RISAFloor

Rapid Interactive Structural Analysis – Floor Systems

Version 18- General Reference

RISA Tech, Inc.

27442 Portola Pkwy, Suite 200

Foothill Ranch, California 92610

(949) 951-5815

(949) 951-5848 (FAX)

risa.com

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 1
Program Limits 2
License Agreement 3
Application Interface **4**
Main Menu 4
Shortcut Menu 7
Toolbars 7
Dynamic View Controls 11
Shortcut Keys and Hot Keys 12
Search Bar 14
Status Bar 16
Windows 16
Modes 17
Beams **19**
Drawing Beams 20
Drawing Infill Beams 24
Modifying Beam Properties 26
Modifying Beam Design Parameters 28
Beam Information Dialog 30
Beams – Primary Data 33
Beams Spreadsheet - Detailing Data 35
Beams - Analysis **36**
Beam Stiffness 36
Column and Wall Stiffness 36
Multi-Span BeamSkip Loading 36
Detail Reports 39
Beam End Offset / Column Eccentricity 40
Concrete Beams (For Beam Supported Floors Only) 40
Beams - Results **41**
Single Angle Results 41
Beam Deflection Results 41
Beam End Reactions 43
Beam Vibration Results 44
Cold Formed Steel - Databases **45**
Custom vs. Manufacturer Shapes 45
Cold Formed Shape Types 46
Cold Formed Steel - Design **48**
Design Parameters 48
Design Results 50
Code Checks 51
Shear Results 52
Bending Results 53
Column Results 54
Assumptions and Limitations 55
Special Messages 57
Columns - Modeling **58**
Drawing Columns 59
Modifying Column Properties 62
Modifying Column Design Parameters 64
Column Stacks and Physical Columns 65
Columns Spreadsheet 67
Column Stacks Spreadsheet 71
Column Stack Manager 73
Columns - Analysis **75**
Multi-Story Column Analysis 75
Lateral Support From Floor/Slab Levels and Framing Elements 75
Column Moments and Shears - Eccentricity 77
Column Skip Loading 79
Hanger Columns 81
Columns - Results **82**
Column Forces Spreadsheet 82
Concrete Member - Database **85**
Custom Rebar Layout Database 85
Concrete Member - Design **90**

Concrete Spans 90
Concrete Design Parameters - Columns 91
Concrete Design Parameters - Beams 92
T-beam & L-beam Sections- RISAFloor ES Only 93
Parabolic vs. Rectangular Stress Blocks 94
Biaxial Bending of Columns 95
British Eurocode Design Parameters (BS EN 1992-1-1: 2004) 96
Limitations - General 96
Limitations - ACI 97
Limitations - Canadian Code 97
Limitations - Australian and New Zealand Codes 97
Limitations - British 98
Limitations - Euro 98
Limitations - Indian 98
Limitations - Saudi Code 98
Special Messages 98
Concrete Member - Design Results 100
Beam Results 100
Column Results 105
Concrete Detail Reports 107
Beam Detail Reports 108
Column Detail Reports 110
Connections 113
Available Connection Types 113
Integrated Connection Design Procedure 114
Troubleshooting RISAConnection Integration 120
Integration Error Messages 122
RISAFloor and RISA-3D Connection Design Interaction 124
Customizing RISAFloor 126
Save as Defaults 126
Application Settings (Preferences) 126
Decks and Diaphragms 131
Creating Diaphragm Edges and Openings 131
Modifying Diaphragm Edges and Openings 132
Assigning Deck Properties 133
Modifying Deck Properties 134
Deck Definitions - General 135
Deck Properties - Material Type 136
Deck Definitions - Loads 140
Diaphragm Edge and Opening Perimeters 141
Member Design Optimization 142
Member Design Lists 142
Member Design Rules – Size / U.C. 142
Member Design Rules – Deflection 143
Member Design Rules – Camber 144
Member Design Rules – Concrete Rebar 145
Member Design Rules Editor 146
Member Redesign Dialog 150
Diaphragms 152
Rigid Diaphragms 152
Semi-Rigid Diaphragms 154
Flexible Diaphragms 156
Diaphragms Spreadsheet - General Tab 157
Diaphragm Modeling Tips 158
Flexible Diaphragms - Analysis and Results 160
Analysis / Loading For Flexible Diaphragms 160
Diaphragms Spreadsheet - Wood Diaphragms Tab 163
Flexible Diaphragm Results - Detail Reports 164
Deflection Calculations 169
Diaphragm Key Plan 170
Diaphragm Design Limitations 170
DXF Files 172
Importing DXF Files 172
Exporting DXF Files 175
DXF Element Numbers 177
DXF File Format 177
File Operations 180
Starting Off 180
Importing and Exporting Files 180

Automatic Backup 181
Floors **182**
Creating a New Floor 182
Deleting a Floor 184
Floors Spreadsheet - General 184
Floors Spreadsheet - Beam Floors 186
Floors Spreadsheet - Concrete Slabs 187
Parent / Child 187
(Global) Model Settings **188**
Description 188
Solution 189
Codes 190
Composite 192
Wind 194
Seismic 194
Concrete 196
Graphic Display **198**
Multiple Windows 198
Controlling the Model View 199
Viewing Part of the Model 201
Saving and Retrieving Model Views 201
Model Display Options **202**
Beams/Columns/Walls 203
Points/Decks/Diaphragms or Points/Shear Caps 205
Slabs 208
Loads 209
Deflection 210
Miscellaneous 212
Design Strips/Cuts 214
User Defined 215
Graphic Editing **216**
Drawing and Modification Features 216
Undo Operations 217
Redo Operations 217
Project Grid 217
Drawing Grid 221
Snap Points 226
Import DXF 227
Copying Model Elements 229
Moving Model Elements 233
Trim/Extend Members 235
Deleting Elements 237
Re-Labeling Selected Elements 239
Graphic Selection **240**
Selection Modes 240
Inverting Selections 241
Criteria Selections 241
Lock & Dim Lock 245
Graphic Selection from Spreadsheets 245
Saving Selections 245
Help Options **246**
Electronic Help File 246
Context Sensitive Help 246
RISA Tech, Inc. Online 246
Tool-tips 246
Tutorials 246
Hot Rolled Steel - Databases **247**
Hot Rolled Steel - Design **252**
Design Parameters Spreadsheet 253
Unbraced Lengths 254
K Factors (Effective Length Factors) 254
Sway Flags 255
Composite 255
Effective Widths (B-eff Left and B-eff Right) 255
Camber Input 255
Web Openings 256
AISC Design Codes - Limitations 261
AISC Design Codes - Special Messages 262
Canadian Design Codes - Limitations 263
Canadian Design Codes - Special Messages 264
British Design Code - Limitations 265
EuroCode Design Codes - Limitations 268

EuroCode Design Codes - Special Messages 269
Indian Design Codes - Limitations 270
Australian & New Zealand Design Codes - Member Design Parameters 270
Australian & New Zealand Design Codes - Limitations 271
Hot Rolled Steel - Composite Beam Design 272
Overview 272
Pre-Composite Design 272
Post-Composite Design 273
Studs and Stud Distribution 274
Composite Decks 275
Moment of Inertia and Deflection 277
General Limitations 278
Limitations / Assumptions for Canadian Composite Design 278
Special Behavior (AISC ASD 9th Edition only) 279
Troubleshooting Composite Design 280
Hot Rolled Steel - Design Results 281
Design Results 281
Code Checks 282
Shear Results 283
Bending Results 284
Column Results 285
Web Opening Results 286
Web Opening Results Spreadsheet 286
Web Opening Detail Report 287
Cardinal Points 289
Overview 289
Detailing Input and Modification: 290
Visualization of the Detailing Layer 293
File I/O 293
Loads 294
Self Weight (Gravity Load) 294
Load Cases 294
Load Categories 295
Modifying Loads 296
Deleting Loads 296
Load Direction 296
Loads - Load Types 298
Area Loads 298
Tapered Area Loads 301
Deck Loads 301
Area Load Query Dialog 301
Line Loads 302
Point Loads 304
Loads – Live Load Reduction 307
Overview 307
Live Load Considerations 311
Integration with RISA-3D and RISAFoundation 312
Limitations 313
Loads – Load Attribution 314
One Way Distribution 314
Two Way Distribution 315
Loads – Load Takedown 317
Beams 317
Columns and Walls 317
Loads - Load Combinations 321
Load Combinations Spreadsheet 321
ASIF Factors for Wind / Seismic Load Combinations 323
Timber Design Load Duration Factor 323
Generating Building Code Combinations 323
Load Generation - Notional Loads 327
Vertical Load used for Notional Force Calculations 327
Notional Load Generation Dialog 327
Notional Load Results 328
Load Generation - Seismic Loads 329
Seismic Weight 329
Seismic Design Parameters 330
Seismic Load Results 332
Semi-Rigid Seismic Loads 333

Load Generation - Wind Loads **335**
Wind Load Parameters 335
Wind Load Results 337
Parapet Wind Loading 341
Mezzanine Levels (No Wind Loads) 349
Sloped Roof Wind Loads 349
Semi-Rigid Wind Loads 350
Other Considerations and Limitations 351
Material Properties **353**
Material Properties Spreadsheet 353
Material Take-Off **357**
Modeling Tips **358**
Dummy Members 358
Stranded Columns 358
Stranded Walls 359
P-Delta - Analysis **361**
Leaning Column Effect 361
P-Little Delta Analysis **362**
P-Little Delta Procedure 362
AISC Direct Analysis Method 363
ACI Concrete Design 363
Points **364**
Point Locations Spreadsheet 364
PointJoint Information Dialog 365
Printing **366**
Printing Reports 367
Printing to a File 369
Graphics Printing 370
Results **372**
Saving Results 372
Results Spreadsheets 372
Excluding Results 373
Graphic Results 373
Clearing Results 373
Member Detail Report 374
Concrete Member Detail Reports 376
RISAFloor and RISA-3D Integration **377**
Lateral System Model Generation 377
Diaphragms 378
Concrete Floor Slabs 378
Gravity Loads 378
Wind Loads 379
Seismic Loads 379
Buckling-Restrained Braced Frame (BRBF) System 379
Detach 379
RISAFoundation Interaction **380**
RISAFoundation Interaction with RISA-3D 380
RISAFoundation Interaction with RISAFloor 382
Limitations 383
Shape Databases **384**
Database Shape Types 384
Hot Rolled Shapes 384
Cold Formed Shapes 384
Concrete Shapes 384
Wood Shapes 384
General Shapes 385
Database Files 388
Sloped Members **390**
Tips for Modeling on Sloped Floors 392
Limitations 392
Solution **394**
Solution Method 394
Differences Between RISA-3D and RISAFloor 394
Difference Between Analysis and Design Sizes 395
Static Solutions / Stiffness Matrix 395
Spreadsheet Operations **397**
Moving and Scrolling 397
Spreadsheet Keyboard Commands 397
Selecting Spreadsheet Cells 398
Undoing Operations 398

Redoing Operations 398
Editing Spreadsheets 399
Moving and Copying Cell Contents 400
Finding in Spreadsheets 401
Default Spreadsheet Data 401
Export to Excel Spreadsheet 402
Special Spreadsheet Functions 403
Steel Products (Joists and Joist Girders) 404
Steel Products Database 404
Steel Products Design 405
Design for Joists with Non-Uniform Loads 406
Joist Girders 407
Composite Joist 408
Design Results - Steel Products 410
Limitations 411
Unbraced Lengths 413
Lb Values (Lb, Lu, Le) 414
Lcomp Values (Lcomp, Le-bend) 415
L-torque 416
Automated Unbraced Length Commands 416
Material Specific Behavior 418
K Factors (Effective Length Factors) 421
Sway Flags 422
Units 423
Standard Imperial Units 423
Standard Metric Units 423
Units Specifications 423
Vibrations - Floor Framing 425
Vibrations Spreadsheet 425
Calculation of Vibration Loads and Deflections 426
Limitations 427
Wall Panels 428
Drawing Wall Panels 428
Modifying Wall Panels 429
Wall Panel Spreadsheets 431
Wall Panel Editor 434
Loading 440
Wall Panel Load Attribution and Load Transfer 441
Wall Panel Load Attribution 441
Wall Panel Load Transfer 445
Wall Panels - Results 449
General Wall Detail Report 449
Wall Panel Results 449
CFS Wall - Design 454
CFS Wall Input 454
General Requirements for Shear Walls 456
General Program Functionality and Limitations 458
CFS Wall - Design Rules 461
Unity Check 461
CFS Wall (Studs) 461
Design Rules - CFS Wall (Fasteners) 462
CFS Wall Results 464
CFS Wall Results Spreadsheets 464
CFS Wall Self Weight 464
CFS Wall Detail Reports 464
Concrete Wall Panel - Design 469
Concrete Wall Input 469
Concrete Design Considerations 470
Concrete Lintel Considerations 472
Concrete Wall Modeling Considerations 473
Concrete Wall - Design Rules 476
Unity Check 476
Concrete Wall (Rebar) Rules 476
Concrete Wall (Cover) Rules 477
Concrete Wall Results 480
Concrete Wall Spreadsheet Results 480
Concrete Reinforcing Spreadsheet Results 480
Concrete Wall Detail Reports 481
Masonry Wall Panel - Modeling 488
Masonry Wall Input 488
Masonry Wall Optimization 492

Masonry Wall Panel - Design **493**
Axial Design - ASD 493
In Plane Design - Strength 494
Unreinforced Masonry Design - ASD 495
Masonry Lintel Design - ASD 495
Masonry Lintel Design - Strength 496
Masonry Wall - Design Rules **498**
Unity Check 498
Masonry Wall General 498
Masonry Wall Out of Plane Design 499
Masonry Wall Lintel Design 500
Masonry Wall Results **502**
Masonry Wall Spreadsheet Results 502
Concrete Reinforcing Spreadsheet Results .. 502
Masonry Wall Detail Reports 503
Wall Report 504
Masonry Detail Reports - Lintels 509
Wood Wall - Design **513**
Wood Wall Input 513
General Requirements for Shear Walls 515
General Program Functionality and Limitations 521
Wood Wall - Design Rules **525**
Unity Check 525
Wood Wall (Studs) 525
Design Rules - Wood Wall (Fasteners) 526
Wood Wall Results **529**
Wood Wall Results Spreadsheets 529
Wood Wall Self Weight 529
Wood Wall Detail Reports 529
Warning Log **535**
Wood - Database **536**
Wood Database Limitations: 538
Custom Wood Sizes: 541
Wood - Design **542**
Glu-Lams 542
Custom Wood Materials 543
Wood Member Design Parameters 544
NDS Adjustment Factors 545
CSA O86 Adjustment Factors 548
Limitations - Wood Design 550
NDS Emin Calculation 550
Beam Design Results 551
Beam Code Checks 552
Beam Shear Results 552
Beam Bending Results 553
Column Results 554
Special Messages - Wood Design 556
Wood Products **558**
Wood Products Database 558
Wood Products Adjustment Factors 559
Design Results - Wood Products 560
Code Checks 561
Wood Product Limitations 562
Elevated Slabs - Overview **563**
Suggested Workflow 563
Slab Modeling Overview 563
Load Path and Reinforcement Placement 564
Elevated Slabs - Modeling **570**
Drawing Slabs 570
Modifying Slabs 571
Thickness 574
Drawing Slab Openings 575
Modify Edges 577
Slab Definitions - General 578
Slab Definitions - Loads 579
Slabs Spreadsheet 579
Elevated Slabs - Design Strip Modeling **581**
Slab Design Rules - General 581
Slab Design Rules - Slab 581
Slab Design Rules - Deflection Parameters . 582
Design Strip Rules Editor 582
Design Strip Terminology 584

Drawing Design Strips - Support Lines 585
Modifying Slab Design Strips 590
Additional Modify Tools for Design Strips . 592
Slab Design Strips Spreadsheet 598
Slab Manual Design Cuts 599
Elevated Slabs - User Defined 601
Slab Design Rules - General 601
User Defined Bottom Rebar 602
User Defined Top Rebar 602
Continuous Top Rebar 603
Top Rebar Over Supports 603
User Defined Deflection 607
User Defined Results 607
User Defined Detail Report 608
Elevated Slabs - Modeling Tips 610
Drawing Long Support Lines 610
Modifying Mesh Size 610
6" Minimum Design Cut Width 610
Varying the Slab Thickness 615
Elevated Slabs - Analysis 617
Mesh Considerations 617
General Slab Deflection Considerations 620
Slab Point Deflections 620
Point Deflections vs Strip Deflections 622
Long Term Deflection Results 625
Elevated Slabs - Design 627
Concrete Reinforcement Design 627
Column Strip Width Calculation 630
Slab Shear Design 633
Limitations 634
Elevated Slabs - Spreadsheet Results 635
Slab Point Deflection 635
Slab Strip Deflection 635
Slab Design Strip Results 636
Slab Design Cut Results 637
Slab Forces 638
Elevated Slabs - Design Strip Detail Reports 640
Design Strip Detail Reports 640
Design Cut Detail Reports 651
Manual Design Cut Detail Reports 656
Elevated Slabs - One Way Slab Considerations 659
Elevated Slabs - Punching Shear Design 662
Punching Shear Basics 662
Punching Shear Calculation Values 662
Shear Cap Considerations 664
Shear Caps Spreadsheet 666
Punching Shear Loading 667
Finding the Controlling Case (Interior, Edge or Corner) 670
Other Punching Shear Considerations/Limitations 672
Elevated Slabs - Punching Shear Results 677
Punching Shear Values 677
Punching Shear Geometries 678
Column Detail Report 679
Appendix A – Redesign Lists 682
Design List Mismatch 682
Locating the Design Lists 682
File Naming Convention for the Design Lists 682
File Format for Redesign Lists 683
Creating or Modifying Design List 684
Appendix B – Error Messages 685
Appendix D – File Format 686
Appendix E - Interfacing with Other Programs 687
Integration with other RISA programs 687
Linking your Autodesk Revit Structure model with RISAFloor 687
Importing or Exporting CIS/2 Files 687
Importing or Exporting DXF Files 687
Appendix F – Wood Shear Wall Files 688
Hold Downs 688

Straps 690
Panel Nailing Schedules 691
Diaphragm Nailing Schedules 693
Appendix G – CFS Shear Wall Files **697**
Panel Nailing Schedules 697
Technical Support **700**

Before You Begin

Welcome to the RISAFloor 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 RISAFloor 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

RISAFloor is an application for modeling and designing multi-story commercial and institutional buildings. Buildings are modeled by the user on a floor-by-floor basis, and the floors are stacked atop each other to create the entire building structure. Buildings may be comprised of many different materials, including hot rolled and cold formed steel, concrete, wood, steel joists, and wood joists.

RISAFloor ES (Elevated Slab) has all of the capabilities of RISAFloor, as well as Concrete Floor Slabs for one-way and two-way concrete slab design. The RISAFloor ES features are only available to users who own a RISAFloor ES license, as opposed to those who just a RISAFloor license.

RISAFloor has both graphical modeling and spreadsheet input capabilities. Not only can you draw your model, there are exceptional generation tools that automatically layout floor elements. To modify your model input data directly, RISAFloor employs a powerful, proprietary spreadsheet. The combination of these features makes modeling both intuitive and powerful. The model can be rapidly edited, solved, viewed, modified, re-solved, etc. The truly interactive nature of RISAFloor is its primary strength.

With RISAFloor, all model editing, model solution, and results browsing is accomplished through the same interface and within the same program. The interactive approach offers several unique advantages such as 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 RISAFloor by using the menu system, or the toolbars. The best way to learn RISAFloor is to go through the Tutorials. The advantage to this is that you are exposed to the tools RISAFloor provides and the ways that you can take advantage of them.

Minimum System Requirements

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)

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.

A subscription provides access to the program and all upgrades during the subscription period. When your subscription expires, you will be given the opportunity to continue the program subscription. You are under no obligation to continue program subscription, of course, but if you decide to discontinue the subscription you will no

longer have access to the full version of the program and technical support. In the event subscription is discontinued, the program will operate in a viewer or demonstration mode.

Complete program support is available to registered owners of RISA and is included in the subscription price. See Technical Support for a list of your support options.

RISA Tech, Inc. will provide technical support only for the current version of RISA.

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

Program Limits

Software Limitations

Floors	200
Walls (per model)	10,000
Openings/Regions (per Wall Panel)	25/100
Columns per Floor	2,500
Beams per Floor	10,000
Points per Floor	25,000
Point Loads per Floor	10,000
Line Loads per Floor	10,000
Area Load Polygons per Floor	1,000
Deck Polygons per Floor	1,000
Materials	500
Custom Wood Species	500
Load Combinations	5,000
Concrete Slabs per Floor	20
Support Lines per Slab	300

Demonstration Version: You can open and solve a model of any size, however you may not:

- Use this version for Professional Design Purposes
- Create more than 2 Concrete Floor Slabs
- Create a Concrete Floor Slab other than 8" thick, 3000 psi concrete
- Save a model with more than 40 beams per floor
- Save a model with more than 3 Floors
- Save a model with more than 1 Concrete Floor Slab
- Save a model with more than 20 columns or 4 walls

- Save a model with more than 10 Support Lines
- Import RISAConnection design results back into the RISAFloor demo.

Also, the Demonstration Version will automatically shut down if left open for 24 continuous hours.

License Agreement

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

Application Interface

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

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

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

Main Menu

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

File Menu

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

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

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

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

Import DXF File will create a new floor using an existing DXF file.

Export will export the current file to a DXF or SDNF file.

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

Print will access RISAFloor printing options.

Page Setup will present page setup options for printing.

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

Exit will close RISAFloor, prompting for saving if necessary.

Edit Menu

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

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

Select Allwill highlight an entire spreadsheet all at once.

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

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

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

Delete Line will delete the current spreadsheet line.

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

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

Unmark Lines will unmark any currently marked lines.

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

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

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

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

Settings

Settings opens the Model Settings.

View Menu

New View will open a new model view window.

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

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

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

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

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

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

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

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

Model Display Optionsopens the Model Display Options.

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

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

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

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

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

Point Labels will turn the display of the point labels on or off, depending on the current setting. A third setting is also available where the points themselves are not shown at all.

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

Insert Menu

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

Modify Menu

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

Spreadsheets Menu

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

Solve Menu

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

Results Menu

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

Tools Menu

Delete All Unattached Points will delete any points that are not connected to beams, columns, decks, deck edges or loads.

RISA-3D Data will allow you delete selected portions of the RISA-3D model that automatically generated by Floor. It will also allow your to "re-start" the optimization of member sizes based on the current RISAFloor sizes.

Relabel Column Stacks assigns new labels to the column stacks in their current order in the **Column Stacks** spreadsheet.

Round Off Point Coordinates will round off the coordinates.

Delete all Wall Panel Regions will delete the regions on all wall panels. The regions will be automatically regenerated next time the model is solved.

Full Model Alignment will shift the coordinates for each floor to align all the vertical elements.

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

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

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

Window Menu

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

Help Menu

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

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

Upgrade to RISAFloor ES: If you do not currently own the Elevated Slab portion of RISAFloor (called RISAFloor ES) then purchasing it will allow you to enter in an RUS password to make it available in your program. See the Elevated Slabs - Overview topic for more information.

Licensing gives information about your subscription license and the option to Borrow or Return a license from your subscription.

About provides RISAFloor version and hardware key information.

Director Menu

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

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

Shortcut Menu

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

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

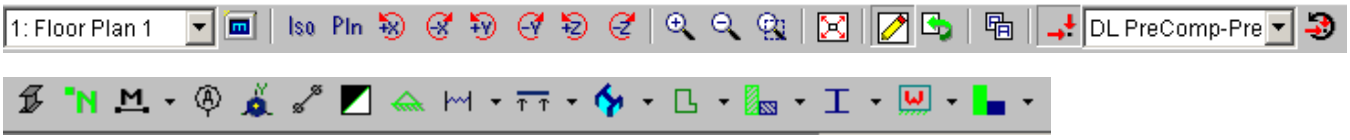
Toolbars

The **Toolbars** provide buttons to help you access common commands and popular options in the menu system discussed above. There are different toolbars that will appear as you work to build your model and browse your results. If at any time you are not sure what a particular button does, simply let your mouse hover over the button and a helpful tip will pop up and explain the button.

RISA Toolbar

The first horizontal toolbar located just below the Main Menu is called the **RISA Toolbar.** The buttons on this bar facilitate file and window access. You may use these buttons to open files and windows and also to analyze the model.

Window Toolbar



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

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

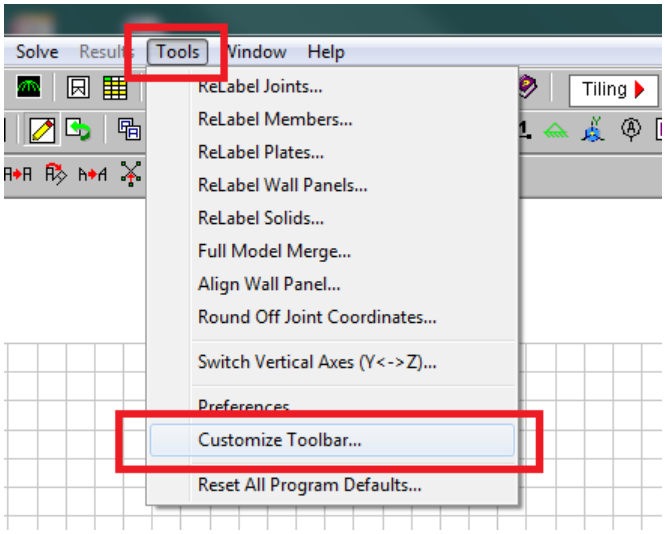
Note:

- - This 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.

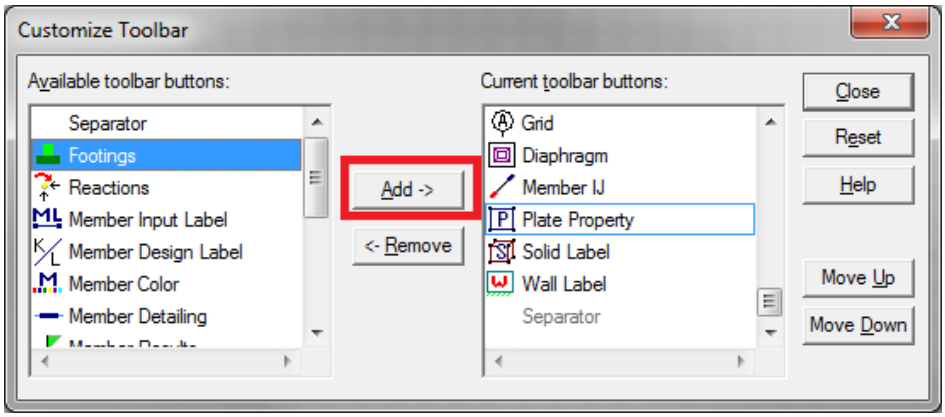
Customizable Model View Toolbar

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

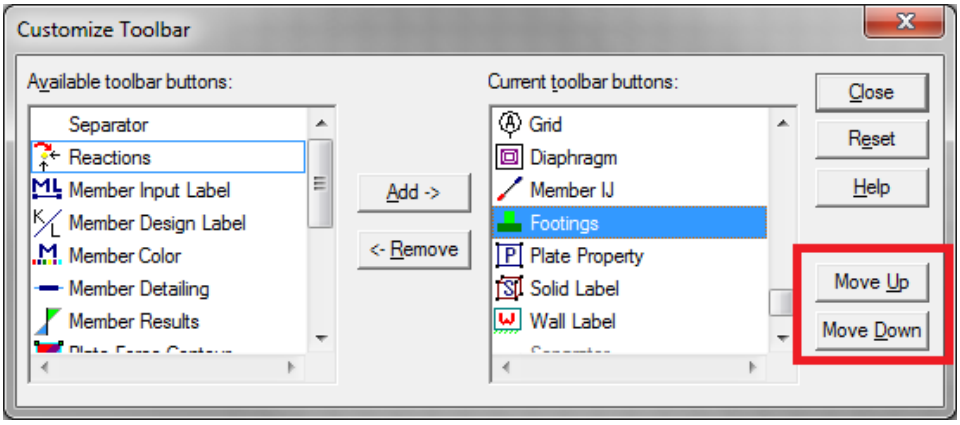
1. Go to Tools menu and select Customize Toolbar.



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



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



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

Note:

- You must have a model view as the current view to see this toolbar.
- If you add more buttons than will fit on the toolbar the buttons that are at the end of the "Current toolbar buttons" will be cut off.
- The changes you have made will automatically be saved on a per-user (Windows User) basis, such that next time you open the program the toolbar will be arranged per your Application Settings. These saved changes are saved in the registry in the HKEY_CURRENT_USER\Software\RISA Technologies***RISAProgram*** location with a String Value called "ToolbarState".

Selection Toolbar

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

Drawing Toolbar

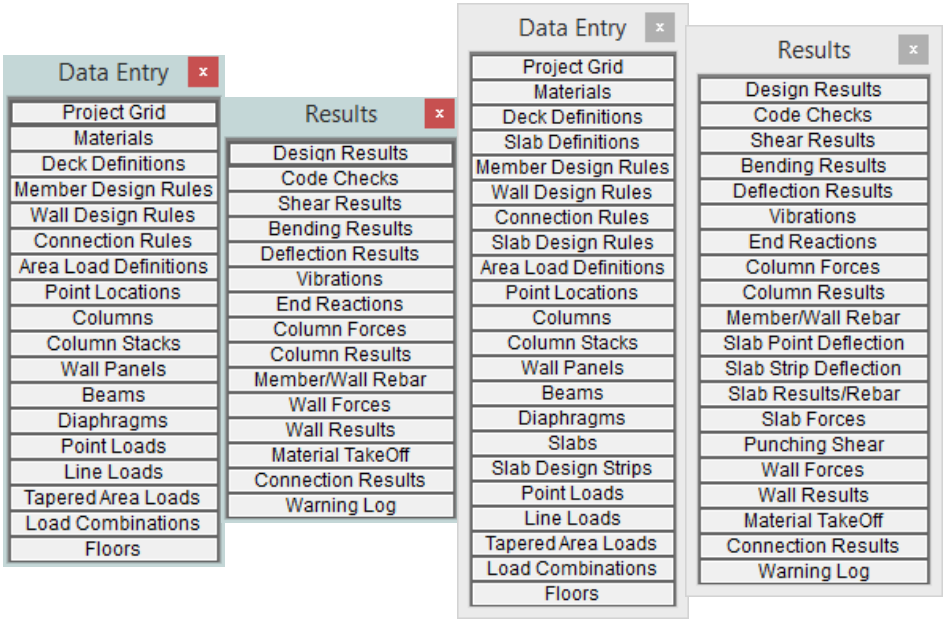
(beam floor)

(slab floor)

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

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

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



RISAFloor Data Entry | RISAFloor Results | RISAFloor ES Data Entry | RISAFloor ES Results

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

Note

- Depending on whether you own RISAFloor ES or just RISAFloor, the Data Entry and Results toolbars will be different.

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

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

Dynamic View Controls

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

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

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

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

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

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

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

Shortcut Keys and Hot Keys

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

General Hot Keys

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

Shortcut Keys available for Specific Windows

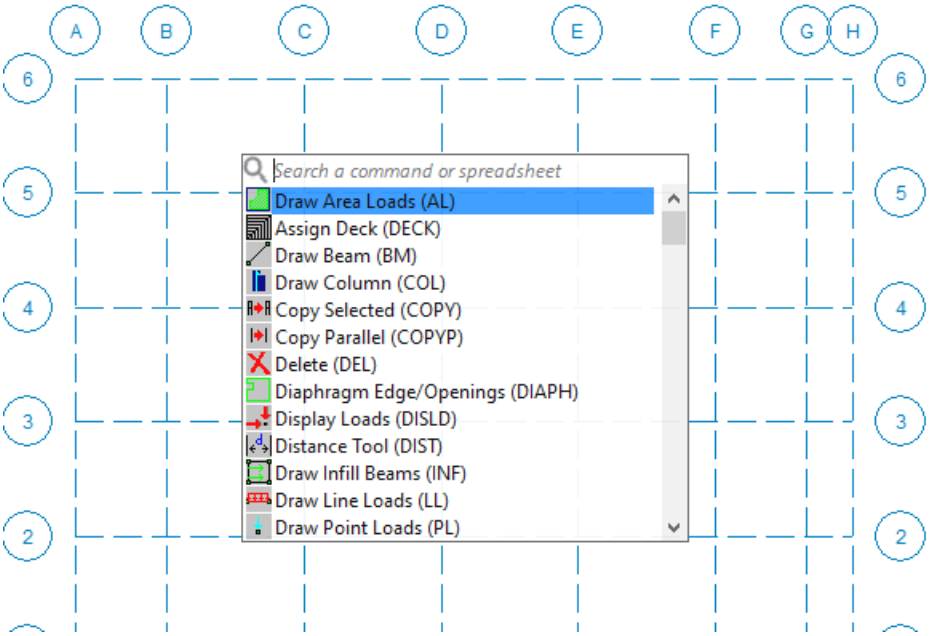
Key Combination	Model View Window	Spreadsheet
B	Open Draw Beams Dialog	
C	Open Draw Columns Dialog	
W	Open Draw Wall Panels Dialog	
I	Open Draw Infill Beams Dialog	
D	Open Create Diaphragm Edge Dialog	
S	Open Draw Slabs Dialog	
O	Create Slab Openings	
K	Open Assign Deck Dialog	
M	Move Selected Part of the Model	
N	Copy Selected Part of the Model	
E	Extend	
T	Trim	
L	Toggle Loads Display	
’	Show Wireframe or Rendering View	
.	Show Node Labels	
/	Show Beam Labels	
`	Lock View	
Delete	Open Delete Items Dialog	
R	Open Detail Report	
Shift-A	Draw Area Loads	
Shift-L	Draw Line Loads	
Shift-P	Draw Point Loads	
Ctrl-D	Open last graphic editing dialog	Delete Marked Lines
Ctrl-G	Toggle Drawing Toolbar	
Ctrl-A	Select All	Select All
Ctrl-U	Unselect all	
Ctrl-F		Block Fill
Ctrl-M		Block Math
Ctrl-I	Invert Selection	
Ctrl-L	Toggle Lock unselected	Unmark lines

Key Combination	Model View Window	Spreadsheet
Ctrl-Enter		Press cell button
F2	Open Model Display Options	Start/Stop Cell Edit
F3		Insert line
F4		Delete Line
F5	Initiates the "Distance" tool	Find
F6	Initiates the "Area Load Query" tool	
F8		Repeat Current Line
+	Zoom In	
-	Zoom Out	
←→↑↓ PgUp PgDwn	Scrolling	Scrolling

Spreadsheet Hot Keys that open spreadsheets

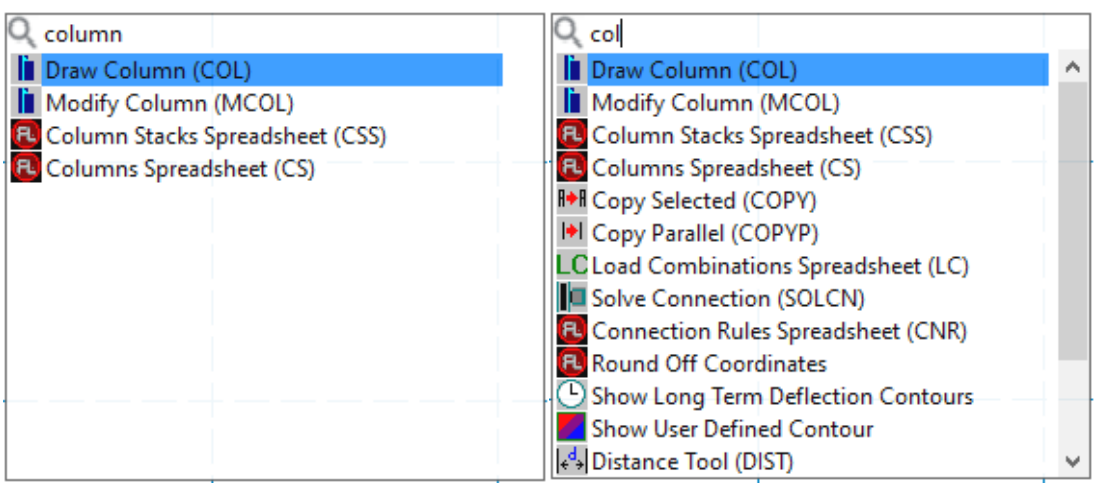
Key Combination	Unsolved Model	Solved Model
Ctrl-Alt-C	Point Coordinates	Column Results
Ctrl-Alt-D	Line Loads	
Ctrl-Alt-E	Beams – Primary Data	Deflection Results
Ctrl-Alt-F		Bending Results
Ctrl-Alt-G	Model Settings	
Ctrl-Alt-H		Design Results
Ctrl-Alt-J	Wall Panel Rules	
Ctrl-Alt-K		Solid Stresses
Ctrl-Alt-L	Load Combinations	
Ctrl-Alt-M	Materials	Material Take Off
Ctrl-Alt-N		Concrete Reinforcing
Ctrl-Alt-P	Point Loads	
Ctrl-Alt-R	Design Rules	End Reactions
Ctrl-Alt-S		Shear Results
Ctrl-Alt-U	Seismic Design Rules	Design Results

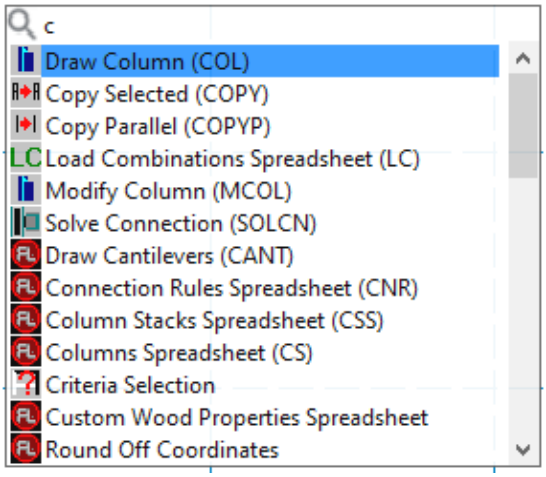
Search Bar



The **Search Bar** provides a quick access to commands and spreadsheets by searching their full names or shortcuts. The Search Bar can be accessed by either clicking the button in the RISA Toolbar, or pressing the **Space** key on the keyboard when in an active model view. Users can use the **Enter** key to confirm the selection of the search results. This will close the Search Bar and bring up the corresponding dialog or spreadsheet. Users can also use the Up or Down Arrow keys or hover the Mouse over different search results to move the selection. After the search bar is prompted, users can press the**Esc** key at any time to exit the search bar. The search bar will always be prompted at the center of the active model view window. Once the Search Bar is prompted, the program will hide other active dialogs or spreadsheets to give focus to the search bar. Pressing the **Esc** key will exit the search bar and regain access to the hidden dialogs or spreadsheets.

In the search bar, users can search either the full names or shortcuts of commands or spreadsheets. For example, typing either "Column..." or "COL" will locate the Draw Column (COL) command. The search results are displayed dynamically as users type, and the best match is highlighted at the top of the list. The search bar remembers recent search results and displays them at the top of the list. The most recent search results are always displayed first, making it easier for users to access their most frequently used commands or spreadsheets. Shortcuts for commands and spreadsheets can be learned from the parentheses in the search result display. For example, most spreadsheets use their acronyms as the search shortcuts. Users can type "CCS" to quickly locate the Code Check Spreadsheet instead of typing the full name, "Code Check...".





Accessing commands or spreadsheets through the Search Bar behaves the same way as clicking the buttons or spreadsheets from the application interface. For example, clicking the button multiple times will switch the display between wireframe and rendering view. Accessing the same tool from the search bar will behave the same way. The memory feature that saves the last search results at the top makes it easier to access the same command repetitively. Users can quickly press **Space** and **Enter** to mimic the toggling behavior of clicking a button. Overall, the search bar is a powerful tool that makes it easier for users to find and access commands and spreadsheets quickly and efficiently in RISA.

Status Bar

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

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

Solution Type	Unsolved	Solved
Static		

After a solution has been performed and modifications are made to the model, the “S” icon changes to (yellow). This indicates that the model has been solved but then modified

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

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

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

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

Windows

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

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

Model Views

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

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

One thing to remember is that the toolbars that are displayed depends upon what window is active. The active window is the one with the blue title bar. For example, if you are looking for the zoom toolbar button and the active window is a spreadsheet you need to select a model view first before you can access the zooming tools.

Spreadsheets

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

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

Dialogs

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

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

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

Note

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

Beams

One of RISAFloor's primary tasks is the modeling, loading, and design of beams. This includes automatic attribution of loads from the deck system, live load reduction, and design optimization for the desired code and criteria. Loads modeled on the deck are attributed to the beams and the beams transfer the loads through the floor framing to walls and columns and ultimately down to the foundation. Automatic live load reductions may also be applied. With the use of beam end releases you may define simply supported or fixed end / continuous beams.

The beams can function to resist only gravity loads, where they are analyzed and designed completely within RISAFloor, or they can function as part of a lateral load resisting system. The lateral systems can be analyzed in RISA-3D for load cases such as wind or seismic events and the beams are designed for combined gravity and lateral forces. See RISA-3D Integration for more information.

Beam data may be viewed and edited in two ways: graphically in the **Information Dialog** or in the **Beams Spreadsheet**. See Beams – Primary Data for descriptions of the beam data.

Beams can be specified as Hot Rolled Steel, Cold Formed Steel, Wood, Concrete, and Steel or Wood on beam supported floors and as Concrete only on slab supported floors. Refer to the specific sections for Steel Design, Wood Design, and Steel or Wood Product Selection for more info on the design or code checking of beams. The shape groups available for each designation are listed below.

Designation	Shape Group
Hot Rolled	Wide Flange
	Channel
	Tube
	Single Angle
	Double Angle
	Pipe
	W_Tee
Cold Formed	CU
	CS
	ZU
	ZS
	HU
Wood	Rectangular
	Rectangular Double
	Rectangular Triple
	Round
	Glulam Western
	Glulam Southern Pine
Concrete	Rectangular
	Rectangular MM (metric dimensions)
Steel Product	Steel Joist
Wood Product	Wood Joist
	TJI Joist
	LPI Joist
	BCI Joist

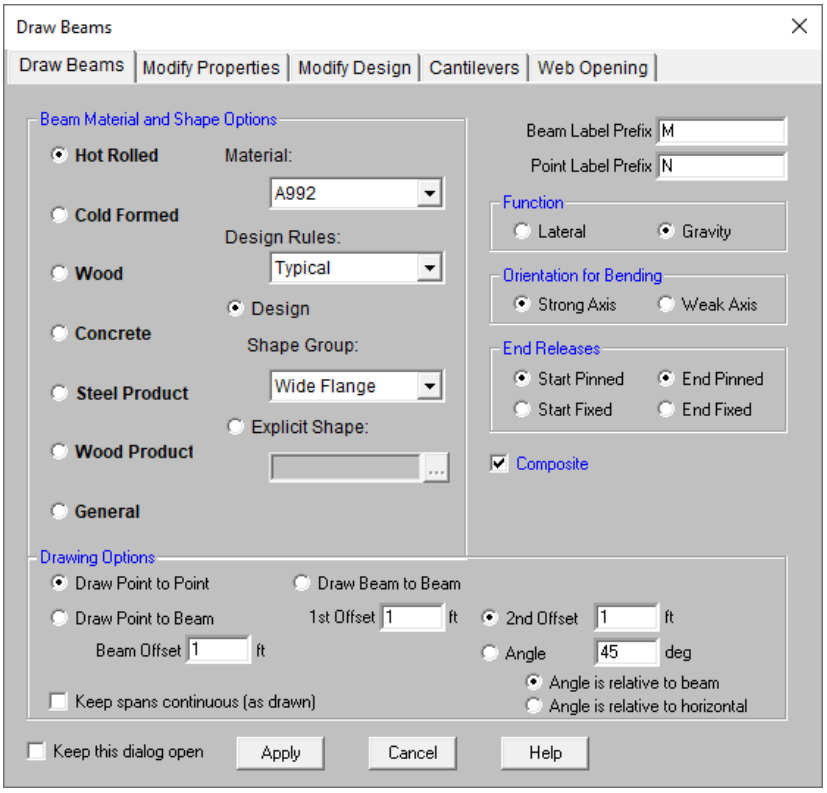
Any available material listed in the **Materials** spreadsheet can be assigned to a beam. See Material Properties for more information. All beam shapes and material properties may be assigned graphically either as you draw or later as a modification to the beams.

Drawing Beams

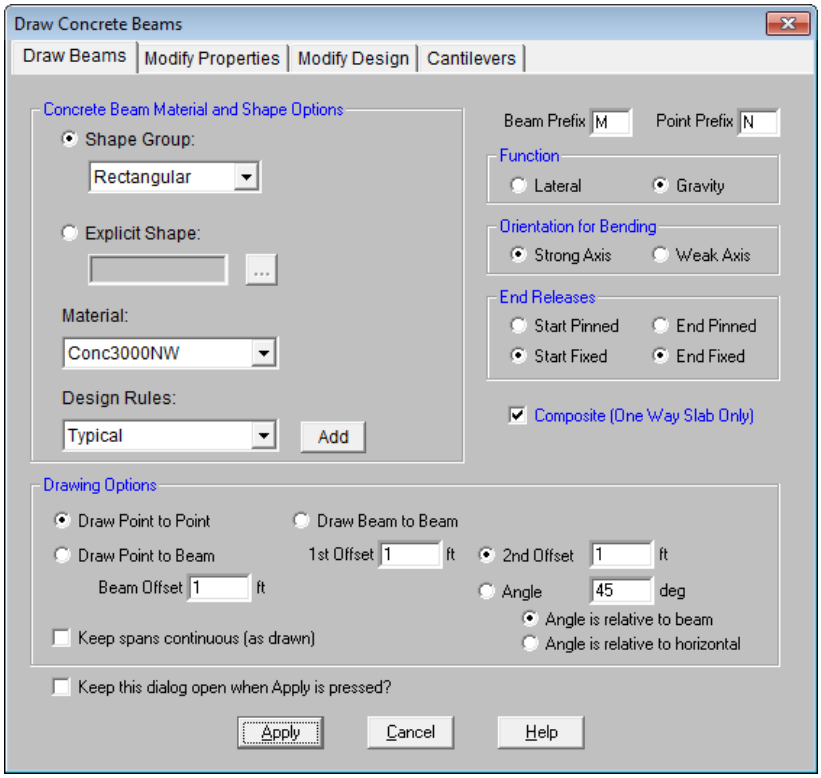
To create new beams you can draw from Point to Point, point to Beam, and Beam to Beam. **Point to Point** is when you want to draw a beam between columns, points, or a combination of column and a point. **Point to Beam** is when you draw a beam from a column or snap point to a beam with an offset distance from the beam end. **Beam to Beam** is when you draw between beams using an offset distance for both members. For **Beam Offsets** click toward the end of the member that you want the offset measured from. You can set all of the beam properties before drawing them or you can modify these properties after you draw a beam. Graphically modifying properties is discussed in the next section. See Beams– Primary Data for information on the beams and beam properties themselves.

To draw beams, click on the **Drawing Toolbar** to open the Draw Beams dialog. Enter the beam type, material, shape group, redesign rules, span type, function, orientation, end releases, and drawing option, click **Apply** and draw beams accordingly.

Beam Supported Floors:



Concrete Slab Floors:



- Composite checkbox applies to One Way Slabs Only to determine if the beam will be designed as T or L beam. If this checkbox is on- the beam will be designed T or L beam. If the checkbox is off- the beam will be rectangular shaped. It should be mentioned that composite beam design is supported in both one-way and two-way slabs in RISAFloor ES. The composite action between the beam and slab is considered in the finite element analysis for both slab types, the only difference is that the beam rebar design uses rectangular shape in two-way slab and T or L shape in one-way slab (if this checkbox is on). See Composite Beams for more information.
- If the **Keep Spans Continuous** is not checked, each beam you draw will be divided up at each beam or column intersection found along its length. The end conditions for each beam will be those specified in the **End Releases** field.
- If the **Keep Spans Continuous** IS checked, the first point you click on is the start of the beam and gets the 'Start' end release that is specified in the End Releases field. The last point you click on gets the 'End' end release that is specified in the End Releases field. For these continuous beams, all intermediate columns or beams that occur along its length will be fully fixed to the continuous beam.

As you draw, the coordinates of the nearest grid point or snap point are shown in the status bar in the lower right corner. The new beams will be shown on screen and will be recorded in the **Beams** spreadsheet.

To Draw Beams

1. If there is not a model view already open then click on the RISA toolbar to open a new view and click to turn on the Drawing toolbar if it is not already displayed.
2. If you do not already have existing columns or walls, you will need to create them first.
3. Click the button and set the beam properties.

4. Click **Apply** to start drawing beams by clicking on columns, beams, or points with the left mouse button. For Beam Offsets click toward the end of the member that you want the offset measured from. The coordinates of the closest grid point or snap point to your cursor are displayed in the lower right hand corner.

   The first click will define the Start-end of the first beam. The second click and each click thereafter will define the End-end of the first beam and also the Start-end of the next beam so that you may continue to draw as if your pencil is down. To pick up the pencil, click the right mouse button. You may then start drawing somewhere else with the left button.

5. To stop drawing altogether click the right mouse button, or press the ESC key.

Note

- Press CTRL-D to quickly recall the last dialog and make changes.
- Instead of clicking on multiple individual columns to draw beams you may also box a set of columns or a line of columns.
- You may also specify or edit beams in the **Beams** spreadsheet.
- You may undo any mistakes by clicking the Undo button.

Drawing Cantilevers

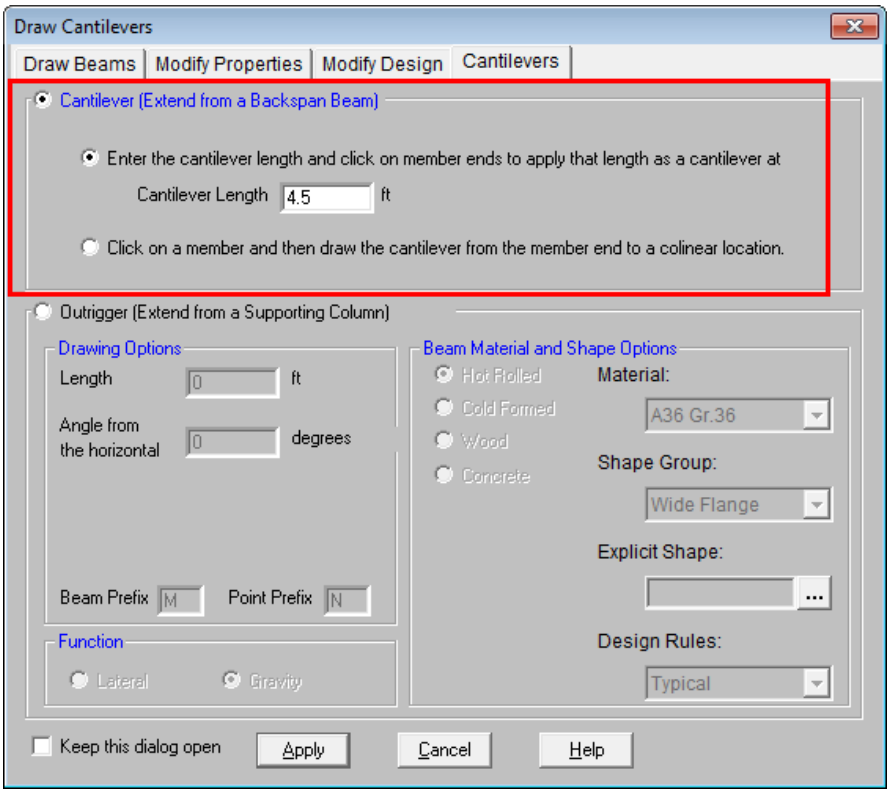
Cantilevers may only be drawn from existing beams. Use the Cantilever tool to extend a cantilever off of an existing beam.

Note:

- Cantilevers are only available on beam supported floors. Beams can be drawn "free-hand" on slab floors, so a cantilever option isn't necessary.

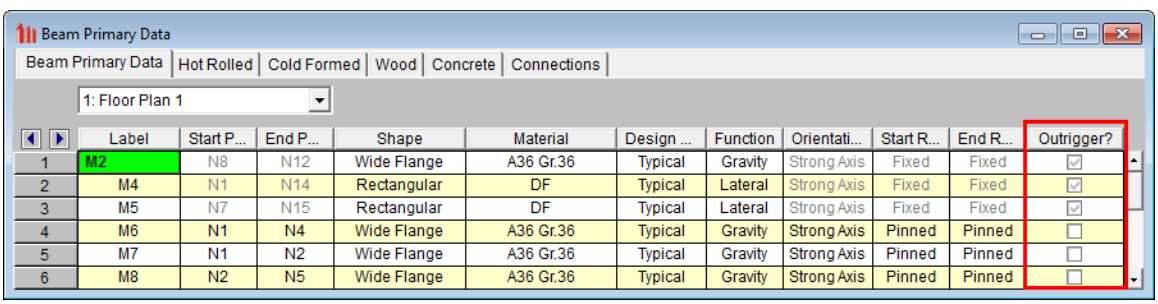
To Draw Cantilevers

1. Select the **Cantilevers** tab from the **Beams** dialog.
2. Click to select **Cantilever (Extend from a Backspan Beam)**.
3. Enter the **Cantilever Length** and click **Apply**.
4. Click toward the end of the beam that you would like to add the cantilever to. The new cantilever will be shown on screen and will be recorded in the **Beams** spreadsheet.



Drawing Outriggers

Outriggers may only be drawn from existing columns. Use the Cantilever tool to define and draw your outrigger member. Once drawn, the outrigger members will have the **Outrigger** check-box selected in the **Beams** spreadsheet to flag that they are outrigger members.



	Label	Start P...	End P...	Shape	Material	Design ...	Function	Orientati...	Start R...	End R...	Outrigger?
1	M2	N8	N12	Wide Flange	A36 Gr.36	Typical	Gravity	Strong Axis	Fixed	Fixed	☑
2	M4	N1	N14	Rectangular	DF	Typical	Lateral	Strong Axis	Fixed	Fixed	☑
3	M5	N7	N15	Rectangular	DF	Typical	Lateral	Strong Axis	Fixed	Fixed	☑
4	M6	N1	N4	Wide Flange	A36 Gr.36	Typical	Gravity	Strong Axis	Pinned	Pinned	☐
5	M7	N1	N2	Wide Flange	A36 Gr.36	Typical	Gravity	Strong Axis	Pinned	Pinned	☐
6	M8	N2	N5	Wide Flange	A36 Gr.36	Typical	Gravity	Strong Axis	Pinned	Pinned	☐

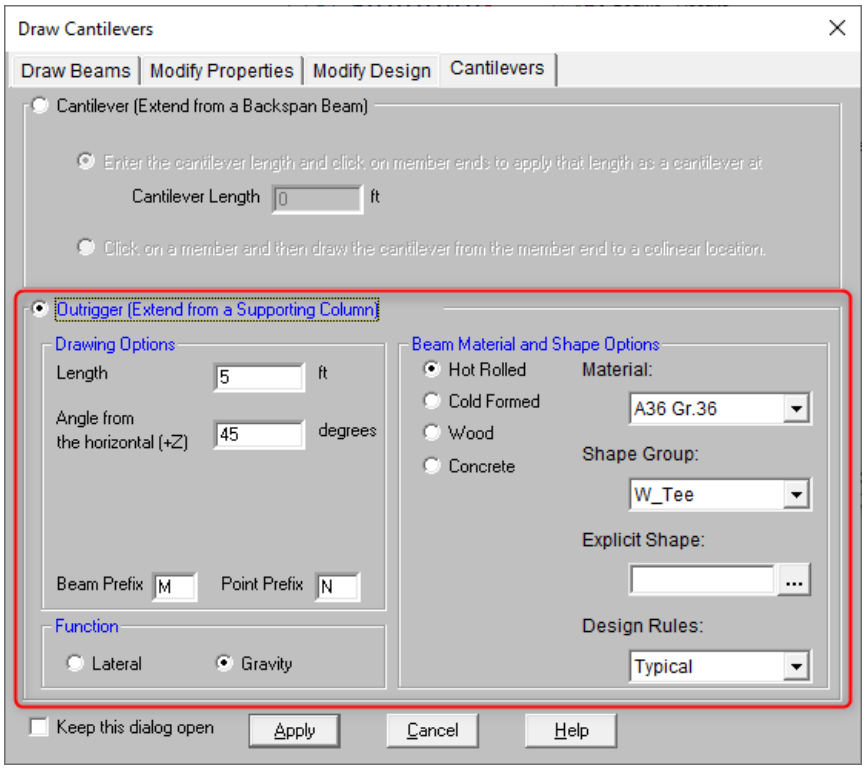
Note:

- When using outriggers, the **Use Column Stiffness** check-box (on the **Solution** tab of **Model Settings**) must be selected in order to give the outrigger a fixed connection to the column.
- Outriggers will automatically be set to strong axis orientation and fixed end releases by default. The spreadsheet can be used to edit the **Orientation** and **End Release** fields.

To Draw Outriggers

1. Select the **Cantilevers** tab from the **Beams** dialog.
2. Click to select **Outrigger (Extend from a Supporting Column)**.
3. Enter the outrigger options by specifying the length and orientation of the outrigger. Orientation of the outrigger is set by specifying the terminating angle of the end node of the outrigger with respect to the Z axis.
4. Click **Apply**.

5. Click on the column that you would like to add the outrigger to. The outrigger will be shown on screen and will be recorded in the **Beams** spreadsheet.



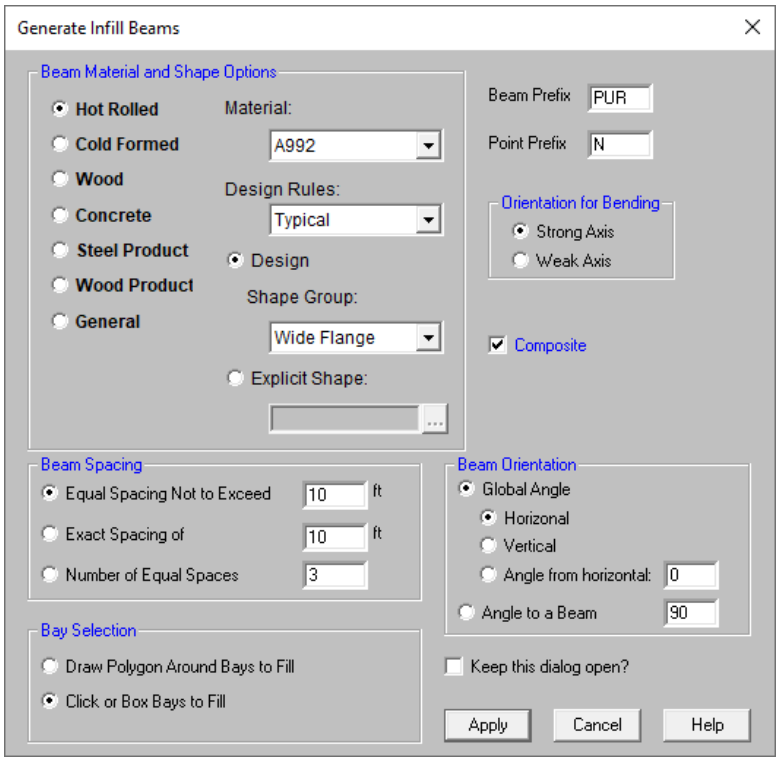
Drawing Infill Beams

Because they tend to be evenly spaced and repetitive, you would not want to draw each and every one of your infill beams or joists. RISAFloor provides special tools on beam supported floors to generate large numbers of infill beams at one time. These tools will allow you to draw an entire bay of new infill beams with a single click. You can set all of the infill beam properties up front or you can modify these properties after you generate them. The procedure for graphically modifying infill beam properties is exactly the same as discussed in the previous sections.

To draw infill beams, click on the **Drawing Toolbar** to open the Generate Infill Beams dialog. Enter the beam type, material, shape group, redesign rules, orientation for bending, beam spacing, placement orientation, and the bay selection option, click **Apply** and click the bay to be filled. You can also draw a polygon around multiple bays or simply drag a box around the bays to be filled. If the placement orientation is referenced to a global angle then no other clicks are required, the new beams will be shown on screen and will be recorded in the **Beams** spreadsheet. If the placement orientation is referenced to a member you must then click the reference member and the new beams will be created.

Note:

- Infill beams are only available on beam supported floors.



To Draw Infill Beams

1. If there is not a model view already open then click on the RISA toolbar to open a new view and click to turn on the Drawing toolbar if it is not already displayed.
2. If you do not already have existing framed bays, you will need to create them first.
3. Click the button and set the beam properties.
4. Click **Apply** to start generating beams by clicking on the bay opening or drawing a box around the bays in which you want the beams generated. If the placement orientation is referenced to a member, you must then click the reference member with the left mouse button. The coordinates of the closest grid point or snap point to your cursor are displayed in the lower right hand corner.
5. To stop drawing click the right mouse button, or press the ESC key.

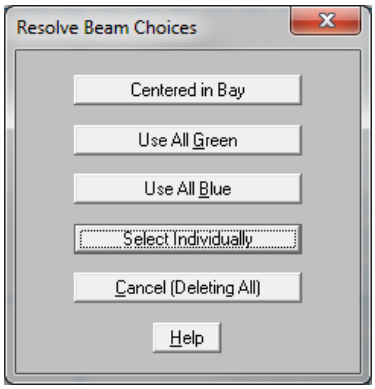
Note

- Press CTRL-D to quickly recall the dialog and make changes.
- You may also specify or edit beams in the **Beams** spreadsheet.
- You may undo any mistakes by clicking the Undo button.
- The items in the material and design rules are listed in the same order in which they appear in the Material and Design Rules spreadsheets.

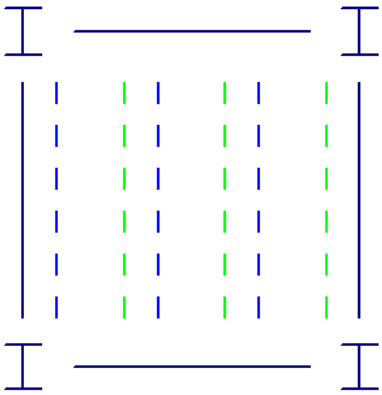
For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Infill Beams**.

Resolve Beam Choices

There are occasions where your input infill beam spacing may not perfectly fit the bay selected. In these cases you will be presented with the Resolve Beam Choices dialog.



This dialog allows you to make a manual; choice between the available spacing options, rather than having the program randomly choose one. For example, if you have a 10' x 10' bay and you ask for an exact spacing of 3 ft, you will be presented with the two options:



The green lines represent a 3 ft spacing starting from the left hand side, while the blue lines represent a 3 ft spacing starting from the right hand side. Rather than assuming one or the other, the program offers you the choice of where you want your framing located.

Choosing **Centered in Bay** will center the beams in the bay, and split the remaining space evenly on both sides.

Modifying Beam Properties

There are two ways to modify beams. You may view and edit the beam data in the **Beams Spreadsheet**.Or, you can use the **Modify Beams** tool of the **Beams** dialog to graphically modify a possibly large selection of beams. See Beams – Primary Data for information on the beams and properties themselves.

The graphical Beam Modify tool discussed here lets you modify the properties of beams that already exist in your model. To use this, you will typically specify parameters you want changed then select the beams that you want to modify. You can modify beams one at a time by selecting the **Click to Apply** option and then click on the beams you wish to modify. You may also modify entire selections of beams by selecting the beams first and then use the **Apply to Selected** option. See Graphic Selection for more on selecting or unselecting portions of your model.

To Modify Beams

1. If there is not a model view already open then click on the RISA toolbar to open a new view and click to turn on the Drawing toolbar if it is not already displayed.

2. Click the button and select the **Modify Properties** tab. Set the parameters for the changed beams. Check the **Use?** Box for the items to apply.

3. You may choose to modify a single beam at a time or an entire selection of beams.

   To modify a few beams choose **Apply Entries by Clicking Items Individually** and click **Apply**. Click on the beams with the left mouse button.

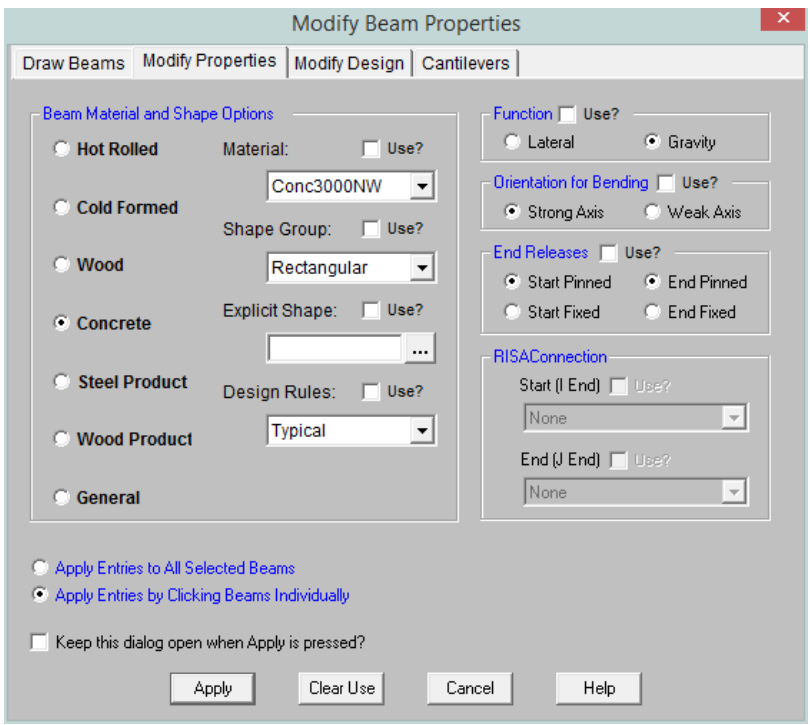
   To modify a selection, choose **Apply Entries to All Selected Items** and click **Apply**.

Note:

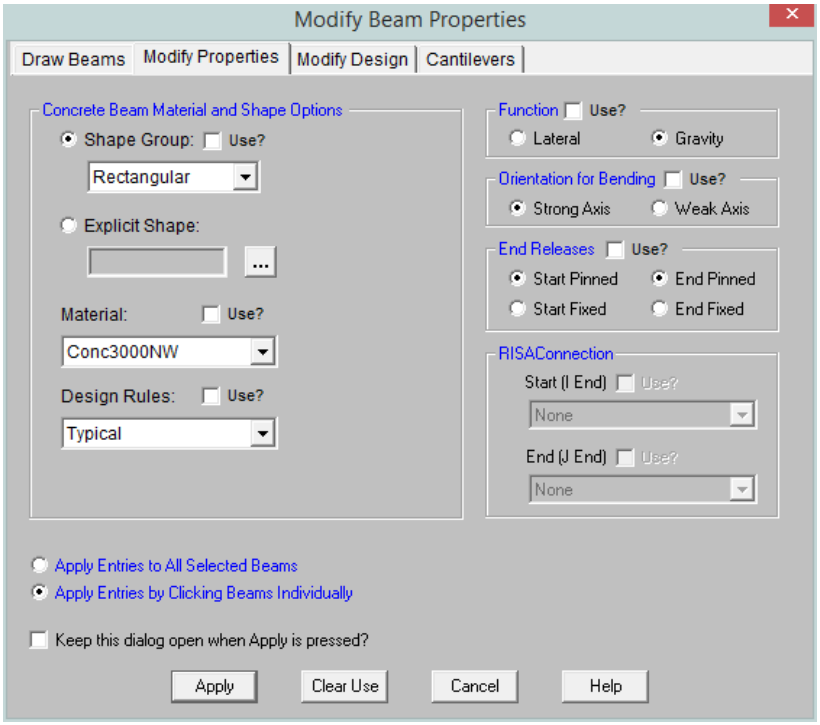
- To modify more beams with different parameters, press CTRL-D to recall the **Modify Beams** settings.
- You may also modify beams in the **Beams** spreadsheet.
- To relabel beams first sort the Beams spreadsheet into the desired order then select the **Tools** menu and choose **Relabel Beams**.
- You may undo any mistakes by clicking the Undo button.

The parameters shown are nearly the same as those used to define new beams.

Beam Supported Floors:



Concrete Slab Floors:



The **Use?** check boxes next to the data fields indicate whether the particular parameter will be used or not when the modification is applied. If the box next to a field is checked, that parameter will be applied to any selected beams. If the box is NOT checked the parameter will NOT be applied, even if a value is entered in the field. This lets you easily change one or two properties on beams without affecting the rest of the properties. Note that if the field is blank and the corresponding **Use?** box is checked, clicking **Apply** will have the effect of clearing the data for these fields.

Note:

- The RISAConnection modification options are only available for Hot-Rolled Steel and only work if you also own RISAConnection. See the RISAConnection Integration topic for more information.

Modifying Beam Design Parameters

You can graphically specify the AISC, AISI, NDS and composite design parameters such as unbraced lengths, K factors, and effective widths. See Hot Rolled Steel - Design, Cold Formed Steel - Design, Wood - Design, Concrete - Design, and Hot Rolled Steel - Composite Beam Design for information on the design parameters themselves.

To do this, you will specify the appropriate beam parameters then select the beams that you want to modify, and then click on the **Apply** button. Alternately you can set the parameters and then choose the **Click to Apply** option, which allows you to then click on the beams you want modified.

To Modify Beam Design Parameters

1. If there is not a model view already open then click on the RISA toolbar to open a new view and click to turn on the Drawing toolbar if it is not already displayed.

2. Click the button and select the **Modify Properties** tab. Set the parameters for the changed beams. Check the **Use?** Box for the items to apply.

3. You may choose to modify a single beam at a time or an entire selection of beams.

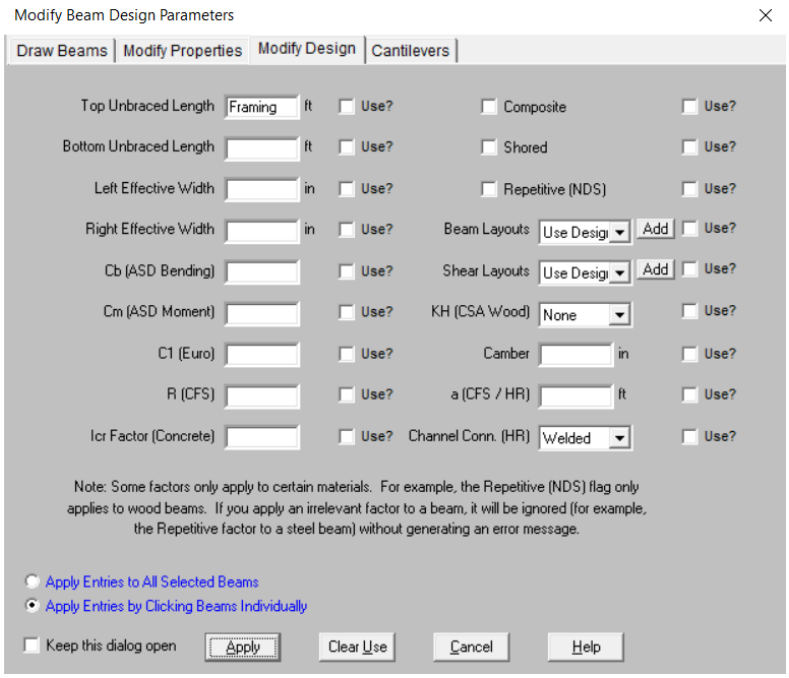
   To modify a few beams choose **Apply Entry by Clicking Items Individually** and click **Apply**. Click on the beams with the left mouse button.

   To modify a selection, choose **Apply Entries to All Selected Items** and click **Apply**.

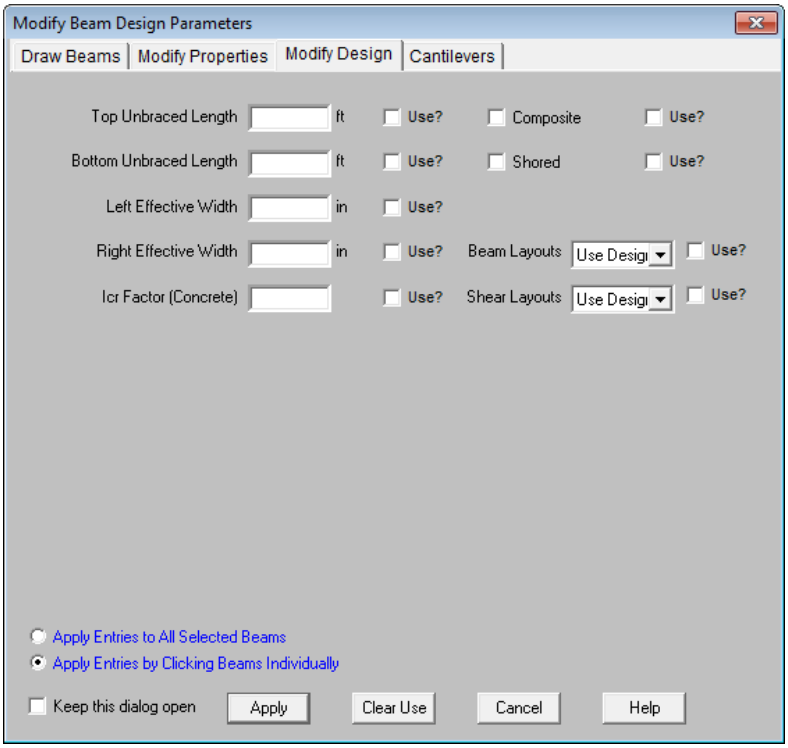
Note:

- To modify more beams with different parameters, press CTRL-D to recall the **Modify Beams** settings.
- You may also specify or edit beams in the **Beams** spreadsheet.
- You may undo any mistakes by clicking the Undo button.

Beam Supported Floors:



Concrete Slab Floors:

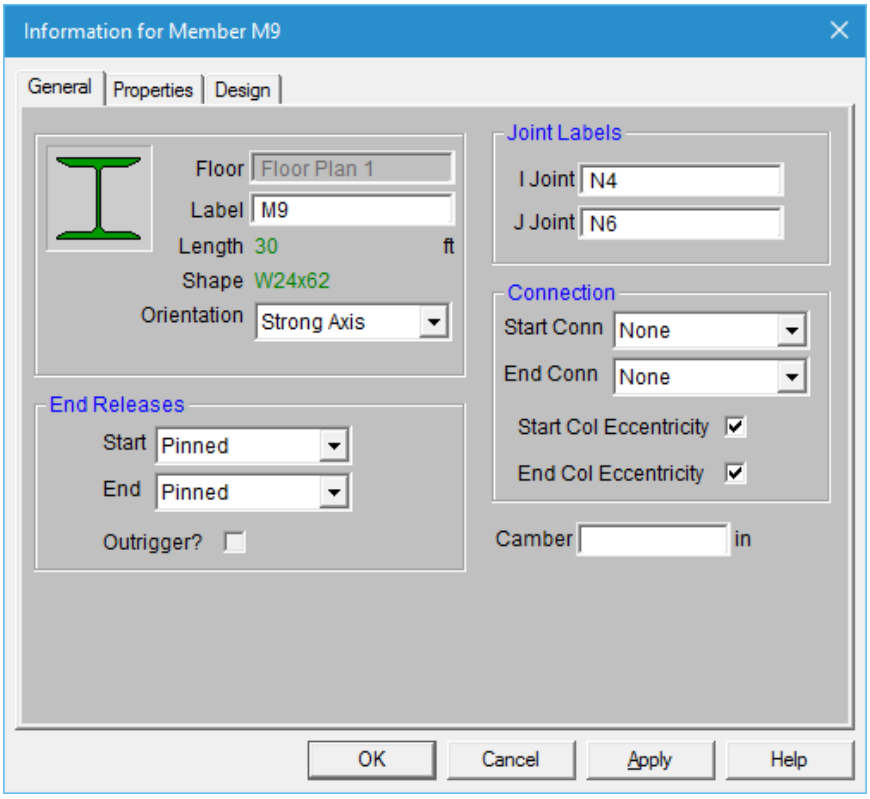


The **Use?** check boxes next to the data fields indicate whether the particular parameter will be used or not when the modification is applied. If the box next to a field is checked, that parameter will be applied to any selected beams, if the box is NOT checked, the parameter will NOT be applied, even if a value is entered in the field. This lets you easily change one or two properties on beams without affecting all the rest of the properties. Note that if the field is blank and the corresponding check box is checked, clicking **Apply** will have the effect of clearing the data for these fields.

Beam Information Dialog

Just as with the points you may double-click any beam member to view it's properties. All of the same information that is stored in the **Beams** spreadsheet is displayed for the member you choose, and may be edited. This is a quick way to view and change member properties. For large selections of members however the spreadsheet and graphic editing tools may be the faster solution.

General Tab



Floor Level - This displays the floor level that the beam is located on. This is not editable in this dialog.

Label - This allows you to view and modify the beam label.

Length - Reports the beam length.

Shape - Reports the beam shape.

Orientation - This allows you to view and modify the orientation of the beam (strong axis or week axis).

End Releases - This tab allows you to view and modify the end releases for each end of the member.

Outrigger - This check-box reports whether or not the member was originally drawn as an outrigger cantilever member. This is not editable in this dialog.

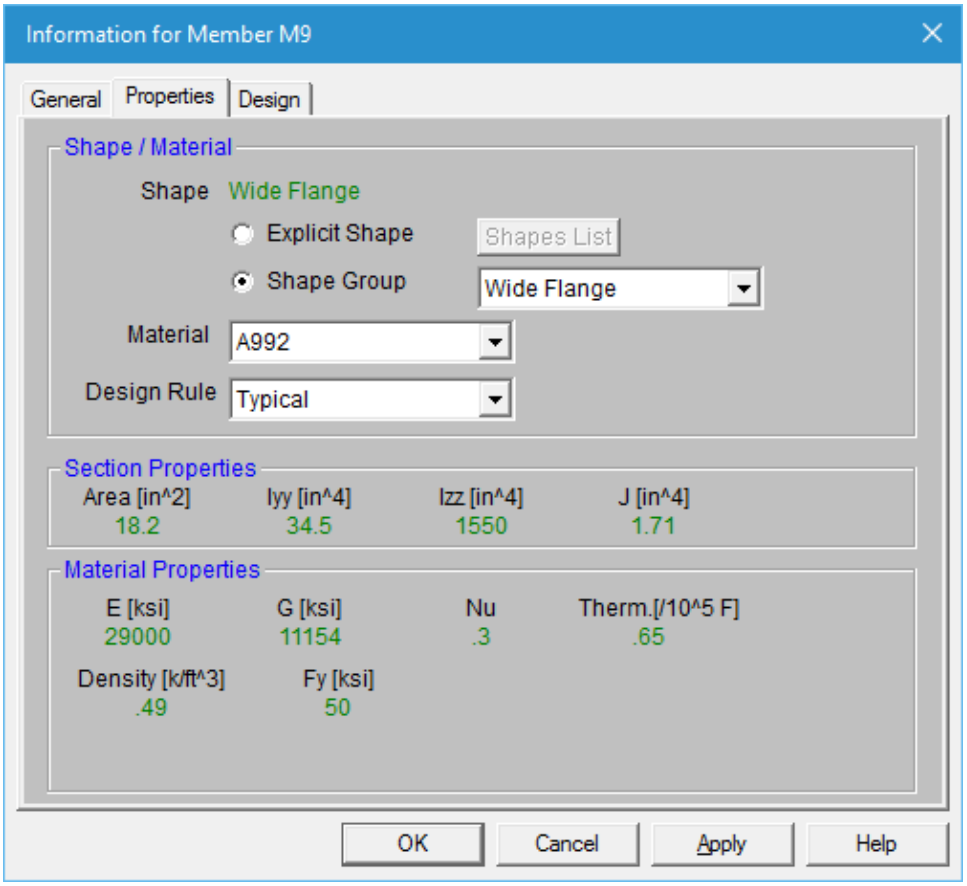
Joint Labels - This allows you to view and modify the end points of the beam.

Start/End Connection - This allows you to define Connection Rules for your hot-rolled steel connection design to be used in conjunction with RISAConnection. See the RISAConnection Integration topic for more information.

Col Eccentricity - This allows you to specify whether the eccentricity between the column centerline and the beam's end reaction resultant should contribute additional moment to the design of the column. This option is only available for pinned beam ends which frame into columns.

