not exceed) the tension demand. The program looks to the **Allowable Tension** field of the strap schedule to choose the design.

Optimization Options

For users who are new to wood wall design within RISA, the best procedure is to utilize the full databases, and to limit the potential designs by utilizing the [design rules spreadsheets](#). This results in a design based on the maximum number of options, which is often the most efficient design.

For experienced users who have more specific limitations in terms of the designs they would like to see, user-defined **Groups** (or families) are the solution. For example, an engineer who prefers to use only one sheathing thickness, or one nail type can create a custom Group that contains only the arrangements they want. For more information on creating these custom groups see [Appendix F- Wood Shear Wall Files](#).

For more information on Wood Walls see [Wood Wall Results](#).

Wood Wall - Design Rules

The wood wall panel element allows you to easily model, analyze and design wood walls for in plane loads. Here we will explain the wood specific inputs and design considerations. For general wall

panel information, see the [Wall Panels](#) topic. For information on wood wall design considerations, see the [Wood Wall - Design](#) topic. For wood wall results interpretation, see the [Wood Wall Results](#) topic.

Unity Check

Wall Panel U.C. Parameters				
Masonry Lintel	Wood Wall (Studs)	Wood Wall (Fasteners)	CFS Wall (Studs)	CFS Wall (Fasteners)
Unity Check	Concrete Wall (Rebar)	Concrete Wall (Misc)	Masonry Wall	Masonry In Masonry Out
	Label	Max Bending Chk	Max Shear Chk	
1	6" Wood Wall	1	1	
2	8" Wood Wall	1	1	

Wall Panel Unity Check Parameters

Column	Description
Max Bending Chk	The Max Bending Chk column defines the hold-down maximum code check value. Typically this will always be 1.0, but allows the option of having the program pick out a hold-down that is not at maximum capacity. Putting a value of 0.8 will choose a hold-down that is at 80% of capacity.
Max Shear Chk	The Max Shear Chk column defines the shear panel selection maximum code check value. Typically this will always be 1.0, but allows the option of having the program pick out a shear panel layout that is not at maximum capacity. Putting a value of 0.8 will choose a panel thickness and nailing that is at 80% of capacity.

Wood Wall (Studs)

Wood Wall Panel Parameters										
Unity Check	Concrete Wall (Rebar)	Concrete Wall (Misc)	Masonry Wall	Masonry In	Masonry Out					
Masonry Lintel	Wood Wall (Studs)			Wood Wall (Fasteners)	CFS Wall (Studs)	CFS Wall (Fasteners)				
	Label	Top Plate	Sill Plate	Studs	Min Stud Space [in]	Max Stud Space [in]	Green Lumber?	Header Size	Header Matl	
1	6" Wood Wall	2-2X8	2X6	2X6	16	16	<input type="checkbox"/>	6x8	Same as Wall	
2	8" Wood Wall	2-2X8	2X6	2X6	16	16	<input type="checkbox"/>	6x8	Same as Wall	

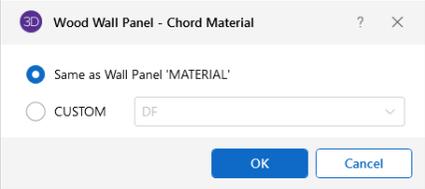
Wood Wall Panel Parameters

Column	Description
Top Plate	The Top Plate column specifies the member to be used as a top plate for your wall. A top plate is a member that runs continuously along the top of the wall studs. Note that you can use multiple plies of nominal lumber, or custom shapes.
Sill Plate	The Sill Plate column specifies the member to be used as a sill plate for your wall. A sill plate is a member that runs continuously along the bottom of the wall studs. Note that you can use multiple plies of nominal lumber, or custom shapes.
Studs	The Studs column specifies the member to be used for studs in your wall. Studs are vertical members in the wall, attached to the sill plate at the bottom and the top plate at the top. Note that you can use multiple plies of nominal lumber, or custom shapes.
Min/Max Stud Space	<p>The Min/Max Stud Space column lets you specify a minimum and maximum spacing of wall studs. The program can then optimize the stud spacing based on axial design only. For information on how the optimization works, see the Wood Wall - Design topic.</p> <ul style="list-style-type: none"> • If you specify the maximum and minimum stud spacing as the same value, then we will use that value exclusively. • Out of plane design is not performed for wood walls, so any optimization is based only on axial forces in the studs.
Green Lumber	<p>The Green Lumber column contains a checkbox that lets you choose a parameter for moisture content.</p> <ul style="list-style-type: none"> • NDS Design: Check this box if your moisture content is greater than 19%. The program multiplies the G_a value of the shear panel by 0.5 per <i>Note 5 of Tables 4.3A and 4.3B</i> of the NDS SDPWS. • CSA O86 Design: Check this box if you have wet service conditions. This effects the Service Condition Factors, K_s per <i>CSA O86-14 Table 6.4.2</i>.
Header Size	<p>The Header Size column defines the default header size for all openings.</p> <p>Note that this can be modified in the Wall Panel Editor by double-clicking the opening and choosing Custom.</p>
Header Matl	The Header Matl columns lets you change the material for the header. By default the program uses the same material for the header as for the studs, chords, etc.

Design Rules - Wood Wall (Fasteners)

Additional Wood Wall Panel Parameters												
Unity Check	Concrete Wall (Rebar)	Concrete Wall (Misc)	Masonry Wall	Masonry In	Masonry Out	Masonry Lintel	Wood Wall (Studs)	Wood Wall (Fasteners)	CFS Wall (Studs)	CFS Wall (Fasteners)		
	Label	Schedule	Min Panel Thick [in]	Max Panel Thick [in]	Double Sided Panel?	Max. Nail Spacing	Min. Nail Spacing	HD Chords	HD Chord Matl	Hold Down	Chord Strap	Eccentricity
1	6" Wood Wall	AWC 2021 OSB	0.375	0.75	Optimum	6-in.	2-in.	2-2X8	Same as Wall	SIMPSON HoldDowns	SIMPSON Chord Straps	<input checked="" type="checkbox"/>
2	8" Wood Wall	AWC 2021 OSB	0.375	0.75	Optimum	6-in.	2-in.	2-2X8	Same as Wall	SIMPSON HoldDowns	SIMPSON Chord Straps	<input checked="" type="checkbox"/>

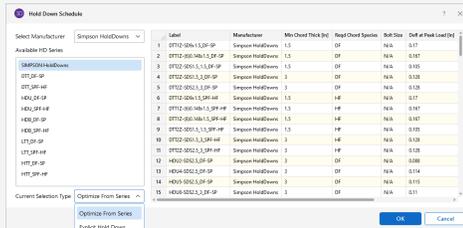
Additional Wood Wall Panel Parameters

Column	Description
Schedule	<p>The Schedule column lets you select the Code, and Panel Group you would like to use for design optimization. By unchecking the Select Entire Panel Group box, an individual panel type may be assigned.</p> <p>For information on how the optimization works, see the Wood Wall - Design topic. For more information on this schedule, as well as information on how to edit or create your own custom schedule, see Appendix F-Wood Design Databases</p>
Min Panel Thick	The Min Panel Thick columns lets you set the minimum thickness of the sheathing that will be designed. If the same value is input for both min and max, that is the thickness used.
Max Panel Thick	The Max Panel Thick columns lets you set the maximum thickness of the sheathing that will be designed. If the same value is input for both max and min, that is the thickness used.
Double Sided	The Double Sided column lets you choose whether you want the program to force sheathing on only one side of the panel, both sides, or to choose the optimum based on weight.
Max Nail Spacing	The Max Nail Spacing column lets you set the maximum spacing of the nails that fasten the sheathing to the boundary members (top plate, sill plate, hold down chords). Note that a 12" spacing is assumed for all field nailing (nails fastening the sheathing to the internal studs).
Min Nail Spacing	The Min Nail Spacing column lets you set the minimum spacing of the nails that fasten the sheathing to the boundary members (top plate, sill plate, hold down chords). Note that a 12" spacing is assumed for all field nailing (nails fastening the sheathing to the internal studs).
HD Chords	You can choose what member size you would like to use for the Hold Down Chords (Posts) at both ends of the wall panel.
HD Chord Material	<p>You can specify whether the hold down chords are of the same material as the wall, or another material.</p> 

Column	Description
--------	-------------

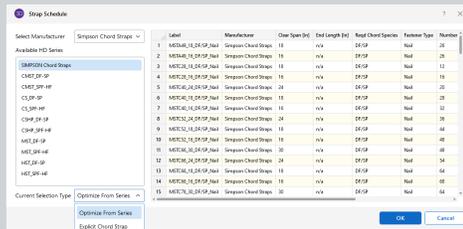
Hold Down

You can select the Manufacturer and Hold Down Series you would like to use for design optimization. By selecting **Explicit Hold Down** from the **Current Selection Type** drop-down you can select an individual hold down product to be assigned. For information on how the optimization works, see the [Wood Wall - Design](#) topic. For more information on this schedule, as well as information on how to edit or create your own custom schedule, see [Appendix F-Wood Shear Wall Files](#)



Chord Strap

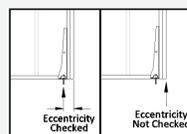
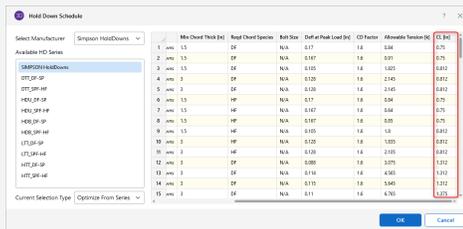
You can select the Manufacturer and Strap Series you would like to use for design optimization. By selecting **Explicit Hold Down** from the **Current Selection Type** drop-down you can select an individual hold down product to be assigned. For information on how the optimization works, see the [Wood Wall - Design](#) topic. For more information on this schedule, as well as information on how to edit or create your own custom schedule, see [Appendix F-Wood Shear Wall Files](#)



Eccentricity

You can choose to include the eccentricity of the Hold-Down location by checking the "Eccentricity" check-box in this spreadsheet. This is used in calculating the [Chord forces](#) and the [Hold Down force](#). The Hold-Down center-line is defined in the Hold-down database.

If the "Eccentricity" check-box is left unchecked the program will use the full length of the wall (ignoring the chord thicknesses and the hold down eccentricity).



Column	Description
	 Note: <ul style="list-style-type: none"> • Eccentricity option is only applicable to hold-downs, not straps. • Eccentricity option does not affect the capacity of the wall sheathing.

Wood Wall Results

Wood Wall results are presented in the **Wall Panel Design** Spreadsheet and the detail reports.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Wood Walls**.

Wall Results Spreadsheets

Wood Wall Axial Results

Concrete In		Concrete Out		Concrete Seismic		Masonry In		Masonry Out		Masonry Lintel	
Masonry Seismic		Wood Wall Axial		Wood Wall In-Plane		Wood Header		CFS Wall Axial		CFS Wall In-Plane	
	Wall Panel	Region	Stud Size	Stud Spacing[in]	Axial Check	Gov LC	Chord Size	Chord Axial Check	Gov LC		
1	WP1	R3	2X6	16	0.121	1	2-2X8	0.032	1		
2		R4	2X6	16	0.076	1	2-2X8	0.023	1		
3		R5	2X6	16	0.121	1	2-2X8	0.032	1		
4		R6	2X6	16	0.076	1	2-2X8	0.023	1		

Wood Wall Panel Axial

Column	Description
Wall Panel	The Wall Panel column lists the wood wall panels that you have defined.
Region	The Region column lists the wall panel region that the results are based on.
Stud Size and Spacing	The Stud Size and Spacing columns show the optimum stud size and spacing chosen for your wall, based on the Design Rules you have defined.


Note: For [Perforated](#) or [Force Transfer Around Openings](#) design methods, regions are not used. Thus, an N/A will be displayed.

Column	Description
Axial Check	The Axial Check value is a code check ratio between the member load and the member capacity.
Gov LC	The Gov LC column adjacent to the Axial Check column shows the governing load combination for the design.
Chord Size	The Chord Size column shows the optimum chord size chosen for your wall, based on the Design Rules you have defined. Note that the chords are the vertical hold-down members/posts at the both ends of the wall.
Chord Axial Check	The Chord Axial Check value is a code check ratio between the member load and the member capacity
Gov LC	The Gov LC column adjacent to the Chord Axial Check column shows the governing load combination for the design.

 **Note:**

- When running the [Segmented](#) design method, the wall panel regions above and below the opening are not considered in design. Thus an NC (no calculation) will be displayed.
- If there are some constraints that will not allow a wall to be designed, an NC (no calculation) will be displayed. Check the **Warning Log** within the program for more information on this.

Wood Wall In-Plane Results

Concrete In		Concrete Out		Concrete Seismic		Masonry In		Masonry Out		Masonry Lintel	
Masonry Seismic		Wood Wall Axial		Wood Wall In-Plane		Wood Header		CFS Wall Axial		CFS Wall In-Plane	
	Wall Panel	Shear Panel Label	Region	Shear Check	Shear Force[k/R]	Gov LC	Hold-Down Label	Chord Strap Label	Tension Check	Tie-Down Force[k]	Gov LC
1	WP1	S1_3/8_8d@6	R3	0.018	0.004	1	Not Req'd	NC	NC	NC	NC
2		S1_3/8_8d@6	R4	0.024	0.005	1	Not Req'd	NC	NC	NC	NC
3		S1_3/8_8d@6	R5	0.018	0.004	1	Not Req'd	NC	NC	NC	NC
4		S1_3/8_8d@6	R6	0.024	0.005	1	Not Req'd	NC	NC	NC	NC

Wood Wall Panel In Plane

Column	Description
Shear Panel Label	The Shear Panel Label shows the optimum shear panel arrangement chosen for your wall, based on the Design Rules you have defined.
Region	The Region gives the region for which the design values are being displayed.

Column	Description
	 Note: The Perforated or Force Transfer Around Openings methods do not consider regions in their design, thus N/A is displayed.
Shear Check	The Shear Check value is a code check ratio between the panel shear load and the panel shear capacity.
Shear Force	The Shear Force column shows the values which governed the design.
Gov LC	The Gov LC adjacent to the Shear Force column show the load combination which governed the design.
Hold-Down Label	The Hold-Down Label shows the optimum hold-down product chosen for your wall, based on the Design Rules you have defined.
Chord Strap Label	The Chord Strap Label shows the optimum strap product chosen for your wall, based on the Design Rules you have defined
Tension Check	The Tension Check value is a code check ratio between the tension load and the hold-down tensile capacity. This column will only be populated for wood walls at the base level of the structure. The program does not do wall to wall strap design
Tie-Down Force	The Tie-Down Force and Gov LC show the maximum value of hold-down/strap which produced the highest strap force. This maximum force could occur on either end of the wall/region.
Gov LC	The Tie-Down Force and Gov LC show the load combination which produced the highest strap force.  Note: <ul style="list-style-type: none"> When running the Segmented design method, the wall panel regions above and below the opening are not considered in design. Thus an NC (no calculation) will be displayed. If running a combined RISAFloor and RISA-3D model use RISAFloor to get header results. If there are some constraints that will not allow a wall to be designed, an NC (no calculation) will be displayed. Check the Warning Log within the program for more information on this. If 'default' is shown in the Hold-Down Label column it means that a hold down is not required for this wall/region.

Wood Wall Header Results

Wood Wall Panel Header Code Checks (AWC NDS-18 / AWC SPDWS-21 ASD)							
Concrete In		Concrete Out		Concrete Seismic		Masonry In	
Masonry Seismic		Wood Wall Axial		Wood Wall In-Plane		Wood Header	
Wall Panel	Header Label	Design Rule	Size	Bend UC	Shear UC	Gov LC	
1	WP1	H1	6" Wood Wall	6x8	NC	NC	NC
2		H2	6" Wood Wall	6x8	NC	NC	NC

Wood Wall Panel Header

Column	Description
Header Label	The Header Label column shows the label of your header. If there is more than one header in a wall, you can see which is which in the wall panel editor.
Design Rule	The Design Rule column shows the rule that is defining the size and material.
Size	The Size column shows the size from the Design Rule.
Bend UC	The Bend UC column shows the bending code check ratio. This is given for the load combination from the Gov LC column.
Shear UC	The Shear UC column shows the shear code check ratio. This is given for the load combination from the Gov LC column.
Gov LC	The Gov LC column shows the load combination that governed the design. The program finds the largest UC value for either bending or shear and reports the associated load combination here.

 **Note:**

- The **Bend UC** and **Shear UC** are always given for the SAME load combination. Whichever load combination produced the highest code check for either bending or shear will have its results shown in these columns.
- The program will only design wood headers for non-EL and non-WL load combinations.

Wood Wall Self Weight

The program calculates the self weight of a wood wall based on the weights of the individual components. Using the material density, the self weight is calculated for the studs, chords, top plates, sill plates, and sheathing. These are all then summed for the total self weight of the wall.

Wood Wall Detail Reports

The detail report gives detailed information about the wall design. The detail reports are specifically molded to the type of design specified. Here we will walk through how to access the different information for each of the types of design: [Segmented](#), [Perforated](#) and [Force Transfer Around Openings](#).



Note:

- RISAFloor only considers the **Segmented** design method. If using RISAFloor together with RISA-3D, simply taking your model into RISA-3D will open up the **Perforated** and **Force Transfer Around Openings** design methods.
- Many of the values for design checks seen below are not performed in RISAFloor as it is strictly a gravity design program.

Accessing the Detail Reports and the Specific Windows

Once you have a solved model, the detail reports become available. They are accessible in two ways:

- If you have the **Wall Panel Design** spreadsheet open,
- If you are in a graphic view of your model, there is a button that opens the detail report window.

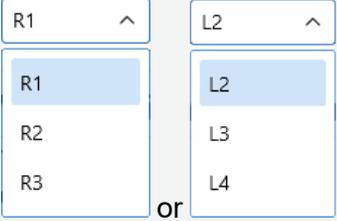
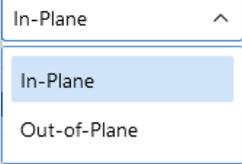
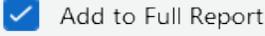


Note: Detail report information is not available for an envelope solution.

Once the detail report window is open, you see an area at the top of the window.

Detail Report Control Options

Option	Name	Description
	<p>Prior/Next</p>	<p>The left and right arrows let you scroll quickly between the different wall panels in your model.</p>
	<p>Wall Panel Part</p>	<p>The first drop-down list lets you choose between individual Region and Opening (Lintel) results and a summary of the entire Wall.</p>

Option	Name	Description
	<p>Regions/Headers</p>	<p>The second drop-down list lets you select between different Regions or Openings within the individual wall panel. “R” represents regions, while “L” represents levels (headers).</p> <p>The is available only when ‘Opening’ or ‘Region’ is chosen in the Wall Panel Part drop-down.</p>
	<p>In/Out Plane</p>	<p>If you have selected a Region, then you have the option of whether to view the in plane or out of plane report.</p> <p>The is available only when ‘Region’ is chosen in the Wall Panel Part drop-down.</p>
<p>The following options can be found at the bottom of the Detail Report window.</p>		
	<p>Print</p>	<p>Lets you print the Detail Report.</p>
	<p>Add to Full Report</p>	<p>Lets you add the current detail report you are viewing to the printed report. View the Printing topic for more information.</p>

The Wall detail report gives an overall summary of your wall, complete with governing code checks and opening information. The Opening detail report gives information to the header design for the opening as well as detailed information for the **Force Transfer Around Openings** method. The Region detail report only applies for **Segmented** walls. Below we have give detailed information on each type of design: [Segmented](#), [Perforated](#) and [Force Transfer Around Openings](#).

Segmented Method Results

The Segmented design method uses each of the three detail report windows to give design information.

Wall Window

This window gives an overview of the wall, giving controlling region information and deflection information. Note that this window only gives information on the full-height segments in your wall, as this is the basis of the **Segmented** method.

Input Echo

This lists information about the wall, similar to the [Region](#) report and also gives an image of the wall. The image shows the location of hold-downs/straps, regions and headers.

Design Summary

Detail Report - Design Summary

Criteria	Description
Enveloped Results	The Enveloped Results shows the code checks for all of the controlling elements in the wall and their associated load combinations.
Region Information	The Regional Information shows the tabulated results of all of the full-height regions in the wall.
Deflection Results	The Deflection Results shows both the calculated NDS deflection (Maximum Region Deflection) the FE deflection for use as a means of comparison. Because the NDS/CSA O86 equations are empirical and take into account many non-elastic considerations such as nail slip, these two values may not be the same. The use of the SSAF helps to make these two values similar.
Opening Information	The Opening Information states that header design cannot be completed with the Segmented method. The regions above and below the opening have their shear stiffnesses set to be zero and this causes the header forces to be invalid.

Opening Window

This window defines the openings in the wall. The segmented wall design method assumes zero stiffness over the opening, therefore, there is no header design for a Segmented wall with openings. Walls specified as a Perforated or FTAO can be analyzed for a header design.

Region Window

This window gives information for your wall on a region by region basis. Note that only full-height regions of the wall panel will have a region detail report. The **Segmented** method only considers these full height segments in the design of the wall.

The region detail report is split into four portions: [input echo](#), [diagrams and design](#), [design details](#), and [cross section detailing](#). Note that in RISAFloor the detail reports are less detailed because RISAFloor does not consider lateral forces which RISA-3D does.

Input Echo

Below is the input echo portion of the detail report.

Criteria section

Criteria	Description
Code	Gives the code used to design your wall panel.
Wall Material	Specifies the wood type assigned to the entire wall
Panel Schedule	Specifies the sheathing/nailing schedule database used to optimize panel selection (set in Design Rules)
Optimize HD	Shows whether or not the program needed to optimize the hold-down, or if the user explicitly defined a hold down
HD Manufacturer	Specifies the manufacturer of chosen hold-down
Strap Manufacturer	Specifies the manufacturer of chosen strap

Materials section

Materials	Description
Wall Studs	Specifies the wood material type assigned to the wall studs
Stud Size	Specifies the member size used for the wall studs
Chord Material	Specifies the wood material type assigned to the chords (vertical members at both ends of the wall)
Chord Size	Specifies the member size used for the chords (vertical members at both ends of the wall)
Top Plate & Sill	Specifies the wood material type assigned to the top and sill plates
Top Plate Size	Specifies the member size used for the top plate
Sill Plate Size	Specifies the member size used for the sill plate

Geometry section

Geometry	Description
Total Height	This is the height of the wall panel region
Total Length	This is the length of the wall panel region
Region H/W Ratio	This is the ratio of wall height to length, using the minimum wall height
Cap. Adj (2w/h)	This is an aspect ratio reduction factor for the shear panel strength per NDS SDPWS section 4.3.3.4 Exception 1 (2015). This factor applies only for seismic loads, thus it will be 1.0 for wind load combinations per NDS SDPWS section 4.3.4.1 (2005/8). This factor is applied separately for each full-height region in your wall.
Aspect Ratio	This is an aspect ratio reduction factor for the shear panel strength per NDS SDPWS section 4.3.4.2 (2015). This factor applies only for seismic and wind loads on Segmented or FTAO walls. This factor is applied separately for each region in your wall.
Gov. H/W Cap.	This is the governing (minimum) factor per the Cap. Adj (2w/h) and Aspect Ratios shown above. Only the minimum shall apply per NDS

Geometry	Description
	SDPWS section 4.3.3.4 Exception 1.
Wind ASIF	The code gives a 40% increase in the tables if the lateral load is wind over seismic. For seismic loads this ASIF will be 1.0 (only applicable to NDS design). For LRFD 2021 SDPWS, the code allows a 60% increase in capacity when wind load governs over seismic
Stud Spacing	This is the optimized stud spacing based on your Design Rules
K	This is the effective length, K Factor used for stud and chord compression design
HD Eccentricity	This is the eccentricity of the hold-down connection. This is based on the manufacturer's catalog and measured as the distance from the center of the chord to the hold-down bolt

Diagrams and Design

Detail Report for - R5 of WP1
— □ ×

< Wall Panel Label WP1 >
Region ▾ R5 ▾
In-Plane ▾

Detail Report: WP1 (Region R5)

CRITERIA	GEOMETRY	MATERIALS
Code: AWC NDS-18 / SDPWS-21:ASD	Total Height (ft): 15	Wall Studs: DF
Wall Material: DF	Total Length (ft): 5	Stud Size: 2X6
Panel Schedule: AWC 2021 OSB	Region H/W: 3.00	Chord Material: DF
Optimize HD: Yes	Cap. Adj. (2w/h): 1.00	Chord Size: 2-2X8
HD Manufacturer: SIMPSON	Aspect Ratio: 0.88	Top Pl. & Sill: DF
	Gov. H/W Factor: 1.00	Top Pl. Size: 2-2X8
		Sill Pl. Size: 2X6
		Specific Gravity: 0.5

ENVELOPE DIAGRAMS

Min: 1.575 at 15 ft
Max: 2.328 at 0 ft

Min: -0.016 at 15 ft
Max: 0.021 at 0 ft

Min: -0.981 at 15 ft
Max: 0.198 at 10.5 ft

Envelope Diagrams

These diagrams show the axial forces, in-plane shear, and in-plane moments of the wall, as well as the maximum and minimum forces and their locations.