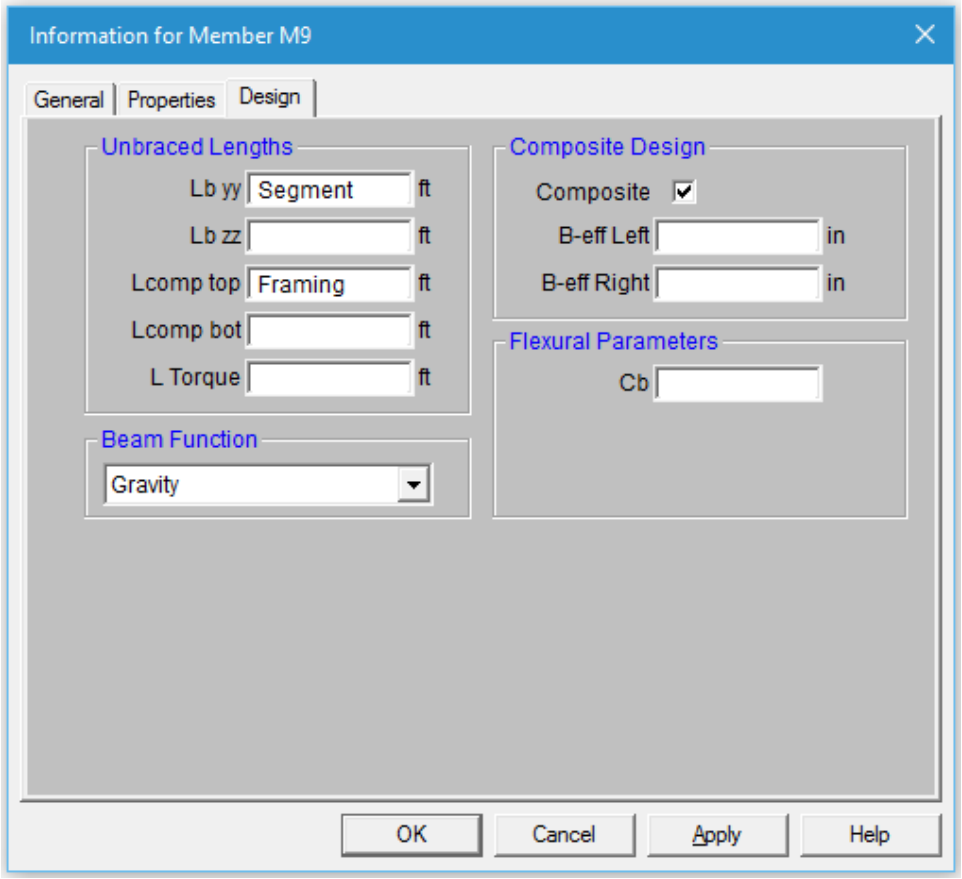
Camber - This allows you to view and modify the beam camber value.

Properties Tab



Shape/Material - This allows you to view and edit the shape properties. The associated **Section Properties** and **Material Properties** are reported below.

Design Tab



Unbraced Lengths - This allows you to view and modify the beam's unbraced length values. See the Unbraced Lengths topic for more information.

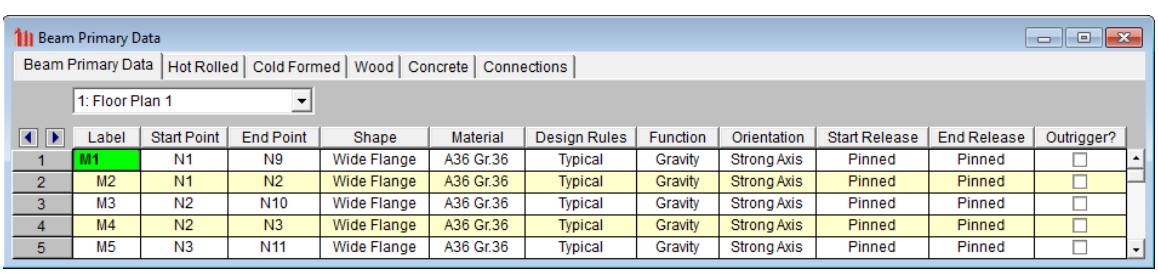
Beam Function - This allows you to view and modify the beam function.

Composite Design - This allows you to view and change the composite beam selections.

Flexural Parameters - This allows you to view and edit the flexural design parameters for member code check calculations.

Beams – Primary Data

The **Beams** spreadsheet records the properties for the beam elements and may be accessed by selecting **Beams** on the **Spreadsheets** menu. The spreadsheet has six tabs: **Primary Data**, **Hot Rolled**, **Cold Formed**, **Wood**, **Concrete** and Connections. The pull down list at the top of the spreadsheet allows you to toggle between floors.



	Label	Start Point	End Point	Shape	Material	Design Rules	Function	Orientation	Start Release	End Release	Outrigger?
1	M1	N1	N9	Wide Flange	A36 Gr.36	Typical	Gravity	Strong Axis	Pinned	Pinned	☐
2	M2	N1	N2	Wide Flange	A36 Gr.36	Typical	Gravity	Strong Axis	Pinned	Pinned	☐
3	M3	N2	N10	Wide Flange	A36 Gr.36	Typical	Gravity	Strong Axis	Pinned	Pinned	☐
4	M4	N2	N3	Wide Flange	A36 Gr.36	Typical	Gravity	Strong Axis	Pinned	Pinned	☐
5	M5	N3	N11	Wide Flange	A36 Gr.36	Typical	Gravity	Strong Axis	Pinned	Pinned	☐

All beams for a particular floor are listed on the **Beam Primary Data** tab. You may choose the floor from the drop down list at the top of the spreadsheet. The **Hot Rolled**, **Cold Formed**, **Wood**, and **Concrete** tabs list only beams that use those materials.

The entries for the **Beam Primary Data** tab are explained below:

Beam Labels

A unique label may be assigned to all of the beams manually or automatically. You can then refer to the beam by its label. You may relabel beams at any time with the **Relabel Beams** option on the **Tools** menu. The label column of the spreadsheet is displayed on all tabs of the spreadsheet.

Beam Points

Start Point and **End Point** list the labels of the start and end points of the beam. See Points for more information about points.

Shape Group & Material

Enter the shape group and material you wish to use for the beam. You can choose both by clicking on the arrow in the cell. The material pull down list presents materials listed in the **Materials** spreadsheet. See Material Properties for more information.

A Shape Group (also known as a Design List) designates the type of beams that will be used to design and optimize the beam. Refer to Appendix A - Redesign Lists for more information.

Design Rules

Enter the set of design rules you wish to use to optimize the design of the beam. You can choose by clicking on the arrow in the cell. This pull down list is related to the **Design Rules** spreadsheet. See Design Optimization for more information.

Function

The **Function** of a beam may be selected by choosing either 'Lateral' or 'Gravity' from the drop down list. Beams that are indicated as 'Lateral' will be designed for gravity loads in RISAFloor and may be optimized for lateral loads (and gravity loads) by RISA-3D. See RISA-3D Integration for more information.

Orientation

Enter the orientation you wish to use for the beam – Strong Axis or Weak Axis. You can choose by clicking on the arrow in the cell.

Note

- To be sure of a beam's orientation you can always view the rendered shape of a beam by clicking the button on the **Window** toolbar.

Start and End Releases

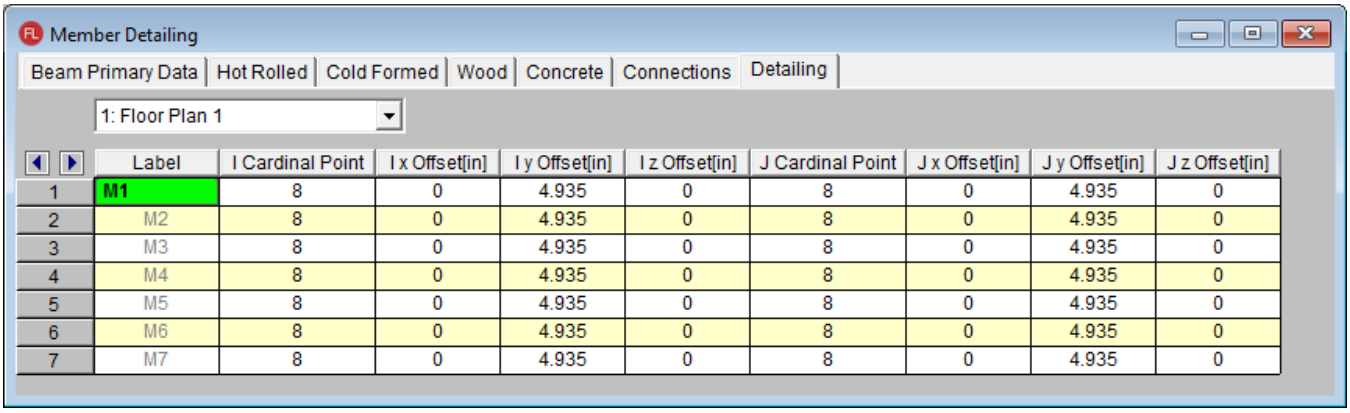
Releases control the forces that may be resisted by a beam. You may use these to define pinned connections, for simply supported gravity beams, or fixed connections. To specify beam releases go to the **Start Release** or **End Release** field for the beam and select the condition from the list.

Outrigger?

This flag denotes whether or not the beam is an outrigger. This cannot be edited within the spreadsheet, it is only set when you draw in the outrigger using the Cantilever tool.

Beams Spreadsheet - Detailing Data

The **Detailing** tab records the detailing data for the beams 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 CIS\2 Translator Page on our website at risa.com.



	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	M1	8	0	4.935	0	8	0	4.935	0
2	M2	8	0	4.935	0	8	0	4.935	0
3	M3	8	0	4.935	0	8	0	4.935	0
4	M4	8	0	4.935	0	8	0	4.935	0
5	M5	8	0	4.935	0	8	0	4.935	0
6	M6	8	0	4.935	0	8	0	4.935	0
7	M7	8	0	4.935	0	8	0	4.935	0

Note:

- The **Detailing** tab may not be visible in the default view. To turn on this option, go to **Tools - Application Settings - General** tab and select the "**Show detailing information in model**" check-box.

Beams - Analysis

RISAFloor performs an analysis of the structure on a floor by floor basis. More details on this Floor by Floor solution are given in the Solution topic. This topic covers only the solution / analysis of beams in a "Beam Supported" floor level. For elevated concrete floor slab systems refer to the Elevated Slabs - Analysis topic.

Beam Stiffness

There is a check box in the (Global) Model Settings which controls whether or not a beams shear deformation is considered in the analysis.

Note:

- Shear deformation is not explicitly considered for simply supported members that do not support other members. This limitation is imposed because it greatly speeds up the solution without greatly affecting the results.

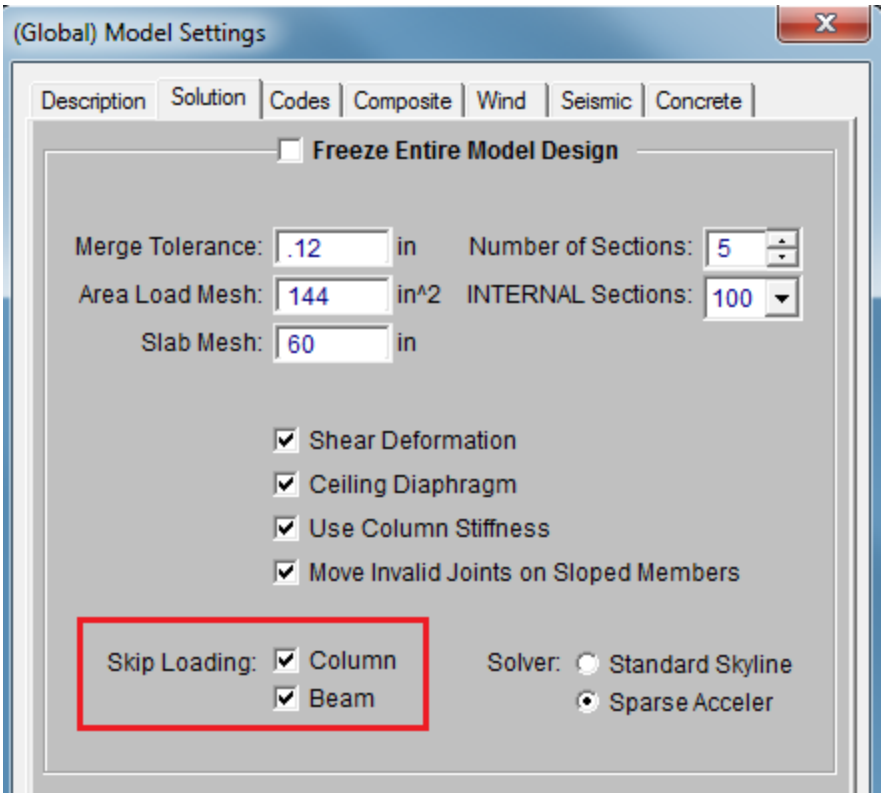
Column and Wall Stiffness

Columns and Walls are assumed to have a rigid vertical stiffness during the solution. The RISAFloor solution does not consider any flexural or rotational stiffness for walls.

When you include the column rotational stiffness in the RISAFloor analysis (see the setting on Model Settings), the program will use a rotational spring for the column based on the orientation and height of the column between floors.

Multi-Span BeamSkip Loading

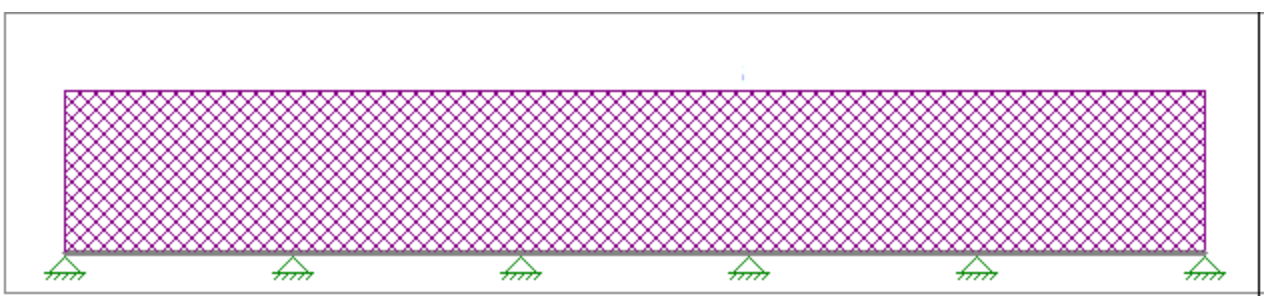
This feature is only enabled when it has been activated in the (Global) Model Settings. It applies to any continuous multi-span beams. If the beam is broken up into separate members for each span, the skip loading feature will NOT be available.



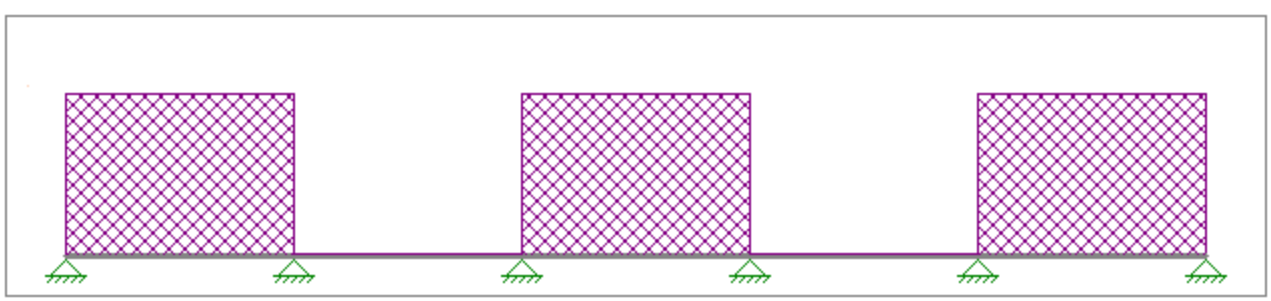
Multi-span beams are included in the overall floor solution with their full loading. However, for their individual design results, the program uses a separate internal beam solver to automatically checks for a number of different patterns for live loading on the beam. This is to ensure that the shear and moment checks account for "partial loading" per ASCE-7 where the live load varies with span.

The various live load patterns used for skip loading are shown in the images / patterns shown below. For normal span ratios, these patterns should identify the worst case load condition for moment and shear. However, when span lengths vary significantly, this is not guaranteed.

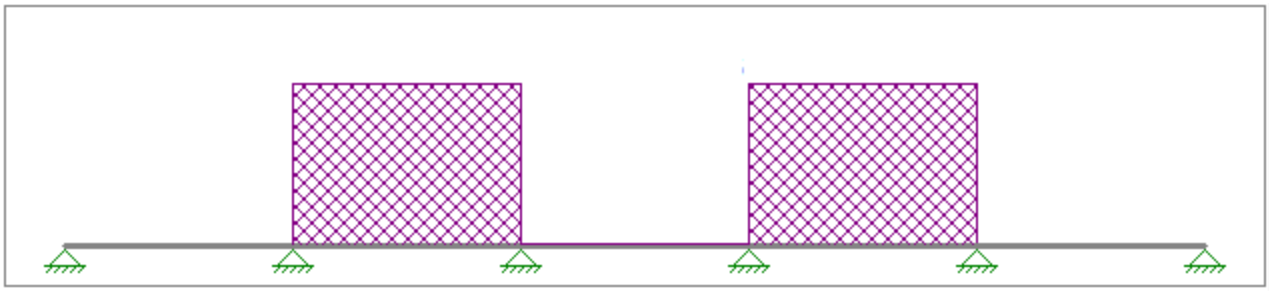
Pattern a: Full load on the beam.



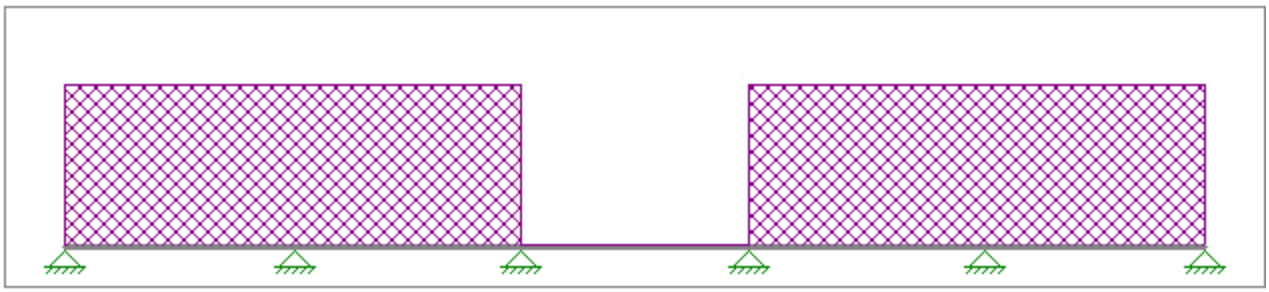
Pattern b: Every other bay is skipped. This is Intended to produce the maximum positive moments for center span (and every other odd span)



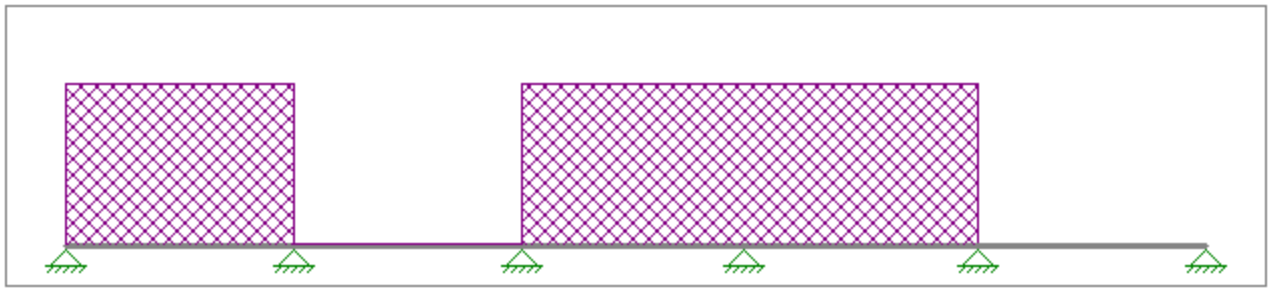
Pattern c: Every other bay is skipped. Intended to produce the maximum negative moment for cantilevers. And / or maximum positive moment on the off-center span beams (and ever other even span beam).

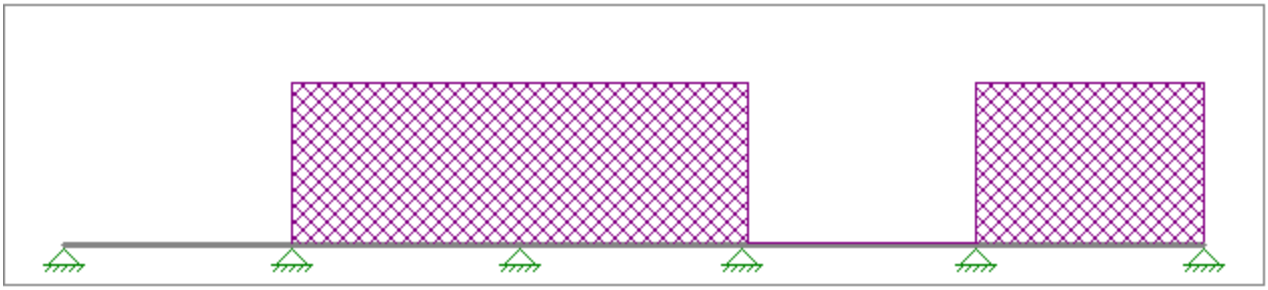


Pattern d: Two adjacent bays loaded with one bay skipped in between. This is intended to produce the maximum reaction for the first interior supports.



Pattern e: Two adjacent bays loaded with one bay skipped in between. This is intended to produce the maximum reaction for the supports that don't get covered by pattern d.





In the images above, the supports are shown as pinned. This is to represent a rigid vertical column support with no rotational stiffness. In the skip loading analysis, columns include vertical spring stiffness as well. For columns, this may also include some rotational stiffness.

Note:

- Sometimes it may be necessary for the program to produce multiple load patterns to produce the desired effect. In cases like this the load patterns will be labels 1 and 2. For example in the above images, there is an e1 and an e2 because the pattern could start from the left or right side.

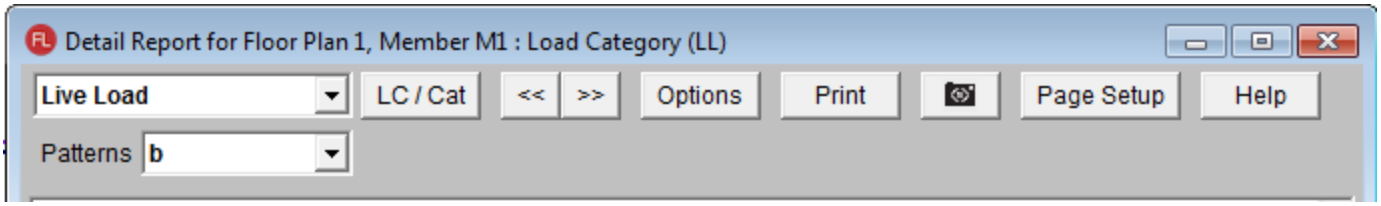
- Only the various types of live load and roof loads are skipped. This included LL, LLS, RLL, SL, SLN, and RL. This does NOT include any form of self weight, dead load, superimposed dead load or user defined OL loads.
- Skip loading is only applied to Multi-Span beams. Multiple beams that are moment connected to each other at each support are not considered for live load pattern / skip loading.
- The pattern based skip loading is only performed in RISAFloor. Any analysis done in RISA-3D on the beams will NOT be skip loaded.
- The effect of skip loading is only considered for the member itself and the end support. Results (absolute deflections) for joists that frame into a multi-span girder will always calculate their results based on the girder being fully loaded for each span.

Detail Reports

View Controls to See Live Load Patterns

When viewing the detail report for a beam that is subject to skip loading, the report may be manipulated to show you exactly what loading was used for each "Load Pattern". This is only available when viewing a live load category where skip loading is applicable (Live Load, Live Load Special, Roof Live Load, Snow Load, and Rain Load).

Merely click on the **Patterns** drop down menu to switch between displayed load patterns.



This drop down menu will affect the displayed load, shear, moment and deflection diagrams. It does NOT affect the summary code checks which are based on all the various load cases.

Detail Report Summary

The member detail report has a summary for beam identifying which pattern controlled the calculations of the reported beam results. This includes the bending check, the shear check, the deflection check and the maximum reactions at the starting and ending supports for the beam.

Beam Skip Loading

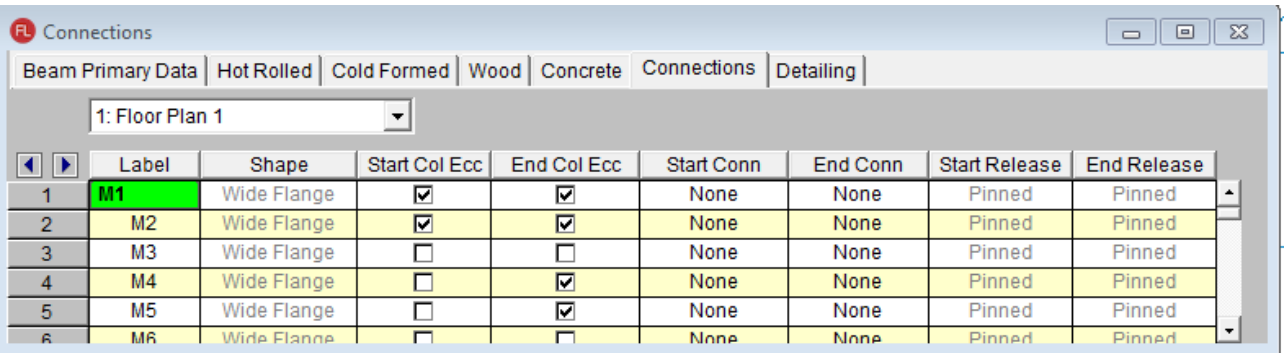
Bending Controlling LC	Bending Controlling Pattern
LC2:Service Dead + Live	**b1: positive moment (left)**
Shear Controlling LC	Shear Controlling Pattern
LC2:Service Dead + Live	**d1: interior reaction (left)**
Start Reaction LC	Start Reaction Pattern
LC2:Service Dead + Live	**b1: positive moment (left)**
End Reaction LC	End Reaction Pattern
LC2:Service Dead + Live	**b1: positive moment (left)**
Deflection Control LC	Deflection Control Pattern
DL+LL	**b1: positive moment (left)**

Note:

- The reactions transferred to internal support points do NOT consider skip loading. Instead they are always based on full loading for all spans.

Beam End Offset / Column Eccentricity

On the Connections tab of the Beams spreadsheet there are check boxes for **Start Col Ecc** and **End Column Ecc**. By default, these boxes are left unchecked when you create new beams. These check boxes do NOT affect the beam analysis, but rather only affect the eccentricities considered when skip loading a column. Refer to the Column Analysis topic for more information.



	Label	Shape	Start Col Ecc	End Col Ecc	Start Conn	End Conn	Start Release	End Release
1	M1	Wide Flange	☑	☑	None	None	Pinned	Pinned
2	M2	Wide Flange	☑	☑	None	None	Pinned	Pinned
3	M3	Wide Flange	☐	☐	None	None	Pinned	Pinned
4	M4	Wide Flange	☐	☑	None	None	Pinned	Pinned
5	M5	Wide Flange	☐	☑	None	None	Pinned	Pinned
6	M6	Wide Flange	☐	☐	None	None	Pinned	Pinned

Concrete Beams (For Beam Supported Floors Only)

The Icr Factor is used to reduce the bending stiffness of concrete beams. This is discussed in more detail in the Concrete - Design topic. The program does not consider different levels of stiffness for deflection checks. This may be over conservative as the Icracked valued used in strength design may result in larger deflections than would generally occur in the concrete members.

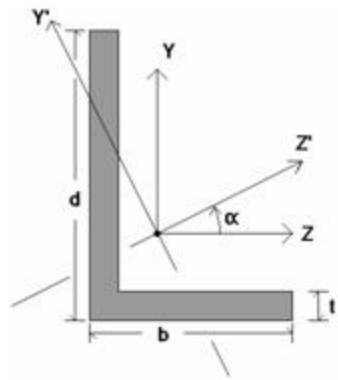
Long term deflections due to creep / shrinkage effects are NOT considered for Beam Support Floors.

Beams - Results

When the model is solved, the results are separated into material specific design results and generic results. The generic beam 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.

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:

$$Mz' = Mz * \cos(\alpha) + My * \sin(\alpha)$$

$$My' = -Mz * \sin(\alpha) + My * \cos(\alpha)$$

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.

Beam Deflection Results

The **Deflection Results Spreadsheet** presents the deflections for beam members and may be accessed by selecting **Deflection** on the **Results Menu**. The spreadsheet has three tabs: Relative Standard, Relative Alternate, and Absolute.

Relative deflections are measured from the deflected end points of the beams. Absolute deflections, however, include the deflection of the beam end points. The pull down list at the top of the spreadsheet allows you to toggle between floors.

Beam Deflections, Relative for Standard Cats

Relative Standard | Relative Alternate | Absolute

1: Floor Plan 1

	Beam	Camber[in]	PreDL[in]	PreDL Ratio	DL[in]	DL Ratio	LL[in]	LL Ratio	DL+LL[in]	DL+LL Ratio
1	B1	2	.988	789	.988	789	0	10000	.988	789
2	B2	2.75	1.137	686	1.166	669	.286	2727	1.45	538
3	B3	0	1.18	305	1.18	305	0	10000	1.18	305
4	B4	1.5	.701	1041	.772	946	.709	1030	1.48	493
5	M5	0	2.3	309	2.3	309	0	10000	2.3	309
6	M6	0	2.278	281	2.278	281	0	10000	2.278	281
7	M7	0	1.425	400	1.425	400	0	10000	1.425	400
8	M8	0	.844	592	.844	592	0	10000	.844	592
9	M9	0	.11	3872	.11	3872	0	10000	.11	3872

The sign convention is such that positive deflections are downward. The deflections are calculated at 100 locations along each beam for each load category and the maximum deflection and deflection ratio are displayed. See Results - Locations of Calculations for more information.

The **Beam Label** column displays the beam label.

The **Camber** column displays the camber required for the percentage of pre-composite dead load specified in the Model Settings. To modify camber requirements see Member Design Rules, or you can manually enter a specific camber value on the Hot Rolled Steel tab of the Members spreadsheet. For an overview on camber see Hot Rolled Steel - Design.

Note

- Camber is only considered for hot-rolled steel members.
- Camber is not considered for continuous beams, cantilevers, or outriggers.
- Camber is taken into account when giving the deflections and deflection ratios in this spreadsheet.

PreDL and PreDL Ratio

These columns display the maximum deflection along the member due to the total, unfactored pre-composite dead load. This includes the deflection due to beam self weight, deck self weight, deck construction DL and pre-composite DL minus the effect of camber. The PreDL column gives the relative deflection on the beam from its supporting members. The PreDL Ratio is the length of the member divided by the maximum PreDL deflection.

DL and DL Ratio

These columns display the maximum deflection along the member due to total unfactored dead load. This includes the deflection due to beam self weight, deck self weight, pre-composite DL and post-composite DL minus the effect of beam camber. The DL column gives the relative deflection on the beam from its supporting members. The DL Ratio is the length of the member divided by the maximum DL deflection.

LL and LL Ratio

These columns display the maximum deflection along the member due to total unfactored live load. This will be due to the summation of all of the various live load categories which can act concurrently. This includes Live Load, Live Load Special, and one of the roof loads (Roof Live Load, Snow Load or Rain Load). The LL column gives the relative deflection on the beam from its supporting members. The LL Ratio is the length of the member divided by the maximum LL deflection.

Note

- Construction live load (LLConst) is not included in the deflection ratio checks. If a category specific deflection restriction has been imposed for construction live load then it will be reported on the Relative Alternate tab.

The DL+LL and DL+LL Ratio

These columns display the maximum deflection along the member due to the sum of total unfactored DL and total unfactored LL. The DL+LL ratio is calculated as the length of the member divided by the maximum DL+LL deflection.

Alternate Deflection Design

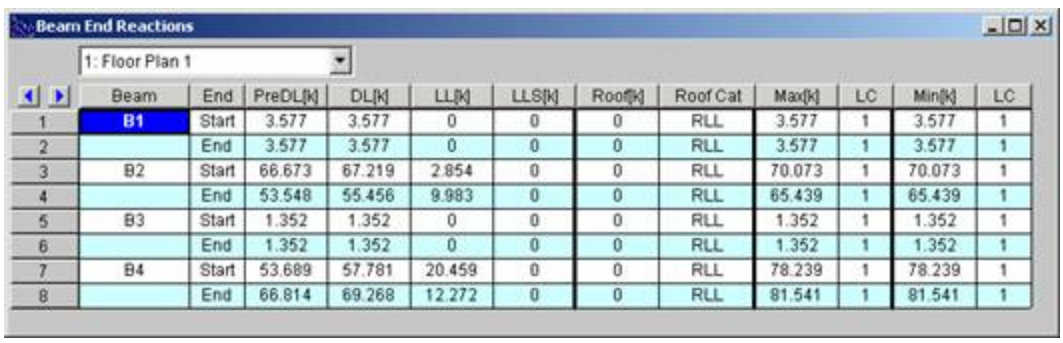
The **Relative Alternate** tab only displays results when additional categories have been specified in the design rules. The category, relative unfactored displacement, and displacement ratio of these additional requirements are listed. See Design Rules – Deflection for more information.

Absolute Deflections

The **Absolute** tab displays the absolute unfactored displacements for the categories described above. Absolute deflections include the effects of the beam end points.

Beam End Reactions

The **End Reactions Spreadsheet** records the end reactions for the beam elements and may be accessed by selecting **End Reactions** on the **Results Menu**. The pull down list at the top of the spreadsheet allows you to toggle between floors.



Beam End Reactions

1: Floor Plan 1

	Beam	End	PreDL[k]	DL[k]	LL[k]	LLS[k]	Roof[k]	Roof Cat	Max[k]	LC	Min[k]	LC
1	B1	Start	3.577	3.577	0	0	0	RLL	3.577	1	3.577	1
2		End	3.577	3.577	0	0	0	RLL	3.577	1	3.577	1
3	B2	Start	66.673	67.219	2.854	0	0	RLL	70.073	1	70.073	1
4		End	53.548	55.456	9.983	0	0	RLL	65.439	1	65.439	1
5	B3	Start	1.352	1.352	0	0	0	RLL	1.352	1	1.352	1
6		End	1.352	1.352	0	0	0	RLL	1.352	1	1.352	1
7	B4	Start	53.689	57.781	20.459	0	0	RLL	78.239	1	78.239	1
8		End	66.814	69.268	12.272	0	0	RLL	81.541	1	81.541	1

For hot-rolled steel members only load combinations with the "Connection" box checked on the Load Combinations spreadsheet will be used to calculate the displayed end reactions. See the Loads - Load Combinations topic for more details. All other beams' end reactions are based on their own material check boxes.

The sign convention assigns positive reactions to downward forces. Negative reactions, if they occur, would indicate uplift.

The **Beam Label** column displays the beam label.

The **End** column identifies for which end of the beam the reaction is being given, with the beam start and end on alternating rows.

Note

- Moment reactions for fixed end beams can be displayed graphically when using the Design Results labeling option for beams from the Model Display Options. However, these moments are not currently reported in the **Beam End Reactions** spreadsheet results, or in the End Reactions section of the beam detail report.

Dead Loads

The **PreDL** column displays the member end reaction due to the pre-composite dead load. This includes the sum of the beam self weight, deck self weight, deck construction dead load and pre-composite dead load.

The **DL** column displays the member end reactions due to the total dead load. This includes the sum of the beam self weight, deck self weight, pre-composite dead load and post-composite dead load.

Live Loads

The **LL** column displays the member end reaction due to the live load. This includes the sum of the live load and live load reducible categories. The portion of this value due to Live Load Reducible has already been reduced, thereby resulting in a final design value which should be appropriate for connection design.

Roof Loads

The **Roof** column displays the member end reaction due to the roof load. This is the maximum of the following load categories: Roof Live Load, Snow Load, Snow Load non-shedding, and Rain Load. The **Roof Cat** cell displays the controlling load category for the roof load end reactions.

Max, Min & LC

The **Max**, **Min** and **LC** columns display the maximum and minimum end reactions and the controlling load combination in which they occur.

Note:

- For hot-rolled steel members only load combinations with the "Connection" checkbox checked in the Load Combinations spreadsheet will be selected. See the Loads - Load Combinations topic for more information.

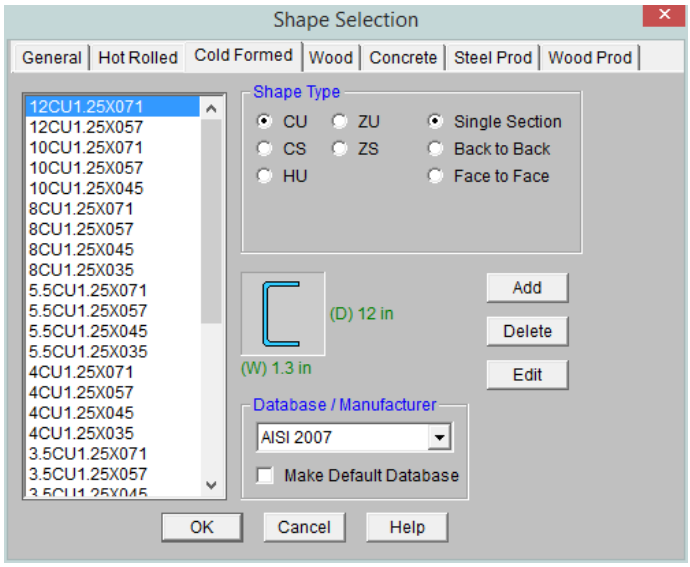
Beam Vibration Results

Vibration calculations are performed for beam members according to the requirements of *AISC Design Guide 11: Floor Vibrations Due to Human Activity*. For information on beam vibration calculations and results, refer to Vibrations - Floor Framing.

Cold Formed Steel - Databases

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

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



To Select a Cold Formed Database Shape

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

Custom vs. Manufacturer Shapes

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

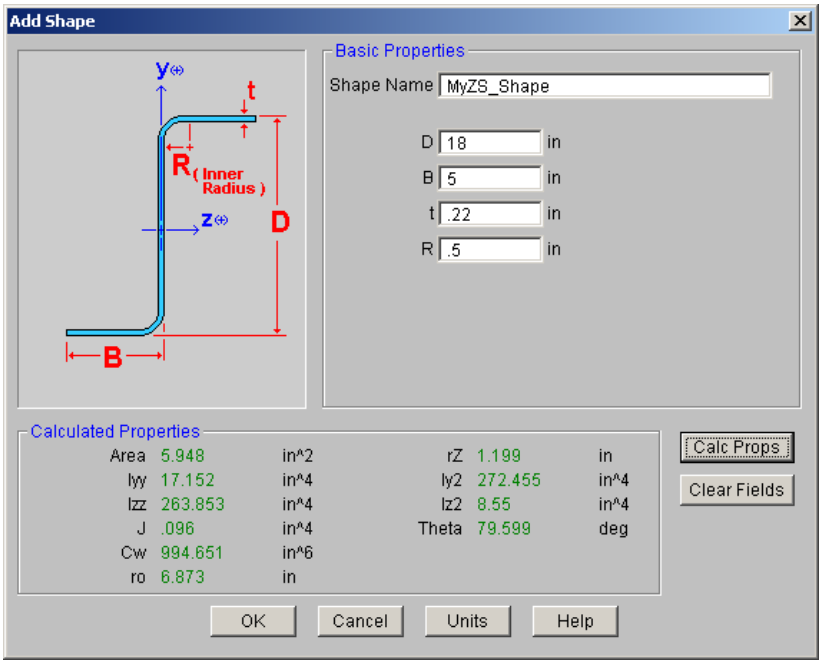
Database Files

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

To Add a Database Shape

1. On the**RISA Toolbar** click the **Edit Shape Database** button.
2. Select the **Cold Formed** tab, then select the shape type you wish to add and click the **Add** button.

3. Specify a name for the shape and fill in the **Basic Properties.**
4. Click **Calc Props** to determine the shape properties.



Note

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

Cold Formed Shape Types

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

C sections without lips (CU)

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

C sections with lips (CS)

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

Z sections without lips (ZU)

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

Z sections with lips (ZS)

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

Hat sections without lips (HU)

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

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)

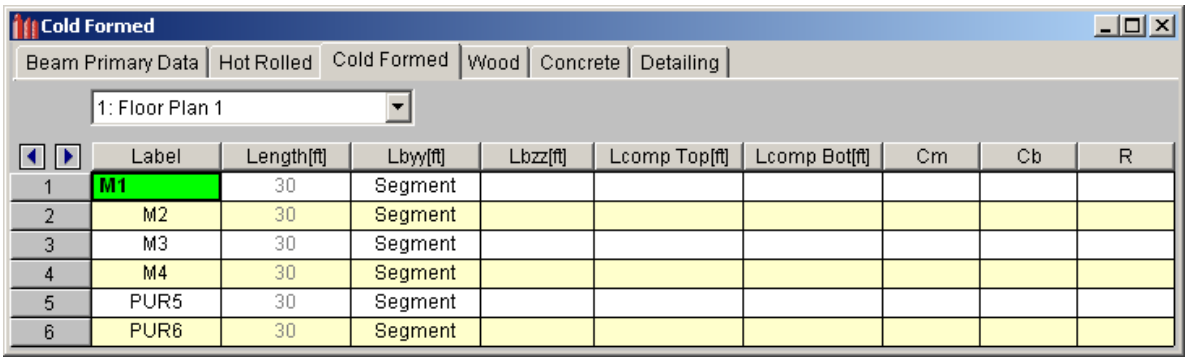
Note:

- 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.

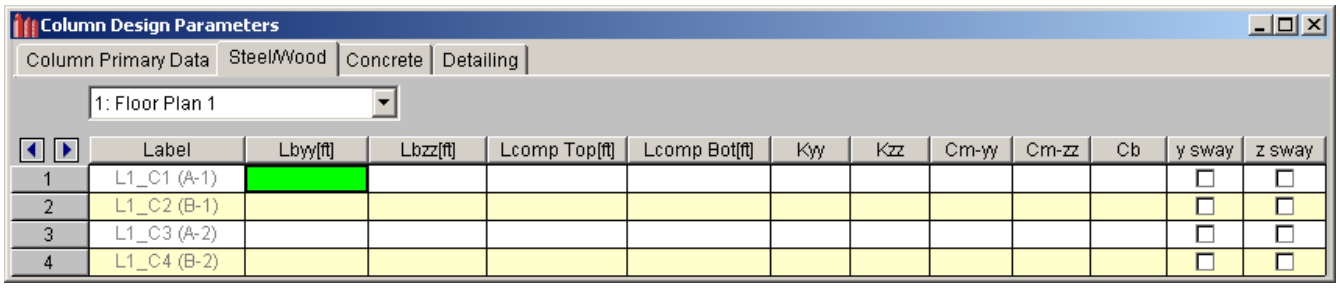
Cold formed shape properties are available in the database and the values are based on the AISI or manufacturer values, whichever is selected (See Cold Formed Steel Database). You may also input your own basic shapes and the properties will be calculated automatically.

Design Parameters

The **Cold Formed** tab on the **Beams Spreadsheet** and the **Steel/Wood** tab on the **Columns Spreadsheet** record the design parameters for the cold formed steel code checks. These parameters may also be assigned graphically. SeeModifying Beam Design or Modifying Column Design to learn how to do this.



	Label	Length[ft]	Lbyy[ft]	Lbzz[ft]	Lcomp Top[ft]	Lcomp Bot[ft]	Cm	Cb	R
1	M1	30	Segment						
2	M2	30	Segment						
3	M3	30	Segment						
4	M4	30	Segment						
5	PUR5	30	Segment						
6	PUR6	30	Segment						



	Label	Lbyy[ft]	Lbzz[ft]	Lcomp Top[ft]	Lcomp Bot[ft]	Kyy	Kzz	Cm-yy	Cm-zz	Cb	y sway	z sway
1	L1_C1 (A-1)										☐	☐
2	L1_C2 (B-1)										☐	☐
3	L1_C3 (A-2)										☐	☐
4	L1_C4 (B-2)										☐	☐

These parameters are defined for each cold formed member. The entries are explained below. The pull down list at the top of the spreadsheet allows you to toggle between floors.

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. The Label field for column members is dictated by the label entry on the **Column Stacks Spreadsheet** and may not be edited here. If you would like to edit this entry, you must do so on the Column Stacks Spreadsheet.

Length

The beam **Length** is reported in the second column. This value may not be edited as it is dependent on the beam start and end points 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. For column members, each segment of a physical column will receive its own calculated value. However, when in RISA-3D via RISAFloor, the *highest* Cm value calculated for a column segment in RISAFloor will be used for the entire physical column in RISA-3D.

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 value is currently not used by RISAFloor for beam members and is in place for use with the future implementation of sloping floor and roof systems. However, Cm values for beam members are "passed" to RISA-3D when accessed via the Director button in RISAFloor.
- In RISAFloor, beam members are assumed to be part of a non sway frame and subsequently braced against sway.

Cb - Bending Coefficients

For the cold formed codes, **Cb Coefficients** are used in the calculation of the nominal flexural strength, Mn. If this entry is left blank, it will be calculated automatically. For column members, each segment of a physical column will receive its own calculated value. However, when in RISA-3D via RISAFloor, the *lowest* Cb value calculated for a column segment in RISAFloor will be used for the entire physical column in RISA-3D.

R Value

The **R Value** for cold formed steel design is described in Section I6.2.1 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 I6.2.1-1 in the AISI Supplement for the actual values).

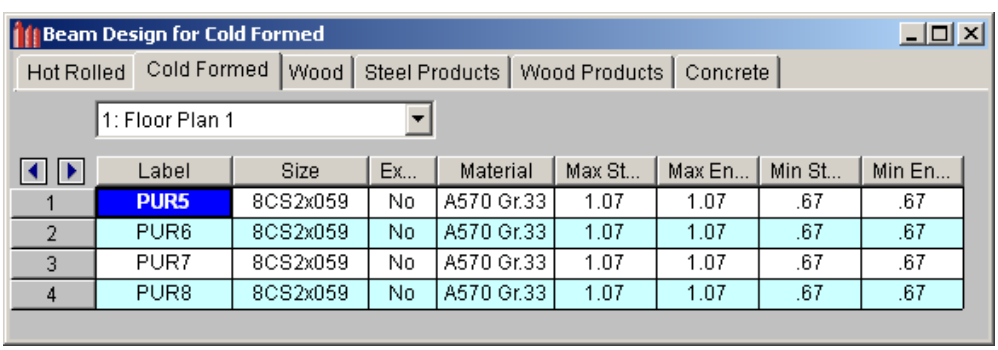
If a value is entered by the user, that value will be used by the program in the moment capacity calculation of the member. There are a number of restrictions that must be met to use this section of the code for moment capacity and the user is responsible to check that these restrictions are satisfied.

Note:

- If the R value is entered, the program will use Section I6.2.1 to check moment capacity calculations.

Design Results

The **Design Results Spreadsheet** displays the optimized design results for the beam elements and may be accessed by selecting **Designs** on the **Results** menu. The spreadsheet has six tabs: Hot Rolled, Cold Formed, Wood, Steel Products, Wood Products, and Concrete. The pull down list at the top of the spreadsheet allows you to toggle between floors.



	Label	Size	Ex...	Material	Max St...	Max En...	Min St...	Min En...
1	PUR5	8CS2x059	No	A570 Gr.33	1.07	1.07	.67	.67
2	PUR6	8CS2x059	No	A570 Gr.33	1.07	1.07	.67	.67
3	PUR7	8CS2x059	No	A570 Gr.33	1.07	1.07	.67	.67
4	PUR8	8CS2x059	No	A570 Gr.33	1.07	1.07	.67	.67

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

The **Size** 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 or deflection requirements (see Design Optimization), or adding more shapes to the available Redesign List (see Appendix A – Redesign Lists).

The **Explicit** column displays "Yes" if the beam has been locked to an explicit beam size by the user. When you have chosen a specific shape to override the programs automatic redesign, that beam becomes "locked" and will not be automatically redesigned by the program.

Note

- To "unlock" a beam, you can use the beam – modify tool to assign a shape group. If the model has already been solved, you may optimize a beam by using the Member Redesign dialog. See Member Redesign for more details.
- Live load reduction effects are considered for the applicable values in this spreadsheet.

The **Material** Column displays the material label assigned to the beam.

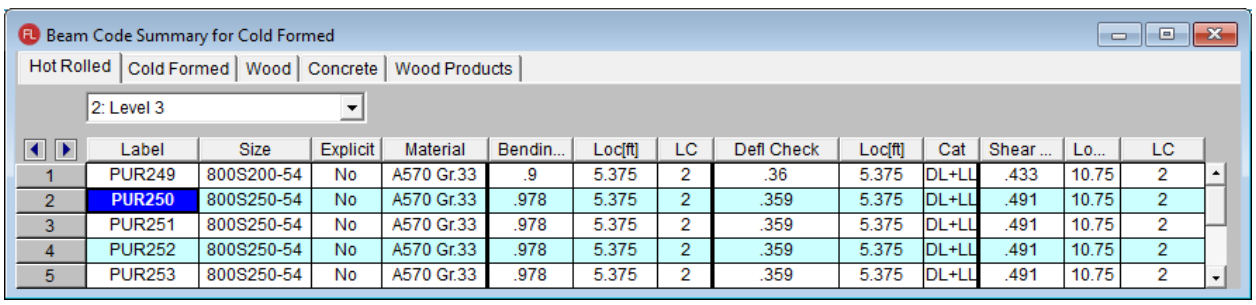
End Reactions

The **Max Start & End Reactions** column displays the maximum start and end reactions of the beam for ALL load combinations. If "Show Factored End Reactions" in **Model Settings** is left unchecked, these displayed forces are not factored. If it is checked, then the displayed forces will have been multiplied by the factors in the load combinations. The sign convention assigns positive reactions to downward forces. Negative reactions, if they occur, would indicate uplift.

The **Min Start & End Reactions** column displays the minimum start and end reaction of the beam.

Code Checks

The **Code Checks Spreadsheet** summarizes the code check results for the beams and may be accessed by selecting **Code Checks** on the **Results** menu. The spreadsheet has five tabs: Hot Rolled, Cold Formed, Wood, Concrete, and Wood Products. The pull down list at the top of the spreadsheet allows you to toggle between floors.


Beam Code Summary for Cold Formed

Hot Rolled | Cold Formed | Wood | Concrete | Wood Products

2: Level 3

	Label	Size	Explicit	Material	Bendin...	Loc[ft]	LC	Defl Check	Loc[ft]	Cat	Shear ...	Lo...	LC
1	PUR249	800S200-54	No	A570 Gr.33	.9	5.375	2	.36	5.375	DL+LL	.433	10.75	2
2	PUR250	800S250-54	No	A570 Gr.33	.978	5.375	2	.359	5.375	DL+LL	.491	10.75	2
3	PUR251	800S250-54	No	A570 Gr.33	.978	5.375	2	.359	5.375	DL+LL	.491	10.75	2
4	PUR252	800S250-54	No	A570 Gr.33	.978	5.375	2	.359	5.375	DL+LL	.491	10.75	2
5	PUR253	800S250-54	No	A570 Gr.33	.978	5.375	2	.359	5.375	DL+LL	.491	10.75	2

The **Label** column displays the beam label.

The **Size** column displays the beam 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 **Explicit** column displays "Yes" if the beam has been locked to an explicit beam size by the user. When you have chosen a specific shape to override the programs automatic redesign, that beam becomes "locked" and will not be automatically redesigned by the program.

Note

- To "unlock" a beam, you can use the beam – modify tool to assign a shape group. If the model has already been solved, you may optimize a beam by using the Member Redesign dialog. See Member Redesign for more details.

The **Material** Column displays the material label assigned to the beam.

Bending, Shear, and Deflection Checks

The **Bending Check** and **Shear Check** columns display the maximum bending check and shear check calculated by the program. This value is equal to the actual bending or shear demand (stress or force) divided by the actually beam resistance (allowable stress or ultimate capacity). You can see the details of these values in the **Bending Results** or **Shear Results** spreadsheet. This check is calculated at 100 stations along each beam for each load combination and the maximum check is reported.

Note

- In the Bending Results spreadsheet, the reported unity value is not solely based on bending analysis. The Bending Unity is determined based on the combined effects of both bending and shear, as per the equation H2-1 specified in AISI S100. This equation takes into account the interaction between bending and shear stresses to provide a comprehensive assessment of the member's structural integrity.

The **Deflection Check** displays the maximum deflection check. This value is equal to the ratio of actual deflection to allowable deflection. You can see the details of these values in the **Deflection Results** spreadsheet. This check is calculated at 100 stations along each beam and the maximum check is reported. See Beam Results - Deflection for more information.

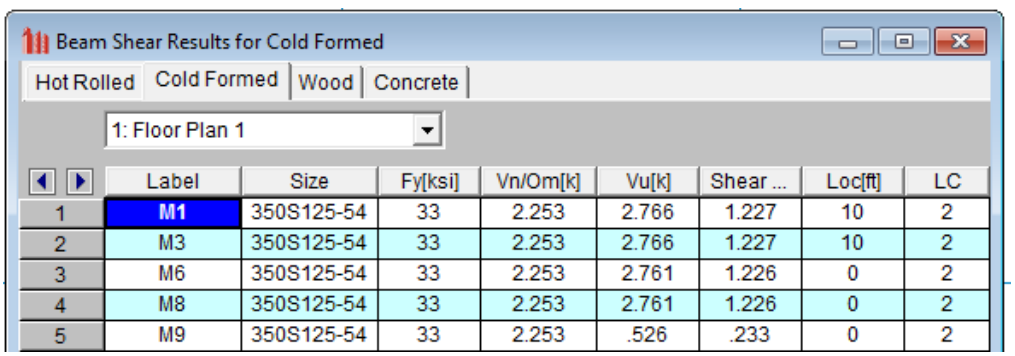
The **Location** columns display the location along the member where the maximum bending, shear, or deflection check occurs.

The **LC** column displays the controlling load combination which resulted in the maximum bending or shear check.

Deflection checks are based on Load Categories (Dead, Live or DL+LL), not Load Combinations. Therefore, the controlling load Category for deflections is reported in the **Cat** column.

Shear Results

The **Shear Results Spreadsheet** records the shear results for the beam elements and may be accessed by selecting **Shear** on the **Results** menu. The spreadsheet has four tabs: Hot Rolled, Cold Formed, Wood, and Concrete. The pull down list at the top of the spreadsheet allows you to toggle between floors.



	Label	Size	Fy[ksi]	Vn/Om[k]	Vu[k]	Shear ...	Loc[ft]	LC
1	M1	350S125-54	33	2.253	2.766	1.227	10	2
2	M3	350S125-54	33	2.253	2.766	1.227	10	2
3	M6	350S125-54	33	2.253	2.761	1.226	0	2
4	M8	350S125-54	33	2.253	2.761	1.226	0	2
5	M9	350S125-54	33	2.253	.526	.233	0	2

The **Label** column displays the beam label.

The **Size** column displays the beam size. When no adequate member could be found from the available shapes, this field will display the text “not designed”. When this occurs, 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 **Fy** column displays the yield strength of the material.

Note:

- Live load reduction effects are considered for the applicable values in this spreadsheet.

Shear Stress and Capacity

The **Vn/Om or phi*Vn** column displays the calculated allowable shear capacity based on Section C3 of the applicable AISI code. The **Vu** column displays the maximum actual shear demand that the member experiences.

The **Shear Check** column displays the maximum shear check. This value is equal to the actual shear demand (force) divided by the actual beam resistance (ultimate capacity). This shear check is calculated at 100 locations along each beam for each load combination. The maximum shear stress, its location, and the controlling load combination are reported.

The **Location** column displays the location along the member where the maximum shear check occurs.

The **LC** column displays the controlling load combination which resulted in the maximum shear check.

Bending Results

The **Bending Results Spreadsheet** records the bending results for the beams and may be accessed by selecting **Bending** on the **Results** menu. The spreadsheet has four tabs: Hot Rolled, Cold Formed, Wood, and Concrete. The pull down list at the top of the spreadsheet allows you to toggle between floors.

Beam Bending Results for Cold Formed

Hot Rolled | Cold Formed | Wood | Concrete

1: Floor Plan 1

	Label	Size	Fy[ksi]	Lb Top...	Lb Bot[ft]	Cb	Cm	Mn/Om[k-ft]	Mu[k-ft]	Bend...	Loc[ft]	LC	R(...	Equati...
1	M1	350S125-54	33	1	10	1	1	.563	7.595	13.491	5	2	1	C5.1.1-1
2	M3	350S125-54	33	1	10	1	1	.563	7.595	13.491	5	2	1	C5.1.1-1
3	M6	350S125-54	33	1	10	1	1	.563	7.596	13.493	5	2	1	C5.1.1-1
4	M8	350S125-54	33	1	10	1	1	.563	7.596	13.493	5	2	1	C5.1.1-1
5	M9	350S125-54	33	3	10	1	1	.527	1.363	2.643	4.612	2	1	C5.2.1-2
6	M11	350S125-54	33	3	10	1	1	.527	1.363	2.643	4.612	2	1	C5.2.1-2

The **Label** column displays the beam label.

The **Size** column displays the beam size. When no adequate member could be found from the available shapes, this field will display the text "not designed". When this occurs, 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 **Fy** column displays the yield strength of the material.

Note:

- Live load reduction effects are considered for the applicable values in this spreadsheet.

Unbraced Lengths

The **Lb top** and **Lb bottom** columns display the unbraced lengths associated with the controlling bending check. See Unbraced Lengths for more information on how these values are calculated.

Steel Design Coefficients

The **Cb** and **Cm** columns display the bending coefficients associated with the controlling bending check. More information on how these values are calculated was presented previously in this chapter.

Bending Stress and Capacity

The **Mn/Om or phi*Mn** column displays the calculated allowable bending moment capacity based on Section C3 of the applicable AISI code. The **Mu** column displays the maximum actual bending moment demand that the member

experiences. The sign convention is defined so that positive bending will result in tension in the bottom fiber.

The **Bending Check** column displays the maximum bending check calculated by the program. This check is equal to the actual demand (bending moment) divided by the beam resistance (moment capacity). This bending check is calculated at 100 locations along each beam for each load combination and the maximum demand (moment), the location, and the controlling load combination are reported.

The **Location** column displays the location along the member where the maximum bending check occurs.

The **LC** column displays the controlling load combination which resulted in the maximum bending check.

The **Equation** column displays the code equation that controlled in the calculation of the bending check.

Column Results

The **Column Results Spreadsheet** summarizes the code check results and records the design results for columns and may be accessed by selecting **Column Results** on the **Results** menu. The spreadsheet has four tabs: Hot Rolled, Cold Formed, Wood, and Concrete.

AISI Cold Formed Steel Column Code Checks

Hot Rolled | Cold Formed | Wood | Concrete

	Stack	Lift	Shape	Code Check	Elev[ft]	LC	Shear Check	Elev[ft]	Dir	L...	phi*Pn[k]	phi*Tn[k]	phi*Mnyy[...	phi*Mnzz[k-ft]	Cb	Cmyy	Cmzz	R...	AISI Eqn
1	CS1 (A-1)	1	4CS4X060	.745	0	2	.000	0	y	2	4.477	21.889	.977	1.715	1	.6	.6	1	C5.2.2-1
2	CS2 (B-1)	1	4CS4X075	.921	0	2	.000	0	y	2	3.637	27.176	.811	2.023	1	.6	.6	1	C5.2.2-1
3	CS4 (A-2)	1	9CS3X060	.675	0	2	.000	0	y	2	9.049	27.235	.98	3.676	1	.6	.6	1	C5.2.2-1
4	CS5 (B-2)	1	11CS3.75X06(	.725	0	2	.000	0	y	2	8.459	33.472	1.42	4.629	1	.6	.6	1	C5.2.2-1
5	CS7 (A-3)	1	4CS4X060	.746	0	2	.000	0	y	2	4.477	21.889	.977	1.715	1	.6	.6	1	C5.2.2-1
6	CS8 (B-3)	1	4CS4X075	.923	0	2	.000	0	y	2	3.637	27.176	.811	2.023	1	.6	.6	1	C5.2.2-1

The **Stack** column displays the column stack label.

The **Lift** column displays the lift number for the physical column. Lift No. 1 is the lowermost physical column in a stack and the lifts are numbered sequentially moving up the column stack.

The **Shape** column displays the physical column 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).

Code Checks

The **Code Check** column displays the maximum combined axial and bending check calculated by the program. This value is equal to the actual combined axial and bending demand (stress or force) divided by the actually column resistance (allowable stress or ultimate capacity). You can see the details of this value in the subsequent **Axial Resistance (Pn/Om** or **phi*Pn** and **Tn/Om** or **phi*Tn)** and **Flexural Resistance (Mnyy/Om & Mnzz/Om** or **phi*Mnyy & phi*Mnzz)** columns of this spreadsheet. This check is calculated at 100 stations along each physical column for each load combination and the maximum check is reported.

The **Shear Check** column displays the maximum shear check calculated by the program. This value is equal to the actual shear demand (stress or force) divided by the actual column resistance (allowable stress or ultimate capacity). This check is calculated at 100 stations along each column for each load combination and the maximum check is reported.

The **Elev** columns display the absolute elevation along the column stack where the maximum code check occurs.

The **LC** column displays the controlling load combination which produced the maximum code check and/or shear check.

The **Dir** column displays the column local axis along which the maximum shear check occurs.

Column Stress and Capacity

The **Pn/Om** (ASD) or **phi*Pn** (LRFD) column displays the calculated allowable compressive capacity based on Section C4 of the applicable AISI code.

The **Tn/Om** (ASD) or **phi*Tn** (LRFD) column displays the calculated allowable tensile capacity based on Section C2 of the applicable AISI code.

The **Mnyy/Om** and **Mnzz/Om** (ASD) or **phi*Mnyy** and **phi*Mnzz** (LRFD) columns display the calculated allowable bending moment capacity based on Section C3 of the applicable AISI code.

The **AISI Eqn** column displays the code equation used in the calculation of the controlling code check.

Steel Design Coefficients

The **Cb**, **Cmyy**, and **Cmzz** columns display the bending coefficients associated with the controlling bending check. More information on how these values are calculated was presented previously in this chapter.

Assumptions and Limitations

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

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

Torsional warping effects are not included. Web crippling is not considered.

See the **Unbraced lengths** for Cold Formed Steel section for the full details.

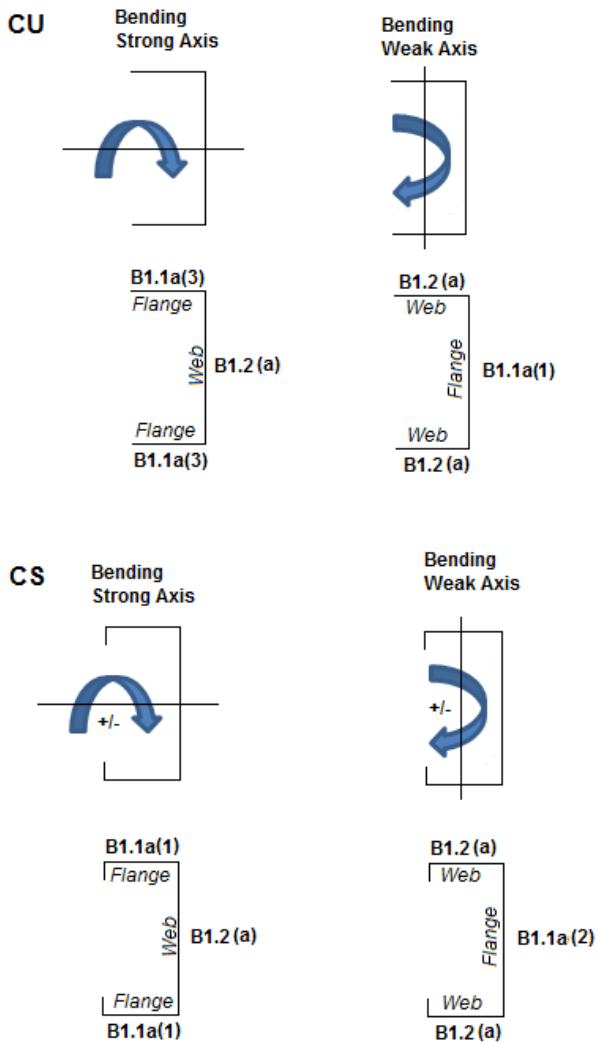
Kt in Section F2.1 is assumed to be 1.0. All conditions listed for the use of C3.1.3-1 are assumed satisfied. Section C3.1.4 is not considered in the calculation of Mn. Effects of shear stiffeners for Section G2 are not considered. Only strong axis bending and strong axis shear are considered for equation C3.3.1 (combined bending and shear).

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

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

H Shapes – Hat sections in bending about the y-y axis such that the brims are in compression are assumed braced such that the brims cannot each fail in lateral torsional buckling independently. 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/ri 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 (My) or shear forces (Vz) 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 (Cw) 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 Cw 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 (gamma) for a simple lip stiffener must be greater or equal to 40 degrees or less than or equal to 140 degrees per the criteria in Commentary Section 1.3 (AISI S100-16) or Section B4.2 (AISI S100-12). The angle gamma for this shape is outside this range.

Can't do code check, flange w/t > 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).

Columns - Modeling

Both columns and walls may be used as supports for beams or concrete slab floors, and both transfer load from floor to floor, and ultimately to the foundation. For beam-supported floors, column design is available for Hot Rolled Steel, Cold Formed Steel, Wood, and Concrete. Only concrete columns may be used to support concrete slab floors. Automatic live load reduction for columns is available as well.

Columns may be defined as either *Gravity* columns or *Lateral* columns. Gravity columns are analyzed and designed completely within RISAFloor for gravity/vertical loads only. Lateral columns function as part of the lateral load resisting system which can be analyzed and designed in RISA-3D for the combined gravity/vertical loads as well as lateral forces. See RISA-3D Integration for more information on lateral system design.

Columns can be specified as Hot Rolled, Cold Formed, Wood, or Concrete. The shapes available for each are listed below:

Designation	Shape Group
Hot Rolled	Wide Flange
	Tube
	Pipe
	Channel
	Single Angle
	Double Angle
	W_Tee
	WF 8
	WF 10
	WF 12
	WF 14
	HSS Pipe
	Square Tube
Cold Formed	CU
	CS
	ZU
	ZS
	HU
Wood	Rectangular
	Rectangular Double
	Rectangular Triple
	Round
Concrete	Rectangular
	Rectangular Square
	Round
	Rectangular_MM
	Rectangular_MM Square
	Round_MM

Any available material listed in the **Materials** spreadsheet can be assigned to a column (See Material Properties). All column shapes and material properties may be defined graphically as you draw or later using the Modify command. Refer to the next two sections for more information.

Column / Beam Interaction

Columns may be modeled as vertical supports with no rotational stiffness, or as vertical supports with a rotational stiffness based on their height, section properties, and orientation. Refer to Model Settings - Solution for more information.

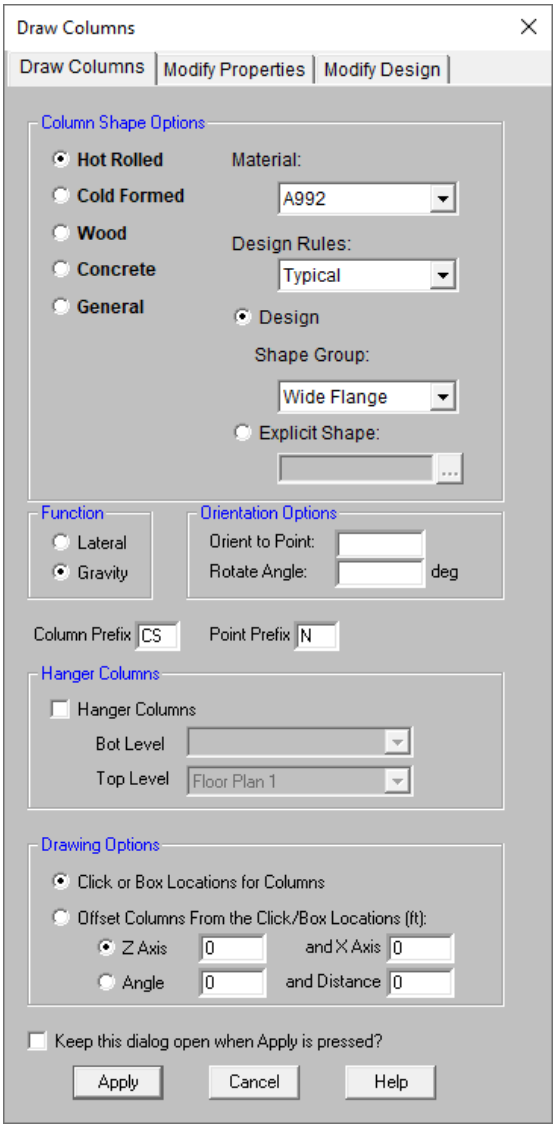
The fixity at the top of a column stack may be controlled through the Columns Spreadsheet.

Drawing Columns

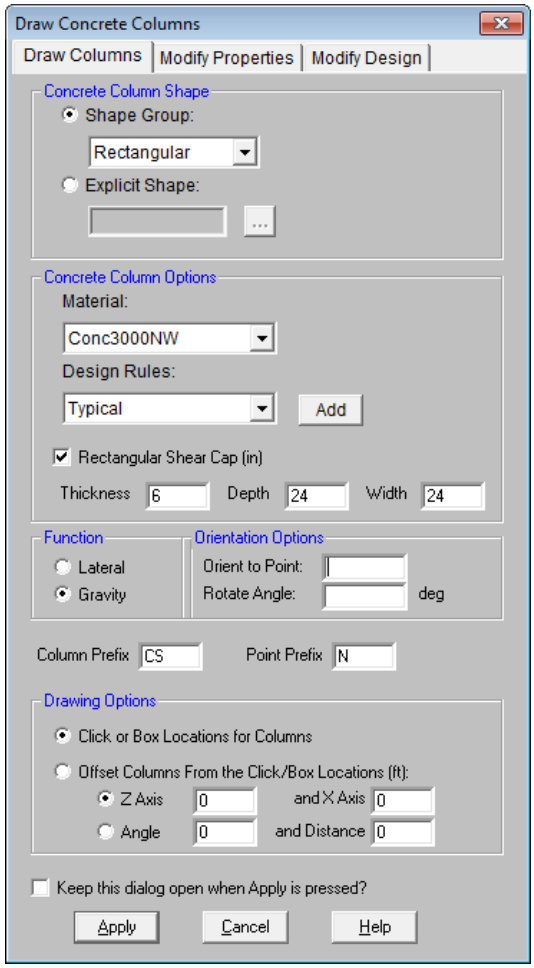
New columns may be drawn in a model view window using a Project Grid, Drawing Grid, or previously entered point locations. You can set all of the column properties before drawing them or you can modify the properties afterward. Modifying properties is discussed in the next section.

To draw columns, click on the Drawing toolbar to open the **Draw Columns** dialog. Enter the column type, material, shape group, function, and orientation followed by the drawing option you wish to use. By selecting the **Click or Box Locations for Columns** option you will click or box existing grids or points after clicking **Apply**. Clicking on a grid intersection or point defines a single column. By boxing multiple grid intersections you can define multiple columns at once. Choosing the **Offset Columns From the Click/Box Locations** option allows you to define a horizontal and/or vertical offset or an angular offset from the selected grid intersections or points. As you draw, the coordinates of the nearest grid point are shown in the status bar in the lower right corner. The new columns will be shown on screen and will be recorded in the **Columns** and **Column Stacks** spreadsheets.

Beam Supported Floors:



Concrete Slab Floors:



To Draw Columns:

1. If there is not a model view already open, click on the RISA toolbar to open a new view and click to turn on the Drawing toolbar if it is not already displayed.
2. If you do not already have a grid or existing point locations available you will need to create them first.
3. Click the button and set the **Column Shape**, **Material** options, **Design Rule**, **Function**, **Orientation**, and **Drawing Options**.
4. Click **Apply** to start drawing columns by clicking or boxing grid intersections and/or points with the left mouse button. The coordinates of the closest grid or point to your cursor are displayed in the status bar on the lower right hand corner of your screen.
5. To stop drawing altogether click the right mouse button, or press the ESC key.

Note:

- Press CTRL-D to quickly recall the dialog and make changes.
- You may also specify or edit columns in the **Columns** or **Column Stacks** spreadsheets.
- Undo any mistakes by clicking the **Undo** button.
- For information on modeling shear caps please see the Punching Shear - Design topic.

To Draw Hanger Columns:

Below are steps to draw hanger columns. For more information, see Hanger Columns.

1. If there is not a model view already open, click on the RISA toolbar to open a new view and click to turn on the Drawing toolbar if it is not already displayed.
2. Verify the floor from which the hanger columns will originate has been modeled.
3. Create a new beam supported floor that is located beneath the floor where the hanger columns originate.
4. Show the Plan View of the floor which the hanger column originates from (supporting floor) to draw the hanger column.
5. Click the button and set the **Column Shape**, **Material** options, **Design Rule**, **Function**, **Orientation**, and **Drawing Options**.
6. Turn on the checkbox for **Hanger Columns** and verify the bottom and top levels. Note that currently hanger columns can only be drawn from the supporting level to its adjacent level below, so users cannot use the dropdown to change the top and bottom levels of hanger columns.
7. Click **Apply** to start drawing columns by clicking anywhere along the length of the beam with the left mouse button. The coordinates of the closest grid or point to your cursor are displayed in the status bar on the lower right hand corner of your screen.
8. To stop drawing altogether click the right mouse button, or press the ESC key.

Note:

- Hanger columns will collect floor loads from connecting beams and take them up to the supporting beams above as concentrated loads.
- Hanger columns can only support a single floor at this time.
- Hanger columns can only be Hot Rolled, Cold Formed, or Wood shapes.
- Hanger columns can only be drawn for Beam Supported Floors.
- Hanger columns cannot be supported by walls or columns from above or below.
- Hanger columns cannot be drawn from framing members that are supported by hanger columns.
- No outriggers are allowed on hanger columns.

Modifying Column Properties

To modify column properties you may edit the column data in the **Columns** and/or **Column Stacks** spreadsheets. Use the **Modify Properties** tab of the **Columns** dialog to graphically modify a large selection of columns at once. See Column Primary Data for specific information on the column properties.

The graphical **Modify Properties** tool lets you modify the properties of columns that already exist in your model. To use this, you will typically specify the parameters you want to change and then select the columns that you want to modify. You can modify columns one at a time by selecting the **Click to Apply** option and then clicking on the columns you wish to modify. You may also modify entire selections of columns by previously selecting the columns and using the **Apply to Selected** option. See Graphic Selection for more information on selecting.

To Modify Column Properties:

1. If there is not a model view already open, click on the RISA toolbar to open a new view and click to turn on the Drawing toolbar if it is not already displayed.
2. Click the button and select the **Modify Properties** tab. Set the parameters for the columns you wish to modify. Check the **Use?** box for each of the parameters you wish to apply.

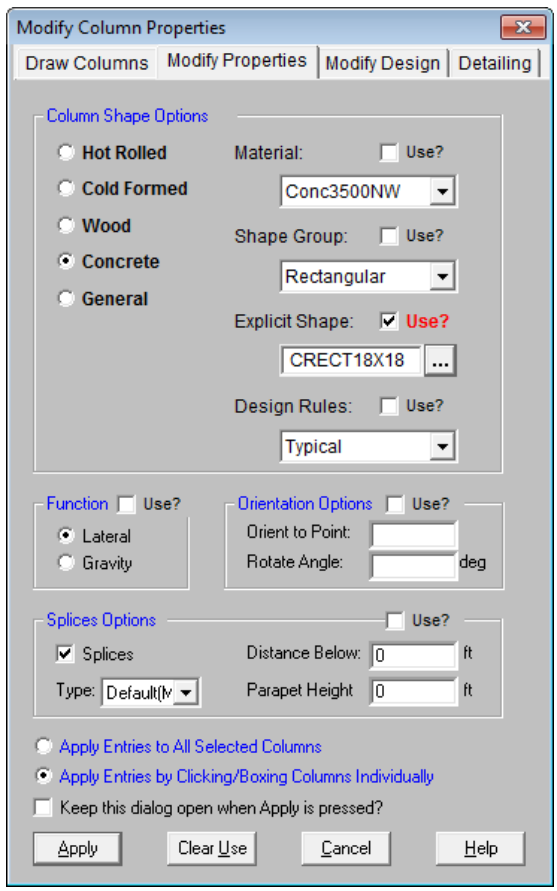
3. To modify columns one at a time choose **Apply Entries by Clicking Items Individually** and click **Apply**. Click on the columns with the left mouse button. To modify a selection of columns, choose **Apply Entries to All Selected Items** and click **Apply**.

Note:

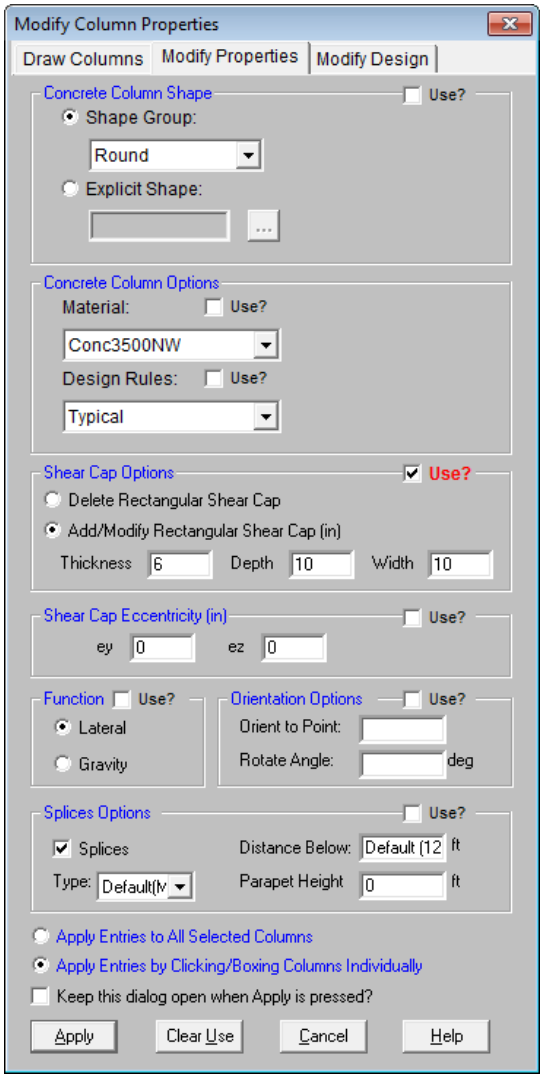
- To modify more columns with different parameters, press CTRL-D to recall the **Modify Properties** settings.
- You may also modify columns in the **Columns** or **Column Stacks** spreadsheets.
- Undo any mistakes by clicking the **Undo** button.

The parameters shown are the same as those used to define new columns.

Beam Supported Floors:



Concrete Slab Floors:



The **Use?** check boxes next to the data fields indicate whether the particular parameter will be used or not when the modification is applied. If the box next to a field is checked, that parameter will be applied to any selected columns, if the box is NOT checked, the parameter will NOT be applied, even if a value is entered in the field. This lets you easily change one or two properties on columns without affecting all the rest of the properties. Note that if the field is blank and the corresponding **Use?** box is checked, clicking **Apply** will have the effect of clearing the data for these fields.

Modifying Column Design Parameters

To modify column design parameters you may edit the column data in the **Columns** spreadsheet. Use the **Modify Design** tab of the **Columns** dialog to graphically modify a large selection of columns at once. See Columns Spreadsheet for specific information on the column properties.

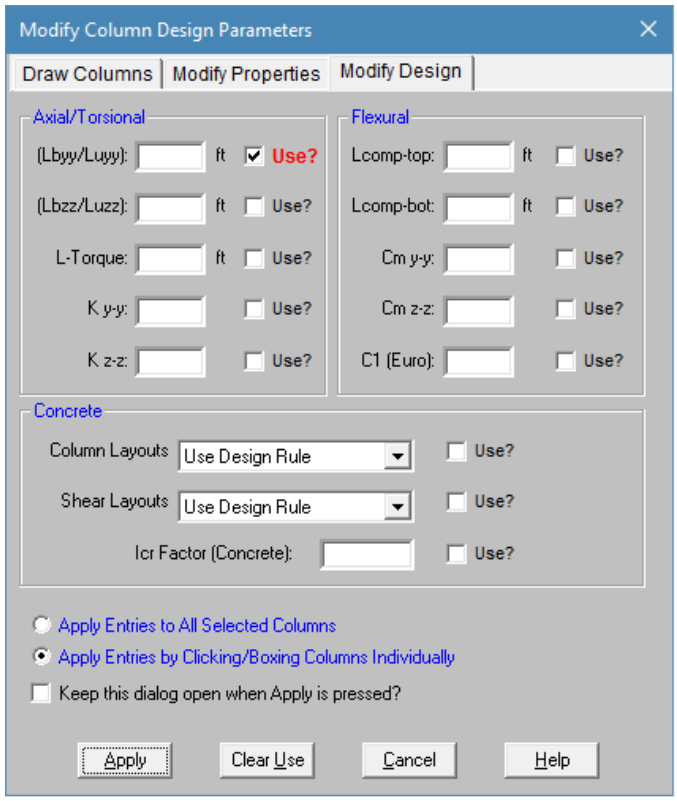
The graphical **Modify Design** tool lets you modify the design parameters of columns that already exist in your model. To use this, you will typically specify the parameters you want to change and then select the columns that you want to modify. You can modify columns one at a time by selecting the **Click to Apply** option and then clicking on the columns you wish to modify. You may also modify entire selections of columns by previously selecting the columns and using the **Apply to Selected** option. See Graphic Selection for more information on selecting.

To Modify Column Design Parameters:

1. If there is not a model view already open, click on the RISA toolbar to open a new view and click to turn on the Drawing toolbar if it is not already displayed.
2. Click the button and select the **Modify Design** tab. Set the design parameters for the columns you wish to modify. Check the **Use?** box for each of the parameters you wish to apply.
3. To modify columns one at a time choose **Apply Entries by Clicking Items Individually** and click **Apply**. Click on the columns with the left mouse button. To modify a selection of columns, choose **Apply Entries to All Selected Items** and click **Apply**.

Note

- To modify more columns with different parameters, press CTRL-D to recall the **Modify Properties** settings.
- You may also modify column design parameters in the **Columns** spreadsheet.
- Undo any mistakes by clicking the **Undo** button.



The **Use?** check boxes next to the data fields indicate whether the particular parameter will be used or not when the modification is applied. If the box next to a field is checked, that parameter will be applied to any selected columns, if the box is NOT checked, the parameter will NOT be applied, even if a value is entered in the field. This lets you easily change one or two properties on columns without affecting all the rest of the properties. Note that if the field is blank and the corresponding **Use?** box is checked, clicking **Apply** will have the effect of clearing the data for these fields.

Column Stacks and Physical Columns

Each column modeled in RISAFloor is considered part of a **Column Stack** which occupies a particular "plan" location within the building model. **Column Stacks** consist of numbered Column Lifts or **Physical Columns**. A physical column or column lift may be continuous through one or more floor levels and are defined by **Splice**

locations within the column stack. **Column Stacks** and **Physical Column Lifts** are defined on the **Column Stacks** spreadsheet.

Column Splices separate one physical column or column lift from the next within a column stack and can be defined as Moment Splices or Shear Splices. Moment Splices transfer both moment and shear from one physical column to the next. Shear Splices transfer only shear forces from one physical column to the next. Column Splices and locations may be assigned on the **Columns** spreadsheet or the **Floors** spreadsheet (defaults).

Columns may also be defined with **Parapet** extensions beyond the uppermost floor level. Parapets can only be applied to physical columns occurring along the perimeter of a deck/diaphragm area. **Parapet Heights** are defined on the **Columns** spreadsheet.

Example:

A particular column stack in a five story building consists of three column lifts or physical columns stacked one on top of the other. The column stack is tied to a particular plan location in building, regardless of elevation, and in this case occurs along the perimeter of the building.

- Column Lift No. 1 is the lowermost physical column in the column stack and begins at the base elevation of the building and continues vertically to a moment splice whose elevation is defined by a distance measured 10 feet below floor level three.
- Column Lift No. 2 begins at the splice defining the top of Column Lift No. 1 (10 feet below floor level three) and continues vertically to a moment splice whose elevation is defined by a distance measured 10 feet below floor level five (the roof in this case).
- The uppermost physical column, Column Lift No. 3, begins 10 feet below floor level five and continues vertically to a parapet height measured 3 feet above floor level five (roof). The definition of a parapet is only allowed because the column is on the perimeter of the building.

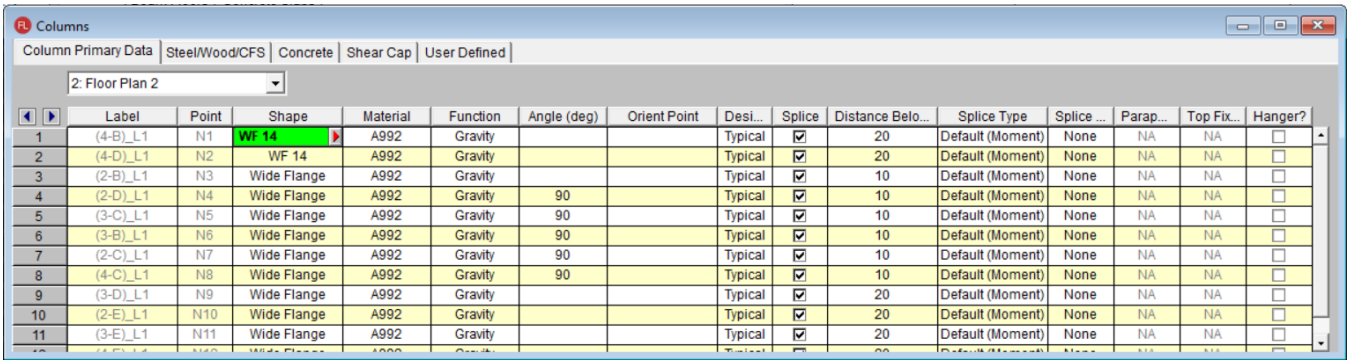
For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Column Splices**.

Columns Spreadsheet

The **Columns** spreadsheet contains all column input data, including primary data and material specific design parameters, and is floor level specific. The **Floor Level** may be selected from the pull down list just above the spreadsheet.

Primary Data

The **Column Primary Data** tab of the **Columns** spreadsheet records the properties for the column elements and may be accessed by selecting **Columns** on the **Spreadsheets** menu. For information on the **Steel/Wood** and **Concrete** tabs, please refer to the Hot Rolled Steel - Design, Cold Formed Steel - Design, Wood - Design, and Concrete - Design sections. For information on shear caps view the Punching Shear - Design topic.



	Label	Point	Shape	Material	Function	Angle (deg)	Orient Point	Desi...	Splice	Distance Belo...	Splice Type	Splice ...	Parap...	Top Fix...	Hanger?
1	(4-B)_L1	N1	WF 14	A992	Gravity			Typical	☑	20	Default (Moment)	None	NA	NA	☐
2	(4-D)_L1	N2	WF 14	A992	Gravity			Typical	☑	20	Default (Moment)	None	NA	NA	☐
3	(2-B)_L1	N3	Wide Flange	A992	Gravity			Typical	☑	10	Default (Moment)	None	NA	NA	☐
4	(2-D)_L1	N4	Wide Flange	A992	Gravity	90		Typical	☑	10	Default (Moment)	None	NA	NA	☐
5	(3-C)_L1	N5	Wide Flange	A992	Gravity	90		Typical	☑	10	Default (Moment)	None	NA	NA	☐
6	(3-B)_L1	N6	Wide Flange	A992	Gravity	90		Typical	☑	10	Default (Moment)	None	NA	NA	☐
7	(2-C)_L1	N7	Wide Flange	A992	Gravity	90		Typical	☑	10	Default (Moment)	None	NA	NA	☐
8	(4-C)_L1	N8	Wide Flange	A992	Gravity	90		Typical	☑	10	Default (Moment)	None	NA	NA	☐
9	(3-D)_L1	N9	Wide Flange	A992	Gravity			Typical	☑	20	Default (Moment)	None	NA	NA	☐
10	(2-E)_L1	N10	Wide Flange	A992	Gravity			Typical	☑	20	Default (Moment)	None	NA	NA	☐
11	(3-E)_L1	N11	Wide Flange	A992	Gravity			Typical	☑	20	Default (Moment)	None	NA	NA	☐

The entries for the **Column Primary Data** are explained below:

Label

Each column is assigned a label based on the corresponding **Stack Label** from the **Column Stacks** spreadsheet. That stack label is repeated here on the **Columns** spreadsheet with an extension, "_L#", that indicates the lift number for the particular column. The label field may not be edited on this spreadsheet but the stack label may be edited on the **Column Stacks** spreadsheet.

Point

The column location is defined by a point in the plane of the floor. These points are created automatically as you draw the columns.

Shape & Material

Enter the **Shape Group** and **Material** you wish to use for the column. You can choose both by clicking on the arrow in the cell. The material pull down list corresponds to the **Materials** spreadsheet. For more information, see Design Optimization - Design Lists or Material Properties for more information. Any entry or editing in these cells will also apply to the **Shape** and **Material** entries on the **Column Stacks** spreadsheet.

Function

Enter the **Function** you wish to use for the column – **Gravity or Lateral**. You can choose by clicking on the pull down list in the cell. Any entry or editing in this cell will also apply to the **Function** entry on the **Column Stacks** spreadsheet.

Angle

RISAFloor uses two methods to explicitly set the orientation of a column. The first is by setting the angle of rotation for the column about vertical global axis. Use the **Angle (deg)** entry to enter the rotation angle you wish to use for the column. If the entry is left blank or an angle of “0” is entered, the column is oriented so that the web is "vertical" or "north/south" in the plan view. The second orientation method is defined by the **Orientation Point** as described below.

Orientation Point

The second method to explicitly define the orientation of a column is by specifying an **Orientation Point** for the column. When used, this orientation point and the column point form a line that defines the orientation of the column web.

Note

- You can combine the **Angle (deg)** and **Orientation Point** to rotate the column with respect to its orientation point.

Design Rules

Enter the **Design Rules** you wish to use for the column. Select from any custom input entered in the Member Design Rules spreadsheet.

Splice

The **Splice** check box indicates whether there is a splice point in the column stack between the current floor level and the floor level below. The **Splice** check box will automatically be checked at the lowest floor level of each column stack, indicating the boundary condition of the column base.

Note

- The RISA-3D Boundary conditions at the bottom of a column will be initially set to perfectly pinned or perfectly fixed based on the splice type in RISAFloor. However, after the initial creation of the 3D model, the splice type is no longer linked to the 3D and it is the user's responsibility to update their boundary conditions to reflect any changes to the base fixity that are initiated in RISAFloor. For additional advice on this topic, please see the RISA Tips & Tricks website: www.risa.com/post/support. Type in Search keywords: **Column Boundary Conditions**.

Distance Below (Splice Location)

The **Distance Below** entry indicates the dimension to a splice located between the current floor level and the floor level below. If the splice check box is checked, the splice elevation will default to the dimension defined in the **Splice Distance Below** column of the **Floors** spreadsheet for the current floor level. If you wish to override the default splice dimension defined on the Floors Spreadsheet you may do so by entering an explicit value here. These entries can also be edited via the **Bot El.** and **Top El.** entries on the **Column Stacks** spreadsheet.

It is not possible to enter a negative dimension for the splice location as all entered values are considered as absolute values. If you would like to define a splice above the current floor level, you will need to enter the splice on the floor level above. The appropriate units are listed at the top of the column.

In the bottom column of a given column stack the **Distance Below** also defines the distance below the current floor elevation to the base of the column.

Note that in models with sloped floors, the distance below is always taken from the floor elevation regardless of the top elevation of a column.

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

Note:

- The distance below may not be set less than 1 inch. This is to prevent divide by zero errors when resolving column moments into shears.

Splice Type

The **Splice Type** entry defines the splice as either a Moment splice which will transfer moment and shear between column lifts in the stack or a Shear splice which will only transfer shear between lifts. If the splice check box is checked, the splice type will default to the type defined in the **Splice Type** column of the **Floors** spreadsheet for the current floor level. If you wish to override the default splice type you may do so by selecting a type from the pull down list in the cell in the **Columns** spreadsheet.

Splice Connection

The **Splice Connection** entry defines the splice connection rule. Only column splice type connections may be applied at upper floors and only baseplate connection types may be applied at the bottom-most floor level. These connections can then be exported into RISAConnection for connection design.

Parapet Height

If the upper most lift of a column stack exists along the perimeter of a diaphragm/deck, the user may assign a **Parapet Height** to that lift indicating the dimension to the top of the column with respect to the current floor level. This "column extension" will be included in the floor tributary mass or area for seismic and wind load generation respectively. The default parapet height is deflned in the Floors spreadsheet.

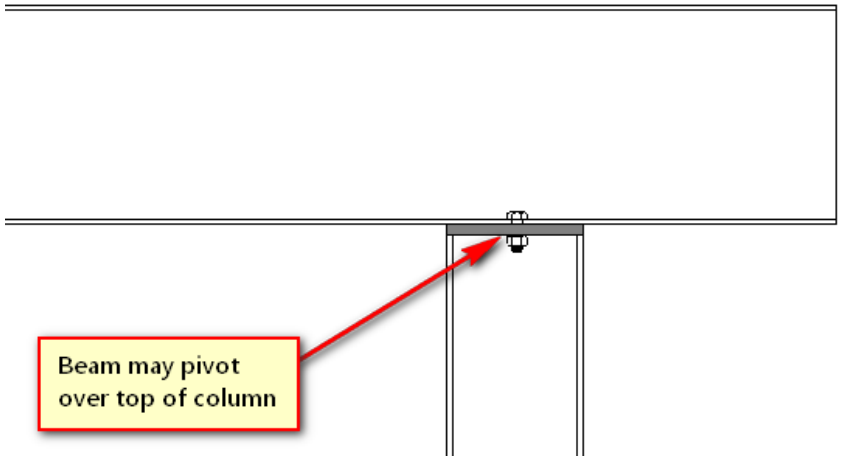
Note:

- This field can not be edited for slab floors. The Default Parapet Height entry on the Floors spreadsheet defines the parapet for slab levels.
- Parapet height for wind considerations changed between V10.0.1 and V11 of RISAFloor. For the current behavior please see the Load Generation - Wind Loads topic.
- Parapets created with columns only have self-weight due to the parapet height of the columns extending up. The parapet itself has no self-weight.

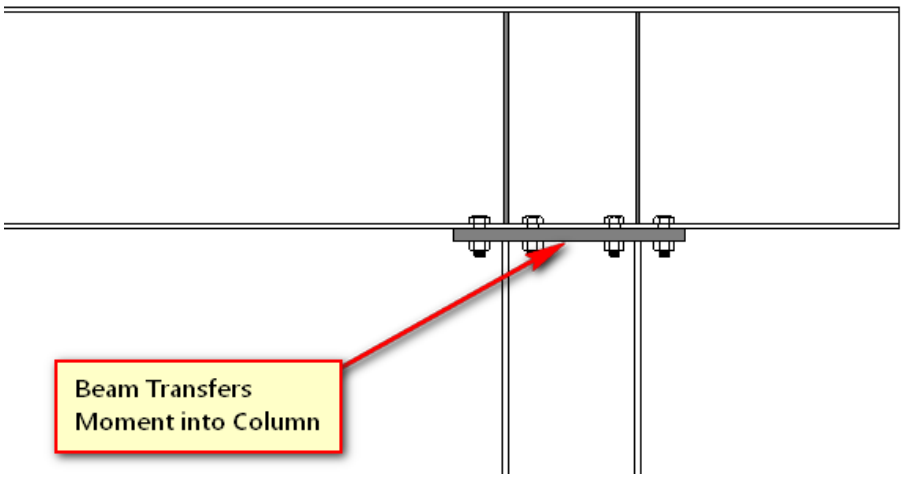
Top Fixity

The entry controls how the column connects to beams which frame into the top of it. When the top of the column is set to Fixed, it will provide rotational restraint for any beams which are framed into it and which also have a fixed end condition. If either the column top or the beam end is set to pinned then no rotational restraint is provided. The primary purpose for this feature is to allow continuous beams to cantilever over the top of a column without transferring any moment into that column (see below)

In the situation illustrated below the column top should be set to Pinned:



In the situation illustrated below the column top should be set to Fixed:



This setting is only available for the top column in the stack. All columns below the top floor for that column stack show "N/A".

Note:

- When setting the Column Top to be Pinned, the column will also ignore any moment that would be caused due to column eccentricity on shear connections.
- There must be a diaphragm present at the level where the top of column terminates to be able to define a pin at these locations.
- If a parapet is defined on this column then the column top must be fixed.

Hanger

This column indicates whether or not the column is a standard column or hanger column. If the entry is checked, it signifies the column is a hanger column.

Columns Spreadsheet - Detailing Data

The **Detailing** tab records the detailing data for the columns 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.

Column Detailing

Column Primary Data | Steel/Wood | Concrete | Shear Cap | Detailing

1: Floor Plan 1 (Slab)

	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	CS1_L1	10	0	0	0	10	0	0	0
2	CS2_L1	10	0	0	0	10	0	0	0
3	CS3_L1	10	0	0	0	10	0	0	0
4	CS4_L1	10	0	0	0	10	0	0	0
5	CS5_L1	10	0	0	0	10	0	0	0
6	CS6_L1	10	0	0	0	10	0	0	0
7	CS7_L1	10	0	0	0	10	0	0	0

Column Stacks Spreadsheet

The **Column Stacks** spreadsheet contains summary information for all column stacks within the building model and may be accessed by selecting **Column Stacks** on the **Spreadsheets** menu. When a column is drawn in RISAFloor, or added to the **Columns** spreadsheet, a Column Stack is automatically generated in the **Column Stacks** spreadsheet.

Column Stacks

	Stack Label	Project Grid	Z [ft]	X [ft]	Lift No.	Length [ft]	Bot El. [ft]	Top El. [ft]	Shape	Material	Function	Design Rules	Flexural Layout	Shear Layout
1	CS1 (F-5)	F-5	119	119	1	18.75	0	18.75	Wide Flange	A992	Gravity	Typical	N/A	N/A
2					2	11.25	18.75	30	WF 14	A992	Gravity	Typical	N/A	N/A
3	CS2 (B-2)	B-2	30	204	1	19	0	19	Wide Flange	A992	Gravity	Typical	N/A	N/A
4					2	11	19	30	WF 14	A992	Gravity	Typical	N/A	N/A
5	CS3 (I-9)	I-9	204	30	1	30	0	30	Wide Flange	A992	Gravity	Typical	N/A	N/A
6	CS4 (E-1)	E-1	0	149	1	30	0	30	Wide Flange	A992	Gravity	Typical	N/A	N/A
7	CS5 (D-1)	D-1	0	174	1	30	0	30	Wide Flange	A992	Gravity	Typical	N/A	N/A

The entries for the **Column Stacks** spreadsheet are explained below:

Stack Label

When a column stack is created in RISAFloor, a **Stack Label** is automatically generated containing the column stack number and the corresponding project grid line intersection, CS1 (A-1) for example. You may overwrite this assigned label with a unique label of your own if you so choose. However, each label must be unique, so if the same label is entered more than once an error message will be displayed.

Project Grid

The values listed indicate the project grid lines on which the column stack is located. These values are not available for editing in this spreadsheet but may be adjusted via the **Label** entries on the **Project Grid Spreadsheet**. If a column stack is not located on a project grid line in a particular direction a "?" will be used in place of a project grid line. If a project grid line is added at the column stack location at a later time, the **Project Grid** value will automatically be updated to reflect the new project grid line.

Column Stack Coordinates

The column stack coordinates, Z and X, contain the coordinates of the **Points** in the horizontal and vertical directions, respectively, that define the location of the column stack. These values represent the offsets of the **Points** from the origin of the global coordinate system (0, 0) and are not available for editing in this spreadsheet but may be adjusted via the entries on the **Point Locations** spreadsheet. The appropriate units are listed at the top of each column.

Lift Number

Each physical column within a column stack is assigned a **Lift Number** based on its location within the column stack. Lift No. "1" is assigned to the lowermost physical column within a column stack with Lift No. "2" assigned to the next physical column and so on, ascending from the bottom of the column stack to the top. These values are not available for editing.

Note

- There may be a "gap" in a column stack where no column occurs in the building plan at certain floor levels. The lifts will still be numbered sequentially from bottom to top. If the "gap" is later filled, the new physical column will merge with the column below unless an additional splice is added to the column stack. In which case, the column lifts will automatically be renumbered from bottom to top in order to account for the new lift and maintain sequential numbering.

Lift Length

The **Length** of each physical column or "lift" is listed here. This is the length of the column from the column base to a splice or top of column, or from splice to splice, or splice to top of column, depending on the position of the lift within the column stack. These values may be editing in two ways. The first is by editing the Bottom and Top Elevation entries that follow in the **Column Stacks** spreadsheet and the second is via the **Distance Below** entries on the **Column Primary Data** tab of the **Columns** spreadsheet.

Bottom/Top Elevation

The elevation of the bottom and top of each lift, **Bot El.** and **Top El.** respectively, are listed here with respect to the base elevation of the model. These elevations are determined by the **Distance Below** entries as defined on the **Column Primary Data** tab of the **Columns** spreadsheet. These elevation entries may be edited and will subsequently update the appropriate entries in this and other spreadsheets.

Shape & Material

Enter the **Shape Group** and **Material** you wish to use for the column. You can choose both by clicking on the arrow in the cell. The material pull down list corresponds to the **Materials** spreadsheet. See Design Optimization - Design Lists or Material Properties for more information. Any entry or editing in these cells will also apply to the **Shape** and **Material** entries on the **Columns** spreadsheet.

Function

Enter the function you wish to use for the column – **Gravity or Lateral**. You can choose by clicking on the pull down list in the cell. Any entry or editing in this cell will also apply to the **Function** entry on the **Columns** spreadsheet.

Design Rules

A **Design Rule** may be applied to each lift of a column stack by making a selection from the pull down list in this cell. The available entries in the pull down list are created on the **Design Rules** spreadsheet. For more information see Design Optimization - Design Rules.

Rebar Layouts

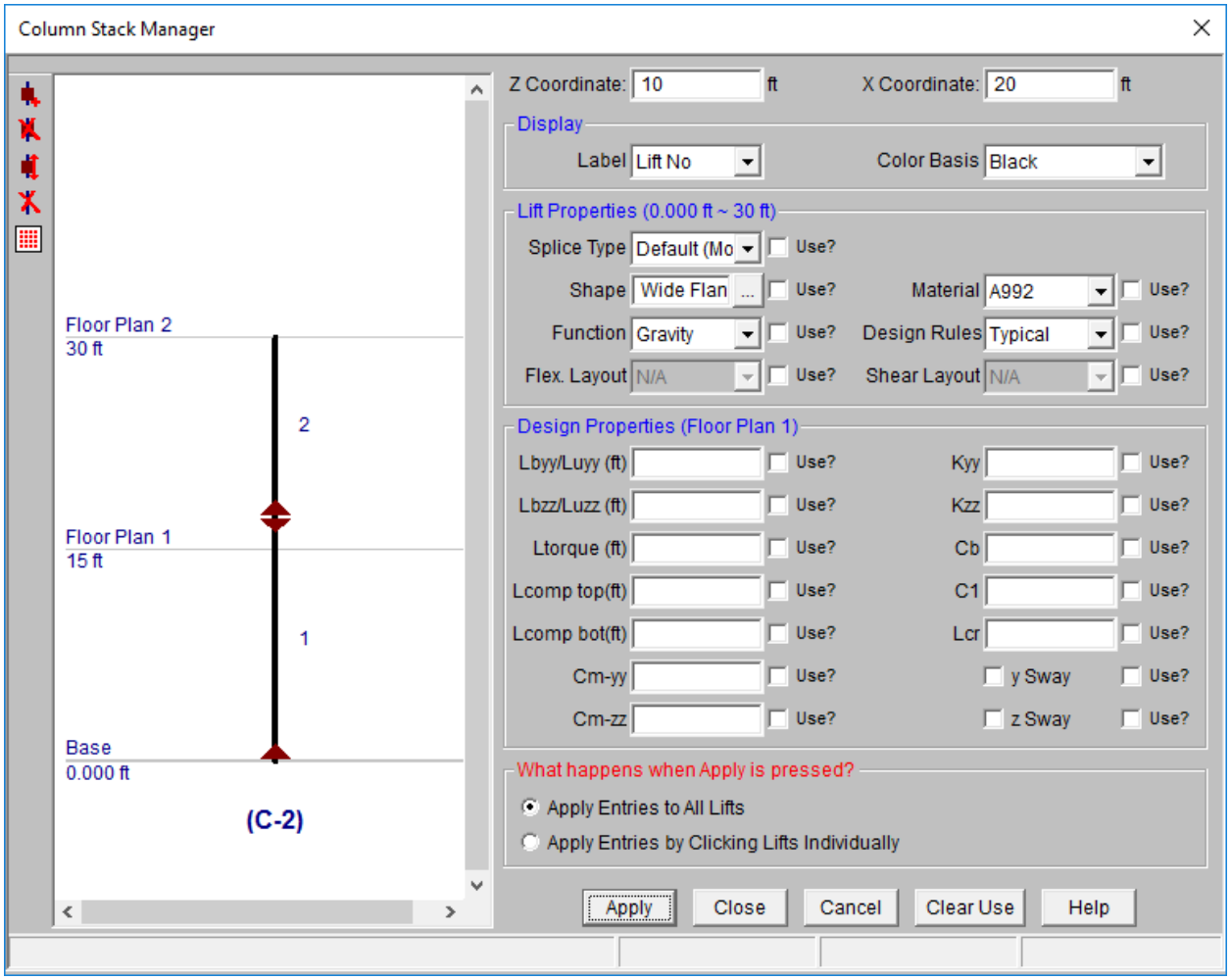
For concrete columns, the **Flexural Layout** and **Shear Layout** entries may be used to assign a custom rebar layout to each lift of a column stack. A custom rebar layout must be used if the user would like to take advantage of bundled bars, multiple layers or reinforcement, or an unequal number of bars per face.

Custom rebar layouts are created in the **Rebar Layout Database** and may be selected from the pull down list in this spreadsheet once they have been created. If *Optimize* is selected from the pull down list, RISAFloor will perform the design and select the appropriate reinforcement for the column lift based on the selected design rule. The program will design for an equal number of bars in each face of a rectangular column (a minimum of four (4) bars will be used in circular column sections) and may vary that reinforcing based on ACI minimums and maximums and the moment and shear demand at each section along the span. If the material used for a column lift is not a concrete material, the entry will default to *N/A*.

See the Concrete Database section and the Rebar Layout Database for more information.

Column Stack Manager

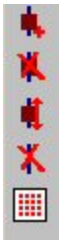
The **Column Stack Manager** may be accessed from a model view by double-clicking any column. This window allows for the display and editing of the entire column stack as well as the coordinates of the application point. It is particularly useful for assigning or moving splices.



Display and Display Controls

The column stack manager will display the entire height of the column stack. The **Label** and the **Color Basis** of the window can be modified by selecting the appropriate setting using the drop down menus. Click **Apply** to accept any display changes.

The column display view on the left is interactive. If the user clicks on the column within the display window, the assigned column properties will be displayed in the text boxes on the right hand side. This is useful for confirming design properties on a lift by lift or floor by floor basis.



In the upper left hand corner of the column stack display window, there exists a number of icons that allow the user to edit the column and column splice data. These allow the user to **Add**, **Delete**, or **Move** a column splice. Similarly, the user may delete an entire column lift if desired. The last icon controls the increment at which a splice can be moved.

Note:

- When moving a splice it is only possible to move it within the height of the existing floor level. If the splice needs to be moved to a different level, then it should be deleted and a new splice added at the appropriate location.
- Moment splices are indicated by displaying a red diamond shape at the splice location.
- Shear splices are indicated by displaying a green circle at the splice location.
- The lowest level of the column splice indicates the boundary condition at the bottom of the column.

Lift Properties

In display mode, the **Lift Properties** section displays the start and ending point of the lift along with all of the basic lift properties. The starting and ending distance are only

In modify mode, the user can choose to modify column lifts one at a time by choose **Apply Entries by Clicking Items Individually** and click **Apply**. Click on the lift with the left mouse button. To modify all of the displayed columns, choose **Apply Entries to All Lifts** and click **Apply**.

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

Note

- When moving a splice it is only possible to move it within the height of the existing floor level. If the splice needs to be moved to a different level, then it should be deleted and a new splice added at the appropriate location.

Design Properties

In display mode, the **Design Properties** section displays the current floor level along with all of the design parameters associated with this floor level.

In modify mode, the user can choose to modify column properties one floor at a time by choose **Apply Entries by Clicking Items Individually** and click **Apply**. Click on the floor, or slightly below it with the left mouse button. To modify all of the displayed floor levels, choose **Apply Entries to All Lifts** and click **Apply**.

The **Use?** check boxes next to the data fields indicate whether the particular parameter will be used or not when the modification is applied. If the box next to a field is checked, that parameter will be applied when the Apply button is pressed. If the box is NOT checked, the parameter will NOT be applied, even if a value is entered in the field. This lets you easily change one or two properties on a lift without affecting all the rest of the properties. Note that if the field is blank and the corresponding **Use?** box is checked, clicking **Apply** will have the effect of clearing the data for these fields.

Columns - Analysis

RISAFloor performs an analysis of columns by taking the individual floor results and using a Modified Hardy-Cross method to distribute moments through the column stack. Here we will talk about some of the details of the RISAFloor column analysis.

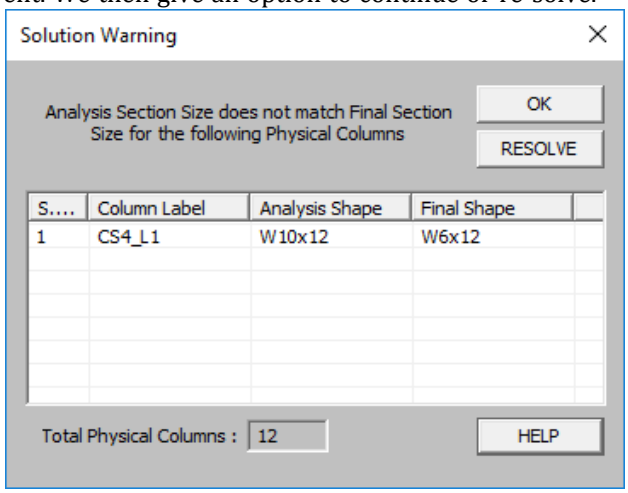
Note:

- For modeling and general information on columns see the Columns - Modeling topic.
- For column results information see the Columns - Results topic.

Multi-Story Column Analysis

RISAFloor designs a structure in a top to bottom manner, thus following load path. At each floor in the model the loads are brought down and applied to the floor below. This can creates some challenges for column analysis in multi-story applications in certain situations. It may be considered as follows:

1. The uppermost floor is analyzed and beams and columns are designed. We will see below that the column size affects the moment distribution to the beams.
2. We then go to the next floor down and the column size taken from the floor above is used in the analysis of the next floor down. During this analysis the beams and columns are again designed. It is probable that the column will need to be upsized due to the additional forces it is seeing from this floor. This upsizing will be considered properly on this floor and the floors below. However, the top-most floor has already been designed/analyzed. The program does not automatically iterate the column for all floors and keep running solutions. This could cause iterative loops that do not converge on a solution.
3. Instead, after solution the program gives the dialog box below. This gives a list of all columns which analysis and design sections were different. We then give an option to continue or re-solve.

   a. 

4. If the "Analysis Shape" and "Final Shape" are very close in size then this difference is likely small. In this case you may choose the OK option which will finish the solution and give results with the mismatched analysis/design sizes. If these shapes are very different then you may choose RESOLVE and an entirely new solution will be run with the analysis shape updated accordingly. Note that you may need to resolve several times if this situation occurs again with the new solution.

Lateral Support From Floor/Slab Levels and Framing Elements

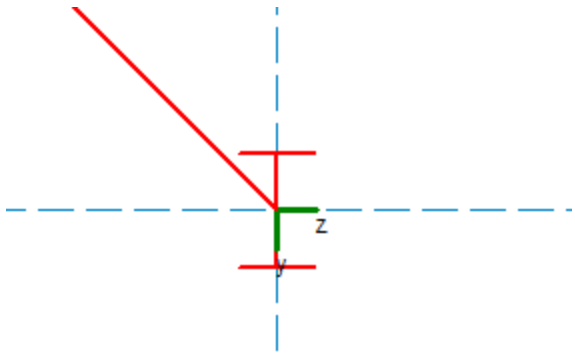
Columns that are multi-story that pass through multiple floors will be checked to determine whether an individual floor provides lateral support. A floor will provide lateral support for a column (for both analysis and design) if:

1. The column passes through a deck or slab.
2. The column has a beam or beams that frame into it that are not outrigger beams. There are some small differences between analysis and design that will be explained.

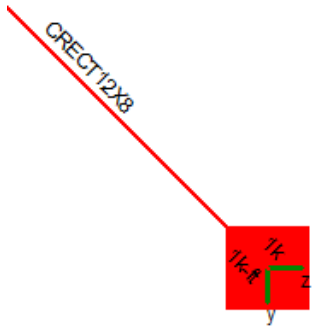
If a beam frames into the column it will brace the column in the direction it is framing in. If a beam frames in that is not in the direction of column local axes then there is different behavior for analysis and design.

Lateral Support for Design (Unbraced Length) Considerations.

For design considerations the beam will brace the direction it would physically frame into. For example, if a beam frames into a wide flange member the program will check to see if the centerline of the beam hits the flange or the web. If it hits the web it will brace the local y-y direction. If it hits the flange it will brace the z-z direction. A beam that frames in at 45 degrees will brace both column local axes. The example below will brace the y-y direction.

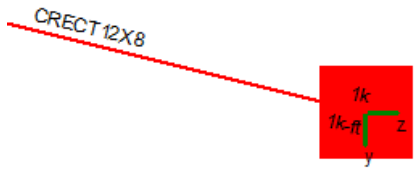


The exception to this occurs if a member frames right at the interface between the strong and weak direction. In that case the program assumes the member is braced only in the z-z direction. See the example below where a concrete beam frames into a square concrete column at a 45 degree angle. In this situation the column is only braced about it's strong axis (z-z) and is not considered braced at this level about it's weak axis (y-y).



Lateral Support for Analysis Considerations.

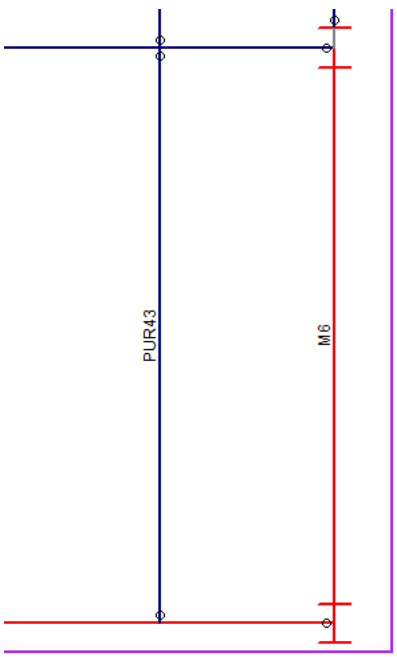
For analysis, if a beam frames in directly along the local axes of the column then the column is only braced in that local axis direction. If a skewed beam frames into a column then it is assumed that the beam braces BOTH local directions, regardless of the angle of the member framing in. In the example below the beam physically frames into the left face of column. However, because it is a skewed member it will be considered to brace the column in both directions for analysis.



Column Moments and Shears - Eccentricity

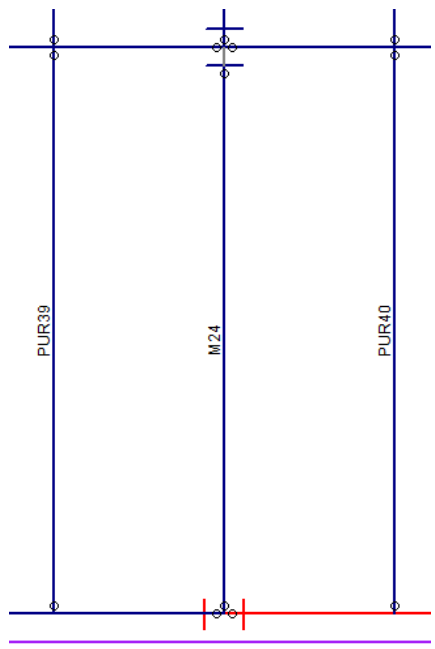
For columns with members framing in without eccentricity the column will be an axial-force only element and the analysis is trivial. However, there are several different ways in which moments can be introduced to columns in RISAFloor. Aside from carrying a moment from higher or lower in the column stack, a column can pick up a moment at a floor from the following methods:

- Beams which have fixed-ends and which frame directly into the column.
- Beams which have pinned-ends and which have **Column Eccentricity** turned on in their **Connection Properties**
- Slab moments that will be transferred into the column (RISAFloor ES only)



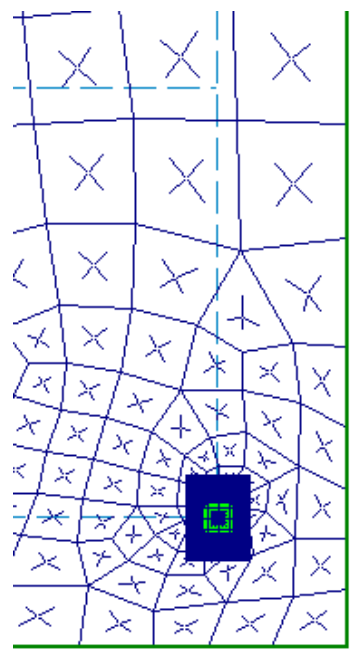
Fixed-End Beams

M6 has fixed end releases on either end, which will impart moment forces into the columns accordingly.



Pinned-End Beams

M24 has Column Eccentricity turned on. Thus, there is an eccentricity defined that will impart a moment to the columns based on the reaction * eccentricity.

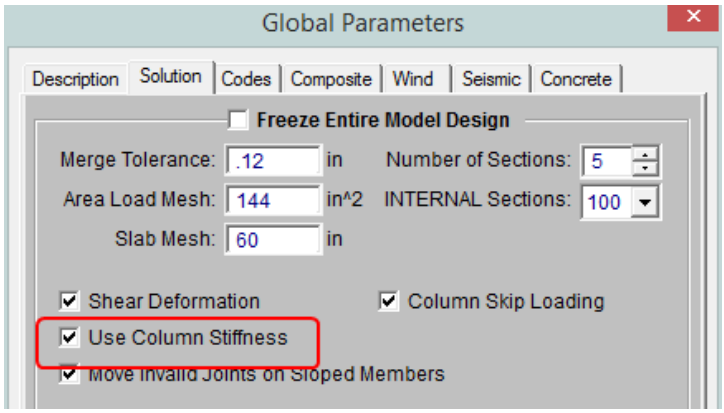


Slab Floor (Floor ES only)

The slab will be moment connected to the column automatically

Fixed-End Beams and Slab Floors

Fixed-end beams can only transfer moment into a column when the Use Column Stiffness checkbox has been checked on the Model Settings.



When this is enabled the column's rotational stiffness is approximated for each axis and is considered in the stiffness matrix solution for that floor. Because RISAFloor performs an independent stiffness matrix solution for each floor the column's rotational stiffness is represented (behind the scenes) as a rotational spring boundary condition. This results in some moment redistribution and can therefore affect the amount of moment which the beam carries.

The amount of rotational stiffness provided by the column (rotational spring boundary condition) is influenced by the following factors:

- The rotational stiffness (E*I) of the column
- The distance (L) from the floor level in question to the floors immediately above and below
- The rotational stiffness of the fixed-end beams on the floors immediately above and below the floor level in question

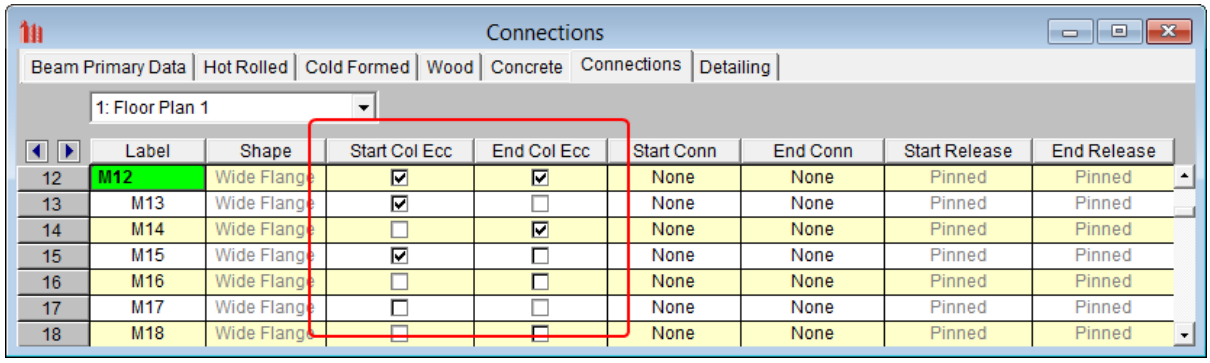
The resultant joint reaction (moment) from the stiffness matrix solution at the spring boundary condition is then distributed to the column. The amount of moment which is sent to the portion of the column above the floor (versus the portion below the floor) is determined using a simplified version of the Hardy Cross Moment Distribution Method.

Note:

- Column flexural properties (E, I and floor heights) are used to determine the distribution factors in the Hardy Cross Moment Distribution Method in the same way they are used to approximate the column rotational stiffnesses.
- The applied moments at each floor are taken directly from the independent stiffness matrix solution for that floor.
- This method is intended to function as a good approximation of the true moments. However, this method does not take into account shear deformation or the presence of axial loads or deformations.
- The Hardy Cross moments only affect the column moments, not any beams that are fixed to the columns. This can result in unbalanced moment between the beam and column.
- The final design of column and beams which require the consideration of three dimensional effects (i.e. moment frames or columns that are supported by transfer girders) should always be performed in RISA-3D. This is because RISA-3D will use a full three dimensional stiffness matrix solution to solve for all forces and moments.

Pinned-End Beams

Pinned-end beams can only transfer moment into a column when the Column Eccentricity flag has been checked in the Connections tab of the Beams spreadsheet.



	Label	Shape	Start Col Ecc	End Col Ecc	Start Conn	End Conn	Start Release	End Release
12	M12	Wide Flange	☑	☑	None	None	Pinned	Pinned
13	M13	Wide Flange	☑	☐	None	None	Pinned	Pinned
14	M14	Wide Flange	☐	☑	None	None	Pinned	Pinned
15	M15	Wide Flange	☑	☐	None	None	Pinned	Pinned
16	M16	Wide Flange	☐	☐	None	None	Pinned	Pinned
17	M17	Wide Flange	☐	☐	None	None	Pinned	Pinned
18	M18	Wide Flange	☐	☐	None	None	Pinned	Pinned

The moment introduced into the column is calculated using the following formula:

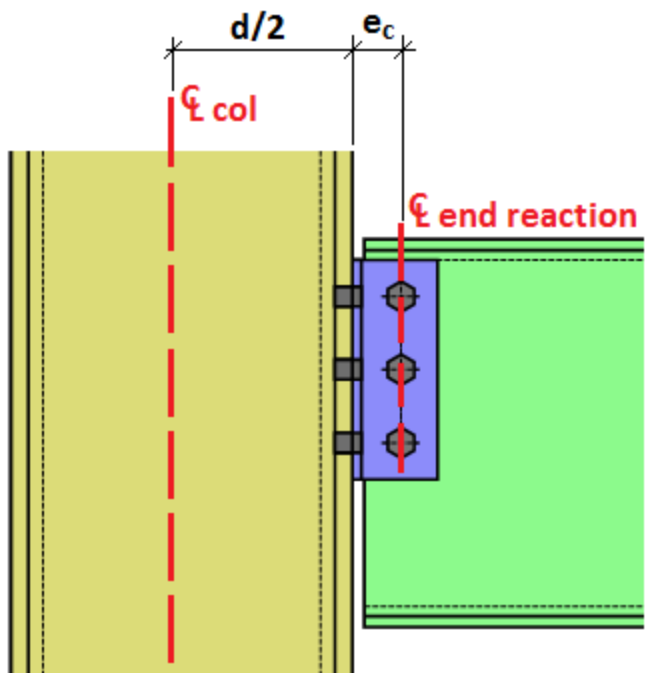
$$\mathbf{M_{column} = R_{beam}*((d_{column}/2) + e_{connection})}$$

where

R = Beam end reaction

d = Column depth in the direction which the beam frames into the column (**d** for strong axis, **b** for weak axis)

e = Eccentricity between the face of the column and the beam's end reaction resultant

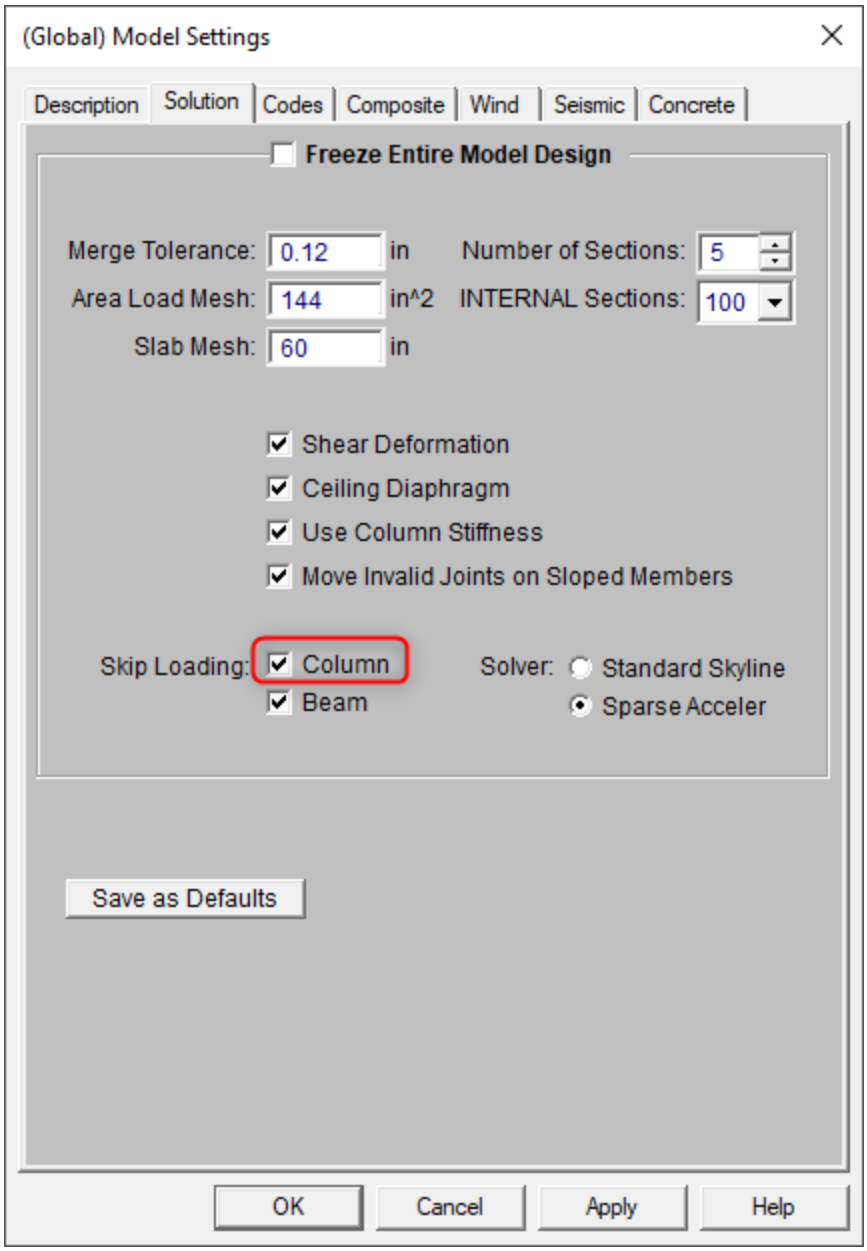


Note:

- Only Hot Rolled Steel beams may be assigned an eccentricity.
- Beams which are skewed (not parallel to either of the column's local axes) will receive a geometrically increased offset to account for the additional eccentricity caused by the additional depth of column.
- For beams framing into the web of **Wide Flange** columns the value (d/2) is taken as zero.
- The moment caused by eccentricity is ignored for beams framing into the top of columns that have a Pinned top fixity.

Column Skip Loading

This feature is only enabled when it has been activated in the Model Settings.

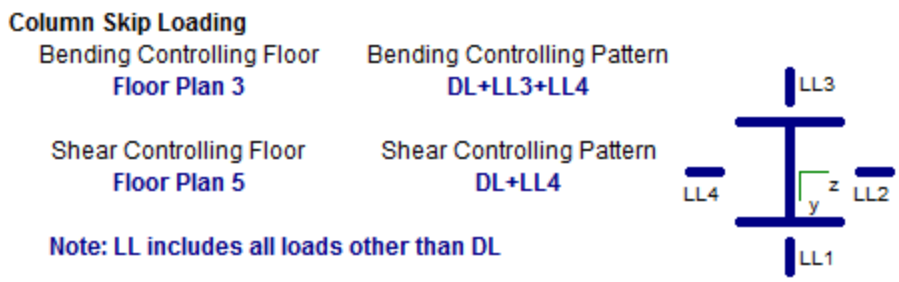


When enabled the program automatically checks for the worst-case unbalanced loading on the columns. The methodology is explained below:

- The program determines the shear and moment end reactions (including moment caused by eccentricity) for each beam that frames into a given column.
- All beams which frame into that column are lumped into four "quadrants", each representing a cardinal direction relative to the column.
    - These quadrants are illustrated in the Detail Report as LL1, LL2, LL3, and LL4. They are relative to the column's local axes.
    - Skewed beams are assigned to the nearest quadrant. Their end reaction is assumed to act entirely within that quadrant.

- During solution the dead load (DL) including Self Weight, PreDL, PostDL, and Const DL is assumed to be fully applied for all four quadrants for all permutations.

- During solution the program solves each permutation of live load patterns for each load combination at each floor of a column lift. While checking the various patterns at a given floor all other floors are assumed to be fully-loaded
- The code check value reported for the column is based on the controlling load combination, with the controlling pattern applied on the controlling floor, and all other floors for that column taken as fully loaded (i.e. the pattern at all other floors is DL + LL1 + LL2 + LL3 + LL4).
- Separate patterns are determined for the Bending/Axial code check and the Shear code check.
- Columns with cantilevers are not skip loaded in the direction of the cantilever. This is unnecessary because the analysis results properly apportion the cantilever moments based on relative rigidity of the back span beam and supporting column.
- Skip loaded LL moments are carried up and down the column stack by quadrant. The worst case unbalanced load in each quadrant for each floor level is used for hardy cross moment redistribution and for the design.



Note:

- A wide flange image is used in the detail report regardless of the actual shape type assigned to the column. This is for strong/weak axis clarity.

Hanger Columns

Hanger columns can be drawn to support a mezzanine floor. A mezzanine floor is like a typical floor with the only difference being that the floor is supported by hanger columns from the floor above. Therefore, the loads originating from the mezzanine floor will distribute to the hanger columns and then ultimately to the floor above. The hanger column must start from a supporting beam from the floor above and can only support the floor directly beneath. Steps to draw hanger column is presented in To Draw Hanger Columns.

Limitations:

- Hanger columns can only support a single floor at this time.
- Hanger columns can only be Hot Rolled, Cold Formed, or Wood shapes.
- Hanger columns can only be drawn for Beam Supported Floors.
- Hanger columns cannot be supported by walls or columns below.
- Hanger columns cannot be drawn from framing members that are supported by hanger columns.
- No outriggers are allowed on hanger columns.

Columns - Results

When the model is solved the results are separated into material-specific design results and general results. The general column 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.

Column Forces Spreadsheet

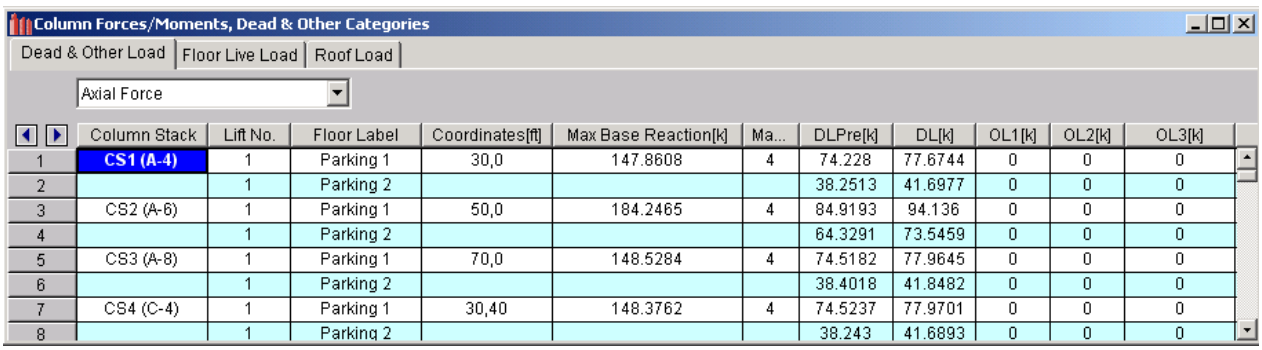
The **Column Forces Spreadsheet** records the forces and moments in physical columns at each floor level and splice location. This spreadsheet may be accessed by selecting **Column Forces** from the **Results Menu**.

The pull down list at the top of the spreadsheet allows you to toggle between various column forces or moments. The choices available are **Axial Force**, **Shear z-z**, **Shear y-y**, **Moment z-z (Top)**, **Moment y-y (Top)**, **Moment z-z (Bot)**, and **Moment y-y (Bot)**.

The spreadsheet has three tabs: Dead & Other Load, Floor Live Load, and Roof Load.

Dead & Other Load Tab

The **Dead & Other Load** tab displays the dead load category forces and moments for physical columns. For more information on load categories, see the Loads topic. The results displayed on this tab are explained below.


Column Forces/Moments, Dead & Other Categories

Dead & Other Load | Floor Live Load | Roof Load

Axial Force

	Column Stack	Lift No.	Floor Label	Coordinates[ft]	Max Base Reaction[k]	Ma...	DLPre[k]	DL[k]	OL1[k]	OL2[k]	OL3[k]
1	CS1 (A-4)	1	Parking 1	30,0	147.8608	4	74.228	77.6744	0	0	0
2		1	Parking 2				38.2513	41.6977	0	0	0
3	CS2 (A-6)	1	Parking 1	50,0	184.2465	4	84.9193	94.136	0	0	0
4		1	Parking 2				64.3291	73.5459	0	0	0
5	CS3 (A-8)	1	Parking 1	70,0	148.5284	4	74.5182	77.9645	0	0	0
6		1	Parking 2				38.4018	41.8482	0	0	0
7	CS4 (C-4)	1	Parking 1	30,40	148.3762	4	74.5237	77.9701	0	0	0
8		1	Parking 2				38.243	41.6893	0	0	0

The **Column Stack** entry displays the column stack label. The parentheses note the gridlines of your column. If your column is only on a single grid line you will get (Grid-). If your column is not on a gridline at all you will get (?-?) or (na).

The **Lift No.** column displays the lift number for the physical column. Lift No. 1 is the lowermost physical column in a stack and the lifts are numbered sequentially moving up the column stack.

The **Floor Label** column displays the floor levels through which each column lift passes. Splice locations between floor levels are also listed and are indicated by a '+' symbol prior to the floor label. An entry of '+3rd Floor' indicates that there is a splice in the column stack above the 3rd Floor.

The **Coordinates** entry displays the plan coordinates of the column stack and is only listed on rows where the column stack label is shown.

The **Max Base Reactions** column records the cumulative forces and/or moments in a physical column lift at the base due to the load categories DL, OL1, OL2, OL3, OL4, LL, LLS, RLL, SL, SLN, and RL, included in the controlling load combination. This value takes into account the live load reduction factors as well as the Load Factors for the controlling load combination.

The **Max Base Reaction LC** column displays the load combination that was used to find the Max Base Reaction.

The **DLPre** and **DL** columns record the cumulative dead load category forces and/or moments in a column lift at the floor level indicated. The forces and moments shown include the load accumulated in the column lift at that floor level in addition to the load accumulated in the floor levels above.

The **OL1**, **OL2**, **OL3**, and **OL4** columns record the cumulative dead load category forces and/or moments in a column lift at the floor level indicated. The forces and moments shown include the load accumulated in the column lift at that floor level in addition to the load accumulated in the floor levels above.

Floor Live Load Tab

The **Floor Live Load** tab displays the floor live load category forces and moments for physical columns. For more information on load categories, see Loads. The results displayed on this tab are explained below.

Column Forces/Moments, Floor Live Load

Dead & Other Load | Floor Live Load | Roof Load

Axial Force

	Column Stack	Lift No.	Floor Label	Coordi...	Reducible ...	LL Reduce	NonRe...	LL[k] (unre...	LLS Redu...	NonReduc...	LLS[k] (unr...
1	CS1 (A-4)	1	Parking 1	30,0	539.3518	.5729	0	18.8426	.8	22.4465	0
2		1	Parking 2		539.3518	.5729	0	18.8426	1	11.2918	0
3	CS2 (A-6)	1	Parking 1	50,0	1078.7036	.4784	0	50.3915	.8	18.2247	0
4		1	Parking 2		1078.7036	.5	0	50.3915	1	9.0078	0
5	CS3 (A-8)	1	Parking 1	70,0	539.3518	.5729	0	18.8426	.8	22.6462	0
6		1	Parking 2		539.3518	.5729	0	18.8426	1	11.3984	0
7	CS4 (C-4)	1	Parking 1	30,40	539.3518	.5729	0	18.8426	.8	22.5469	0
8		1	Parking 2		539.3518	.5729	0	18.8426	1	11.2862	0

The **Column Stack** entry displays the column stack label.

The **Lift No.** column displays the lift number for the physical column. Lift No. 1 is the lowermost physical column in a stack and the lifts are numbered sequentially moving up the column stack.

The **Floor Label** column displays the floor levels through which each column lift passes. Splice locations between floor levels are also listed and are indicated by a '+' symbol prior to the floor label. An entry of '+3rd Floor' indicates that there is a splice in the column stack above the 3rd Floor.

The **Coordinates** entry displays the plan coordinates of the column stack and is only listed on rows where the column stack label is shown.

The **Reducible Area** column records the cumulative floor area eligible for live load reduction that is tributary to the column lift at the floor level indicated. The area shown includes the area accumulated in the column lift at that floor level in addition to the area accumulated in the floor levels above.

The **LL Reduce** and **LLS Reduce** columns display the floor live load reduction factors applicable to the column lift at the floor level indicated. These reduction factors are calculated based on the values listed in the 'Reducible Area' column of the spreadsheet.

The **LL (unreduced)** and **LLS (unreduced)** columns record the cumulative *Reducible* floor live load category forces and/or moments in a column lift at the floor level indicated. The forces and moments shown include the load accumulated in the column lift at that floor level in addition to the load accumulated in the floor levels above.

Note

- The *Reducible* floor live load category forces and/or moments shown have NOT been reduced by the applicable reduction factors listed in the spreadsheet.

The **NonReduce LL** and **NonReduce LLS** columns record the cumulative *Non-Reducible* floor live load category forces and/or moments in a column lift at the floor level indicated. The forces and moments shown include the load accumulated in the column lift at that floor level in addition to the load accumulated in the floor levels above.

Roof Load Tab

The **Roof Load** tab displays the roof load category forces and moments for physical columns. For more information on load categories, see Loads. The results displayed on this tab are explained below.

Column Forces/Moments, Roof Load

Dead & Other Load | Floor Live Load | Roof Load

Axial Force

	Column Stack	Lift ...	Floor Label	Coordinat...	Reducible ...	RLL Redu...	NonReduc...	RLL[k] (un...	SL[k]	SLN[k]	RL[k]
1	CS1 (A-4)	1	Parking 1	30,0	539.3518	.6606	0	4.4319	0	0	0
2		1	Parking 2		539.3518	.6606	0	4.4319	0	0	0
3	CS2 (A-6)	1	Parking 1	50,0	1078.7036	.6	0	11.8524	0	0	0
4		1	Parking 2		1078.7036	.6	0	11.8524	0	0	0
5	CS3 (A-8)	1	Parking 1	70,0	539.3518	.6606	0	4.4319	0	0	0
6		1	Parking 2		539.3518	.6606	0	4.4319	0	0	0

The **Column Stack** entry displays the column stack label.

The **Lift No.** column displays the lift number for the physical column. Lift No. 1 is the lowermost physical column in a stack and the lifts are numbered sequentially moving up the column stack.

The **Floor Label** column displays the floor levels through which each column lift passes. Splice locations between floor levels are also listed and are indicated by a '+' symbol prior to the floor label. An entry of '+3rd Floor' indicates that there is a splice in the column stack above the 3rd Floor.

The **Coordinates** entry displays the plan coordinates of the column stack and is only listed on rows where the column stack label is shown.

The **Reducible Area** column records the cumulative roof area eligible for live load reduction that is tributary to the column lift at the floor level indicated. The area shown includes the area accumulated in the column lift at that floor level in addition to the area accumulated in the floor levels above.

The **RLL Reduce** column displays the roof live load reduction factors applicable to the column lift at the floor level indicated. These reduction factors are calculated based on the values listed in the 'Reducible Area' column of the spreadsheet.

The **RLL (unreduced)** column records the cumulative *Reducible* roof live load category forces and/or moments in a column lift at the floor level indicated. The forces and moments shown include the load accumulated in the column lift at that floor level in addition to the load accumulated in the floor levels above.

Note

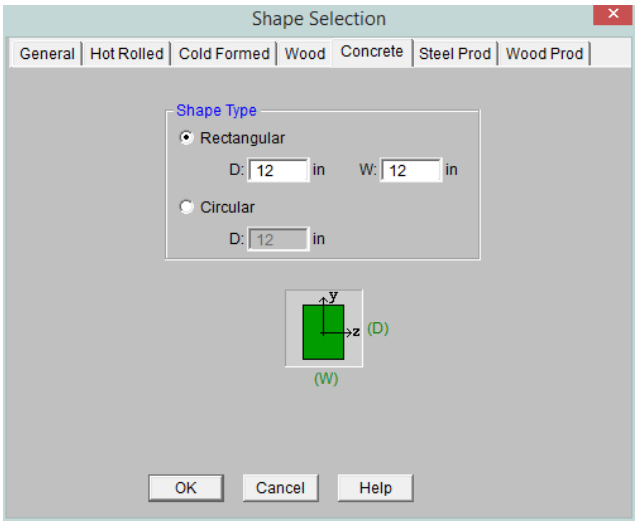
- The *Reducible* roof live load category forces and/or moments shown have NOT been reduced by the applicable reduction factors listed in the spreadsheet.

The **NonReduce RLL** column records the cumulative *Non-Reducible* roof live load category forces and/or moments in a column lift at the floor level indicated. The forces and moments shown include the load accumulated in the column lift at that floor level in addition to the load accumulated in the floor levels above.

The **SL**, **SLN**, and **RL** columns record the cumulative roof snow and rain load category forces and/or moments in a column lift at the floor level indicated. The forces and moments shown include the load accumulated in the column lift at that floor level in addition to the load accumulated in the floor levels above.

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 may click the button to have the program generate the desired shape definition for you.



Rectangular Sections

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

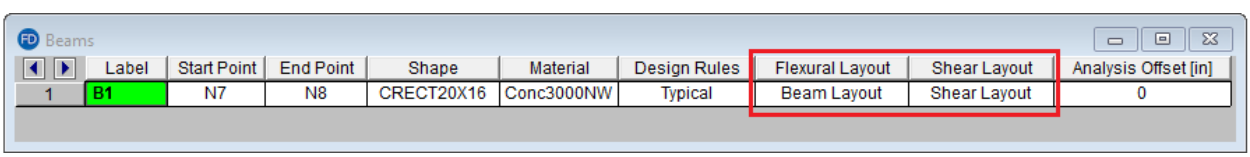
Circular Sections

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

Custom Rebar Layout Database

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

These reinforcement layouts may be assigned to beam or column members in the same way as the other concrete design parameters are assigned. This can be done from the **Concrete** tabs of the **Beams Spreadsheet** or **Column Stacks Spreadsheet**, or the **Modify Beams** or **Modify Columns Dialog**.

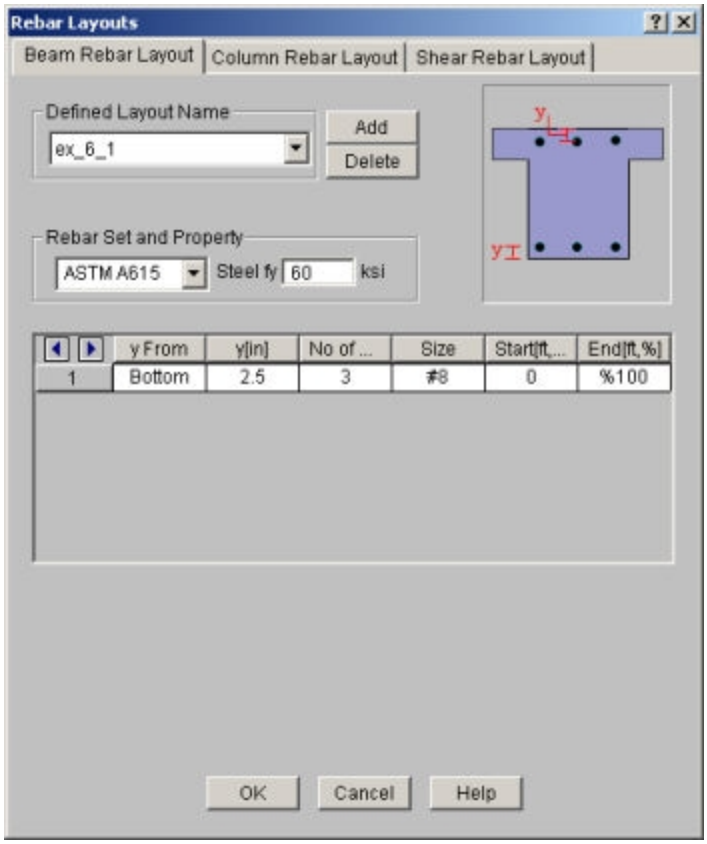


	Label	Start Point	End Point	Shape	Material	Design Rules	Flexural Layout	Shear Layout	Analysis Offset [in]
1	B1	N7	N8	CRECT20X16	Conc3000NW	Typical	Beam Layout	Shear Layout	0

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



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

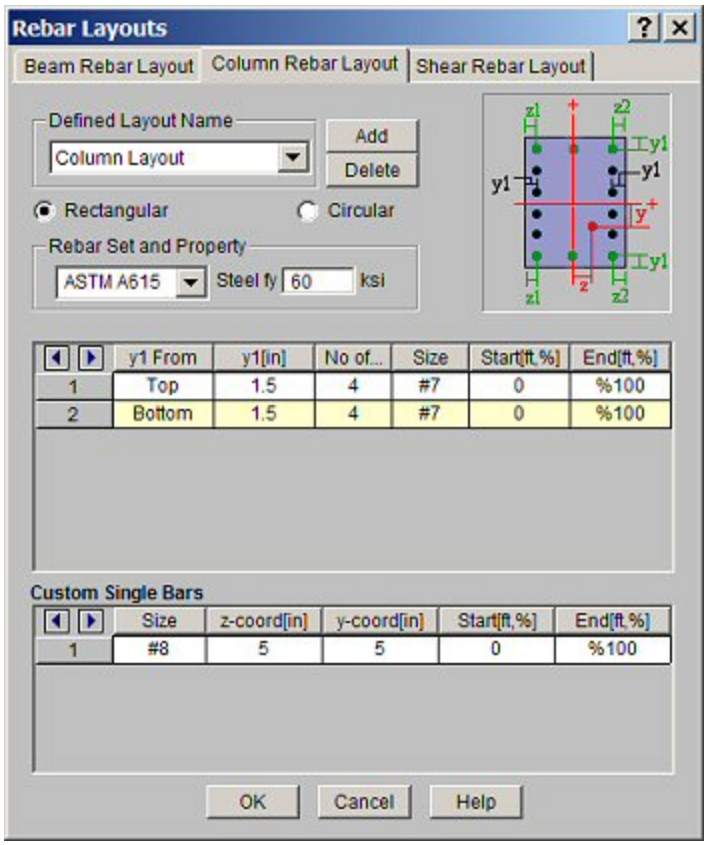
Note

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

Rectangular Column Rebar Layouts

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

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



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

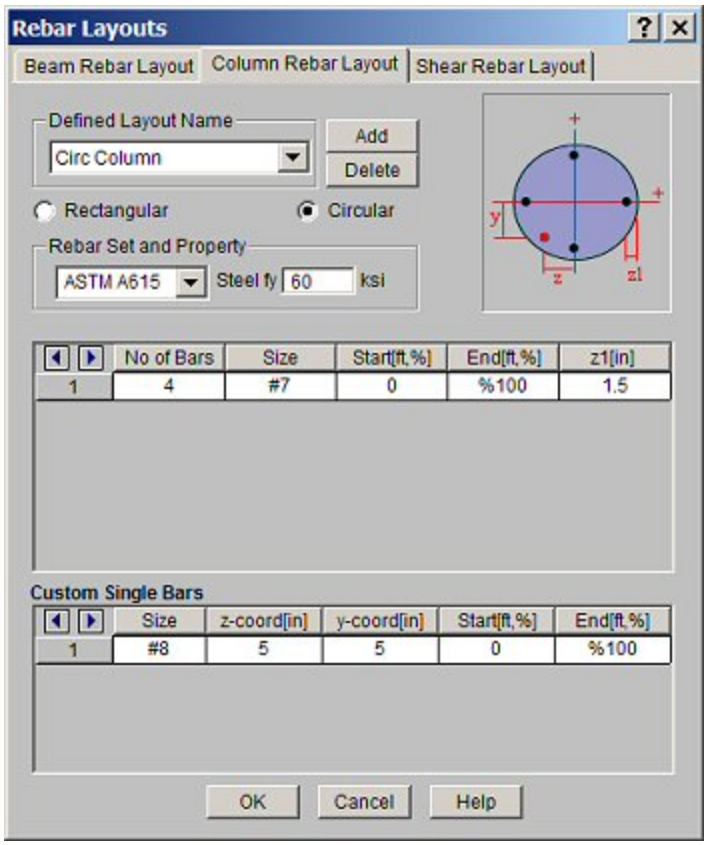
Custom single bars are specified by their y and z coordinates measured from the local y and z-axis respectively. A positive y coordinate would place the bar closer to the top fiber and a negative y coordinate would place the bar closer to the bottom fiber. Similarly, a positive z coordinate would place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

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

Circular Column Rebar Layouts

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

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



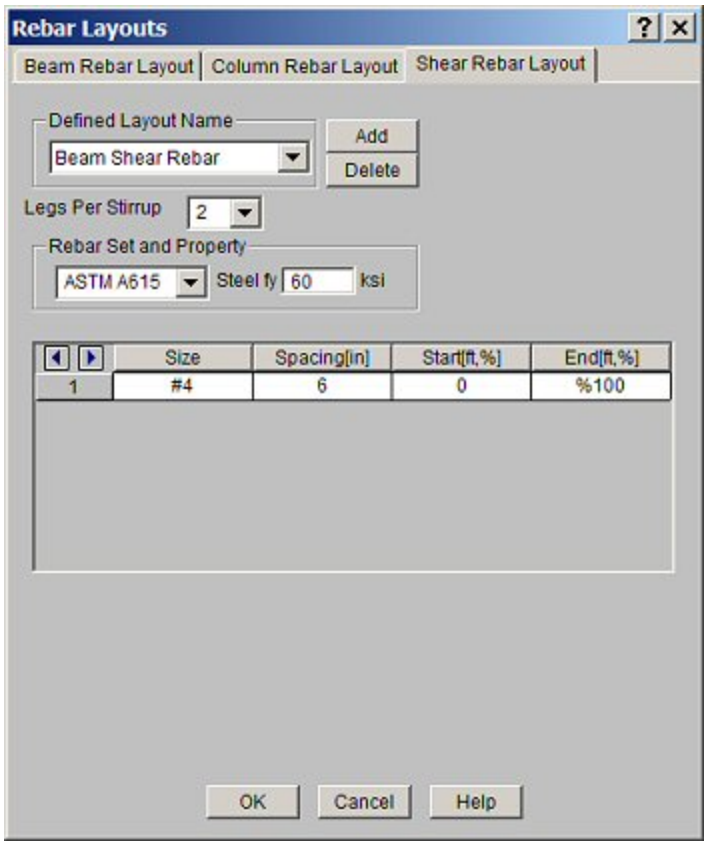
The **Start** and **End** locations dictate the location along the length of the member where these bars will be present. You can use these entries to specify partial length bars that will only be present in locations with a higher moment demand. If the bar should be present for the entire length of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

Note

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

Shear Rebar Layouts

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



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

Note

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

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

To Apply a Concrete Design Code

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

Concrete Spans

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

Beam members are supported by the following: Column Members and other Beam Members that are supporting that member.

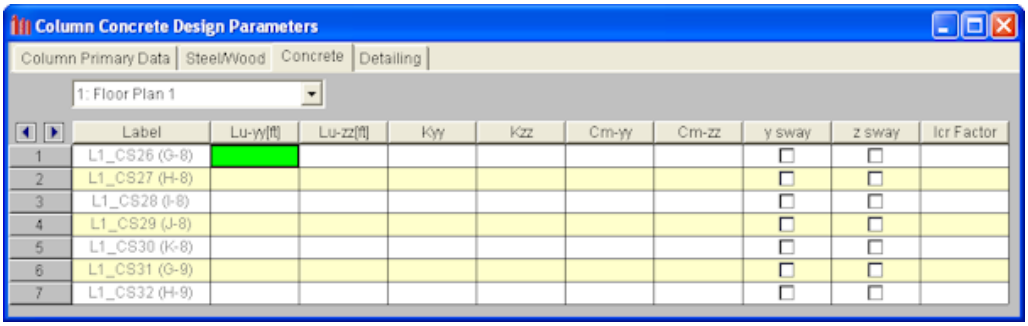
Column members are supported by the following: Beam Members and Rigid Diaphragms or Decks.

Note

- The program's ability to recognize spans is important because it will give you more relevant span to span information without overwhelming you with independent design results for each finite element segment that comprises your physical member.
- For continuous beam members, the program will evaluate the framing to determine which beams elements are supporting other beam elements so that only supporting members are treated as supports and not vice versa.

Concrete Design Parameters - Columns

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



The following parameters can be defined for each concrete column. The pull down list at the top of the spreadsheet allows you to toggle between floors.

Label

The **Label** field for column members is dictated by the label entry on the **Column Stacks Spreadsheet** and may not be edited here. If you would like to edit this entry, you must do so on the Column Stacks Spreadsheet.

Unbraced Length

You may specify unbraced lengths or have RISAFloor 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. They are listed on the **Concrete** tab of the **Columns Spreadsheet**. These **Lu** values are used to check the column for Euler buckling, and for the Moment Magnification Procedure in RISAFloor and older editions of the ACI code.

If the **Lu** values are not entered (left blank), the unbraced lengths for each segment of a physical column or lift will be automatically calculated by RISAFloor based on the distances between floor levels and/or splices. This means that each physical column can have multiple unbraced lengths if the entry is left blank. However, if a number is entered, RISAFloor will use that value for ALL segments of the physical column. Unbraced lengths that have been set or modified in RISA-3D will also be used in RISAFloor.

Note

- The calculated unbraced lengths are listed on the **Member Detail** report.
- The "**Segment**" code cannot be entered for column members in RISAFloor.

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. **Kyy** is for column type buckling about the member's local y-y axis and **Kzz** is for buckling about the local z-z axis.

If a value is entered for a **K Factor**, that value will be used for the segment of the physical column between the current floor level or splice above and/or the floor level or splice below. When in RISA-3D via RISAFloor, the largest K factor entered for any segment of a physical column will be used for the entire physical column. If an entry is not

made (left blank), the value will internally default to '1' for that column segment. See ACI 318-14 Section R6.2.5 (ACI 318-11 Section R10.10.1) for an explanation of how to calculate K Factors.

Sway Flags

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

Sway flags may be applied to any column segment at any floor level. However, when in RISA-3D via RISAFloor, if a sway flag is checked for any segment of a physical column, the entire physical column will be assumed subject to sway. Beam members in RISAFloor are assumed to be braced against sway.

Cm – Equivalent Moment Correction Factor

The **Cm Coefficients** are used to check the column for Euler buckling, and for the Moment Magnification Procedure in RISAFloor and older editions of the ACI code. **Cm-yy** is for bending about the columns's local y-y axis and **Cm-zz** is for bending about the local z-z axis. If these entries are left blank they will be automatically calculated. Each segment of a physical column will receive its own calculated value. However, when in RISA-3D via RISAFloor, the *highest* Cm value calculated for a column segment in RISAFloor will be used for the entire physical column in RISA-3D.

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

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 will be used.

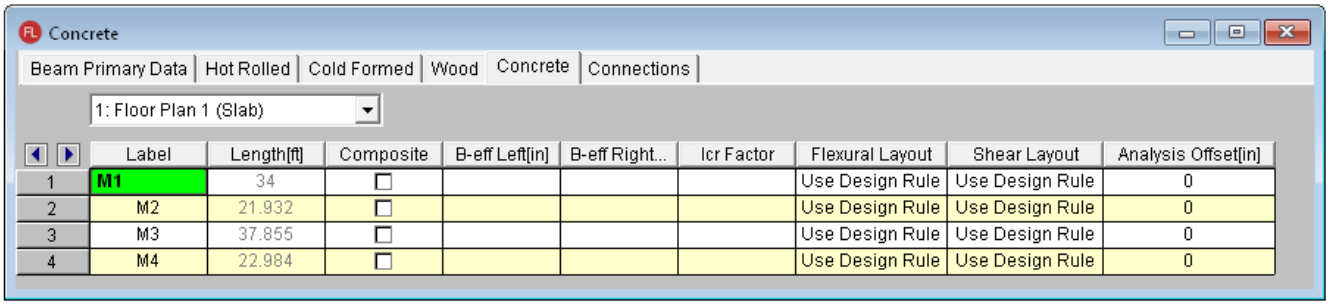
Note

- The **Icr Factor** will be 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.

Note

Concrete Design Parameters - Beams

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



	Label	Length[ft]	Composite	B-eff Left[in]	B-eff Right...	Icr Factor	Flexural Layout	Shear Layout	Analysis Offset[in]
1	M1	34	☐				Use Design Rule	Use Design Rule	0
2	M2	21.932	☐				Use Design Rule	Use Design Rule	0
3	M3	37.855	☐				Use Design Rule	Use Design Rule	0
4	M4	22.984	☐				Use Design Rule	Use Design Rule	0

The following parameters can be defined for each concrete member.

Label

You may assign a unique **Label** to all of the members. Each label must be unique, so if you try to enter the same label more than once you will get an error message.

Length

The beam **Length** is reported in the second column. This value may not be edited as it is dependent on the beam start and end points listed on the **Primary Data** tab. It is listed here as a reference only.

Flexural and Shear Rebar Layout

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

Icr Factors (Cracked Moment of Inertia Factors)

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

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 will default 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 will default to a value of 0.4 for beams and 0.8 for columns.

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

Note

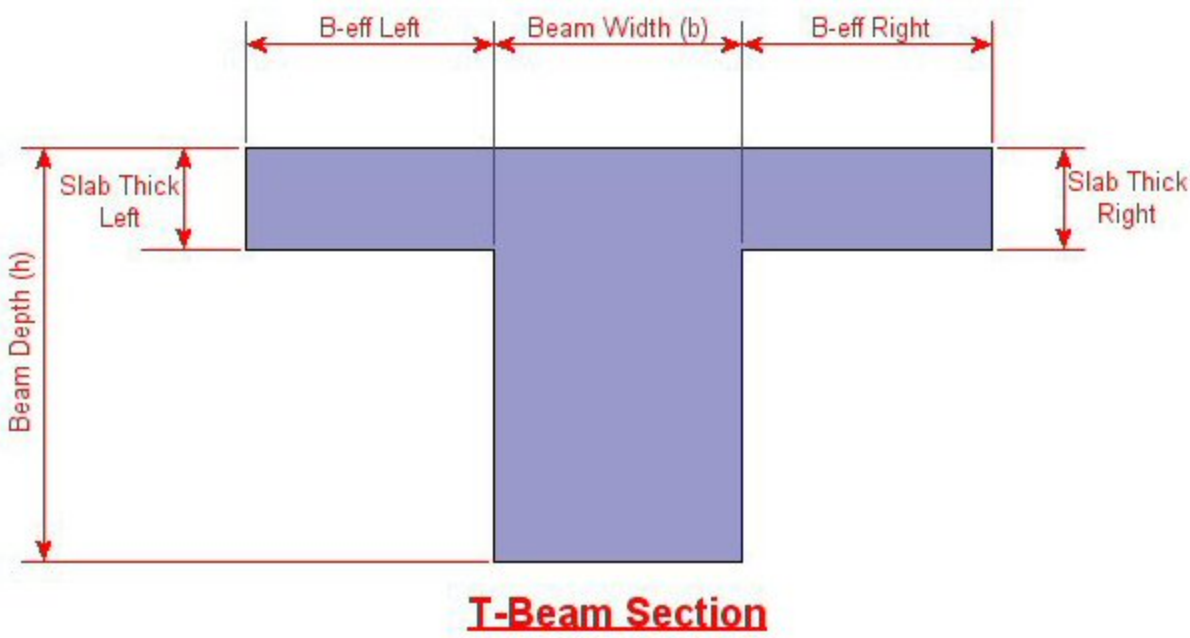
- The **Icr Factor** will be ignored if the **"Use Cracked Stiffness"** box is not checked on the **Concrete** tab of the **Model Settings** dialog.