**Note:**

- The diagrams are not actually enveloped.
 - The **Axial** diagram shown is the diagram from the governing load combination for *Stud* design.
 - The **Shear** diagram is the diagram from the governing load combination for *Shear Panel* design.
 - The **Moment** diagram is the diagram from the governing load combination for *Hold-Down* design.

Design Summary

This portion gives you the capacity and strength values at the section in the wall where the combined check is maximum, as well as the governing load combination. Much of this information is also reported in the **Wood Wall Panel Design** spreadsheets.

Design Summary Results

Section	Description
Shear Panel	<p>This section displays the Shear Panel code check results.</p> <p>The provided capacity of the shear panel is taken from the Shear Capacity column of the panel database. This is the allowable shear value from Table 2306.4.1 from the 2006 IBC (for NDS design) or Table 9.3A, 9.3B, and 9.3C from the CSA O86-14 (for Canadian design). This capacity automatically considers whether the loading is based on wind or seismic loads. The Governing LC explicitly states if the controlling load combination was based on Wind or Seismic. The program does a unity check for all LCs that are being solved, finds the maximum value and reports that information.</p> <div data-bbox="493 1409 1409 1780" style="border: 1px solid black; padding: 5px;">  <p>Note:</p> <ul style="list-style-type: none"> • A shear panel design will be chosen for the worst-case region in a segmented wall. That panel will then be used for all regions in that wall. The worst-case region is the one that has the highest Shear UC value. • Because wind and seismic loading allows for different design capacities, the highest shear in the wall may not be the governing shear (if that highest shear was due to wind). </div>
Chord Design	<p>This section displays the Chord Design code check results.</p> <p>The provided capacities of these members are calculated using the</p>

Section	Description
	<p>standard provisions for tension/compression members. These members are assumed to be fully braced in the weak axis, and unbraced in the strong axis. For more information on the chord force calculations, see the Wood Wall - Design topic.</p> <div style="border: 1px solid black; padding: 5px; margin-top: 10px;">  Note: For chord results, the tension/compression capacity is computed using the reduced cross-sectional properties caused by the hold-down bolt hole. </div>
Stud Design	<p>This section displays the Stud Design code check results.</p> <p>The provided capacities of these members are calculated using the standard provisions for tension/compression members. These members are assumed to be fully braced in the weak axis, and unbraced in the strong axis. For more information on the chord force calculations, see the Wood Wall - Design topic.</p>
Hold Down Design	<p>This section displays the Hold-Down Design code check results.</p> <p>The provided capacity of the hold-down is taken from the Allowable Tension column of the hold-down database. This is information supplied by the manufacturer. Note that we are modifying the Cd value for the hold-down based on taking a ratio of the assumed Cd values from the database and the Cd called for in the Load Combinations spreadsheet.</p>
Chord Straps	<p>This section displays the Chord Straps results.</p> <p>The provided capacity of the chord strap is taken from the Allowable Tension column of the chord strap database. This is information supplied by the manufacturer. Note that we are modifying the Cd value for the chord strap based on taking a ratio of the assumed Cd values from the database and the Cd called for in the Load Combinations spreadsheet.</p>
Deflection	<p>This section displays the Deflection results.</p> <p>The deflection listed in the detail report is based on an approximation from the design code.</p>

NDS Design Deflection

Per the NDS 2015 *Special Design Provisions for Wind and Seismic*, Equation 4.3-1:

$$\delta_{sw} = \frac{8vh^3}{EAb} + \frac{vh}{1000G_a} + \frac{h\Delta_a}{b}$$

Where:

- b = Shear wall length,
- Δ_a = Total vertical elongation of wall anchorage system (including fastener slip, device elongation, rod elongation, etc.) at the induced unit shear in the shear wall, in (This value is taken from the hold down database and scaled per the actual tension force; hence you multiply this value by the holddown ratio given in the output)
- E = Modulus of elasticity of end posts (chords), psi
- A = Area of end post (chord) cross-section, in²
- G_a = Apparent shear wall shear stiffness from nail slip and panel shear deformation, kips/in. (taken from shear panel database)
- h = Shear wall height, ft
- v = Induced unit shear, lbs/ft
- δ_{sw} = Total shear wall deflection determined by elastic analysis.

The first term of the above equation determines the Bending Component of the deflection.

The second term of the above equation determines the Shear Component of the deflection.

The third term of the above equation determines the Hold-Down Elongation, which causes additional deflection.



Note:

- This is the theoretical deflection of the wall. This may differ from the deflection of the wall as performed by finite element analysis within RISA. Therefore, this deflection value may not coincide with the reported deflection value in the deflections spreadsheets. For information on making the FEM deflections similar to the reported deflections from the NDS calculated deflections, see the [Shear Stiffness Adjustment Factor](#) information.
- For **Perforated** design deflections, the total deflection is to be divided by C_o per section 2305.3.8.2.9 of the IBC 2006. Because the unit shear values have already been amplified by C_o , the only portion of the deflection that needs to be divided is the hold-down portion.
- The hold-down deflection is reported for the maximum shear LC, which may not result in the largest hold-down component, but typically results in the highest total deflection.

CSA O86 Design Deflection

Per the CSA O86-14 clause 11.7.1.2 (or CSA O86-09 clause 9.7.1.1):

$$\Delta_{sw} := \frac{2 \cdot v \cdot H_s^3}{3 \cdot E \cdot A \cdot L_s} + \frac{v \cdot H_s}{B_v} + \frac{H_s}{L_s} \cdot d_a$$

Where:

- d_a = Total vertical elongation of wall anchorage system (including fastener slip, device elongation, rod elongation, etc.) at the induced unit shear in the shear wall, mm. (This value is taken from the hold down database and scaled per the actual tension force; hence you multiply this value by the hold down ratio given in the output)
- E = Modulus of elasticity of boundary elements (chords), N/mm²
- A = Area of end post (chord) cross-section, mm²
- B_v = Shear through-thickness rigidity of the sheathing, N/mm (taken from shear panel database)
- H_s = Shear wall segment height, mm
- L_s = Shear wall segment length, mm
- v = Induced unit shear, N/mm
- Δ_{sw} = Total shear wall deflection determined by elastic analysis.

The first term of the above equation determines the Bending Component of the deflection.

The second term of the above equation determines the Shear Component of the deflection.

The third term of the above equation determines the Hold-Down Elongation, which causes additional deflection.



Note:

- This is the theoretical deflection of the wall. This may differ from the deflection of the wall as performed by finite element analysis within RISA. Therefore, this deflection value may not coincide with the reported deflection value in the deflections spreadsheets. For information on making the FEM deflections similar to the reported deflections from the CSA calculated deflections, see the [Shear Stiffness Adjustment Factor](#) information.
- The hold-down deflection is reported for the maximum shear LC, which may not result in the largest hold-down component, but typically results in the highest total deflection.

Design Details

DESIGN SUMMARY			
STUDS			
Required Cap (k):	0.466	Governing LC:	1
Provided Cap (k):	3.835	Stud Spacing (in):	16
Ratio:	0.121	Studs in Region:	5
CHORDS			
Max Comp Force (k):	0.522	Gov Comp LC:	1
Comp Capacity (k):	16.095	Max Tens Force (k):	0
Comp Ratio:	0.032	Tens Capacity (k):	17.617
SHEAR PANEL			
Selected Shear Panel:	S1_3/8_8d@6	Ratio:	0.018
Reqd. Shear Cap. (k/ft):	0.004	Panel Grade:	St-I
Shear Cap. (k/ft):	0.230	Panel Thick (in):	0.375
Adj. Shear Cap. (k/ft):	0.230	Reqd Pen (in):	1.375
Governing LC:	1	Nail Size:	8d
Num. Sides:	One	Reqd. Spacing (in):	6
		Over Gyp. Board:	No
DEFLECTIONS			
Flexure Comp (in):	0.001	HD Elong (in):	0
Shear Comp (in):	0.003	Tot Deflection (in):	0.004
HOLD-DOWNS			
Hold-Downs Not Required.			

Design Details Results

Section	Description
Shear Panel	<p>The Shear Panel section displays the call-out from the shear panel database. The information below it is the information describing the call-out.</p> <div style="border: 1px solid black; padding: 10px; margin-top: 10px;">  Note: There will also be an "Adjusted Shear Capacity" that takes the given shear capacity from the design code and divides it by any appropriate adjustment factors from the Geometry section at the top of the report. </div>
Hold Down Design	<p>The Hold-Down section displays the call-out from the hold-down database. The information below it is the information describing the call-out. The "Base Capacity" is the capacity from the manufacturer divided by the assumed Cd value from the database. The "CD factor" that is</p>

Section	Description
	displayed is the value from the load combinations spreadsheet for the controlling load combination. The actual capacity of the hold-down is the Base Capacity*CD factor.
Chord Straps	The Chord Strap section displays the call-out from the strap database. The information below it is the information describing the call-out. The "Base Capacity" is the capacity from the manufacturer divided by the assumed Cd value from the database. The "CD factor" that is displayed is the value from the load combinations spreadsheet for the controlling load combination. The actual capacity of the chord strap is the Base Capacity*CD factor.

The above section of the report echoes the database information for the selected shear panel and hold-down. For more information on these properties refer to [Appendix F-Wood Shear Wall Files](#).

Cross Section Detailing

The last section of the detail report consists of the wall detailing information. This information is provided as a visual confirmation of the wall design. The wall thickness, and stud spacing are shown as dimensions. The triangle shows sheathing on one side of the wall, with the abbreviated panel designation. The chord sizes/forces and hold down or strap designations/forces are shown at either end. If either chord is only experiencing a compression force, the hold down or strap will not be drawn.



Note: The displayed chord force is not necessarily the force used in the hold-down design because hold-down optimization only considers the governing tension LC.

Perforated Method Results

Opening Window

This window is similar to the [Force Transfer](#) information.

Wall Window

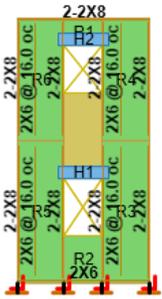
This is where the majority of the information is located for the perforated method.

Detail Report for - WP1
— □ ×

<
Wall Panel Label WP1
>
Wall
v

Detail Report: WP1

Wood Wall



GENERAL	GEOMETRY	MATERIALS
Code: AWC NDS-18 / SDPWS-21:ASD	Total Height (ft): 28	Top Pl. DF 2-2X8
Design Method: Segmented	Total Length (ft): 14	Sill DF 2X6
Wall Material: DF	Max H/W Ratio: NC	Wall Stud DF 2X6
Panel Schedule: AWC 2021 OSB		Chord DF 2-2X8
Sel. Shear Panel: S1_3/8_8d@6		

Perforated Method Wall Window

Section	Description
Criteria (General)	This section displays some of the important parameters in designing the wall.
Geometry	<p>This section displays dimensions and ratios for the wall panel.</p> <ul style="list-style-type: none"> The Wall H/W Ratio is checked against the aspect ratio limits given in Table 4.3.4 of the NDS 2015 <i>Special Design Provisions for Wind & Seismic</i> (SDPWS). The Max Opening Ht is used in the calculation of C_o. The % Full Ht Sheathed is used in the calculation of C_o. The Full Ht Sheathed is the Sum Li value for C_o.
Materials	This section displays the sizes of the members that are not explicitly talked about in the detail report.

Note: The top plate, sill plate and trimmer sizes are used only for the Material Take Off.

Detail Report for - WP1
— □ ×

<
Wall Panel Label WP1
>
Wall
v

ENVELOPED RESULTS

Controlling Shear Region	Shear Panel	Shear UC	Shear LC	Controlling Hold Down	Hold Down UC	Hold Down LC	Chord UC	Chord LC	Stud UC	Stud LC
R4	S1_3/8_8d@6	0.024	1	Not Req'd	NC	NC	0.032	1	0.121	1

REGION INFORMATION

Region Label	H/W Ratio	Shear UC	Shear LC	Hold-Down	Hold-Down UC	Hold-Down LC	Chord UC	Chord LC	Stud UC	Stud LC
R3	3.00	0.018	1	Not Req'd	NC	NC	0.032	1	0.121	1
R4	2.60	0.024	1	Not Req'd	NC	NC	0.023	1	0.076	1
R5	3.00	0.018	1	Not Req'd	NC	NC	0.032	1	0.121	1
R6	2.60	0.024	1	Not Req'd	NC	NC	0.023	1	0.076	1

DEFLECTION RESULTS

Maximum Region Deflection (in)	Deflection LC	Finite Element Deflection (in)	Shear Stiffness Adjustment Factor
0.004 (R4)	1	8.158e-6	1

OPENING INFORMATION

Headers of openings are not designed for Segmented walls.
Please choose Perforated or FTAO design method to get header design.

Print
 Add to Full Report

Close

Perforated Method Wall Window

Section	Description
Design Details	The Design Details section displays adjustment factors and some other values used to calculate C_o . See the Wall Panels topic for more information on the Shear Stiffness Adjustment Factor . See the Wood Wall - Design topic for more information on C_o .
Deflections	The Deflections section displays the calculated deflection for the three term shear wall equation from the NDS. See above for more information on deflections.
Wall Results	<p>The Wall Results section displays the following information:</p> <ul style="list-style-type: none"> Governing LC - The load combination that produced the highest allowable code check. This will also state if this load combination was a wind or seismic LC. Total Shear - The total shear in the wall for the governing LC. Max. Unit Shear - The maximum shear in the shear panel and it is what is used to optimize the sheathing/nailing selection from the shear panel database. Note that the Max Unit Shear

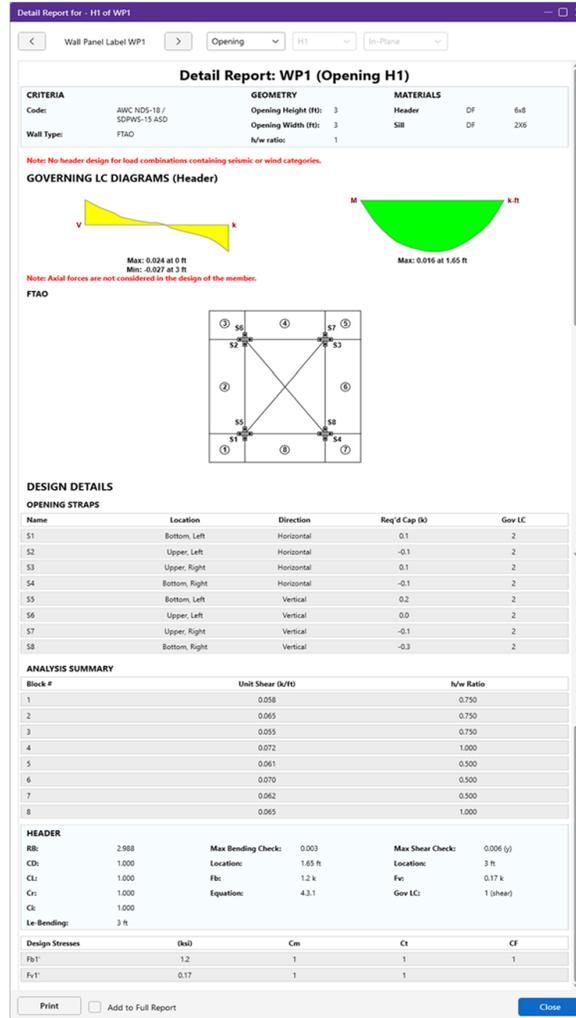
Section	Description
	<p>may not be the absolute maximum, because shear walls governed by wind are allowed a 40% stress increase. Thus, a wind Max Unit Shear would need to be 40% higher than a seismic Max Unit Shear to govern.</p> <ul style="list-style-type: none"> • Shear Ratio - A ratio of the Max Unit Shear over the Shear Capacity of the shear panel selected from the database.
Selected Shear Panel	The Selected Shear Panel section displays the same shear panel information that was given in the Segmented region report .
Selected Hold-Down	The Selected Hold-Down section displays the same shear hold down information that was given in the Segmented region report .
Selected Strap	The Selected Strap section displays the same strap information that was given in the Segmented region report .
Cross Section Detailing	The Cross Section Detailing section displays a detailed view of the wall. For more information see the Segmented section.

Force Transfer Method Results

Wall Window

This is the overall wall information and is essentially identical to the [Perforated method](#) wall window information. There is some geometry information that is not necessary for FTAO that is omitted.

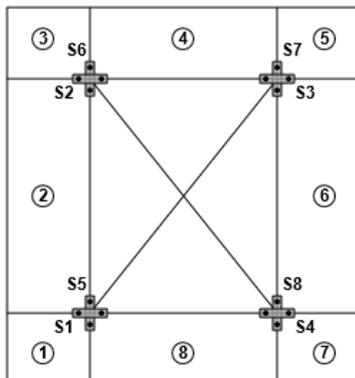
Opening Window



Opening Window

Section	Description
Criteria	The Criteria section gives the code being used and which design method used.
Geometry	The Geometry section gives the opening dimensions and the h/w ratio.
Materials	The Materials section gives dimensions for some of the members in the wall.
Envelope Diagrams	The Envelope Diagrams give the enveloped shear and moment diagrams for the header beam above the opening
FTAO	The FTAO graphic shows the design block numbers around the wall panel opening along with the strap numbers.

Design Details



DESIGN DETAILS

OPENING STRAPS

Name	Location	Direction	Req'd Cap (k)	Gov LC
S1	Bottom, Left	Horizontal	0.1	2
S2	Upper, Left	Horizontal	-0.1	2
S3	Upper, Right	Horizontal	0.1	2
S4	Bottom, Right	Horizontal	-0.1	2
S5	Bottom, Left	Vertical	0.2	2
S6	Upper, Left	Vertical	0.0	2
S7	Upper, Right	Vertical	-0.1	2
S8	Bottom, Right	Vertical	-0.3	2

ANALYSIS SUMMARY

Block #	Unit Shear (k/ft)	h/w Ratio
1	0.058	0.750
2	0.065	0.750
3	0.055	0.750
4	0.072	1.000
5	0.061	0.500
6	0.070	0.500
7	0.062	0.500
8	0.065	1.000

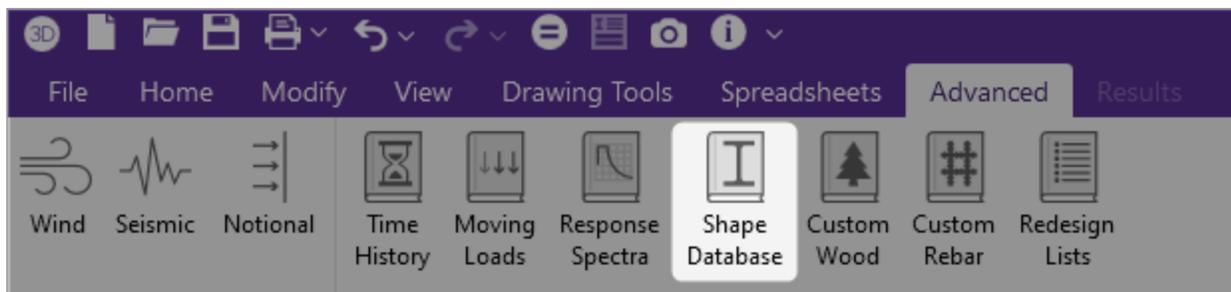
The **Design Details** section is split into two different tables: 'Opening Straps' and 'Analysis Summary'.

Design Details

Section	Description
Opening Straps	<p>The Opening Straps information gives the location, direction, force in the strap and the load combination that caused that force.</p> <div data-bbox="492 464 1409 726" style="border: 1px solid black; background-color: #e6f2ff; padding: 10px;"><p> Note:</p><ul style="list-style-type: none">• For more information on how the strap forces and unit shears are calculated, see the Wood Wall - Design topic.• The program does not design the straps around the opening, just presents the forces.</div>
Analysis Summary	<p>The Analysis Summary provides the unit shear and the h/w ratio for each of the blocks.</p> <p>The information below the Analysis Summary section is the code check information for the 'Header' member. This is identical to the information given for Segmented Window information.</p>

Wood - Database

The Wood Database may be accessed from the **Wood** tab of the **Section Sets** spreadsheet by clicking in the **Shape** field and then clicking , or by clicking the **Shape Database** button on the **Advanced** ribbon and then clicking the **Wood** tab of the **Shape Database** window.



3D Shape Selection
?
✕

⊞
 General

I
 Hot Rolled

C
 Cold Formed

■
Wood

□
 Aluminum

I
 Stainless

Database NDS Wood Shapes

Shape Type Nominal Sawn Lumber

Wood Type

Solid Sawn

Glulam

SCL

Custom

Material DF ...

2X2
 2X3
2X4
 2X5
 2X6

Plies 1

Ply Connection Nails

Properties

F_b	1	ksi	E	1700	ksi
F_t	0.675	ksi	E_{mod}	1	
F_v	0.18	ksi	COV_E	0.25	
F_c	1.5	ksi	E_{min}	621.025	ksi
SG	0.5				

Preview

3 1/2 in (d)

1 1/2 in (b)

Type Solid Sawn, Visually Graded
 Species Douglas Fir-Larch
 Grade No.1

Save Database as default

OK
Cancel

Wood Material/Shape Selection

Unlike any other material, the design properties for wood members depends both on the material selection as well as the shape selection. Therefore, the **Wood** tab of the **Shape Selection** database contains shape and material selections.

Material Selection

The 'Material Selection' menu reads in material options that are currently listed on the **Wood** tab of your [Materials](#) spreadsheet. You can select one of these existing materials, or click on the

Create/Edit Wood Material button  to open the Create/Edit Wood Material window, which lets you add a new material, or edit an existing material, without having to go to the Materials spreadsheet.

Create/Edit Wood Material Window

The Create/Edit Wood Material window lets you create or edit a material that is then automatically added to your [Materials](#) spreadsheet. This allows you to enter/select this information directly from the **Draw Members** Property Panel without having to first manually enter the information in the Materials spreadsheet.

 Edit Wood Material
? ×

Label

Wood Properties

Wood Type

Database

Species

Grade

Design Properties

Wet Properties, Cm

Elastic Modulus Modifier, Emod

Poisson's Ratio, U

Coefficient of Thermal Expansion /10⁵ °F

Density k/ft³

Allowable Stresses

F _b	1	ksi
F _t	0.675	ksi
F _v	0.18	ksi
F _c	1.5	ksi
E	1700	ksi
COV _E	0.25	
E _{min}	621.025	ksi
Specific Gravity	0.5	

To create a new custom wood material:

1. Choose **Custom** from the Wood Type drop down.
2. Click the **New** button to manually enter the design values for a new custom wood material.

This information will then be automatically saved to the [Custom Wood Species](#) spreadsheet.

Size Selection

Currently there are two ways to define a member size. In the **Shape Type** dropdown the options available are from either the defined list of **Nominal Sawm Lumber** sizes, or by using the **Net Rectangular/Round** option (typically referred to as Full Sawn).

- The **Nominal Sawm Lumber** option populates a list of nominal sized shapes for the NDS design code and a list of net sized shapes for the CSA O86 design code. When the NDS design code is chosen, the program automatically reduces the shape properties to their net sizes. If the CSA O86 design code is selected, a list of *actual* metric sizes are shown.
- Either the **Net Rectangular** or **Net Round** option lets you manually enter any *dressed* size that you like.
 - Rectangular imperial full sawn (as opposed to nominal dimensions) shapes will be designated with a "FS" suffix. You can also use this nomenclature to type the size directly into the spreadsheet. For example, a 7.25"x12" net rectangular member can be typed in as "7.25x12FS".
 - Rectangular metric full sawn (as opposed to nominal dimensions) rectangular shapes will be designated with a "MFS" suffix. You can also use this nomenclature to type the size directly into the spreadsheet. For example, a 184mmx305mm net rectangular member can be typed in as "184x305MFS".
 - Round imperial shapes will be designated with a "RND" suffix. You can also use this nomenclature to type the size directly into the spreadsheet. For example, a 10" net round member can be typed as "10RND".
 - Round metric shapes will be designated with a "MRND" suffix. You can also use this nomenclature to type the size directly into the spreadsheet. For example, a 254mm net round member can be typed as "254MRND".

**Note:**

- Multiple-ply lumber is assumed to be fastened sufficiently to achieve full composite behavior between plies. Therefore, a two-ply member will have eight times greater minor-axis flexural stiffness than a single ply member (as opposed to double the stiffness).
- If using multiple plies there is a **Bolted** check-box. If this box is unchecked it is assumed that the plies are nailed together. If bolted is checked then it is assumed the plies are bolted together. This affects the Kf value from section 15.3.2 of the NDS 2005/2008, 2012, 2015, and 2018. If you define the multi-ply section as bolted a "B" (or "MB" for metric sizes) will appear after the shape name.

View Design Properties

Once you have selected the appropriate member size and material, you can view the un-adjusted wood design values (Fb, Fv, Ft, etc.) for the selected combination in the **Properties** table at the bottom of the window. The final adjusted values (Fb', Fv', Ft', etc.) will then be displayed after solution in the [Member Detail Reports](#).