T-beam & L-beam Sections- RISAFloor ES Only

T-beams and **L-beams** may be specified by indicating a beam as composite when creating the beam or after the beam is created by checking the composite check box on the **Concrete** tab of the **Beams Spreadsheet**. This modification may also be made graphically via the **Modify Properties** tab of the **Draw Beams** tool.

Note:

- The T-beam & L-beam design is done only when a one-way slab is defined. See One-Way Slab Design for more information.
- Composite beam design is supported in both one-way and two-way slabs in RISAFloor ES. The composite action between the beam and slab is considered in the finite element analysis for both slab types, the only difference is that the beam rebar design uses rectangular shape in two-way slab and T or L shape in one-way slab (if Composite checkbox is on).



RISAFloor will automatically calculate the effective slab widths, **B-eff Left** and **B-eff Right**, based on ACI 318-14 Table 6.3.2.1) ACI 318-11 Sections 8.12.2 and 8.12.3) if no values are entered by the user in the Beams Spreadsheet. You may override the values calculated by RISAFloor by entering your own values for B-eff Left and B-eff Right in the spreadsheet. It should be noted that if the value entered is in excess of the values allowed by ACI 318-14 Table 6.3.2.1 (ACI 318-11 Sections 8.12.2 and 8.12.3), RISAFloor will reduce the value to meet the code requirements.

If the composite check box is un-checked, the beam will be designed as the shape indicated on the **Beam Primary Data** tab and any values entered for either **B-eff Left** or **B-eff Right** will be ignored.

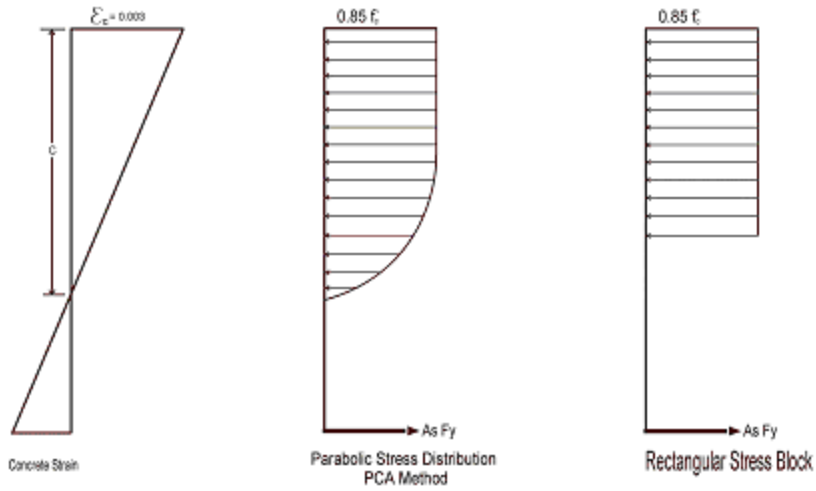
RISAFloor automatically calculates the slab thicknesses for T-beams and L-beams based on the slab thickness defined in each slab area. See the Slab Definitions Spreadsheet for information on defining slab thicknesses.

Note:

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

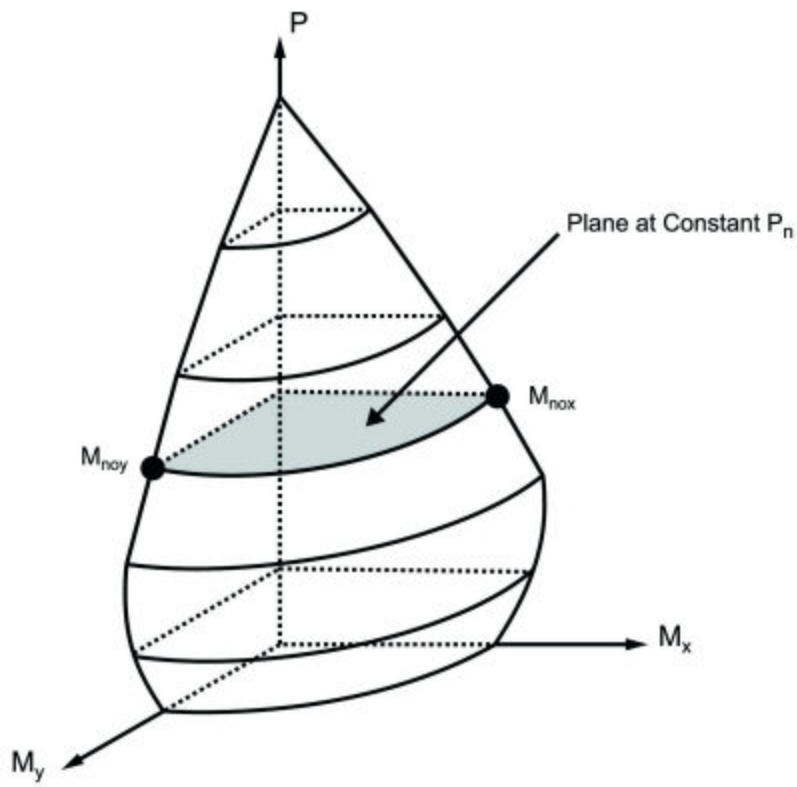
Parabolic vs. Rectangular Stress Blocks

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



Biaxial Bending of Columns

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



Biaxial Failure Surface
PCA Load Contour Method

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*span length and $b_{eff,i}$ =L_o/10
- $\varphi_{concrete}$ = 1.5
- φ_{rebar} = 1.15
- Maximum bar spacing for beams = 300 mm

Tension Development Length

- α_{ct}= 1 (assumed in Eq 3.16)
- η_1= η_2=1 (assumed in Eq 8.2 to calculate bond stress)
- λ_3, λ_5, λ_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/γ_c for Eq 6.2.a
- v_{min} =0.035 $k^{3/2}$ $f_{ck}^{1/2}$
- ν =.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
- λ_{lim} = 20 A B C/$n^{1/2}$ (A=.7, B= 1.1, C=0.7 for unbraced)
- K_ϕ =1 in Eq 5.34; the effect of creep is neglected.

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**.

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 will currently be 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 (Vc) uses equations in Table 22.5.5.1. Note that for members meeting the minimum shear reinforcement requirement (Av≥Av,min), Vc is taken as the larger of the results calculated by the equations (a) and (b) in the table. ACI 318-19 code suggests ρw 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 ρw 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*λ*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).

Limitations - Canadian Code

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

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

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

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

Limitations - 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 – Ku in AS code is always assumed to be less than 0.4.

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

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

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

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

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

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

Limitations - British

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

Cracked Sections – Icracked defaults to 1.0 for the British code. But, a user entered value may be entered if desired. 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 British specification rather than performing a complete biaxial condition.

Limitations - Euro

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

Cracked Sections – Icracked defaults to 1.0 for EuroCode. But, a user entered value may be entered if desired. 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 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, a user entered value may be entered if desired. 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 may be missed and an un-conservative design may result.

Warning: No design for spans less than 1 ft.

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

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

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

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

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

Concrete Member - Design Results

You can access the **Concrete Results Spreadsheets** by selecting the **Results Menu** and then selecting **Designs**, **Code Checks**, **Shear**, or **Bending** for beam members or **Column Results** for column members. 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.

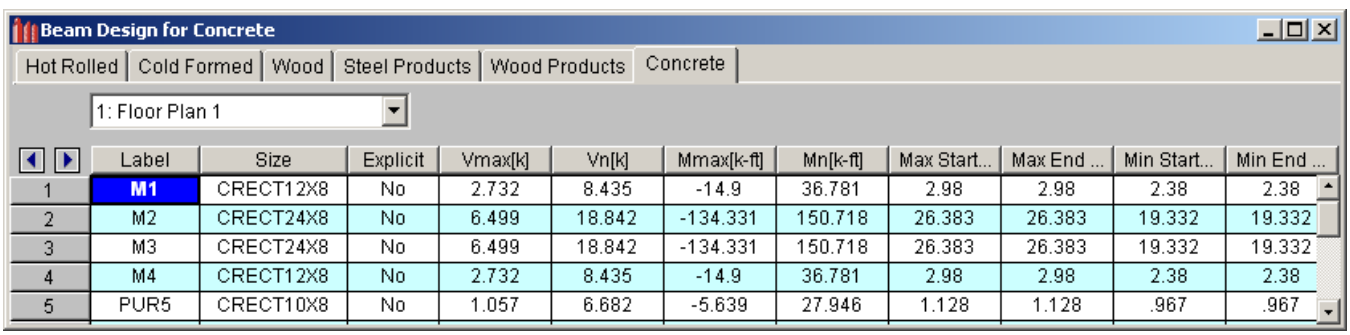
For beam flexural design, the required bars are based on the envelope moment diagrams. For column flexural design, the required bars for each load combination are calculated at various sections for the moments and axial forces at those sections. The required bars for all load combinations are then enveloped. 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 six following spreadsheets: **Design Results**, **Code Checks**, **Shear Results**, **Bending Results**, **Beam Bending Reinforcement**, and **Beam Shear Reinforcement**.

Design Results Spreadsheet

The **Design Results Spreadsheet** shows the controlling ultimate strengths and corresponding nominal strengths for each beam. The spreadsheet can be accessed by clicking on the **Results Menu** and clicking **Design**. The concrete results are listed on the **Concrete** tab. The pull down list at the top of the spreadsheet allows you to toggle between floor levels.



	Label	Size	Explicit	Vmax[k]	Vn[k]	Mmax[k-ft]	Mn[k-ft]	Max Start...	Max End ...	Min Start...	Min End ...
1	M1	CRECT12X8	No	2.732	8.435	-14.9	36.781	2.98	2.98	2.38	2.38
2	M2	CRECT24X8	No	6.499	18.842	-134.331	150.718	26.383	26.383	19.332	19.332
3	M3	CRECT24X8	No	6.499	18.842	-134.331	150.718	26.383	26.383	19.332	19.332
4	M4	CRECT12X8	No	2.732	8.435	-14.9	36.781	2.98	2.98	2.38	2.38
5	PUR5	CRECT10X8	No	1.057	6.682	-5.639	27.946	1.128	1.128	.967	.967

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

The **Size** 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 or deflection requirements (see Design Optimization), or adding more shapes to the available Redesign List (see Appendix A – Redesign Lists).

The **Explicit** column displays "Yes" if the beam has been locked to an explicit beam size by the user and "No" when the program has selected the shape used. When you have chosen a specific shape to override the programs automatic redesign, that beam shape becomes "locked" and will not be automatically redesigned by the program.

Note

- To "unlock" a beam, you can use the beam – modify tool to assign a shape group. If the model has already been solved, you may optimize a beam by using the Member Redesign dialog. See Member Redesign for more details.
- Live load reduction effects are considered for the applicable values in this spreadsheet.

The **Vmax** column shows the governing maximum ultimate shear and the corresponding nominal shear strength of the beam is shown in the **Vn** column. Note that the nominal shear strength shown, Vn, has NOT been reduced by the strength reduction factor, Phi.

The **Mmax** column shows the governing maximum ultimate bending moment and the corresponding nominal moment strength of the beam is shown in the **Mn** column. Note that the nominal moment strength shown, Mn, has NOT been reduced by the strength reduction factor, Phi.

The **Max Start & End Reaction** columns display the maximum "start" and "end" reactions of the beam for ALL load combinations. If the "Show Factored End Reactions" check box in **Model Settings** dialog is left unchecked, these displayed forces are not factored. If the box is checked, then the displayed forces will have been multiplied by the factors in the load combinations. The sign convention assigns positive reactions to downward forces. Negative reactions, if they occur, would indicate uplift.

The **Min Start & End Reaction** columns display the minimum start and end reaction of the beam.

Code Checks Spreadsheet

The **Code Checks Spreadsheet** summarizes the code check results for the beams and may be accessed by selecting **Code Checks** on the **Results Menu**. The spreadsheet has five tabs: Hot Rolled, Cold Formed, Wood, Concrete, and Wood Products. The concrete results are listed on the **Concrete** tab. The pull down list at the top of the spreadsheet allows you to toggle between floor levels.

Beam Code Summary for Concrete

Hot Rolled | Cold Formed | Wood | Concrete | Wood Products

2: Level 3

	Label	Size	Explicit	Material	Bending Check	Loc[ft]	LC	Defl Check	Loc[ft]	Cat	Shear Check	Loc[ft]	LC
1	PUR310	CRECT8X8	No	Conc3000N	.025	1.75	2	.036	0	LL	.074	.474	2
2	PUR311	CRECT8X8	No	Conc3000N	.025	1.75	2	.036	0	LL	.074	.474	2
3	PUR312	CRECT8X8	No	Conc3000N	.025	1.75	2	.036	0	LL	.074	.474	2
4	PUR313	CRECT8X8	No	Conc3000N	.025	1.75	2	.036	0	LL	.074	.474	2
5	PUR314	CRECT8X8	No	Conc3000N	.025	1.75	2	.036	0	LL	.074	.474	2

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

The **Size** 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 or deflection requirements (see Design Optimization), or adding more shapes to the available Redesign List (see Appendix A – Redesign Lists).

The **Explicit** column displays "Yes" if the beam has been locked to an explicit beam size by the user and "No" when the program has selected the shape used. When you have chosen a specific shape to override the programs automatic redesign, that beam shape becomes "locked" and will not be automatically redesigned by the program.

Note

- To "unlock" a beam, you can use the beam – modify tool to assign a shape group. If the model has already been solved, you may optimize a beam by using the Member Redesign dialog. See Member Redesign for more details.

The **Material** column displays the material label from the **Concrete** tab of the **Materials Spreadsheet** that was assigned to the beam.

The **Bending Check** and **Shear Check** columns display the maximum bending check and shear check calculated by the program. These values are equal to the governing ultimate moment or shear divided by the nominal moment or shear strength. You can see the details of these values in the **Bending Results** or **Shear Results Spreadsheets**. This check is calculated at 100 stations along each beam for each load combination and the maximum check is reported. See Results Spreadsheet for more information.

The **Deflection Check** displays the maximum deflection check calculated by the program. This value is equal to the ratio of actual deflection to allowable deflection. You can see the details of these values in the **Deflection Results Spreadsheet**. This check is calculated at 100 stations along each beam and the maximum check is reported. See Beam Results - Deflection for more information.

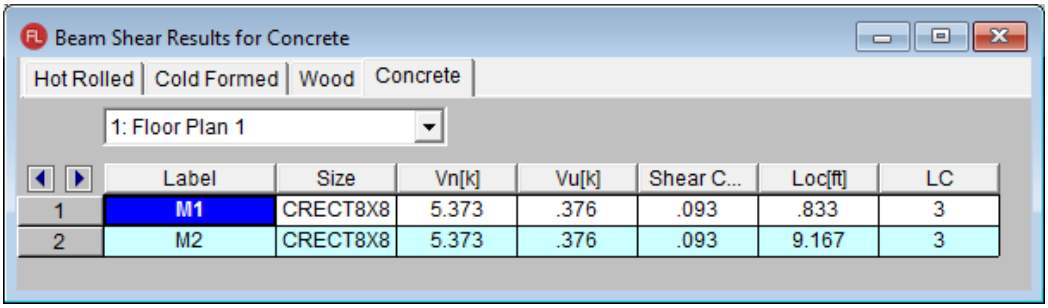
The **Loc** columns display the location along the member where the maximum bending, shear, or deflection check occurs.

The **LC** columns display the governing load combination that produced the maximum code check.

The **Cat** column displays the controlling load category which resulted in the maximum deflection check.

Shear Results Spreadsheet

The **Shear Results Spreadsheet** records the shear results for each beam and may be accessed by selecting **Shear** on the **Results Menu**. The concrete results are listed on the **Concrete** tab. The pull down list at the top of the spreadsheet allows you to toggle between floor levels.



	Label	Size	Vn[k]	Vu[k]	Shear C...	Loc[ft]	LC
1	M1	CRECT8X8	5.373	.376	.093	.833	3
2	M2	CRECT8X8	5.373	.376	.093	9.167	3

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

The **Size** 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 or deflection requirements (see Design Optimization), or adding more shapes to the available Redesign List (see Appendix A – Redesign Lists).

The **Vn** column displays the calculated nominal shear strength of the beam based on the applicable code. The **Vu** column displays the maximum ultimate shear force that the member experiences.

Note

- The nominal shear strength shown, Vn, has NOT been reduced by the strength reduction factor, Phi.
- Live load reduction effects are considered for the applicable values in this spreadsheet.

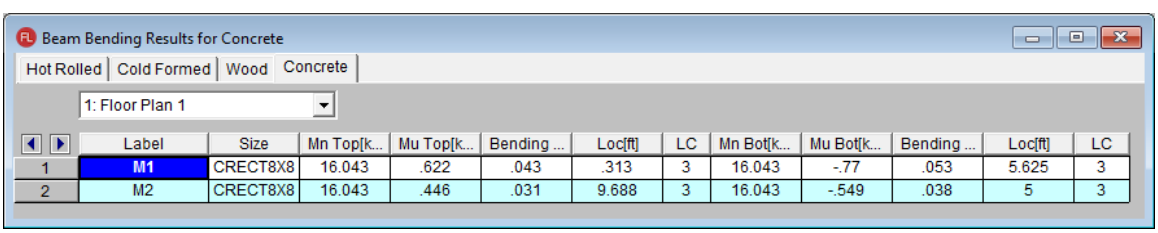
The **Shear Check** column displays the maximum shear check calculated by the program. This value is equal to the governing ultimate shear force divided by the nominal shear strength. This check is calculated at 100 stations along each beam for each load combination and the maximum check is reported. See Results Spreadsheet for more information.

The **Loc** column displays the location along the member where the maximum shear check occurs.

The **LC** column display the governing load combination that produced the maximum code check.

Bending Results Spreadsheet

The **Bending Results Spreadsheet** records the bending results for each beam and may be accessed by selecting **Bending** on the **Results Menu**. The concrete results are listed on the **Concrete** tab. The pull down list at the top of the spreadsheet allows you to toggle between floor levels.



	Label	Size	Mn Top[k...	Mu Top[k...	Bending ...	Loc[ft]	LC	Mn Bot[k...	Mu Bot[k...	Bending ...	Loc[ft]	LC
1	M1	CRECT8X8	16.043	.622	.043	.313	3	16.043	-.77	.053	5.625	3
2	M2	CRECT8X8	16.043	.446	.031	9.688	3	16.043	-.549	.038	5	3

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

The **Size** 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 or deflection requirements (see Design Optimization), or adding more shapes to the available Redesign List (see Appendix A – Redesign Lists).

The **Mn Top** and **Mn Bot** columns display the calculated nominal moment strength of each beam based on the applicable code. The **Mu Top** and **Mu Bot** columns display the calculated ultimate bending moment of each beam.

Note

- The nominal moment strengths shown, **Mn Top** and **Mn Bot**, have NOT been reduced by the strength reduction factor, Phi.
- The **Mn Top** and **Mu Top** values represent bending that produces tension in the top fiber of the beam.
- The **Mn Bot** and **Mu Bot** values represent bending that produces tension in the bottom fiber of the beam.
- The sign convention for **Mu Top & Bot** is defined so that positive bending will result in tension in the top fiber of the beam. **The Mn Top & Bot** values are absolute values and therefore always shown as positive.
- Live load reduction effects are considered for the applicable values in this spreadsheet.

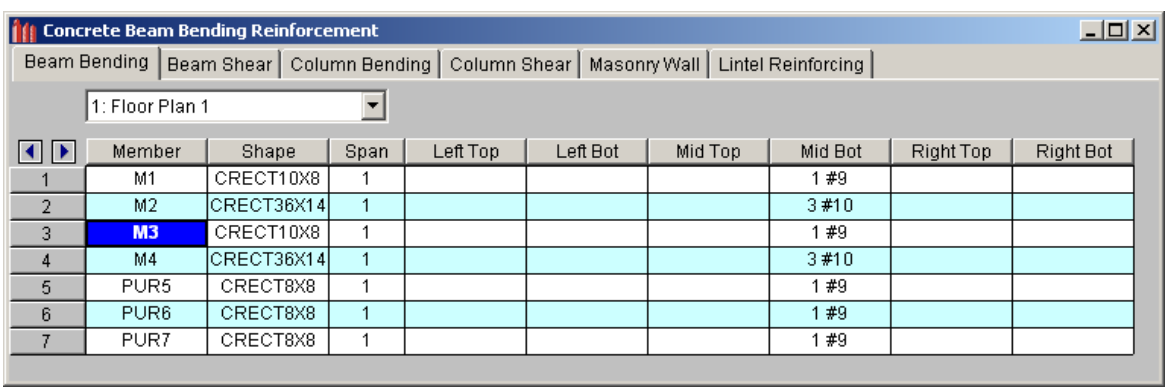
The **Bending Chk Top** and **Bending Chk Bot** columns display the maximum bending checks calculated by the program. These values are equal to the governing ultimate moment divided by the nominal moment strength. The **Bending Chk Top** value represents bending that produces tension in the top fiber of the beam and the **Bend Chk Bot** value represents bending that produces tension in the bottom fiber of the beam. These checks are calculated at 100 stations along each beam for each load combination and the maximum checks are reported. See Results Spreadsheet for more information.

The **Loc** column displays the location along the member where the maximum bending checks occur.

The **LC** columns display the governing load combination that produced the maximum code check.

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 **Beams ▸ Concrete Reinforcing** on the **Results Menu** and the results are listed on the **Beam Bending** tab. The pull down list at the top of the spreadsheet allows you to toggle between floor levels.



	Member	Shape	Span	Left Top	Left Bot	Mid Top	Mid Bot	Right Top	Right Bot
1	M1	CRECT10X8	1				1 #9		
2	M2	CRECT36X14	1				3 #10		
3	M3	CRECT10X8	1				1 #9		
4	M4	CRECT36X14	1				3 #10		
5	PUR5	CRECT8X8	1				1 #9		
6	PUR6	CRECT8X8	1				1 #9		
7	PUR7	CRECT8X8	1				1 #9		

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

The **Shape** column displays the beam size. When no adequate beam could be found from the available shapes list, 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 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.

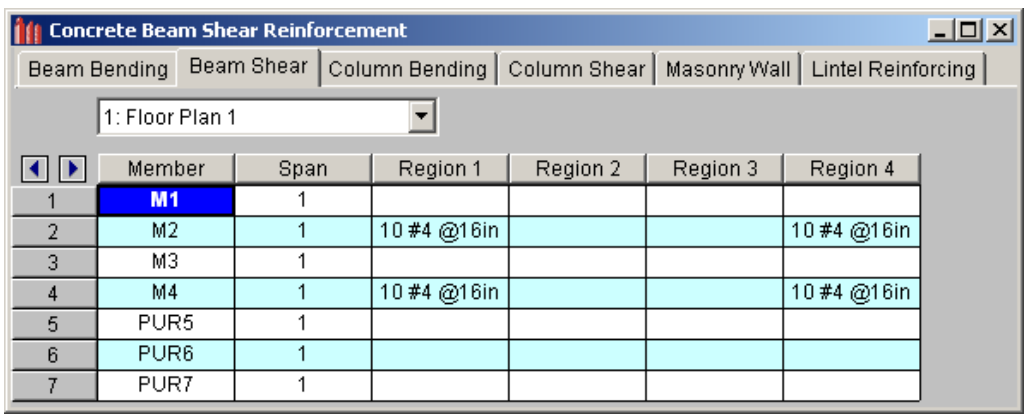
The **Left Top**, **Left Bot**, **Mid Top**, **Mid Bot**, **Right Top**, and **Right Bot** entries record the number and size of flexural reinforcement bars that are required in each of the six **Reinforcement Sections**. The first number indicates the number of parallel reinforcement bars in that section. 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 **Beams** ▸ **Concrete Reinforcing** on the **Results Menu** and the results are listed on the **Beam Shear** tab. The pull down list at the top of the spreadsheet allows you to toggle between floor levels.



	Member	Span	Region 1	Region 2	Region 3	Region 4
1	M1	1				
2	M2	1	10 #4 @16in			10 #4 @16in
3	M3	1				
4	M4	1	10 #4 @16in			10 #4 @16in
5	PUR5	1				
6	PUR6	1				
7	PUR7	1				

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

Note

- If '2' shear regions are selected on the **Concrete** tab of the **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 Vu and Mu themselves.

Column Results

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

Column Results

The **Column Results Spreadsheet** summarizes the code check results and records the design results for columns and may be accessed by selecting **Column Results** on the **Results** menu. The concrete results are listed on the **Concrete** tab.

Concrete Column Code Checks

Hot Rolled | Cold Formed | Wood | Concrete

	Stack	Lift	Shape	UC Max	Elev[ft]	UC LC	Shear ...	UC LC	Elev[ft]	Dir	Phi used
1	(K-1)	1	CRECT8X8	.004	0	3	.005	3	0	z	.9
2	(K-2)	1	CRECT8X8	.001	0	3	0	3	0	z	.9
3	(K-3)	1	CRECT8X8	.004	0	3	.005	3	0	z	.9

The **Stack** column displays the column stack label.

The **Lift** column displays the lift number for the physical column. Lift No. 1 is the lowermost physical column in a stack and the lifts are numbered sequentially moving up the column stack.

The **Shape** column displays the physical column 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 **UC Max** column displays the maximum combined axial and bending check calculated by the program. This value is equal to the combined ultimate axial and ultimate bending demand divided by the actually column nominal capacity or strength. You can see the details of this value in the subsequent **Axial Capacity (Pn)** and **Moment Capacity (Mny & Mnz)** columns of this spreadsheet. This check is calculated at 100 stations along each physical column for each load combination and the maximum check is reported. See Results Spreadsheet for more information.

The **Elev** columns displays the absolute elevation along the column stack where the maximum axial and bending code check occurs.

The **UC LC** column displays the controlling load combination which produced the maximum axial and bending code check.

The **Shear UC** column displays the maximum shear check calculated by the program. This value is equal to the ultimate shear demand divided by the actual shear capacity or strength of the column. This check is calculated at 100 stations along each column for each load combination and the maximum check is reported. See Results Spreadsheets for more information.

The **UC LC** column displays the controlling load combination which produced the maximum shear code check.

The **Elev** columns displays the absolute elevation along the column stack where the maximum shear code check occurs.

The **Dir** column displays the column local axis along which the maximum shear check occurs.

The **Phi Used** column lists the applicable **Strength Reduction Factor** ***(ϕ)*** to be applied to the nominal capacities listed in the subsequent strength columns.

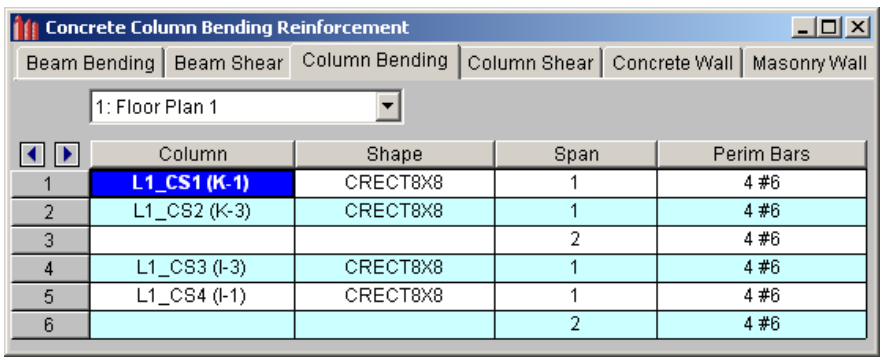
The **Pn** column displays the calculated compressive capacity or strength of each column lift based on the applicable code. Note that this value has NOT been reduced by the strength reduction factor, Phi.

The **Mny** and **Mnz** columns display the calculated nominal moment strength of each column lift based on the applicable code. Note that this value has NOT been reduced by the strength reduction factor, Phi.

The **Vny** and **Vnz** columns display the calculated nominal shear strength of each column lift based on the applicable code. Note that this value has NOT been reduced by the strength reduction factor, Phi.

Column Bending Reinforcement Spreadsheet

The **Column Bending Reinforcement Spreadsheet** shows the perimeter flexural reinforcement steel required in each span of each column lift. This spreadsheet may be accessed by selecting **Concrete Reinforcing** on the **Results Menu** and the results are listed on the **Column Bending** tab. The pull down list at the top of the spreadsheet allows you to toggle between floor levels.



	Column	Shape	Span	Perim Bars
1	L1_CS1 (K-1)	CRECT8X8	1	4 #6
2	L1_CS2 (K-3)	CRECT8X8	1	4 #6
3			2	4 #6
4	L1_CS3 (I-3)	CRECT8X8	1	4 #6
5	L1_CS4 (I-1)	CRECT8X8	1	4 #6
6			2	4 #6

The **Column** field displays the column label. The column label is a combination of the stack label and the lift number. The column stack label is the portion of the label preceding the underscore symbol, "_". The column lift number follows the extension, "_L".

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 bottom of the column lift and subsequent spans are numbered '2', '3', '4', and so forth moving from the bottom to the top of the column lift.

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.

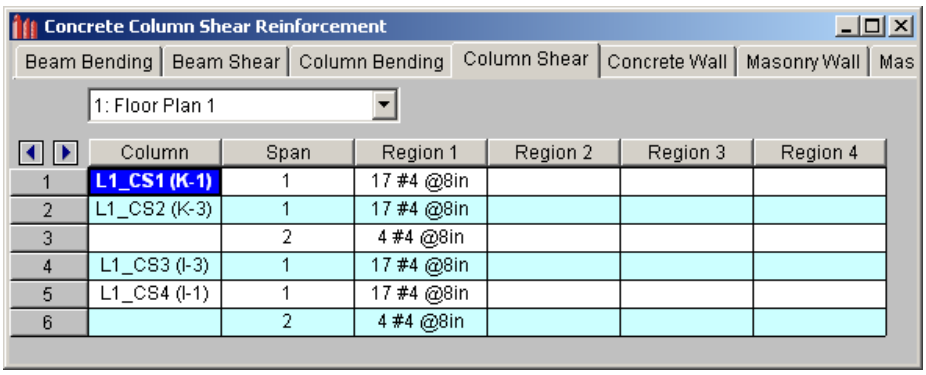
Note

- 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 lift. This spreadsheet may be accessed by selecting **Concrete Reinforcing** on the **Results Menu** and the results are listed on the **Column Shear** tab. The pull down list at the top of the spreadsheet allows you to toggle between floor levels.



	Column	Span	Region 1	Region 2	Region 3	Region 4
1	L1_CS1 (K-1)	1	17 #4 @8in			
2	L1_CS2 (K-3)	1	17 #4 @8in			
3		2	4 #4 @8in			
4	L1_CS3 (I-3)	1	17 #4 @8in			
5	L1_CS4 (I-1)	1	17 #4 @8in			
6		2	4 #4 @8in			

The **Column** field displays the column label. The column label is a combination of the stack label and the lift number. The column stack label is the portion of the label preceding the underscore symbol, "_". The column lift number follows the extension, "_L".

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

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

Note

- If '2' shear regions are selected on the Concrete tab of the 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 RISAFloor 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:

- - This 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.

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.

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%

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, **Vn**, in each region. The portion of the nominal shear strength provided by the concrete and the steel, **Vc** and **Vs** respectively, is given for each region. The area of steel required, **As Reqd**, and the area of steel provided, **As Prvd**, are also given for each shear region and are reported as 'area of steel per unit dimension', i.e. in^2/in or mm^2/mm.

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

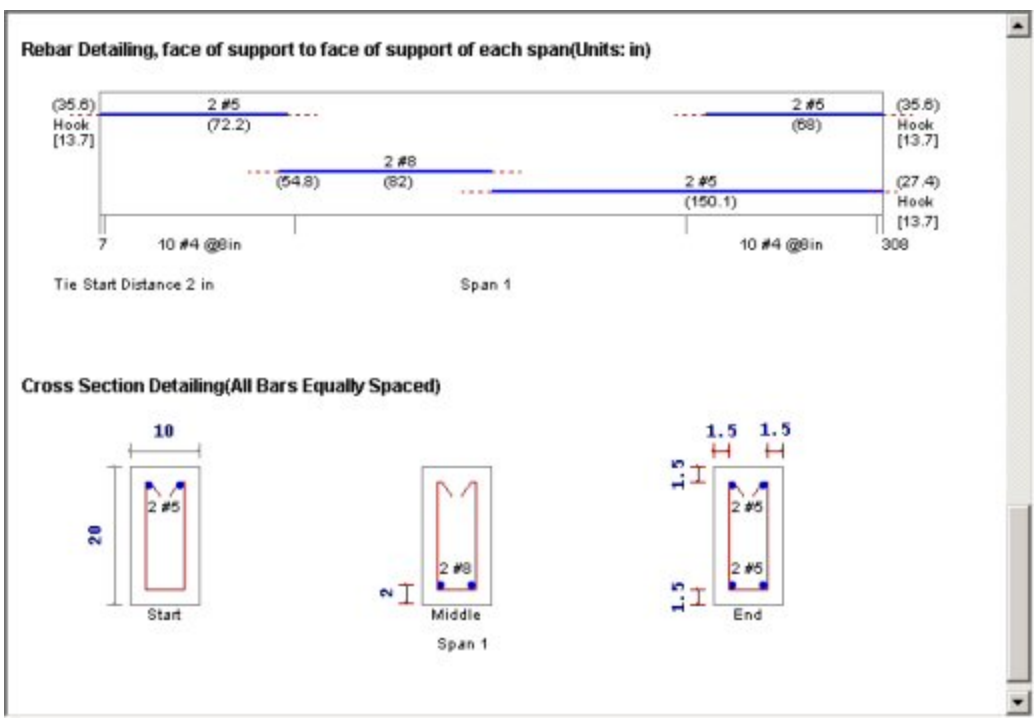
The last section of the detail report shows the **Beam Reinforcement Detailing Diagrams**. The **Rebar Detailing** portion of the report shows elevation views of the beam complete with top and bottom flexural reinforcement indicated for the left, middle, and right portions of each span. The number and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

Note:

- Development lengths are calculated per ACI 318-14 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.



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

Column Detail Reports

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

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

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

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

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

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

Note:

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

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

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

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

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

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

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

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.

Magnified Moments / Slenderness Effects

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

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

For Non-Sway frames, the assumption is that EI = 0.25*Ec*Ig. This is equivalent to setting B_{dns} to 0.6 in ACI 318-14 Equation 6.6.4.4.4c (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.

Connections

The **Connection Rules** spreadsheets and settings are used for two purposes in RISAFloor:

1. The analysis and design of columns with respect to the eccentricity effects caused by connections.
2. The integration with RISAConnection for the design of hot-rolled steel connections.

Note:

- A RISAConnection license is ***not*** required for the analysis of connection eccentricity in the RISAFloor model.

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 list below represents the available connection options from the current version of RISAConnection:

- **Beam to Column Shear Connections** (shear tab, clip angle, end plate).
    - 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).
    - 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).
    - 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).
    - Direct Weld and Flange Plate moment connection columns may be Wide Flanges or HSS Rectangular Tubes.
    - Extended End Plate and Flush Plate moment connection columns may only be Wide Flanges.
    - Beams may only be Wide Flanges.
- **Beam to Column Web Moment Connections** (direct weld, flange plates).
    - Columns may only be Wide Flanges.
    - Beam may only be Wide Flanges.
- **Beam to Column Web Moment Connections** (Continuous beam over column).
    - 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).
    - Columns may only be Wide Flanges.
    - Beams may only be Wide Flanges.
- **Single Column Base Plate Connections**
    - Column may be a Wide Flange, Tube, or Pipe.
- **Brace to Base Plate Connections**
    - Column may be a Wide Flange, Tube, or Pipe.
- **Brace to Column Base Plate Connections**
    - Column may be a Wide Flange, Tube, or Pipe.
    - Brace may be HSS Tube, HSS Pipe, Single Angle, Double Angle, WTs, or Channels.

See Troubleshooting below for more assistance with connection type limitations.

Integrated Connection Design Procedure

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

1. Completing 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. Defining Connection Rules

Next, you must define Connection Rules. These rules allow you to define which types of connections you want to design in your model. You must have a rule for each type of connection you want considered. You may also want to create two separate rules for the same connection Type so you can manipulate these groups separately.

Connection Rules

Hot Rolled

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

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

- **Label** - This is how you will identify your Connection Rule in other areas within the program. Each Connection Rule must have a unique **Label**.
- **Conn Type** - The **Conn Type** drop-down filters the list of **Types** into Shear, Moment, or Brace connections.
- **Type** - The **Type** refers to the different types of connections currently considered in RISAConnection. You will need to set up at least one connection rule for each connection type in your project. When you solve the connections in RISAConnection the member ends assigned to an individual Connection Rule will be grouped together.
    - Note: You may have many double angle shear connections in your project. However, some are bolted with A325 bolts and some use A490 bolts. You should define these as separate Connection Rules.
- **Beam Conn** - This option defines how the beam is connected to the supporting pieces (plate, clip angle, etc). When you go to RISAConnection all connections grouped in this Connection Rule will default to this type of connection.
    - Note:Many connections do not have options for these, so "N/A" will be shown
- **Col/Girder Conn** - This option defines how the supporting member (column or girder) is connected to the supporting pieces (plate, clip angle, etc.). When you go to RISAConnection all connections grouped in this Connection Rule will default to this type of connection.
    - Note: Many connections do not have options for these, so "N/A" will be shown
- **Eccentricity** - This value has no effect on the RISAConnection integration. Instead it is used for analysis purposes. In models integrated with RISAFloor, beam/column shear connections add an automatic eccentricity equal to half the column's depth automatically (see the Model Settings-Solution tab for more information on this). However an *additional* eccentricity away from the column face may be entered in the Connection Rules spreadsheet. The value entered here should be equal to the distance between the face of column and the resultant beam end reaction (i.e. centroid of bolt/weld group on beam web).

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

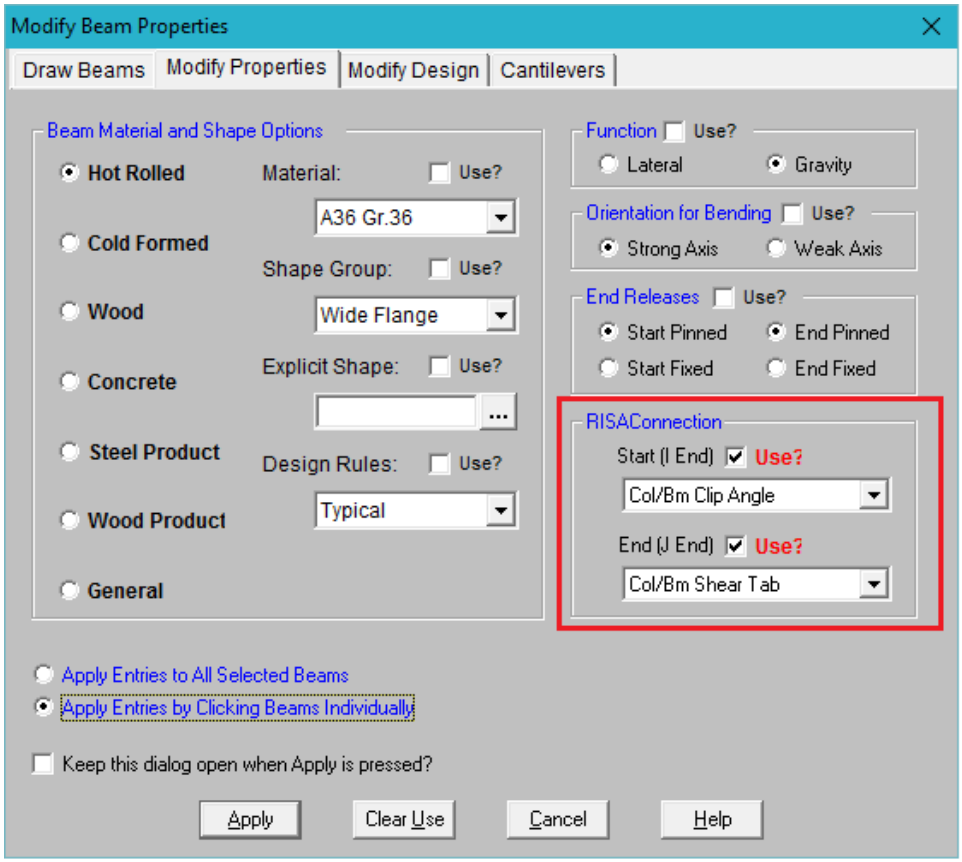
3. Assigning Connection Rules

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

- 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.
- Column Splice connections are applied as the Splice Connection Type in the Columns spreadsheet.
    - Baseplate connections (as the Column Splice Connection Type) may only be applied at the bottom-most floor level.

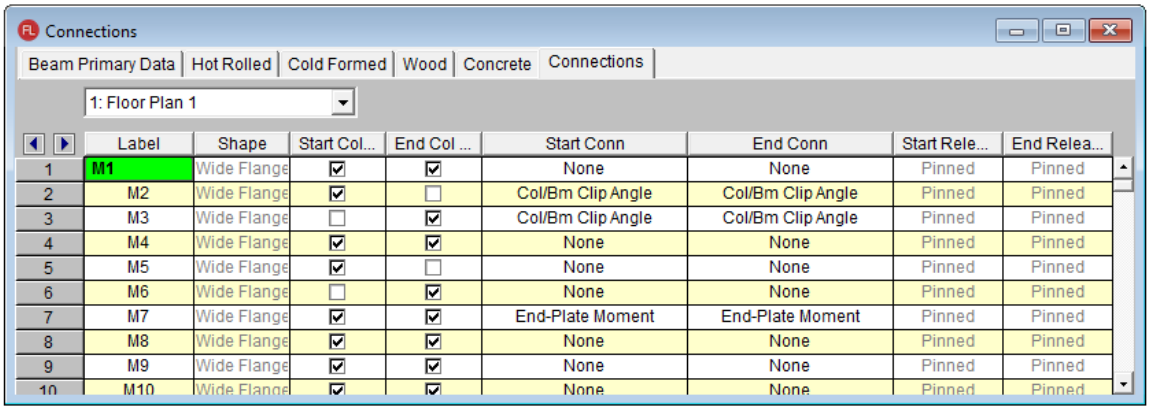
Assigning the Connection Rules Graphically:

- To assign **Connection Rules** graphically click on the Drawing toolbar and click on the **Modify Properties** tab.
- Select the proper Connection Rule for both the **Start** and **End** of the member and check the **Use** check boxes. You can then either apply the entries to all selected members or by clicking members individually.



Assigning the Connection Rules in the Beams Spreadsheet:

The **Connections** tab of the **Beams** spreadsheet provides a place to assign, edit and view Connection Rules and column eccentricity settings as they apply to individual members.

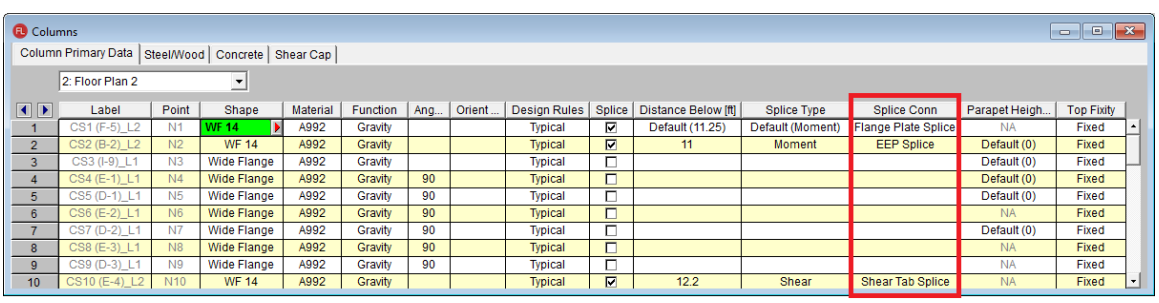


	Label	Shape	Start Col...	End Col ...	Start Conn	End Conn	Start Rele...	End Relea...
1	M1	Wide Flange	☑	☑	None	None	Pinned	Pinned
2	M2	Wide Flange	☑	☐	Col/Bm Clip Angle	Col/Bm Clip Angle	Pinned	Pinned
3	M3	Wide Flange	☐	☑	Col/Bm Clip Angle	Col/Bm Clip Angle	Pinned	Pinned
4	M4	Wide Flange	☑	☑	None	None	Pinned	Pinned
5	M5	Wide Flange	☑	☐	None	None	Pinned	Pinned
6	M6	Wide Flange	☐	☑	None	None	Pinned	Pinned
7	M7	Wide Flange	☑	☑	End-Plate Moment	End-Plate Moment	Pinned	Pinned
8	M8	Wide Flange	☑	☑	None	None	Pinned	Pinned
9	M9	Wide Flange	☑	☑	None	None	Pinned	Pinned
10	M10	Wide Flange	☑	☑	None	None	Pinned	Pinned

- **Label** - These are the member labels for all Hot Rolled Steel members in the floor.
- **Shape** - This allows you to view what shape is assigned to the member.
- **Start/End Col Ecc** - This allows you to specify whether the eccentricity between the column center-line and the beam's end reaction resultant should contribute additional moment to the design of the column. This option is only available for pinned beam ends which frame into columns.
- **Start Conn/End Conn** - These entries allow you to select a rule from the **Connection Rules** spreadsheet. You will need to know which end of the member is the Start (I end) and the End (J end), which you can view graphically.
- **Start Release/End Release** - The end release for the beam is reported here. This is useful in verifying the that the chosen connection for the beam is valid for its end fixity (i.e. shear connections for pinned-end beams).

Note:

- Connections can only be designed for Hot Rolled Steel members, so only those members are shown on this tab in the spreadsheet.
- Column splices can be applied in the **Columns** spreadsheet. The splice fixity (shear vs. moment) will be determined based on the column **Splice Type**.
    - Only base plate "splices" may be applied at the lowest level columns.

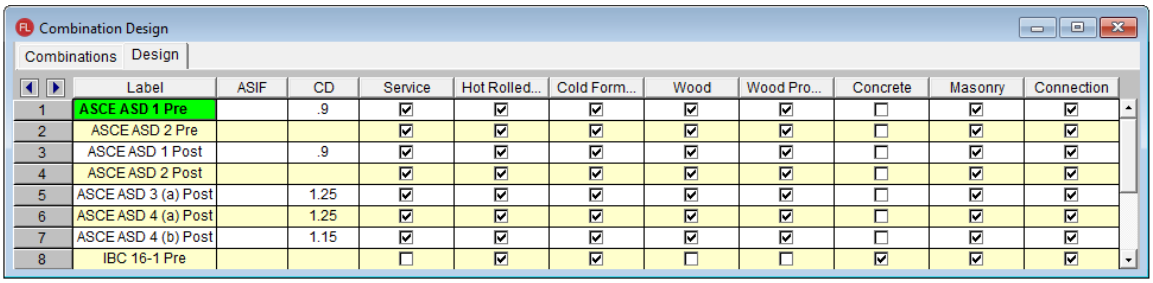


	Label	Point	Shape	Material	Function	Ang...	Orient ...	Design Rules	Splice	Distance Below [ft]	Splice Type	Splice Conn	Parapet Heigh...	Top Fixity
1	CS1 (F-5)_L2	N1	WF 14	A992	Gravity			Typical	☑	Default (11.25)	Default (Moment)	Flange Plate Splice	NA	Fixed
2	CS2 (B-2)_L2	N2	WF 14	A992	Gravity			Typical	☑	11	Moment	EEP Splice	Default (0)	Fixed
3	CS3 (I-9)_L1	N3	Wide Flange	A992	Gravity			Typical	☐				Default (0)	Fixed
4	CS4 (E-1)_L1	N4	Wide Flange	A992	Gravity	90		Typical	☐				Default (0)	Fixed
5	CS5 (D-1)_L1	N5	Wide Flange	A992	Gravity	90		Typical	☐				Default (0)	Fixed
6	CS6 (E-2)_L1	N6	Wide Flange	A992	Gravity	90		Typical	☐				NA	Fixed
7	CS7 (D-2)_L1	N7	Wide Flange	A992	Gravity	90		Typical	☐				Default (0)	Fixed
8	CS8 (E-3)_L1	N8	Wide Flange	A992	Gravity	90		Typical	☐				NA	Fixed
9	CS9 (D-3)_L1	N9	Wide Flange	A992	Gravity	90		Typical	☐				NA	Fixed
10	CS10 (E-4)_L2	N10	WF 14	A992	Gravity			Typical	☑	12.2	Shear	Shear Tab Splice	NA	Fixed

4. Assigning Load Combinations

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

On the **Design** tab of the **Load Combinations** spreadsheet there is a check box for **Connection**. This check box defines whether you want your connections designed for that LC or not.



	Label	ASIF	CD	Service	Hot Rolled...	Cold Form...	Wood	Wood Pro...	Concrete	Masonry	Connection
1	ASCE ASD 1 Pre		.9	☑	☑	☑	☑	☑	☐	☑	☑
2	ASCE ASD 2 Pre			☑	☑	☑	☑	☑	☐	☑	☑
3	ASCE ASD 1 Post		.9	☑	☑	☑	☑	☑	☐	☑	☑
4	ASCE ASD 2 Post			☑	☑	☑	☑	☑	☐	☑	☑
5	ASCE ASD 3 (a) Post		1.25	☑	☑	☑	☑	☑	☐	☑	☑
6	ASCE ASD 4 (a) Post		1.25	☑	☑	☑	☑	☑	☐	☑	☑
7	ASCE ASD 4 (b) Post		1.15	☑	☑	☑	☑	☑	☐	☑	☑
8	IBC 16-1 Pre			☐	☑	☑	☐	☐	☑	☑	☑

There are times where the member design may be designed by LRFD methods and the connections designed by ASD methods. This check box allows for that flexibility.

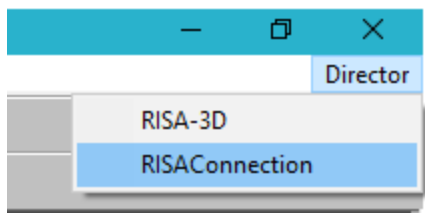
5. Designing Connections

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.

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

This option will not automatically open RISAConnection. Instead, it will run in the background, using whatever default or previous settings are in RISAConnection. Once you've done this the **Connection Results** browser will be populated. To fully export into RISAConnection, use the **Director** tool, as shown next.

2. Choose **Director - RISAConnection**.



This option will automatically launch 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 RISAFloor to populate the **Connection Results** browser.

RISAConnection File Creation and Workflow

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

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

Note that any items defined by the RISAFloor models can not be modified (beam/column sizes, bolted vs welded connections, design code, etc). Any of these changes must be taken care of in RISAFloor 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 will be available in RISAFloor. You can view the results browser or see color-coded results graphically.

Connection Results Browser

Connection Design Results

Hot Rolled Steel

1: Floor Plan 1

	Label	Member End	Connection Rule	Pass/F...	Max UC	Gov LC	Limit State
8	PUR93	J	Girder/Bm Shear Tab	Pass	.6439	11	Bolt Shear at Beam
9	PUR94	I	Girder/Bm Shear Tab	Pass	.6439	11	Bolt Shear at Beam
10	PUR94	J	Girder/Bm Shear Tab	Pass	.6439	11	Bolt Shear at Beam
11	M144	I	Girder/Bm Shear Tab	Fail	7.9316	10	Plate Flexural Rupture
12	M144	J	Girder/Bm Shear Tab	Fail	4.2949	10	Plate Flexural Rupture
13	M13	I	End-Plate Moment	Fail	2.0017	10	Bolt Shear Strength
14	M13	J	End-Plate Moment	Fail	2.0017	10	Bolt Shear Strength
15	M16	I	End-Plate Moment	Fail	1.3746	11	Bolt Shear Strength

- **Label/Member End** - These fields give the location in the model that this connection result corresponds to.
- **Connection Rule** - Shows the Connection Rule assigned to this location.
- **Pass/Fail** - Tells whether the connection passes ALL connection checks or not.
- **Max UC** - Gives the maximum unity check for the worst case LC.
- **Gov LC** - Gives which LC provided the Max UC.
- **Limit State** - Gives the governing Limit State which produced the worst case code check or failing criteria.

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

Detail Results for Current Connection

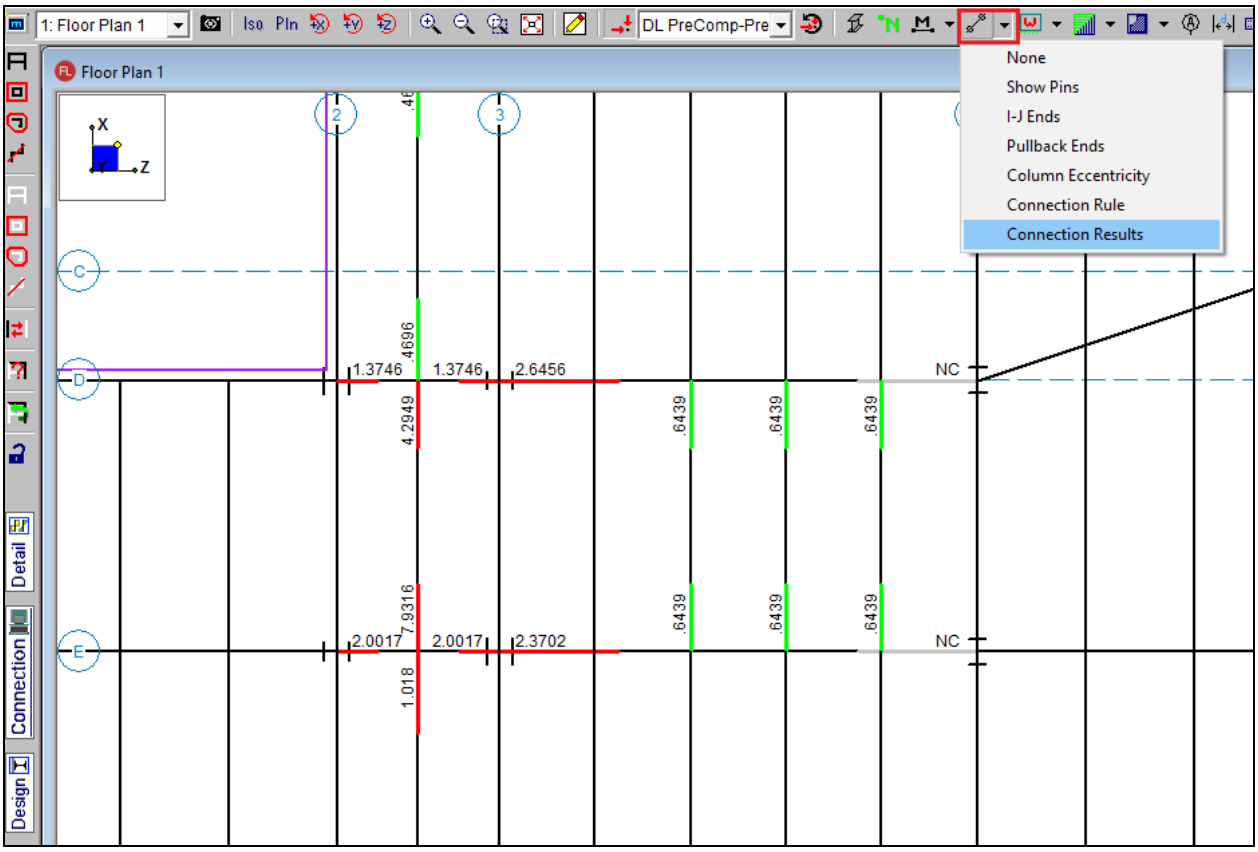
Clicking this button will open up RISAConnection to this specific connection, allowing you to take a further look at the connection results and allow you to edit connection properties in a quick, efficient manner.

Note

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

Viewing Results Graphically

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



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

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

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 will be saved. This means that if the RISAFloor/RISA-3D model is modified and the connections are re-designed you will not lose the connection information that you have modified.
    - One exception to this is if you make a change to the Connection Rule for that member end. If that occurs then the basic input of the connection is modified and your connection changes will be deleted and replaced with the new connection's default settings.
- Thus, you are able to work on a RISAConnection file separately from the RISAFloor/RISA-3D file. You can then re-locate them back in the same directory and they will be able to work together, as long as none of the

Connection Rules changed in RISAFloor/RISA-3D. Any change of an individual member end's Connection Rule will wipe out any changes made in the RISAConnection file.

Troubleshooting 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 below 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 column 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 most 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 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.

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:

- ***Connection not supported*** : This message will occur when there is a connection configuration that RISAConnection can not design. An example would be a horizontal brace connection where brace members frame into a beam web.
- ***Invalid or missing supporting connection members***: This message will occur if the member(s) that a member is framing into is not of the proper material, shape, member type, etc. 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 will occur if the member's material is not hot-rolled steel.
- ***Invalid connection member shape***: This message will occur 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 will occur if the member being connected has an invalid slope. Currently, beam/column connection design is only for orthogonal connections. If the angle between members is greater than 15 degrees from orthogonal then this message is given.
- ***Invalid member/brace rotation (more than 15 degrees)***: This message will occur if the member being connected has an invalid rotation. Currently, members can only be designed at certain orientations. For example, weak axis beam to column design is not currently supported and will instead give this message.
- ***Invalid member skew (more than 15 degrees)***: This message will occur if the member being connected has an invalid skew. Currently, beam/column connection design is only for orthogonal connections. If the angle skew between members is greater than 15 degrees from orthogonal then this message is given.
- ***Connections on skewed members are not supported***: This is a similar message to the invalid member skew message.
- ***Invalid vertical brace connection***: This message will occur if there was not a valid configuration. If there is not a valid column, beam and vertical brace intersection (at proper orientation, member type, member material, etc) then you may see this message. Note that for integration you must assign BOTH the beam and the brace to the vertical brace Connection Rule.
- ***Brace angle is invalid***: For a corner brace or chevron vertical brace connection the brace angle with the column and beam must be > 10 degrees.
- ***Members have different/missing connection rules***: If there is a vertical brace connection and the beam and brace have different Connection Rules, you will get this message.
- ***Beam must be a WF in a Chevron brace connection***: This message will come up 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 connection. Connection is applied to a gravity member***: This message will occur if a seismic connection rule is applied to a gravity member. These type of members require lateral solution results from RISA-3D, so it does not make sense to apply the connection to a gravity only member.
- ***Invalid code. HR Steel and Connection design codes must be the same***: This message will occur 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 will occur if an IMF or SMF seismic 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 will occur 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, we will transfer the connection to RISAConnection as an orthogonal connection. If the skew or slope is greater than 15 degrees than the connection will not be transferred.
- Horizontal brace connections.

RISAFloor and 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:

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

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

RISAFloor Gravity Member Connections (Scenario 1)

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

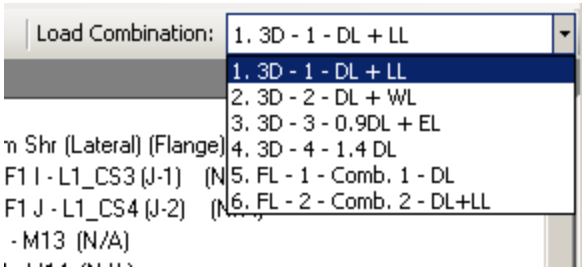
The difference is when you use the Director to go into RISA-3D and then into RISAConnection, the Gravity connections are still in the RISAConnection project. The .rcn file contains BOTH RISAFloor AND RISA-3D connections. Therefore, when entering RISAConnection from RISA-3D, you will see more connections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

RISAFloor Lateral Member Connections (Scenario 2)

Additional items to consider are:

Load Combinations

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



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

Note:

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

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

Connection Design Results

Hot Rolled Steel

2: Floor Plan 2

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

Naming Convention

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

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

Connection Design Results

Hot Rolled Steel

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

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

Customizing RISAFloor

You may customize many of the default parameters, design and analysis options in RISAFloor. 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 on the **Tools Menu** provides you control over the behavior of the software. The **Save as Defaults** feature allows you to specify the default settings for new model files. These features are discussed below. Custom reports may also be defined and saved for future use. See Printing to learn how to build a custom report.

Save as Defaults

You may use the **Save as Defaults** feature in the following dialog boxes by entering the default information in the dialog and clicking the **Save as Defaults** button: **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 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**.

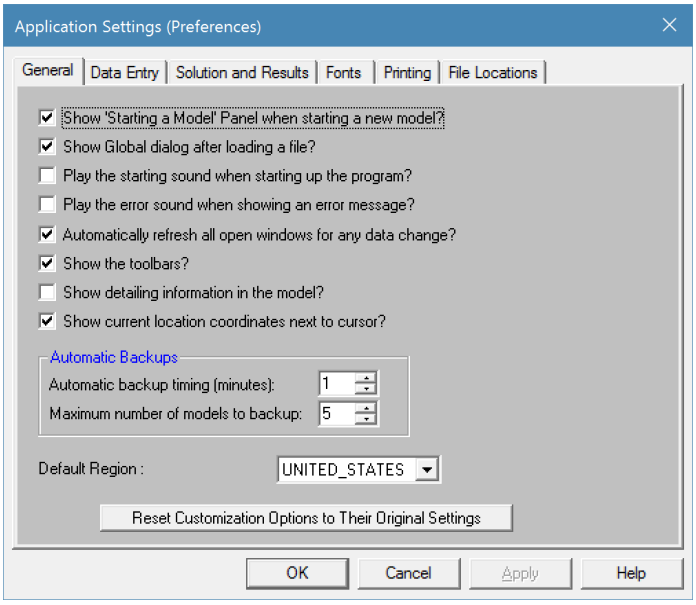
Once you create a new file you may redefine any of the default data and settings for that particular file so the **Save as Defaults** feature may be used to give you a good starting point for new files but won't hold you to those settings.

Application Settings (Preferences)

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

General Application Settings (Preferences)

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



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

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

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

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

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

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

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.

Maximum Number 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.

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

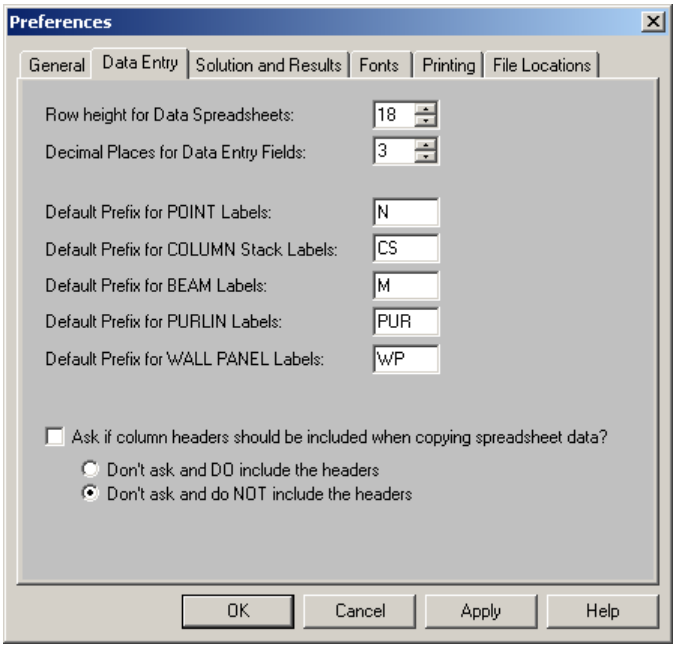
Default Region - Choose the default region for your projects.

Show detailing information in the model - Choose to display the Detailing information in the spreadsheets, draw dialogs and member information dialogs.

Data Entry Application Settings (Preferences)

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

If you wish to use a prefix with your point, beam, and wall labels, such as “P” with points, you can specify the default prefix. These prefixes may be changed as you build your model.



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

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

Default prefix for POINT labels – Sets the default prefix to be used in point labels.

Default prefix for COLUMN labels - Sets the default prefix to be used in column member labels.

Default prefix for BEAM labels – Sets the default prefix to be used in beam labels.

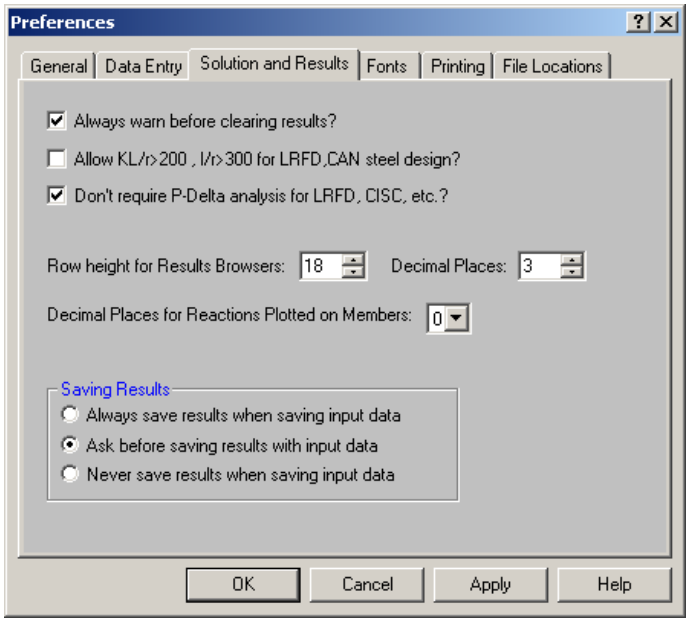
Default prefix for PURLIN labels - Sets the default prefix to be used in purlin or 'in-fill beam' labels.

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

Ask if column headers should be included when copying... - If this checkbox is checked then the program will ask if you want to copy the spreadsheet header any time you attempt a copy command in a spreadsheet. If this checkbox is not checked then you have the option of either always copying the spreadsheet headers, or never copying them.

Solution and Results Application Settings (Preferences)

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



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

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

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**.

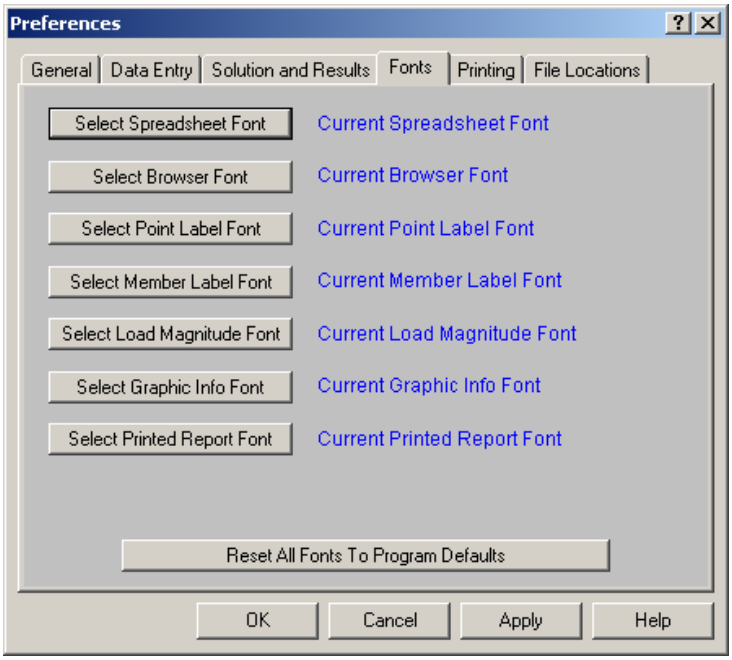
Row height for Results Browsers – Sets the row height and font size for results spreadsheets.

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

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

Font Application Settings (Preferences)

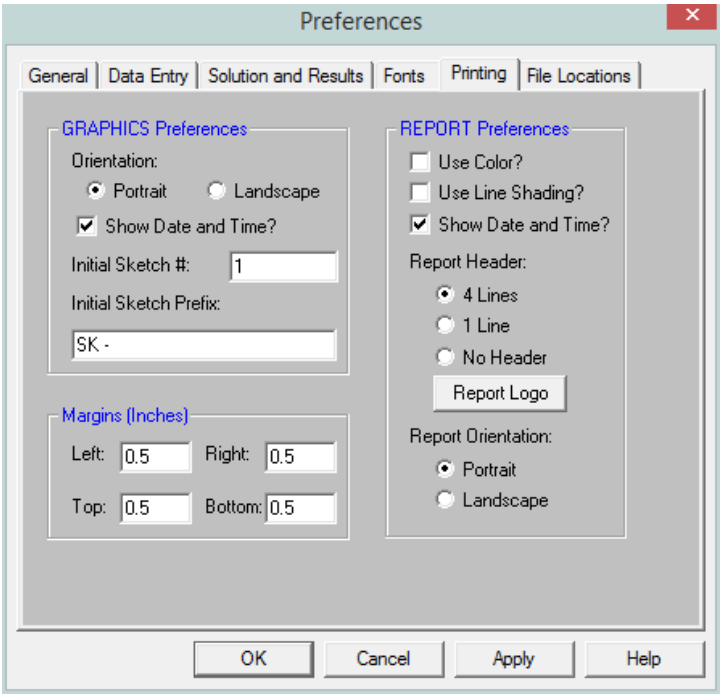
The font Application Settings can be used to adjust the fonts used by the spreadsheets, results browsers, and graphics. The font changes will affect both the on-screen displayed data and the printed data. The exceptions to this are the spreadsheet and browser fonts which may be changed for on-screen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



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

Printing Application Settings (Preferences)

This tab gives many options for printing output. See Printing for additional information.



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

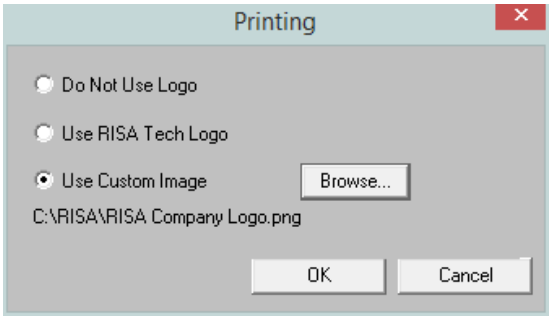
Sketch Information - Sets sketch numbers and prefixes for graphic printing.

Margins – Sets the default printing margins.

Report Application Settings – These allow you to define information about colors, date and time, headers, report logo and overall report orientation.

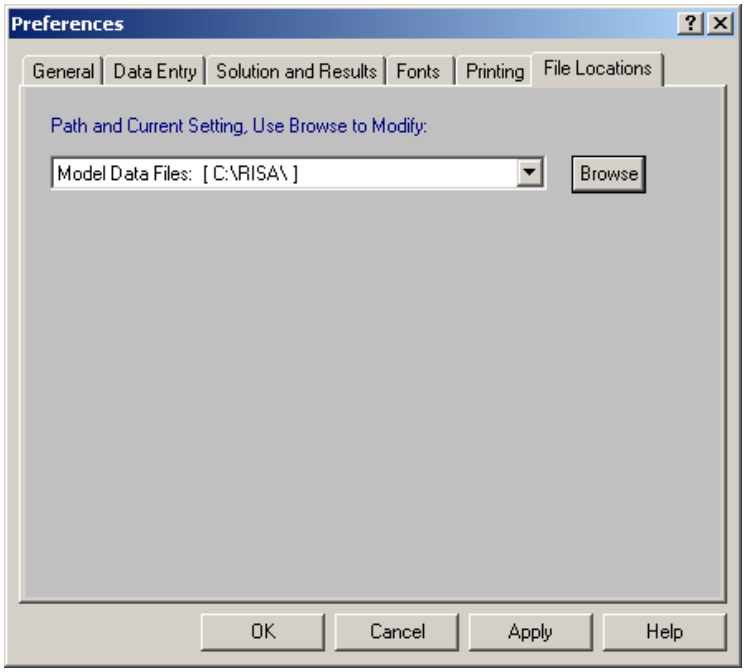
Use Line Shading? - Defines whether the spreadsheet output shades every other line. This can give contrast to make the information easier to read.

Report Logo - Allows you to add either a RISA logo or any company logo to your printed output.



File Application Settings (Preferences)

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



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

Decks and Diaphragms

Decks may be defined for load attribution as well as composite beam design. Unbraced lengths for the top flanges of beams may also be controlled through the deck properties. Loads may be associated with the deck, or defined with the applied loads. Allowable span lengths can also be specified to avoid violating maximum span capabilities of the deck. Deck direction also controls the span of wood structural panels for flexible diaphragms.

Note

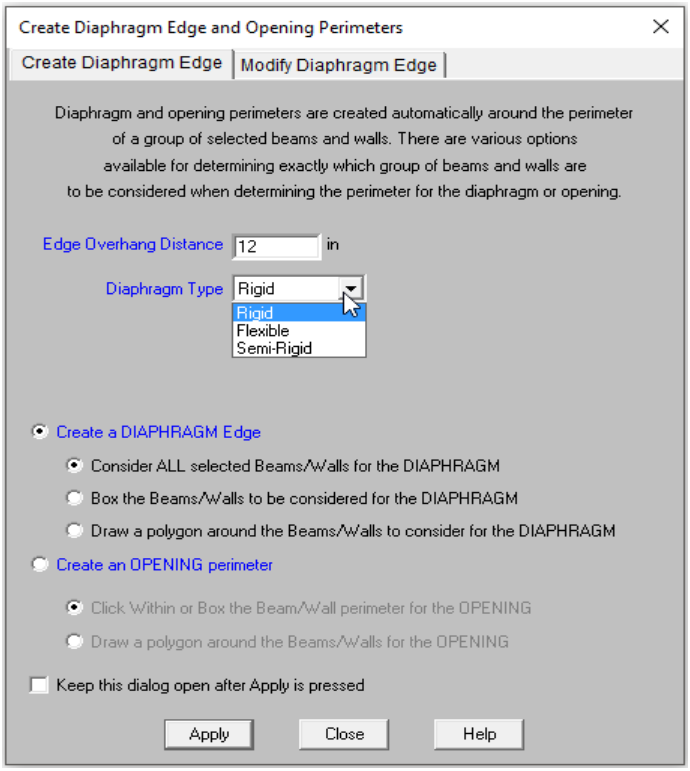
- The term "deck" refers to different types of floor construction including wood, metal, concrete and composite type construction. RISAFloor uses the term "deck" when discussing properties in an area.
- Diaphragms may be defined as rigid, semi-rigid or flexible, and as such they determine the diaphragm behavior within themselves.

Diaphragm edges and openings must be defined to establish the extents of the deck. Properties for the deck are defined in the **Deck General Properties Spreadsheet.** A default deck is listed for each floor on the **Floors Spreadsheet.** However, local deck systems may be defined where deviation from the default deck system occurs.

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

Creating Diaphragm Edges and Openings

To create diaphragm edges or openings click on the **Drawing Toolbar** to open the Create Diaphragm Edge and Opening Perimeters dialog. Enter the edge overhang distance and specify whether it is a diaphragm edge or opening. For diaphragm edges, select whether you want the area to be treated as a rigid or flexible diaphragm. For Diaphragm perimeter, select whether you want ALL selected Member/Walls to be considered for the diaphragm perimeter or do you want to box or polygon the diaphragm perimeter, click **Apply**. For an opening, select whether you want to click within or box the framed opening or draw a polygon around the opening, click **Apply**. The new diaphragm edge or opening will be shown on screen.



To Create Diaphragm Edges or Openings

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. If you do not already have beams or walls modeled to support the diaphragm edge you will need to create them first.
3. Click the button and set the edge distance.
4. Select the type of Diaphragm: Rigid, Flexible or Semi-Rigid. See Diaphragms for more information.
5. Click **Apply.** For diaphragm edges:

- If Consider ALL selected members was chosen then you should already see the diaphragm perimeter.
- If box or polygon was selected then start drawing by windowing a diaphragm perimeter or by clicking with the left mouse button outlining the perimeter.

For Diaphragm Openings:

- Start drawing by clicking within or windowing a framed opening or by clicking with the left mouse button outlining the opening.
- To stop drawing altogether click the right mouse button, or press the Esc key.

Note

- For diaphragm openings the minimum edge overhang distance is 2-inches.
- Press CTRL-D to quickly recall any of the dialogs and make changes.
- The coordinates of the closest grid or point to your cursor are displayed in the lower right hand corner.
- You may undo any mistakes by clicking the Undo button.

Modifying Diaphragm Edges and Openings

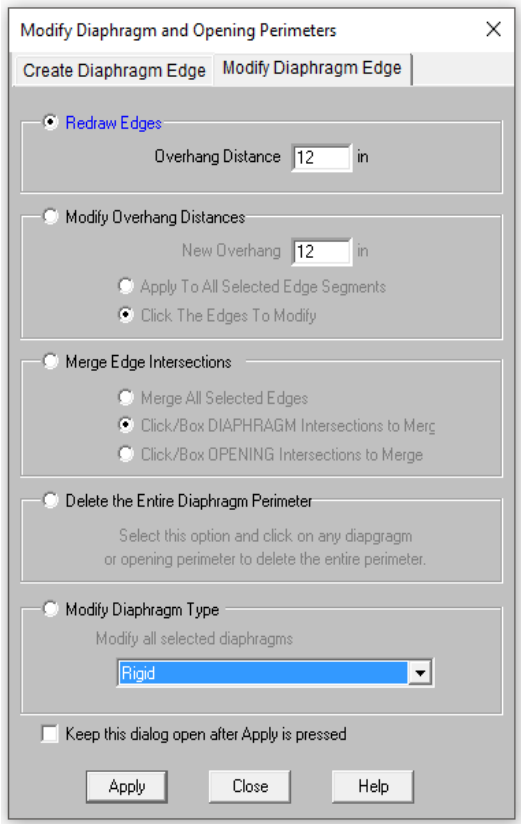
The graphical Modify Edge tool lets you modify the diaphragm edges or openings that already exist in your model. You can redraw diaphragm edges, modify the overhang distance, modify edge intersections, delete an entire diaphragm edge or opening, or change the diaphragm type. You can modify diaphragm edges or openings one at a time by selecting the **Click The Edges** or **Click Intersecting Points** option and then click on the diaphragm edges or points you wish to modify. You may also modify multiple edges by selecting the diaphragm edges first and then using the **Apply To ALL** or **Merge ALL** option. See Graphic Selection for more on selecting.

To Modify diaphragm Edge and Openings

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Click the button and select the option you wish to use.
3. To redraw an edge: Enter the **Overhang Distance** and click **Apply**. Select and redraw the edge you wish to modify.
4. To modify overhang distances: Enter the **Overhang Distance** and whether you want to **Apply To All Selected** or **Click the Edges to Modify**. Click **Apply**.
5. To merge edges: You can choose **Apply To All Selected** to merge all the selected edges or **Click the Edges to Modify** to merge edges one by one. Click **Apply**.
6. To delete an entire perimeter: Select this option and click **Apply.** You then click on the edge or opening perimeter to be deleted.

Note

- It may be necessary to merge edges prior to redrawing them.
- You may undo any mistakes by clicking the Undo button.



To Modify Diaphragm Type

1. Click on the Modify Diaphragm Type radio button
2. Select whether you want to convert the diaphragm in question into a Rigid, Semi-Rigid or Flexible diaphragm
3. Click Apply. The modification will be applied to all selected edges

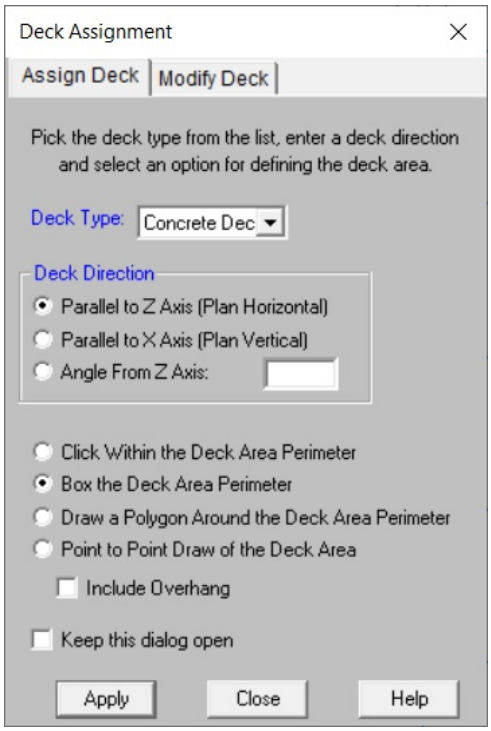
Assigning Deck Properties

The default diaphragm and deck properties and the deck angle are defined in the Floor Spreadsheet. If you have areas of your floor that are oriented differently or have different deck properties, you will have to assign new local deck properties to these areas.

To assign deck properties click on the **Drawing Toolbar** to open the Deck Assignment dialog. Select a **Deck Type** from the pull down list. See Deck Properties - General for more information on Decks or to add or remove decks from the list.

Specify a **Deck Direction** by choosing Parallel to Horizontal, Parallel to Vertical or entering an angle that the span direction makes with the horizontal axis.

You may apply the deck properties by clicking within or boxing a deck area perimeter, by drawing a polygon around the deck area perimeter, or by drawing point to point.