Wood Database Limitations

Round Shapes

Per section 3.7.3 of the NDS, the design of a round cross section shall be based on the design calculations for a square shape with the same cross-sectional area. Therefore the moment of inertia values will be calculated per the equivalent square shape, not the round cross section. However, the section modulus value, S is calculated per the actual round shape.

Glulam Databases

The program includes five tables of available glulam materials: NDS Tables 5A - 5D and CSA O86 Table 6.3. The values in NDS tables 5B & 5D depend on the number of laminations in the glulam. Therefore you will see three listings (2 Laminations, 3 Laminations, or 4 Laminations) for each material. If you have a custom material that is not available from these lists, you can always enter your appropriate design values as a [Custom Wood Material](#) for use in your model.

**Note:**

- The design values used per NDS tables 5B & 5D take into account footnote #1 to reduce the reference shear design value Fvy depending on the number of laminations.
- None of the other footnotes in either of these tables is taken into account.

Mechanically Graded Lumber

Mechanically graded lumber grades (MSR & MEL) are available for use with the *ANSI/AWC NDS-2015* and the *CSA O86-2014* design codes. Older versions of the design codes will not support design of mechanically graded materials from the program databases. However, you can always enter the properties as a Custom Wood Material to get the design calculations for these materials in older code versions.

Hot Rolled	Cold Formed	Wood	Concrete	Masonry	Aluminum	Stainless	General						
Label	Type	Database	Species	Grade	Cm	Cl	Emod	Nu	Therm. Coeff. [1e ⁶ F ⁻¹]	Density [k/ft ³]			
2	SP	Solid Sawn	Visually Graded	Southern Pine	No.1	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035		
3	HF	Solid Sawn	Visually Graded	Hem-Fir	No.1	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035		
4	SPF	Solid Sawn	Visually Graded	Spruce-Pine-fir	No.1	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035		
5	24F-1.8E DF Balanced	Glulam	NDS Table 5A	24F-1.8E_DF_BAL	na	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035		
6	24F-1.8E DF Unbalanced	Glulam	NDS Table 5A	24F-1.8E_DF_UNBAL	na	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035		
7	24F-1.8E SP Balanced	Glulam	NDS Table 5A	24F-1.8E_SP_BAL	na	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035		
8	24F-1.8E SP Unbalanced	Glulam	NDS Table 5A	24F-1.8E_SP_UNBAL	na	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035		
9	1.3E-1600F_VERSALAM	SCL	Boise Cascade	1.3E-1600F_VERSALAM	na	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035		
10	1.35E LSL_SolidStart	SCL	Louisiana Pacific	1.35E LSL_SolidStart	na	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035		
11	1.4E_RIGIDLAM LVL	SCL	Roseburg Forest Products_2023	1.4E_RIGIDLAM LVL	na	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035		
12	2.0E_DF Parallam PSL	SCL	TrusJoist	2.0E_DF Parallam PSL	na	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035		
13	LVL_PRL_1.5E_2250F	Custom	N/A	LVL_PRL_1.5E_2250F	na	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035		
14	LVL_Microlam_1.9E_2600F	Custom	N/A	LVL_Microlam_1.9E_2600F	na	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035		
15	PSL_Parallam_2.0E_2900F	Custom	N/A	PSL_Parallam_2.0E_2900F	na	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035		
16	LSL_TimberStrand_1.55E_2325F	Custom	N/A	LSL_TimberStrand_1.55E_2325F	na	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035		
17	Mech Graded DFL	Solid Sawn	Mechanically Graded	Douglas Fir- South	MSR_750F-1.4E	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035		

In order to assign a mechanically graded lumber material, you must select both the species and then the mechanical (MEL or MSR) grade. The species selection determines the shear capacity and compression perpendicular to grain capacity design values.

See *Footnote #2* from *NDS 2015 Table 4C* which says that all species from *NDS 2015 Table 4A* are applicable. See *clause 6.3.2* from *CSA O86-14* which says that specified shear strengths shall be taken from *CSA O86 Table 6.3.1A*.

The other design values are defined per the mechanical grade. See *NDS 2015 Table 4C* and *CSA O86-14 Tables 6.3.2 and 6.3.3*.

Structural Composite Lumber (SCL)

Each SCL manufacturer has documentation of their proprietary material design values. Because these are updated so often, RISA-3D relies instead on the design data from approved product reports (APA and ICC-ES) to build the material databases:

Manufacturer	Product Report	Notes
Ainsworth Engineered Wood*	APA PR-L260	*In 2015, Ainsworth Engineered was merged with Norbord Inc.
APA PRL-501	APA PR-L501	-
Boise Cascade	ICC-ES ESR-1040	-
Georgia Pacific**	APA PR-272	** Georgia Pacific sold to Boise Cascade in 2016
	APA PR-L257	
	APA PR-L266	
Global LVL	APA PR-L301	-
Louisiana Pacific	APA PR-L280	-
Murphy Engineered Wood	APA PR-L283	-
Pacific Woodtech	APA PR-L233	-
Roseburg Forest Products	APA PR-L289	-
	APA PR-L270	-
TrusJoist	ICC-ES ESR-1387	-

Footnote Limitations:

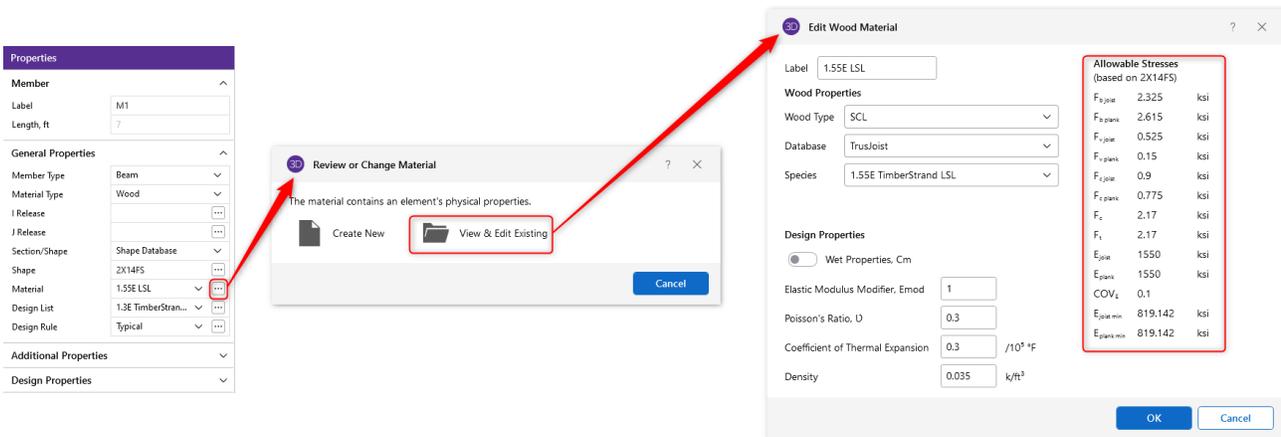
Each of the above reports includes tables with design values which were used to create the RISA-3D and RISAFloor SCL design property material databases. However, due to the difficulty of design variation, the following limitations are present:

- The Fb design values are not factored in accordance with the depth of the member. Currently RISA-3D always uses the single tabular Fb value regardless of member size.
- The Ft design values are not factored in accordance with the length or thickness of the member. Currently RISA-3D always uses the single tabular Ft value regardless of member size.
- The Fb design value will be increased by 4% if the member is designated as a repetitive member using the Cr check-box on the Wood tab of the Members spreadsheet.

Alternatively, you can always enter the appropriate design values, per the applicable footnotes, as a [Custom Wood Material](#).

Design Values

The design properties may be verified by clicking on the ellipsis next to the Material section of the specific member's information in the Property Panel.



The values are as follows:

- **Fb_joist:** Bending capacity due to joist/beam edge loading
- **Fb_plank:** Bending capacity due to plank/flat edge loading
- **Fv_joist:** Shear capacity due to joist/beam edge loading
- **Fv_plank:** Shear capacity due to plank/flat edge loading
- **Fc_joist:** Compressive (perpendicular to grain) axial capacity due to joist/beam edge loading
- **Fc_plank:** Compressive (perpendicular to grain) axial capacity due to plank/flat edge loading
- **Fc:** Compressive (parallel to grain) axial capacity, regardless of load direction
- **Ft:** Tensile axial capacity, regardless of load direction
- **E_joist:** Modulus of elasticity due to joist/beam edge loading
- **E_plank:** Modulus of elasticity due to plank/flat edge loading
- **E_mod:** Young's Modulus modifier (always set as 1.0)
- **COV_E:** Coefficient of Variation in Modulus of Elasticity per NDS Table F1. This value is used in the Emin calculation. For SCL materials COV_E always equals 0.10.
- **E_joist_min:** Emin (as calculated per NDS Appendix D equation D-4) due to joist/beam loading
- **E_plank_min:** Emin (as calculated per NDS Appendix D equation D-4) due to plank/flat edge loading

Currently the program only designs using the *Joist* properties of the SCL materials. This assumes that the loading is always on the small dimension side of the member. In a future version we will be taking the Plank values into consideration as well. In the meantime you can always enter the appropriate design values from these tables as a [Custom Wood Material](#) for use in your model.



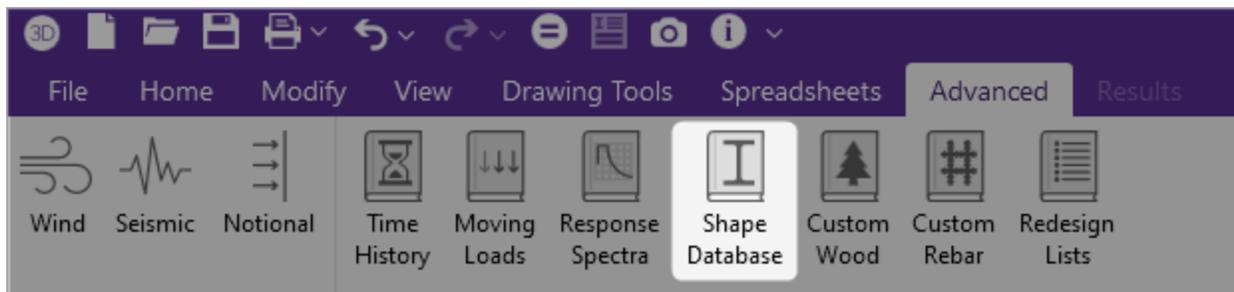
Note: Canadian wood member design only includes SCL design for the CSA O86-14 code.

Custom Wood Sizes

If you would like to enter explicit dimensions of a member or if the member is "Not Dressed", the member must be designated as a "Full Sawn" member by choosing **Net Rectangular** as the **Shape Type** and entering the exact dimensions of the member in the boxes below. This applies to regular wood species, custom wood species, Structural Composite Lumber (SCL), and Glu-Lam members.

Wood - Database

The Wood Database may be accessed from the **Wood** tab of the **Section Sets** spreadsheet by clicking in the **Shape** field and then clicking , or by clicking the **Shape Database** button on the **Advanced** ribbon and then clicking the **Wood** tab of the **Shape Database** window.



3D Shape Selection
? X

General

Hot Rolled

Cold Formed

Wood

Aluminum

Stainless

Database NDS Wood Shapes ▾

Wood Type

Solid Sawn

Glulam

SCL

Custom

Material DF ▾ ...

Shape Type Nominal Sawn Lumber ▾

2X2

2X3

2X4

2X5

2X6

Plies 1 ▾

Ply Connection Nails ▾

Properties

F_b	1	ksi	E	1700	ksi
F_t	0.675	ksi	E_{mod}	1	
F_v	0.18	ksi	COV_E	0.25	
F_c	1.5	ksi	E_{min}	621.025	ksi
SG	0.5				

Preview

3 1/2 in (d)

1 1/2 in (b)

Type Solid Sawn, Visually Graded
 Species Douglas Fir-Larch
 Grade No.1

Save Database as default

OK
Cancel

Wood Material/Shape Selection

Unlike any other material, the design properties for wood members depends both on the material selection as well as the shape selection. Therefore, the **Wood** tab of the **Shape Selection** database contains shape and material selections.

Material Selection

The 'Material Selection' menu reads in material options that are currently listed on the **Wood** tab of your [Materials](#) spreadsheet. You can select one of these existing materials, or click on the

Create/Edit Wood Material button  to open the Create/Edit Wood Material window, which lets you add a new material, or edit an existing material, without having to go to the Materials spreadsheet.

Create/Edit Wood Material Window

The Create/Edit Wood Material window lets you create or edit a material that is then automatically added to your [Materials](#) spreadsheet. This allows you to enter/select this information directly from the **Draw Members** Property Panel without having to first manually enter the information in the Materials spreadsheet.

 Edit Wood Material
? ×

Label

Wood Properties

Wood Type

Database

Species

Grade

Design Properties

Wet Properties, Cm

Elastic Modulus Modifier, Emod

Poisson's Ratio, U

Coefficient of Thermal Expansion /10⁵ °F

Density k/ft³

Allowable Stresses

F _b	1	ksi
F _t	0.675	ksi
F _v	0.18	ksi
F _c	1.5	ksi
E	1700	ksi
COV _E	0.25	
E _{min}	621.025	ksi
Specific Gravity	0.5	

To create a new custom wood material:

1. Choose **Custom** from the Wood Type drop down.
2. Click the **New** button to manually enter the design values for a new custom wood material.

This information will then be automatically saved to the [Custom Wood Species](#) spreadsheet.

Size Selection

Currently there are two ways to define a member size. In the **Shape Type** dropdown the options available are from either the defined list of **Nominal Saw Lumber** sizes, or by using the **Net Rectangular/Round** option (typically referred to as Full Sawn).

- The **Nominal Saw Lumber** option populates a list of nominal sized shapes for the NDS design code and a list of net sized shapes for the CSA O86 design code. When the NDS design code is chosen, the program automatically reduces the shape properties to their net sizes. If the CSA O86 design code is selected, a list of *actual* metric sizes are shown.
- Either the **Net Rectangular** or **Net Round** option lets you manually enter any *dressed* size that you like.
 - *Rectangular imperial* full sawn (as opposed to nominal dimensions) shapes will be designated with a "FS" suffix. You can also use this nomenclature to type the size directly into the spreadsheet. For example, a 7.25"x12" net rectangular member can be typed in as "7.25x12FS".
 - *Rectangular metric* full sawn (as opposed to nominal dimensions) rectangular shapes will be designated with a "MFS" suffix. You can also use this nomenclature to type the size directly into the spreadsheet. For example, a 184mmx305mm net rectangular member can be typed in as "184x305MFS".
 - *Round imperial* shapes will be designated with a "RND" suffix. You can also use this nomenclature to type the size directly into the spreadsheet. For example, a 10" net round member can be typed as "10RND".
 - *Round metric* shapes will be designated with a "MRND" suffix. You can also use this nomenclature to type the size directly into the spreadsheet. For example, a 254mm net round member can be typed as "254MRND".

**Note:**

- Multiple-ply lumber is assumed to be fastened sufficiently to achieve full composite behavior between plies. Therefore, a two-ply member will have eight times greater minor-axis flexural stiffness than a single ply member (as opposed to double the stiffness).
- If using multiple plies there is a **Bolted** check-box. If this box is unchecked it is assumed that the plies are nailed together. If bolted is checked then it is assumed the plies are bolted together. This affects the Kf value from section 15.3.2 of the NDS 2005/2008, 2012, 2015, and 2018. If you define the multi-ply section as bolted a "B" (or "MB" for metric sizes) will appear after the shape name.

View Design Properties

Once you have selected the appropriate member size and material, you can view the un-adjusted wood design values (F_b , F_v , F_t , etc.) for the selected combination in the **Properties** table at the bottom of the window. The final adjusted values (F_b' , F_v' , F_t' , etc.) will then be displayed after solution in the [Member Detail Reports](#).

Wood Database Limitations

Round Shapes

Per section 3.7.3 of the NDS, the design of a round cross section shall be based on the design calculations for a square shape with the same cross-sectional area. Therefore the moment of inertia values will be calculated per the equivalent square shape, not the round cross section. However, the section modulus value, S is calculated per the actual round shape.

Glulam Databases

The program includes five tables of available glulam materials: NDS Tables 5A - 5D and CSA O86 Table 6.3. The values in NDS tables 5B & 5D depend on the number of laminations in the glulam. Therefore you will see three listings (2 Laminations, 3 Laminations, or 4 Laminations) for each material. If you have a custom material that is not available from these lists, you can always enter your appropriate design values as a [Custom Wood Material](#) for use in your model.

**Note:**

- The design values used per NDS tables 5B & 5D take into account footnote #1 to reduce the reference shear design value F_{vy} depending on the number of laminations.
- None of the other footnotes in either of these tables is taken into account.

Mechanically Graded Lumber

Mechanically graded lumber grades (MSR & MEL) are available for use with the *ANSI/AWC NDS-2015* and the *CSA O86-2014* design codes. Older versions of the design codes will not support design of mechanically graded materials from the program databases. However, you can always enter the properties as a Custom Wood Material to get the design calculations for these materials in older code versions.

Hot Rolled	Cold Formed	Wood	Concrete	Masonry	Aluminum	Stainless	General							
Label	Type	Database	Species	Grade	Cm	Cl	Emod	Nu	Therm. Coeff. [1e ⁻⁶ F ⁻¹]	Density [k/ft ³]				
2	SP	Solid Sawn	Visually Graded	Southern Pine	No.1	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035			
3	HF	Solid Sawn	Visually Graded	Hem-Fir	No.1	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035			
4	SPF	Solid Sawn	Visually Graded	Spruce-Pine-fir	No.1	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035			
5	24F-1.8E DF Balanced	Glulam	NDS Table 5A	24F-1.8E_DF_BAL	na	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035			
6	24F-1.8E DF Unbalanced	Glulam	NDS Table 5A	24F-1.8E_DF_UNBAL	na	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035			
7	24F-1.8E SP Balanced	Glulam	NDS Table 5A	24F-1.8E_SP_BAL	na	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035			
8	24F-1.8E SP Unbalanced	Glulam	NDS Table 5A	24F-1.8E_SP_UNBAL	na	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035			
9	1.3E-1600F_VERSALAM	SCL	Boise Cascade	1.3E-1600F_VERSALAM	na	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035			
10	1.35E LSL_SolidStart	SCL	Louisiana Pacific	1.35E LSL_SolidStart	na	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035			
11	1.4E_RIGIDLAM LVL	SCL	Roseburg Forest Products_2023	1.4E_RIGIDLAM LVL	na	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035			
12	2.0E_DF Parallam PSL	SCL	TrusJoist	2.0E_DF Parallam PSL	na	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035			
13	LVL_PRL_1.5E_2250F	Custom	N/A	LVL_PRL_1.5E_2250F	na	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035			
14	LVL_Microlam_1.9E_2600F	Custom	N/A	LVL_Microlam_1.9E_2600F	na	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035			
15	PSL_Parallam_2.0E_2900F	Custom	N/A	PSL_Parallam_2.0E_2900F	na	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035			
16	LSL_TimberStrand_1.55E_2325F	Custom	N/A	LSL_TimberStrand_1.55E_2325F	na	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035			
17	Mech Graded DFL	Solid Sawn	Mechanically Graded	Douglas Fir- South	MSR_750F-1.4E	<input type="checkbox"/>	<input type="checkbox"/>	1	0.3	0.3	0.035			

In order to assign a mechanically graded lumber material, you must select both the species and then the mechanical (MEL or MSR) grade. The species selection determines the shear capacity and compression perpendicular to grain capacity design values.

See *Footnote #2* from *NDS 2015 Table 4C* which says that all species from *NDS 2015 Table 4A* are applicable. See *clause 6.3.2* from *CSA O86-14* which says that specified shear strengths shall be taken from *CSA O86 Table 6.3.1A*.

The other design values are defined per the mechanical grade. See *NDS 2015 Table 4C* and *CSA O86-14 Tables 6.3.2 and 6.3.3*.

Structural Composite Lumber (SCL)

Each SCL manufacturer has documentation of their proprietary material design values. Because these are updated so often, RISA-3D relies instead on the design data from approved product reports (APA and ICC-ES) to build the material databases:

Manufacturer	Product Report	Notes
Ainsworth Engineered Wood*	APA PR-L260	*In 2015, Ainsworth Engineered was merged with Norbord Inc.
APA PRL-501	APA PR-L501	-
Boise Cascade	ICC-ES ESR-1040	-
Georgia Pacific**	APA PR-272	** Georgia Pacific sold to Boise Cascade in 2016
	APA PR-L257	
	APA PR-L266	
Global LVL	APA PR-L301	-
Louisiana Pacific	APA PR-L280	-
Murphy Engineered Wood	APA PR-L283	-
Pacific Woodtech	APA PR-L233	-
Roseburg Forest Products	APA PR-L289	-
	APA PR-L270	
TrusJoist	ICC-ES ESR-1387	-

Footnote Limitations:

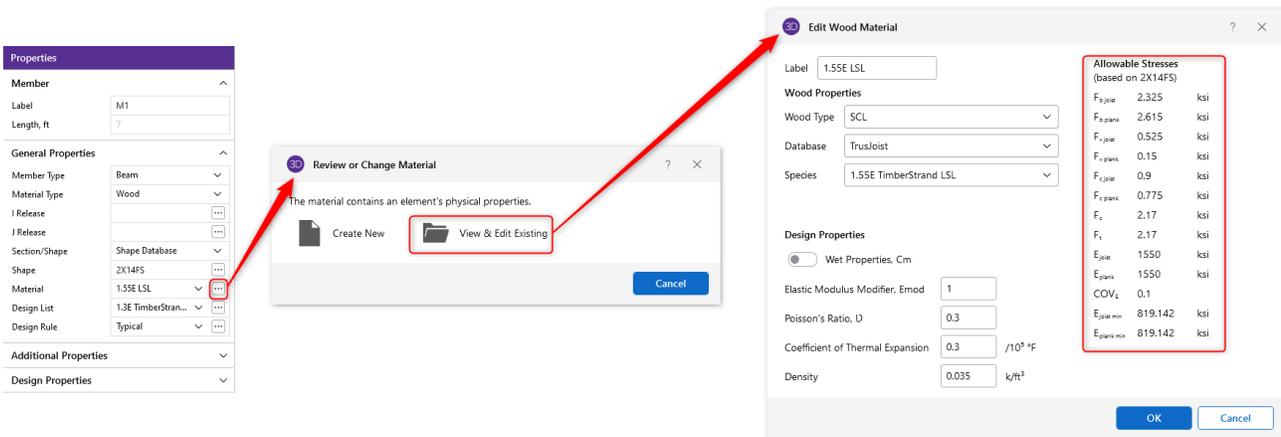
Each of the above reports includes tables with design values which were used to create the RISA-3D and RISAFloor SCL design property material databases. However, due to the difficulty of design variation, the following limitations are present:

- The Fb design values are not factored in accordance with the depth of the member. Currently RISA-3D always uses the single tabular Fb value regardless of member size.
- The Ft design values are not factored in accordance with the length or thickness of the member. Currently RISA-3D always uses the single tabular Ft value regardless of member size.
- The Fb design value will be increased by 4% if the member is designated as a repetitive member using the Cr check-box on the Wood tab of the Members spreadsheet.

Alternatively, you can always enter the appropriate design values, per the applicable footnotes, as a [Custom Wood Material](#).

Design Values

The design properties may be verified by clicking on the ellipsis next to the Material section of the specific member's information in the Property Panel.



The values are as follows:

- **Fb_joist:** Bending capacity due to joist/beam edge loading
- **Fb_plank:** Bending capacity due to plank/flat edge loading
- **Fv_joist:** Shear capacity due to joist/beam edge loading
- **Fv_plank:** Shear capacity due to plank/flat edge loading
- **Fc_joist:** Compressive (perpendicular to grain) axial capacity due to joist/beam edge loading
- **Fc_plank:** Compressive (perpendicular to grain) axial capacity due to plank/flat edge loading
- **Fc:** Compressive (parallel to grain) axial capacity, regardless of load direction
- **Ft:** Tensile axial capacity, regardless of load direction
- **E_joist:** Modulus of elasticity due to joist/beam edge loading
- **E_plank:** Modulus of elasticity due to plank/flat edge loading
- **E_mod:** Young's Modulus modifier (always set as 1.0)
- **COV_E:** Coefficient of Variation in Modulus of Elasticity per NDS Table F1. This value is used in the Emin calculation. For SCL materials COV_E always equals 0.10.
- **E_joist_min:** Emin (as calculated per NDS Appendix D equation D-4) due to joist/beam loading
- **E_plank_min:** Emin (as calculated per NDS Appendix D equation D-4) due to plank/flat edge loading

Currently the program only designs using the *Joist* properties of the SCL materials. This assumes that the loading is always on the small dimension side of the member. In a future version we will be taking the Plank values into consideration as well. In the meantime you can always enter the appropriate design values from these tables as a [Custom Wood Material](#) for use in your model.



Note: Canadian wood member design only includes SCL design for the CSA O86-14 code.

Custom Wood Sizes

If you would like to enter explicit dimensions of a member or if the member is "Not Dressed", the member must be designated as a "Full Sawn" member by choosing **Net Rectangular** as the **Shape Type** and entering the exact dimensions of the member in the boxes below. This applies to regular wood species, custom wood species, Structural Composite Lumber (SCL), and Glu-Lam members.

Wood - Design

Full code checking can be performed on Dimension Lumber and Post and Timber size wood shapes based on the following codes:

- The 2018 edition of the NDS (with 2015 and 2021 SDPWS)
- The 2015 edition of the NDS
- The 2012 edition of the NDS
- The 2005/08 edition of the NDS
- The 2001 edition of the NDS
- The 1991 / 1997 editions of the NDS
- The 2014 edition of the CSA O86 Canadian wood design code
- The 2009 edition of the CSA O86 Canadian wood design code



Note: When the 1991 / 1997 NDS is selected, the 1991 NDS specification will be used with the 1997 stress tables. This is consistent with the requirements of the 1997 UBC.

Glu-Lams

Glu-Lams are treated as any other wood species and may be selected from the list of species on the **Wood** tab of the Materials spreadsheet.

Glulam Material Properties

Available Glulam Materials are per Tables 5A and 5C of the NDS Supplement and Table 6.3 of the CSA O86. When a Glu-Lam is selected, the grade will be listed as "na" or not applicable.

If you prefer to use a material that is not listed in the design code glulam tables, please enter the material type as a [Custom Wood Species](#).

**Note:**

- Glu-Lams from Table 5A are always assumed to have the special tension laminations. Therefore, the F_{bx} value is not reduced.
- RISA is NOT applying any of the footnotes to Table 5A and 5C at this time except for the following:
 - Footnote #1 from Table 5A - For balanced materials F_{bx-} shall equal F_{bx+} for the stress class.
 - Footnote #3 from Table 5A - F_{vx} and F_{vy} are increased for Southern Pine Glulam materials.
- Commentary to CSA O86 Clause 6.2 notes that the Hem Fir glulam species listed in Table 6.3 are not common and therefore they are not included by default in RISA. If you prefer to use one of these materials, please enter it as a Custom Wood Species.

Glulam Dimensions

All Glu-Lam members should be dimensioned as "Full Sawn" using the format $wXdFS$ (or $wXdMFS$ for metric sizes), where "w" and "d" are the actual width and depth dimensions. If the size is entered as wXd without the *FS* designation, then the size will be assumed to be regular dimensional lumber.

Glulam Redesign Lists/ Optimization

RISA includes two redesign lists for Glu-Lams: Glu-Lam_Western for Western Species and Hardwoods (HW), and Glu-Lam_SouthernPine for Southern Pine (SP/SP).

Glulam Limitations

Please note that glulam design is not supported for the 91/97 NDS design code.

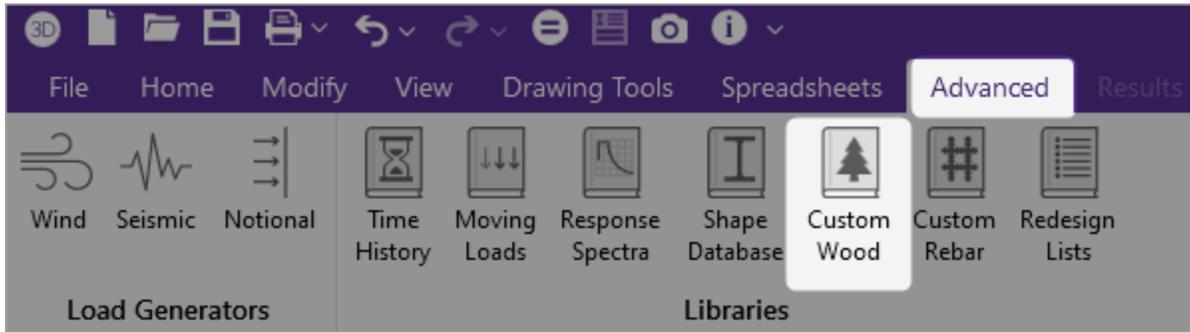
Custom Wood Materials & Structural Composite Lumber

To use a custom wood material that is not part of the standard NDS or CSA O86 databases, you need to define the custom design properties. This can be done through the **Materials** spreadsheet, or by clicking the **Custom Wood** icon on the **Advanced** ribbon.

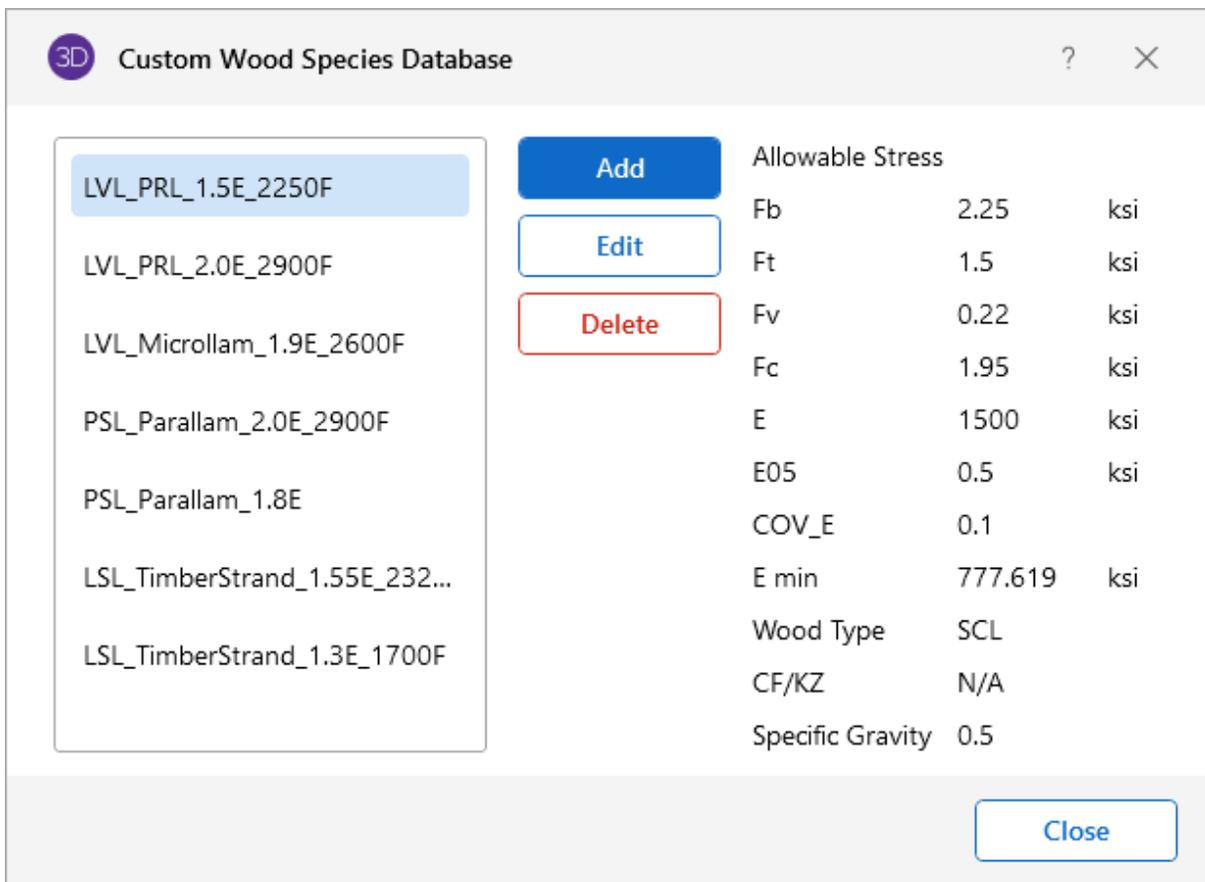
Custom Wood Properties

To access the Custom Wood Properties input window:

1. Go to the **Advanced** ribbon.
2. Click the **Custom Wood** icon.



The Custom Wood Species Database window opens.



Follow the steps below that apply to what you want to do:

Add a new custom wood

- a. Click the **Add** button to open the Add Custom Wood Species window.

3D Add Custom Wood Species ? X

Label

Custom Wood Properties

Bending Stress (Fb) ksi

Tension Stress (Ft) ksi

Shear Stress (Fv) ksi

Compressive Stress Parallel to Grain (Fc) ksi

Elastic Modulus (E) ksi

Fifth Percentile Modulus of Elasticity ksi

Structural Composite Lumber ▼

CF/KZ

Specific Gravity

OK Cancel

- b. In the **Label** box, type a name for the new custom wood.
- c. In the **Custom Wood Properties** section, enter all the applicable properties, referring to the [Allowable Stress Properties](#) table for descriptions.
- d. Click **OK** to save the new custom wood and close the Add Custom Wood Species window.

Edit an existing custom wood

- a. Scroll through the list and click on the existing custom wood you want to modify.
- b. Click the **Edit** button to open the Edit Custom Wood Species window.

3D Edit Custom Wood Species ? X

Label: LVL_PRL_1.5E_2250F

Custom Wood Properties

Bending Stress (Fb): 2.25 ksi

Tension Stress (Ft): 1.5 ksi

Shear Stress (Fv): 0.22 ksi

Compressive Stress Parallel to Grain (Fc): 1.95 ksi

Elastic Modulus (E): 1500 ksi

Fifth Percentile Modulus of Elasticity: 0.5 ksi

Structural Composite Lumber: SCL

CF/KZ:

Specific Gravity: 0.5

OK Cancel

- c. Enter the changes you want to make, referring to the [Allowable Stress Properties](#) table for descriptions.
- d. Click **OK** to save your changes to the custom wood and close the Edit Custom Wood Species window.

The following table provides descriptions for each of the Allowable Stress properties.

Allowable Stress Properties

Property	Description
Bending Stress (Fb)	Bending Stress (Fb) represents the Bending Stress limit applied to the custom wood.
Tension Stress (Ft)	Tension Stress Limit (Ft) represents the Tension Stress limit applied to the custom wood.
Shear Stress (Fv)	Shear Stress Limit (Fv) represents the Shear Stress limit applied to the custom wood.
Compression Parallel	Compression Parallel to Grain Stress (Fc) Limit represents

Property	Description
to Grain (Fc)	the Compression Parallel to Grain Stress limit applied to the custom wood.
Elastic Modulus (E)	Modulus of Elasticity (E) represents the Modulus of Elasticity applied to the custom wood.
Fifth Percentile Modulus of Elasticity	Fifth Percentile Modulus of Elasticity only applies for Canadian CSA O86 design; it is ignored for NDS design. It represents the 5th Percentile Modulus of Elasticity applied to the custom wood.
Structural Composite Lumber	Structural Composite Lumber is used to determine design variables (COV_E , E_{min} , c , etc.) and design calculations.
CF/KZ	CF/KZ represents the Wood Size Factor. The program defaults to 1.0, unless you manually enter a different value.
Specific Gravity	Specific Gravity represents the Specific Gravity applied to the custom wood.

3. Click the **Close** button to close the Custom Wood Species Database window.



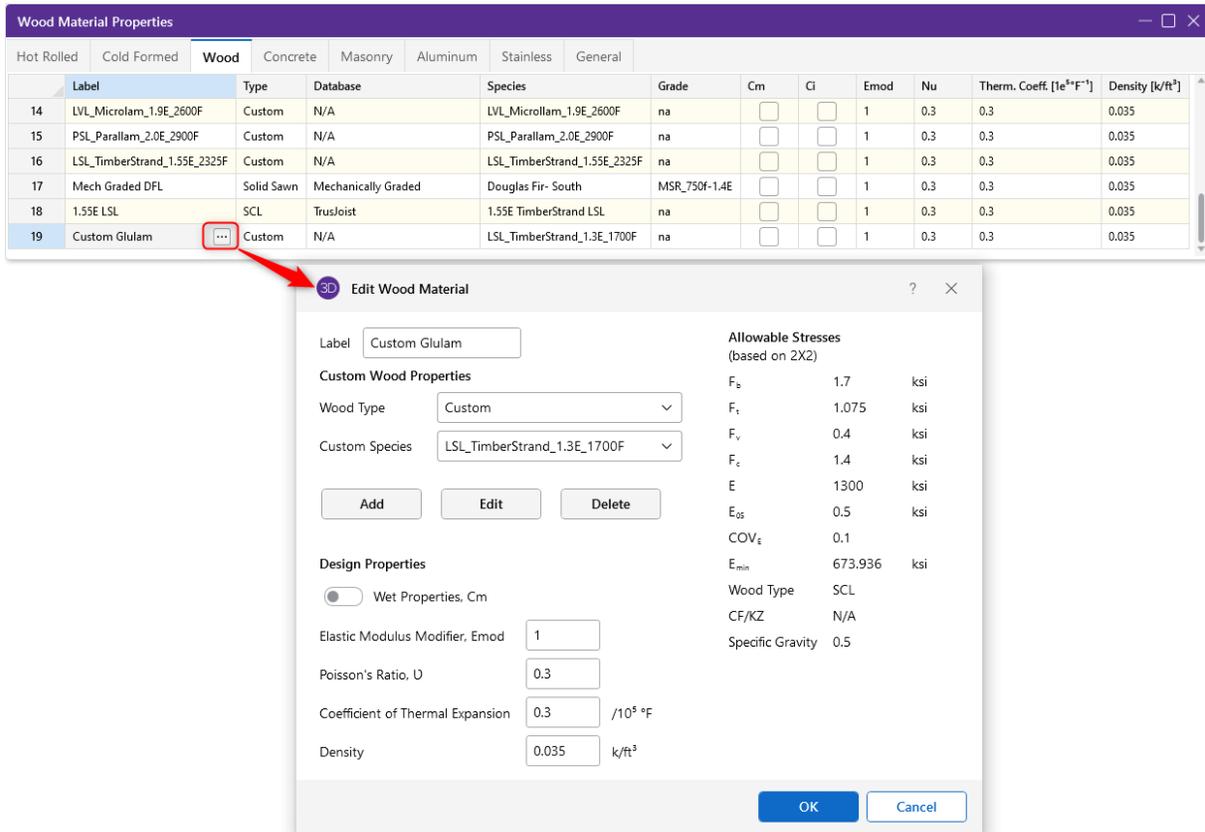
Note:

- Most wood materials (sawn lumber visually & mechanically graded, glulam, SCL, and LVL) are available in the current wood materials database. Please see the **Wood** tab of the [Materials spreadsheet](#) for more information.
- Due to the limited entries in the Custom Wood spreadsheet, Glulam members are always assumed to be balanced, uniform material:
 - F_b is taken as the F_{bx+} , F_{bx-} , and F_{by} values for glulam design.
 - F_v is taken as the F_{vx} and F_{vy} values for glulam design.
 - E is taken as the E_x and E_y values for glulam design.
 - Glulams will always use the CV volume factor equation per NDS Table 5C (this is the same equation as Tables 5A, 5B, and 5D assuming that the material is not Southern Pine).

Create and Apply a Custom Wood Material

To create and apply custom material:

1. Open the **Materials** spreadsheet.
2. Click on the **Wood** tab.



3. Select the last row in the spreadsheet and press **Enter** to create a new entry.
4. In the **Label** box, type a name for the new custom property.
5. Click on the **Wood Type** arrow and choose **Custom** from the list.
6. Do one of the following to define the custom species of the property you are adding.
 - Click on the **Species** arrow and choose from the existing custom species to include. The list of available species is read in from the Custom Wood Properties window. Note that you can click the 'Edit' button to open the Edit Custom Wood Species window in which you can modify the properties of this species.
 - Click the **Add** button to open the Add Custom Species window in which you can add a new custom species.
7. (Optional) Use the **Design Properties** options to change any of the default design properties for the custom wood you are adding.

Note that the Elastic Modulus Modifier Emod is a factor that is applied to the Young's modulus modifier to reflect the NDS Appendix F criteria. This is not applicable to the CSA O86 design code.
8. Click **OK** to apply the new custom wood material and close the Edit Wood Material window.



Note: The **Type** menu is used to designate whether the entry is a “Visually Graded” sawn lumber, a “MSR” mechanically graded lumber, a “MEL” mechanically graded lumber, a “Glulam” material, or a “SCL” (structural composite lumber). This selection effects the design factors and the applicable provisions of the selected design code.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Custom Wood Species**.

Wood Design Parameters

The **Wood Design Parameters** spreadsheet records the design parameters for the timber code checks and may be accessed by selecting **Members** from the **Data Entry** toolbar on the **Explorer** panel and then clicking on the **Wood** tab. These parameters may also be assigned graphically. See [Modifying Member Design](#) to learn how to do this.

Wood Design Parameters													
Primary	Advanced	Hot Rolled	Cold Formed	Wood	Concrete Beam	Concrete Column	Aluminum	Stainless	RISACONNECTION				
	Label	Shape	Length [ft]	le2 [ft]	le1 [ft]	le-bend top [ft]	le-bend bot [ft]	K y-y	K z-z	CV	Cr	y sway	z sway
1	M1	2X14FS	7		Segment	Lbyy					<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
2	M2	2X14FS	7	2	Segment	0.5					<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
3	M3	2X14FS	7			Lbyy					<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
4	M4	2X14FS	7	2		0.5					<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
5	M5	2X14FS	7			Lbyy					<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

These parameters are defined for each member.

Wood Design Parameters

Parameter	Description
Label	The Label column lets you assign a unique Label to all of the members. Each label must be unique, so if you try to enter the same label more than once you get an error message. You can relabel at any time by right clicking in the spreadsheet and choosing either Relabel All Members or Relabel Selected Members .
Shape	The member Shape (or Section Set) column value is listed for reference only and can not be edited, as it is dictated by the entry in the 'Section/Shape' column on the <i>Primary</i> tab.
Length	The member Length column lists the length of the material. This value can not be edited, as it is dependent on the member end coordinates listed on the <i>Primary Data</i> tab. Length is listed here only as a reference for unbraced lengths, which are discussed in the next section.
Unbraced Length	See the Unbraced Lengths topic.
K Factors (Effective Length Factors)	See the Unbraced Lengths topic.

Parameter	Description
Adjustment Factors	Please see below for information about the various wood adjustment factors.
Sway Flags	See the Unbraced Lengths topic.

Timber Design Adjustment Factors

The Timber Design Adjustment Factors (a.k.a. NDS Adjustment Factor) design codes have a number of adjustment factors that are applied to the various allowable stresses to determine the capacity of the member. The adjustment factors are summarized in section 2.3 of the code. The following topics help to summarize how adjustment factors are obtained and used.



Note:

- C_b (Bearing Area Factor), C_T (Buckling Stiffness Factor), C_c (Curvature Factor), C_i (Stress Interaction Factor), and C_{vr} (Shear Reduction Factor) are NOT used in the RISA analysis.
- If you are using the Canadian CSA O86 design code, see the section below for the [CSA adjustment factors](#).

Timber Design CD (Load Duration Factor- NDS)

CD is the Load Duration adjustment factor used for ASD codes. It is entered on the [Load Combinations](#) spreadsheet for each load combination for which you want wood code check results. The CD factor must be entered for each individual load combination because the CD factor is dependent on the types of loads that are applied in each load combination. Therefore, different load combinations could have different CD factors. For example, per the NDS 2018 specification, a load combination that had only dead load, would have a CD factor of “0.9”, while another combination that was comprised of dead load plus wind load would have a CD factor of “1.6”.

The CD factor will only be applied to wood code checks on wood members. See Table 2.3.2 in the NDS 2018 specification for the CD factors to be applied for typical loads. Appendix B has additional information about the Load Duration Factor.



Note: The CD factor used for a load combination should be for the load with the shortest load duration in that load combination.

Timber Design C_m (Wet Service Factor)

C_m is the Wet Service adjustment factor. It is applied when you check the **C_m** checkbox in the [Materials Spreadsheet](#).

Timber Design Ci (Incision Factor- NDS)

Ci is the Incision factor per Table 4.3.8 of NDS code. It is applied when you check the **Ci** checkbox in the [Materials Spreadsheet](#).

Timber Design Ct (Temperature Factor)

Ct is the Temperature adjustment factor. It is calculated internally based on the wood **Temperature** value set on the **Codes** tab of [Model Settings](#). See section 2.3.3 of the NDS 2018 for more information on this factor.

Timber Design CP and CL (Column/Beam Stability Factors - NDS)

The Column Stability Factor, **CP**, and the Beam Stability Factor, **CL**, are calculated internally. These calculated values are shown on the **Wood** tab of the **Code Check Spreadsheet**, as well as in the **Member Detail Reports**. See NDS 2018 section 3.3.3 for information on the CL factor and NDS 2018 section 3.7.1 for information on the CP factor.

The value of Emin used for the calculation of these factors is calculated using equation D-4 from appendix D of the 2018 NDS. For some members (especially for glulams) this equation may produce a slightly more accurate value of Emin that shown in the NDS tables.



Note: The column stability factor, CP, is affected by the Kf factor from NDS 2018 section 15.3.2, depending on whether multi-ply members are bolted or nailed together. Bolted columns will have a shape name with a **B** after. See the [Wood-Database](#) topic for information on how to define bolted vs nailed multi-ply members.

Timber Design CF (Size Factor- NDS)

CF is the Size adjustment factor. It is applied automatically when you assign a wood shape from the NDS shape database. See Tables 4A, 4B, 4D, and 4E in the NDS supplement for information on the CF factor.



Note: Prior to the 2012 NDS, Table 4D (Reference Values for Timbers) combined the Size Factor(CF) and the Flat Use Factor (Cfu) under the Size Factor (CF) description. Per the code commentary, you are required to apply both the CF per the equation in Table 4D and the CF per the tabular values. To clarify between the two, the program calls the calculated value (per the equation) CF and the tabular value Cfu. This is verified with clarification in the NDS 2012 design code.

Timber Design CV (Volume Factor)

CV is the Volume adjustment factor. It is applied automatically when you assign a glulam or SCL material member. The user can override the calculated value by manually entering the factor on the **Wood** tab of the [Members Spreadsheet](#).

**Note:**

- In the calculation of CV, RISA takes L as the full length of the member. This is a conservative assumption.
- This factor is not available for the NDS 1991/1997 design option.

Timber Design Cfu (Flat Use Factor)

Cfu is the Flat Use adjustment factor. is automatically applied to the weak axis allowable bending stress of a wood member whenever weak axis moments are present. See the tables in the NDS supplement for more information on this factor.



Note: Prior to the 2012 NDS, Table 4D (Reference Values for Timbers) combined the Size Factor(CF) and the Flat Use Factor (Cfu) under the Size Factor (CF) description. Per the code commentary, you are required to apply both the CF per the equation in Table 4D and the CF per the tabular values. To clarify between the two, the program calls the calculated value (per the equation) CF and the tabular value Cfu. This is verified with clarification in the NDS 2012 design code.

Timber Design Cr (Repetitive Factor)

Cr is the Repetitive Member adjustment factor. This factor specifies if the beam is one of a group of repetitive members. This design parameter can be set on the **Wood** tab of the [Members Spreadsheet](#). If you put a check in the Cr field, a factor of 1.15 will be applied to beam members that are 2" to 4" thick. See the section 4.3.9 of the NDS 2018 for information on this factor.

**Note:**

- This flag will be ignored for a NDS shape that is thicker than 4".
- A value of '1.0' will be used for Wood Products.
- Different restrictions apply to the use of the Cr factor for Structural Composite Lumber and Glu-Lams.

Timber Design CH (Shear Stress Factor)

CH is the Shear Stress adjustment factor. This design parameter can be set on the **Wood** tab of the [Members Spreadsheet](#). If left blank the program will use a default value of 1.0. See the tables in the NDS supplement for information on other CH factors.

**Note:**

- The CH factor is only available for the 1991/1997 NDS option. For other codes, this entry will be ignored.
- Only tables 4A, 4B, and 4D are used.

Timber Design Cf (Form Factor)

Cf is the Form adjustment factor. It is applied automatically when designing by the NDS 91/97 or 2001 Specification and a 'Round' shape is selected from the NDS shape database. See section 2.3.8 in the NDS (91/97, 2001) for information on the Cf factor.

**Note:**

- This factor is not applied when designing with the NDS 2005 or 2012 specifications.
- 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 Kf (Format Conversation Factor)

Kf is the format conversion factor for LRFD design only. The tabulated reference design values provided in the NDS Supplement contain safety adjustments appropriate for ASD. The Kf factor converts these values to nominal design values for LRFD. These factors are provided in NDS Table 4.3.1 and Appendix N.

Timber Design Phi (Resistance Factor)

Phi is the resistance factor for LRFD design only. These values are provided in NDS Table 4.3.1 and are dependent on the property ranging from 0.75 for shear and 0.90 for compression.

Timber Design lambda (Time Effect Factor- NDS)

Lambda is the Time Effect adjustment factor used for LRFD codes. It is entered on the [Load Combinations](#) spreadsheet for each load combination for which you want wood code check results. The lambda factor must be entered for each individual load combination because the lambda

factor is dependent on the combination of loads. Therefore, different load combinations could have different lambda factors. For example, per the NDS 2018 specification, a load combination that had only dead load, would have a CD factor of “0.6”, while another combination that was comprised of dead load plus wind load would have a CD factor of “1.0”.