To Assign Deck Properties

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. If you do not already have beams or walls supporting the area you will need to create them first.
3. Click the button and set the deck properties.
4. Select how you would like to apply the local deck and click **Apply.** Start drawing the deck by clicking within or boxing the area, drawing a polygon around the area perimeter, or clicking on grid or existing point locations with the left mouse button.
5. The include Overhang box allows you to include the deck's overhang when applying a deck.
6. To stop drawing altogether click the right mouse button, or press the Esc key.

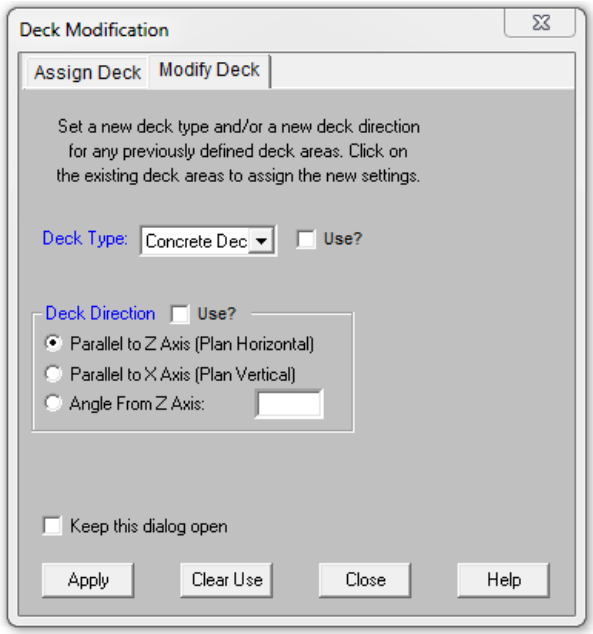
Note

- Except on diaphragm edges, the local decks must be supported on all sides by beams or walls.
- The default deck properties and the deck angle are defined in the Floor Spreadsheet. If these default properties are sufficient, you do not need to draw in local deck assignments.
- The coordinates of the closest grid or point to your cursor are displayed in the lower right hand corner.
- Press CTRL-D to quickly recall the dialog and make changes.
- You may undo any mistakes by clicking the Undo button.
- The listed decks are presented in the same order in which they appear in the Deck Definitions spreadsheet.

Modifying Deck Properties

For existing decks that have been previously defined, you can use the Modify Decks dialog to make changes.

To modify deck properties click on the **Drawing Toolbar** and click to the Modify Deck tab. Make your changes and make sure to check the **Use?** checkbox, as only selections with **Use?** checked will actually change in the model.

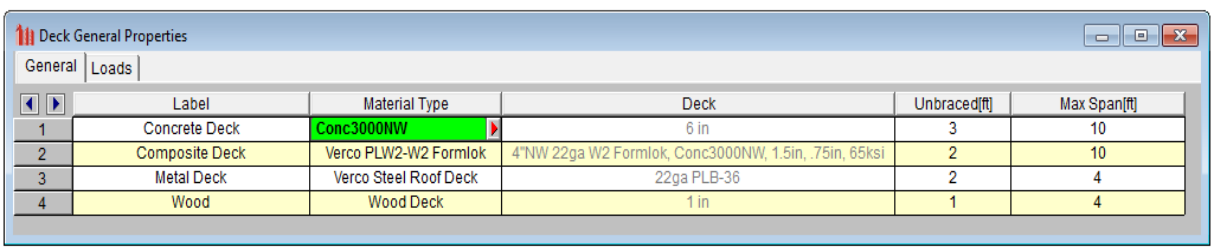


Note

- This dialog will NOT make changes to a default deck. To make changes to the default deck see the Floors spreadsheet.

Deck Definitions - General

The **Deck Properties Spreadsheet** records the deck properties and may be accessed by selecting **Deck Properties** on the **Spreadsheets** menu. The spreadsheet has two tabs: General, and Loads.



	Label	Material Type	Deck	Unbraced[ft]	Max Span[ft]
1	Concrete Deck	Conc3000NW	6 in	3	10
2	Composite Deck	Verco PLW2-W2 Formlok	4"NW 22ga W2 Formlok, Conc3000NW, 1.5in, .75in, 65ksi	2	10
3	Metal Deck	Verco Steel Roof Deck	22ga PLB-36	2	4
4	Wood	Wood Deck	1 in	1	4

The **General** tab holds the general data for the decks. The Loads tab contains load information for the deck.

Define a new Deck

1. On the **Spreadsheets** menu click on **Deck Properties** to open the spreadsheet. There are four default decks available.
2. Place the mouse in the last row of the spreadsheet and press Enter to create a new deck.
3. Enter the deck label. Choose the Material Type by clicking on the arrow in the cells. Enter the material information and press OK to return to the Deck Properties spreadsheet.
4. Enter the Unbraced Length and Max Span.
5. Click on the Loads tab to enter all the deck loading.

The entries for the **General** tab are explained below:

Label

A unique label may be assigned to all of the decks manually or automatically. You can then refer to the deck by its label. You may relabel the deck at any time without affecting the application of the properties to the model.

Material Type

There are four material types available; Concrete Deck, Composite Deck, Metal Deck and Wood Deck. You can select the deck by clicking on the arrow . See Material Type for more information.

Deck

This shows the selection from the Material Type dialogue. This is for display only, you cannot edit this directly. To change this information, you must go back to the Material Type dialogue.

Unbraced

The unbraced length that the deck provides to the top flange of the beams may be entered here. If left blank the deck is assumed not to brace the beam top flange and the unbraced length is determined from the spacing of the framing, the full length of the beam, and any values specified in the Beams spreadsheet. See Unbraced Lengths for more on how the unbraced length is determined.

Max Span

Enter the maximum safe span allowed for the deck. If the span is exceeded, the location will be reported in the warning log.

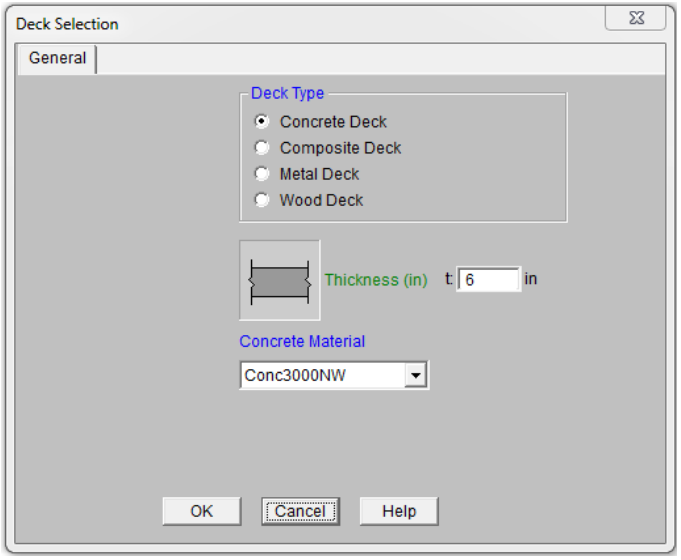
Deck Properties - Material Type

Concrete Deck

You enter the thickness and select the concrete material. The thickness is defined as the full thickness of a flat slab of concrete above the beams. The concrete material is a list from the Materials spreadsheet.

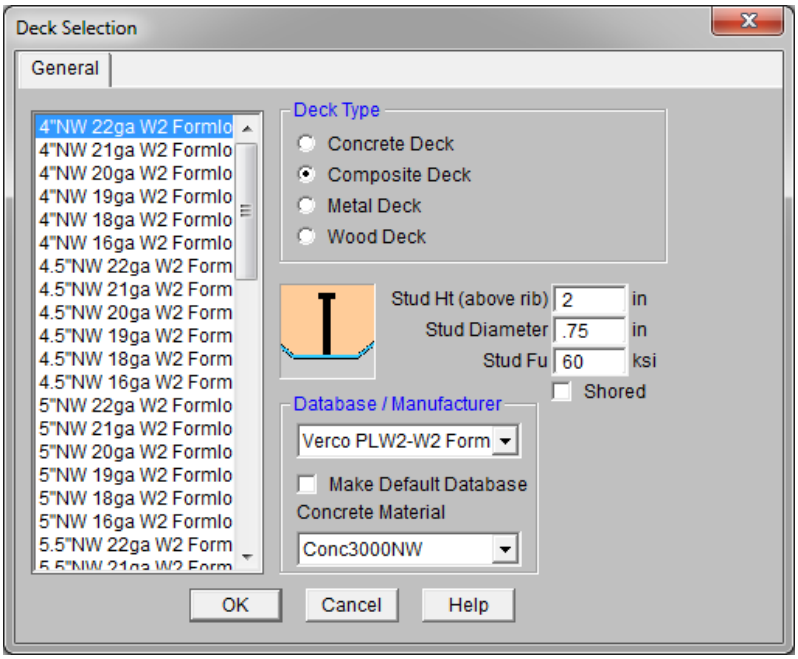
Note:

- To design a Composite Flat Slab use the Composite Deck option, and select the Concrete Deck database.



Composite Deck

The decks are organized by **Database/Manufacturer** from the drop-down list, then you can select the deck.



Composite Deck Database Files

RISAFloor uses the deck properties for Composite Decks based on a database XML file in the "RISA_Decks" sub-directory of the RISA folder. The location of this directory is based on the information of the Tools- Application Settings dialog. The program comes pre-loaded with XML files, each of which contains the required information for the composite beam design. The name of the XML file itself will be used in the list of the Database/Manufacturer.

The Self Wt is from the XML database is used for the deck self weight, and you can confirm this in the Deck Loads tab of the Deck Definitions dialogue.

	A	B	C	D	E	F	G	H	I
1	Label	Composite	Self Wt	Conc Assumed	Slab Thickness	Rib Ht	Rib Top Width	Rib Bot Width	Pitch
2			ksf	kcf	in	in	in	in	in
3	4"NW 22ga W2 Formlok	Yes	0.0363	0.1450	4.000	2	7	5	12
4	4"NW 21ga W2 Formlok	Yes	0.0363	0.1450	4.000	2	7	5	12
5	4"NW 20ga W2 Formlok	Yes	0.0363	0.1450	4.000	2	7	5	12
6	4"NW 19ga W2 Formlok	Yes	0.0363	0.1450	4.000	2	7	5	12
7	4"NW 18ga W2 Formlok	Yes	0.0363	0.1450	4.000	2	7	5	12
8	4"NW 16ga W2 Formlok	Yes	0.0363	0.1450	4.000	2	7	5	12

Note:

- For the Vulcraft composite decks (1.5 VLR, 1.5VL-VLI, 2VLI and 3VLI) the **Self Wt** of the deck is taken directly from the deck manufacturer tables. These tables assume 18 gauge deck for all cases when calculating the self-weight, which is conservative in most instances.
- For the Verco composite decks (PLW2-W2, PLW3-W3, PLB-B and PLN-N) the **Self Wt** of the deck is provided which takes the weight due to concrete plus the weight due to the deck. Thus, an exact value of the deck weight is used, different from how the Vulcraft tables were formulated.

Concrete Material

After selecting the deck, the corresponding Concrete Material must be selected. This list is the from the Materials spreadsheet. The weight of the concrete material will be checked to match the database value. The modulus of elasticity will be used to design composite beams, including the determination of the stud capacities.

Rib Height

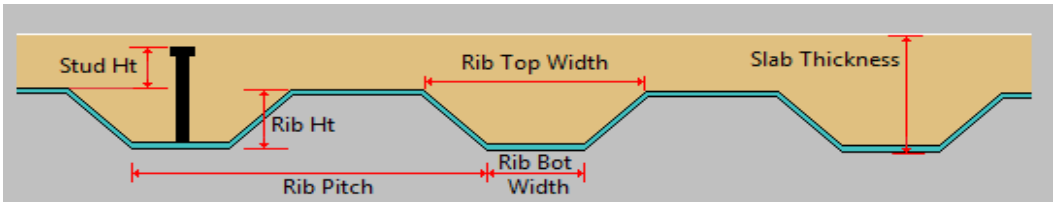
Rib Height defines the total height of the deck ribs. Enter 0 for a flat slab.

Rib Widths

Rib Widths define the top and bottom widths of the rib. If the Rib Height is set to 0 the Rib Width values are ignored.

Rib Pitch

Rib Pitch defines the spacing of the ribs. If the Rib Height is set to 0 the Rib Pitch value is ignored.



Slab Thickness

For the thicknesses given in the composite deck database, the thickness is defined as the total slab depth.

Stud Height

Stud Height defines the height of the shear stud **above the rib** to be used for composite beam design. There are a number of checks regarding stud height defined in the AISC.

Note

- The slab thickness and stud height definitions changed in Version 6.0. Prior to this version, the slab thickness was measured from top of rib to the top of the slab and the stud height was measured from the bottom of the rib to the top of the stud.

Stud Diameter

Stud Diameter defines the diameter of the shear stud.

Stud Fu

Stud Fu defines the Ultimate Tensile Strength of the shear stud.

Shored

When the Shored box is unchecked, composite beams will be checked for pre-composite/construction loads using their bare beam section properties, and all other loads using their composite properties.

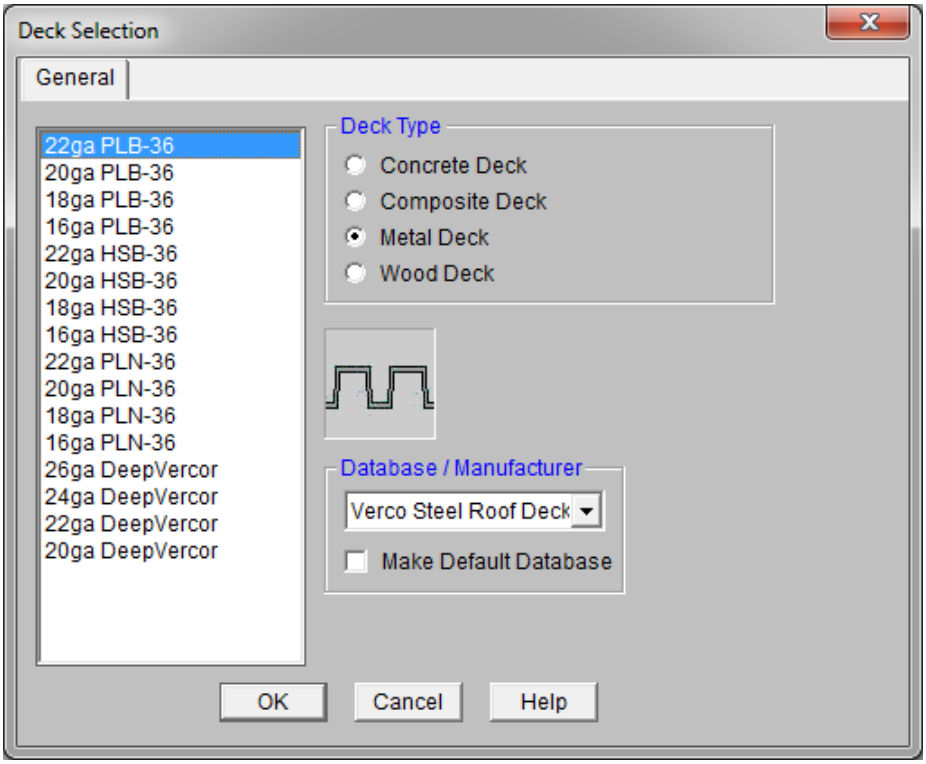
When the Shored box is checked, composite beams will be checked for all loads using their composite properties.

Note:

- Beams that are shored will not be cambered.

Metal Deck

The decks are organized by **Database/Manufacturer** from the drop-down list, then you can select the deck to be used from the display window on the left side of the dialogue.



Metal Deck Database Files

RISAFloor uses the deck properties for Metal Decks based on a database XML file in the "RISA_Decks" sub-directory of the RISA folder. The location of this directory is based on the information of the Tools- Application Settings dialog. The program comes pre-loaded with XML files, each of which contains the required information for the composite beam design. The name of the XML file itself will be used in the list of the Database/Manufacturer.

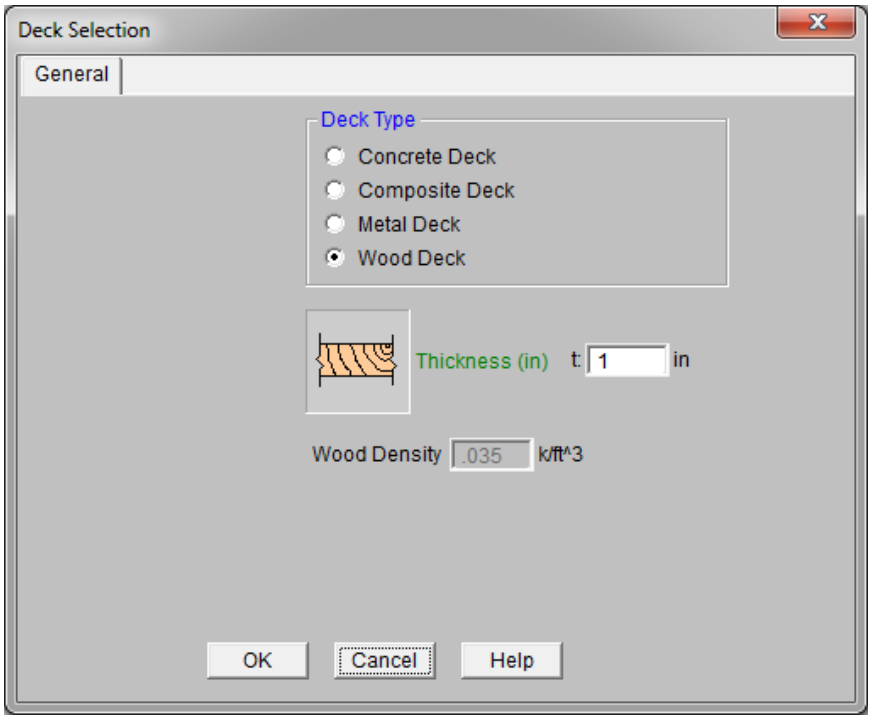
Note:

- Currently only the Self Weight is used in the RISAFloor model.

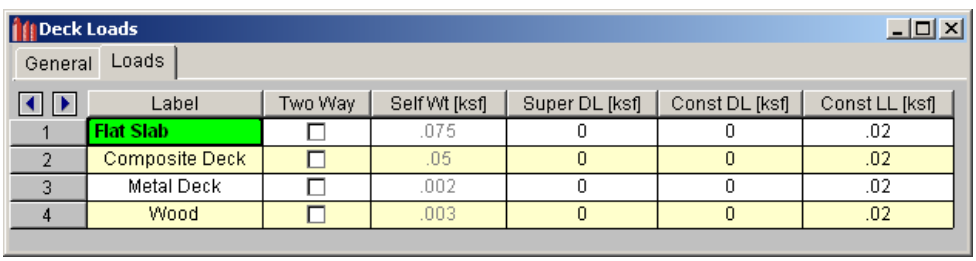
	A	B	C	D	E	F	G	H
1	Label	Composite	Self Wt	Rib Ht	Rib Top Width	Rib Bot Width	Shear Capacity	Ga
2			ksf	in	in	in		
3	1.5 B24	No	0.0015	1.500	3.5	1.75	0	0.00
4	1.5 B22	No	0.0018	1.500	3.5	1.75	0	0.00
5	1.5 B20	No	0.0021	1.500	3.5	1.75	0	0.00
6	1.5 B19	No	0.0025	1.500	3.5	1.75	0	0.00
7	1.5 B18	No	0.0028	1.500	3.5	1.75	0	0.00

Wood Deck

You enter the thickness of the wood deck. The self weight will be calculated based on the shown wood density. The density shown is not editable and is based on the first wood entry in the Materials spreadsheet under the Wood tab. You can change the density in the Material spreadsheet, or create a new wood material to change this value.



Deck Definitions - Loads



	Label	Two Way	Self Wt [ksf]	Super DL [ksf]	Const DL [ksf]	Const LL [ksf]
1	Flat Slab	☐	.075	0	0	.02
2	Composite Deck	☐	.05	0	0	.02
3	Metal Deck	☐	.002	0	0	.02
4	Wood	☐	.003	0	0	.02

The Loads tab holds all the load data for the decks. See Load Types - Deck Loads or Loads - Vertical for more information on how loads are combined for solution.

Two Way

The decks can be specified as one-way or two-way to control the attribution of the applied loads. Check this box if you want the area loads applied to the deck to be distributed based on two-way action. See Loads – Attribution for more information.

Self Weight

Self Weight is the weight of the deck system. This weight will be used as part of the pre-composite dead load (PreDL) and also as part of the post-composite dead load (DL) when creating load combinations for solution.

Note:

- The self-weight and Super DL entries here are used in the camber design, depending on what loading option you selected in your Camber Design Rules.

Superimposed Dead Load

The Superimposed Dead Load (Super DL) is additional load that will be treated as additional deck self weight. This might be useful to account for deck topping or flooring materials that will be part of the dead load of the floor system. This load will be included for any calculations that include deck self weight (seismic mass, beam camber for hot-rolled steel and any places the refer to load categories DLPre and DL).

Construction Dead Load

You may define construction dead load that occurs during construction but is not part of the final load on the system. This weight will be used as part of the pre-composite dead load (PreDL) but not included as part of the post-composite dead load (DL) in the load combinations for solution.

Construction Live Load

You may define construction live load that occurs during construction but is not part of the final load on the system. This weight will be used as part of the pre-composite live load (LLConst) but not included as part of the post-composite live load (LL) in the load combinations for solution.

Diaphragm Edge and Opening Perimeters

Diaphragm edge perimeters **must** be specified to define the outer most extents of the floor deck or diaphragm if automatic load attribution or composite beam design is desired. The edge defines both the area available for loading and the available effective width for composite beams. While load and deck areas may be defined beyond the diaphragm edge, the portions outside the edge are ignored at solution time.

Diaphragm edges perimeters and openings can be generated automatically. See Creating Diaphragm Edges and Openings to learn how to draw and modify diaphragm edges and openings.

Diaphragm edges along with diaphragm openings may also be used to define rigid diaphragms for lateral analysis in RISA-3D. See RISAFloor & RISA-3D Interaction for more information.

Member Design Optimization

RISAFloor will optimize Hot Rolled Steel, Cold Formed Steel, Wood, and Concrete members, as well as Steel and Wood Products. The criteria used for this optimization are the selected design code and the **Design Rules** assigned to the beam or slab design strip/cut. 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 or later as a modification. See Modifying Beam 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 may be accessed by selecting **Design Rules** on the **Spreadsheets Menu**. You may create and name any number of design rules and assign different rules to various members.

Design Size/U.C. Parameters

Size/UC | Deflection (Beams) | HR Camber | Concrete Rebar

	Label	Max Depth[in]	Min Depth[in]	Max Width[in]	Min Width[in]	Max Bending Chk	Max Shear Chk
1	Joists	12				1	1
2	Exterior Columns					.9	1
3	Interior Columns			16	16	.85	1
4	Girders	24	12			1	1
5	Floor Diaphragms					1	1
6	Roof Diaphragms					1	1

The spreadsheet has four tabs: **Size/UC**, **Deflection**, **Camber** and **Concrete Rebar**. The entries for the **Size / UC** tab are explained below:

Design Rule Labels

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

Max/Min Depth

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

Max/Min Width

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

Max Bending and Shear Check

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

Member Design Rules – Deflection

Design Deflection Parameters

Size/UC | Deflection (Beams) | HR Camber | Concrete Rebar

	Label	DL Def...	DL Ratio	LL Defl...	LL Ratio	DL+LL...	DL+LL...	Categ...	Defl[in]	Ratio	Categ...	Defl[in]	Ratio
1	Joists		240		360		240	None	.3	360	None		360
2	Exterior Columns		240		360		360	None		360	None		360
3	Interior Columns		240		360		360	None		360	None		360
4	Girders		240		360		360	None		360	None		360
5	Floor Diaphragms		240		360		360	None		360	None		360
6	Roof Diaphragms		240		360		360	None		360	None		360

The entries for the **Deflection** tab are explained below.

DL Deflection and Ratio

Enter the maximum allowable dead load deflection and/or the minimum dead load deflection to beam length ratio.

If no redesign rules are assigned to a beam, then a default ratio of L/240 will be used for all shape groups except Steel Products and Wood Products which do not have a maximum DL Deflection Ratio.

LL Deflection and Ratio

Enter the maximum live load deflection and/or the minimum live load deflection to beam length ratio.

If no redesign rules are assigned to a beam, then a default ratio of L/360 will be used for all shape groups except Wood Products which have a default of L/480.

Note

- By default construction live load (LLConst) is not included in the deflection ratio checks. If needed this can be added in as a category specific deflection ratio check.

DL+LL Deflection and Ratio

Enter the maximum dead + live load deflection and/or the minimum dead + live load deflection ratio.

If no redesign rules are assigned to a beam, then no constraints will be used except Wood Products which have a default of L/240.

Category Specific Deflection and Ratio

The remaining columns allow you to set deflection criteria for other load categories. Enter the category, the maximum deflection and/or the minimum deflection ratio.

Cantilever Deflection Ratios

For a cantilever the program uses 2*Cantilever Length in determining the L/ratio. For a 10' cantilever that is deflecting 0.842" the ratio will be **2***10*12/0.842" = 285 (L/285).

Member Design Rules – Camber

Design Camber Parameters

Size/UC | Deflection (Beams) | HR Camber | Concrete Rebar

	Label	Min Camber[in]	Max Camber[in]	Increment[in]	% DL	DL Included	Min Beam Length[ft]
1	Joists	.75	3	.25	75	SW Only	30
2	Exterior Columns	.75	3	.25	75	Total DL	30
3	Interior Columns	.75	3	.25	75	Total DL	30
4	Girders	.75	3	.25	75	Total DL	30
5	Floor Diaphragms	.75	3	.25	75	SW + SuperDL	30
6	Roof Diaphragms	.75	3	.25	75	SW + SuperDL	30

The entries for the **Camber** tab are explained below.

Minimum and Maximum Camber

Enter the minimum and maximum camber values for the design.

Increment

Enter the cambering increment value.

% DL

Percentage of the selected load (see "DL Included") below to be considered for the camber design calculations.

DL Included

Select the loading to be considered for the camber design calculation. Options include:

- **SW + SuperDL** - Both self weight loading (of the member and the deck) and any applied load from the "Superimposed DL" load cases (from the Deck Definitions spreadsheet) will be included for the camber calculation.
- **Total DL**- All dead loads (self weight of member and deck, Superimposed DL, PreDL, and PostDL) will be included for the camber calculation.

Min. Beam Length

Enter the minimum beam length for camber design. Only the members that are equal to or exceed this length will be designed for camber.

Member Design Rules – Concrete Rebar

Parameters for Concrete Rebar

Size/UC | Deflection (Beams) | Concrete Rebar

	Label	Optimize ...	Min Flex Bar	Max Flex Bar	Shear Bar	Legs per Stirrup	Top (Colu...	Bottom Co...	Side Cove...	Top/Botto...	Add'l Side ...	Shear ...
1	Joists	Optimize	#8	#10	#4	2	1.5	1.5	1.5	2	1	12
2	Exterior Columns	Optimize	#5	#10	#4	2	1.5	1.5	1.5	2	1	12
3	Interior Columns	Optimize	#5	#10	#4	2	1.5	1.5	1.5	2	1	12
4	Girders	Optimize	#5	#10	#4	2	1.5	1.5	1.5	2	1	12
5	Floor Diaphragms	Optimize	#5	#10	#4	2	1.5	1.5	1.5	2	1	12
6	Roof Diaphragms	Optimize	#5	#10	#4	2	1.5	1.5	1.5	2	1	12

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

Note

- If you would like to define specific flexural and shear rebar layouts for beams and columns, either use the **Explict** option below or see Rebar Layout Database for Custom Rebar layout.

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 greyed 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 - Concrete. You can force the program to analyze one bar size by setting the Min and Max values to be the same bar.

Shear Ties

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

Legs per Stirrup

Use the **Legs per Stirrup** column to enter the specific information about how may 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

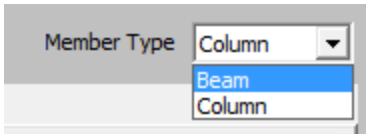
The **Member Design Rules Editor** provides a way to create or modify member design rules in a graphical way. This editor automatically updates any information input into the **Member Design Rules** spreadsheet.

Note:

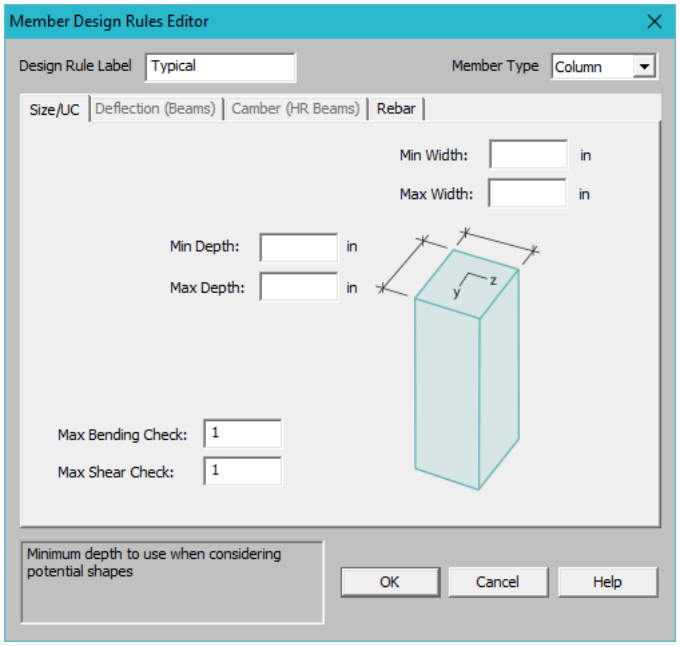
- Keep in mind that this dialog has four tabs. Make sure to fill out the information on all the tabs before clicking **OK**.
- See the information in the Status Bar for more information on each of these entries.

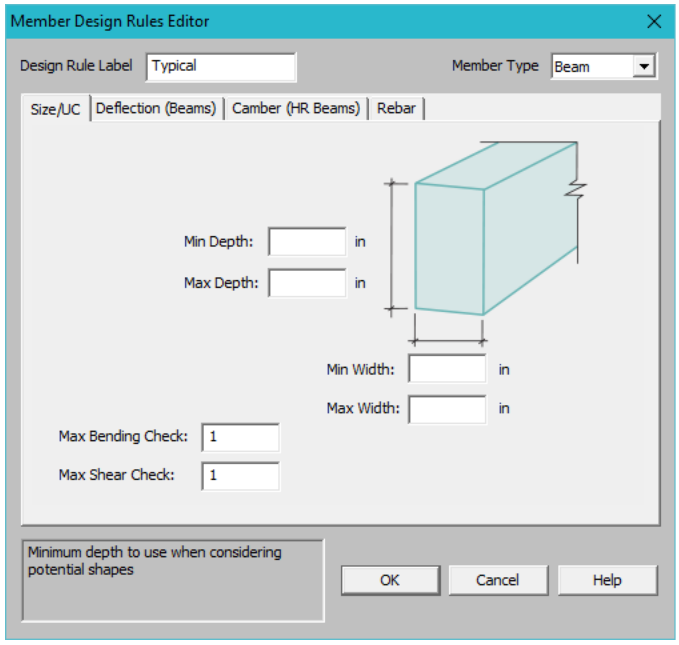
The **Design Rule Label** is editable and will change the spreadsheet label. It is useful to name the design rule with a descriptive name like "First Floor Columns".

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.



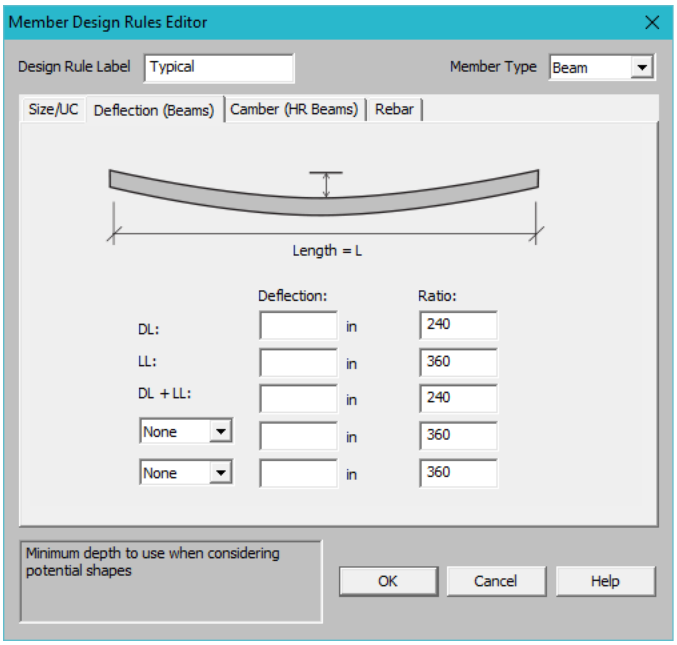
The Size/UC tab:





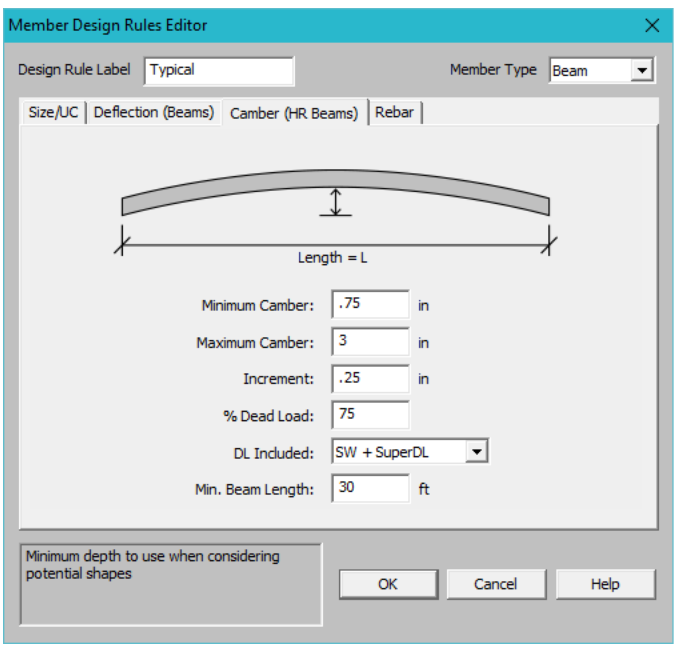
Deflection (Beams) tab:

This only applies to Beam members.



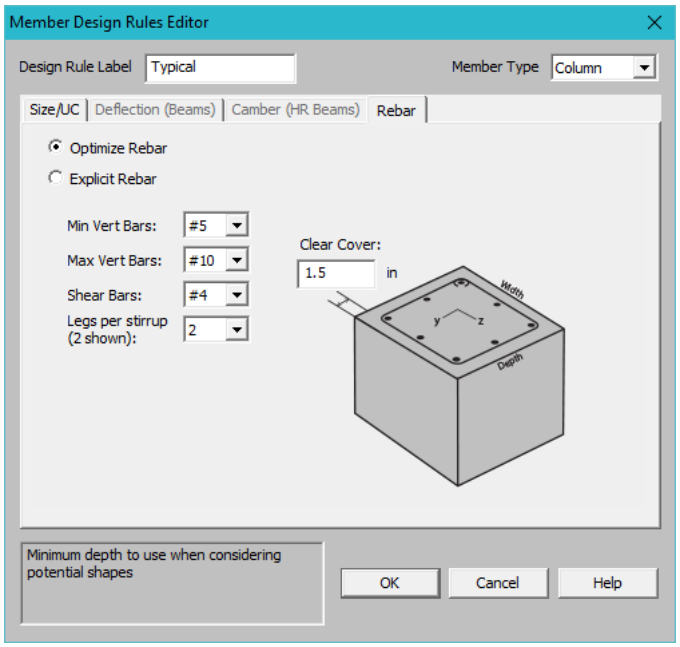
Camber (HR Beams) tab:

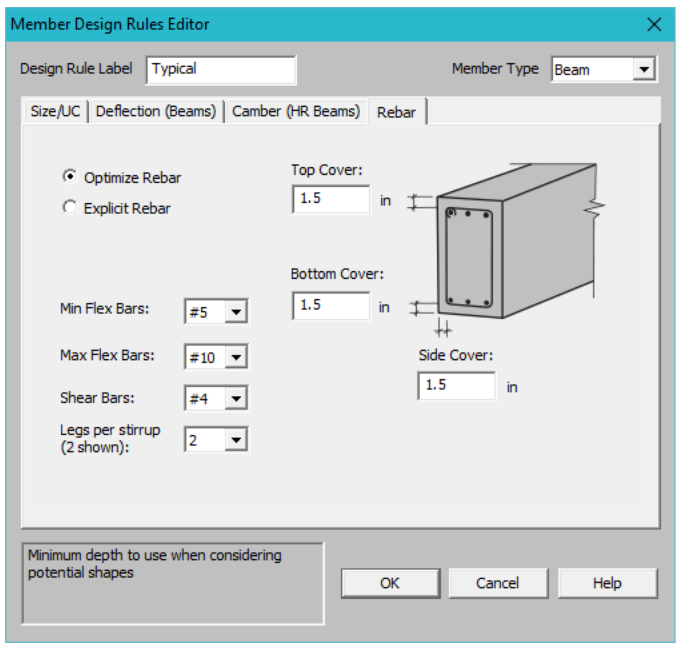
This only applies to Beam members.



Rebar tab:

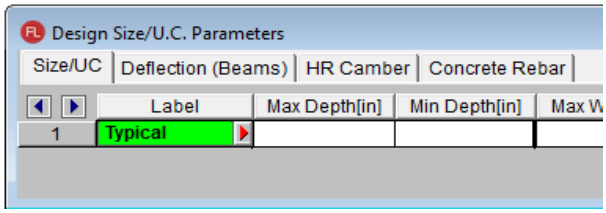
You can select **Optimize** or **Explict** to set the rebar design parameters.





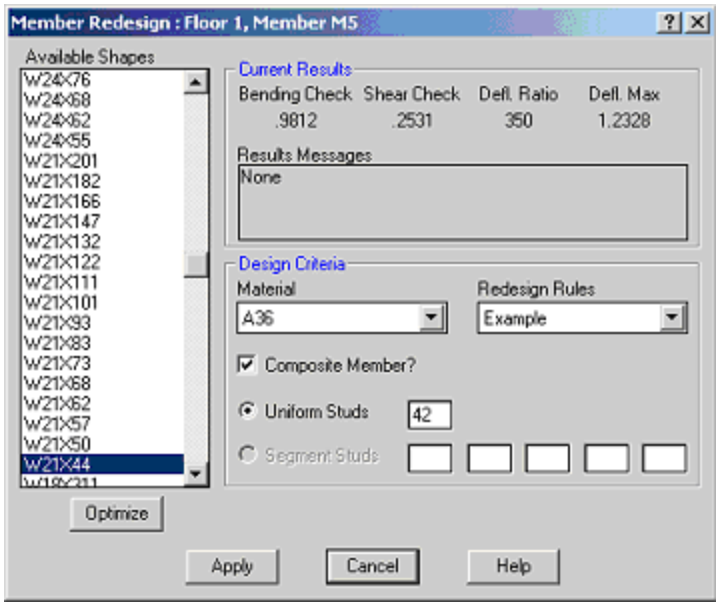
Accessing the Member Design Rules Editor

To access this editor you can click the red arrow on any tab in the **Member Design Rules** spreadsheet (click in the **Label** cell to make the red arrow appear).



Member Redesign Dialog

The member redesign dialog allows you to select one member at a time to view the current beam size, redesign rules, and calculated code check limits. You may then interactively select another size for investigation and instantly see the new calculated code check limits. If you are happy with the new size you may then apply that new size to the member.



To Perform a Member Redesign

1. To perform a member redesign after running a solution, click on the Design button on the **Selection** Toolbar and then click on a member. If you do not already have the model view open click the **New View** button on the **RISA Toolbar**.
2. From the available shapes list, select the new size for investigation.
3. Select **Apply** to assign the new size to the member.

Assumptions of the Redesign Dialog

The member Redesign Tool is intended to allow the user to assign a specific shape to their beam after the solution is run. For composite beams, the amount of studs may also be adjusted in this dialog.

The Redesign dialog optimizes the beams creating a list of shapes using the following assumptions:

1. The Design List is based on the shape type assigned to the member to create a list of available shapes to start with.
2. The Redesign shape list is organized from least weight to greatest weight.
3. This list of available shapes will be reduced based on the criteria set in the Member Design Rules for geometry and maximum code check restrictions.
    1. For composite design, the program will also use the restrictions from the Model Settings - Composite tab.

When a user makes a change in the Redesign dialog, the following occurs:

1. When a new member size is selected, this size becomes an explicit shape size assigned to the member. The shape will no longer be optimized upon subsequent solutions.
2. The member results will be updated to reflect the new shape's code checks. However, the changes in member forces (due to changes in self-weight for example) will not be considered until after the model is re-

solved. This tool allows the user to manipulate the model and get reasonable results for size changes, however the model has to be re-solved to get the final results using the updated sizes.

3. If the user selects the "Optimize" button, this removes the previous explicit shape assigned to the member and the program will optimize the shape again for subsequent solutions.

Note

- Member Redesign is not available for wood members.

Diaphragms

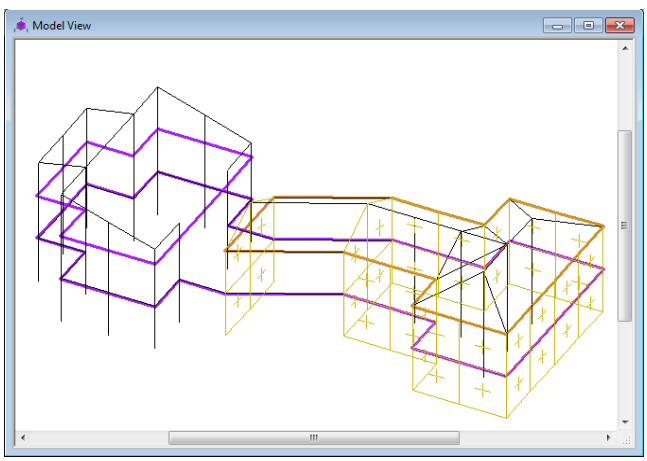
Diaphragms provide lateral load distribution functions, and are necessary for automatic Wind and Seismic load generation. There are three fundamental types of diaphragms in structural modeling:

- Rigid Diaphragms
- Semi-Rigid Diaphragms
- Flexible Diaphragms

Note:

- When considering diaphragms in RISAFloor, understand that diaphragm modeling here is for taking the model into RISA-3D for the lateral analysis/design. RISAFloor is a gravity-only program.
- 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**.



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 joints within the diaphragm plane together for both translation and rotation, but only within the plane of the diaphragm. This is typical behavior for most slabs and decks, which attribute vertical loads based on the tributary area of their supporting members.

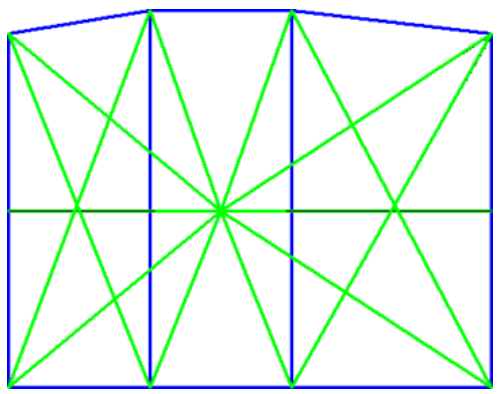
Note:

- 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 joints in the diaphragm, therefore preserving the diaphragm's rotational degree of freedom (something which traditional nodal slaving is incapable of).



Connectivity

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

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

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

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

Rigid Diaphragms in a Combined RISAFloor/RISA-3D Model

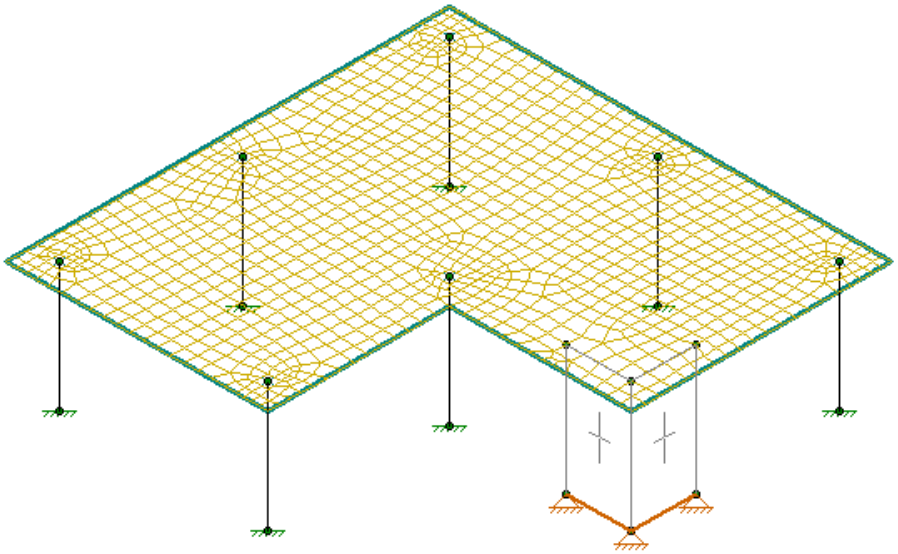
A RISA-3D model that is linked up to RISAFloor has an automatic rigid diaphragm analysis and design. Each individual slab/deck polygon is converted to a rigid diaphragm within RISA-3D. Therefore, it is possible to have multiple independent diaphragms at any given floor elevation.

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.

Semi-Rigid Diaphragms in a RISAFloor/RISA-3D Model

The Semi-Rigid diaphragm in RISA-3D will help 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. 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..



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 in RISA-3D. The **Icr** slab stiffness only affects the out-of-plane stiffness. For further information on cracked slabs, see Elevated Slabs - Analysis. 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.

Note:

- Sloped semi-rigid diaphragms are not supported. Therefore, semi-rigid diaphragms always exist in a flat horizontal plane.

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.

1. Contours: The contours will display the global axis forces over the slab. See the Model Display Options - Slab section 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.
2. 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 finds 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 the IFST Topic.

Semi-Rigid Modeling Tips for Beam Supported Floors

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.

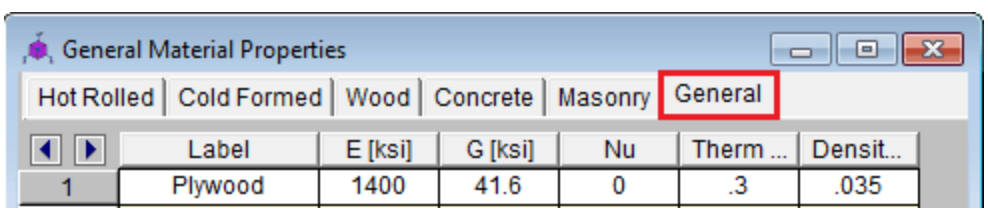
Diaphragm Material: The Semi-Rigid diaphragm material uses the General Materials. There are a list of default General Materials, however you can add to this list to match your diaphragm. See Materials for further information on creating your own General Material.

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 truely one-way, you may consider using a Flexible diaphragm.

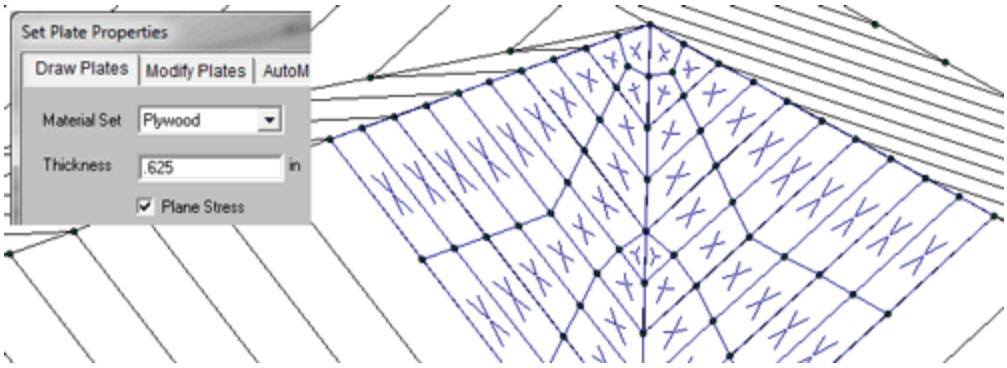
Semi-Rigid Diaphragms for RISA-3D Only Models

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:

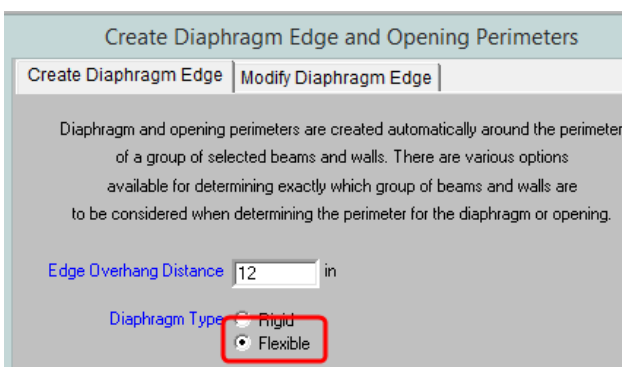
- When meshing a floor diaphragm around members, you want to make sure that the corners of your plates meet at members. Otherwise, when applying surface loads to plates, the members will not see that load.

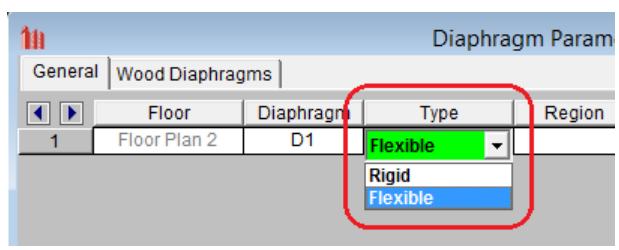
Flexible Diaphragms

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 Combined RISAFloor/RISA-3D Model

A flexible diaphragm can be defined in a combined RISAFloor/RISA-3D model. For wood diaphragms, the program will actually design 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.





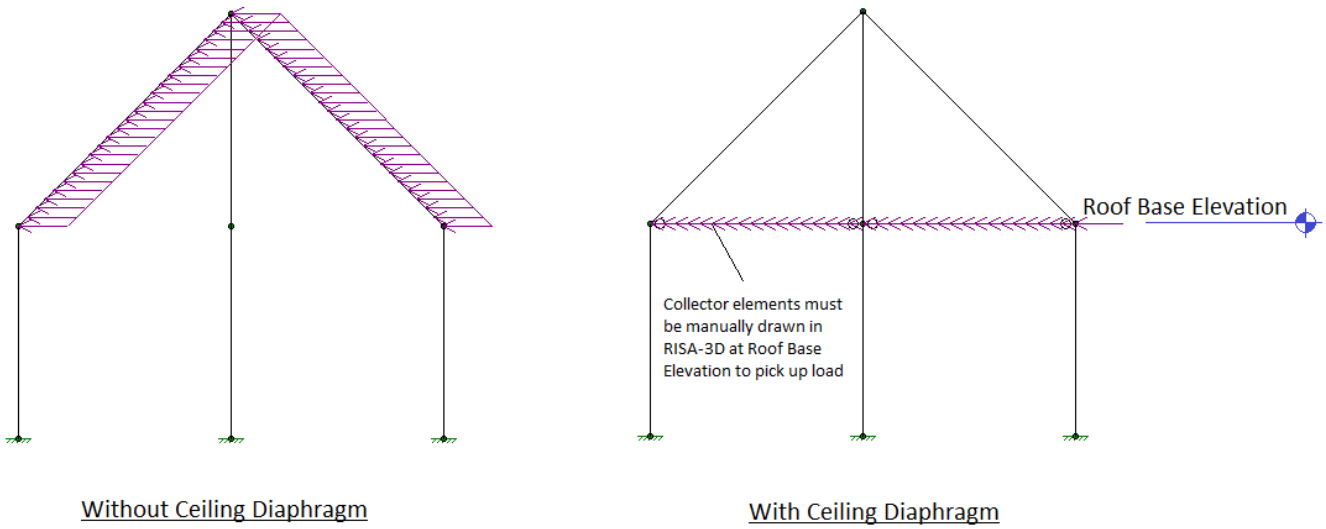
Note:

- 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)

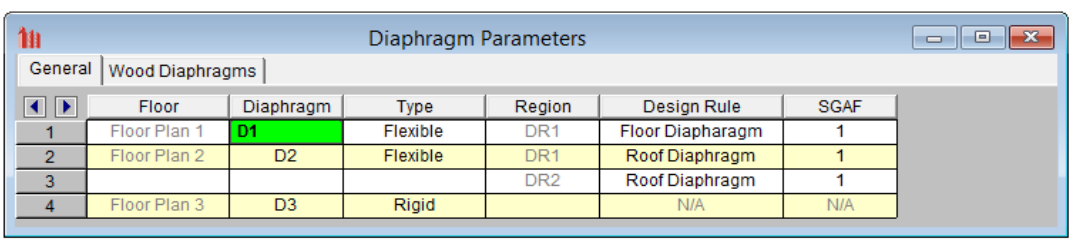
When a flexible diaphragm is applied to a sloped roof in RISAFloor, a setting exists to specify whether a ceiling diaphragm is present.



The load attribution for the flexible diaphragm will occur 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 and may be accessed by selecting **Diaphragms** on the **Spreadsheets** menu under **Data Entry**. Alternatively, it can be accessed by clicking the button on the Data Entry Toolbar.



	Floor	Diaphragm	Type	Region	Design Rule	SGAF
1	Floor Plan 1	D1	Flexible	DR1	Floor Diapharagm	1
2	Floor Plan 2	D2	Flexible	DR1	Roof Diaphragm	1
3				DR2	Roof Diaphragm	1
4	Floor Plan 3	D3	Rigid		N/A	N/A

The diaphragms spreadsheet within RISAFloor is a simplified version of the diaphragms spreadsheet that is available from within the RISA-3D interface, which can be accessed via the Director Tool.

Floor displays the name of the Floor on which each diaphragm is located.

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

Note:

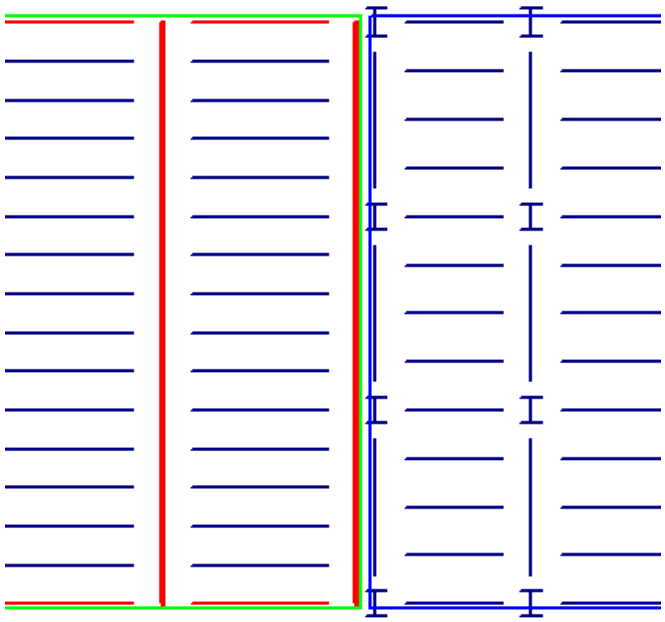
- For information about the **Wood Diaphragms** tab see the Diaphragms - Analysis and Results topic.

Diaphragm Modeling Tips

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.

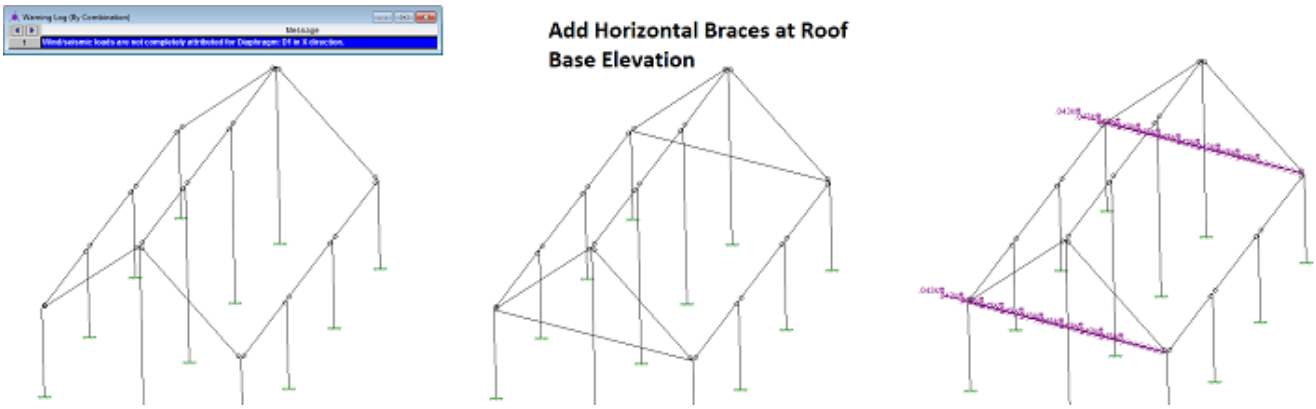
Below is a screenshot of the gap for example:



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

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

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



For additional advice on this topic, please see the RISA 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.

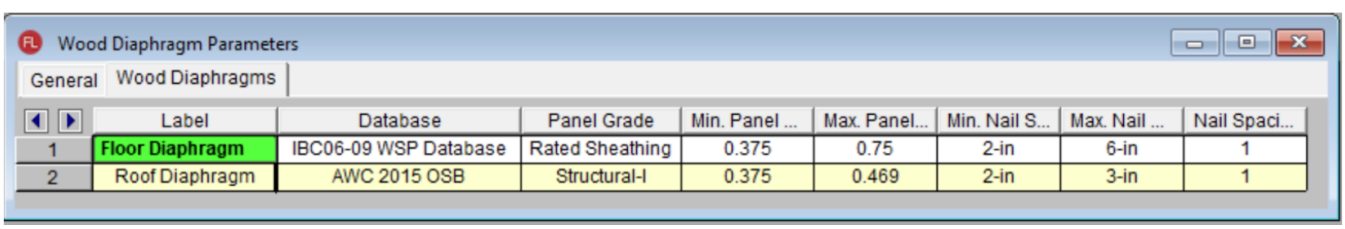
These area loads then gets broken down into a series of "transient" distributed loads that are applied to the members which support the diaphragm. These transient load can be viewed as Basic Load Cases. Refer the section on Member Area Loads for more information about how these transient loads are generated.

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.

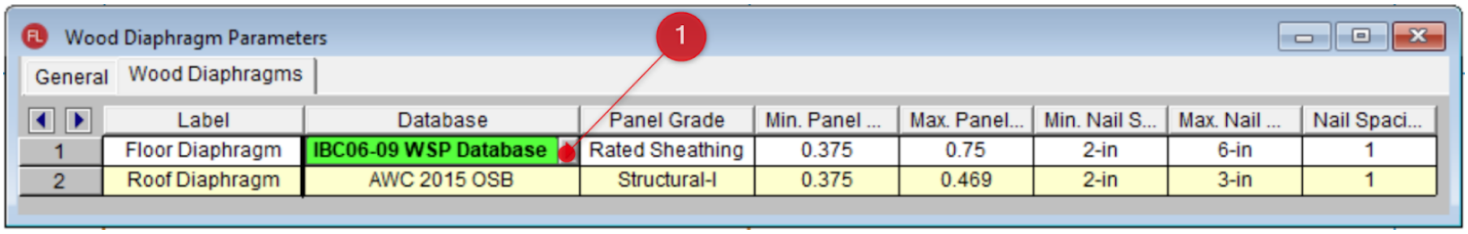
Wood Diaphragm Design Rules

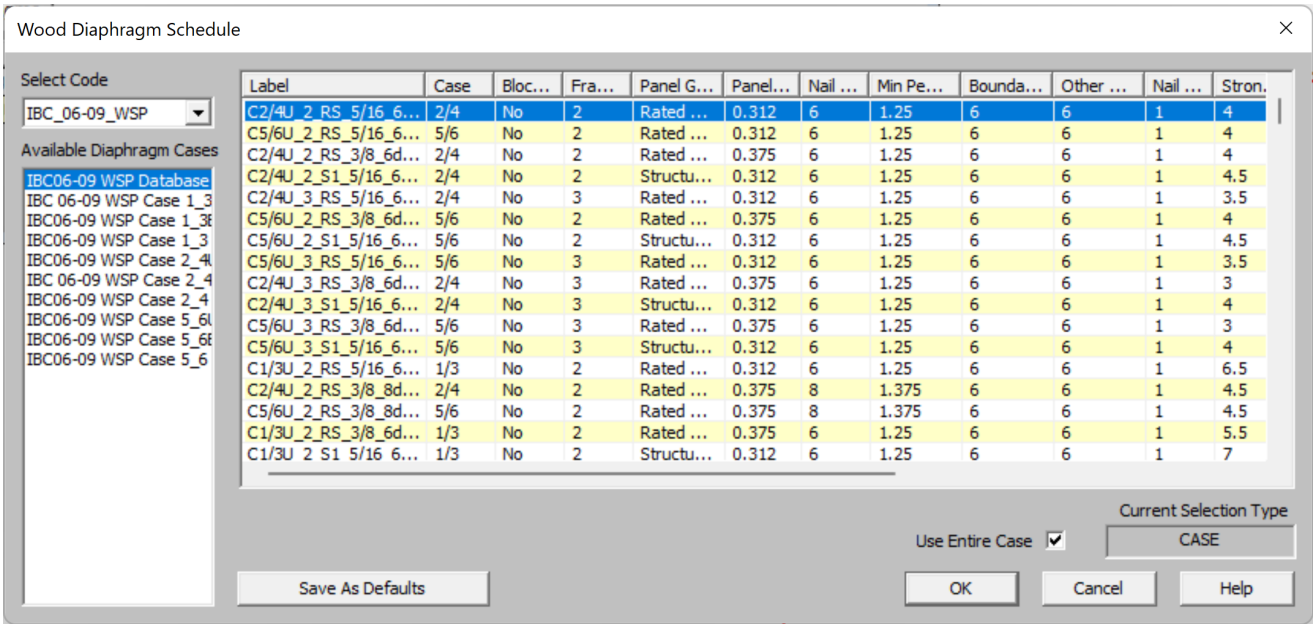
Note that these are only available in a combined RISAFloor/RISA-3D model. Here you can input the parameters for your diaphragm design.



	Label	Database	Panel Grade	Min. Panel ...	Max. Panel...	Min. Nail S...	Max. Nail ...	Nail Spaci...
1	Floor Diaphragm	IBC06-09 WSP Database	Rated Sheathing	0.375	0.75	2-in	6-in	1
2	Roof Diaphragm	AWC 2015 OSB	Structural-I	0.375	0.469	2-in	3-in	1

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

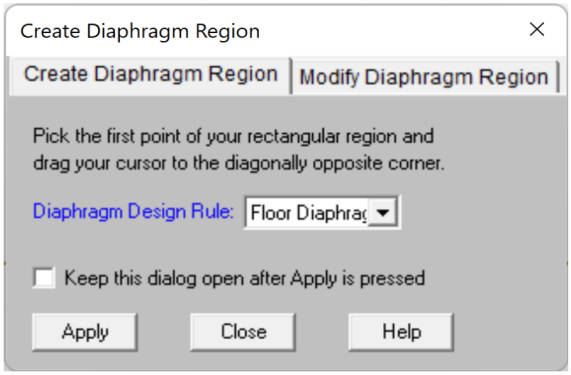




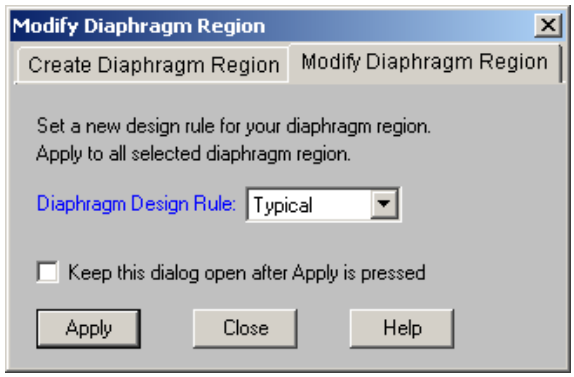
For more information on this, see the Appendix F - Wood Shear Wall Database Files topic.

Diaphragm Regions

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



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



Below are some limitations for drawing diaphragm regions:

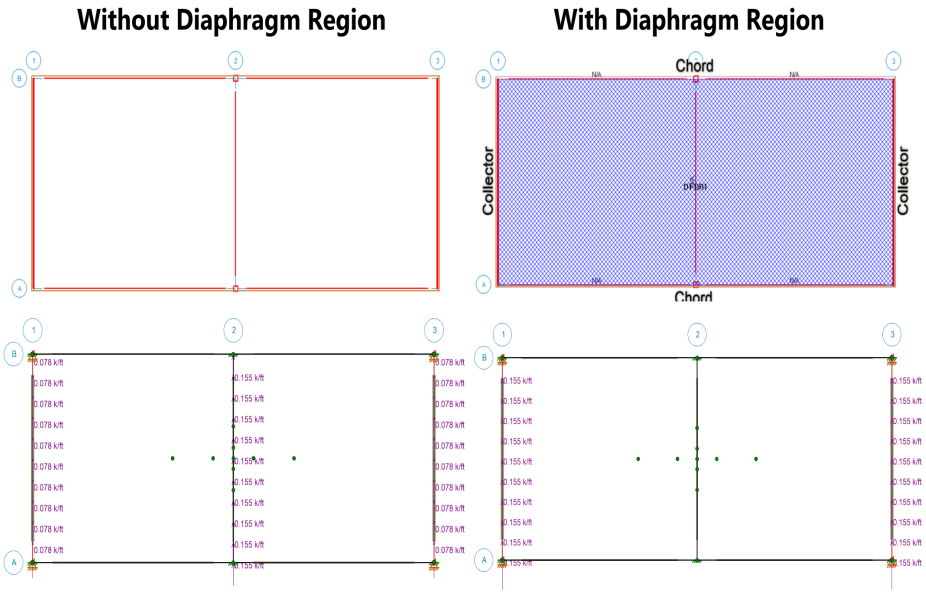
- Diaphragm Regions can only be drawn on flexible diaphragms. Therefore, a Flexible slab edge must be present around the portion of the floor where the diaphragm region is drawn.
- Diaphragm Regions must be rectangular.
- Diaphragm Regions must have a closed-circuit of **lateral** members around their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

Load Attribution

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

If Diaphragm Regions have not been drawn then loads are attributed to all horizontal members within the floor plane (except members perpendicular to load direction). Therefore, drag struts and collectors MUST be modeled in order to get proper load attribution into lateral frames.

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



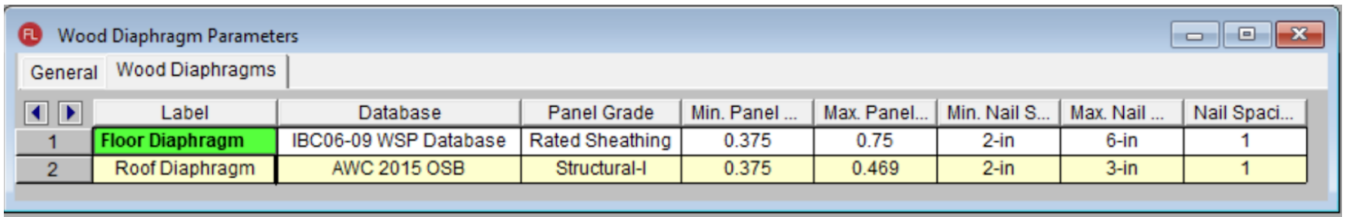
If a diaphragm region has been drawn on a Floor, the total generated lateral load within that region will be attributed to the edges of that diaphragm region, and no other members within that region will receive lateral loads. This allows you to use diaphragm regions to explicitly define load paths in 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



	Label	Database	Panel Grade	Min. Panel ...	Max. Panel...	Min. Nail S...	Max. Nail ...	Nail Spaci...
1	Floor Diaphragm	IBC06-09 WSP Database	Rated Sheathing	0.375	0.75	2-in	6-in	1
2	Roof Diaphragm	AWC 2015 OSB	Structural-I	0.375	0.469	2-in	3-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		

Code – Reports the code used to design the diaphragm.

Design Rule - Lists the Design Rule which has been assigned to this diaphragm region.

Panel Grade – Reports the grade of the panels used in the design (Structural-I, Rated Sheathing, or Other)

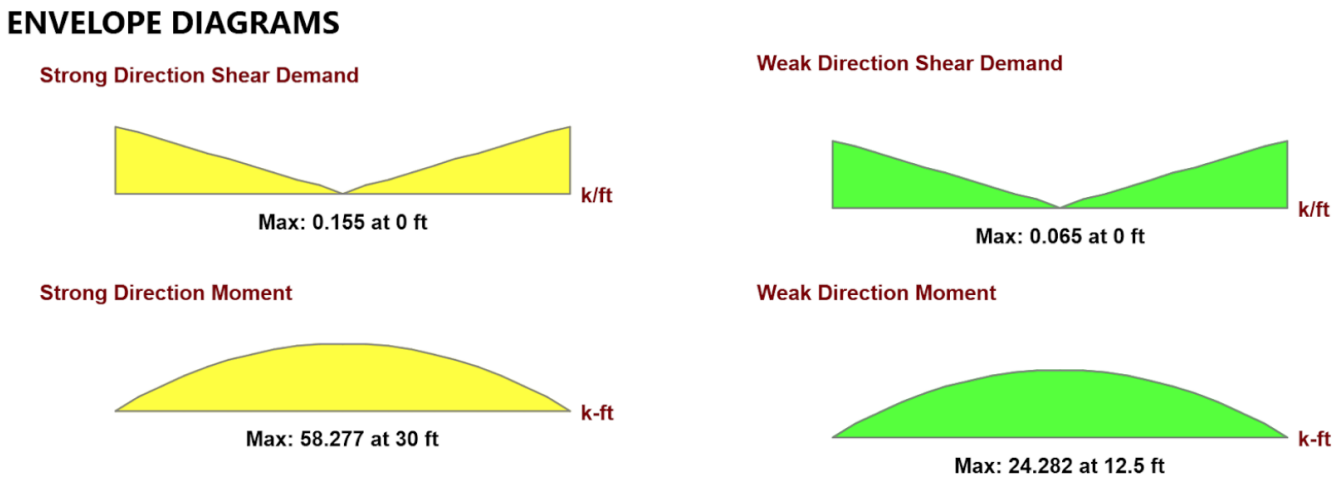
Panel Schedule – Reports the nailing schedule database used to optimize panel selection

SGAF - The specific gravity adjustment factor, input in the Diaphragms spreadsheet, reduces the capacity of the diaphragm. Note 2 from Tables A.4.2A-A.4.2C of the NDS 2005 SDPWS gives the calculation of this value as: SGAF = (1 - (0.5 - G)), where G = Specific gravity of the floor framing members.

In addition the basic geometry information for the diaphragm (**Total Length**, **Total Width**, **Elevation**, and **L/W Ratio**) are also reported here.

Envelope Diagrams

The detail report presents envelope diagrams for the shear and moment demand seen on the flexible diaphragm for both the strong and weak directions. The diagrams are color coded yellow for strong direction and green for weak direction for quick reference.



Design Summary

The results for design summary are displayed for both strong and weak directions as shown below.

DESIGN SUMMARY			
STRONG DIRECTION		**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)

Panel Required Capacity is the scaled maximum design shear, also reported on the envelope diagram. The maximum shear as determined from analysis is compared against Fp for seismic loads. If Fp exceeds the shear as determined by analysis then that shear is scaled up to match Fp. For more information see ASCE7-10, Section 12.10.1.1.

Panel Provided Capacity is the capacity of the designed diaphragm. For diaphragms with multiple nailing zones, this will be reported as capacity at the point of maximum shear. For more information on how the program selects between the various nailing schedules provided in the database, please refer to Appendix F.

The **Ratio** provides the code check for the diaphragm based on shear demand versus shear capacity.

Governing LC gives the load combination that controls the design. This is based on whichever load combination resulted in the highest ratio for shear capacity. In parentheses it is stated whether the governing load combination was seismic or wind based. If the panel was governed by wind, then you are allowed to use a 40% increase in panel strength. The program will automatically consider this, as long as you have checked the **Increase Nailing Capacity for Wind** checkbox in the Model Settings-Solution tab.

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

The **deflection** of a diaphragm is comprised of two main terms, depending on the equation used. The terms are as follows:

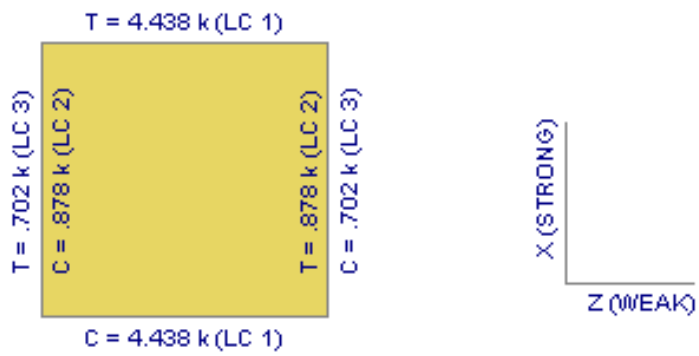
The **Flexure Component** accounts for the deflection of the diaphragm based on bending stresses.

The **Shear Component** accounts for the deflection of the diaphragm based on shear stresses.

This deflection is based on the actual maximum shears in the diaphragm, which are given. It is not based on the scaled Fp forces, thus we report the unscaled shears.

Chord Force Summary

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.

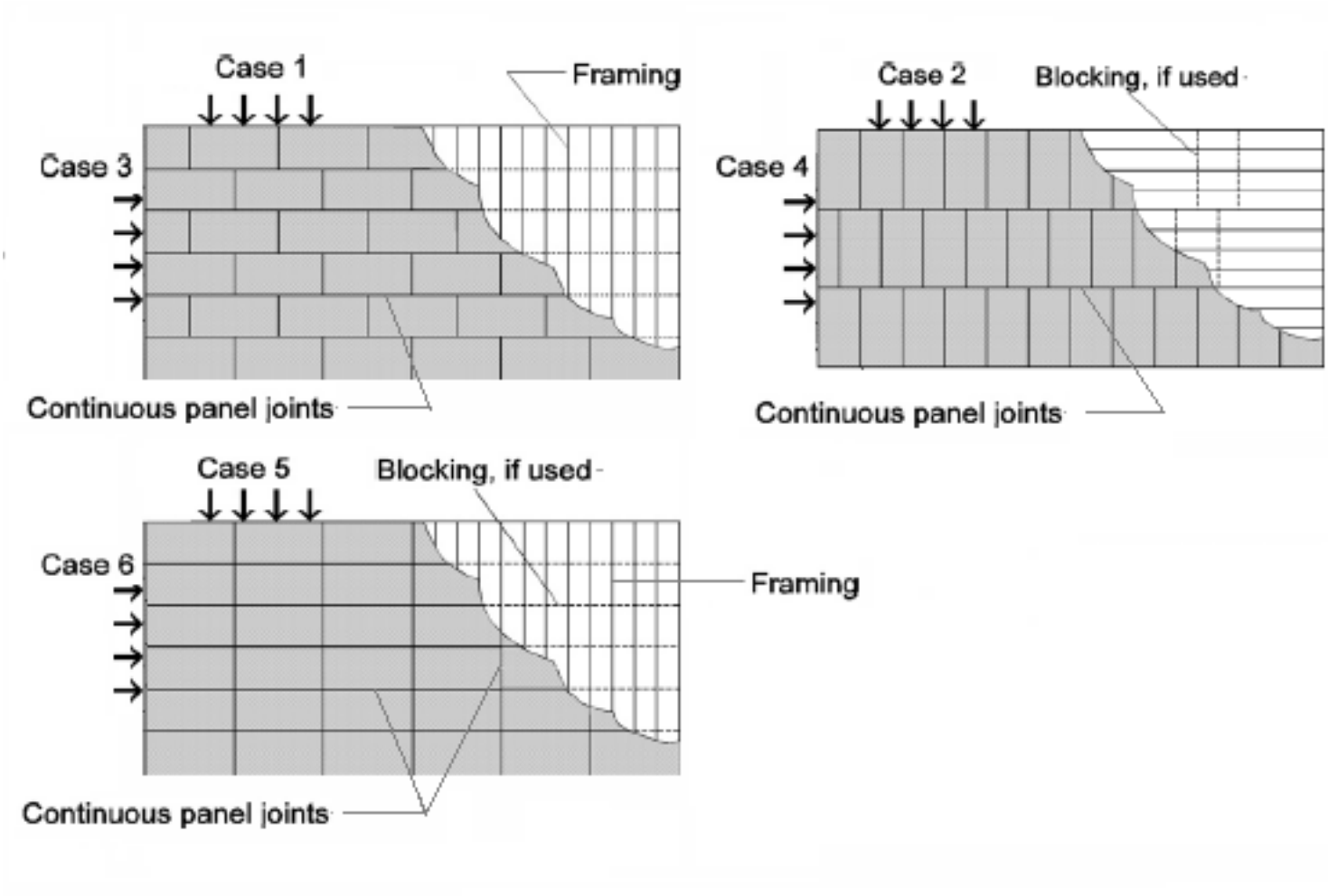
Design Details

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

Panel Label - This is the call-out that the program uses to describe the specific panel and nailing. This information is expanded upon afterward.

Layout Case- This reports the case (panel layout) used for the diaphragm design. Because each diaphragm has a strong and weak direction, the program will actually report two cases associated with each diaphragm.



Blocked – This reports whether the diaphragm was designed as blocked or unblocked

Panel Thickness – This reports the decimal thickness of the wood panels. For convenience the following table lists the decimals that correspond with common panel thicknesses:

Panel Thickness	Decimal Equivalent
5/16	0.3125
3/8	0.375
7/16	0.4375
15/32	0.4688
19/32	0.5938

Required Nail Penetration – This is the minimum nail penetration as required by the IBC/NDS. This value is taken from the diaphragm nailing schedule/database. For more information on the database refer to Appendix F.

Nail Size - This is the size of the nail required for this call-out.

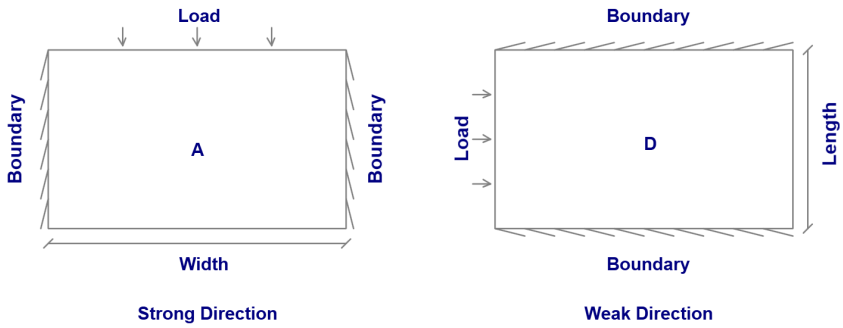
Nail Spacing Schedule

A diaphragm region is split into various nailing zones. The optimum nailing is chosen from the diaphragm nailing database for the highest point of shear of the diaphragm region in each direction. The program considers the Nail Spacing Increment as defined in the Design Rules. If a nailing arrangement is available that has spacing wider than the sum of the nailing increment and the spacing in the "higher" zone then a new nailing zone can be created. The arrangement of the nailing zones is such that they line up with the threshold shears of the capacities of the zones.

NAIL SPACING SCHEDULE

Zone	Location (ft)	Label	Lines	Framing Width (in)	Boundary (in)	Cont Edge (in)	Other Edge (in)
A	0	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:

The **Zone** gives the zone label so you can compare it to the zone layout below.

The **Location** defines the distance to that zone from the edge of the diaphragm. Note that this is a symmetric distance from either side of the diaphragm.

The **Label** is giving the call-out for the diaphragm panel.

The **Lines** entry gives the number of lines of nails on each panel edge. Normally, this value is greater than one only for High-Load diaphragms. This entry is taken directly from the diaphragm nailing schedule database and is not otherwise used in the diaphragm design.

Note:

- If there is only one zone, the location value will just be zero.

The **Framing Width** displays the nominal width required of the framing members that support the diaphragm panels. This entry is taken directly from the diaphragm nailing schedule database and is not otherwise used in the diaphragm design.

The **Boundary** column gives the required boundary edge nail spacing.

The **Cont Edge** column gives the required nail spacing at continuous boundary edges that are parallel to the direction of load.

The **Other Edge** column gives the required nail spacing at all other edges. i.e. edges that are not considered to be boundaries or continuous edges parallel to load.

Collector Forces

The collector forces are not currently reported in the Detail Report. However, the applied seismic forces are considered in the normal member design of the collectors. The only issue here is that the seismic forces have not been amplified by the seismic overstrength factor (Ω).