The lambda factor will only be applied to wood code checks on wood members. See Table N3 in the NDS 2018 specification for the lambda factors to be applied for typical load combinations.

CSA O86 Adjustment Factors

The CSA O86 design code has a number of adjustment factors that are applied to the various allowable stresses to determine the capacity of the member. The adjustment factors are summarized in clause 4.3 of the code. The following topics help to summarize how adjustment factors are obtained and used.



Note: KB (Length of Bearing Factor), KE (End Fixity Factor), KF (Foundation Factor for plywood), KM (Bending Capacity Modification Factor), KN (Notch Factor), KR (Radial Stress Factor), KT (Treatment Factor), and KX (Curvature Factor) are NOT used in the RISA analysis.

Timber Design KD (Load Duration Factor - CSA)

KD is the Load Duration adjustment factor. It is entered on the [Load Combinations](#) spreadsheet for each load combination for which you want wood code check results. The KD factor must be entered for each individual load combination because the factor is dependent on the types of loads that are applied in each load combination. Therefore, different load combinations could have different KD factors. For example, per the CSA O86 -2009 specification, a load combination that had only dead load, would have a KD factor of 0.65, while another combination that was comprised of dead load plus wind load would have a KD factor of 1.15.

The KD factor will only be applied to wood code checks on wood members. See Table 5.3.2.2 in the CSA O86 - 2014 specification for the KD factors to be applied for typical loads.



Note: The KD factor used for a load combination should be for the load with the shortest load duration in that load combination.

Timber Design Ks (Service Condition Factor)

Ks is the Service Condition adjustment factor. It is applied when you check the **Ks** check-box in the [Materials Spreadsheet](#). See clause 6.4.2 (sawn lumber) or clause 7.4.2 (glulams) in the CSA O86-14 for more information on this factor.

Timber Design CV (Shear Load Coefficient-CSA)

CV is the Shear Load coefficient for glulam members. It is applied automatically when you assign a material from the **CSA Table 7.3** glulam material database. By default this value will always be taken as 1.0. However, the user can override this value by manually entering the factor on the **Wood** tab of the **Members Spreadsheet**.

Timber Design KH (System Factor)

KH is the System adjustment factor. This factor depends on the System Factor selection applied to the member on the **Wood** tab of the **Members** spreadsheet:

- **None** - Assumes that the member is NOT in a system of repetitive members and therefore $KH = 1.0$.
- **Sheathed** - Assumes that the member is in a system of repetitive members and the members are sheathed with plywood. This is defined as "Case 2" in clause 6.4.4.2 for sawn lumber. In this case KH will come from Table 6.4.4 (2014 code).
- **UnSheathed** - Assumes that the member is in a system of repetitive members but the members are NOT sheathed with plywood. This is defined as "Case 1" in clause 6.4.4.1 for sawn lumber. In this case KH will come from Table 6.4.4 (2014 code).



Note:

- If the member is a glulam, selecting either "Sheathed" or "Unsheathed" will apply the KH factors per clause 7.4.3. If "None" is selected than KH will be taken as just 1.0.
- Always assumed 1.0 for wood wall design per the CSA O86 design codes.

Timber Design KZ (Size Factor- CSA)

KZ is the Size factor. It is applied automatically when you assign a wood shape from the CSA shape database. See Table 6.4.5 in the *CSA O86-14* design code for information on this factor.

Timber Design KL (Lateral Stability Factor - CSA)

KL is the Lateral Stability factor. This factor is calculated internally per the equation given in clause 7.5.6.4.4 for both glulam and full sawn members.

The final calculated values of both CB and KL are shown on the **Wood** tab of the **Code Check Spreadsheet**, as well as in the **Member Detail Reports**.



Note: Clause 6.5.4.2.1 allows full sawn members to use the glulam clause 7.5.6.4 to determine KL.

Timber Design KC (Slenderness Factor)

KC is the Slenderness factor. This factor is calculated internally per the equation given in clause 6.5.6.2.4 for full sawn members and per clause 7.5.8.5 for glulam members.



Note: The program will use the maximum C_c value (per clause 6.5.6.2.2 or 7.5.8.2) in the KC calculation.

Timber Design Flat Use Factor -CSA

The Flat Use factor is just called "**Flat Use**" in the member detail report. There is no explicit factor for this defined in the *CSA O86-14* design code. However, there is a note for Table 6.3.1C (*Material Strengths for Beams and Stringers*) that includes a flat use adjustment factor. The program will determine this factor based on the asterisk table under Table 6.3.1C.

Limitations

Wood Design Limitations

- It is assumed that the load on the member is occurring through the member's shear center. This means secondary torsional moments that may occur if the load is not applied through the shear center are not considered.
- Currently the program only considers the Joist loaded edge design values for SCL members per NDS design.
- CSA O86 wood design will not automatically account for the E_s value (per clause 5.4.1). If the user wants to account for this modified modulus of elasticity, then you must manually adjust the material properties.

Adjustment Factor Limitations

- **Buckling Stiffness Factor** - The NDS buckling stiffness factor, C_T , is not currently accounted for.
- **Bearing Area Factor** - The NDS bearing area factor, C_b , is not currently accounted for.
- **Curvature Factor** - The NDS curvature factor for glulams, C_c , is not currently accounted for.
- **Stress Interaction Factor** - The NDS stress interaction factor for glulams, C_I , is not currently accounted for.
- **Shear Reduction Factor** - The NDS shear reduction factor for glulams, C_{vr} , is not currently accounted for.

- **Length of Bearing Factor** - The CSA O86 length of bearing factor, KB, is not currently accounted for.
- **End Fixity Factor** - The CSA O86 end fixity factor, KE, is not currently accounted for.
- **Foundation Factor** - The CSA O86 foundation factor (for plywood), KF, is not currently accounted for.
- **Bending Capacity Modification Factor** - The CSA O86 bending capacity modification factor, KM, is not currently accounted for.
- **Notch Factor** - The CSA O86 notch factor, KN, is not currently accounted for.
- **Shear Load Coefficient** - The CSA O86 shear load coefficient for glulams, CV, will always default to 1.0. The user can override this by manually typing in the appropriate **CV** value on the **Wood** tab of the **Members** spreadsheet.
- **Radial Stress Factor** - The CSA O86 radial stress factor, KR, is not currently accounted for.
- **Treatment Factor** - The CSA O86 treatment factor, KT, is not currently accounted for.
- **Curvature Factor** - The CSA O86 curvature factor for glulams, KX, is not currently accounted for.

NDS Emin Calculation

RISA will calculate the Emin value for NDS wood materials rather than read it in from the design tables. In the 2018 edition of the NDS, Emin is calculated per equation (D-4) from Appendix D.

$$E_{min} = E * (1 - 1.645 * COV_E) * \left(\frac{1.03}{1.66} \right)$$

COV_E (the coefficient of variation in modulus of elasticity) comes from Table F1 in Appendix F.



Note: If your member is a glulam or SCL material, the Emin equation becomes:

$$E_{min} = E * (1 - 1.645 * COV_E) * \left(\frac{1.05}{1.66} \right)$$

Wood Code Check Results

Access the **Wood Code Checks Spreadsheet** by selecting the **Code Check** spreadsheet from the **Results** toolbar on the **Explorer** Panel and then clicking the **Wood** tab.

Member Wood Code Checks (By Combination)																	
Hot Rolled Steel		Cold Formed Steel		Wood		Aluminum		Stainless		Concrete							
	LC	Member	Shape	UC Max	Loc[ft]	Shear UC	Loc[ft]	Dir	Fc' [ksi]	Ft' [ksi]	Fb1' [ksi]	Fb2' [ksi]	Fv' [ksi]	RB	CL	CP	Eqn
1	1	M1	2X14FS	0.005	7	0.002	7	y	0.374	1.6	2.136	2.325	0.525	17.146	0.919	0.172	3.6.3
2	1	M2	2X14FS	0.003	0	0.002	7	y	2.017	1.6	2.136	2.325	0.525	17.146	0.919	0.929	3.9-1
3	1	M3	2X14FS	0.005	7	0.002	7	y	0.374	1.6	2.136	2.325	0.525	17.146	0.919	0.172	3.6.3
4	1	M4	2X14FS	0.003	0	0.002	7	y	2.017	1.6	2.136	2.325	0.525	17.146	0.919	0.929	3.9-1
5	1	M5	2X14FS	0.005	7	0.002	7	y	0.374	1.6	2.136	2.325	0.525	17.146	0.919	0.172	3.6.3

Code Checks

The final result of the design solution are the code check values (ratios of actual stress to allowable stress). So, if these values are less than 1.0, the member passes. If they are greater than 1.0, the member fails.



Note:

- If the value is greater than 9.999 it will be listed as "9.999".
- The [Member Detail Report](#) gives a more detailed view of the values used to perform the code check.
- See [Results View Settings - Members](#) to learn how to view the code check results graphically.

The **UC Max** value represents the combined bending and axial force stresses. The governing equation that was used to calculate the UC Max value is listed at the far right of the spreadsheet in the **Eqn** column.

The **Shear UC** is the maximum ratio of actual to allowable shear stress.

The **Loc** fields that are to the right of the code check fields tells at what location the maximum code check occurs measured from the I-joint location of the member.

The **Dir** field tells us in which local member direction the maximum shear is occurring (y or z).

Capacity Values

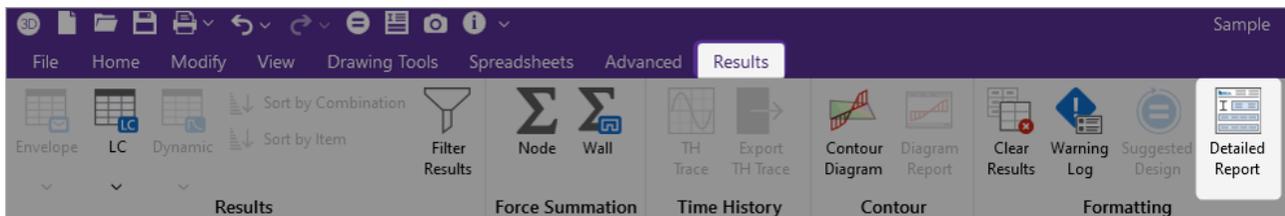
The values (Fc', Ft', Fb1', Fb2', Fv') are the factored allowable stresses per NDS wood member design.

- 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 2018 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.

The values (Pr, Tr, Mr, Vr/Wr) are the allowable forces per CSA O86 wood member design. These will only be visible when you have selected the **CSA O86-09: Ultimate** or **CSA O86-14: Ultimate** as your wood design code in [.Model Settings](#).

- Reference equations are reported in the member detail report for the calculation of each capacity value.
- The **Vr/Wr** shear resistance value for glulams depends on the [CV](#) value. Unless manually entered by the user, this value is always assumed 1.0.
 - This value is called "Vr" in the 2009 version of the code and "Wr" in the 2014 version of the code.
- The **Pr**, axial compressive resistance for sawn lumber members will have a 75% reduction taken into account (per clause 6.5.6.4.3) for bolted multi-ply members.
- The **Tr**, axial tensile resistance for glulams only considers the gross area of the member when calculated per clause 7.5.11.
- The glulam material Table 7.3 in the CSA O86-09 does not include weak axis bending or shear values. Per the commentary to clause 7.5, glulams will use the sawn lumber values for the glulams species' grade No. 2 listing in Table 6.3.1A.

Finally, the **Equation** controlling the code check is listed. For NDS wood member design, this will be either Eqn. 3.9-1 or 3.9-3. Eqn. 3.9-2 is not checked since this equation includes the tension stress in a beneficial (non-conservative) manner. All other requirements in Section 3.9 are also checked, such as $f_c < F_c E_1$, 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 by right clicking in a spreadsheet row and clicking **Detailed Report** or by clicking the **Detailed Report** button on the **Results** ribbon.



Note:

- For Enveloped results the combination that produced the listed code and shear checks is given in the "LC" column. The other values are the corresponding values and are not necessarily the maximums across all the combinations.
- Moving Load results are enveloped and the governing load combination and step location is shown for each result value under the "LC" column. The first number is the load combination, the second is the step number: (load combination - step number). See [Moving Loads](#) to learn more.

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:

Code Check Not Calculated

This is the general message displayed when code checks were not performed for a member. It could mean that you have not selected a Design Code in [Model Settings](#), or you have not included any load combinations for this material type in your solution. Check the **Design** tab of the **Load Combinations** spreadsheet.

RB value is greater than 50

Section 3.3.3.7 of the NDS code limits the slenderness ratio RB to a maximum of 50. Similarly, clause 7.5.6.4.3 of the CSA O86 limits CB to a maximum of 50. You will 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 code limits the column slenderness ratio of L_e1/b or L_e2/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 3.9.2 of the NDS 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)

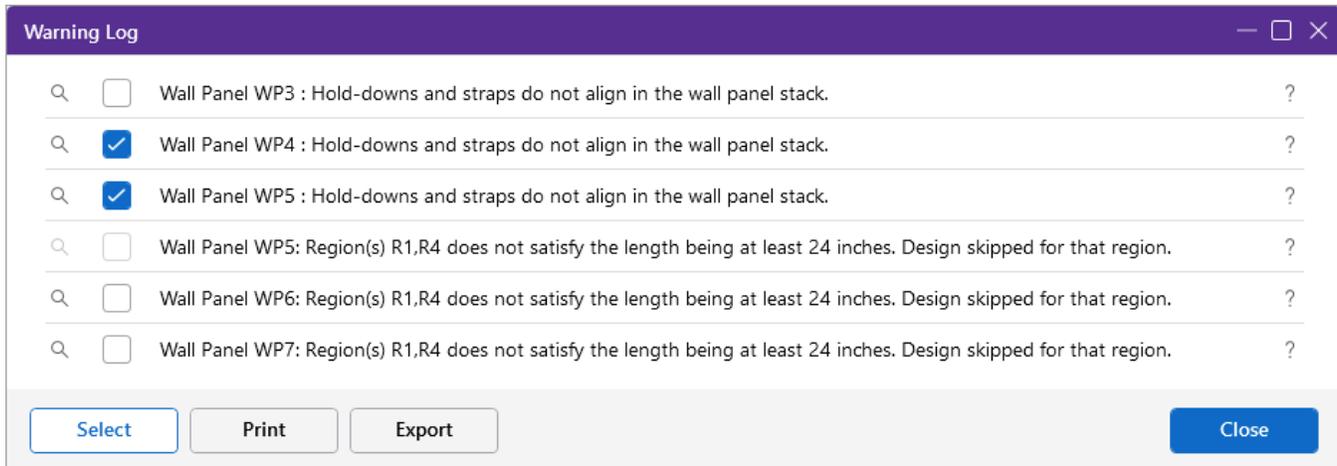
fc is greater than FcE2

Section 3.9.2 of the NDS 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

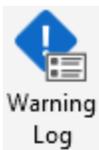
Section 3.9.2 of the NDS code limits the actual strong axis bending compressive stress to be less than the term FbE. This term is approximately the lateral buckling stress.

Warning Log



The **Warning Log Spreadsheet** provides you with a record of any warnings or errors that occurred during the solution of your model. The log should be reviewed for warnings or errors that would affect the design of your structure. The error log reports back the item label for which the error occurred.

- Click the **Help** button to jump to the specific warning within the Warning Log section of the Help File.
- Click the **Find** button to zoom into the specific element reported in the warning.
- Use the **Select** checkbox and Select button at the bottom to select the elements reported for those particular warnings.
- You can view the warning log by clicking on the Spreadsheets menu item and then clicking on the Warning Log selection.
- Click the Warning Log hyperlink to open the help that provides further explanation on this item.



The following sections include some of the common warnings which might require additional explanation.

Sum of reactions is not equal to the sum of the loads (LC xx)! Check for any small rigid links or fixed boundary conditions.

When the solution is complete, the program checks the sum of the reaction forces in each direction if this is more than 0.1% different from the sum of the applied loads, then this warning message will be

displayed. The most common causes for this warning message are the following:

- A joint instability which has been automatically LOCKED by the program. If the joint is locked then the reaction at that location is not computed. Once you rectify the instability then this warning log message should also go away. You may also need to uncheck the "[Lock isolated ROTATIONAL instabilities without notification](#)" box as this may conceal some instabilities. See the [Stability](#) section for more information.
- A user assigned boundary condition which was used the "Fixed, reaction will not be calculated" option rather than the "Reaction" option. See the [Boundary Conditions](#) section for more information. Using the term "Fixed" suppresses the reaction output in the Joint Reactions and thus the applied loads does not equal joint reactions. In this case the Warning Log could likely be ignored, as there is a valid reason for this discrepancy.
- A "ghost reaction" may have developed where one or more rigid elements (Diaphragms, rigid end offsets, or rigid links) have become so stiff that they actually became stiffer than the internal stiffness used by the program to define boundary conditions. In this rare case forces may be leaving the model at locations other than boundary conditions. Having a combination of a rigid diaphragm, rigid links, rigid end offsets, top of member offsets, etc., all in a localized place in the model could cause these.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Sum of Reactions**.

*Please refer to the **Help File** in RISA-3D for detailed information about each error message.*

*The Help File may be accessed by selecting  **The RISA Help** in the upper right-hand corner. Choose the **Warning Log** topic*

Appendix A – Redesign Lists

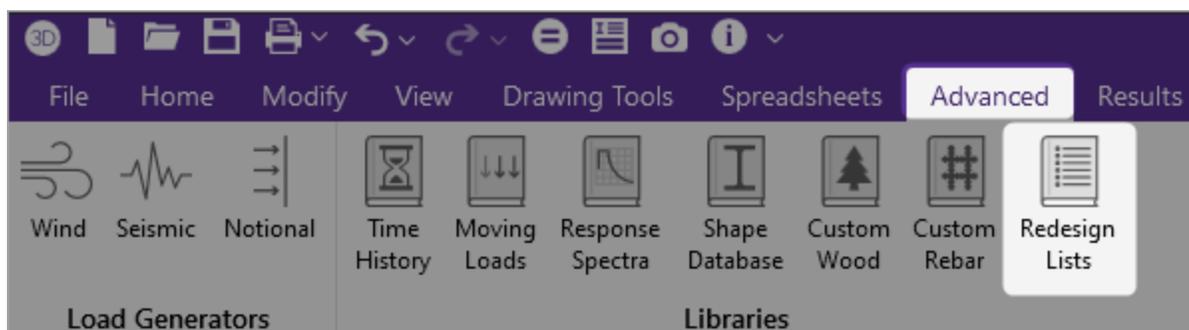
RISA-3D has Redesign Lists that are used to optimize hot-rolled steel, cold-formed steel, dimensional lumber, and concrete beams and columns. Although, the criteria used for this optimization is the **Design Rules**, the sizes must be chosen from the available sizes in the Redesign List.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Redesign**.

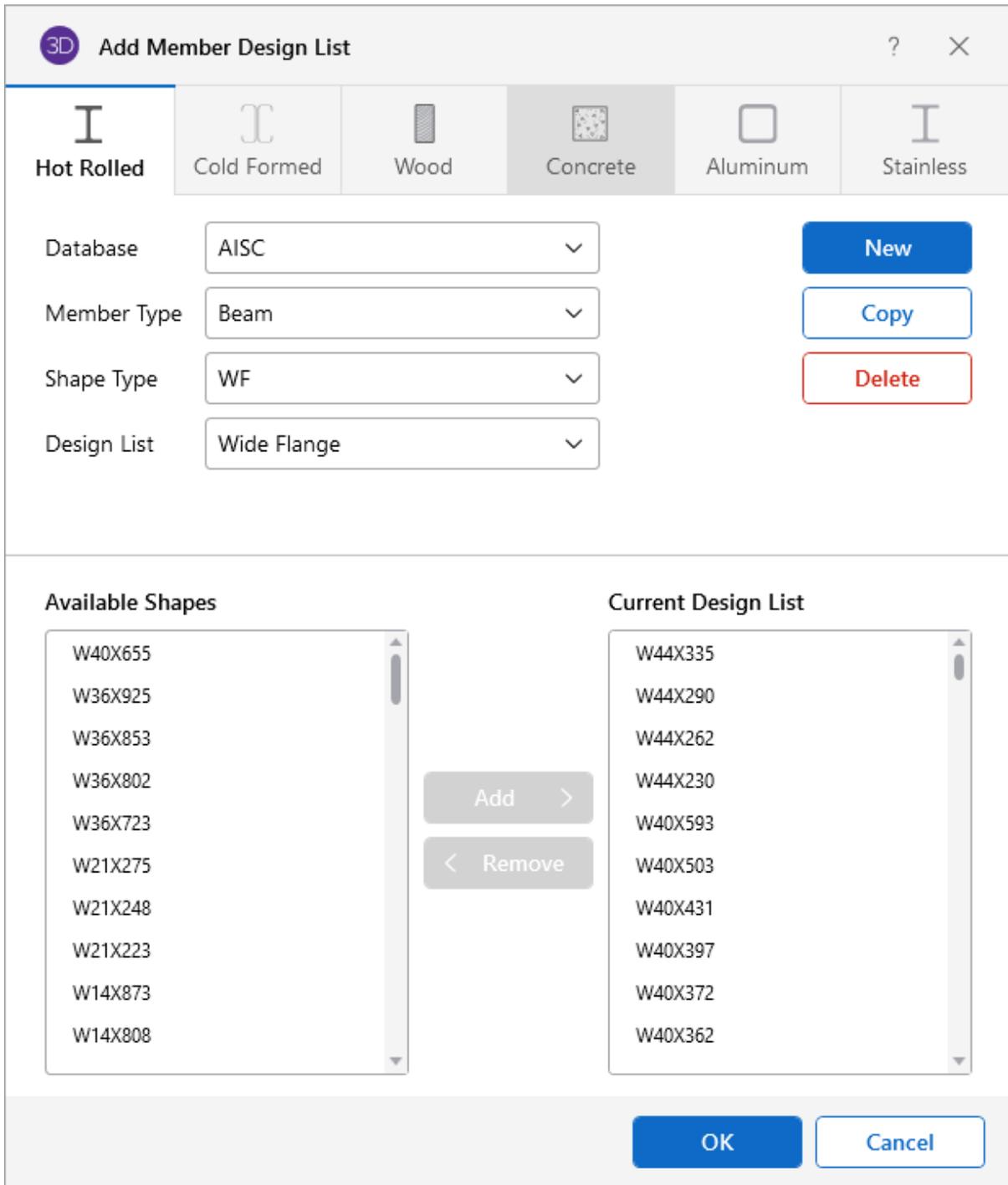
Creating or Modifying Redesign Lists

To create or modify a Redesign List:

1. Click on the **Advanced** ribbon tab to open it.



2. Click the **Redesign Lists** icon.
The Add member Design List window opens.



3. Select a material using the tabs at the top of the dialog.
For each material, select a Database, Member Type, Shape Type, and Design List.
4. Do one of the following, depending on what you want to do.

- Click **New** to create a new Redesign List for the selected Member Type and Shape Type.
- Click **Delete** to permanently delete the currently selected Redesign List.
- Click **Copy** to copy and rename the currently selected Redesign List.

The **Available Shapes** on the left are listed based on the selected Database, Member Type, and Shape Type.

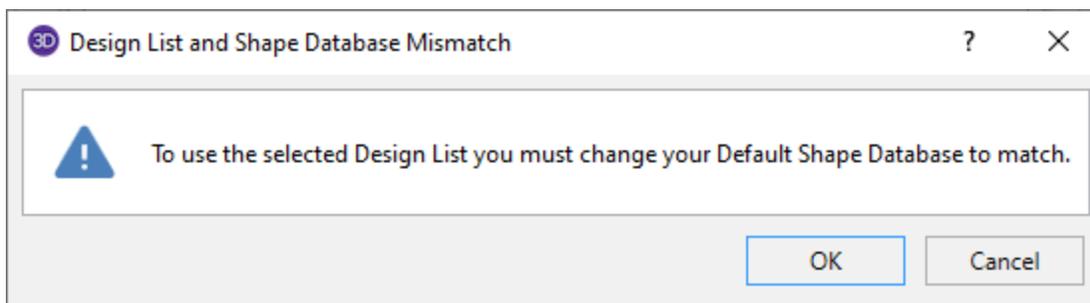
The **Current Design List** on the right lists all the shapes in the selected Design List.

5. Select shapes in the **Available Shapes** or **Current Design List**.
6. Use the **Add** or **Remove** buttons to modify the Current Design List.
7. Use the **Move Up** and **Move Down** buttons to re-order the shapes in the Current Design List.
8. Click **OK** to save your changes .

These design lists can then be assigned to a specific member or Section Set.

Design List Mismatch

If you see a Design List and Shape Database Mismatch dialog, then there is a mismatch between your default shape database and the design list you have currently selected. Either choose a Design List to match the default Shape Database, or change your default Shape Database so that you can use the Design List you want. To check your default Shape Database, open the Shape Database which is located on the Advanced tab of the ribbon toolbar. In the Shape Database, your default databases will currently be selected for each material.



Locating the Design Lists

By default the redesign list files exist in the Redesign Lists folder in the user's C:\RISA User Data\[USER NAME]\Redesign Lists. However, this folder can be changed by selecting the **File** button ▶ Application Settings ▶ [File Locations](#).

File Naming Convention for the Design Lists

The design lists must be ASCII text files with a file extension of ASC. In addition, the first 3 characters of the Design indicate which Region the lists will be available for. The region can be changed by

selecting the RISA-3D button ▶ Application Settings ▶ General ▶ [Advanced](#). Each region corresponds to the following file name prefix.

Region	File Prefix
United States	US_
Canada	CA_
Britain	BS_
Europe	EU_
India	IN_
Australia	AU_
New Zealand	AU_
Mexico / South America	MX_

File Format for Redesign Lists

The following is an example of a user defined redesign list. You can view these files using any text editor such as Window's Notepad. However, this file must be saved in the `risa__redesign_lists` folder as an ASCII text file with a file extension `ASC`. If the file is saved with a `txt` extension, it will not be read into RISA-3D. The various entries for the input fields are described below:

The first field `[RISA_REDESIGN_LIST]` identifies this as a RISA Redesign list. If this line is not present the file will not be recognized by the program.

The second field `[NAME]` identifies that the next entry will be the name of the Shape Group that will identify this list in the program. In this case, this Shape Group is given the name *Available W24s*.

The next field `[MATERIAL_TYPE]` identifies that the next entry will be the material type that this shape group belongs to. This material type must be given one of the following designations: Hot Rolled Steel, Cold Formed Steel, Wood, Steel Products, or Wood Products.

The next field `[MEMBER_TYPE]` identifies that the next entry will be the designation of the member type. This entry must be designated as either a Beam or Column.

The next field `[ASSOCIATED DATABASE]` identifies that the next entry will be the RISA database name in which the entered shapes are defined.

The next field `[UNITS]` identifies the units used to define the member sizes in the redesign list. Entries may be Inches, Millimeters, or Centimeters. This field is only applicable to files similar to the concrete and wood lists which have the dimensions built right into the list. For files which are based on database shape names, this field will have no effect.

The next field `[END HEADER]` identifies the end of the descriptive information and the beginning of the shape list.

The next field [AVAILABLE_SHAPES] identifies that the next series of lines will signify all of the shapes available for this Shape Group. The end of this field is reached when the [SHAPE_END] entry is read. The file below uses an asterisk (*) to specify that W24x68's and W24x76's are NOT available for this particular job.

The next field [OPTIMUM_SHAPES] identifies that the next series of lines will signify the preferred shapes for this Shape Group. The program will attempt to only use optimum shapes for beams that are significantly braced. By significantly braced, we mean that the unbraced length of the top flange is less than or equal to three feet. The end of this field is reached when the [SHAPE_END] entry is read.

The last field [END] identifies the end of the file.

[NAME]

Available W24s

[MATERIAL_TYPE]

Hot Rolled Steel

[MEMBER_TYPE]

Beam

[ASSOCIATED_DATABASE]

AISC

[END_HEADER]

[AVAILABLE_SHAPES]

W24X104

W24X103

W24X94

W24X84

*W24X76

*W24X68

W24X62

W24X55

[SHAPE_END]

[OPTIMUM_SHAPES]

W24X55

W24X62

*W24X68

W24X76

[SHAPE_END]

[END]

Manual Modification

By default, RISA installs a series of default redesign lists for each material and member type. The default lists are assumed to be a compilation of what engineers would typically choose. To view the available sections for each list, you can open up the file located in the Redesign Lists folder using a word editing program.

To modify an existing redesign list, open the file in a word editing program to override any of the variables. The same formatting as the original file should be followed as well as saving the file to the same location, otherwise the program will not recognize the changes.

To create a new redesign list, open an existing file and run a Save As to save the file with a different file name in the same location. The [NAME] input as described in the above section will be the label you will see within the program so it is important to give your design list a unique name. The [MATERIAL_TYPE], [MEMBER_TYPE], and [ASSOCIATED DATABASE] are important criteria your member must have in order for the design list to be available. If a member in the model includes all three of the criteria defined in a particular design list, this design list will be available within Members spreadsheet Design List pull down box (see [Design Lists](#) for more information).



Note: If you want to list a member size in the redesign list file but not have it considered as part of the redesign list, you can put an asterisk in front of the shape name.

Appendix A – Redesign Lists

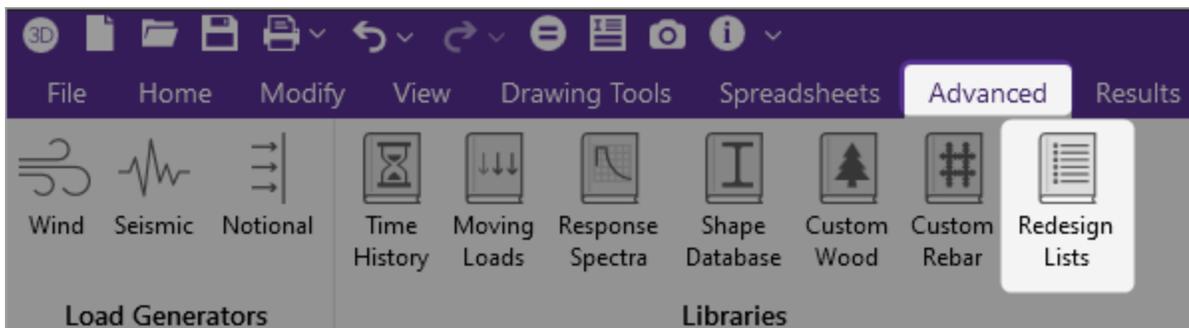
RISA-3D has Redesign Lists that are used to optimize hot-rolled steel, cold-formed steel, dimensional lumber, and concrete beams and columns. Although, the criteria used for this optimization is the **Design Rules**, the sizes must be chosen from the available sizes in the Redesign List.

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Redesign**.

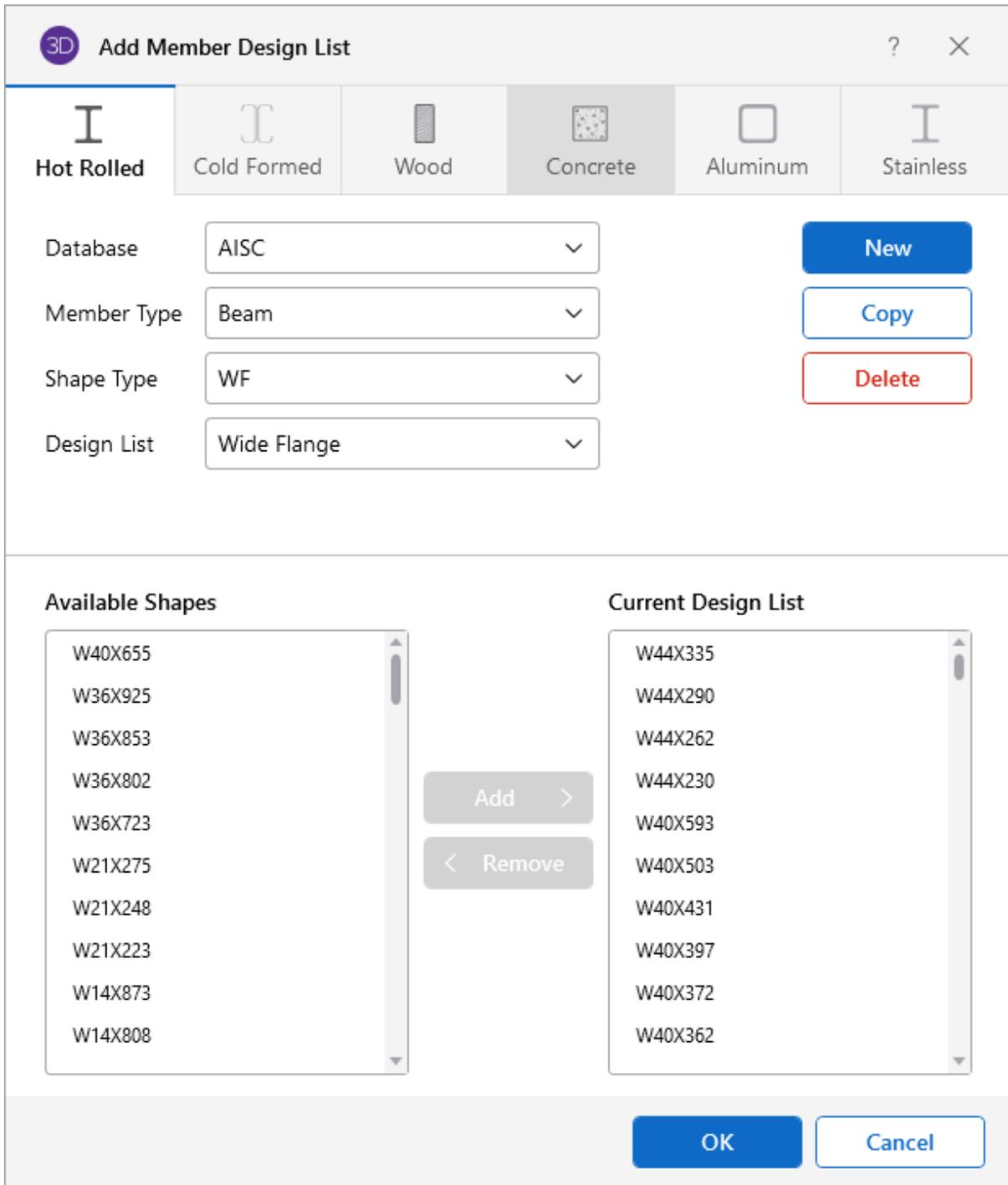
Creating or Modifying Redesign Lists

To create or modify a Redesign List:

1. Click on the **Advanced** ribbon tab to open it.



2. Click the **Redesign Lists** icon.
The Add member Design List window opens.



3. Select a material using the tabs at the top of the dialog.
For each material, select a Database, Member Type, Shape Type, and Design List.
4. Do one of the following, depending on what you want to do.

- Click **New** to create a new Redesign List for the selected Member Type and Shape Type.
- Click **Delete** to permanently delete the currently selected Redesign List.
- Click **Copy** to copy and rename the currently selected Redesign List.

The **Available Shapes** on the left are listed based on the selected Database, Member Type, and Shape Type.

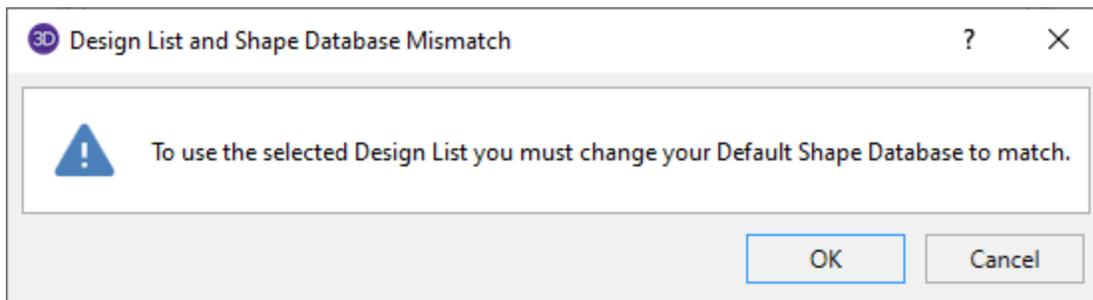
The **Current Design List** on the right lists all the shapes in the selected Design List.

5. Select shapes in the **Available Shapes** or **Current Design List**.
6. Use the **Add** or **Remove** buttons to modify the Current Design List.
7. Use the **Move Up** and **Move Down** buttons to re-order the shapes in the Current Design List.
8. Click **OK** to save your changes .

These design lists can then be assigned to a specific member or Section Set.

Design List Mismatch

If you see a Design List and Shape Database Mismatch dialog, then there is a mismatch between your default shape database and the design list you have currently selected. Either choose a Design List to match the default Shape Database, or change your default Shape Database so that you can use the Design List you want. To check your default Shape Database, open the Shape Database which is located on the Advanced tab of the ribbon toolbar. In the Shape Database, your default databases will currently be selected for each material.



Locating the Design Lists

By default the redesign list files exist in the Redesign Lists folder in the user's C:\RISA User Data\[USER NAME]\Redesign Lists. However, this folder can be changed by selecting the **File** button ▶ Application Settings ▶ [File Locations](#).

File Naming Convention for the Design Lists

The design lists must be ASCII text files with a file extension of ASC. In addition, the first 3 characters of the Design indicate which Region the lists will be available for. The region can be changed by

selecting the RISA-3D button ▶ Application Settings ▶ General ▶ [Advanced](#). Each region corresponds to the following file name prefix.

Region	File Prefix
United States	US_
Canada	CA_
Britain	BS_
Europe	EU_
India	IN_
Australia	AU_
New Zealand	AU_
Mexico / South America	MX_

File Format for Redesign Lists

The following is an example of a user defined redesign list. You can view these files using any text editor such as Window's Notepad. However, this file must be saved in the `risa__redesign_lists` folder as an ASCII text file with a file extension `ASC`. If the file is saved with a `txt` extension, it will not be read into RISA-3D. The various entries for the input fields are described below:

The first field [`RISA_REDESIGN_LIST`] identifies this as a RISA Redesign list. If this line is not present the file will not be recognized by the program.

The second field [`NAME`] identifies that the next entry will be the name of the Shape Group that will identify this list in the program. In this case, this Shape Group is given the name *Available W24s*.

The next field [`MATERIAL_TYPE`] identifies that the next entry will be the material type that this shape group belongs to. This material type must be given one of the following designations: Hot Rolled Steel, Cold Formed Steel, Wood, Steel Products, or Wood Products.

The next field [`MEMBER_TYPE`] identifies that the next entry will be the designation of the member type. This entry must be designated as either a Beam or Column.

The next field [`ASSOCIATED DATABASE`] identifies that the next entry will be the RISA database name in which the entered shapes are defined.

The next field [`UNITS`] identifies the units used to define the member sizes in the redesign list. Entries may be Inches, Millimeters, or Centimeters. This field is only applicable to files similar to the concrete and wood lists which have the dimensions built right into the list. For files which are based on database shape names, this field will have no effect.

The next field [`END HEADER`] identifies the end of the descriptive information and the beginning of the shape list.

The next field [AVAILABLE_SHAPES] identifies that the next series of lines will signify all of the shapes available for this Shape Group. The end of this field is reached when the [SHAPE_END] entry is read. The file below uses an asterisk (*) to specify that W24x68's and W24x76's are NOT available for this particular job.

The next field [OPTIMUM_SHAPES] identifies that the next series of lines will signify the preferred shapes for this Shape Group. The program will attempt to only use optimum shapes for beams that are significantly braced. By significantly braced, we mean that the unbraced length of the top flange is less than or equal to three feet. The end of this field is reached when the [SHAPE_END] entry is read.

The last field [END] identifies the end of the file.

[NAME]

Available W24s

[MATERIAL_TYPE]

Hot Rolled Steel

[MEMBER_TYPE]

Beam

[ASSOCIATED_DATABASE]

AISC

[END_HEADER]

[AVAILABLE_SHAPES]

W24X104

W24X103

W24X94

W24X84

*W24X76

*W24X68

W24X62

W24X55

[SHAPE_END]

[OPTIMUM_SHAPES]

W24X55

W24X62

*W24X68

W24X76

[SHAPE_END]

[END]

Manual Modification

By default, RISA installs a series of default redesign lists for each material and member type. The default lists are assumed to be a compilation of what engineers would typically choose. To view the available sections for each list, you can open up the file located in the Redesign Lists folder using a word editing program.

To modify an existing redesign list, open the file in a word editing program to override any of the variables. The same formatting as the original file should be followed as well as saving the file to the same location, otherwise the program will not recognize the changes.

To create a new redesign list, open an existing file and run a Save As to save the file with a different file name in the same location. The [NAME] input as described in the above section will be the label you will see within the program so it is important to give your design list a unique name. The [MATERIAL_TYPE], [MEMBER_TYPE], and [ASSOCIATED DATABASE] are important criteria your member must have in order for the design list to be available. If a member in the model includes all three of the criteria defined in a particular design list, this design list will be available within Members spreadsheet Design List pull down box (see [Design Lists](#) for more information).



Note: If you want to list a member size in the redesign list file but not have it considered as part of the redesign list, you can put an asterisk in front of the shape name.

Appendix C – STAAD® Files

STAAD files may be imported into RISA-3D by choosing File > Import and then specifying STAAD as the file type. RISA-3D can translate files produced by the STAAD III or STAAD/Pro programs.

The translation process will cause model information including geometry data, member and element properties, load information, some advanced modeling information, and AISC steel code check information to be read into RISA-3D.

Translated geometry data are the joints, members, and plate/shell elements. Supported member and element properties are the material property information, element thickness and member section shape data. Loading information includes joint loads, support displacements, member distributed and point loads, and element surface loads. The advanced modeling information that are translated are such things as the joint boundary conditions, including springs, the member end releases, and the “Truss” members. Note that the STAAD model type (i.e. Space, Plane, etc.) will also be detected and this information will be used to help translate the model.

The translator has been tested with files as old as STAAD, version 10. If you’re having a problem translating an older or newer file some features may not be available.

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 pops up and tells 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

STAAD General Keywords

Keyword	Description
UNIT	Statements that cause model data to be interpreted in the specified units. All STAAD unit types are supported.
SET Z UP	A statement that causes the vertical axis setting on the Global Parameters to be set to the Z-axis. (Default vertical in RISA-3D is the Y-axis)
FINISH	A keyword 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.

STAAD Model Type Keywords

Keyword	Description
PLA	"Plane" models that are assumed to be in the X-Y plane at a Z-coordinate of

Keyword	Description
	zero. Thus, only the X and Y coordinates are read and the Z coordinates of all joints are set to zero.
SPA	“Space” models that are read in as is.
TRU	“Truss” models that cause member end releases to be set for all members so that members 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 can 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 ALLBoundary Condition code to apply a very soft spring to the in-plane rotational DOF. For space trusses, you can use the ALL code to apply very soft springs to all the rotational DOF’s (MX, MY, MZ) for all free joints. See Stability for more information.
FLO	“Floor” models that 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

STAAD Joint Keywords

Keyword	Description
JOInt COOrdinates	Only Cartesian coordinates are supported. If you have a model in cylindrical or reverse cylindrical coordinate, you need to read the model into STAAD and then save it back out. This causes 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

STAAD Member Keywords

Keyword	Description
MEMber INCidences	Repeat keywords and command file data generation are not

Keyword	Description
MEMber PROperties	supported. 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 CANadia	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	<p>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.</p> <p>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.</p>
TABle	<p>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.</p> <p>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.</p>

Keyword	Description
	For Canadian shapes as listed in the S16.1-94 standard, the following “type_spec” words are supported: ST , and T . All wide flange, channel, and WT shapes are supported. The HSS shapes are supported, however, STAAD uses the AISC names for the HSS shapes. All pipe shapes, single angles, double angles, built up box type tube shapes, double channels, and built up plate girders are not supported.
MEMber RELEase	All full member end releases are recognized. Partial releases are not supported or translated.
MEMber TRUss	Members which are assigned this property are given an I-end release of ALLpin and a J-end release of BENpin. The member will take moment if a distributed load or self weight load is applied.
MEMber LOAd	Most member loads are supported. Unsupported member loads include projected point loads and projected moments, loads with a shear center offset, distributed moment loads, and triangular loads with the maximum at the center of the member specified using the LIN load option.
STArt GROup DEFinition	This feature is used in STAAD to give a frequently used list of member/element items an easier to reference “name”. There is a limit of 32,000 groups, and 50,000 total group items that RISA-3D will use when translating the STAAD file.
CONstants	The constant keywords STEEL, CONCRETE, and ALUMINUM are supported. Note that since RISA-3D ties material properties and section properties together by using Section Sets, members with the same geometric properties but different material properties will be assigned to different Section Sets.
DEFine MATerial STArt	This feature is fully supported for ISOTROPIC materials only. NON-Isotropic material definitions will be replaced by the first default material in RISA-3D.

STAAD Element Keywords

STAAD Element Keywords

Keyword	Description
ELEment INCidences	Repeat keywords and command file data generation are not supported.
ELEment PROperty	Only uniform element thicknesses are supported. If multiple thicknesses are specified for an element, only the first thickness is read and used as the thickness for the whole element.
ELEment LOAd	Only uniform surface loads are supported.
STArt GROup DEFinition	This feature is used in STAAD to give a frequently used list of member/element items an easier to reference “name”. There is a

Keyword	Description
	limit of 32,000 groups, and 50,000 total group items that RISA-3D will use when translating the STAAD file.
CONstants	The constant keywords STEEL, CONCRETE, and ALUMINUM are supported. Note that since RISA-3D ties material properties and section properties together by using Section Sets, members with the same geometric properties but different material properties will be assigned to different Section Sets.
DEFine MATerial STArt	This feature is fully supported for ISOTROPIC materials only. NON-Isotropic material definitions will be replaced by the first default material in RISA-3D.

STAAD Load Keywords

STAAD Load Keywords

Keyword	Description
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

STAAD AISC Parameters

Parameter	Description
PARameter	<p>Only the AISC (ASD 9th or LRFD 2nd) codes and the Canadian CAN/CSA S16.1-94 code are supported in RISA-3D.</p> <p>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.</p>

Unsupported STAAD Features

In general, you will want to examine your STAAD translation log file to note all lines that were not read in and translated. Typically a line will only be written to the log file if it is not recognized and translated successfully. This will give a good indication of any features that weren't brought into RISA-3D.

- STAAD solves LOAD cases and LOAD Combinations; RISA-3D only solves Load Combinations. You will need to have additional load combinations containing only one basic load case per combination to solve your basic load cases.
- P-Delta analyses are specified for each load combination in RISA-3D. The P-Delta flag will NOT be set automatically. You will need to go set it for combinations where you want to include P-Delta effects.
- RISA-3D does not translate any of the information in the JOB Information block of model files.
- Members with K-joint's cause joints to be created at the K-joint coordinates. These joints have their degrees of freedom locked automatically during model solution.
- RISA-3D has a limit of 8 Basic Load Cases per Load Combination. If a STAAD model has more than 8 LOAD cases in a Load Combination, only the first 8 will be used and a warning message will be written to the log file.
- Any response spectra entered in your STAAD file will need to be entered in RISA-3D's spectra database. STAAD stores each spectra with a particular data file, whereas RISA-3D maintains a library of spectra which are accessible from any data file.
- Shapes that are defined using a User defined shape database file will need to be entered into RISA-3D's shape database. RISA can automatically convert these shapes based on a Mapping file defined in the STAAD Mapping File section at the end of this appendix.
- RISA-3D doesn't support non-isotropic materials. If you translate a model with a non-isotropic material specified, we will still translate the model, but we'll use the first default material instead.

STAAD User's Overview

Folks who have a lot a structural modeling experience with STAAD can usually come up to speed with RISA-3D fairly quickly. The only thing that'll slow you down is figuring out how to do in RISA-3D what you knew how to do in STAAD. The RISA-3D User's Guide is a great place to start, in spite of the fact that it covers a lot of basic modeling concepts, because it shows you the most common ways to get things done in RISA-3D. In STAAD, you were probably accustomed to generating the model by manually editing the command text file, or maybe starting the model with the graphical pre-processor and then fine tuning the model by hand in the text file. With RISA-3D the steps are similar, except that you won't ever be directly editing the text file. You will do all manual data editing using our spreadsheets. Things like Section Sets and Material Properties are good examples of data that will always be entered via the spreadsheets. The actual model geometry and the application of boundary conditions, loads, and design parameters is usually done quickest using the Model Generation functions or the Graphics Editing functions. Most of these tools will require that you spend a few minutes the first time you use them to study what they can do and how it can help you model. There is

a full explanation of all the input parameters for each graphical tool in this Help file under each graphical tool topic.

Many STAAD users who are now using RISA-3D often want to know about the differences in the way modeling is performed between the two programs. You may want to read about some of the differences between RISA-3D and STAAD that we've documented and discussed in the relevant Help file topics.

A big plus for people who've used STAAD for a while and built up a library of models is that RISA-3D can read STAAD input files.

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 Differences from RISA-3D

STAAD is a "batch" mode program, where you are building a text input file either by hand or using their pre-processor. The latest STAAD/Pro program is very nearly an interactive program, with the only external programs being the solvers. RISA-3D is completely an interactive program in that we do not write an intermediate file. All input, solution, and results are performed using the same program.

For manual data entry, the input file can be directly edited in STAAD, whereas in RISA-3D you edit your data manually in custom spreadsheets that error check your input as it goes in. RISA-3D also has many built in spreadsheet functions to assist manual editing of the model data. You can cut and paste from other programs and spreadsheet directly into the RISA-3D data spreadsheets. The *.R3D file format uses keyword delimited format that may be edited directly from a text editor such as notepad. See Appendix D for more information on the RISA file format. Directly editing the *.R3D file also bypasses many of the error-checking features that would catch syntactical errors in the model data (having your model data integrity assured before you even run the model will save you lots of time in the long run).