Deflection Calculations

Flexure Component

If you think of the diaphragm as a beam with the chord members acting as the flanges, then this is the deflection due to the tension and compression forces that develop in the chords.

$$\delta_{flex} = \frac{5vL^3}{8EAW}$$

v = Maximum shear load in diaphragm region

L = Diaphragm dimension perpendicular to the direction of applied load

E = Chord modulus of elasticity

A = Chord cross-sectional area

W = Diaphragm dimension parallel to the direction of applied load

Shear Component

If you think of the diaphragm as a beam with the diaphragm sheathing acting as the web of the beam, then this component represents the deflection due to the shear forces in the sheathing. This term is based on the apparent shear stiffness (Ga) as described in the NDS *Special Design Provisions for Wind and Seismic*. As such, this term includes the effects of both elastic shear deformation of the sheathing and nail slip of the panels.

$$\delta_{shear} = \frac{0.25vL}{1000G_a}$$

v = Maximum shear load in diaphragm region

L = Diaphragm dimension perpendicular to the direction of applied load

Ga = Apparent diaphragm shear stiffness

For diaphragm regions that have only one zone in each direction the value of Ga is taken directly from the nailing database. For multi-zone diaphragm regions the program internally calculates an equivalent Ga using the formula below:

$$G_a = \frac{1.4v_s}{\frac{1.4v_s}{Gt} + 0.75\alpha e_n}$$

vs = Diaphragm shear capacity (ASD value taken from nailing database)

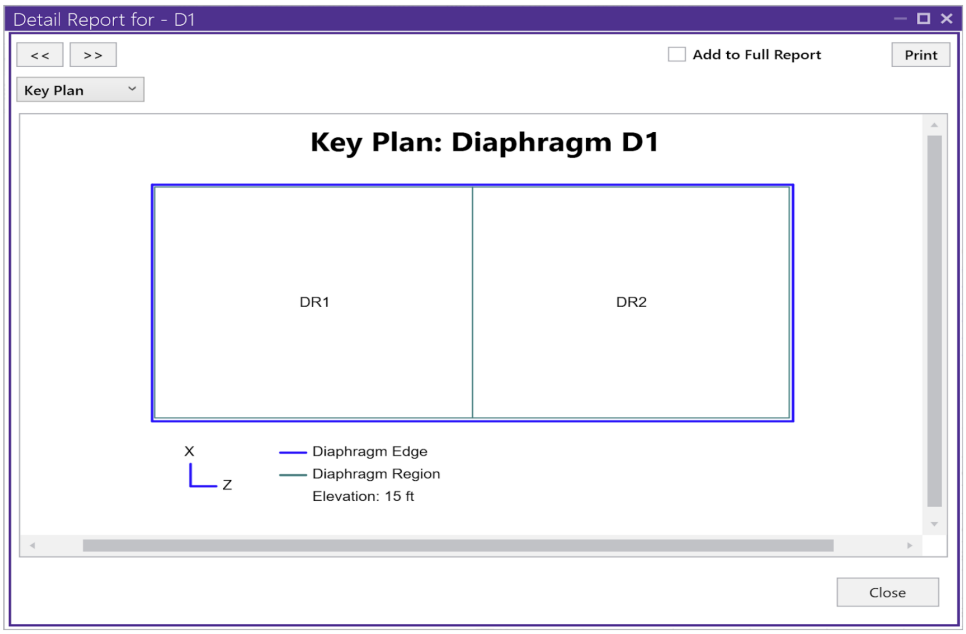
Gt = Shear stiffness of panel depth (taken from nailing database)

en = Nail slip (taken from nailing diaphragm nailing schedule / database)

α = Nail slip adjustment factor. This factor accounts for non-uniform nail slip across multiple zones, and is derived from the process outlined in the ATC-7 and further explained in the APA design / Construction Guide on Diaphragms and Shears Walls.

Diaphragm Key Plan

The diaphragm key plan is just a plan view of the diaphragms at a given level. The key plan can be accessed by clicking 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

Rigid Diaphragms - Currently, the program does not provide any design or code check information for Rigid diaphragms. The rigid diaphragms are solely used to attribute load to the supporting frames or walls.

Unblocked Diaphragm - The program will not optimize multiple nailing zones for unblocked diaphragms.

Chord Slip - The deformation due to chord slip is not considered in the deflections calculations.

Required Framing Size - The detail report lists a required framing nominal width, however no check is performed to ensure that the framing modeled actually meets this requirement.

Diaphragm Loading - The only load seen by flexible diaphragms is the load coming directly from the automatically generated loading. Any loading added in RISAFloor or RISA-3D will not be considered in the design of the diaphragms and will not be attributed to the shear walls.

Sloped Diaphragms - Currently the program will not design sloped flexible diaphragms. At solution in RISAFloor the program will force you to change the diaphragm to rigid. This feature will be added in an upcoming release.

Moisture Content - The program is not considering a reduction in strength due to moisture content.

Orientation - No diaphragm design will be done for regions placed over deck that is not parallel to the Global X and Z axes.

2021 SDPWS Provisions - 2021 SDPWS provisions are not considered for flexible diaphragms.

DXF Files

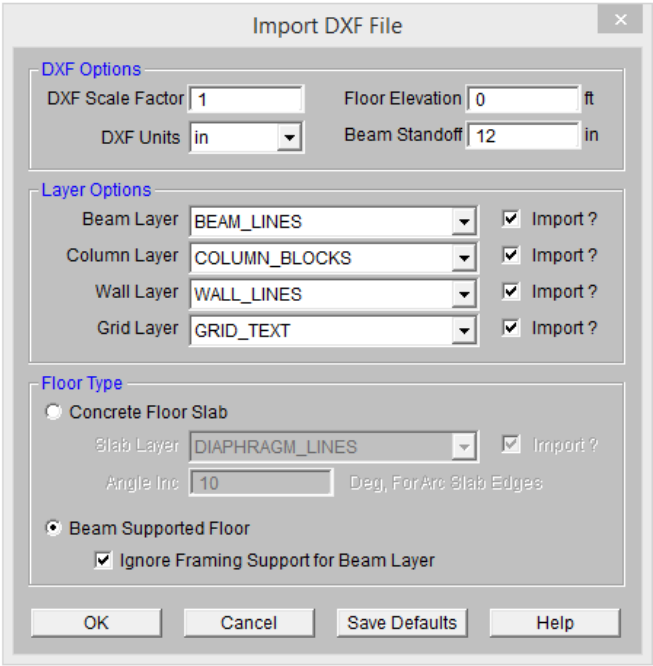
You may read and writeexport DXF files. Generally, you would read in a DXF file to create the geometry for a new structural model, or you could write out a DXF file from an existing model to form the basis for a model or a CAD drawing. This feature provides two-way compatibility with any other program that can read and write DXF files. This includes most major CAD programs and many analysis programs.

Note

- 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.

Importing DXF Files

You may import columns, beams, walls and a project grid. Each DXF file should contain only one Floor level. You have several options available for controlling how DXF files are imported. They are as follows:



DXF Units

Select the units you used in the CAD model. The options are none, inches, feet, mm, cm, and meters.

DXF Scale

Enter the scale factor that will scale the DXF file to its full size. For instance, if you had created a scaled model in AutoCAD at a scale of 1/4"=12", then the appropriate scale factor to produce a full size RISAFloor model would be 48.

Floor Elevation

Enter the desired elevation for the new floor level that will be created from the DXF file.

Note

- DXF will not be import a new floor if there is an existing floor at the defined elevation.

Beam Standoff

In CAD drawings, beams are not typically drawn all the way to the working point where they connect. Instead, they are shortened by a standoff distance at each support. Enter the maximum standoff distance used in the DXF file. A typical value would be 12 inches. RISAFloor will use this distance when determining the supports for each beam.

Note: this entry will be ignored if you also have the **Ignore Framing Support for Beam Layer** checkbox selected.

Column Layer

Specify the layer where the column blocks exist. The program will search for any blocks in this layer and import them as column locations.

Note

- The "insertion point" of the block will be viewed as the centerline of the column.

Beam Layer

Enter the layer where the beam data is defined.

Note

- Unless, the **Ignore Framing Support for Beam Layer** option is checked, then only fully supported beams will be imported.

Wall Layer

Enter the layer where the wall data is defined. Walls must be defined as POLYLINES or they will not be imported. The *width* of the Polyline will determine the thickness of the wall element in RISAFloor.

Note

- Polylines are treated differently in earlier versions of the DXF file format. Therefore, walls cannot be imported from a DXF file format older than that used in AutoCAD2000.
- While walls are only imported if they are defined as polylines, they are always exported as regular lines. Therefore, adjustment is always necessary if you are to re-import a file that was originally exported from RISAFloor.

Grid Layer

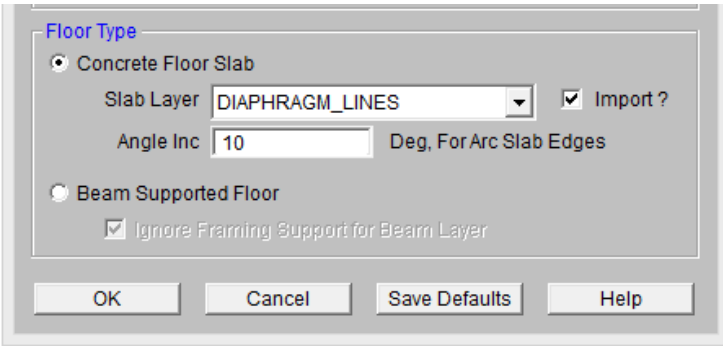
Enter the layer where the project grid data is specified. This will replace any existing project grid data and replace it with the project grid from the DXF file. Therefore, it may be desirable to un-check the **Import?** box to prevent the current Project Grid from being replaced.

Limitations

Cantilevered beams Cantilever beams are unsupported at the cantilever end. Therefore, they are not imported unless the **Ignore Framing Support for Beam Layer** option is checked. When this box is checked, the cantilevers will still require some manipulation within RISAFloor. However, your basic model geometry and secondary framing will be more complete.

Floor Type

The Floor Type can be set to Concrete Floor Slab or Beam Supported Slab. A RISAFloor ES license is required in order to create a Concrete Floor Slab.



Concrete Floor Slab

The Slab Layer will be imported based on the layer selected. Concrete Floor Slab don't require framing of beams, columns or walls to be imported however all the options above can be imported with the slab if they are checked on.

- The slab can be drawn in DXF with lines, polylines or arcs. Splines are not currently supported.
- The slab outline must be a **closed shape** connected at end to end. A single line segment or an open outline (i.e. an U shape) will not import.
- The entire slab outline must be on the same layer.
- The slab will only import from a 2D DXF file.

Angle Inc

The slab will break an arc into straight line elements automatically during the import. This number indicates the number of degrees that an imported arc should be broken into.

DXF File Vertical Axis

Although it is not specifically noted in the AutoCAD documentation, the implied default vertical axis is the positive Z-axis of the current User Coordinate System.

The default vertical axis in RISA is usually the positive Y-axis and may be specified on the **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.

Translate Section Sets to Layer Names

This is a Yes/No check box. If you check the box “Yes”, the program will translate Section Set Labels to layer names. Layers will be created in the DXF file corresponding to the section set labels in the RISA model. A “Yes” choice here overrides any layer name entered for the member layer.

The program will add a prefix to each section set layer to designate what type of material that section is. The prefixes are as follows:

Material Type	Layer Prefix
Cold Formed Steel	CF_
Concrete	CN_
Hot Rolled Steel	HR_
General Materials	GN_
Wood	WD_

For example, let's say you have designed a structure with Hot Rolled steel section sets called "Column"and " Girder", as well as a Wood section set called "Joist". If you type in a member layer name such as "STEEL" then all members, regardless of size, will appear on a layer named "STEEL". However, if you choose "Yes" for the *translate section sets to layers* option, then all the member that are assigned to the Column section set will appear on a layer named "HR_Column", the girders on a layer named "HR_Girder", and the joists on a layer called "WD_Joist".

Translate Shapes to Layer Names

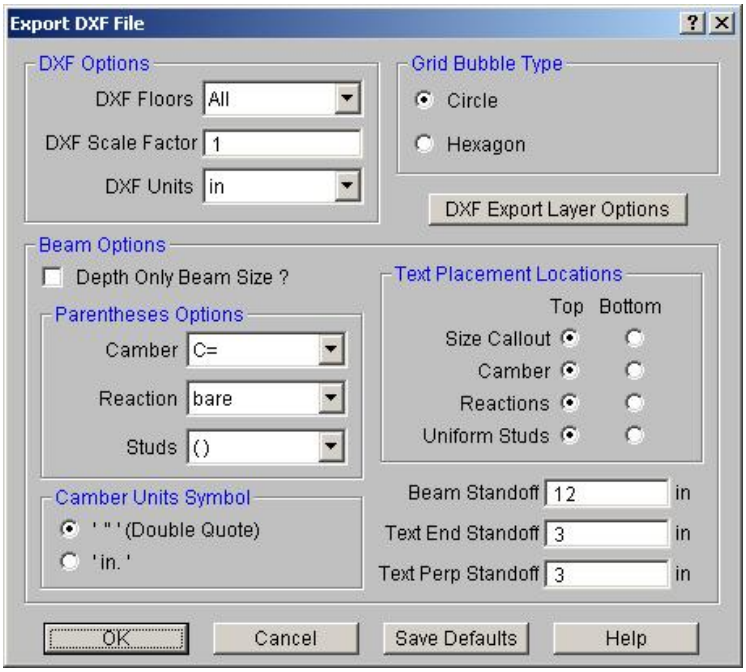
This is a Yes/No choice. If you choose "Yes", the program will take members and assign them to a layer which uses their shape label as the layer name. Layers will be created in the DXF file corresponding to the shape labels in the RISA model. A "Yes" choice here overrides any layer name entered for the member layer.

Note:

- If you check BOTH the *Translate Section Sets* and *Translate Shapes* boxes, then only the explicitly defined shapes will be put placed on a layers according to their shape labels. All members defined with section sets will still be placed on layers according to their section set label.
- Please note that if the section set database shape designation includes one or more decimal point (".") characters, the export will translate each occurrence of a decimal point character into an underscore ("_") character. For instance, a section set or shape label such as a C10X15.3 will translate into a layer name of "C10X15_3" or "HR_C10X15_3".
- The DXF format does not properly recognize certain ASCII text characters for layer names (< > / \ " : ; ? * | = '). Therefore, these characters should be avoided for shape or section sets when using the "translate to layer names" options.

Exporting DXF Files

Points, columns, walls, beams, and text can be exported to an ASCII DXF file. Any other information such as the boundary conditions, loads, member end releases, etc. are not exported. You have several options available for controlling how DXF files are exported as follows:



Floors

You can export all the floors of the model or individually select the floor you wish to export.

DXF Scale

Enter the factor that will cause the full scale RISA model to be scaled up or down to the desired drawing scale. For example, if you created a full scale model that you wanted scaled down to 1/4"=12", the factor would be 0.020833, which is (.25/12).

DXF Units

Select the units you desire the CAD model to be created in. The options for the DXF units are none, inches, feet, mm, cm, and meters.

Grid Bubbles

You can export the project grid bubbles either as Circles or Hexagons.

Beam Options

You can create DXF files that give the Beam Size, Camber, End Reactions, and Number of Studs. Alternatively, you can create a DXF file that only shows the beam depth.

Beam Standoff

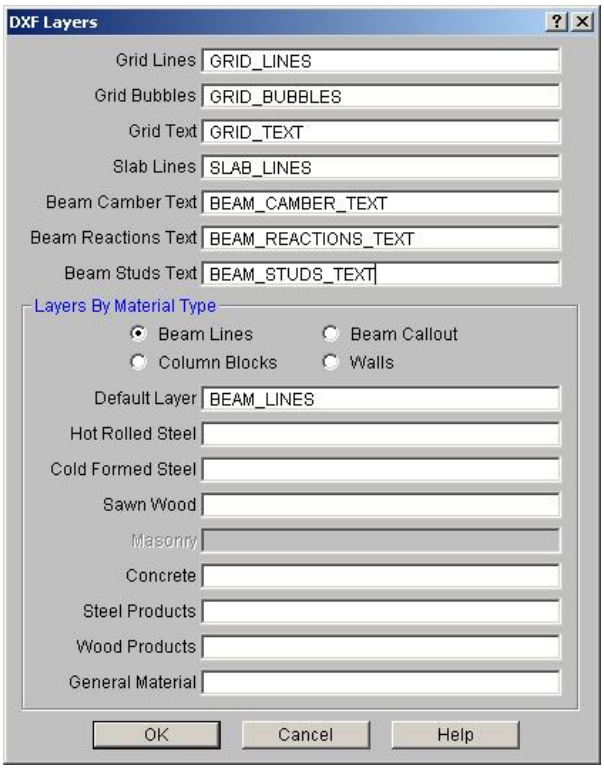
Typically beams are not drawn all the way to the ends of the beams or columns to which they connect. Instead, they are shortened by a standoff distance at each support. Enter the standoff distance used in the DXF file.

Text Standoffs

Enter the distance you wish to have the text “stand off” from the beams and beam ends.

DXF Export Layer Options

In addition to the standard exporting features, you can further customize the DXF file that is created using the DXF Export Layer Options.



This dialog gives you layer control over the DXF file that will be created. By accessing these controls, you can create different layers for your different material types.

DXF Element Numbers

Different CAD packages handle ordering of geometric data in their DXF files in two basic ways.

Method 1:

Entities are written out into the DXF file based on the order in which they were created within the CAD program itself regardless of the order in which they were selected at the time the DXF file was made. Different operations such as copying, mirroring, arraying, etc. can produce unexpected results and it therefore becomes necessary to consult your CAD program documentation to understand how it stores and orders the geometry that you create via these various operations.

Method 2:

Entities are written out into the DXF file based on the order in which they were selected at the time the DXF file was made. AutoCAD is such a program. In order to control the ordering of the LINE entities, you must select the "Entities" option under the DXFOUT command and then select the lines in the order that you want them to appear in the RISA model.

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. The DXF read/write feature was written based on the DXF documentation for AutoCAD release 14.

The following is a short excerpt of the AutoCAD ASCII DXF format. This information is provided to help you debug any problems you may be having with DXF files that you are trying to read. For more complete information, consult your CAD documentation.

General

A DXF file is composed of sections of data. Each section of data is composed of records. Each record is stored on it's own line. Each particular item is stored as two records, the first record is a group code and the second record is the data or a keyword. RISA only reads the ENTITIES section.

Group Codes

Each 2 record item start with an integer group code. RISA recognizes the following group codes:

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

First and Last Records for a DXF file

Each DXF file must start with the first record as the group code "0". The 2nd record must be the keyword "SECTION". Each DXF file must have the 2nd to last record as the group code "0". The last record must be the keyword "EOF".

Entities Section

The ENTITIES section will be identified by a group code of "0", followed in the next record by the keyword "SECTION". The next record will be the group code 2, followed in the next record by the keyword "ENTITIES".

Item Formats within the ENTITIES Section

The POINT format is started by a group code of "0" followed by the keyword "POINT". The layer name for the POINT will start with a group code record of 8, followed by a record with the actual layer name.

The coordinates for the point will be started by the 10, 20, and 30 group codes respectively for the X, Y, and Z coordinates. Other group codes and data may be present within the POINT data but these will be ignored.

The LINE format is started by a group code of "0" followed by the keyword "LINE". The layer name for the LINE will start with a group code record of 8, followed by a record with the actual layer name.

The coordinates for the first point will be started by the 10, 20, and 30 group codes respectively for the X, Y, and Z coordinates. The coordinates for the second point will be started by the 11, 21, and 31 group codes respectively for the X, Y, and Z coordinates. Other group codes and data may be present within the LINE data but these will be ignored by RISAFloor.

The BLOCK format is started by a group code of “0” followed by the keyword “BLOCK”. The layer name for the BLOCK will start with a group code record of 8, followed by a record with the actual layer name.

The coordinates for the definition point of the block will be started by the 10, 20, and 30 group codes respectively for the X, Y, and Z coordinates. Other group codes and data may be present within the BLOCK data but these will be ignored by RISAFloor.

AutoCAD Layer Names

The only valid characters in an AutoCAD layer name are the letters A to Z, the numbers 0 to 9, and the three following characters: the dollar sign “$”, the underscore “_”, and the dash “-”.

File Operations

You may create, save, and recall files from the **File Menu** or from the toolbar buttons just beneath the menu on the **RISA Toolbar**. Importing and exporting other file types may also be accomplished from the **File Menu**.

Starting Off

When you open the program or start a new model, you are presented with the following option: Create a New Floor

The first option is to help you to create a new model by starting with a new floor plan. The second group of options helps you open existing files by showing you the most recent files or letting you find the model you want in the Open File Dialog.

Creating a New Model

- On the RISA toolbar click the **New Model** button. You will be prompted to 'Save' if you have made any changes to the current model that have not yet been saved.

Opening an Existing Model

1. On the RISA toolbar click the **Open File** button.
2. In the **Look In** field click the drive, folder, or location that contains the file.
3. In the folder list, double-click the folders until you open the folder that contains the file you want.
4. Double-click the file you want to open.

Tip

- To open a model you have used recently, click the **File Menu** and select from the file names at the bottom of the menu.

Saving a New Untitled Model

1. On the RISA toolbar click the **Save Model** button.
2. To save the model in a different folder, click a different drive in the **Save in** field, or double-click a different folder in the folder list. To save the model in a new folder click the **Create New Folder** button.
3. In the **File name** field type a name for the document. You can use long, descriptive filenames if you want.
4. Click **Save**.

Saving an Existing Model

1. On the RISA toolbar click the **Save Model** button.

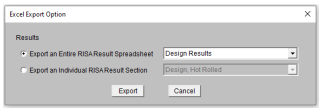
Importing and Exporting Files

You may import DXF drawing files and BIM Exchange files. See DXF Files and Appendix E for more on these file types.

Exporting to Excel Spreadsheet

You may export Result spreadsheets into Excel spreadsheet:

1. On the **File** menu click **Export**.
2. Choose **Excel File**.
3. Select export options , you can select to export an entire RISA result spreadsheet or an individual RISA result section.

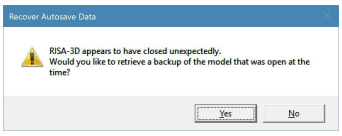


4. Select the folder to export the spreadsheet and export.

Automatic Backup

RISAFloor provides an automatic backup system for systematically backing up your models. The purpose of this is to provide a means for you to recover your data in the event RISAFloor or your computer closes unexpectedly. As you work within RISAFloor 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** on the **Tools 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** option on the **Tools Menu**.

If RISAFloor does close unexpectedly, the next time you start the program the following message will appear:

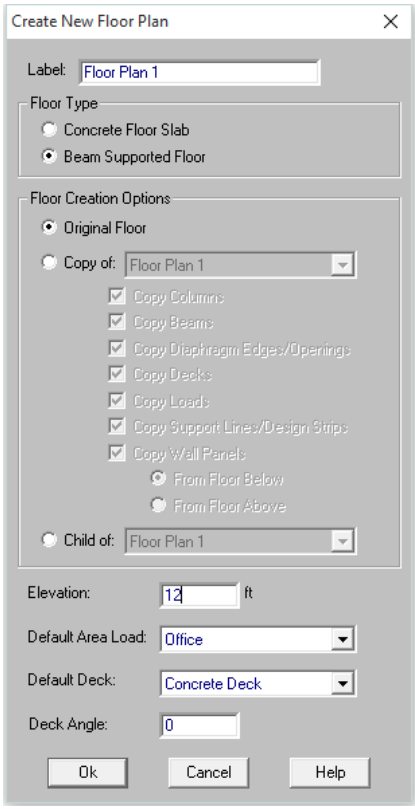


Floors

The primary function of RISAFloor is to design floor systems. The first thing RISAFloor does when you start a new file is prompt you to create a floor. A floor can be an original floor, a copy of an existing floor or a child floor dependent on an existing parent floor. A default area load and deck can be assigned to each floor for modeling simplicity. These options are discussed below.

Creating a New Floor

To create a floor, select **Floors...** on the **Insert** menu or press the button.



Floor Label

This is how the floor will be referenced on the **Floors** spreadsheet.

Floor Type

Concrete Floor Slab

This option is available for users of RISAFloor ES for one-way and two-way floor slab design.

Beam Supported Floor

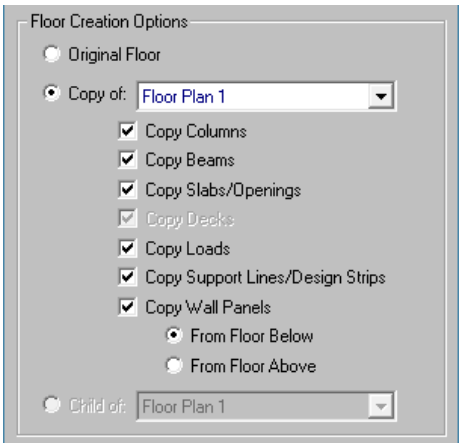
This option is available for all users of RISAFloor for beam supported floors.

Original/Copy/Child

If the floor is a copy of another floor, select the floor which you wish to copy and select the individual elements you want copied. If the floor is a child, specify the parent floor to which the child is bound.

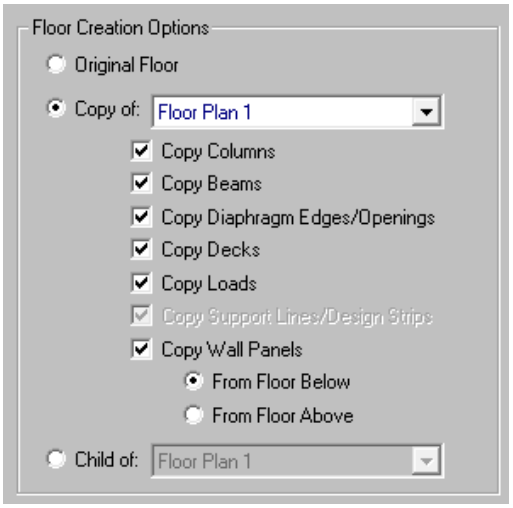
Concrete Floor Slab

This allows you to copy a Concrete Floor Slab from one floor to another. Child is not yet available for Concrete Slab floors.



Beam Supported Floor

This allows you to copy a Beam Supported floor from one floor to another.



Note:

- It is not possible to copy a slab floor to a beam supported floor or vice-versa.
- See the Parent/Child section for more information.

Copy Wall Panels

The copy feature for walls has differing behavior for wood walls vs masonry and general walls. Masonry and general walls will be extended up or down such that one wall spans between multiple floors. Wood walls will be copied up or down such that each floor has a separate wall that forms a stack of wood walls. This allows for strap

forces to be calculated for the wall connections at each floor level. Openings and regions in wood walls will also be copied up/down, and will be referenced from the bottom of the wall.

Elevation

Enter an elevation for the floor from the base.

Default Area Loads and Default Decks

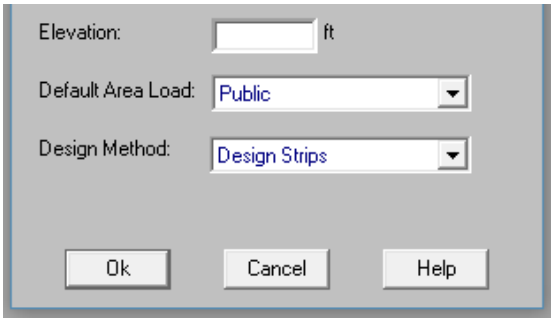
If a deck/slab edge is drawn on a floor level then the program will automatically apply a default area load and a default deck (for beam supported floors) to the area encompassed by the deck/slab edge. These defaults can be edited in the **Floors** spreadsheet.

Note:

- Default decks are not used on slab floors utilizing RISAFloor ES, thus you can ignore this input on slab floors.

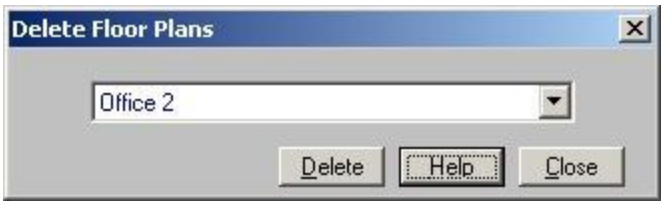
Design Method

There are two methods available; Design Strips or User Defined. Design Strips use the traditional methods that use Column/Middle Strips and the User Defined allows the user to define a uniform rebar.



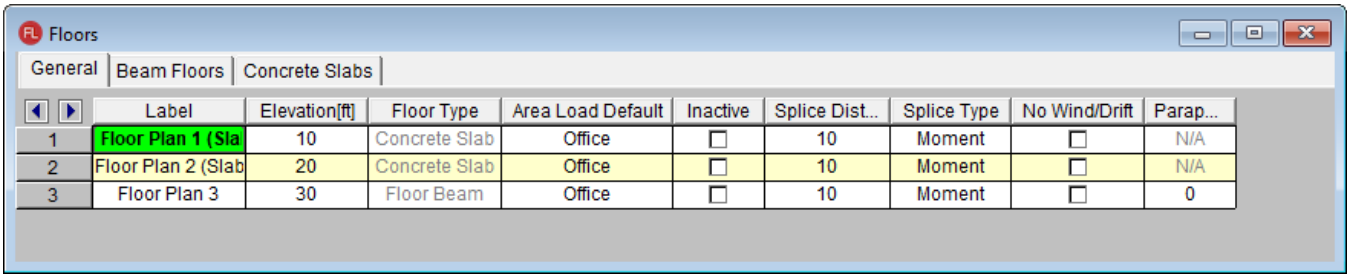
Deleting a Floor

To delete a floor, select the delete floor icon from the RISA Toolbar. Select the floor to be deleted from the drop down list and click the delete button. .



Floors Spreadsheet - General

The **Floors Spreadsheet** records the properties of each floor and may be accessed by selecting **Floors** on the **Spreadsheets** menu. The spreadsheet has three tabs: General, Beam Floors, and Concrete Slabs.



	Label	Elevation[ft]	Floor Type	Area Load Default	Inactive	Splice Dist...	Splice Type	No Wind/Drift	Parap...
1	Floor Plan 1 (Sla	10	Concrete Slab	Office	☐	10	Moment	☐	N/A
2	Floor Plan 2 (Slab	20	Concrete Slab	Office	☐	10	Moment	☐	N/A
3	Floor Plan 3	30	Floor Beam	Office	☐	10	Moment	☐	0

The **General** tab holds the general properties for all floors. The Beam Floors tab contains properties specific to beam supported floors. The Concrete Slabs tab contains properties specific to concrete slab floors.

Create a new Floor

1. On the **Spreadsheets** menu click on **Floors...** to open the spreadsheet.
2. Place the mouse in the last row of the spreadsheet and press Enter to create a new floor. This opens the Create New Floor Plan dialog box.

The entries for the **General** tab are explained below:

Floor Labels

You may assign a unique label to the floors. You can then refer to the floor 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.

Floor Elevation

Enter the elevation of the floor from the base.

Floor Type

This is a column that shows whether the floor plan is a Concrete Floor Slab- Concrete Slab or Beam Supported Floor- Floor Beam. Only when RISAFloor ES is licensed, you can create a Concrete Floor Slab. Note that it is not possible to change a slab floor to a beam floor or vice versa.

Area Load Default

Enter the default area load to be used. Click on the arrow in the cell to choose from the area loads defined in the **Area Load Definitions** spreadsheet. The default area load will be applied to the entire floor except where local Area Loads are defined. See Drawing Area Loads to define local area loads.

Note

- Area loads can only be applied within an edge perimeter. If a given section of the floor is not contained within a diaphragm edge then the area loads will not be assigned to that section. See Creating Diaphragm Edges and Openings for more information.

Inactive

Check this box if you want to inactivate the floor. The solution will behave as if the inactive floor is not present. Using an inactive floor you can define a core portion of the building that is not actually part of the solution model, but which has many child floors that depend on it.

Splice Distance Below

The **Splice Distance Below** defines the default dimension relative to the floor elevation for all splices that occur between the current floor level and the floor below. See Column Primary Data for more information on splice distances.

Note:

- The distance below may not be set less than 1 inch. This is to prevent divide by zero errors when resolving column moments into shears.

Splice Type

The **Splice Type** defines the default splice type as either a Moment splice or a Shear splice for all splices that occur between the current floor level and the floor below. See Column Primary Data for more information on splice types.

No Wind/Drift

You may designate any floor level as a **No Wind** (i.e. mezzanine) level in order to omit it from the generated wind load and drift calculations. See the Load Generation - Wind Loads topic for more information.

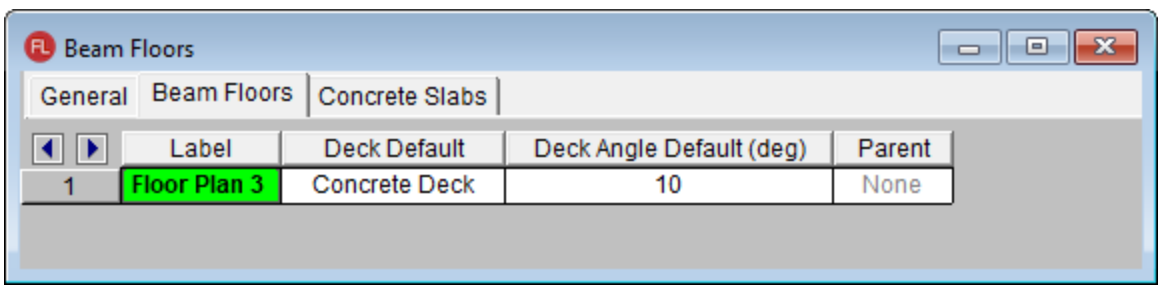
Parapet Height Default

This defines a default parapet for this floor level. For beam supported floors lower and upper level roofs can have a parapet. For slab floors a parapet can only exist at the highest level in the model.

Note:

- Parapets can be different heights around a roof level for a beam supported floor. This would need to be defined in the Columns or Wall Panels spreadsheet.
- Parapets must be a uniform height on slab floors and cannot be edited in the Columns or Wall Panels spreadsheet.
- For more information on column parapets see the Columns topic.
- For more information on wall parapets see the Wall Panels topic.
- For wind load calculation considerations see the Load Generation - Wind Loads topic.

Floors Spreadsheet - Beam Floors



The **Beam Floors** tab holds the floor properties specific to beam supported floors.

Deck and Angle Default

Enter the default Deck that will be used on the floor. Click on the arrow in the cell to choose from the decks defined in the **Deck Definitions** spreadsheet. The default deck will be applied to the entire floor except where local decks are defined. See Assigning Diaphragm and Deck Properties for more information.

Note

- Deck can only be applied within an edge perimeter. If a given portion of the floor is not contained within a diaphragm edge then the deck properties will not be assigned to that area. See Creating Diaphragm Edges and Openings for more information.

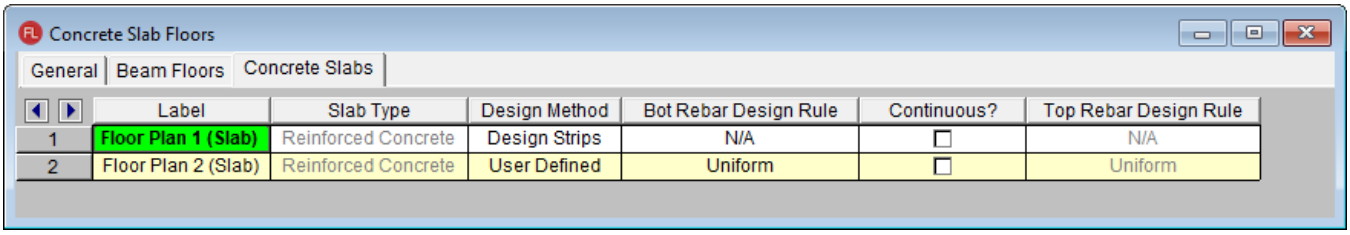
The default deck angle defines the span direction of the default deck. An angle of “0” spans parallel to the horizontal axis. An angle of “90” spans parallel to the vertical axis. This value is ignored for two way decks.

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

Parent

Displays the parent floor. This cell cannot be edited. You may detach the child from the parent by using the right click menu and selecting **Detach Child From Parent** on the child floor in the General tab of the **Floors** spreadsheet. See the Parent/Child section for more information.

Floors Spreadsheet - Concrete Slabs



	Label	Slab Type	Design Method	Bot Rebar Design Rule	Continuous?	Top Rebar Design Rule
1	Floor Plan 1 (Slab)	Reinforced Concrete	Design Strips	N/A	☐	N/A
2	Floor Plan 2 (Slab)	Reinforced Concrete	User Defined	Uniform	☐	Uniform

.

The **Concrete Slabs** tab holds the floor properties for concrete floor slabs.

Design Method

There are two methods available; Design Strips or User Defined. Design Strips use the traditional methods that use Column/Middle Strips and the User Defined allows the user to define a uniform rebar.

Bot Rebar Design Rule

This option only applies to User Defined floor levels. The uniform bottom reinforcement will be based on this Design Rule.

Continuous?

This checkbox only applies to User Defined floor levels. When this checkbox is turned on, the program will put continuous top reinforcement throughout the entire slab. If it is unchecked, the program will only put top reinforcement over the regions defined over the columns and walls spreadsheets. See User Defined for further information.

Top Rebar Design Rule

This option only applies to User Defined floor levels. When the Continuous checkbox is turned on, the top reinforcement for the entire slab will be based on this design rule.

Parent / Child

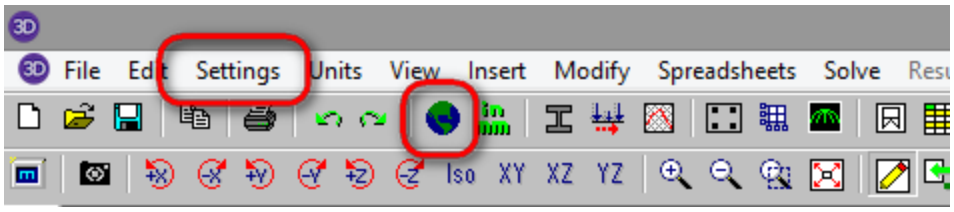
A floor can be an original floor, a copy of an existing floor or a child floor dependent on an existing parent floor. When a floor is a child of another floor, it means that everything that occurs on the parent floor will be mimicked automatically by the child floor. For example: adding or editing columns, beams, walls, diaphragm / decks, etc. on the parent will automatically force the same additions or changes to occur on the child floor. This dependence does not go both ways; changes made to the child are **not** applied on the parent floor.

Note:

- The Parent/Child behavior is not allowed for slab floors.

(Global) Model Settings

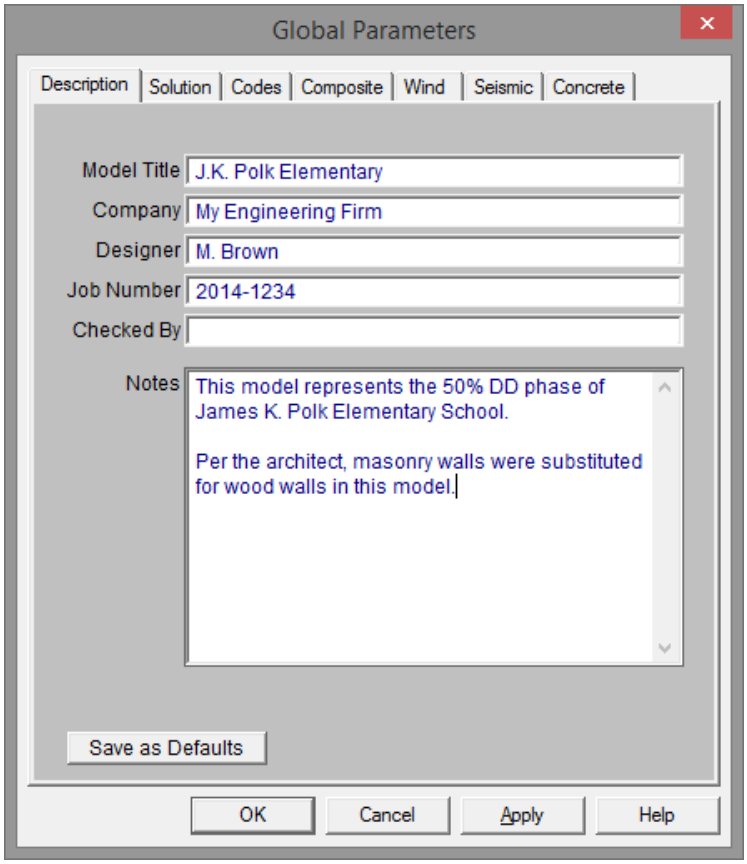
The **Model Settings** may be accessed through the menu or toolbar buttons highlighted below.



The Model Settings are settings 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 may save any of the settings in the Model Settings as the default setting, such that new models you create will begin with the settings that you saved as Defaults. To do this, simply enter the information that you want to save and click the **Save as Defaults** button.

Description

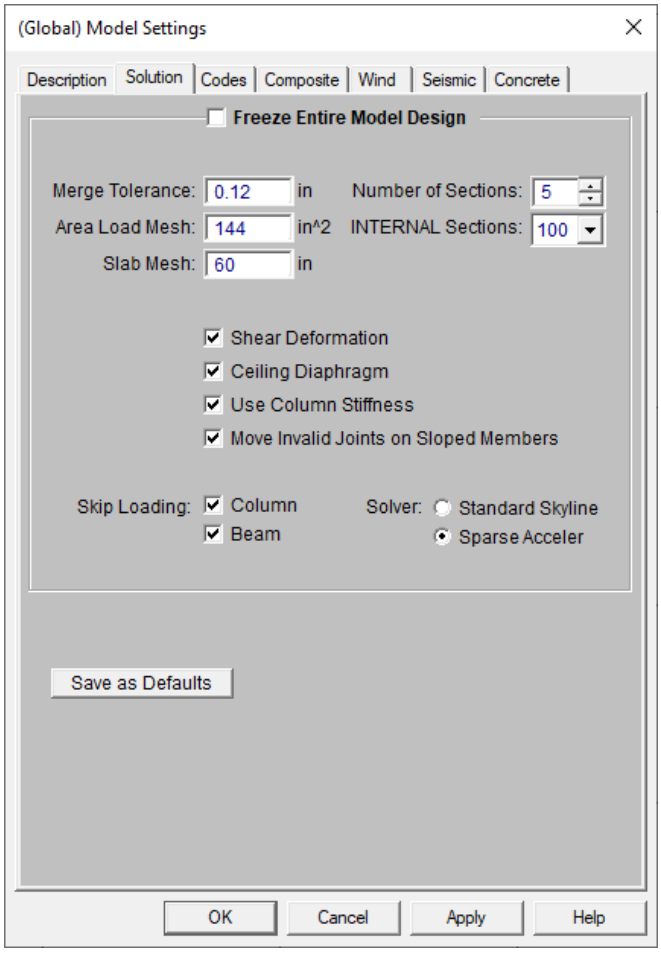


The entries under the **Description** tab are used to enter descriptive information such as a title for the particular model being defined, the name of the designer and a job number. The title may then be printed at the top of each sheet of the output, and on the graphic printouts of the model. Note that for any of these tabs, you can click on **Save as Defaults** and then your settings will be saved the next time you open the program.

The **Checked By** field is intended for the model reviewer to initial upon completion of a model review. Their initials will then be printed on each sheet of the output.

The **Notes** field is intended for information about the model that may be useful for the engineer or reviewer to refer to.

Solution



The entries under the **Solution** tab are used to control settings that affect the general solution of the model.

Number of Sections controls how many places you receive **reported** member force, stress, torsion, and deflection results. This only affects the amount of data displayed in the results spreadsheets, and has no effect on the solution of the model or the code checks. See Printing and Member Results for more information.

Number of Internal Sections controls how many places along each member the software calculates and stores results such as deflections and code checks. The member force diagrams displayed in the model view and the detail plot are also drawn from these results. Increasing this value means that the program will make more "cuts" along the beam's length, which means it is more likely to hit the theoretical maximum and minimum values for code checks.

Note:

- Number of Sections cannot exceed 20. Also, Number of Internal Sections cannot be less than twice the Number of Sections. If unacceptable values are entered for either of these fields the program will automatically reset them to acceptable values.
- 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.

When checked, the **Freeze Entire Model Design** box will disable the program's redesign capabilities and no beam or columns will be resized during a solution.

Check the **Shear Deformation** box if shear deformation considerations are to be included in the model solution. See Member Shear Deformations for more information.

The **Area Load Mesh** is used to determine the maximum size when meshing an area load and attributing the load to members. See Area Loads to learn more about this.

The **Slab Mesh** is used to define the maximum mesh size of finite elements created for slab floors utilizing RISAFloor ES. No plate generated will have a side dimension larger than what is defined here. There are minimum numbers of plates required between control points, so smaller meshed areas will occur in places with more closely spaced elements.

The **Merge Tolerance** is used as the maximum distance 2 points 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 beams.

The **Solver** to be used during solution may be selected by clicking the radial button next to **Standard Skyline** or **Sparse Accelerated**. See Solution for more information on these two options.

Note:

We have removed the **Show Factored End Reactions** check-box. This box was previously used to define whether you want your beam end reactions to show up as being factored or not (using 1.0 instead). The graphical end reactions will now be shown based on which load combinations are checked in the "Connection" column of the Load Combinations - Design tab. If you want to design your members to factored loads but want the unfactored reactions to show up graphically you would create two sets of load combinations where the factored LC's have the "Hot Rolled Steel" column checked and a set where the "Connection" column is checked.

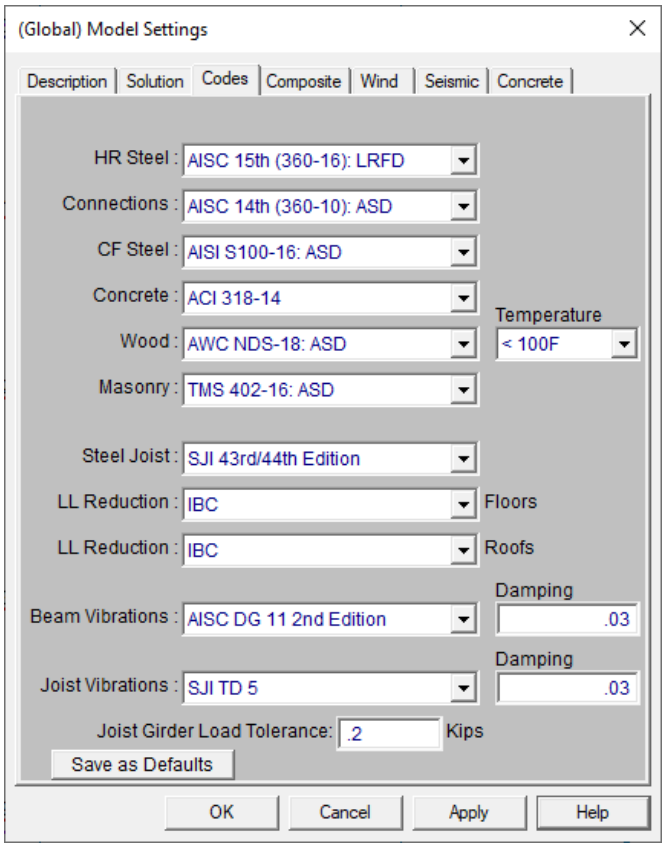
If the **Use Column Stiffness** box is not checked, columns will be viewed as simple supports with no rotational stiffness of their own. If this box is checked the column's rotational stiffness will be calculated based upon the column material, moment of inertia, orientation and story height. For more information see the Distribution of Moments section of the Column Results.

The **Move Invalid Joints on Sloped Members** is used when you are adding joints to a member after it has already been sloped. If you slope a roof level, then later add members or point loads to the sloping members, these joints are created at the base elevation of the floor. If this box is checked then, at solution time the program will move those joints up to the sloping member. This check-box is checked by default. If you have problems at solution time with your sloping members, you may consider un-checking this box for troubleshooting.

If the **Skip Loading** box for **Column** or **Beam** is checked, the program will automatically design the members for the controlling combination of unbalanced live load at each floor. When the box is unchecked for columns, the columns will only be checked for the case where gravity loads from all sides of the column act concurrently. For more information see Column Skip Loading.

The **Ceiling Diaphragm** setting is only applicable when the roof of the model is sloped. When the roof is sloped and a rigid or semi-rigid diaphragm is used, this setting must always be turned on because the program does not support sloped rigid or semi-rigid diaphragms. For more information on how this setting affects Flexible diagrams see the Diaphragms topic.

Codes



The settings under the **Codes** tab control which design code is used for code checks for each material type, as well as the codes to be used for live load reduction and floor vibration checks.

The **HR Steel** entry indicates which code is to be used for the design of hot rolled steel. For more information see Hot Rolled Steel Design.

Note:

- 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.

The **Connection** entry indicates which code is to be used for connection design. This entry is only considered if you are also using RISAConnection. See the RISAConnection Integration topic for more information.

The **CF Steel** entry indicates which steel code is to be used for the design of cold formed steel. For more information see Cold Formed Steel Design.

The **Wood** entry 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.

Note:

- 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.

The **Concrete** entry indicates which concrete code is to be used in the design of concrete members, slabs, and walls. For more information see Concrete Design.

Note:

- A significant change in the 2014 ACI code is a change to the minimum flexural reinforcement for elevated slabs. Please see ACI 318-14 Sections 7.6.1 and 8.6.1 (and Chapter 13's references to Chapter 7 and Chapter 8) for more information
- 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.

The **Masonry** entry indicates which masonry code is to be used for the design of masonry walls. For more information see Masonry Design.

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.

The option for **Beam Vibration** checks is the AISC Design Guide #11, both the 1st and 2nd Edition. When this calculation is turned on, you may adjust the **Damping ratio** used your floor system.

The **Joist Vibrations** check is based on the Steel Joist Institute Technical Digest 5. You can adjust the **Damping ratio** used in these calculations.

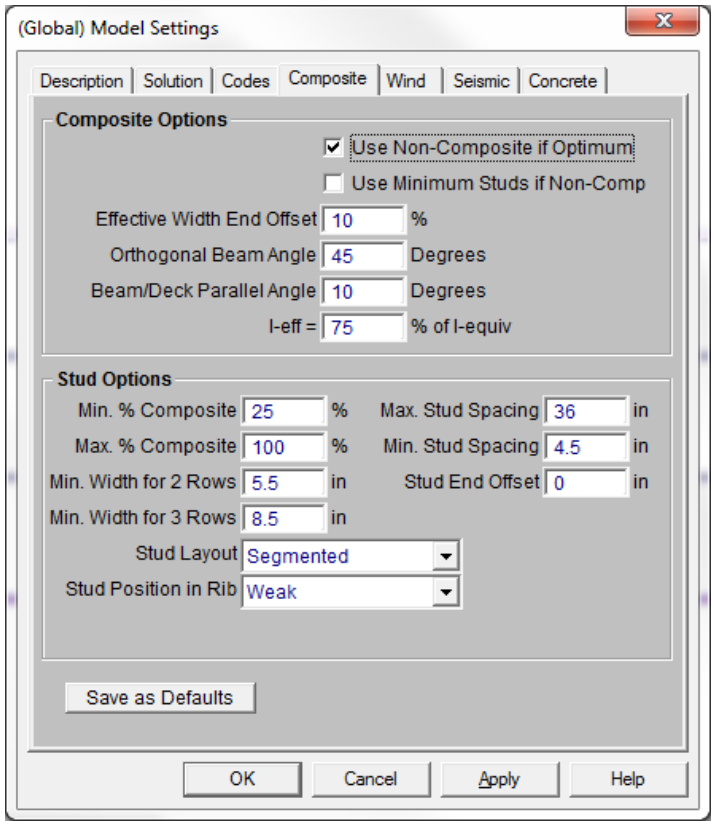
The **Live Load Reduction** criteria entry indicates which code to use for live load reduction. Floor live load reduction applies to the Live Load and Live Load Special load categories. The **Roof** option applies to Roof Live Loads.If you do not want live loads to be reduced you may specify " none".

When you have multiple point loads framing into a joist-girder, the **Joist Girder Load Tolerance** sets the maximum load variation to be allowed when specifying that standard joist girder call-out. When the load variation exceeds this, the call-out will switch to a more complex format for a custom joist girder call-out.

If you do not want a particular code check to be performed you may specify "None" in the drop down list of available codes.

Composite

The options on the **Composite** tab are specifically related to composite steel beam design.



When the **Use Non-Composite if Optimum** checkbox is unchecked, all beams which are marked as Composite and which have Composite Deck on them are forced to be composite, meeting the Min. Percent Composite specified on this same tab in the Model Settings. This can result in beams or stud layouts that are over-conservative with respect to strength and serviceability requirements. When this box is checked, the program will design the beam as non-composite if the non-composite beam size is more economical than the composite beam size which meets the Min. Percent Composite.

The **Use Minimum Studs if Non-Comp** box is only available if the Use Non-Composite if Optimum box is checked. When this box is unchecked, non-composite beams will not be detailed with any studs. When this box is checked, a small number of studs are detailed on the beam, equal to the beam length divided by the Max Stud Spacing. These studs serve no purpose for strength or serviceability, and do not affect any capacities or code checks in the program.

The **Effective Width End Offset** specifies the percent of the beam ends to ignore in determining the effective width. See Effective Widths for more information.

The **Orthogonal Beam Angle** gives the minimum angle that two beams can make with each other and still be considered perpendicular. Perpendicular beams will not encroach on each other's effective widths. See Effective Widths for more information.

The **Beam / Deck Parallel Angle** gives the maximum angle between a beam and the deck for which they can be considered parallel. It is recommended to set this value greater than zero, to allow for minor tolerances within the program.

The **I-Effective = %I-Equivalent** factor allows the user to make the effective moment of inertia a certain percentage of the equivalent / elastic value based on elastic theory. Commentary to the AISC specification would suggest appropriate values in the range of 70% to 85%. Refer to the Hot Rolled Steel - Composite Beam Design topic for more details.

Min. Percent Composite specifies the minimum percent composite action allowed for composite beams.

Max. Percent Composite specifies the maximum percent composite action allowed for composite beams.

The **Min Stud Spacing** and **Max Stud Spacing** control the allowable stud spacing.

The **Stud End Offset** is used to specify the length of the beam ends that may be unsuitable for stud placement due to construction or connection issues.

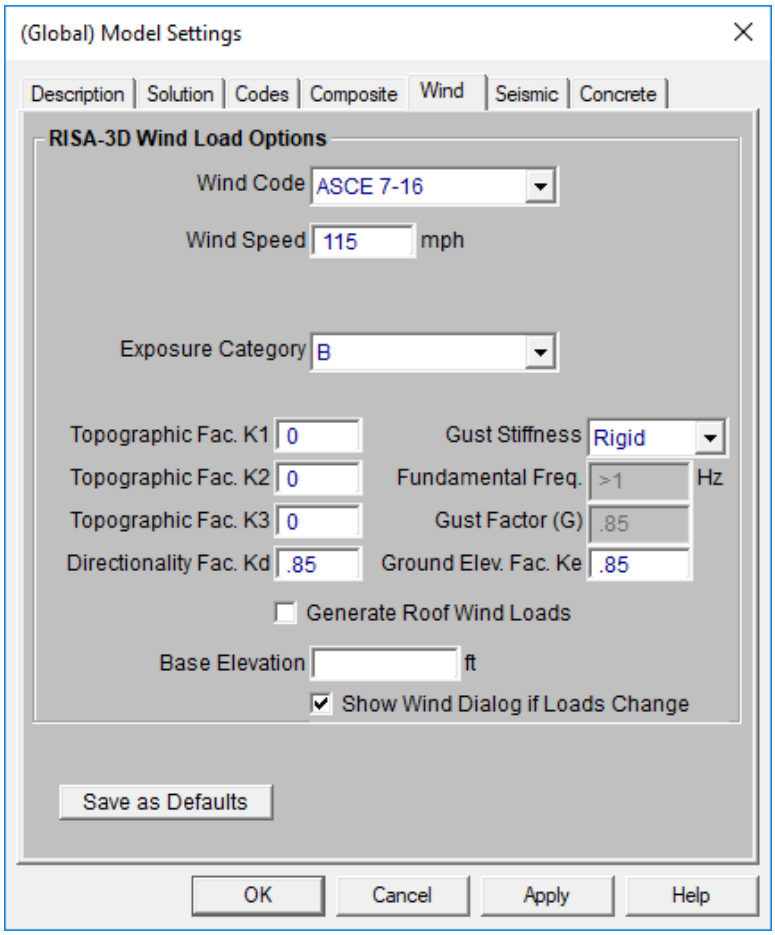
The **Minimum Width for 2 Rows** and **Minimum Width for 3 Rows** specify the flange widths that can accommodate multiple rows of studs.

The **Stud Layout** option gives you the ability to choose between uniform stud spacing and segmented spacing.

The **Stud Position in Rib** option gives you the ability to choose between strong and weak position of studs (Rp, for the AISC code). For the Canadian code this give you the option to choose between studs centered in the rib deck and studs off center in the rib deck. Refer to the Studs section of the Composite Beam Design topic for more details.

Wind

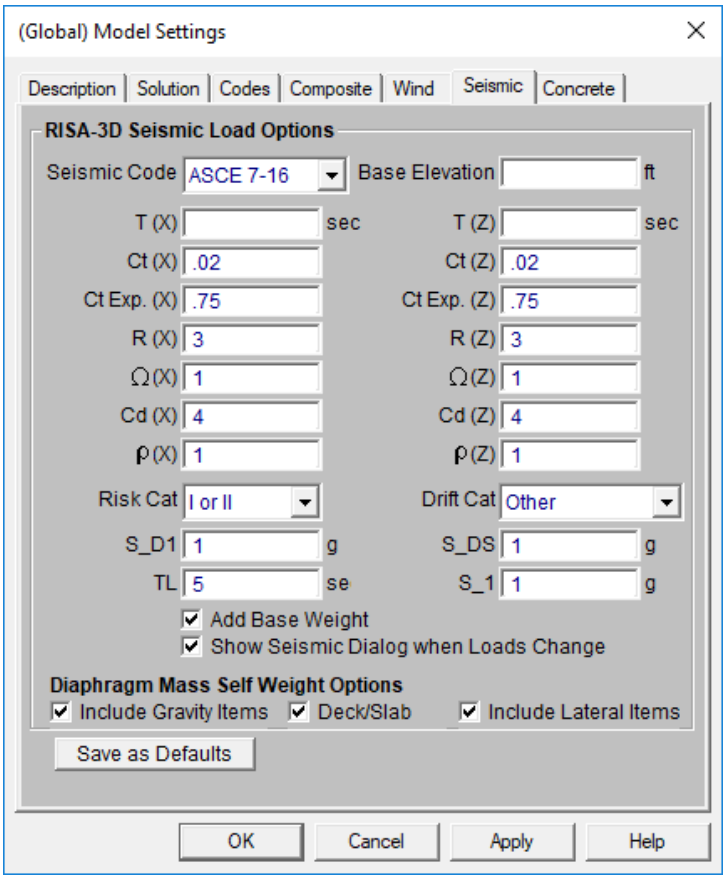
The options on the **Wind** tab present the Model Settings that are specifically related to calculation of code prescribed Wind Loads. This information will be used by RISA-3D to automatically generate the lateral loading on your structure.



See the Wind Load topic for more information on these inputs.

Seismic

The options on the **Seismic** tab present the Model 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.



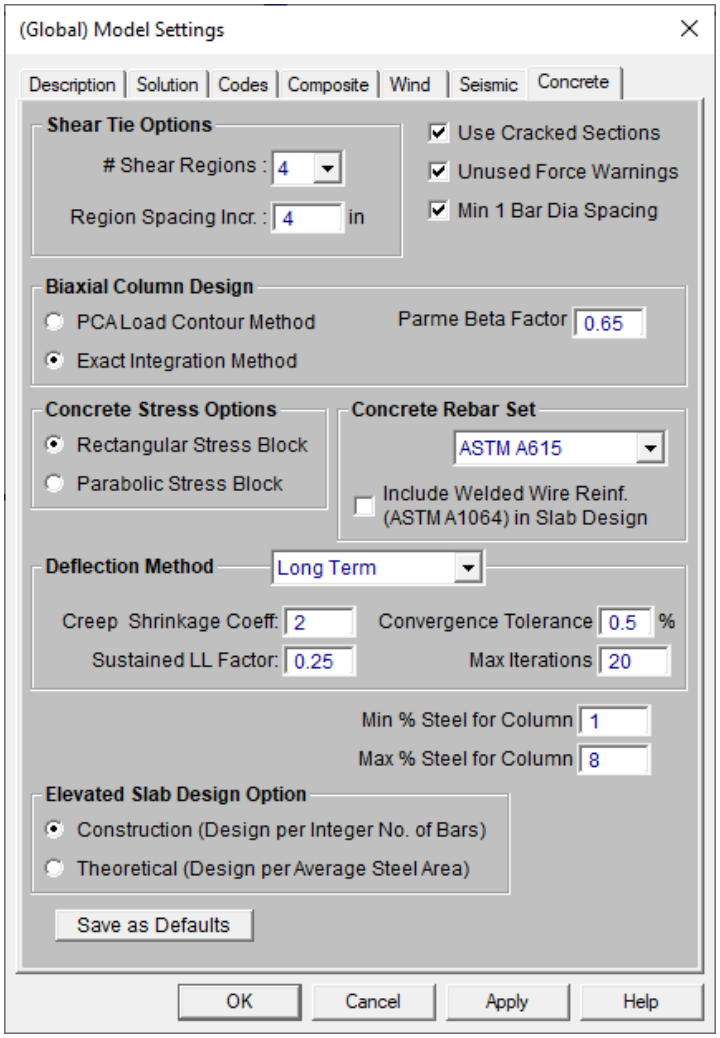
Most of the input fields on this tab are discussed in the Seismic Load topic. Below are a few inputs which are specific to this tab. See the section Drift Calculations to see how Rho, Cd, Drift Category and Risk Category affect the calculation of story drift results.

(Ω) is the **Overstrength Factor**. This can be included in the seismic load combinations generated by the LC Generator.

The **Redundancy Factor** (ρ) is based on the extent of redundancy in your structure. This can be included in the seismic load combinations generated by the LC Generator.

The **Diaphragm Mass Self Weight Options** allows the user to discount certain portions of the dead weight of the structure for seismic mass calculations. By default all the checkboxes are checked. This may be used if you want to manually add all of the seismic mass to the structure for a specific application.

Concrete



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

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

The **Biaxial Column Design Options** controls which method will be used to determine the biaxial column capacity. The options are the **PCA Load Contour Method** and the **Exact Integration Method**. The **Parme Beta Factor** is used to approximate the column's 3D interaction surface when using the PCA Load Contour Method. See Biaxial Bending of Columns for more information.

The **Concrete Stress Options** allow you to choose what type of stress block to consider in your analysis. The options are the constant **Rectangular Stress Block** and the **Parabolic Stress Block**. See Parabolic vs. Rectangular Stress Blocks for more information.

Check the **Used Cracked Sections** box if you want to modify the member and wall stiffnesses by the **Icr Factor** as described in both the Concrete - Design and Wall Panels topics.

Check the **Unused Force Warnings** check box to see force warnings on the detail report.

Checking the **Minimum 1 Bar Dia Spacing** box will allow a minimum spacing of one bar diameter between parallel bars. Otherwise, RISA will default to a two bar diameter or one inch clear spacing, whichever is greater, to allow for lap splices and continue to maintain adequate spacing between parallel bars.

The **Concrete Rebar Set** allows you to choose from the standard ASTM A615 (imperial), ASTM A615M ("hard" metric, i.e. #8M is an 8mm bar), prENV 10080 (Euro), CSA G30.18 (Canadian), and AS/NZS 4671:2001 (Australian/New Zealand) reinforcement standards.

The **Include Welded Wire Reinf. (ASTMA1064) in Slab Design** allows you to design a slab using Welded Wire Reinforcement.

Note:

- Users shall choose the correct material in the Materials Spreadsheet to account for ASTM A1064 steel.
- Welded wire reinforcement is only available for Elevated Slabs.
- Only ASTM A1064 deformed bar sizes are available.

Deflection Method options allows you to choose from None, **I Cracked/Elastic** or **Long Term**. This only applies to a concrete slab floor.

- None: This will use uncracked moment of interia for the slab.
- Use Cracked Slab: This option will modify the moment of inertia of the slab for the Service Load Combination. I if you want to modify the concrete slab stiffnesses by the **Icr Factor** as described in the Elevated Slab - Modeling topic and also Service Load Combinations for further information.
- Long Term: This option will turn on the iterative anlalysis that accounts for long term deflection including creep and shrinkage. See Long Term Deflection for further information.

The **Minimum % Steel for Columns** allows you to choose the minimum percentage of reinforcement steel to be used in a concrete column section. The percentage entered will be multiplied by the gross area of the column section to determine the minimum amount of reinforcement required in each column. It should be noted that the minimum percentage allowed by ACI 318-14 Section 10.6.1.1 (ACI 318-11 Section 10.9.1) is 1%.

The **Maximum % Steel for Columns** allows you to choose the maximum percentage of reinforcement steel to be used in a concrete column section. The percentage entered will be multiplied by 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%.

The **Elevated Slab Design Option** is only visible when you have elevated slabs. This option offers a choice between two methods, namely the Construction method and the Theoretical method, for the design of the slab rebar. The Construction method, which is the default method employed by the program for a significant period, determines the design of the slab based on an integer number of rebar. This method operates under the assumption that if the strip width divided by rebar spacing is not an integer, an additional rebar will be provided to guarantee the presence of a rebar at each end of the design strip. On the other hand, the Theoretical method is a new method added later that enables the program to calculate the rebar area based on the average value, where the rebar area equals As (single rebar) multiplied by the Strip Width divided by Spacing. However, it is crucial to note that this method may not always result in an integer number of rebar so unlike the Construction method, the results for Theoretical method will have no info of rebar amount.

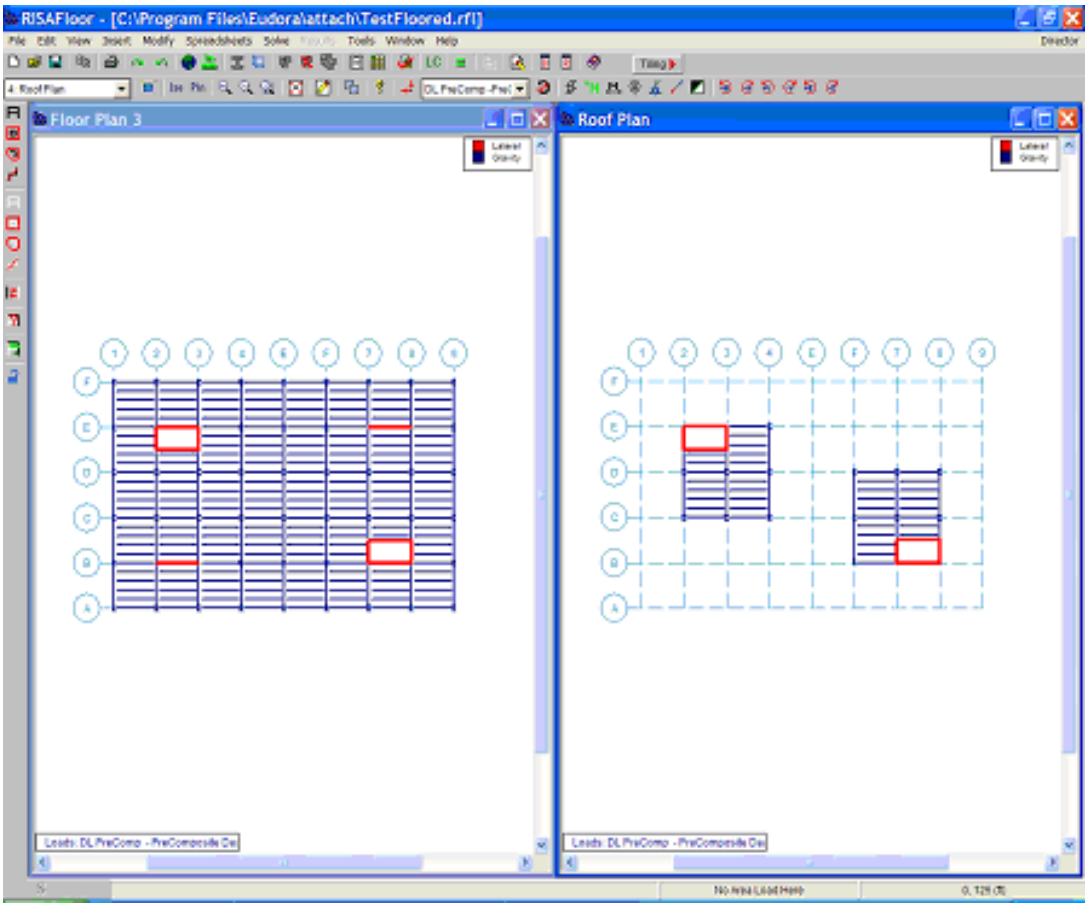
Note:

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.

Graphic Display

RISAFloor's robust graphics make building and understanding your model much easier. You may have numerous independent views open and active at the same time. Fast redraw times, responsive panning, true to scale rendering, and graphic editing tools all help you build and understand your model. You may draw the model and specify loads graphically. These items may be modified graphically as well. Verification of your model is made simple with true-to-scale rendering and color coded code checks.

Multiple Windows



You may have multiple views of the model open at the same time, each with it's own characteristics. For example you can simultaneously view and work with members in one window and plates in another. You can have multiple spreadsheets open as well.

To Open a New View or Clone an Existing View

- To open a new view of the model click the **New View** button. Alternately, you can open a new view that is an exact replica of the current view by clicking the **Clone View** button.

Note

- With the model view active you may then use the **Window Toolbar** to adjust the view of the model.

Controlling the Model View

The two main tools to help you manipulate the views of the model are the **Window Toolbar** and the **Model Display Options**.

The **Window Toolbar,** discussed in this section, is the second toolbar from the top when a model view is active. You may use these buttons to rotate the model, zoom in and out, render the model, and display common items such as point labels and loads.

The **Model Display Options Dialog** (See Model Display Options for more information) is opened by the first toolbar button and provides various options not otherwise available on the toolbar. It allows you to specify how to plot points, beams, slabs, and loads and the information that is to accompany them. The tabs across the top of the dialog break things up into basic elements to help you find the options.

The **Snapshot** tool is a great way to add images of model views and detail reports in your report printing. See Printing for more information.

Adjusting the Viewpoint

The remaining buttons on the **Window Toolbar** help you change the viewpoint by rotating, snapping, and zooming the view. You may use the scroll bars and the mouse wheel (if you have one) to pan the view.

To Rotate a View

- When holding down the Shift key, the mouse wheel button can be used to dynamically rotate the structure. See Application Interface - Dynamic Controls for more information.
- To rotate a view of the model about a global axis, click any of the **Rotate** buttons on the **Window Toolbar:**
- To snap to an isometric or planar view of the model click any of the **Snap** buttons on the **Window Toolbar:** Iso Pln

To Zoom In and Out

- Use the wheel button on the mouse to dynamically zoom in or out the model view. Rotating the wheel forward zooms in and rotating the wheel backward zooms out.
- To zoom in towards the center of the model view click the **Zoom In** button or press the **+** key.
- To zoom out from the center of the model view click the **Zoom Out** button or press the **-** key.
- To zoom in on a specific area of the model view click the **Zoom Box** button and then draw a box around the zoom area by clicking and dragging with the left mouse button.
- To snap the model to a full model view click the **Full Model View** button.
- The Function Keys, F3 and F4, are for the Zoom Previous and Zoom Next functions respectively. The system holds a total of 10 zoom-states. The F3 or F4 keystroke moves the active pointer forward or backward on the list of zoom states. If multiple model views are present, each window will have its own zoom list.

To Pan the View

- Clicking and holding the mouse wheel button triggers a dynamic pan and allows the user to drag the current model view to the limits of scroll bars.
- If you do not have a mouse wheel you may drag the scroll bars and click the arrows to pan the view.

Enabling Graphic Editing

The **Graphic Editing Toolbar** or **Drawing Toolbar** may be turned on to graphically edit the model at any time and it may be turned off to provide a larger view. See Graphic Editing for more information.

To Toggle the Graphic Editing Toolbar

- To toggle the graphic editing toolbar on or off, click the button on the **Window Toolbar**.

Saving Views

Click the button to save and recall views. If you have a view that you like to work with or have created a view that took some time to set up, save it with this feature. All of the Model Display Options are saved with the model for later recall. See Saving and Retrieving Model Views for more information.

Cloning Views

You can clone a view. This allows you to preserve the original view and modify the clone. This is useful in achieving two views that have similarities in some way.

To Clone the Model View

- Click the **Clone View** button on the **Window Toolbar.**

Displaying Loads

Use these buttons DL-Dead Load to control the display of loads. You may display an entire load category or load combination. The first button turns the display of loads on and off in the current view. The pull down list controls which category or combination is shown in the view. Click the last button to choose between plotting load categories or load combinations. More control over the display of loads is provided in the **Model Display Options Dialog** discussed in the next section.

Toggle Buttons

The last buttons on the **Window Toolbar** are toggles that plot information in the current view.

Click to toggle model rendering. See Model Display Options for more information on rendering.

Click to toggle the point labels.

Click to toggle the beam labels on and off. If rendering is on, beam labels will not be visible.

Click to toggle the display of end releases.

Click to toggle deck assignments.

Click to toggle area load types.

Click to toggle the display of the project grid.

Click to use the distance tool.

Click to use the Area Load Query tool.

Click to toggle the display of the concrete slab reinforcement bars.

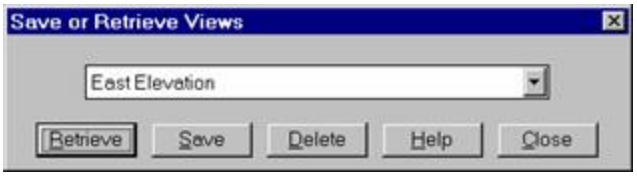
Click to toggle display of out-of-plane moment and shear.

Viewing Part of the Model

You may want to view only part of your model, especially if it is a big one. You may use the selection tools to give you the view that you want. You can graphically unselect parts of the model that you don't wish to see, or you can use a range of coordinates or other criteria to specify what to view. See Graphic Selection for more information.

Saving and Retrieving Model Views

You can save and recall views for a model. Saved views are model dependent so any views you save will stay with the model. A saved view includes information such as the current view angle, zoom state, pan location, plot option settings, etc. Saved views do NOT include the selection state for the model. You can save selection states separately. See Saving Selections for more information.



To Save the Model View

- Click the **Save View** button on the **Window Toolbar.**
- Click the **Save** button and enter a name and then click **OK.**

To Retrieve a Saved Model View

- Click the **Save View** button on the **Window Toolbar.**
- From the drop down list, choose the view you wish to recall and then click the **Retrieve** button.

To Delete a Saved Model View

- Click the **Save View** button on the **Window Toolbar.**
- From the drop down list, choose the view you wish to delete and then click the **Delete** button.

Model Display Options

The Window toolbar (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 are located in the **Model Display Options** dialog which may be accessed by clicking the button on the Window toolbar or pressing the F2 function key.

The options are organized into groups of items. Access each group by clicking on the tabs along the top of the dialog box. Some of the options are mutually exclusive and others are conditional. Typically, only one option in a vertical row may be chosen at a time and indented options will not be applied if the heading option is not also applied.

Each of the groups presents the option to turn off those particular items in the view. This is not the same as unselecting the items. Turning items on and off is independent from selecting and unselecting. For example if you make a selection of beams and then turn off beams all together, the selection still stands for any modification applied to beams.

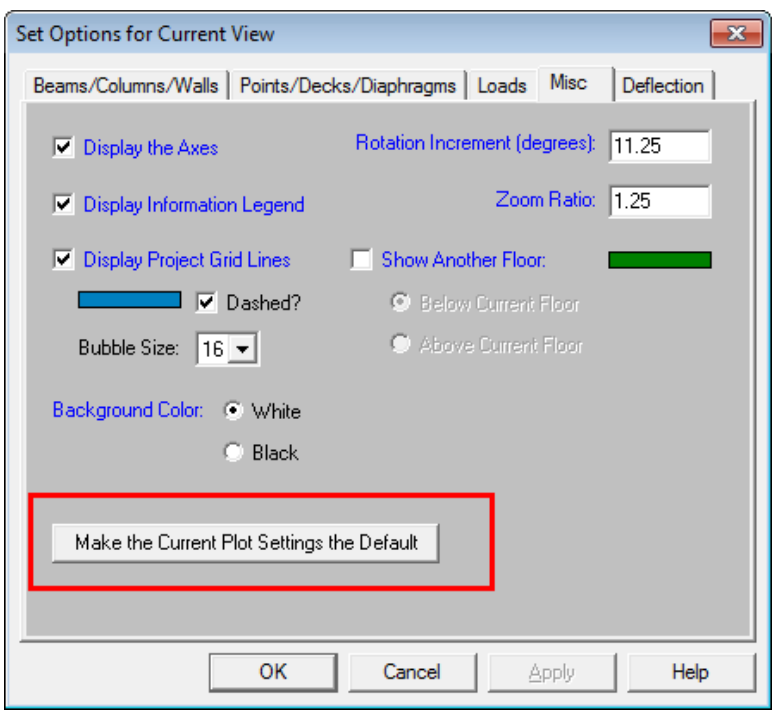
Defaults

You may make any Plot Option settings the default start-up settings. To do this, go through all the Model Display Options tabs and set all the options to what you want as the start-up default. Once that is done, select the **Misc** tab and press the button labeled **Make Current Plot Settings the Default**.

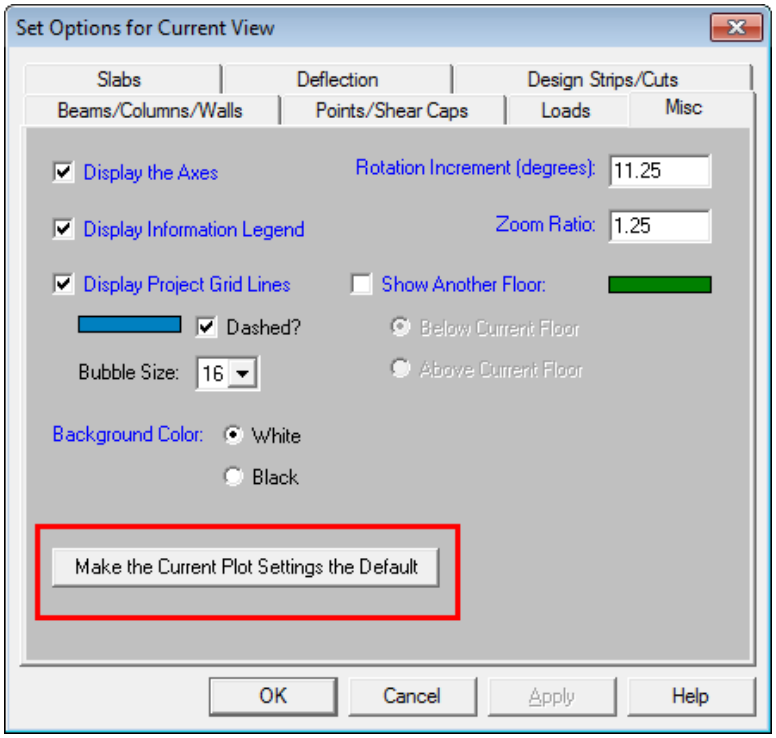
Note:

- Remember, this button applies to ALL tabs in the Model Display Options dialog.

Beam Supported Floors:



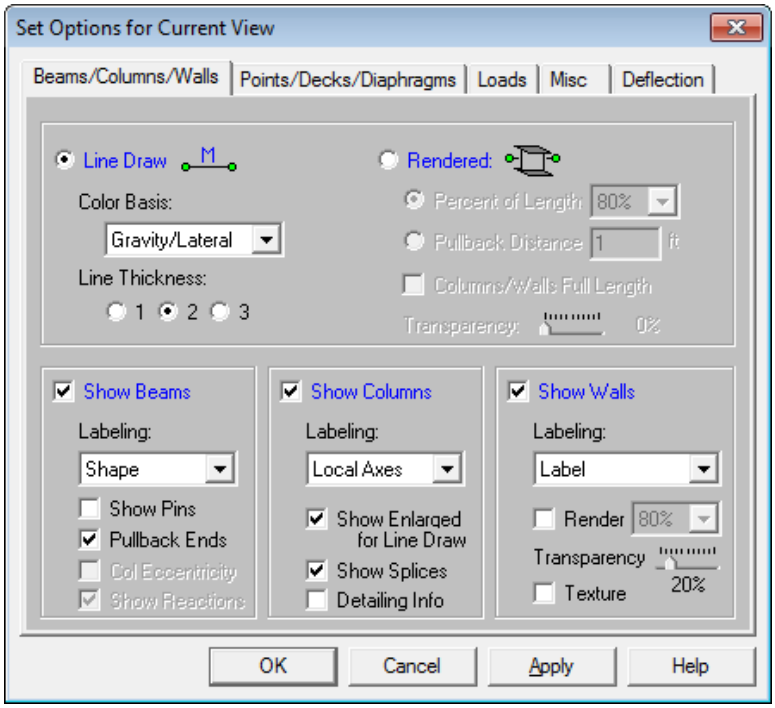
Concrete Slab Floors:



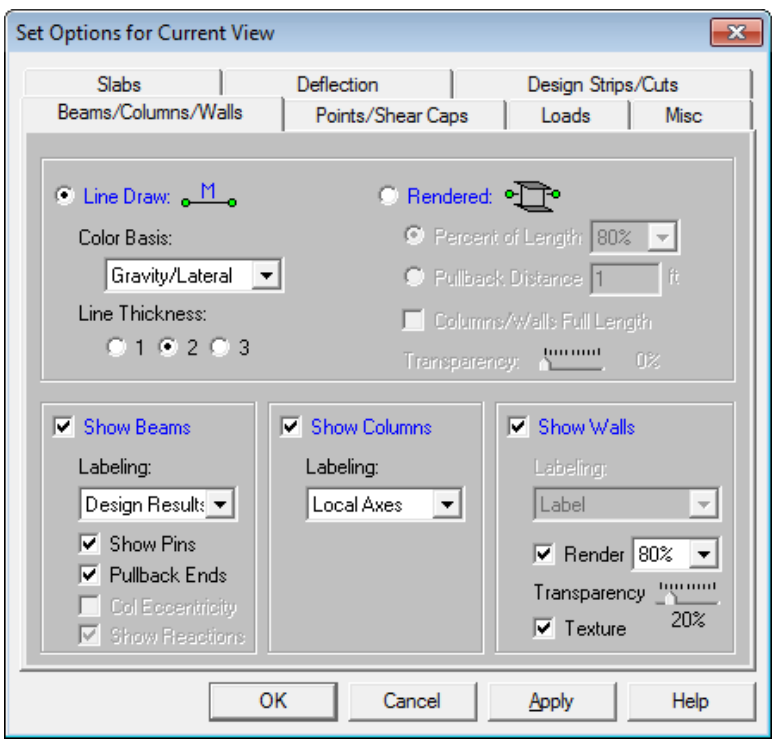
Beams/Columns/Walls

Access the Model Display Options for **Beams/Columns/Walls** by clicking **Model Display Options** dialog button on the Window toolbar and selecting the **Beams/Columns/Walls** tab shown below.

Beam Supported Floors:



Concrete Slab Floors:



You may specify whether or not to show beams, columns and/or walls by checking or un-checking their proper box. If shown, you can also have them drawn as lines or as an actual rendered shape. For a line draw, you can specify the color basis of the line and the line thickness.

If the **Line Draw** option is selected, you may include textual information alongside each beam, column, and/or wall. The **Labeling** drop down list provides the choices you have as to what information will be displayed. For beams you can also display pinned end conditions and you can pullback the line drawn from the end point. For the columns you have the ability to show the column enlarged for line draw. None of these options are available if the **Rendered** display option is selected.

Color Basis draws the beams using various colors to represent particular information such as function (gravity or lateral), material set, etc. The **Color Basis** drop down list provides the choices you have as to how the colors are to be assigned to each member. If a solution has been performed you may color the members by the bending check, shear check, or deflection check.

If the **Rendered** option is selected, true to scale representations of the members will be drawn. Selecting the **Percent of Length** option and a value from the drop down list will render only that percentage of the length of each member and the left over portions at each end will be drawn as lines. Entering a percentage of '100' will render the full length of all members. Selecting the **Pullback Distance** option and entering a value in the text box will render each member with the specified length at each end drawn as lines. Entering a length of '0' will render the full length of all members.

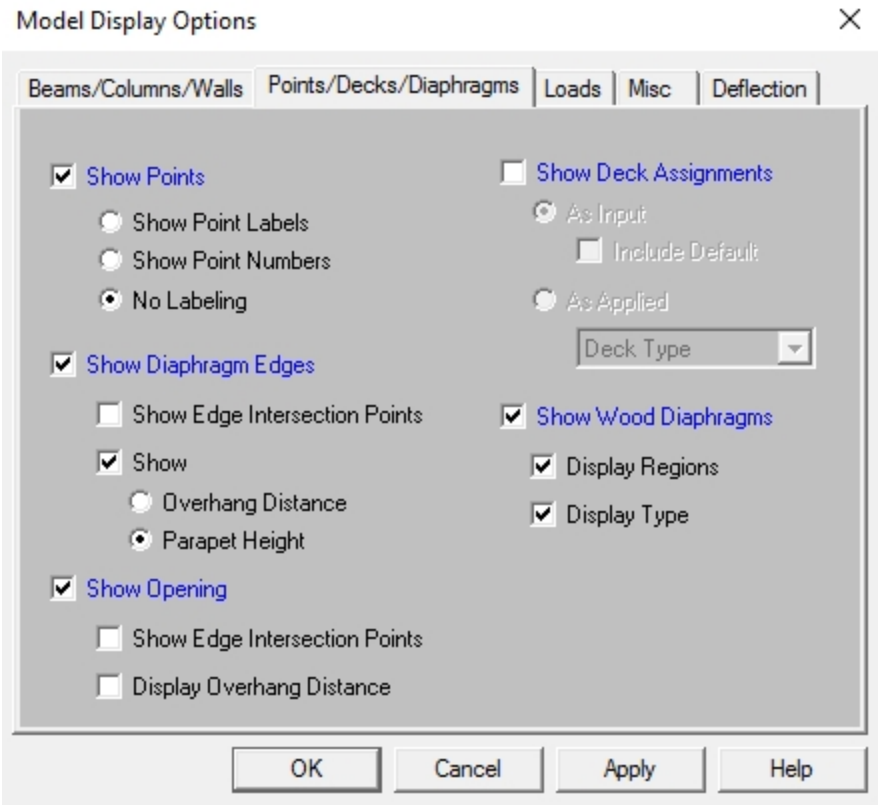
The rendered option is very useful for verifying member orientations and modifying the rendering options can provide a better understanding of intersection areas and a better view of the member cross-section.

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.

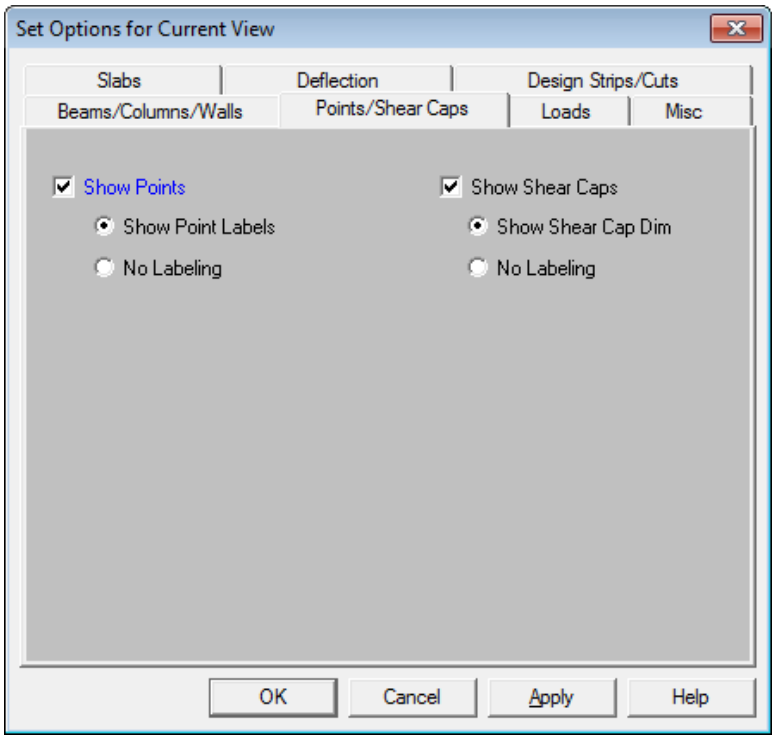
Points/Decks/Diaphragms or Points/Shear Caps

Access the graphical Model Display Options for points, decks, and diaphragms (beam supported floors only) or points and shear caps (slab supported floors only) by clicking the **Model Display Options** button on the Window toolbar and selecting the appropriate tab shown below:

Beam Supported Floors:



Concrete Slab Floors:



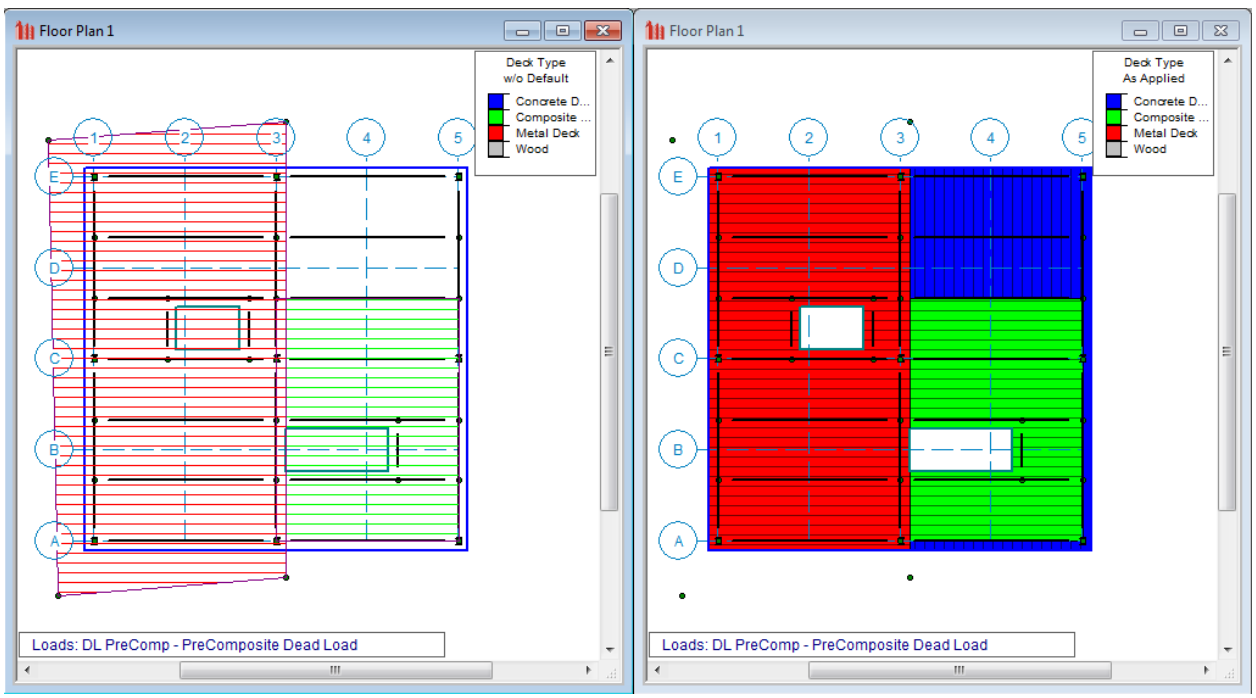
Checking the **Show Points** box will turn on the display of points in the current model view. You may include the point labels, point numbers, or turn off labeling by checking the corresponding radio buttons.

Beam Supported Floor Options:

Checking the **Show Diaphragm Edges** and/or **Show Openings** box will turn on the display of diaphragm and/or opening edges. The display of edge intersection points can be turned on by checking the **Show Edge Intersection Points** box. The diaphragm/opening overhang dimensions can be displayed by checking the **Display Overhang Distance** box. The dimension shown in the model view is the distance from the edge of the diaphragm edge or opening to the center-line of supporting beam or wall.

Checking the **Show Deck Assignments** box will turn on the display of deck/slab assignments in the current model view. Assignments may be shown '**As Input**' or '**As Applied**' by clicking the corresponding radial button.

- The **As Input** option shows the actual polygons as they were drawn in the model.
- The **As Applied** option shows how the deck/diaphragm edge is internally interpreted by the program and subsequently how it will be applied to the supporting members. Openings are best viewed with this option as can be seen in the figure below.



The image on the left above shows the deck as drawn in by the user. The approximate orientation of the deck is shown with the hatching lines. The color of the hatching lines corresponds to the deck properties as defined in the **Deck Definitions** spreadsheet. The legend at the top right of the screen shows which decks correspond to which colors.

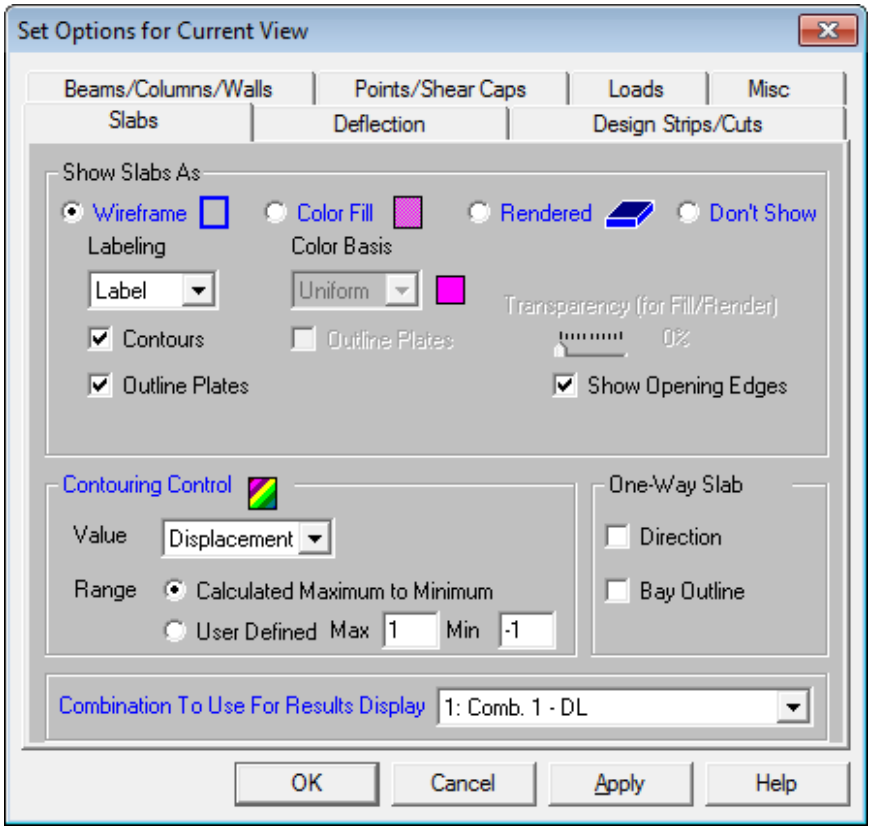
The image on the right shows the deck as it was applied *within the diaphragm edges* by the program. This shows how the deck is actually applied to the floor system and where it is not applied- such as outside the diaphragm edge or within openings. The color corresponds to the deck properties as defined in the **Deck Definitions** spreadsheet. The legend at the top right of the screen shows which decks correspond to which colors.

Concrete Slab Floor Options:

Checking the **Show Shear Caps** box will turn on the display of column shear caps in the current model view. The shear cap dimension can be turned on by checking the **Show Shear Cap Dim** box. Note: this is only available for slab supported floors.

Slabs

Access the graphical Model Display Options for slabs by clicking the **Model Display Options** button on the Window toolbar and selecting the **Slabs** tab shown below.



Note: This tab is only available for concrete slab supported floors.

The **Show Slabs As...** section of this tab allows you to specify that slabs are to be drawn as wireframe, color filled, or rendered elements, or that they are not to be shown at all.

When the **Wireframe** option is selected, you may include information on the slabs when plotted. The **Labeling** drop down list provides the choices you have as to what information will be displayed on each slab, such as the slab label, material, thickness, or the slabnumber. The wireframe option also allows for the display of line contours representing the force or stress results for the slabs. Each color line represents a specific value. You must choose which load combination you want to view in the **Combination To Use For Results Display** drop down list.

When the **Color Filled** option is selected, slabs are plotted with different colors that are mapped in a key shown in the upper right corner of the model view. You may color the plates within a slab by material set or specify a uniform color for all plates. You also have the option to plot the stress and force results using filled color contours where each fill color represents a range of values. The color filled plates themselves may be presented with varying **Transparency** with 100% transparency meaning completely see through and 0% indicating completely solid. You must choose which combination you want to view.

Note:

- The transparency setting does not apply when "**Contour**" is selected as the color basis from the drop down list.

The **Color Contours** are plotted with a global smoothing algorithm that allows the contour to vary across the plates within a slab. Because of this, the plotted contour results will differ slightly from the tabulated results in the

spreadsheets. This effect will be heightened in regions of a high rate of change such as loads or boundaries. The contours may be drawn as either lines or as color filled areas.

The **Contouring Control** section determines what force or stress result is to be contoured and how the contour colors are to be assigned. The **Value** drop down list is where you select the specific result to be contoured (QZ, MXZ, etc.). The **Range** controls determine how the contour colors are assigned. You can either contour the full range of the results by choosing **Calculated Maximum to Minimum**, or, if you are only interested in a specific range of values, you can choose **User Defined** and enter your own max and min values.

When the **Rendered** option is selected, true to scale representations of the slabs will be drawn with thickness.

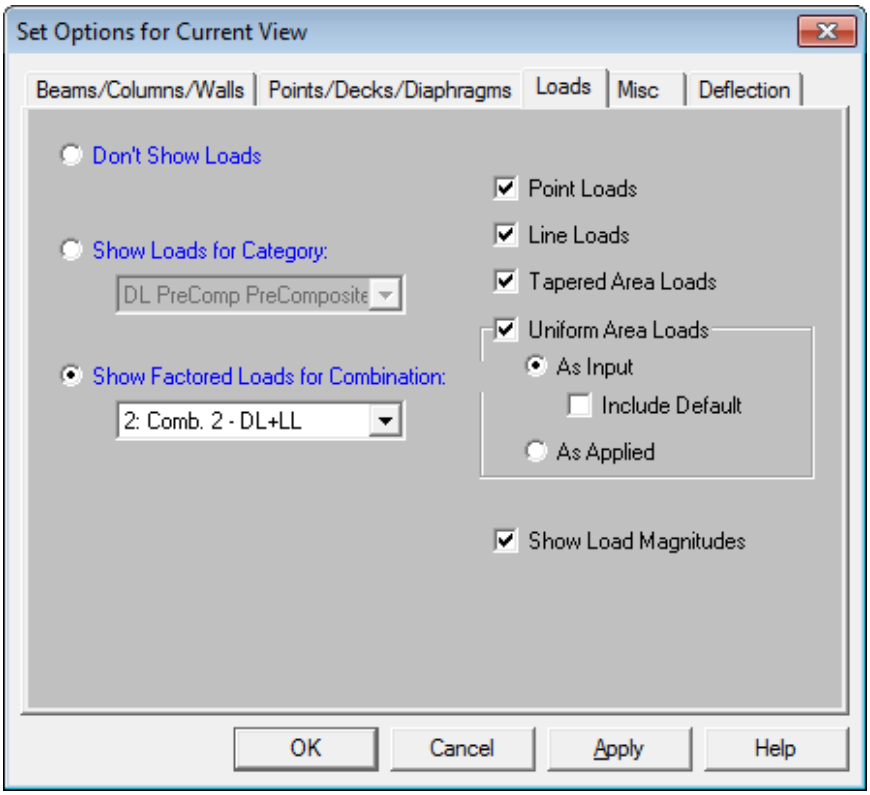
The **Transparency** setting described above for Color Filled plates also applies to Rendered plates.

The **One-Way Slab** check boxes control the display of the one way slab. The **Direction** check box toggles on and off the display of a direction arrow. Selecting the **Bay Outline** check box will turn on a highlighted view of the bay area over which the one way load is distributed.

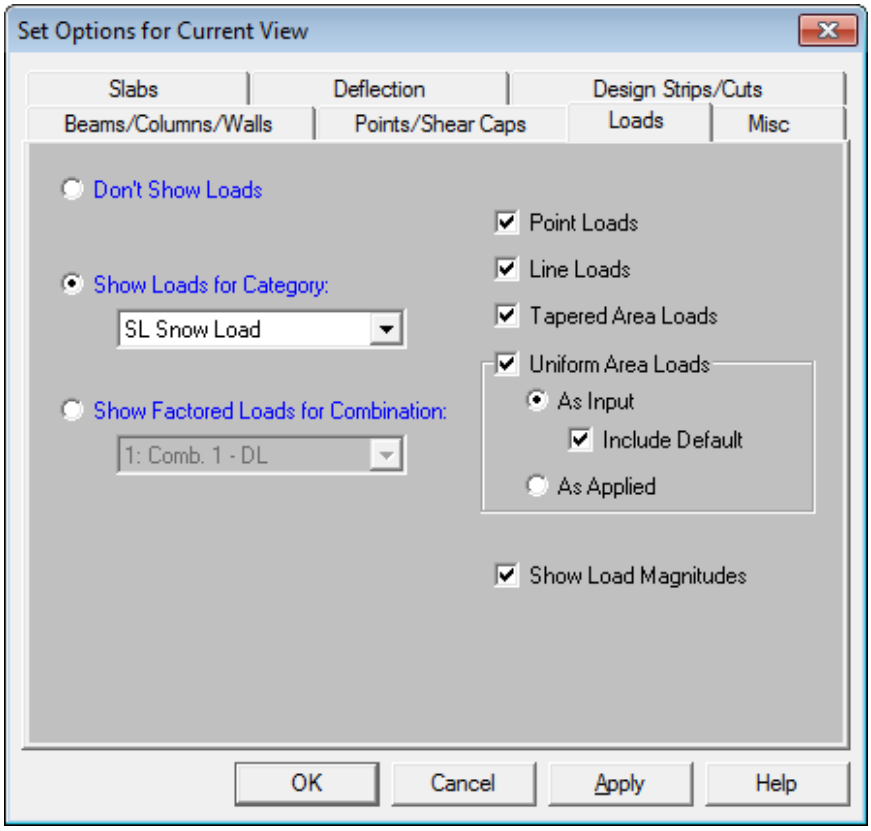
Loads

Access the graphical Model Display Options for loads by clicking the **Model Display Options** button on the Window toolbar and selecting the **Loads** tab shown below.

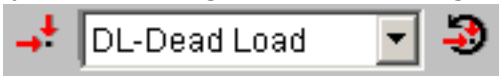
Beam Supported Floors:



Concrete Slab Floors:



You may specify that the loads be drawn as Load Categories or Load Combinations by selecting the corresponding radio button. The load types to be displayed can be selected by checking the corresponding boxes.

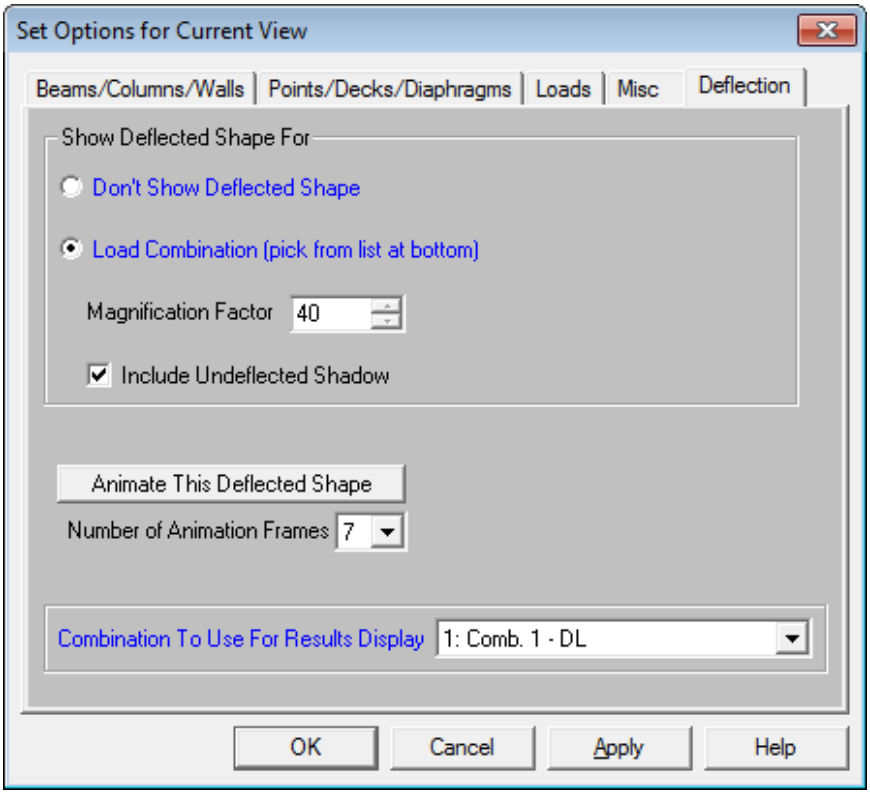
Once you set the Model Display Options for loads, you may then run through all of the load categories or load combinations by choosing them from the drop down list 

on the Window toolbar.

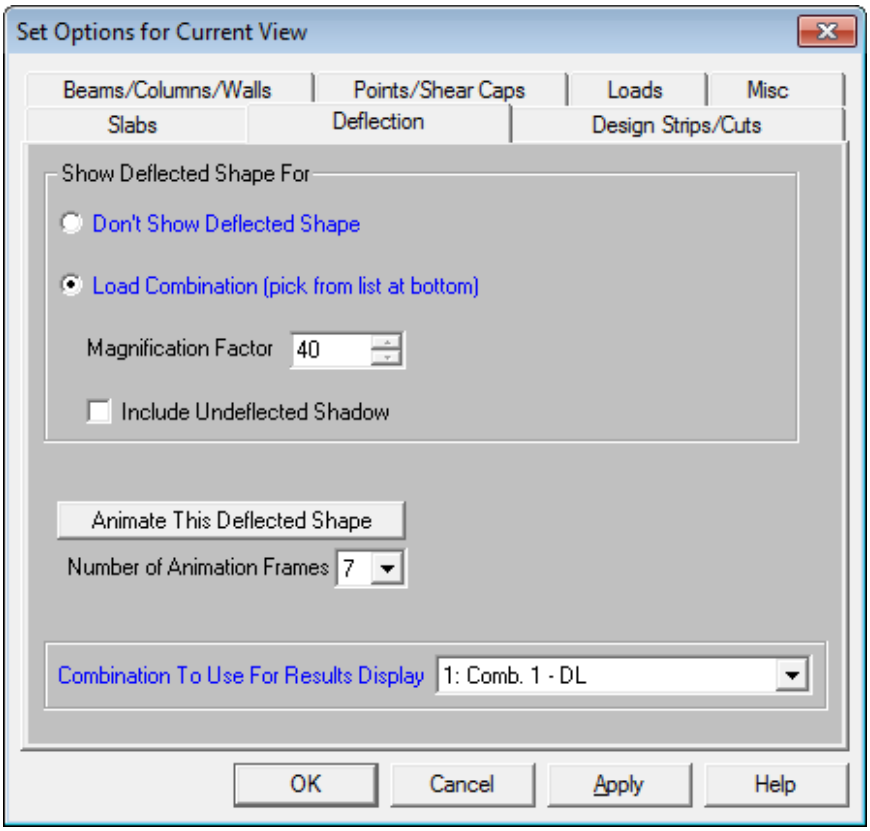
Deflection

Access the Model Display Options for deflections by clicking the **Model Display Options** button on the Window toolbar and selecting the **Deflection** tab shown below.

Beam Supported Floors:



Concrete Slab Floors:



In the **Show Deflected Shape For...** section of the tab you may specify that a deflected shape be drawn in the current model view based on a load combination. You can control the magnification of the deflections with the **Magnification Factor** text box and specify whether an undeflected shadow is to be shown by checking the **Include Undeflected Shadow** box. You must choose which combination you want to view in the **Combination To Use For Results Display** drop down list.

To animate a particular deflected shape, first select the deflected shape (as described above) and then simply click the **Animate This Deflected Shape** button. A new model view will be created with the animation. The **Number of Animation Frames** drop down list allows you to specify how many frames are used in the animation to move from the undeflected state of the model to the fully deflected state.

In the model view of the animated deflected shape, the **Capture** feature allows users to save video file format of the animated deflected shape. To use this feature, adjust to the desired number of frames and click on the video button. The user will be prompted to save the video file.

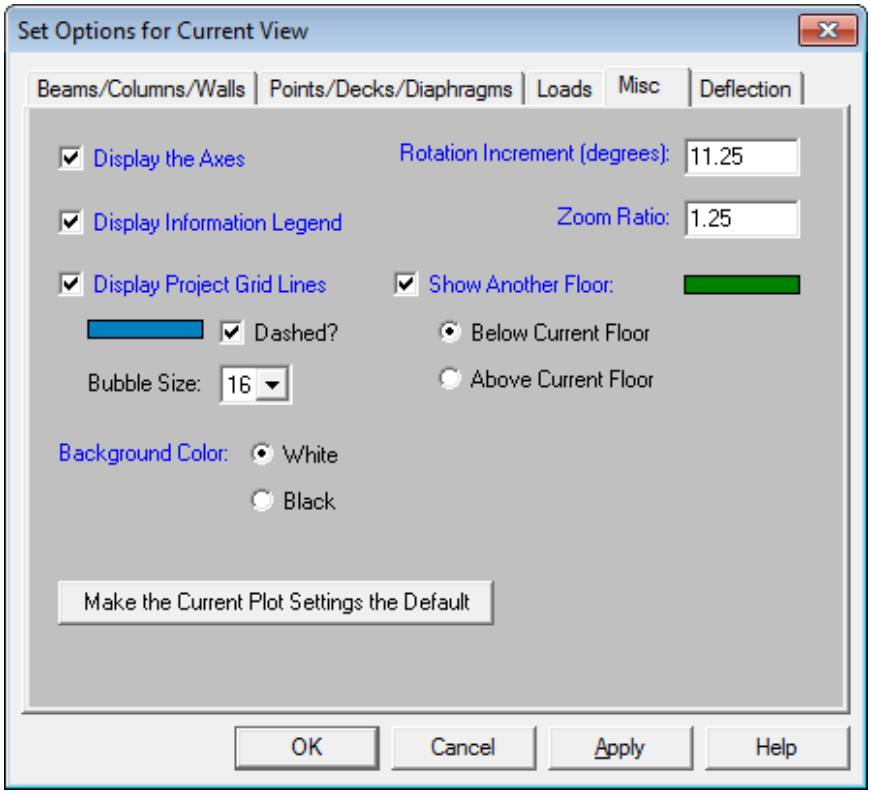
Note:

- The animation that is created is placed behind the original view. You may shrink the original view or use the **Window** menu to bring the animation into view.

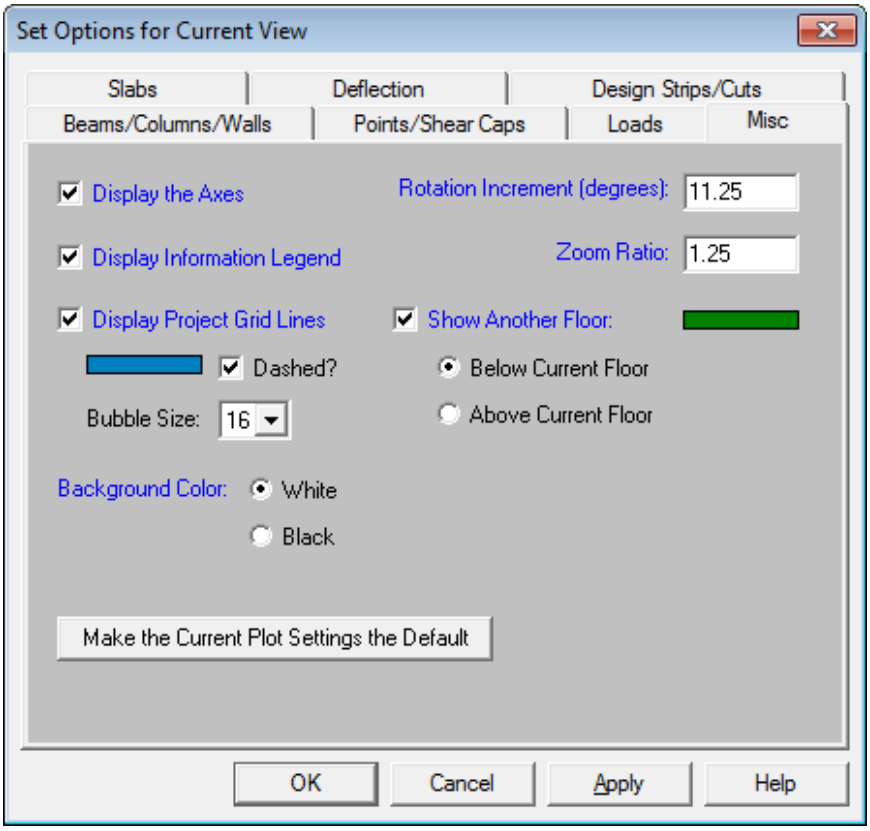
Miscellaneous

Access miscellaneous plotting options by clicking the **Model Display Options** button on the Window toolbar and selecting the **Misc** tab shown below.

Beam Supported Floors:



Concrete Slab Floors:



You may indicate whether the global axes icon is to be displayed in the upper left corner of the model view by checking the **Display the Axes** box. You also have the option of displaying the informational legend drawn in the bottom left corner of the model view by checking the **Display Information Legend** box. This legend has information such as which combination has been solved, which loads are being displayed, etc.

Checking the **Display Project Grid Lines** box will turn on the display of the project grid in the current model view. The color and line type for the grid may also be customized. Click the colored rectangle to choose a custom color. Check the **Dashed?** box to display grid lines as dashed lines. The **Bubble Size** drop down list allows you to choose a font size for the grid labels and the bubbles that encompass them.

You may specify whether the **Background Color** for the model view is to be black or white by clicking the corresponding radial button. A black background shows colors more vividly, but a white background shows a more realistic representation of what you'll get if you print the graphic view.

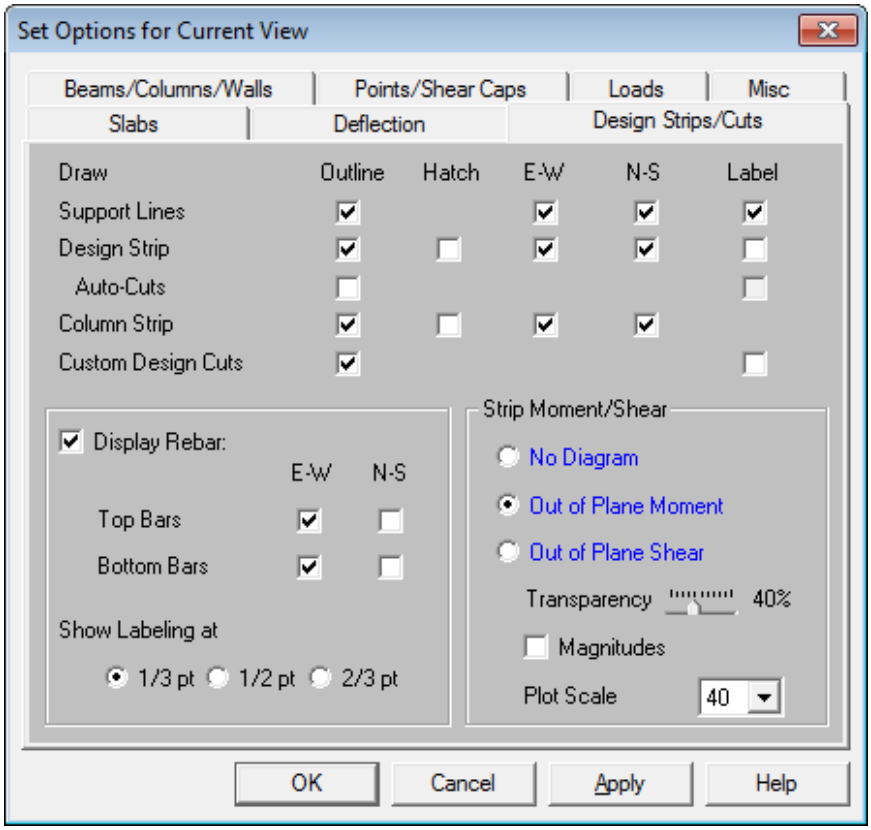
By entering values in the text boxes, you may control the **Rotation Increment** and **Zoom Ratio** that will be used with each click of the corresponding toolbar buttons on the **Window Toolbar**.

When the **Show Another Floor** option is selected you will see members from the floor above or below superimposed in a different color on the Plan/Iso view of your current floor. This is useful for laying out new floors, in that you can see where the columns/walls exist on the floors above or below. You can also use these elements as snap points for modeling.

Finally, you may make any plot option settings the default startup settings. To do this, go through all the Model Display Options tabs and set all the options to what you want as the startup default. Once that is done, press the **Make Current Plot Settings the Default** button. This button applies to ALL the Model Display Options tabs.

Design Strips/Cuts

Access the graphical Model Display Options for design strips by clicking the **Model Display Options** button on the Window toolbar and selecting the **Design Strips/Cuts** tab shown below.



Note: this plot option is only available on concrete slab floors.

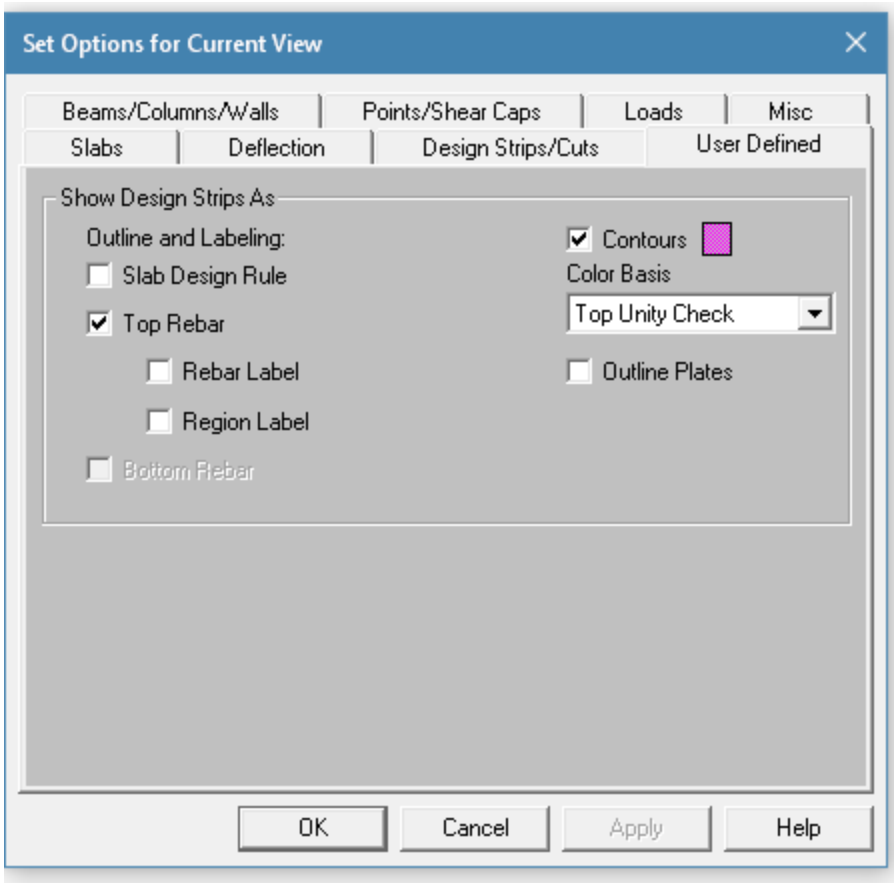
The **Draw** table allows you to check on or off the selected view options. These include slab **Support Lines**, **Design Strips**, **AutoCuts**, **Column Strips**, and **Custom Design Cuts**. Notice that you have various display options for each selection. **Outline** and **Hatch** control how the item regionis displayed. **E-W** (East - West) and **N-S** (North - South) control the directional selections and **Label** allows you to view the item's label.

The **Strip Moment/Shear** section of this tab controls the display of moment and shear diagrams on each design strip. **Out of Plane Moment** diagrams and **Out of Plane Shear** diagrams can be plotted on each design strip by clicking the corresponding radial button. The diagrams themselves may be presented with varying degrees of **Transparency** with 100% transparency meaning an outline of the force diagram and 0% indicating a solidly filled diagram. The transparency is controlled by clicking and dragging the slider bar until the desired percentage is displayed below the bar. Checking the **Magnitudes** box will turn on the display of the force magnitudes corresponding to each design cut within the design strips. The relative graphical scale of the moment and shear diagrams can be controlled by selecting a factor from the **Plot Scale** drop down list.

Note:

- The Moment and Shear Diagrams that are displayed are Enveloped to capture the maximum positive and negative force magnitudes along the length of each design strip.

User Defined



Note: this plot option is only available on concrete slab floors.

Slab Design Rule will display the name of the Design Rule on the slab. Checking the **Top Rebar** will show the Region above the Columns/Walls with the Rebar label shown after solution and the Region Label will show the size of the Top Rebar region. **Bottom Rebar** will be available after solution and it will display the rebar for the entire slab.

The Contours are displayed for Top and Bottom Unity Check with an option to turn on the plate outlines. These options are available after solution.

Graphic Editing

You may draw, edit, and load your model directly in the model views. You can draw columns, beams, walls, and slab edges between existing points or draw to grid intersections and have the points created automatically.

Drawing and Modification Features

There are several graphic editing features that make the creation and modification of models quite easy. Use the **Insert** and **Modify Menus** or the **Drawing Toolbar** to utilize these features in the model view. To create new beams, columns, etc., you can draw them using a drawing grid or draw "dot to dot" from existing points. Once you have created these items you may use other graphic features to load the model.

- See Columns for information on drawing and modifying **Columns**.
- See Walls for information on drawing and modifying **Walls**.
- See Beams for information on drawing and modifying **Beams**.
- See Decks and Slabs for information on drawing and modifying **Decks/Slabs**.
- See Loads - Load Types for information on drawing **Loads**.
- See Graphic Selection to learn how to make selections.

All model data is automatically recorded in spreadsheets and the spreadsheets and model views are always in tune (unless you turn this feature off in the **Application Settings** on the **Tools Menu**). As you edit a model graphically the spreadsheets are automatically updated and as you make changes in the spreadsheets the model views reflect these changes immediately.

All of the graphical modeling tools may be found on the **Drawing Toolbar** shown below. This toolbar may be turned on and off by clicking the button on the **Window Toolbar**.

RISAFloor Beam Supported Floor:

RISAFloor Concrete Floor Slab:

Where to Start

When you first open a new model in RISAFloor, you are prompted to create a new floor. Once a floor is created a project grid may be useful. This allows you to set up grid lines, which may then be used to define your columns or walls which in turn support the rest of your floor.

Apply Options

Some of the graphic editing options offer more than one way to apply a modification. This is because there are times when each option is useful. For example, changing the material of a beam from A36 steel to A572 steel can be accomplished using the **Beams Spreadsheet**. If you had to apply this change to 100 beams however you would not want to do that for each of them. A better way to do this would be to graphically select all of the beams and then apply the changes all at once.

Use the following options to specify how you want to choose the items to modify. Choosing **Apply Entries to All Selected Items** allows you to use the tools on the **Selection Toolbar** to choose the items you want and then apply the modifications to all selected items at once. Choose **Apply Entries by Clicking Items Individually** to then click on the items you wish to modify individually. See Graphic Selection for more information on the selection tools.

Note

- The selection and viewing tools override the graphic editing modes so that as you are editing the model you can alter the selected state. You will be returned to the current editing mode when you exit a selection mode.

Undo Operations

RISAFloor provides you with unlimited 'Undo' capability so that you may easily correct mistakes or just back up to try different possibilities. Simply click the button on the **RISA Toolbar** as many times as you wish to undo your previous actions. The model view and the spreadsheets will visually display the "undoing". Remember that spreadsheet edits are undone as well.

Note

- Changes made to the selection state of the model, the zoom level or rotation of the model view, or to the Model Display Options settings are NOT undone. Only a change to the model data can be undone.

Redo Operations

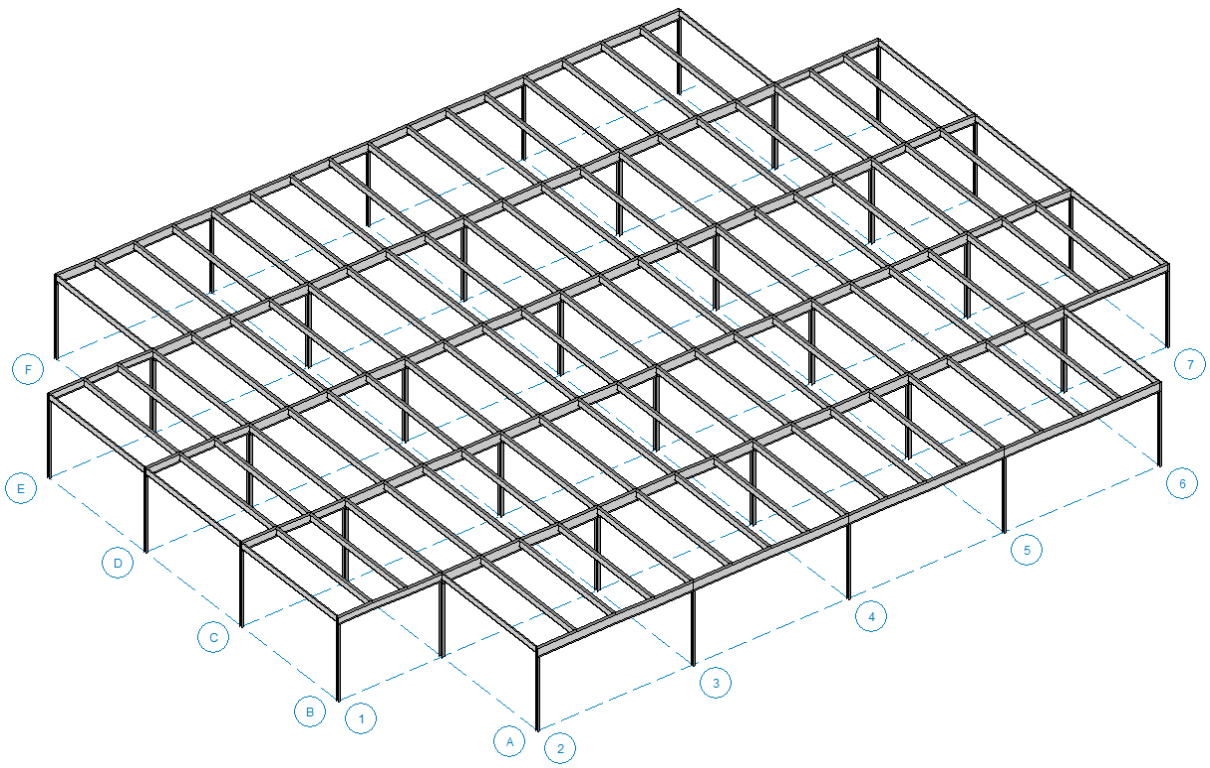
RISAFloor provides you with unlimited 'Redo' capability so that you may easily redo any actions that were previously undone using the 'Undo' button. Simply click the button on the **RISA Toolbar** as many times as you wish to redo actions that were previously undone. The model view and the spreadsheets will visually display the “redoing”. Remember that spreadsheet edits are redone as well.

Note

- Changes made to the selection state of the model, the zoom level or rotation of the model view, or to the 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.

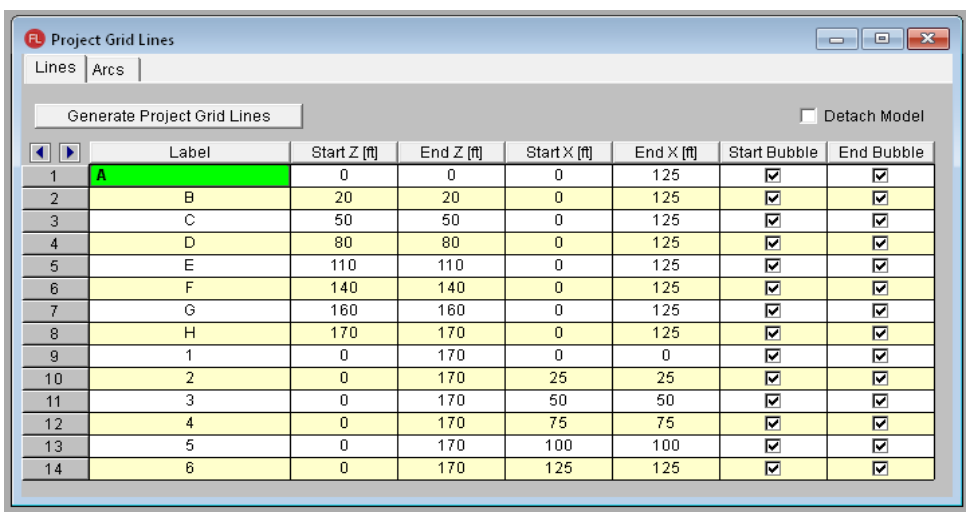
Project Grid

The Project Grid provides convenient snap points for modeling columns and walls in buildings and building-type structures. It also provides a 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.



Project Grid Spreadsheet

Click the button on the **RISA Toolbar**, or click the Project Grid button on the **Data Entry Toolbar** to open the **Project Grid Spreadsheet** shown below.



Project Grid Lines

Lines | Arcs

Generate Project Grid Lines

Detach Model

	Label	Start Z [ft]	End Z [ft]	Start X [ft]	End X [ft]	Start Bubble	End Bubble
1	A	0	0	0	125	☑	☑
2	B	20	20	0	125	☑	☑
3	C	50	50	0	125	☑	☑
4	D	80	80	0	125	☑	☑
5	E	110	110	0	125	☑	☑
6	F	140	140	0	125	☑	☑
7	G	160	160	0	125	☑	☑
8	H	170	170	0	125	☑	☑
9	1	0	170	0	0	☑	☑
10	2	0	170	25	25	☑	☑
11	3	0	170	50	50	☑	☑
12	4	0	170	75	75	☑	☑
13	5	0	170	100	100	☑	☑
14	6	0	170	125	125	☑	☑

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.

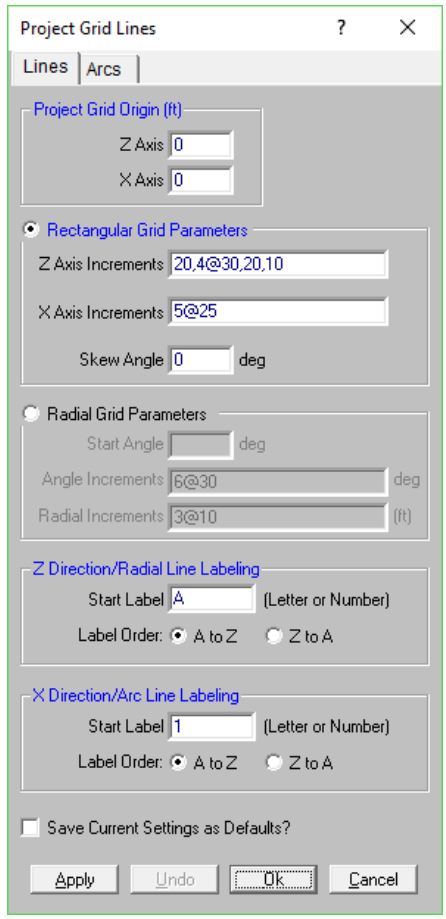
Detach Model

The columns and walls will move when the project grid is moved. The detach model check-box will allow you to move the project grid lines without moving the model.

Note: The model will move with the associate grid line/arc that moved. For example if you increase the radius of an Arc, the columns on that Arc grid will follow the arc. If you move a Line grid line, the columns/walls will move with that Line grid. This may mean that the column/wall is no longer at the intersection.

Project Grid Line Generation

The Project Grid Line generator can be accessed using the Generate Project Grid Lines button in the Project Grids spreadsheet, or by clicking the button on the graphic editing toolbar.



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 may use symbols such as "@", "/" and "," when entering the increments.

The "@" entry may be used to specify multiple, equally spaced, grid increments. For example, if you wanted 7 increments at 10 units each, you would type "7@10" in the increment field.

The "/" entry subdivides a larger increment into smaller equal increments. For example, the entry "12/4" would create 4 increments of 3 units each.

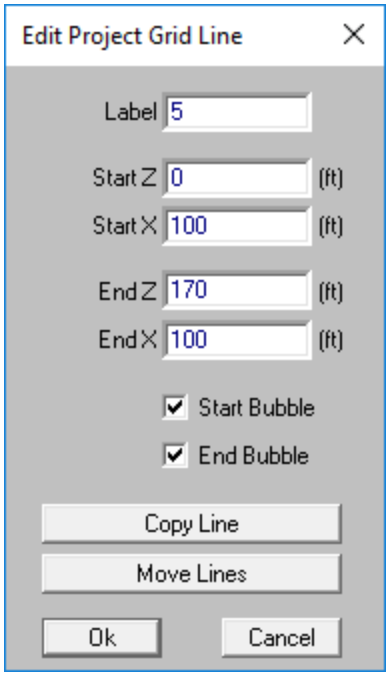
Use commas (",") to enter multiple increments in the increment field. For example, if you wanted to define increments of 3, 4, 7 and 2 units, you could enter "3,4,7,2" in the increment field.

The Apply and Undo buttons provide an easy way to preview a Project Grid before adding one to the model.

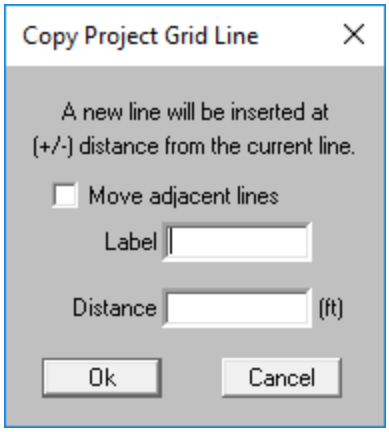
Once the project grid is specified and displayed in the model view, it will provide snap points while drawing your model.

Editing the Project Grid (Adding or Moving Grid Lines)

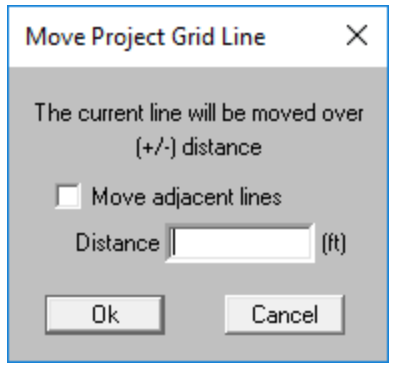
Once a Project Grid has been created the grid lines can be moved, or additional grid lines can be created, by double clicking on a grid line in the model view.



The Copy Line button allows a line to be created by offsetting an existing grid line. If the 'Move adjacent lines' is checked, a new grid line will be created and will shift existing grid lines in the same direction as well. If the 'Move adjacent lines' is unchecked, the new grid line will be created at the offset distance specified, without altering adjacent existing grid lines.



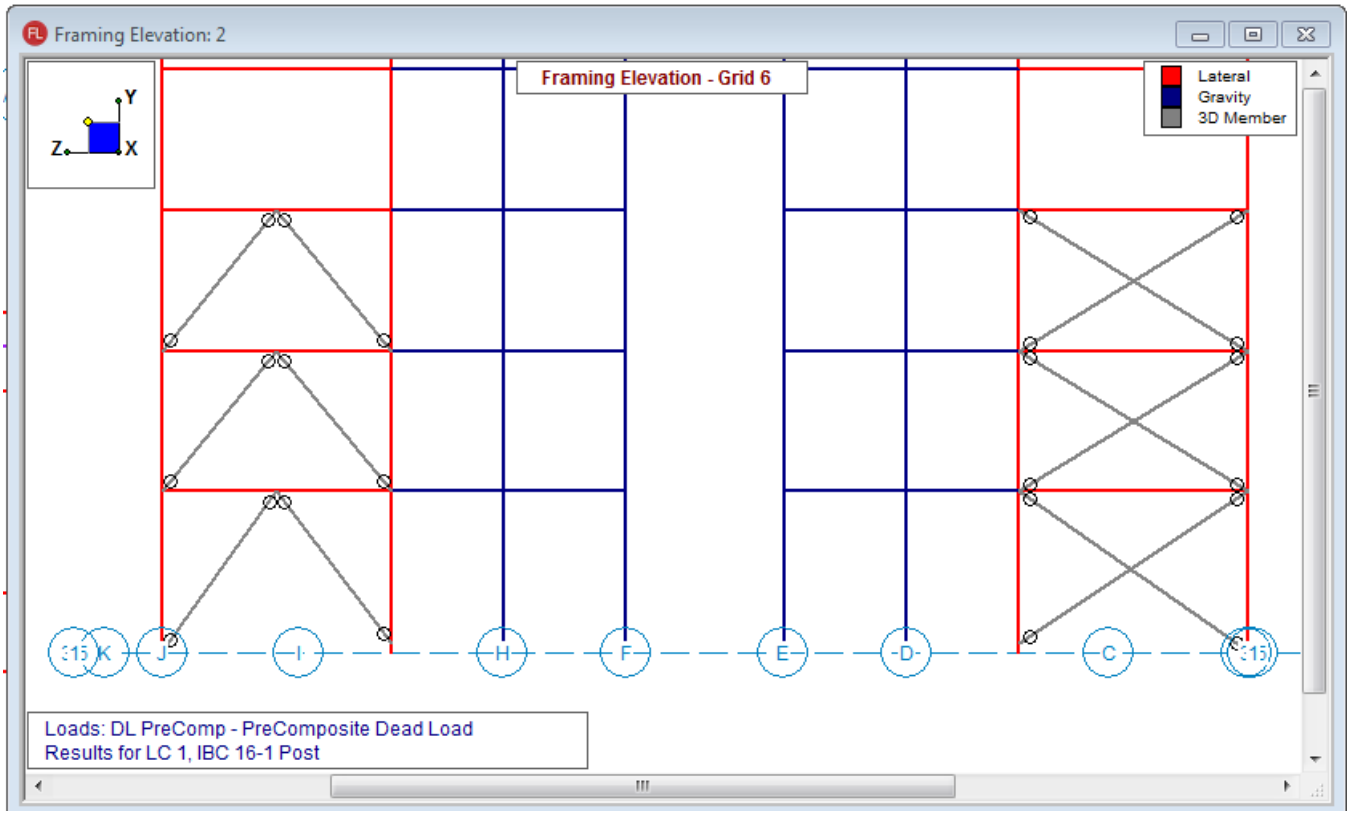
The Move Lines button allows a grid line to be moved by a specified distance. A positive distance will move the line in the positive global direction, while a negative distance will move the line in the opposite direction. If 'Move adjacent lines' is checked, all lines in the same direction will be moved by the same distance as well. This allows for a particular bay in a structure to be made larger/smaller without affecting the other bay sizes.



Elevation Framing View

In the RISAFloor interface, below the selection toolbar there is a an

button. This will initiate the Framing Elevation selection tool. The mouse cursor changes to a cross hair icon. Then when you click on a project grid line you will see the elevation view of that project grid line in a new window. The display and label options of this window are based on those in the currently selected model view, but can be changed by clicking on the plot options.

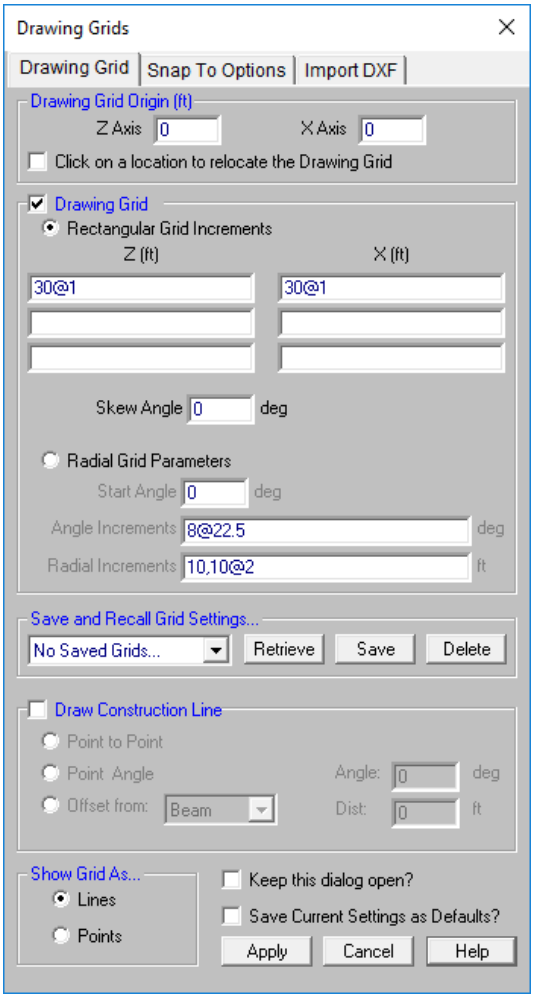


Drawing Grid

The **Drawing Grid** is a tool that lets you draw new columns, beams, and decks 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 columns, beams, and decks the points used to define them are created automatically. These points remain in their locations if the grid is relocated.

Drawing Grid Dialog

Click the button on the **Drawing Toolbar** to open the **Drawing Grid Dialog** shown above. The display of the **Drawing Grid** in the current model view may be turned on and off by clicking the button on the **Drawing Toolbar**. If the Drawing Toolbar is not visible in the current model view, click the button on the **Window Toolbar**.



You may choose to view construction lines and/or drawing grids. For drawing grids, you may choose between a rectangular drawing grid or a radial drawing grid. Also, rectangular drawing grids may be skewed from perpendicular to the global axis plane.

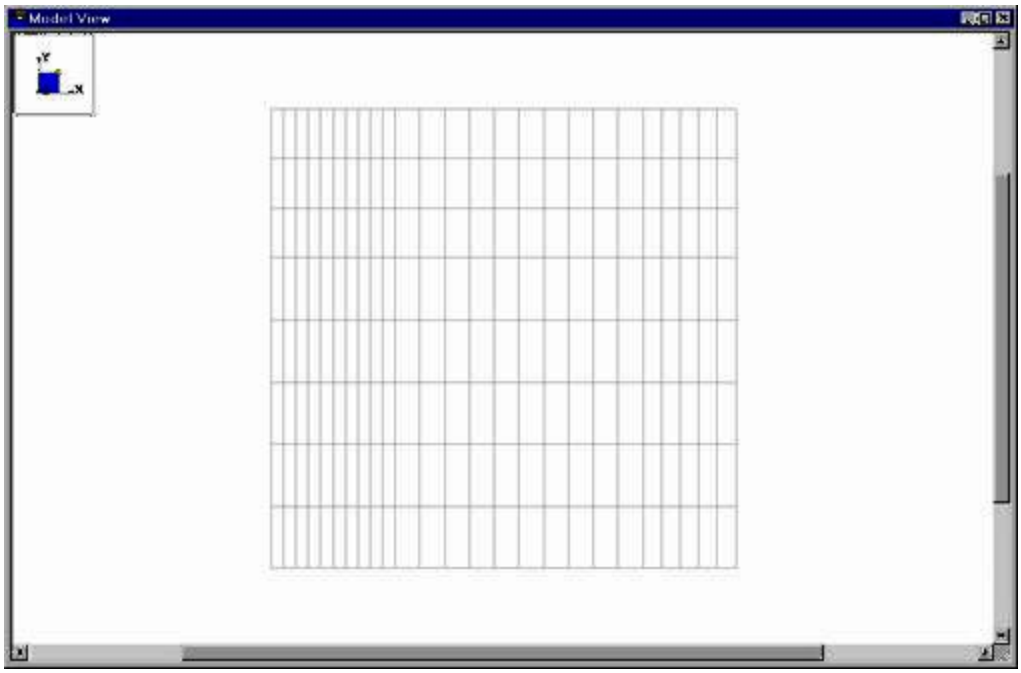
The **Save and Recall Grid Settings...** section allows for drawing grids may be saved and recalled for later use. Saved drawing grids are model independent, i.e. when you save a grid, you can reuse it in any other model you are working with in the future. To save a drawing grid, click the **Save** button after defining the grid. You will be prompted for a name for the drawing grid and it will then be saved and added to the list of grids available for recall. To recall a previously defined drawing grid, select the grid name from the drop down list and click the **Retrieve** button. If you wish to delete a previously saved drawing grid, select the grid name you want to delete from the drop down list and click the **Delete** button.

The **Show Grid As...** section gives the option of displaying the drawing grid as lines from grid point to grid point or simply as a grid of points.

You may save any of the grid information as the default setting so that when you start a new model that information is already there. To do this, simply enter the information that you want to save in the **Drawing Grids Dialog**, check the **Save Current Settings as Defaults** box, and then click the **OK** button.

Rectangular Drawing Grid

The rectangular drawing grid is defined by increments in two directions. The **Drawing Grid Origin** is where you want the grid increments to start. The **Rectangular Grid Increments** are the distances between the grid points or lines in two global directions.



You may use symbols such as "@", "/" and "," when entering the drawing grid increments.

The "@" entry may be used to specify multiple, equally spaced, grid increments. For example, if you wanted 7 increments at 10 units each, you would type "7@10" in the increment field.

The "/" entry subdivides a larger increment into smaller equal increments. For example, the entry "12/4" would create 4 increments of 3 units each.

Use commas (",") to enter multiple increments in the increment field. For example, if you wanted to define increments of 3, 4, 7 and 2 units, you could enter "3,4,7,2" in the increment field.

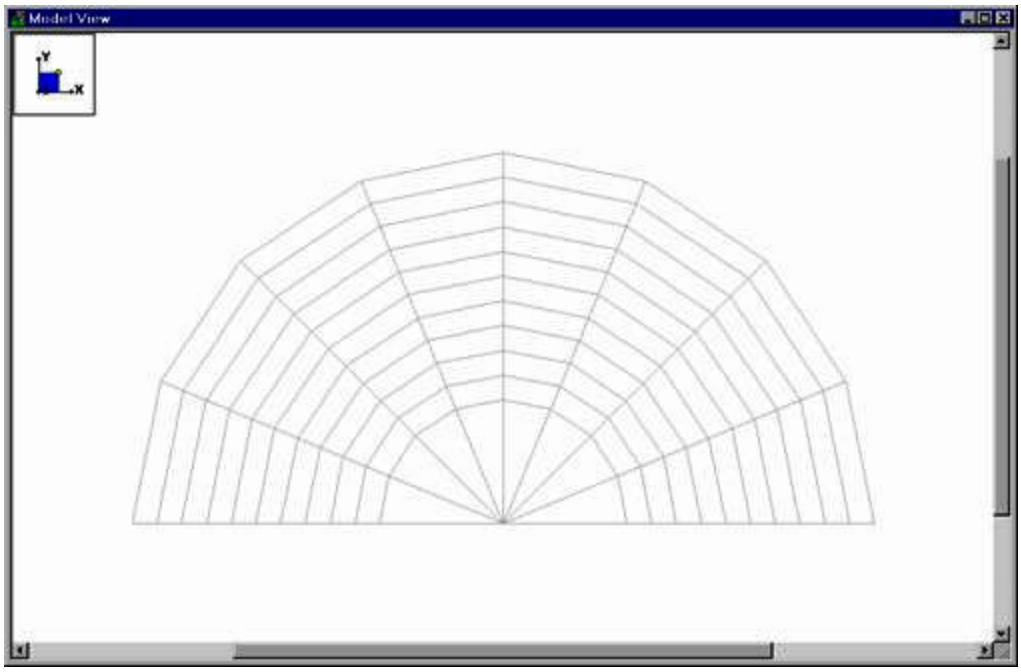
Once the drawing grid is specified and displayed in the model view, it will provide snap points while drawing your model.

Skewed Drawing Grid

The rectangular drawing grid can be skewed by specifying a skew angle. This option is available in the **Rectangular Grid Increments** section of the **Drawing Grids Dialog**. This skew angle will allow the creation of a regular rectangular drawing grid, but displayed in the model view at the specified angle, inclined from the global axis.

Radial Drawing Grid

Increments in two polar directions define a radial drawing grid. The **Drawing Grid Origin** is the point about which the grid will rotate. The default is at the global origin (0,0). The **Start Angle** defines the angle from the global axis that the first spoke will be drawn. The **Angle Increment** controls the number and angular spacing of the spokes in the grid. The **Radial Increments** controls the number and location of the rings in the grid.



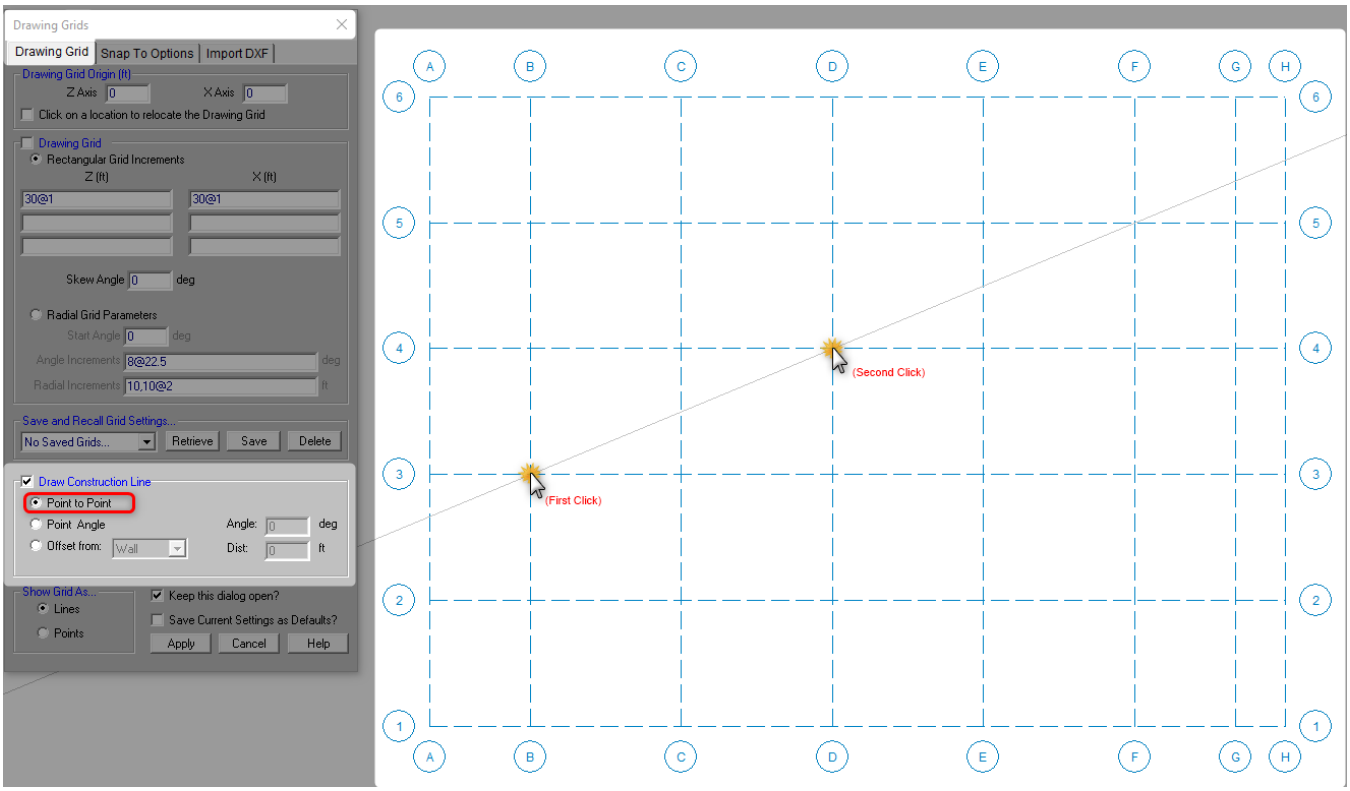
Relocating the Drawing Grid

The origin of the drawing grid may be specified in two ways. The first is to enter the exact global coordinates for the origin. This can be done by entering the values in the X and Z fields of the **Drawing Grid Origin** section. The second option is to specify the origin by clicking on an existing point in the model. This option is available by checking the **Click on a location to relocate Origin** box, clicking the **OK** button, and then clicking on the specific location in the model view where you wish the drawing grid origin to be located. The drawing grid origin will then be moved to this point.

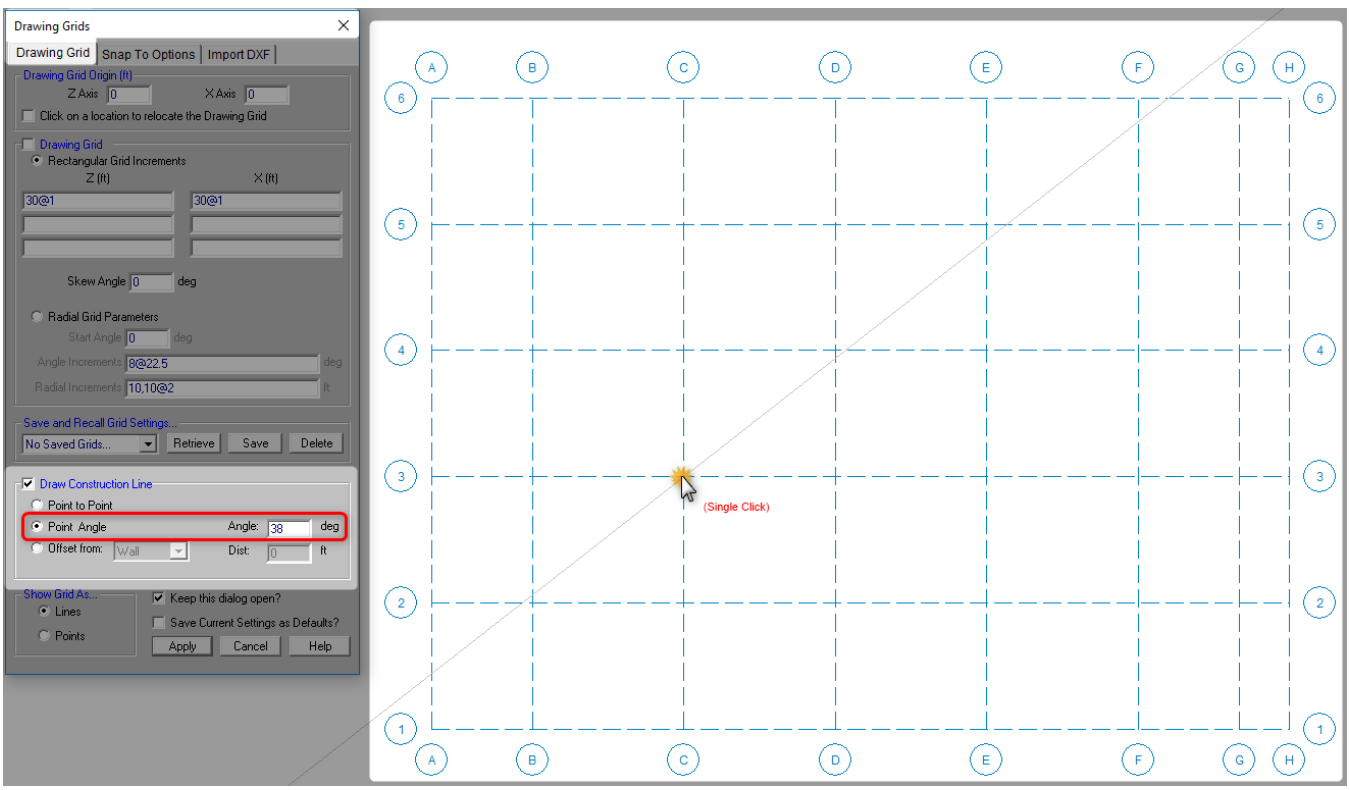
Construction Lines

The **Draw Construction Line** option adds the ability to draw Construction Lines as a type of Drawing Grid. Construction Lines are drawing grid lines that are infinite and allow you to snap to them just as if they were typical drawing grids. Construction Lines can be drawn by either **Point to Point**, **Point Angle** or **Offset**.

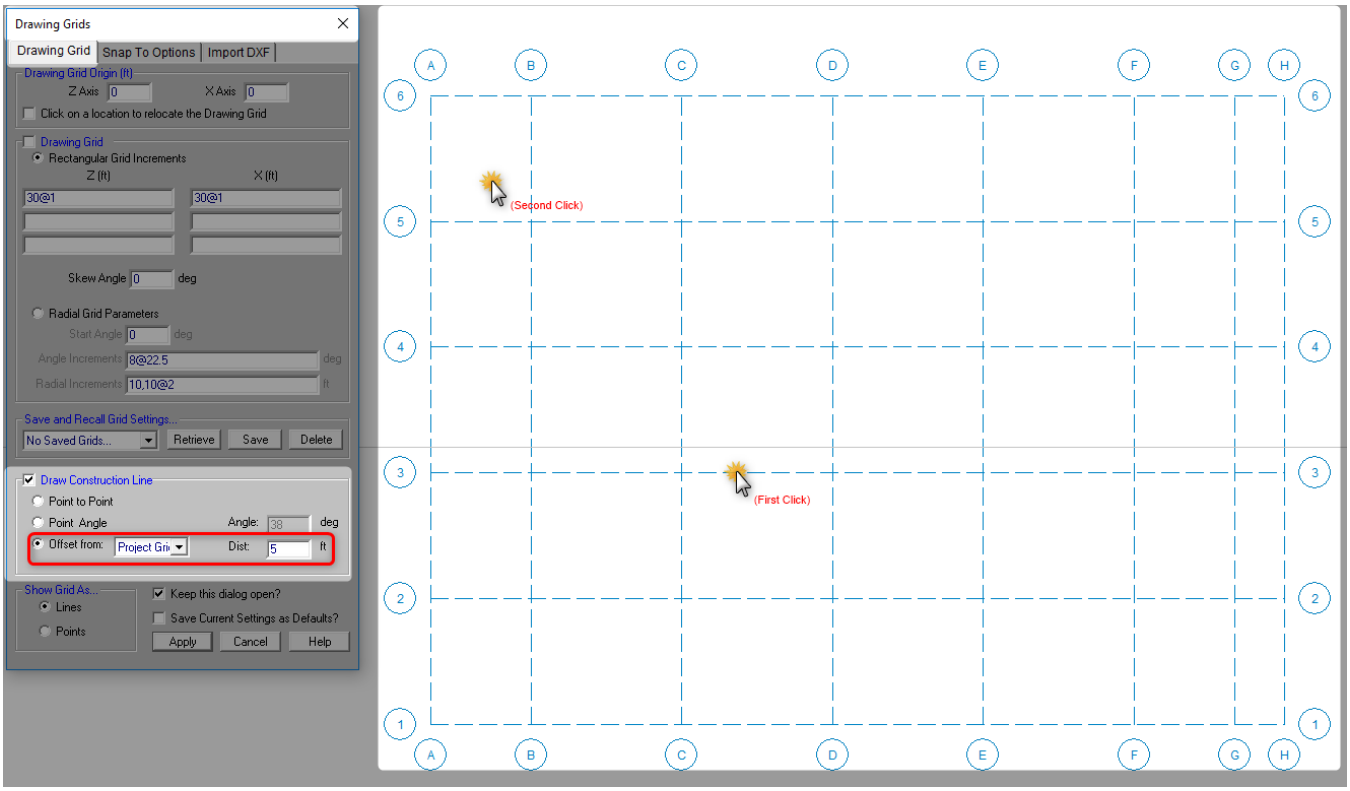
Point to Point will draw a construction line that passes through two selected points.



Point Angle will allow the user to define an angle within the dialog and then select a point for construction line to pass through at that specified angle.

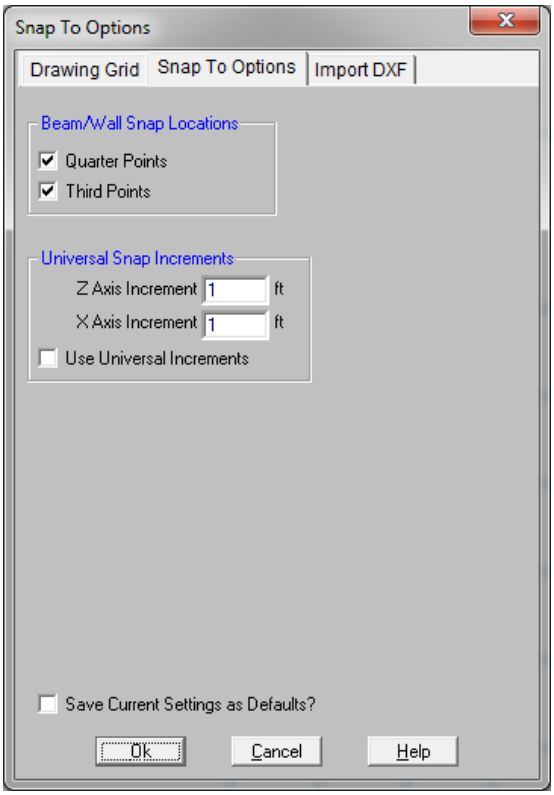


Offset will draw a construction line that is parallel and offset by a specified distance to any of three element types chosen by the user; Walls, Beams, or Project Grids. First select the item you would like to reference, then select on which side you would like the construction line to be drawn on.