Member Data

RISA-3D uses a Section Set to relate a set of members to a particular shape. The analog in STAAD is their "Groups". The RISA-3D Section Set combines a material and a shape into a one entity, which is then assigned to members. In RISA-3D, steel redesign is performed on a Section Set basis, so the worst-case member in a section set will control the size of all the members in that set.

Load Data

STAAD solves Primary LOAD cases and LOAD Combinations; RISA-3D only solves Load Combinations. If you'd like to run all your Basic Load Cases in addition to your Load Combinations, you will need to set up additional load combinations that only contain one basic load case per combination. RISA-3D has a limit of 8 Basic Load Cases per Load Combination. To learn how to get more than 8 BLC's per Load Combination, see [Nesting Load Combinations](#) for more information.

Analysis Types

A P-Delta analysis is a specific analysis option for a STAAD file and usually applies to all the loads in the current data file. In RISA-3D, a P-Delta analysis can be specified by setting a P-Delta flag for each combination on the Load Combination spreadsheet where you would like to include 2nd order effects.

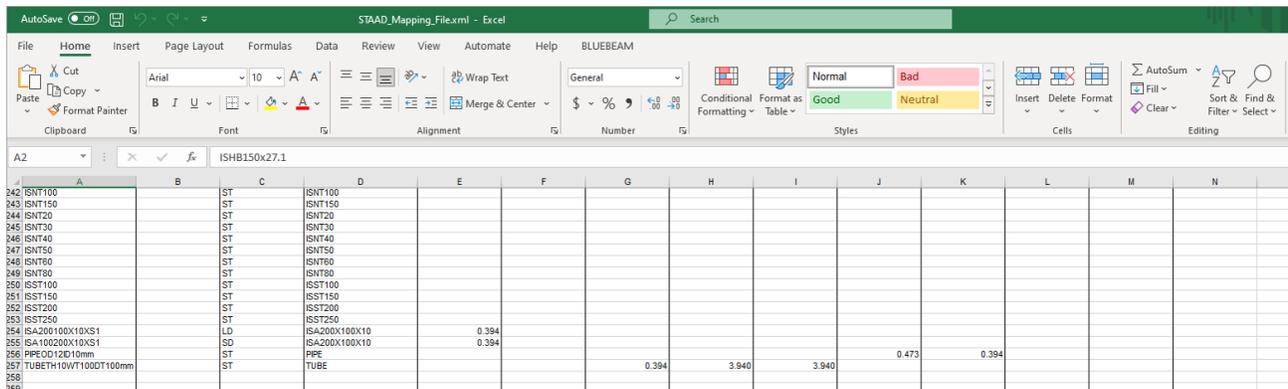
Dynamic/Response Spectrum

Any response spectra used by your STAAD file will need to be entered in RISA-3D’s spectra database. STAAD stores each spectra with a particular data file, whereas RISA-3D maintains a library of spectra which are accessible from any data file.

For dynamic analysis in RISA-3D, mass is assigned in the vertical direction and then assumed to act in all three global directions. In STAAD, you have to specify your mass in all the directions in which you want it to act.

STAAD Mapping File

The STAAD importing feature now accepts a mapping file (STAAD_Mapping_File.XML). This file format allows RISA to import User Defined database or table shapes from STAAD. All parameters in the tables are assumed to be based on units of kips and inches.



STAAD Mapping File Column Headers

Header	Description
RISA ShapeName	The Shape Name in RISA programs.
type-spec	Can be any of the following: ST, RA, D, LD, SD, T, CM, TC, BC, TB
table-name	Table section name like W6X9, C9X15 etc.
SP	Spacing between angles
WP	Width of Cover Plate
TH	Thickness of plate or tubes
WT	Width of tubes

Header	Description
DT	Depth of tubes
OD	Outer Diameter
ID	Inner Diameter
CT	Concrete Thickness for Composite Sections
FC	Compressive Strength of Concrete for Composite Sections.

Appendix E - Interfacing with Other Programs

RISA-3D interacts with a number of other RISA programs within the RISA suite (see below) as well as a the 3rd party programs listed in the sections below.

Integration with other RISA programs

For information on the integration between Floor and 3D see [RISA-3D Integration](#).

For information on the integration between RISA-3D and RISASection see [Shape Databases](#).

For information on the integration between RISA-3D and Foundation see [RISAFoundation Integration](#).

For information on the integration between RISA-3D and RISAConnection see [RISAConnection Integration](#).

Exporting to ADAPT Builder

RISA-3D has the ability to export Node Reactions in an ADAPT Builder file format(.b3d file extension). This allows for the reactions to be imported as loads in ADAPT Builder.

Exporting to SDS2

RISA-3D has the ability to export Enveloped Member End Reactions in a Flat file format(.txt file extension).

Linking your Autodesk Revit Structure model with RISA-3D

AutoDesk® Revit® Structure users can now link directly with RISA-3D and/or RISAFloor. This link is being continuously improved and updated. Therefore, for the most up to date information on this link, please visit the AutoDesk and RISA web sites.

risa.com/products/risa-revit-link

autodesk.com/products/revit/overview

For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Revit**.

Importing or Exporting SAF Files

RISA-3D has the ability to import and export files in the SAF file format(.xlsx file extension). This is an open exchange file format that allows for exchanging data between structural analysis software. This is an initiative from Nemetschek Group to improve collaboration.

For more information and to download an SAF Viewer, visit: <https://www.saf.guide/en/stable/getting-started/what-is-saf.html>

Importing or Exporting CIS/2 Files

RISA-3D and RISAFloor have the ability to import and export files in the CIS/2 file format(.stp file extension). This is a generic file format that allows different pieces of software to communicate with each other. Our three main partners that accept and produce this type of file are:

- SDS2 Design Data
- Tekla Structures
- StruCad

For more information and to download the CIS/2 translator, visit our website at: risa.com/products/risa-cis2-translator

Importing or Exporting DXF Files

For DXF importing / exporting, refer to the [DXF Files](#) section of the main manual.

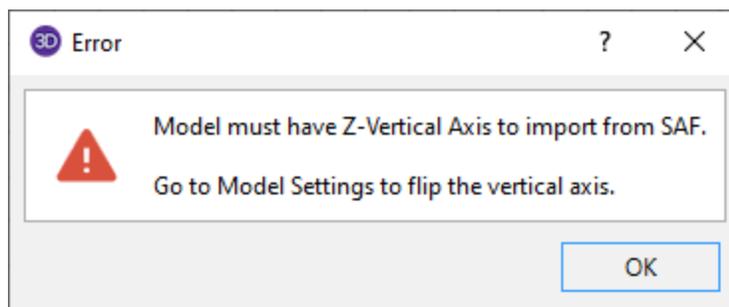
Importing STAAD Files

Refer to [Appendix C](#) for more information on importing STAAD files.

Importing SAF Files

RISA is actively involved in the SAF Open Exchange Initiative with other Nemetschek brands. For more information, visit the SAF Open Exchange Guide: <https://saf.guide/>

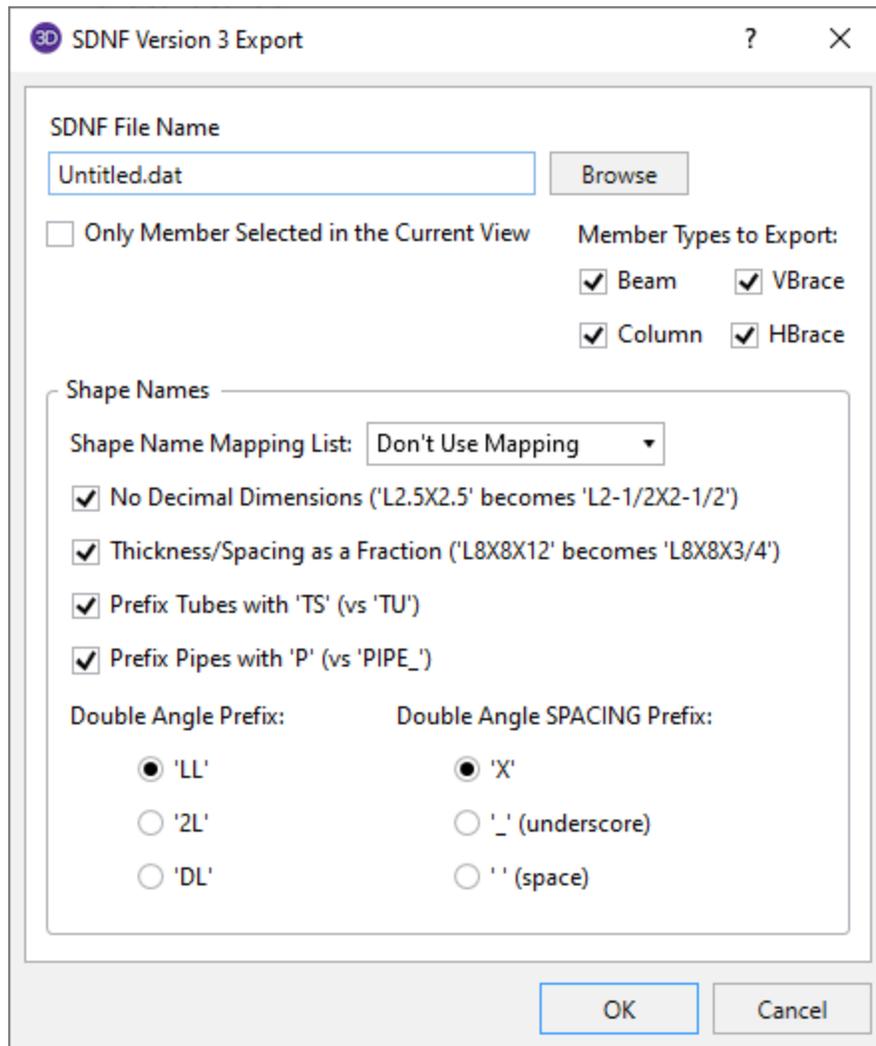
In order to import an SAF file, the vertical axis in RISA-3D must be Z.



Exporting to an SDNF File Format

The Steel Detailing Neutral File is a file format that is used by a number of CAD and steel detailing packages. This format is superior to a DXF export in that it contains member size and orientation information as well. The SDNF export dialog may be obtained by selecting File - Export - SDNF from the Main Menu Toolbar.

Because the SDNF data exchange is so heavily dependent on naming conventions for the structural steel, there are a number of options related to the steel shape prefixes and naming conventions as shown in the dialog below.



**Note:**

- SDNF exporting includes member information only. It does not export Wall Panel Elements, solids or plates.
- The SDNF exporting does not link to the Member Detailing information. Setting cardinal points and offsets in the Member Detailing will have no effect on this export.

Structural Desktop

Structural Desktop is a 3rd party program that works inside AutoCAD and is capable of importing RISA models into AutoCAD for the purposes of drawing production. Structural Desktop is also capable of exporting RISA files as well.

Structural Desktop also includes a library of steel joists that can be used directly inside of RISA-3D or 2D. Refer to the [Shape Database - General Shapes](#) section of the manual for more information.

For more information, see the RISA and Structural Desktop web sites:

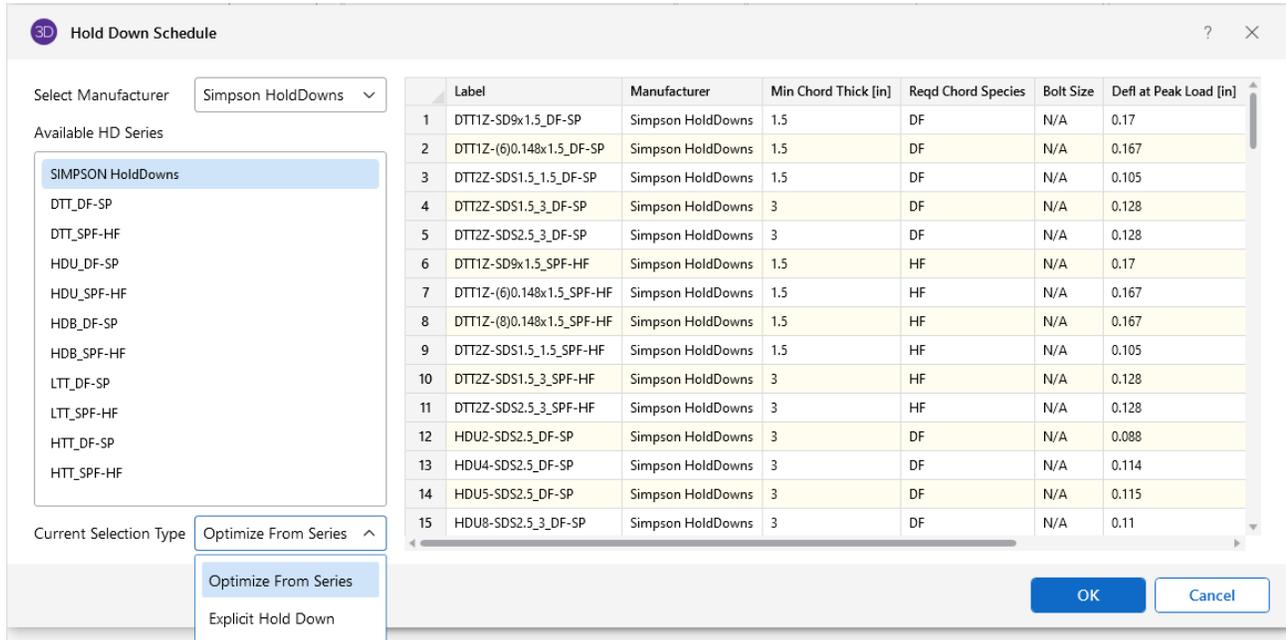
risa.com/partners/prt_sdt.html

www.structuraldesktop.com

Appendix F – Wood Shear Wall Files

RISA-3D has design databases for wood shear walls and diaphragms which are used to optimize the nailing and hold downs for the wall. The criteria used for this optimization is specified on the Wood Wall (Fasteners) and Wood Diaphragms tabs of the **Design Rules** spreadsheet. In addition to this basic criteria, the user may specify a subset of the overall database from which the design / optimization must be performed.

Hold Downs



Each database of hold downs is specified by an XML file in the "Hold Downs" sub-directory of the Wood Wall panels directory. The location of this directory is based on the information in the File Locations tab of the [Application Settings](#) window.

The program comes pre-loaded with three XML files, one for Simpson hold-downs, one for Canadian Simpson hold-downs, and one for USP hold-downs. The name of the XML file itself will be used in the list of databases in the Hold Down Schedule Dialog.

The first sheet of the XML file should always be descriptive of the contents of the database (such as **Simpson HoldDowns**). This is because the name used here is the name used in the Design Rules spreadsheet. This sheet contains all of the identifier, design and code check information used for each hold down. These entries are described below.

Full Database - Required Fields

The following fields are *required* information. If they are not provided or are left blank, then that hold down will not be available for use in that database.

Hold Downs: Full Database - Required Fields

Field	Description
Label	The 'Label' field is used to identify the hold down. This field must be referenced on the sheets that identify families or groups of hold downs. Maximum numbers of characters for this field is 32.
Deflection at Peak Load	The 'Deflection at Peak Load' entry is used to calculate the deflection of the shear wall per APA / NDS formulas. This deflection is then reported

Field	Description
	on the shear wall detail report for each wall panel.
CD Factor	The 'CD Factor' is the assumed load duration factor that was used as the basis for specifying the listed Allowable Tension value for that hold down.
Allowable Tension	A load combination may be solved with a load duration factor different from the CD Factor described above. When this is the case, the 'Allowable Tension' for that hold down will be adjusted based on the difference between the assumed and actual load duration factors.

Full Database - Optional Fields

The following fields are optional. They are not currently used in the design or capacity calculations, but are reported on the detail reports for reference purposes only.

Hold Downs: Full Database - Optional Fields

Field	Description
Manufacturer	The 'Manufacturer' field is an identifier for the hold down. It is provided so that the engineer can more easily identify the call-outs for their final design drawings.
Min Chord Thickness	The allowable capacity of the hold down will vary based on the Chord Thickness. Therefore, the 'Min Chord Thickness' gives the minimum chord thickness that will yield the listed allowable tension load. However, this field is NOT currently used in the calculations. A future revision may provide a warning message if the actual chord thickness provided is less than required.
Required Chord Species	The allowable capacity of the hold down will vary based on the density of the wood species being used. Therefore, the 'Required Chord Species' lists the density assumed for the entered allowable tension. However, this field is NOT currently checked in the calculations. A future revision may provide a warning message if the actual chord density provided is less than required.
Bolt Size	The 'Bolt Size' when specified is used to reduce the axial capacity of the hold down chord itself. The only change to the calculation is that the program will perform the allowable tension check on the net area of the chord member rather than the gross area.

Grouping Hold Down Schedules for Design Optimization

The other sheets allow the user to group hold downs together into families for optimization purposes. These additional sheets CANNOT be the first sheet in the XML file as that first sheet must always be the one where the full database information resides.

The hold down labels specified on these sheets refer only to hold downs that have already been defined on the full database sheet. The information in this sheet need not be organized in a specific order. Instead, they will always be optimized based on the assumption that the hold down cost is directly related to the tension capacity. Therefore, when this group is selected, then the hold down within the group with the code check closest to unity, but still less than 1.0 will get selected during the optimization process.

Straps

The screenshot shows the 'Strap Schedule' dialog box. On the left, there is a 'Select Manufacturer' dropdown set to 'Simpson Chord Straps'. Below it, a list of 'Available HD Series' is shown, with 'SIMPSON Chord Straps' selected. The 'Current Selection Type' is set to 'Optimize From Series'. On the right, a table lists 15 different strap configurations with their respective properties.

	Label	Manufacturer	Clear Span [in]	End Length [in]	Reqd Chord Species	Fastener Type	Number
1	MSTA49_18_DF/SP_Nail	Simpson Chord Straps	18	n/a	DF/SP	Nail	26
2	MSTA49_16_DF/SP_Nail	Simpson Chord Straps	16	n/a	DF/SP	Nail	26
3	MSTC28_18_DF/SP_Nail	Simpson Chord Straps	18	n/a	DF/SP	Nail	12
4	MSTC28_16_DF/SP_Nail	Simpson Chord Straps	16	n/a	DF/SP	Nail	16
5	MSTC40_24_DF/SP_Nail	Simpson Chord Straps	24	n/a	DF/SP	Nail	20
6	MSTC40_18_DF/SP_Nail	Simpson Chord Straps	18	n/a	DF/SP	Nail	28
7	MSTC40_16_DF/SP_Nail	Simpson Chord Straps	16	n/a	DF/SP	Nail	32
8	MSTC52_24_DF/SP_Nail	Simpson Chord Straps	24	n/a	DF/SP	Nail	36
9	MSTC52_18_DF/SP_Nail	Simpson Chord Straps	18	n/a	DF/SP	Nail	44
10	MSTC52_16_DF/SP_Nail	Simpson Chord Straps	16	n/a	DF/SP	Nail	48
11	MSTC66_30_DF/SP_Nail	Simpson Chord Straps	30	n/a	DF/SP	Nail	48
12	MSTC66_24_DF/SP_Nail	Simpson Chord Straps	24	n/a	DF/SP	Nail	54
13	MSTC66_18_DF/SP_Nail	Simpson Chord Straps	18	n/a	DF/SP	Nail	64
14	MSTC66_16_DF/SP_Nail	Simpson Chord Straps	16	n/a	DF/SP	Nail	68
15	MSTC78_30_DF/SP_Nail	Simpson Chord Straps	30	n/a	DF/SP	Nail	64

Each database of straps is specified by an XML file in the "Straps" sub-directory of the Wood Wall panels directory. The location of this directory is based on the information in the File Locations tab of the [Application Settings](#) window.

The program comes pre-loaded with a database for Simpson Chord Straps. The name of the XML file itself will be used in the list of databases in the Strap Schedule Dialog.

The first sheet of the XML file should always be descriptive of the contents of the database (such as **Simpson Chord Straps**). This is because the name used here is the name used in the Design Rules spreadsheet. This sheet contains all of the identifier, design and code check information used for each strap. These entries are described below.

Full Database - Required Fields

The following fields are *required* information. If they are not provided or are left blank, then that strap will not be available for use in that database.

Straps: Full Database - Required Fields

Field	Description
Label	The 'Label' field is used to identify the strap. This field must be referenced on the sheets that identify families or groups of straps. Maximum numbers of characters for this field is 32.
CD Factor	The 'CD Factor' is the assumed load duration factor that was used as the basis for specifying the listed Allowable Tension value for that hold down. A load combination may be solved with a load duration factor different from the CD Factor described above. When this is the case, the Allowable Tension for that hold down will be adjusted based on the difference between the assumed and actual load duration factors.
Allowable Tension	'Allowable Tension' is the value the program will use when designing the strap to be used.

Full Database - Optional Fields

The following fields are optional. They are not currently used in the design or capacity calculations, but are reported on the detail reports for reference purposes only.

Straps: Full Database - Optional Fields

Field	Description
Manufacturer	The 'Manufacturer' field is an identifier for the strap. It is provided so that the engineer can more easily identify the callouts for their final design drawings.
Clear Span	The 'Clear Span' is the distance the strap is required to span between elements it is resisting the tension of.
End Length	'End Length' is the length of the strap that extends past the clear span. The allowable capacity of the strap will vary based on the density of the wood species being used. Therefore, the Required Chord Species lists the density assumed for the entered allowable tension. However, this field is NOT currently checked in the calculations. A future revision may provide a warning message if the actual chord density provided is less than required.
Fastener Type	The 'Fastener Type' shows whether Nails or Bolts is used to fasten the strap to the tension chords.
Number of Fasteners	The 'Number of Fasteners' field show how many fasteners are required to be used to meet the requirements of the tension capacity given by the strap.

Grouping Strap Schedules for Design Optimization

The other sheets allow the user to group straps together into families for optimization purposes. These additional sheets CANNOT be the first sheet in the XML file as that first sheet must always be the one where the full database information resides.

The strap labels specified on these sheets refer only to straps that have already been defined on the full database sheet. The information in this sheet need not be organized in a specific order. Instead, they will always be optimized based on the assumption that the strap cost is directly related to the tension capacity. Therefore, when this group is selected, then the strap within the group with the code check closest to unity, but still less than 1.0 will get selected during the optimization process.

Panel Nailing Schedules

	Label	Grade	Min Thick [in]	Min Penetration	Over Gypsum	One/Two Sided	Nail Size	Staple Size	Nail
1	S1_5/16_6d@6	St-I	0.312	1.25	No	1	6.000000	n/a	6
2	S1_(2)5/16_6d@6	St-I	0.312	1.25	No	2	6.000000	n/a	6
3	S1_3/8_8d@6	St-I	0.375	1.375	No	1	8.000000	n/a	6
4	S1_(2)3/8_8d@6	St-I	0.375	1.375	No	2	8.000000	n/a	6
5	S1_7/16_8d@6	St-I	0.438	1.375	No	1	8.000000	n/a	6
6	S1_(2)7/16_8d@6	St-I	0.438	1.375	No	2	8.000000	n/a	6
7	S1_15/32_8d@6	St-I	0.469	1.375	No	1	8.000000	n/a	6
8	S1_(2)15/32_8d@6	St-I	0.469	1.375	No	2	8.000000	n/a	6
9	S1_15/32_10d@6	St-I	0.469	1.5	No	1	10.000000	n/a	6
10	S1_(2)15/32_10d@6	St-I	0.469	1.5	No	2	10.000000	n/a	6
11	S1_5/16_6d@4	St-I	0.312	1.25	No	1	6.000000	n/a	4
12	S1_(2)5/16_6d@4	St-I	0.312	1.25	No	2	6.000000	n/a	4
13	S1_3/8_8d@4	St-I	0.375	1.375	No	1	8.000000	n/a	4
14	S1_(2)3/8_8d@4	St-I	0.375	1.375	No	2	8.000000	n/a	4
15	S1_7/16_8d@4	St-I	0.438	1.375	No	1	8.000000	n/a	4

Each database of wall panels is specified by an XML file in the "Shear Panels" sub-directory of the Wood Wall panels directory. This directory is located based on the information File Locations tab of the [Application Settings](#) window.

The program comes preloaded with XML files, including OSB and Plywood databases from the American Wood Council's 2015 *Special Design Provisions for Wind and Seismic*.



Note: The 2015 IBC refers directly to AWC SDPWS 2015 instead of publishing separate IBC tables as in previous versions.

The first sheet of the XML file should always be descriptive of the contents of the database (such as **IBC 2012**). This is because the name used here is the name used in the Design Rules

spreadsheet. This sheet contains all of the identifier, nailing, design and code check information for each nailing schedule. These entries are described below.

Full Database - Required Fields

The following fields are *required* information. If they are not provided or are left blank, then that nailing schedule will not be available for use in the database.

Panel Nailing Schedule: Full Database - Required Fields

Field	Description
Label	<p>The 'Label' field is used to identify the panel schedule and its nailing requirements. This field must be referenced on the sheets that identify families or groups of panels. Maximum numbers of characters for this field is 32.</p> <div style="border: 1px solid black; padding: 5px; margin-top: 10px;">  Note: Panels with a label containing the characters "_W" together will be ignored during design optimization. </div>
Min Panel Thickness	<p>The 'Min Panel Thickness' is used during the design optimization to limit the selected panels based on the Design Rules chosen by the user. It is also used to help set the elastic stiffness of the wall panel used during the FEM solution.</p>
Ga	<p>The 'Ga' value is the Apparent Shear Stiffness from nail slip and panel deformation as defined in equation 4.3-1 of the NDS' Special Design Provisions for Wind and Seismic. This value (in combination with the Min Panel Thickness defined above) is used to set the elastic stiffness of the wall panel that will be used during the FEM solution.</p> <div style="border: 1px solid black; padding: 5px; margin-top: 10px;">  Note: When a family or group of panels / nailing schedules are assigned to a shear wall, the lowest value of Ga and Min Panel thickness will be used to determine the elastic stiffness of the plate elements in the FEM solution. </div>
One/Two Sided	<p>The 'One/Two Sided' field is used during the design optimization to limit the available panels based on the Design Rules specified by the user.</p>
Boundary Nail Spacing	<p>The 'Boundary Nail Spacing' field is used during the design optimization to limit the available panels based on the Design Rules specified by the user.</p>

Field	Description
	 Note: The maximum field spacing is never entered in the program but is generally equal to 12 inches for the nailing schedules defined in the 1997 UBC and 2006 IBC databases. If a different nail spacing is present, then the user should add in a new nailing schedule to the existing database with a user defined shear capacity.
Shear Capacity	The 'Shear Capacity' listed in the spreadsheet is the primary value that controls the code checking of the shear wall. This is the seismic capacity, which the program can automatically increase for wind loads if the Wind ASIF function is enabled.

Full Database - Optional Fields

The following fields are optional. They are not currently used in the design or capacity calculations, but are reported on the detail reports for informational purposes only.

Panel Nailing Schedule: Full Database - Optional Fields

Field	Description
Panel Grade	The 'Panel Grade' field is an identifier for the engineer, but is not used in the design calculations. It is provided so that the engineer can more easily identify the panels in their design results and drawings.
Min Penetration	The 'Min Penetration' field is an identifier for the engineer, but is not used in the design calculations. It is provided so that the engineer can more easily identify the panels in their design results and drawings.
Panel Applied Over Gypsum	The 'Panel Applied Over Gypsum' field is also a identifier for the engineer that will not be used in the design calculations.
Nail Size	The 'Nail Size' listed in the spreadsheet is intended to refer to the Common nail size, but is reported only for reference purposes and are NOT used in the capacity calculations. If the nail size is changed by the user, then the user should also change the Shear Capacity entry accordingly. Below is a reference table for common, box, and sinker nails.

Field	Description					
	Diameter				Length	
Penny Weight	Common (in)	Common (Wire Gage)	Box (in)	Sinker (in)	Box and Common (in)	Sinker (in)
6d	0.113	11.5	0.099	0.092	2	1.875
7d	0.113	11.5	0.099	0.099	2.25	2.125
8d	0.131	10	0.113	0.113	2.5	2.375
10d	0.148	9	0.128	0.120	3	2.875
12d	0.148	9	0.128	0.135	3.25	3.125
16d	0.162	8	0.135	0.148	3.5	3.25

The Staple size listed in the database is reported for reference purposes only. If the staple size is entered or changed by the user, then the user should also change the shear capacity entry to the appropriate value.

Grouping Panel / Nailing Schedules for Design Optimization

The other sheets in the database allow the user to organize multiple nailing schedules into groups or families for design optimization purposes. These additional sheets CANNOT be the first sheet in the XML file as that first sheet must always be the one where the full database information resides.

The panel labels specified on these sheets refer only to panel / nailing schedules that have already been defined on the full database sheet. The information in this sheet need not be organized in a specific order. Instead, they will always be optimized based on the assumption that the installed cost is directly related to the shear capacity. Therefore, when a group or family is selected, then the nailing schedule within the group with the code check closest to unity, but still less than 1.0 will get selected during the optimization process.

Diaphragm Nailing Schedules

Wood Diaphragm Schedule

Select Code: AWC 2015 OSB

Available Diaphragm Cases:

- AWC 2015 OSB Case 1_3
- AWC 2015 OSB Case 1_3
- AWC 2015 OSB Case 1_3
- AWC 2015 OSB Case 2_4
- AWC 2015 OSB Case 2_4
- AWC 2015 OSB Case 2_4
- AWC 2015 OSB Case 5_6
- AWC 2015 OSB Case 5_6
- AWC 2015 OSB Case 5_6

Label	Case	Bloc...	Fra...	Panel G...	Panel...	Nail ...	Min Pe...	Bounda...	Other ...	Nail ...	Stron...
C2/4U_2_RS_5/16_6...	2/4	No	2	Rated ...	0.312	6	1.25	6	6	1	6
C5/6U_2_RS_5/16_6...	5/6	No	2	Rated ...	0.312	6	1.25	6	6	1	6
C2/4U_2_RS_3/8_6d...	2/4	No	2	Rated ...	0.375	6	1.25	6	6	1	5
C2/4U_2_S1_5/16_6...	2/4	No	2	Structu...	0.312	6	1.25	6	6	1	6
C2/4U_3_RS_5/16_6...	2/4	No	3	Rated ...	0.312	6	1.25	6	6	1	5
C5/6U_2_RS_3/8_6d...	5/6	No	2	Rated ...	0.375	6	1.25	6	6	1	5
C5/6U_2_S1_5/16_6...	5/6	No	2	Structu...	0.312	6	1.25	6	6	1	6
C5/6U_3_RS_5/16_6...	5/6	No	3	Rated ...	0.312	6	1.25	6	6	1	5
C2/4U_3_RS_3/8_6d...	2/4	No	3	Rated ...	0.375	6	1.25	6	6	1	4
C2/4U_3_S1_5/16_6...	2/4	No	3	Structu...	0.312	6	1.25	6	6	1	4.5
C5/6U_3_RS_3/8_6d...	5/6	No	3	Rated ...	0.375	6	1.25	6	6	1	4
C5/6U_3_S1_5/16_6...	5/6	No	3	Structu...	0.312	6	1.25	6	6	1	4.5
C1/3U_2_RS_5/16_6...	1/3	No	2	Rated ...	0.312	6	1.25	6	6	1	9
C2/4U_2_RS_3/8_8d...	2/4	No	2	Rated ...	0.375	8	1.375	6	6	1	6
C5/6U_2_RS_3/8_8d...	5/6	No	2	Rated ...	0.375	8	1.375	6	6	1	6
C1/3U_2_RS_3/8_6d...	1/3	No	2	Rated ...	0.375	6	1.25	6	6	1	7.5
C1/3U_2_S1_5/16_6...	1/3	No	2	Structu...	0.312	6	1.25	6	6	1	9

Current Selection Type: CASE

Use Entire Case:

Buttons: Save As Defaults, OK, Cancel, Help

Each database of diaphragms is specified by an XML file in the "Diaphragms" sub-directory of the Wood schedules directory. This directory is located based on the information on the File Locations tab of the [Application Settings](#) window.

The program comes preloaded with XML files, including OSB and Plywood databases from the American Wood Council's 2015 *Special Design Provisions for Wind and Seismic*.



Note:

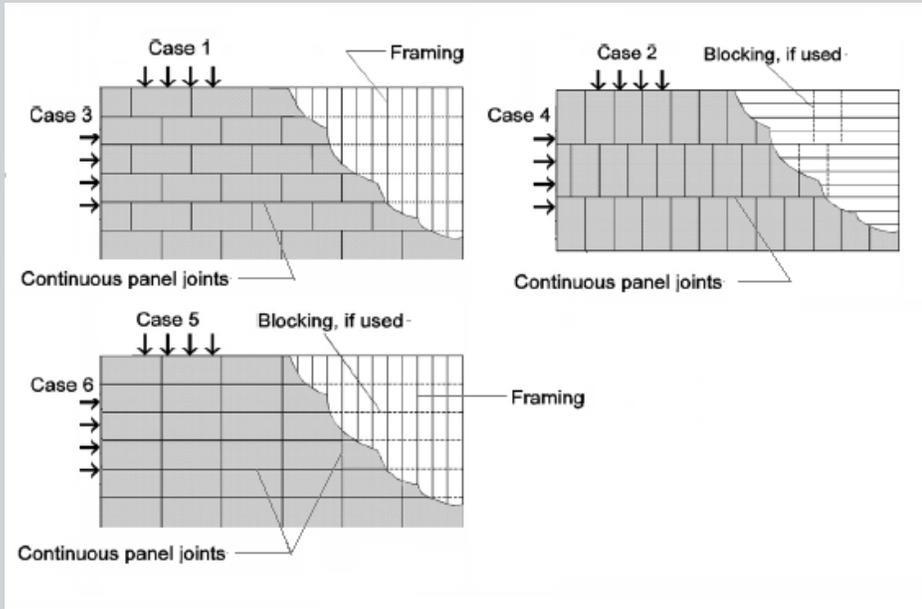
- The 2015 IBC refers directly to AWC SDPWS 2015 instead of publishing separate IBC tables as in previous versions.
- Diaphragm design is currently only available for flexible diaphragms that were created in RISA-Floor and brought into RISA-3D.

The first sheet of the XML file should always be descriptive of the contents of the database (such as **IBC_06-09_OSB**). This is because the name used here is the name used in the Design Rules spreadsheet. This sheet contains all of the identifier, nailing, design and code check information for each nailing schedule. These entries are described below:

Diaphragm Nailing Schedules: Full Database - Required Fields

The following fields are required information. If they are not provided or are left blank, then that nailing schedule will not be available for use in the database.

Diaphragm Nailing Schedules: Full Database - Required Fields

Field	Description
Label	<p>The 'Label' field is used to identify the diaphragm nailing. This field must be referenced on the sheets that identify families or groups of panels.</p> <div style="border: 1px solid black; padding: 5px; margin-top: 10px;">  <p>Note: Panels with a label containing the characters "_W" together will be ignored during design optimization.</p> </div>
Case	<p>The 'Case' field is used to specify the layout of the shear panels as shown below. Any diaphragm that has a Case 1 layout also has a Case 3 layout, and the same goes for 2/4 and 5/6.</p> <div style="text-align: center; margin: 10px 0;">  </div> <p>The program considers the deck span defined in RISAFloor to coincide with the long direction of the plywood. Therefore, Cases 2, 3 and 6 are considered parallel to the to the RISAFloor deck span. Whereas, Cases 1, 2 and 5 would be considered perpendicular to the same deck span.</p>
Blocked	<p>The 'Blocked' field specifies whether blocking is used to achieve the associated design strength. It also determines the method by which the diaphragm deflection will be calculated. For more information see Diaphragm Deflection.</p>
Panel Grade	<p>The 'Panel Grade' field specifies what grade of structural panel is used in the diaphragm. This may be set to either "Structural-I", "Rated Sheathing" or "Other" and is used as a criteria in the Design Rules.</p>
Panel Thickness	<p>The 'Panel Thickness' field specifies the thickness of the structural panel used for the diaphragm. This is a decimal value that is rounded to four places for reporting convenience. For example, a 15/32" panel is listed as 0.4688</p>

Field	Description
Boundary/Cont Edge Spacing	The 'Boundary/Cont Edge Spacing' field specifies the nail spacing at the boundary and along any continuous edges. These must be specified as the same value.
Other Edge Spacing	The 'Other Edge Spacing' field specifies the nail spacing at non-continuous edges.
Nail Lines	The 'Nail Lines' field specifies the number of lines of nails along each panel edge. This value is greater than (1) for High Load diaphragms. This value is not currently used in design optimization, but is reported on the output for reference purposes only.
Strong Shear Capacity	The 'Strong Shear Capacity' field specifies the shear strength of the diaphragm (lbs/ft) based on its stronger case. For example, while Case 1/3 represents the same panel layout, Case 1 has greater strength than Case 3 (based on load direction). This is the seismic capacity, which the program can automatically increase for wind loads if the Wind ASIF function is enabled.
Weak Shear Capacity	The 'Weak Shear Capacity' field specifies the shear strength of the diaphragm (lbs/ft) based on its weaker case. There are many situations where strong and weak capacities are identical. In these cases the same value must be specified for both fields. This is the seismic capacity, which the program can automatically increase for wind loads if the Wind ASIF function is enabled.
Strong Ga	The Strong Ga' field specifies the apparent shear stiffness (kips/in) of the diaphragm as specified in the NDS document <i>Special Design Provisions for Wind and Seismic</i> . Since this is the strong direction it will be based on the stronger direction / case for loading. For example, while Case 1/3 represents the same panel layout, Case 1 has greater stiffness than Case 3 (based on load direction). For more information see Diaphragm Deflection .
Weak Ga	The 'Weak Ga' field specifies the apparent shear stiffness (kips/in) of the diaphragm as specified in the NDS document <i>Special Design Provisions for Wind and Seismic</i> .
Gt	The 'Gt' field specifies the shear stiffness of panel depth. It is always the same for both strong weak directions, hence it does not need to be specified twice. For more information see Diaphragm Deflection .
Strong Nail Slip (en)	The 'Strong Nail Slip (en)' field specifies the nail slip used for deflection calculations based on the stronger case. For example, while Case 1/3 represents the same panel layout, Case 1 may have less nail slip than Case 3 (based on load direction). For more information see Diaphragm Deflection .
Weak Nail Slip (en)	The 'Weak Nail Slip (en)' field specifies the nail slip used for deflection calculations based on the weaker case. There are many situations where strong and weak nail slips are identical. In these cases the same value must be specified for both fields. For more information see Diaphragm Deflection .

Field	Description
	 Note: The Gt and Nail Slip fields are ignored for unblocked diaphragms.

Full Database - Optional Fields

The following fields are optional. They are not currently used in the design, capacity or deflection calculations, but are reported on the detail reports for informational purposes only.

Diaphragm Nailing Schedules: Full Database - Optional Fields

Field	Description
Framing Width	The 'Framing Width' field identifies the minimum required framing width for the nailing layout. A higher shear capacity can typically be achieved for a diaphragm by using wider supporting framing, thereby reducing the tension perpendicular to the grain of supporting members.
Minimum Penetration	The 'Minimum Penetration' field identifies the minimum required nail penetration specified in the IBC/NDS tables.
Nail Size	The 'Nail Size' listed in the spreadsheet is intended to refer to the Common nail size, but is reported only for reference purposes and are NOT used in the capacity calculations. If the nail size is changed by the user, then the user should also change the Shear Capacity entry accordingly. The section on shear walls contains a good reference table for common, box and sinker nails.

Grouping Panel / Nailing Schedules for Design Optimization

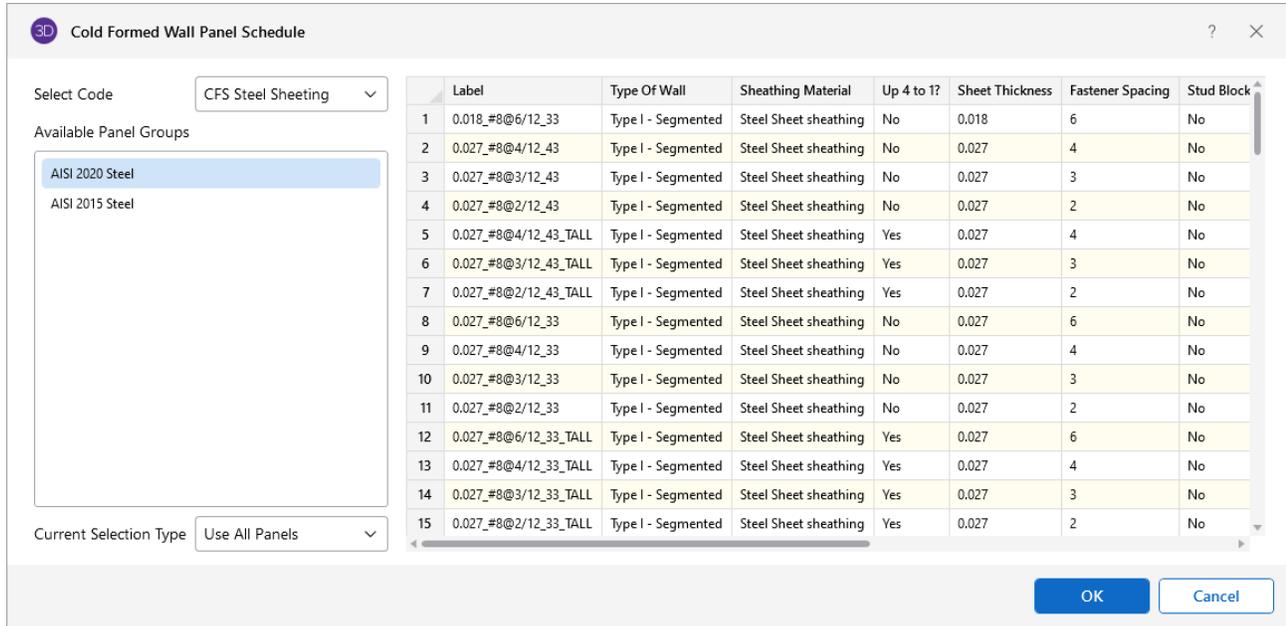
The other sheets in the database allow the user to organize multiple nailing schedules into groups or families for design optimization purposes. These additional sheets CANNOT be the first sheet in the XML file as that first sheet must always be the one where the full database information resides.

The labels specified on these sheets refer only to nailing schedules that have already been defined on the first (full) database sheet.

Appendix G – CFS Shear Wall Files

RISA-3D has design databases for CFS shear walls which are used to optimize the nailing for the wall. The criteria used for this optimization is specified on the CFS Wall (Fasteners) tab of the **Design Rules** spreadsheet. In addition to this basic criteria, the user may specify a subset of the overall database from which the design / optimization must be performed.

Panel Nailing Schedules



Each database of wall panels is specified by an XML file in the "Shear Panels" sub-directory of the CFS Wall panels directory. This directory is located based on the information File Locations tab of the [Application Settings](#) window.

The program comes preloaded with XML files, including Wood, Steel sheet, Fiberboard, and Gypsum databases based on the AISI S240-15/20 and AISI S400-15/20 codes.

The first sheet of the XML file should always be descriptive of the contents of the database (such as **AISI 2015 Wood**). This is because the name used here is the name used in the Design Rules spreadsheet. This sheet contains all of the identifier, nailing, design and code check information for each nailing schedule. These entries are described below.

Full Database - Required Fields

The following fields are *required* information. If they are not provided or are left blank, then that nailing schedule will not be available for use in the database.

Full Database - Required Fields

Field	Description
Label	The 'Label' field is a used to identify the panel schedule and its nailing requirements. This field must be referenced on the sheets that identify families or groups of panels. Maximum numbers of characters for this field is 32.

Field	Description								
	 Note: <ul style="list-style-type: none"> • Panels with a label containing the characters "_W" together will be ignored during design optimization. • Panels containing the same exact label name as a preceding panel will be ignored and will not be loaded into the program. 								
Type of Wall	The 'Type of Wall' dictates the design method that this row should be associated with.								
Sheathing Material	The 'Sheathing Material' indicates which material of the sheathing. This should be Wood Structural, Steel Sheet, Fiberboard or Gypsum Board Panels.								
2:1 to 4:1?	The '2:1 to 4:1?' column should only contain a yes or no entry. "No" means that the row will only apply for walls with height-to-width ratios up to 2:1. "Yes" means that the row only applies for walls with height-to-width ratios greater than 2:1 but less than 4:1.								
Thickness	The 'Thickness' column is used during the design to help set the elastic stiffness of the wall panel used during the FEM solution.								
Fastener Spacing	The 'Fastener Spacing' is used during the design optimization to limit the available panels based on the Design Rules specified by the user.								
Min Member Thickness	<p>The 'Min Member Thickness' is used in the design optimization to pick a sheathing configuration that satisfies the minimum required stud thickness. For wood sheathing, this column is treated as the required thickness since AISI S400 does not permit any other thickness to be used for the wood sheathing case.</p> <table border="1" style="width: 100%; border-collapse: collapse; text-align: center;"> <thead> <tr> <th style="color: #4a4a8a;">Stud thickness</th> <th style="color: #4a4a8a;">Sheathing entries with Min Member Thickness values equal to:</th> </tr> </thead> <tbody> <tr> <td>XXSXXX - 33</td> <td>33</td> </tr> <tr> <td>XXSXXX - 43</td> <td>33, 43</td> </tr> <tr> <td>XXSXXX - 54</td> <td>33, 43, 54</td> </tr> </tbody> </table>	Stud thickness	Sheathing entries with Min Member Thickness values equal to:	XXSXXX - 33	33	XXSXXX - 43	33, 43	XXSXXX - 54	33, 43, 54
Stud thickness	Sheathing entries with Min Member Thickness values equal to:								
XXSXXX - 33	33								
XXSXXX - 43	33, 43								
XXSXXX - 54	33, 43, 54								
Gov by S400 (EL)	The 'Gov by S400 (EL)' determines whether this sheathing configuration comes from the AISI S400-15/20 code or not. This column should only contain a yes or no entry. A "no" entry means that this configuration comes from AISI S240-15/20 and will only be applied to load combinations that do not contain earthquake load categories. A "yes" entry means that this configuration is pulled from the AISI S400-15/20 provisions and will only be applied when the load combination contains an earthquake load category (EL, ELX, ELZ, etc).								
Shear Capacity	The 'Shear Capacity' listed in the spreadsheet is the primary value that controls the code checking of the shear wall. This is the nominal lateral capacity of the shear wall that is pulled directly from the tables in AISI								

Field	Description
	S240 and S400. The program will then apply the appropriate factors depending on if ASD or LRFD design is selected.
Shear Modulus	The 'Shear Modulus' is (in combination with the panel Thickness defined above) is used to set the elastic stiffness of the wall panel that will be used during the FEM solution. This value is also used in the reported design deflection calculation for wood or steel sheathed panels.

For 7/16" OSB, we are assuming 24/16 OSB rated sheathing and approximating G based on AISI S240-15/20-C pg. 30. $CGGVtV = 177,300$ psi

For Struct 1 sheathing (4-ply) we are using IBC values assuming 24/16 rated sheathing. The values reported in the table are given in lb/in. For the values input in the spreadsheets that we'll use, we take this value and divide by the panel thickness to get this in psi which is required in the deflection calculation.



Note: When a family or group of panels/nailing schedules are assigned to a shear wall, the lowest value of Shear Modulus and Min Panel thickness will be used to determine the elastic stiffness of the plate elements in the FEM solution.

Full Database - Optional Fields

The 'Min Screw Size' listed in the spreadsheet is reported only for reference purposes and are NOT used in the capacity calculations.

Grouping Panel / Nailing Schedules for Design Optimization

The other sheets in the database allow the user to organize multiple nailing schedules into groups or families for design optimization purposes. These additional sheets CANNOT be the first sheet in the XML file as that first sheet must always be the one where the full database information resides.

The panel labels specified on these sheets refer only to panel / nailing schedules that have already been defined on the full database sheet. The information in this sheet need not be organized in a specific order. Instead, they will always be optimized based on the assumption that the installed cost is directly related to the shear capacity. Therefore, when a group or family is selected, then the nailing schedule within the group with the code check closest to unity, but still less than 1.0 will get selected during the optimization process.

Technical Support

Technical support is an important part of the RISA-3D package. There is no charge for technical support for all users with a **current** subscription of RISA-3D. Technical support is very important to the staff at RISA Tech, Inc.

Email Support

Email: support@risa.com. If your question is related to a specific model, please send us that model. Doing so makes it much easier for us to debug a problem or answer questions about specific issues. The model file has a .R3D extension. Make sure you tell us your name, company name, serial number, Key ID (or Cloud ID), phone number, and give a thorough problem description. If you have multiple members, plates, or load combinations, make sure you specify which ones to look at.

Phone Support

Hours: 6AM to 5PM Pacific Time, Monday through Friday (excluding holidays)

Phone Support: (949) 951-5815. Feel free to call, especially if you need a quick answer and your question is not model specific and therefore doesn't require us to look at your file.

Before contacting technical support, you should typically do the following:

1. **Please search the Help File or General Reference Manual.**

The Help File and General Reference are meant to cover most common questions asked about RISA-3D. The Table of Contents or Search can also be used, to find specific topics. This is a good first step in finding the answer to your question. It is also good practice to go through all of the 'Tutorials' when you first get the program. This helps you get a feel for the program and will likely increase your modeling confidence.

2. Ensure that you are running the latest version of the software.

Click the  **Help/Info** icon on the Quick Access Toolbar.

3. Visit our website at risa.com.

4. Make sure you understand the problem, and make sure your question is related to the program or structural modeling.

5. Take a few minutes to experiment with the problem to try to understand and re solve it.