Snap Points

Snap Points let you draw in the model view without the use of grids. To view or modify the snap point settings, click the button on the **Drawing Toolbar** and select the **Snap To Options** tab shown below.



In the **Beam Snap Locations** section, you can set the program to automatically snap to the **Quarter Points** and/or **Third Points** of a beam by checking the appropriate boxes.

The **Universal Snap Increments** section is used to define a snap grid for "free" drawing to any incremental location on a plane. To activate this feature, check the **Use Universal Increments** box. Or, the feature may be toggled on and off in the current model view by clicking the button on the **Drawing Toolbar**. If the Drawing Toolbar is not visible in the current model view, click the button on the **Window Toolbar**.

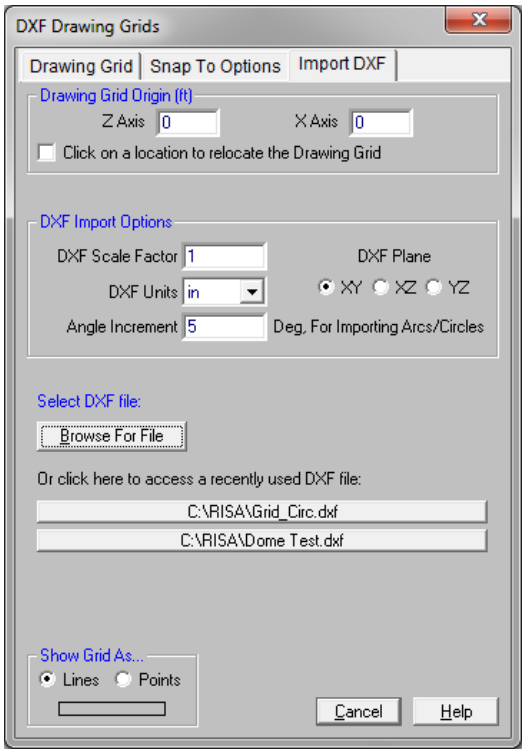
When snap points are activated, a red dot or asterisk will appear on your screen as you move your drawing cursor over one of these points. The exact coordinates of this point, and whether it is a 1/3 or 1/4 point of a beam, are reported in the status bar at the very bottom of the main application window just beneath the workspace.

Import DXF

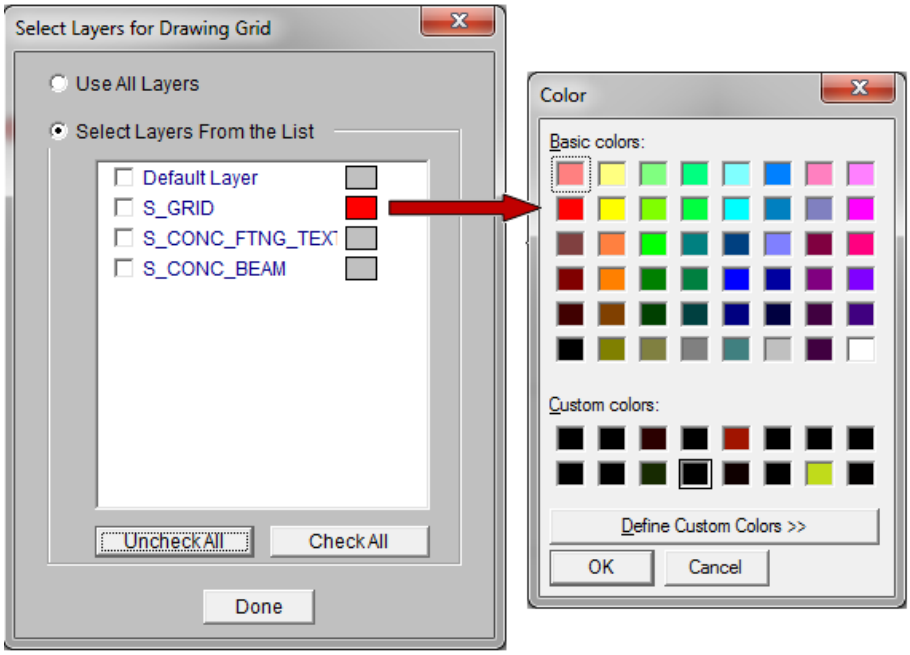
Import DXF lets you import a DXF drawing into the model view as a grid. The DXF image is imported so that you can snap to any point or intersection to aid in the drawing of your model. The DXF Import Drawing Grid supports Lines, Polylines, Circles, Arcs, Polylines with Arcs, and Points.

To Import DXF Drawing Grid:

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Select the Drawing Grid Origin or click to checkbox to manually locate it after the dialogue closes.
3. Assign the DXF Import Options and DXF Plane.
4. Select Lines or Points for grid lines.
5. Select the DXF File from a file or from the list of recently used files. The program will remember the last 5 files imported.
6. Once the file is imported, you will be prompted to "Use All Layers" or "Select Layers from the List". You can check or uncheck selected layers, and select colors of the layers display by clicking on the gray box (defaults to gray).



The program will default to use All Layers. Or you can select the color of the grid lines/points by clicking on the color box next to each layer name and select the color.



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 and Z fields of the **Drawing Grid Origin** section. The second option is to specify the origin by clicking on an existing point in the model. This option is available by checking the **Click on a location to relocate Origin** box, clicking the **OK** button, and then clicking on the specific location in the model view where you wish the drawing grid origin to be located. The drawing grid origin will then be moved to this point.

Drawing Plane

Select the RISA drawing plane for the DXF file shall be placed.

DXF Scale Factor

Enter the scale factor that will cause the DXF file to be scaled up or down to full scale. For instance, if you had created a scaled model in AutoCAD at a scale of 1/4"=12", then the appropriate scale factor to produce a full size RISA- model would be 48. The default is 1.0.

DXF Plane

Select 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 Units

Select the units you used in the CAD model from which you produced the DXF file. The supported DXF units are none, inches, feet, mm, cm, and meters.

Angle Increment

The program will break an arc or circle into straight line segments to allow you to snap to them. This indicates the number of degrees the that an imported arc or circle should be broken into.

Copying Model Elements

To Copy Selected Items

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Select the items you want to copy.
3. To copy the selection *linearly* click the **Copy** button and select one of the following options:
   - **Copy by Point Location**
   - **Copy by Increment**
   - **Copy by Angle**
   - **Copy to Another Floor**

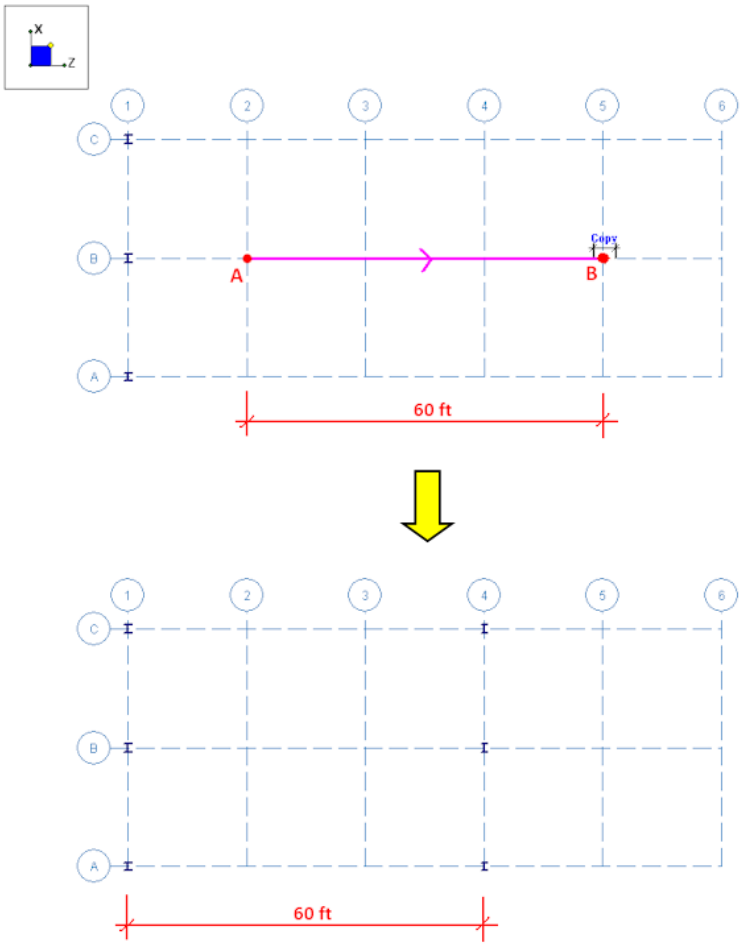
Note:

- Use the "@" symbol to specify multiple equal increments. For example specifying "3@10" will give you 3 copies at 10 units apart.
- You may undo any mistakes by clicking the **Undo** button.

Copy by Point Location

This allows you to copy the selected items by clicking on any two points. The selected items will copy the distance between the two points in the direction of the first to second click.

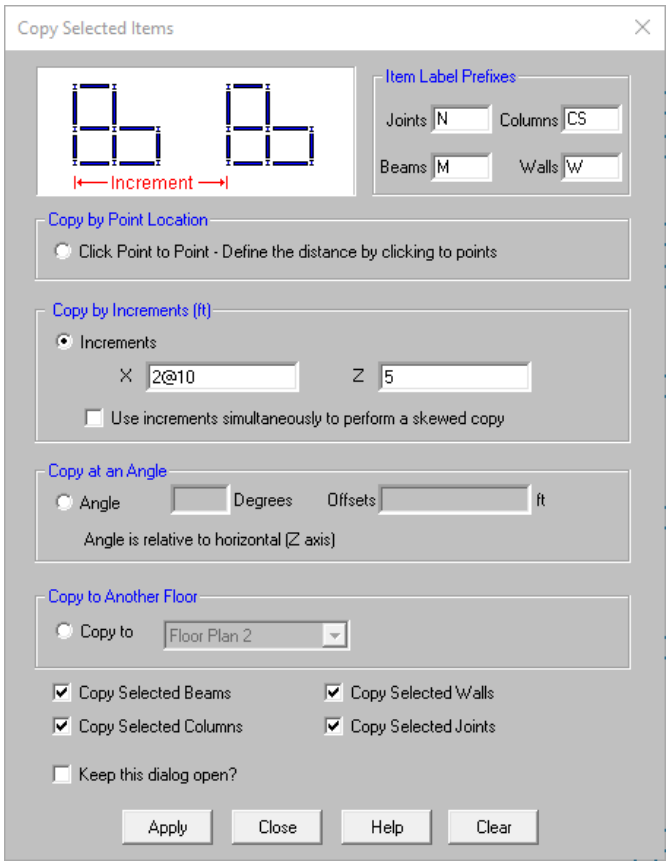
For example, if you click two points: A then B in the model below, the columns will copy 60 feet in the positive Z direction.



Copy by Increments

This allows you to copy the selected items by entering in increments in any or all of the global directions. The selected items will copy the increment distance(s) that you have entered.

Use the "@" symbol to specify multiple equal increments. For example specifying "3@10" will give you 3 copies at 10 units apart.



Select the **Use increments simultaneously to perform a skewed copy** checkbox to combine the orthogonal increments into a single resultant increment vector to copy the elements in a direction other than the three global orthogonal directions.

Select **Copy to Another Floor** to graphically copy beams, columns, walls and joints on a floor level to another existing floor level.

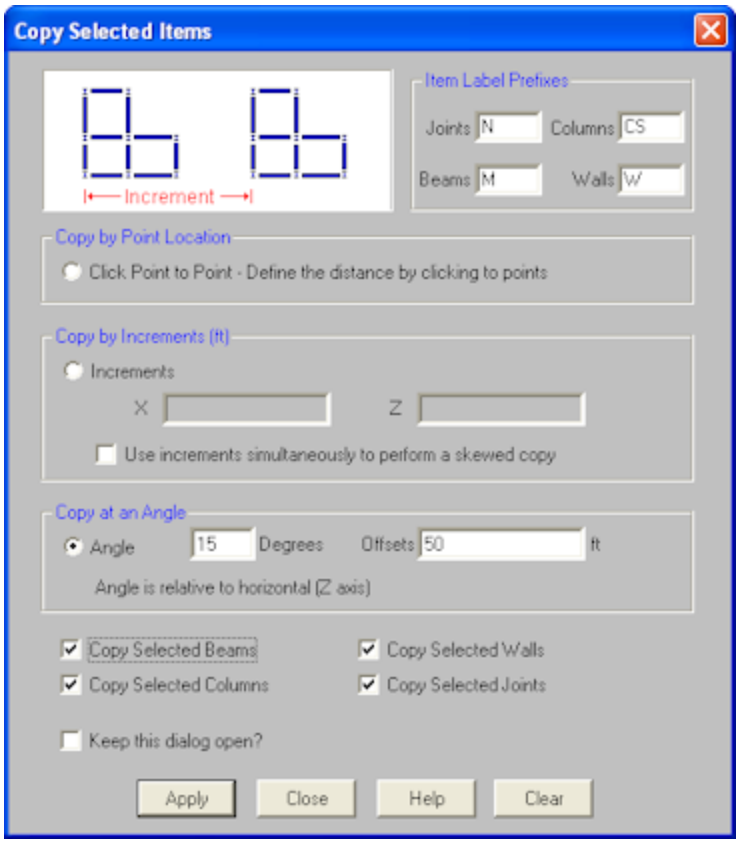
Note:

- Copy tool does not support floor to floor copies on Parent/Child floors.

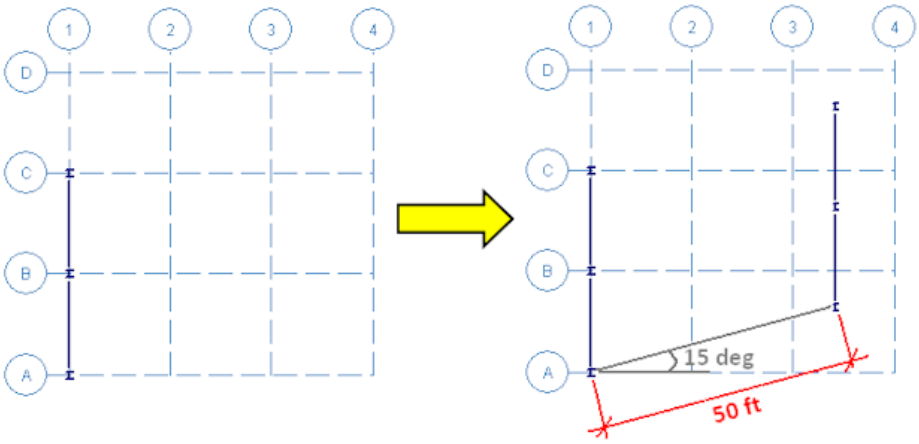
There are many **Copy Selected** checkboxes at the bottom of the dialog to define which elements in the model you wish to copy.

Copy by Angle

This allows you to copy selected items a certain distance at an input angle. Simply enter the angle (relative to the global Z axis) and the increment and your selected items will be copied in that direction.



For example, if you enter an angle of 15 degrees and an increment of 50 feet in the model below, it will copy the frame 40 feet at 45 degrees.



Copy to Another Floor

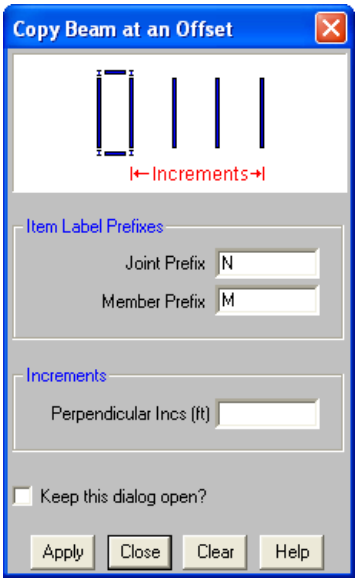
The copy to another floor option will duplicate selected beams, columns, walls, and joints on a level to another existing floor level. To use this tool, select these elements in your model view and choose an existing floor level to copy those elements on. The copied elements will be duplicated on the selected floor level. The location of the elements will remain in the same X and Z coordinates as the original floor they were copied from.

Note:

- Elements can only be copied to similar floor types, e.g. beam to beam supported floors and slab to slab floors.
- Elements cannot be copied to or from parent/ child floors.

Copy Offset

This tool allows you to copy selected beams at an offset simply by entering the offset distance, clicking on the member, and then clicking on the side of the member at which you'd like it to be offset.



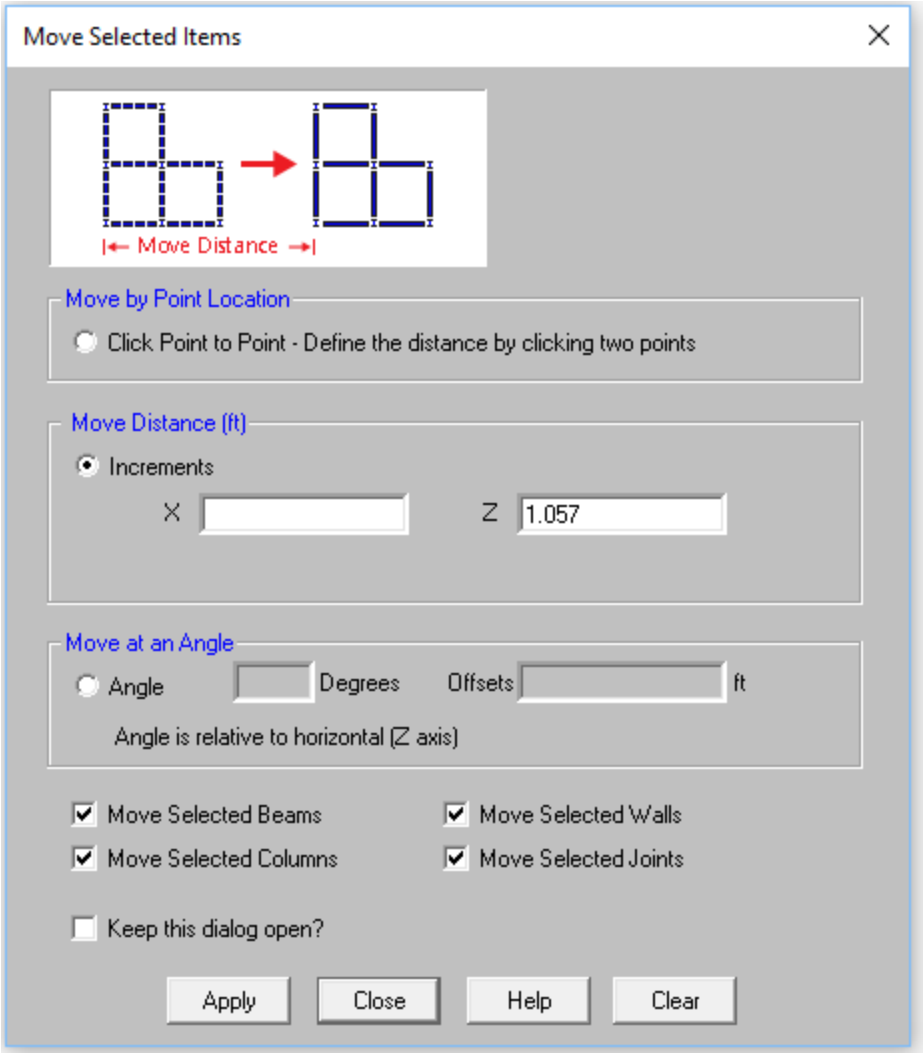
Moving Model Elements

To Move Selected Items

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Select the items you want to move.
3. To move the selection linearly click the **Move Linear** button and specify the offset distances.

Note:

- You may undo any mistakes by clicking the Undo button.



To Move Selected Items

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Select the items you want to move.
3. To move the selection items *linearly* click the **Move** button and select on of the following options:
   - **Move by Point Location:** This allows you to move the selected items by clicking on any two points. The selected items will move the distance between the two points in the direction of the first to second click.
   - **Move Distance:** This allows you to move the selected items by entering in an increment of the global directions. If you enter both X and Z directions, the selected items will move diagonally.
   - **Move at an Angle:**This allows you to move selected items a certain distance at an input angle. Simply enter the angle (relative to the global Z axis) and the increment and your selected items will be moved in that direction.

Note

- You may undo any mistakes by clicking the Undo button.

Trim/Extend Members

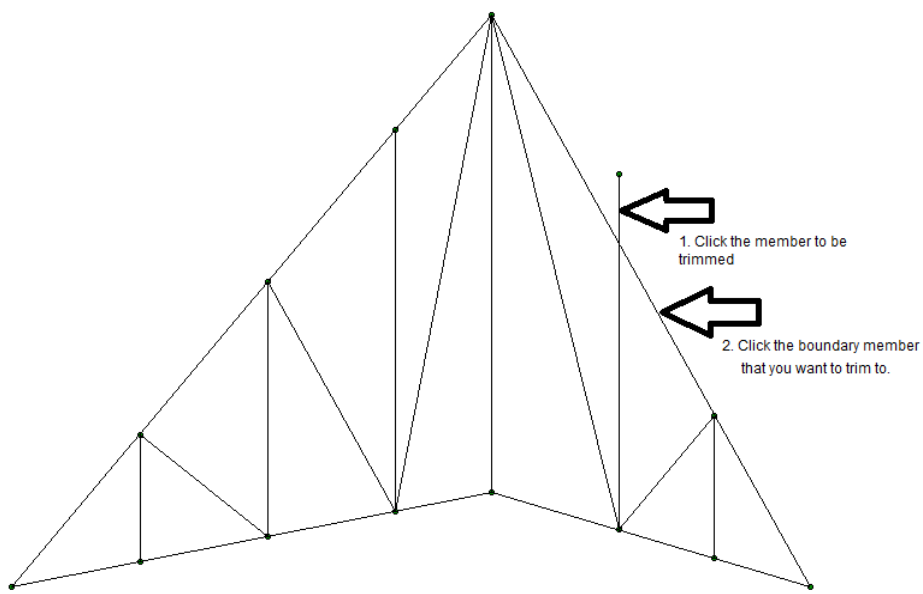
As you modify the model, you may find that you will need to trim or extend a member. The Trim/Extend tool will allow you to modify the ends of any member in relation to another member. This extension is for members only, not walls or plates.

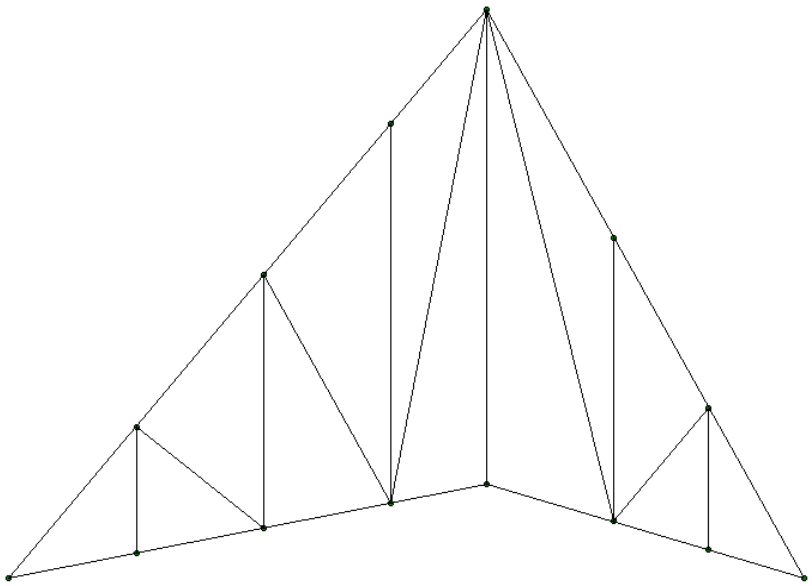
Note

- 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 joint. This tool is modifying the length of the member.

To Perform a Trim

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Select the items you wish to trim.
3. Click the **Trim** button.
4. Click on the member end to Trim. Then click on the boundary member.

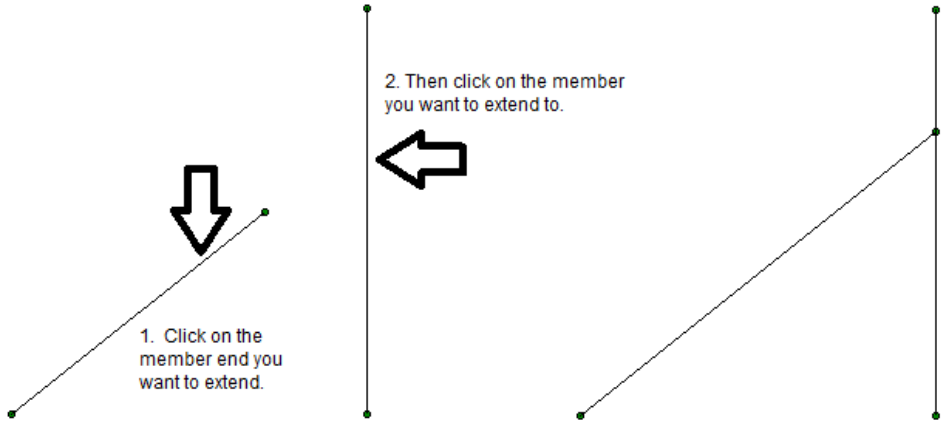




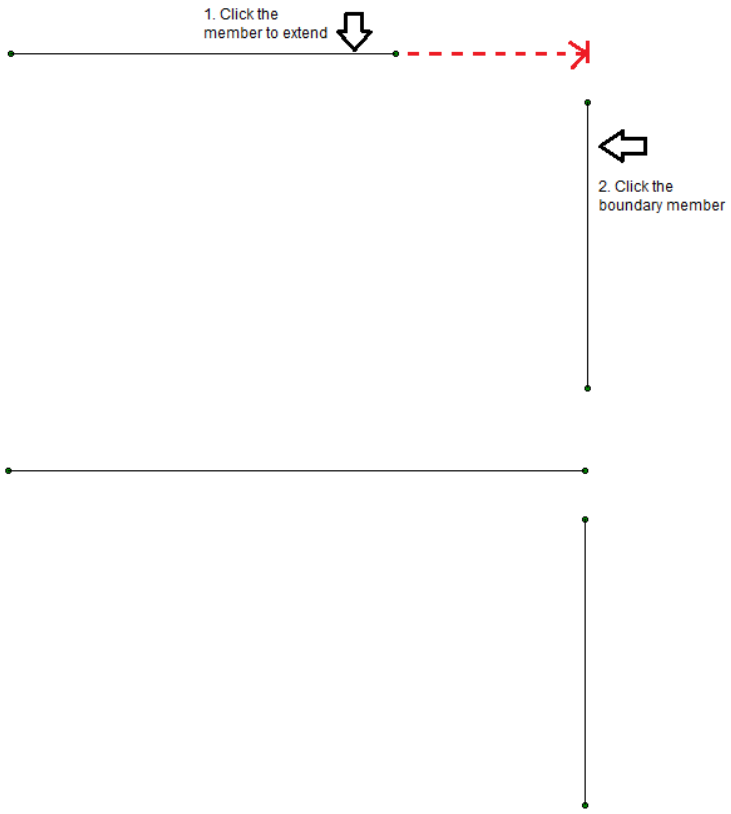
To Perform an Extend

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Select the items you wish to merge.
3. Click the **Extend** button twice or select Extension by clicking the arrow.
4. Click on the member end to Extend. Then 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.



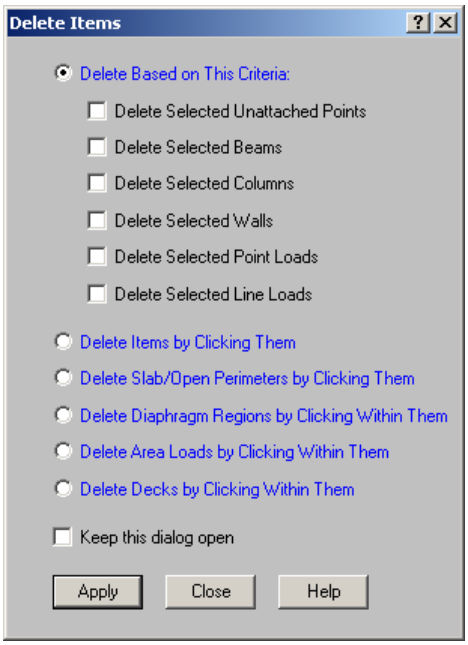
Deleting Elements

You can delete parts of the model based on the current selection state, or you can click on the items you wish to delete individually. If you accidentally delete something you didn't want deleted, you can use the Undo feature to repair the damage.

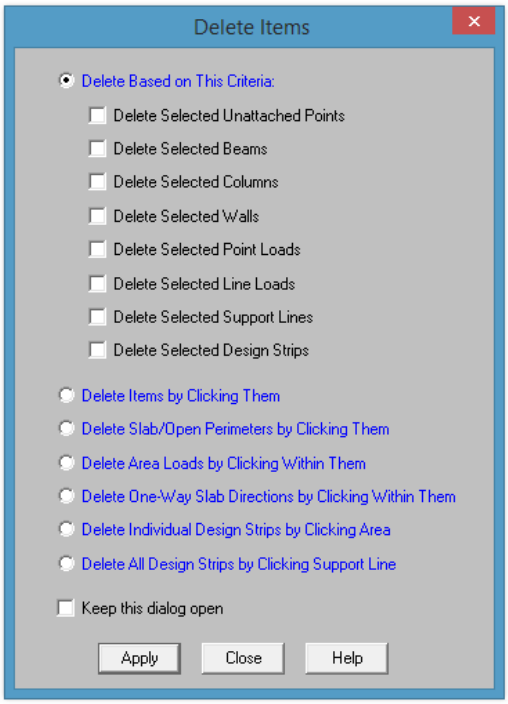
If you wish to delete based on the current selection state, you must use the check boxes to define the criteria the program will use to perform the deletion. Only items that are selected and that have their check boxes "checked" will be deleted. The choices let you delete points, beams, columns, decks/slabs, edges, and/or loads.

Keep in mind that if you delete points, any beams, columns, etc. attached to the deleted points MUST also be deleted, regardless of whether those elements are selected or not.

Beam Supported Floors:



Concrete Slab Floor:



A noteworthy feature is the ability to delete unattached points. Sometimes in the process of modeling you accidentally create unwanted unattached points. This parameter gives you a convenient way to remove these unwanted points.

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 Display Options Dialog**, you can easily delete specific types of loads for particular load categories or load combinations.

Alternatively, you can choose to delete items by clicking on them individually. Select the radial button next to **Delete Items by Clicking Them** and use the mouse to click on any individual item you wish deleted.

Remember to click the **Apply** button to make your choices active.

To Delete Selected Items

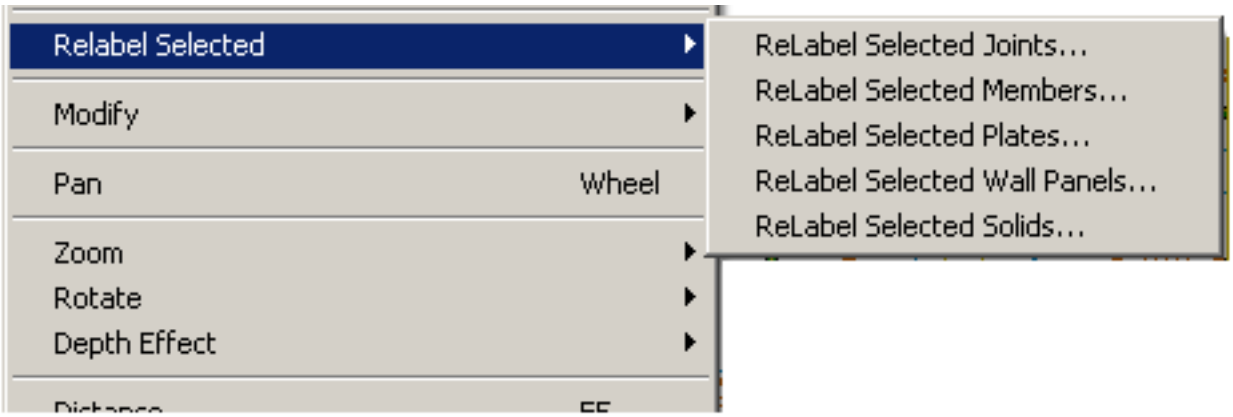
1. If there is not a model view already open then click the button on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Select the items you want to delete.
3. To delete the items click the **Delete** and specify the types of items.

Note

- You may undo any mistakes by clicking the Undo button.

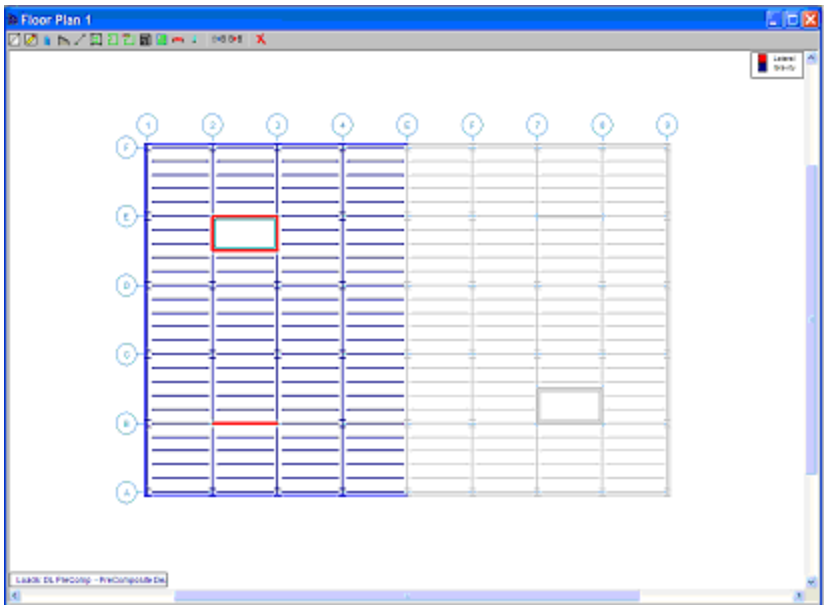
Re-Labeling Selected Elements

You can use the right-click menu in the program to re-label only the selected items. This is useful when you want all members at a certain elevation to have a prefix which denotes the floor on which it is located.



Graphic Selection

You may graphically select items in order to view or modify part of the model or to print part of the results. When used in conjunction with the graphic editing features it allows you to quickly model and make modifications to the model. If used with results it allows you to view and print only the input or results that you want.



The elements that you see in the model views have two possible states; selected and unselected. By default, all items are selected, and therefore fully displayed. If you unselect any items, they are drawn in a gray or “ghosted” shade. To select or unselect an item simply click on it with the left mouse button. You may also use one of the selection tools below to select multiple items.

Note

- As an alternative to using the selection tools, some operations offer a **Click to Apply** option, which allows you to modify items by clicking or boxing them with the mouse. This is useful when working with a few items.
- 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 Display Options Dialog**, they remain in the current selected state and can still be selected/unselected.

Selection Modes

The **Selection Toolbar** is the vertical toolbar located on the left side of the screen. This toolbar is for selecting points, beams, and slabs in model views. This toolbar will not be available when a model view is not active. You may also access the selection tools from the **View Menu**.

Some of the tools are for one-time applications such as **Select All**. Other tools, such as **Box Select**, place you in a selection mode that remains active until you cancel it. The current mode is indicated by the mouse pointer and by the state of the button. While in a selection mode the button will stay depressed until you click it again, choose another button, or press the ESC key. You may have more than one model view open and may be in different modes in each view.

Note

- There are other types of graphic modes such as **editing** and **viewing** (zooming) modes. The viewing mode overrides the selection mode, which overrides the editing mode. This is so that while you are editing the model you can alter the selected state. You will be returned to the current editing mode when you terminate a selection mode.
- To cancel a selection mode re-click the same button so that it is in the "up" position, or press the ESC key, or click the right mouse button.
- You can cancel a box, line, or polygon selection that is underway by dragging the mouse off the model view while still holding the mouse button down.

Select All and Unselect All

Clicking **Select All** and **Unselect All** tools will select or unselect all of the active points, beams, and columns in the model.

Box Select and Unselect Modes

The **Box Select** and **Box Unselect** tools allow you to draw a box around the items that you wish to select or unselect. Beams and columns must be entirely within the box in order for them to be affected.

Polygon Select and Unselect

The **Polygon Select** and **Polygon Unselect** tools allow you to draw a polygon around the items that you wish to select or unselect. Beams and columns must be entirely within the polygon in order for them to be affected.

Line Select and Unselect

The **Line Select** and **Line Unselect** tools allow you to draw a line through the items that you wish to select or unselect. Any element the line crosses will be affected. This is useful when choosing items between other items.

Inverting Selections

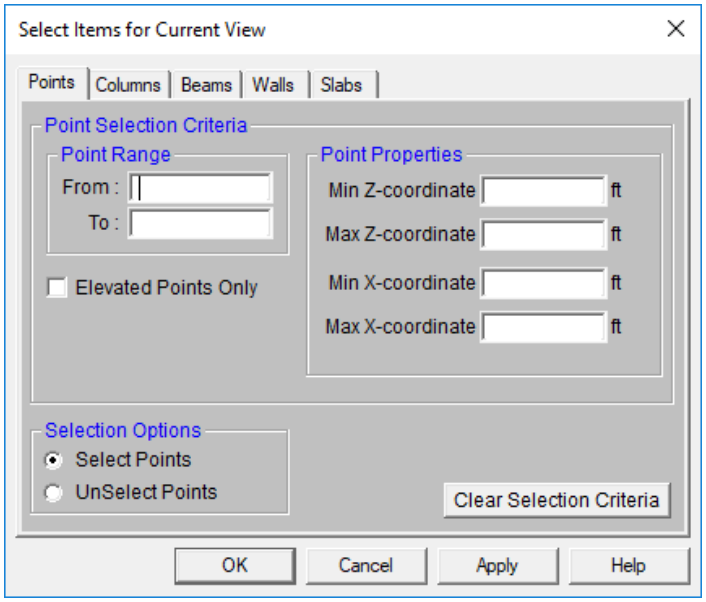
The **Invert Selection** button is used to invert the selected state of the model. When clicked, all selected items are made unselected and all previously unselected items are made selected. For example, this can be very useful when just a few items are to be selected in a large model. Simply click on the desired items to unselect them, then click the **Invert Selection** button to make them selected and everything else unselected.

Criteria Selections

The **Criteria Selection** button allows you to select items based on a wide range of criteria such as coordinates, labels and many other conditions. When the **Criteria Select** button is clicked, you will be presented with the Criteria Selection Dialog with options grouped by tabs across the top. Each tab represents groups of criteria that you may use to refine your selection.

The options are numerous making it easy to quickly achieve complicated selections. This is a powerful tool so it is worth taking the time to experiment with the options so that you will know how to use it to your advantage. The various tabs in the dialog are described below.

Points



You may **Select** or **Unselect** Points as follows.

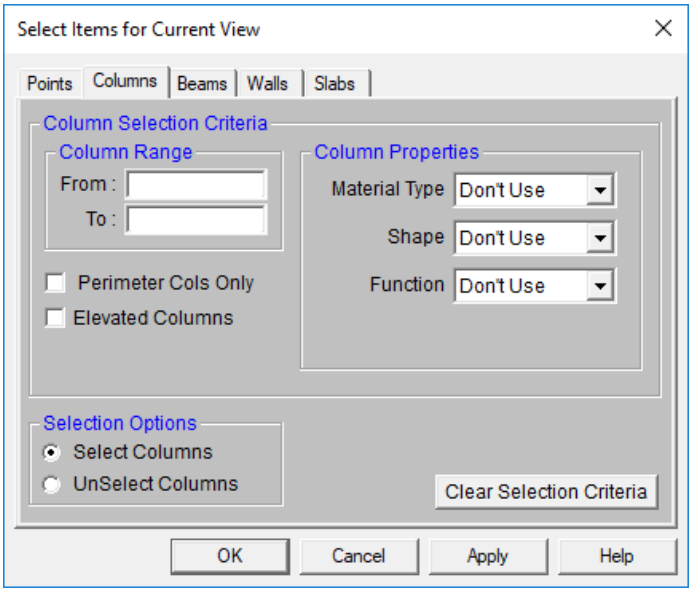
Point Range - You may specify a range of point labels. Specifying only one label selects just that one point.

Point Properties - You may specify a Min Horizontal, Max Horizontal, Min Vertical, and/or Max Vertical coordinate location.

Note

- If no criteria is specified, all the Points will be selected/unselected.

Columns



You may **Select** or **Unselect** Columns as follows.

Column Range – You may specify a range of Column labels. Specifying only one label selects just that one Column.

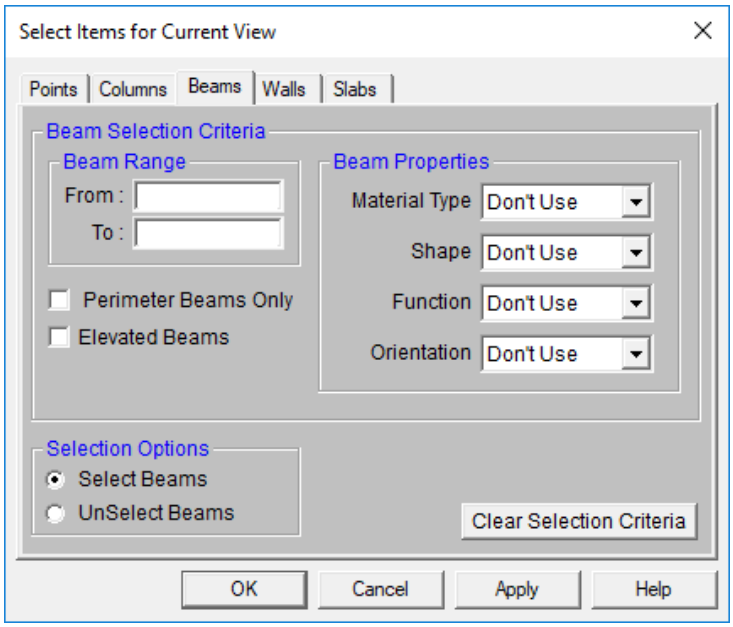
Column Properties – You may specify a Material Type, Shape, or Function when selecting columns.

Perimeter Selection – Checking this box will select only the perimeter columns of your building.

Note

- If no criteria is specified, all the Columns will be selected/unselected.

Beams



You may **Select** or **Unselect** beams as follows.

Beam Range – You may specify a range of Beam labels. Specifying only one label selects just that one beam.

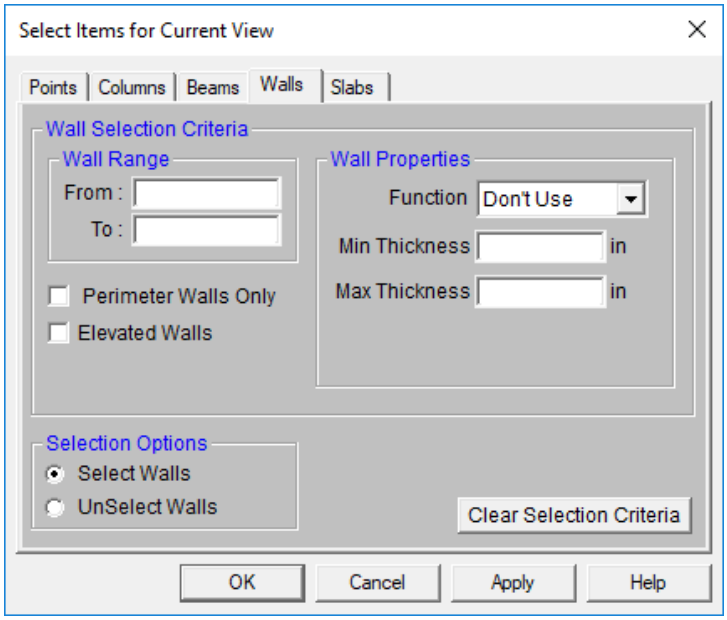
Beam Properties – You may specify a Material Type, Shape, Function, or Orientation when selecting beams.

Perimeter Selection – Checking this box will select only the buildings perimeter beams.

Note

- If no criteria are specified, all the Beams will be selected/unselected.

Walls



You may **Select** or **Unselect** walls as follows.

Wall Range – You may specify a range of Wall labels. Specifying only one label selects just that one wall.

Wall Properties – You may specify a Function or Thickness when selecting walls.

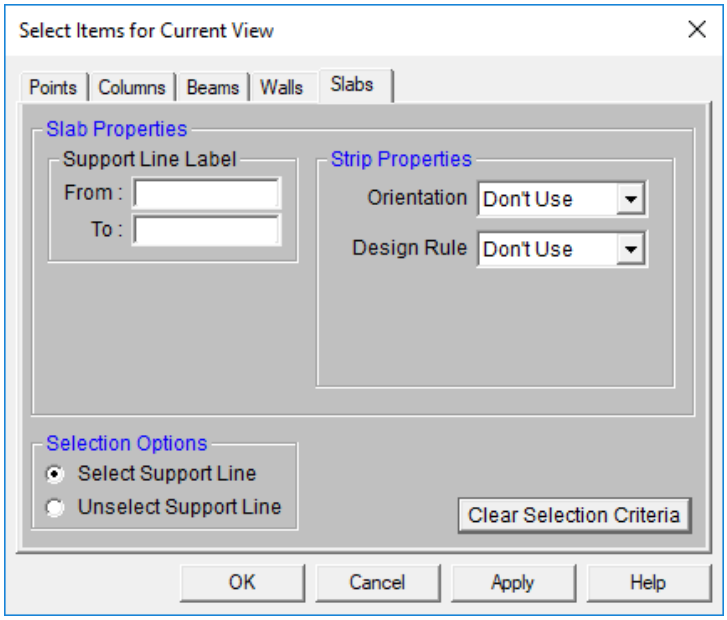
Perimeter Selection – Checking this box will select only the buildings perimeter walls.

Elevated Walls – Checking this box will select only the walls that have been modified using the Elevation Offset tool.

Note

- If no criteria are specified, all the Walls will be selected/unselected.

Slabs



You may **Select** or **Unselect** slabs as follows.

Support Line Label – You may specify a range of Support Line labels. Specifying only one label selects just that one support line.

Strip Properties – You may specify an Orientation or Design Rule when selecting support lines.

Note

- If no criteria are specified, all the Support Lines 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.

To Unlock the unselected members in the model view, click the **Unlock Unselected** button and the unselected items that were previously "removed" from the model view will be returned to the view in the unselected state.

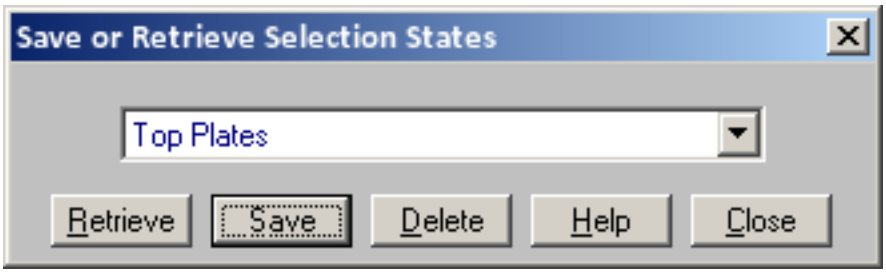
Graphic Selection from Spreadsheets

In many spreadsheets, both Data Entry and Results, you can select rows on the spreadsheet and use the right-click menu to select the S**elect Marked Lines in Current View** or **Unselect Marked Lines in the Current View** options. This will graphically select or unselect the items corresponding to the selected rows in the top most model view window. This is very useful for highlighting failing members in a spreadsheet and then having them graphically selected in a model view to see where the problems are.

Saving Selections

You can save and recall various selection states for a model. If the model is altered after a selection state has been saved, the saved selection state will also be altered. Any new items (points, beams…) will be set to "selected" in any selection states saved prior to the creation of the new item.

To save a selection state, click the **Save Selection State** button on the **Selection Toolbar**.



Click the **Save** button and provide a name for the saved selection. You can have up to 15 different saved selections in a model. To retrieve a saved selection, choose the selection state from the drop down list and click the **Retrieve** button. To delete a saved selection, choose the selection state from the drop down list and click the **Delete** button.

Help Options

RISA 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 many ways that you can get help in understanding the software.

Electronic Help File

The **Help File** is designed to help you get your work done as quickly as possible and is intended to provide:

- Procedures that lead users through the steps of completing tasks
- Context-sensitive Help topics that provide users with quick descriptions of items on their screens
- Troubleshooting topics that guide users through solutions to common problems
- Extensive discussions for a thorough understanding of engineering and modeling topics
- Easy access to related topics

The electronic help file can be accessed by clicking the **Help File** button on the **RISA Toolbar**. A new window containing a Table of Contents will be opened. Click on any item in the Table of Contents for extensive information on the topic.

Context Sensitive Help

Context Sensitive Help is help that a user can access in context while working in a program. It provides you with the information you need where and when you want it.

You may get detailed help when working in a dialog, spreadsheet or graphic view by pressing **F1**. This will launch a Help File window displaying the topic that is related to the window in which you are working. The topic will be explained and links to related topics may also be provided.

RISA Tech, Inc. 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.

Tool-tips

Are you uncertain what a toolbar button is for? Simply hold your mouse pointer over that button without clicking. **Tool-tips** are displayed that will explain what the button will do should you decide to press it.

Tutorials

The comprehensive **Tutorials** guide you through using most features. It is a real-world example of building and solving a model, making changes, and optimizing. This is the best way to quickly get up and running. The 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.

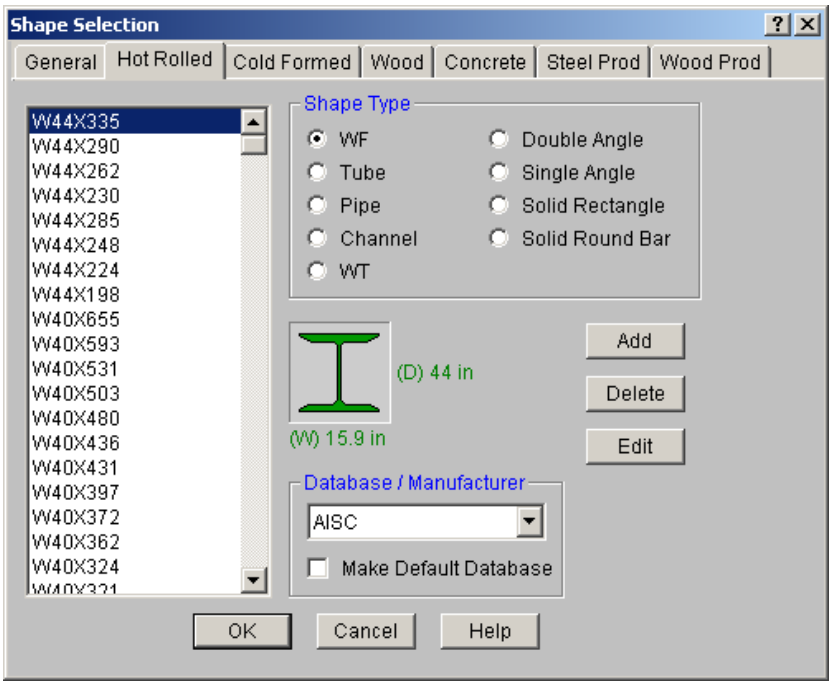
Hot Rolled Steel - Databases

Shapes are organized by **Shape Type** and **Database / Manufacturer**. Common shapes are supported such as wide flanges, tubes, pipes, channels, etc. You may type in the names directly, select shapes from these databases or add your own shapes.

RISA currently supports the following common Hot Rolled steel databases: AISC (United States), Australian, British, Canadian, Chilean, Chinese, European, Indian, and Mexican.

Note:

- Older AISC shapes which are no longer part of the AISC database will be automatically moved / exported to a new AISC_backup database during installation or update.

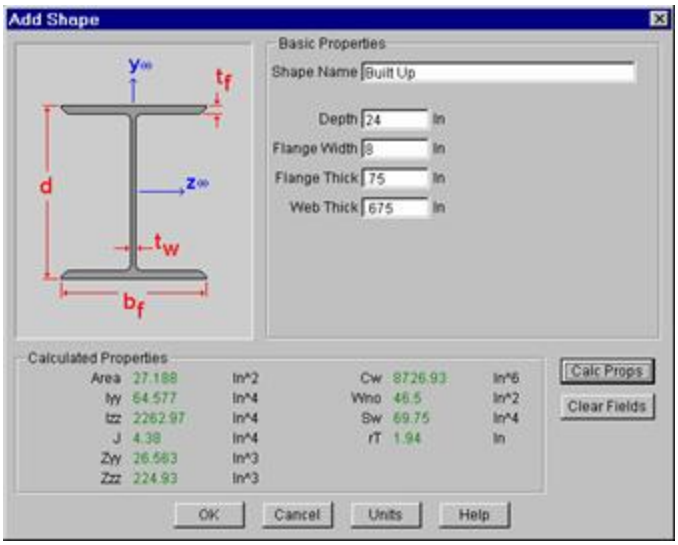


To Select a Hot Rolled Database Shape

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

To Add a Database Shape

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



Note

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

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 a have a design wall thickness of .465" = .93*1/2" (8/16ths) . A HSS32X24X10 would be 32" deep by 24" wide, and have a design wall thickness of 5/8"(10/16ths)

For the Canadian database, tubes also have a "HSS" prefix and the dimensions are all called out in millimeters. British shapes use the prefix "SHS" for square tubes and "RHS" for rectangular tubes.

Note:

- The prefix for older AISC tube shapes is "TU". These shapes reflect the properties that were published with the older AISC 9th edition. As such, these shapes may only exist in the AISC_backup database.
- Tubes using the TU prefix will have a design wall thickness of the nominal wall thickness. The syntax is "TUdepthXwidthXthick", where "depth" is the tube depth, "width" is the tube width and "thick" is the tube thickness in number of 1/16ths. For example, TU16X12X8 would be a 16" deep, 12" wide tube with a thickness of 1/2" (8/16ths).

Pipe Database Shapes

Pipe shapes, which are hollow circular shapes, are entered as on-line shapes. The syntax for these shapes is "PIdiaXthick", where "dia" is the pipe outside diameter and "thick" is the pipe thickness (in inches or centimeters). For example (assuming US Standard units), PI10X.5 would be a 10" diameter pipe with a wall thickness of 1/2".

Channel Database Shapes

Channel shapes are entered with the "C" or “MC” prefix. For example C15X50 would be a valid entry. For Canadian and British shapes, the depth is called out in millimeters and the mass per length is in kg/meter. 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.

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.

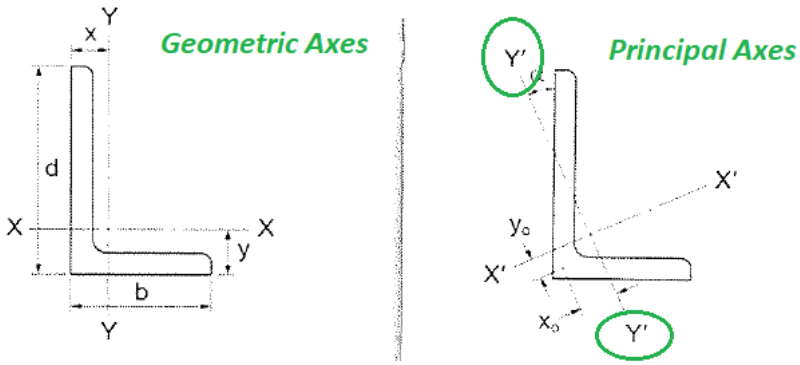
Single Angle Database Shapes

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:

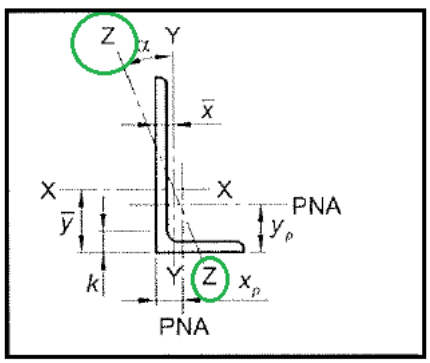
- **rZ** is the Radius of Gyration along the rotated (Principal) vertical axis (local y). The Canadian CSA code calls this variable ry' but RISA calls it rZ based on the naming convention of the AISC 360.

Per CSA S16-16 (11th edition):



-

Per AISC 360-16 (15th edition):



Solid 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

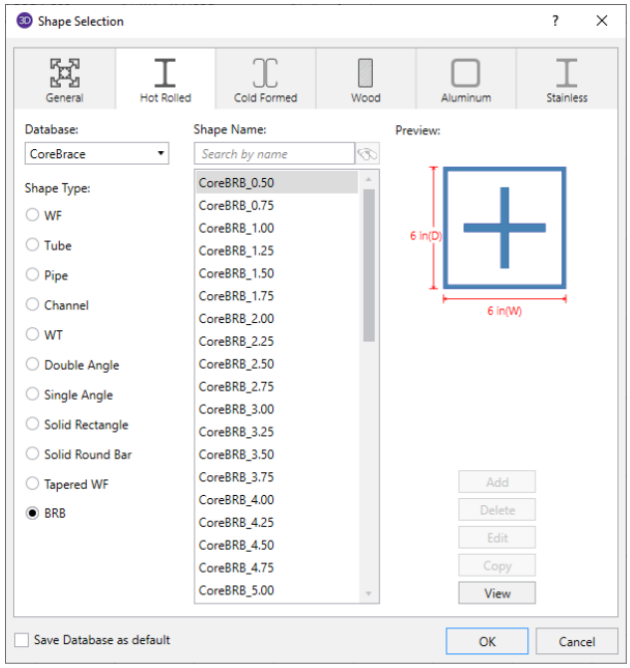
These shapes are defined as on-line shapes. The syntax is "BARdia", where "dia" is the circle diameter. For example (assuming metric units), BAR2 would be a circular bar with a diameter of 2 cm.

Buckling-Restrained Brace Database Shapes

Buckling-restrained brace (BRB) shapes are only available in RISAFloor-RISA3D integration models. They 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 in^2 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.



Note:

- The BRB shape database is only available for RISAFloor/RISA-3D integration models.

Hot Rolled Steel - Design

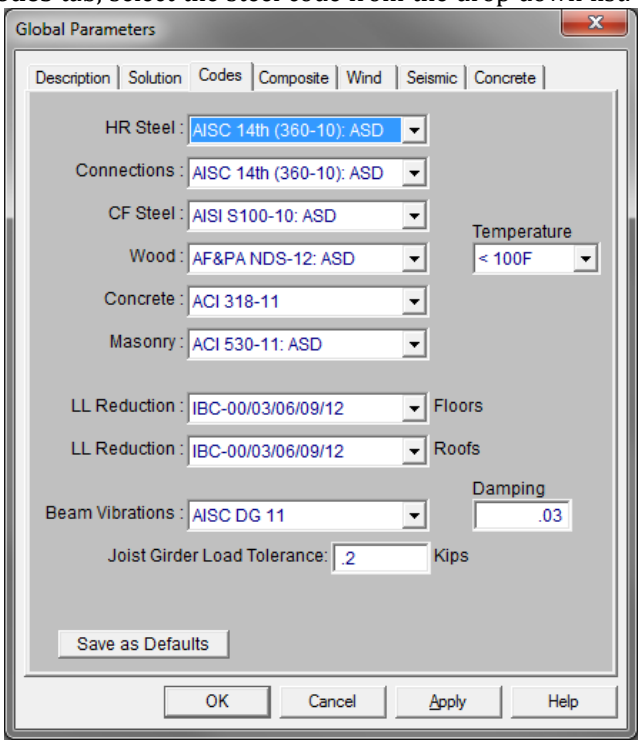
Full code checking member optimization can be applied to standard steel shapes based on the following codes:

- United States:
    - AISC 360-16 (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

How To Apply a Steel Design Code

1. On the **Model Settings** - **Codes** tab, select the steel code from the drop down list.



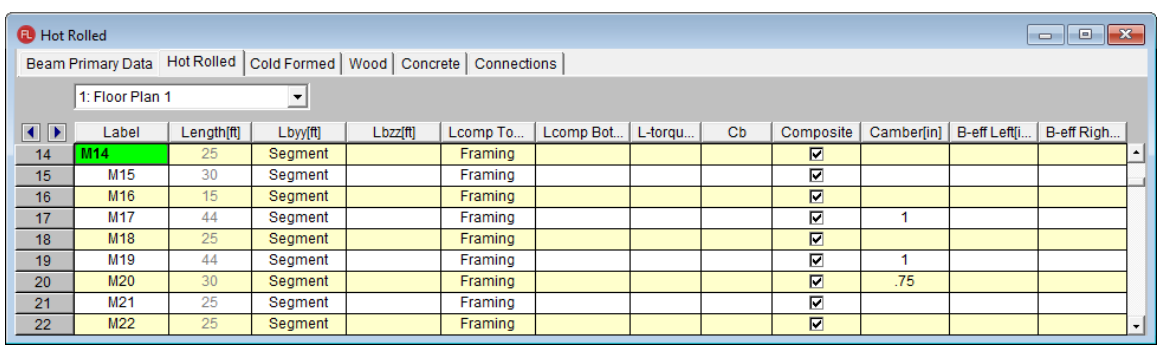
2. On the **Hot Rolled** tab of the **Beams Spreadsheet** or the **Steel** tab of the **Columns Spreadsheet**, enter the appropriate bracing information and factors.

Note:

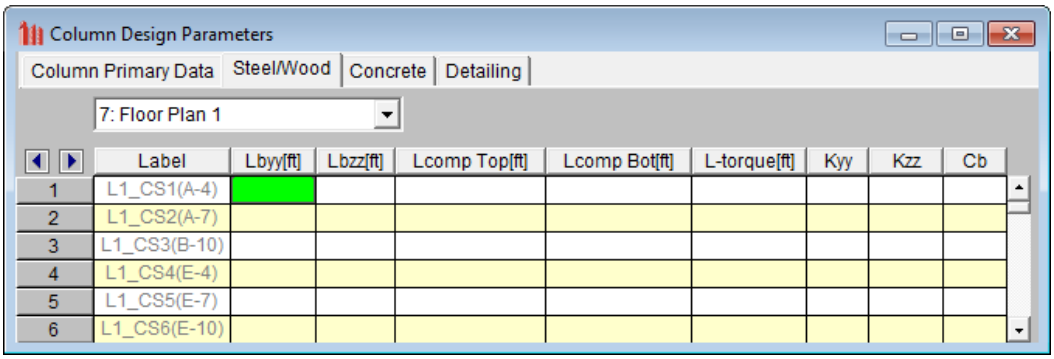
- For code checking to be performed on a member, the member must be defined with a database shape or shape group on the **Beams**, **Columns**, or **Column Stacks Spreadsheet**.

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 **Beams** or **Columns** on the **Spreadsheets** menu and then selecting the **Hot Rolled** or **Steel/Wood** tab. These parameters are defined for each individual member and may also be assigned graphically. See Modifying Beam Design or Modifying Column Designto learn how to do this.



	Label	Length[ft]	Lbyy[ft]	Lbzz[ft]	Lcomp To...	Lcomp Bot...	L-torqu...	Cb	Composite	Camber[in]	B-eff Left[i...	B-eff Righ...
14	M14	25	Segment		Framing				☑			
15	M15	30	Segment		Framing				☑			
16	M16	15	Segment		Framing				☑			
17	M17	44	Segment		Framing				☑	1		
18	M18	25	Segment		Framing				☑			
19	M19	44	Segment		Framing				☑	1		
20	M20	30	Segment		Framing				☑	.75		
21	M21	25	Segment		Framing				☑			
22	M22	25	Segment		Framing				☑			



The pull down list at the top of the spreadsheet allows you to toggle between floors.

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. The Label field for column members is dictated by the label entry on the **Column Stacks** Spreadsheet and may not be edited here. If you would like to edit this entry, you must do so on the **Column Stacks** Spreadsheet.

Length

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

RISAFloor Beam Parameters

In RISAFloor, the beam design is performed for each member based on the design parameters defined for that beam. If these values are not entered, they will be automatically calculated based on deck and framing.

When RISAFloor creates a 3D model, all the Lateral Beams that came originally from Floor will have a "link list" that defines those beam parameters for every section along its length. In this way, the unbraced lengths used in RISAFloor should be identical to the ones used in RISA-3D.

RISAFloor Column Parameters

In RISAFloor, the column design is performed for each floor based on the design parameters defined for that floor level. That way, each level may have a different K value, unbraced length, Cb value and such.

When RISAFloor creates the 3D model for lateral analysis, the column information transferred into the RISA-3D model will not necessarily be the same as the values used in the RISAFloor analysis. Instead, the most conservative value for unbraced length will be used for the entire length of that column lift unless a user defined value is entered in RISA-3D.

Member Design Parameters - General

Unbraced Lengths

See the Unbraced Lengths topic.

K Factors (Effective Length Factors)

See the Unbraced Lengths topic.

Sway Flags

See the Unbraced Lengths topic.

Composite

This box is used to indicate whether the beam is to be designed as composite with a Composite Deck (if present). For more information see Composite Beam Design.

Note:

- A beam that is indicated as "composite" AND "lateral" in RISAFloor will not be analyzed or designed as a composite beam in RISA-3D. The beam will be designed as a "non-composite" beam with only the 'bare' beam's section properties considered in the analysis. If the beam size is changed in RISA-3D, the new size will be reflected automatically in RISAFloor. Also, the solution flag will change to yellow indicating that a change has been made post-solution and the model must be solved again. A new solution must be run which will use the new beam size dictated by RISA-3D and the composite beam calculations will be updated accordingly.
- Beams which are specified as Composite will only be designed of there is Composite Deck placed on at least one side of the beam.
- For beams which have Composite Deck on only one side of the beam, but which are specified as Composite, the program is assuming that the actual constructed detail provides sufficient side cover for the studs to allow composite action as an "L" beam. For more detail on this assumption see Composite Beam Design

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

B-eff Left and **B-eff Right** are the effective widths of the slab for composite beam design. If this value is left blank the effective widths will be calculated automatically as the smaller of half the distance to the adjacent beam and one eighth of the beam span. The calculated widths are listed on the **Member Detail** report. See Hot Rolled Steel Design – Composite for more information on how these are automatically calculated by the program.

Camber Input

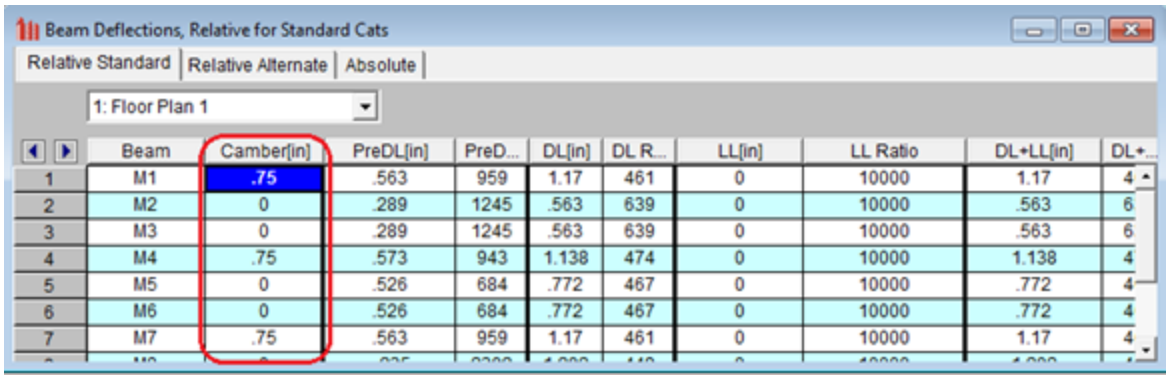
Use this field to manually enter an explicit camber value. If this entry is left blank, the program will design camber based on your Member Design Rules.

After solution, the camber is reported in the **Member Detail Report** and **Deflection Results** spreadsheet. The member detail report displays the camber under the deflections category. The deflection and span ratio results in the code checks section are calculated after the beam has been cambered. The deflection diagram in the report does NOT include the camber.

Deflections: 50.6% Capacity at 22.5ft. (Camber = .75in)

	PreDL	DL	LL	DL+LL	None	None
Deflection (in):	.573	1.138	0	1.138	0	0
Span Ratio	943	474	10000	474	10000	10000

The **Deflection Results** spreadsheet displays a column with the camber specified for each member. Again, the deflection and span ratio results in the code checks section are calculated after the beam has been cambered.



	Beam	Camber[in]	PreDL[in]	PreD...	DL[in]	DL R...	LL[in]	LL Ratio	DL+LL[in]	DL+...
1	M1	.75	.563	959	1.17	461	0	10000	1.17	4
2	M2	0	.289	1245	.563	639	0	10000	.563	6
3	M3	0	.289	1245	.563	639	0	10000	.563	6
4	M4	.75	.573	943	1.138	474	0	10000	1.138	4
5	M5	0	.526	684	.772	467	0	10000	.772	4
6	M6	0	.526	684	.772	467	0	10000	.772	4
7	M7	.75	.563	959	1.17	461	0	10000	1.17	4

Note

- Camber is considered for both composite and non-composite members.
- A beam cannot be both cambered and shored.
- Camber is not considered for continuous beams, cantilevers, or outriggers.

Web Openings

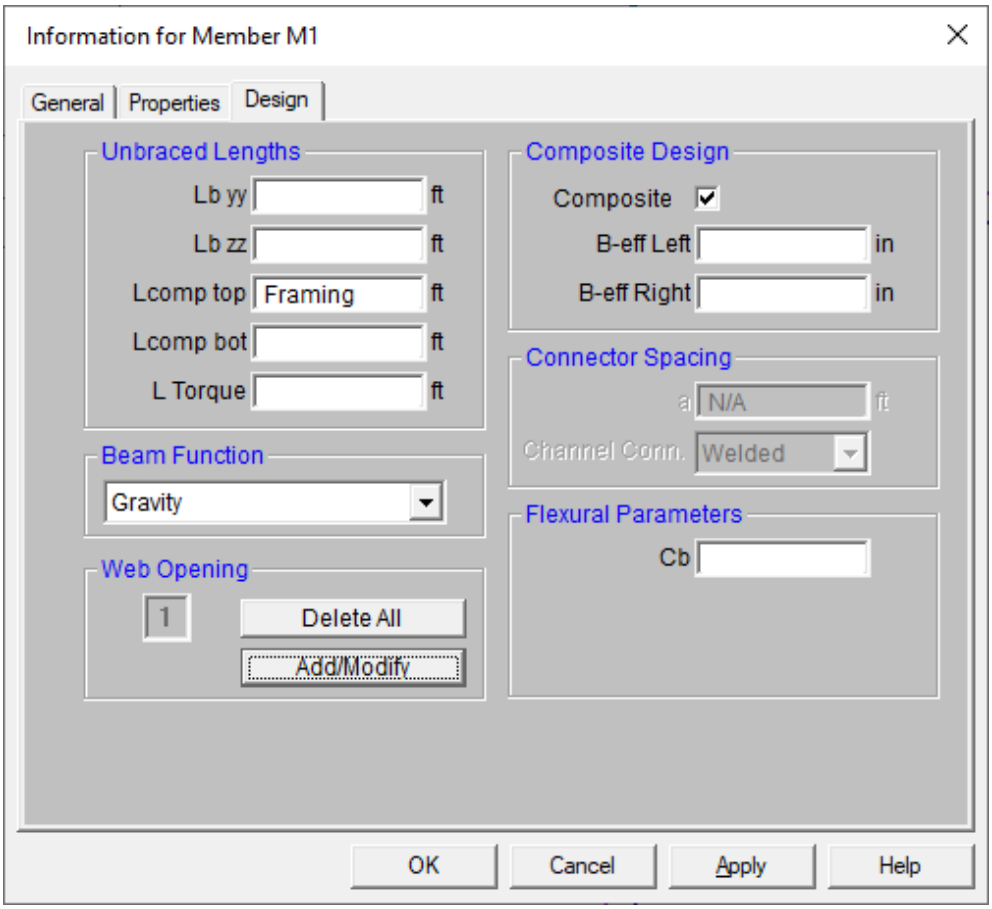
RISAFloor is capable of designing wide flange steel beam members for openings in the web. Both rectangular and circular web openings are supported and may be placed in wide flanges of both non-composite and composite beams. Additionally, reinforcements may be considered along the top and bottom edge to reinforce the web openings.

The design of the web openings follow the guidelines set forth by the *AISC Design Guide #2 - Steel and Composite Beams with Web Openings*.

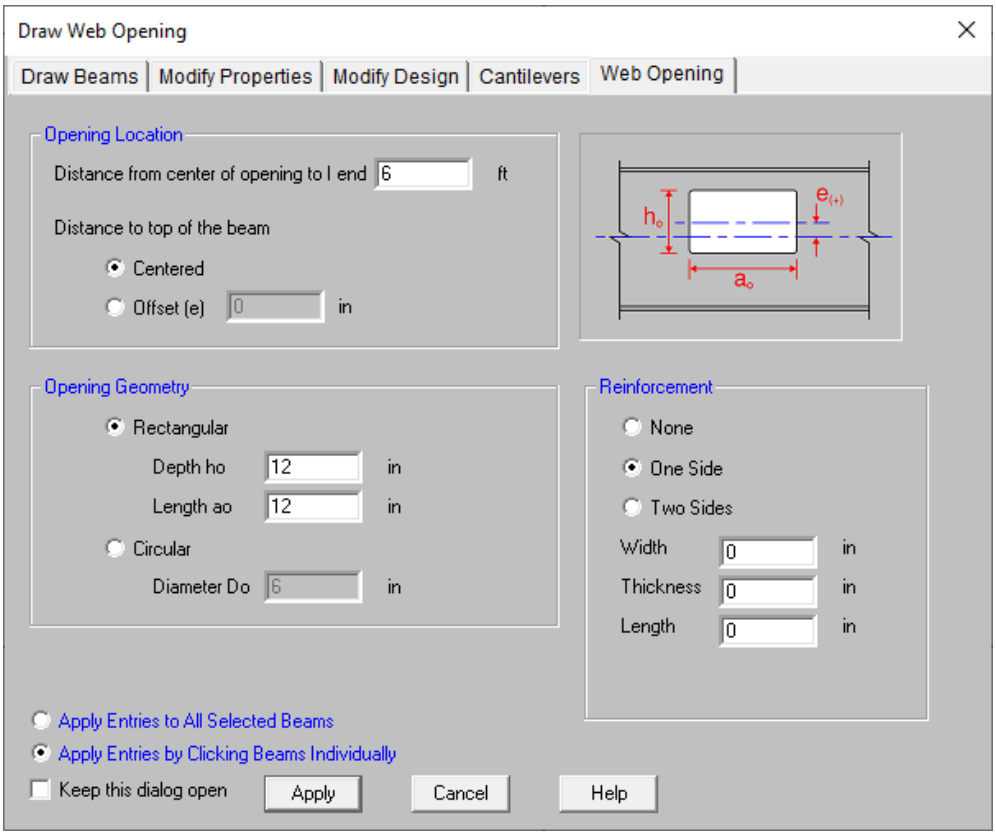
The following Design Codes are supported for Web Openings:

- AISC 360-16 (15th Edition) ASD & LRFD
- AISC 360-10 (14th Edition) ASD & LRFD
- AISC 360-05 (13th Edition) ASD & LRFD

Web openings may be added to an individual beam by double clicking on the beam and navigating to the **Design** tab and clicking on the **Add/Modify** button as shown below:



Web openings may also be added several beams at once by going to the **Modify** --> **Beams** --> **Beams** from the Modify Menu (or click on the **Draw or Modify Beam** button in the Graphic Editing Toolbar) and then navigating to the **Web Opening** tab as shown below:



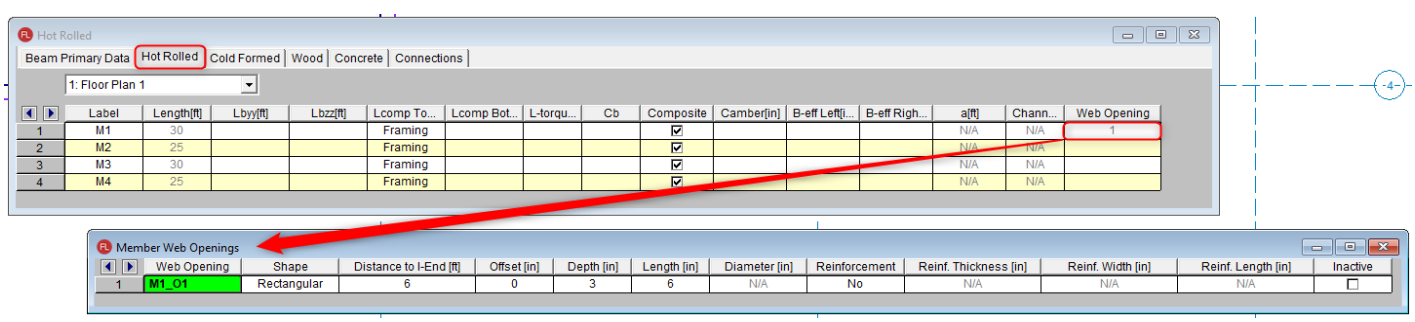
Opening Location - This allows you to specify the location of the web opening along the length of the beam and if it is to be centered on the beam or offset from the center line of the beam.

Opening Geometry - This allows you to choose the shape of the web opening as well as the dimensions of the opening.

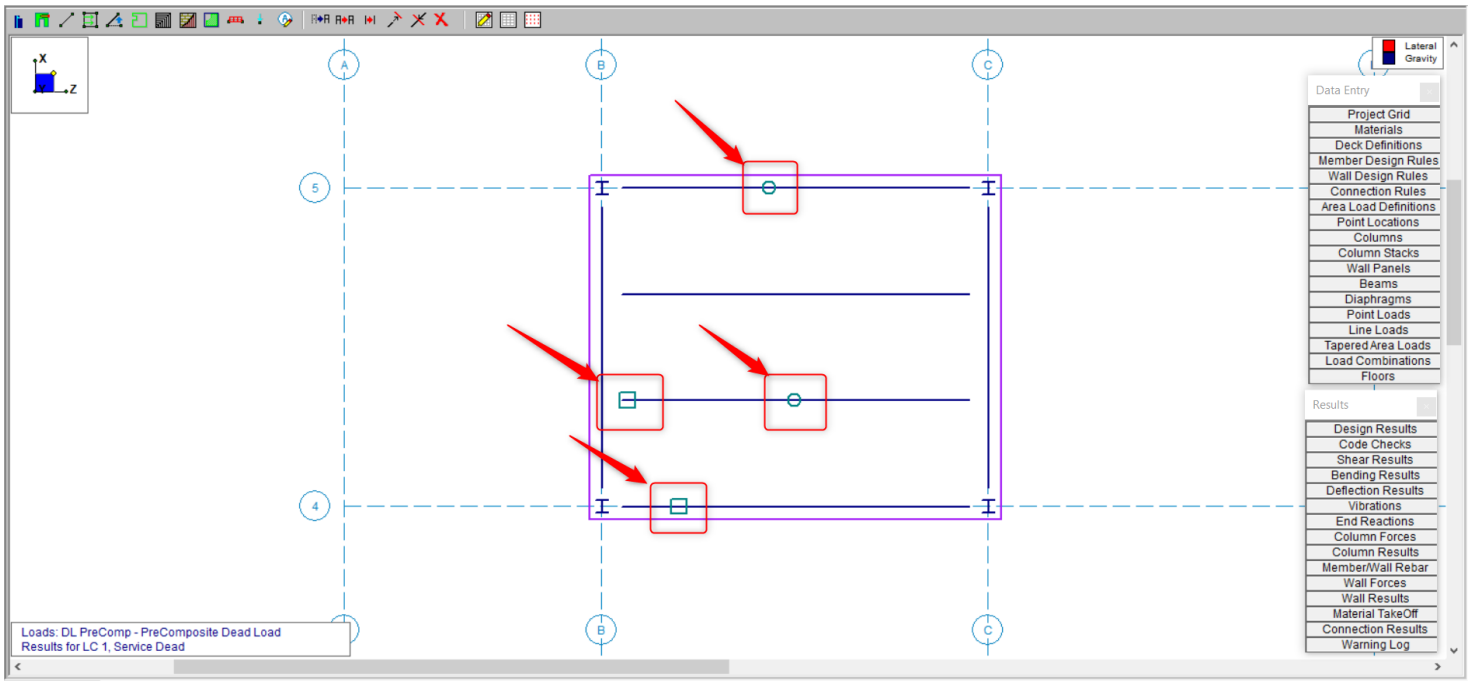
Reinforcement - This allows you to specify the reinforcement of the web opening.

- **One Side/ Two Sides** - This specifies if the reinforcements are added to one side, or both sides of the beam web. In both cases, reinforcements are always added top and bottom of the opening.
- **Width** - This is the width of the plate reinforcement that is perpendicular to the beam web.
- **Thickness** - This defines the thickness of the plate reinforcement.
- **Length** - This is the length of the plate reinforcement that is parallel to the beam length.

Alternatively, web openings can be added, modified, or deleted through the **Beams** spreadsheet as shown below:



When a web opening has been defined onto a beam, the plan view will graphically show the location of the web opening as well as the shape (rectangular or circular) as shown below:



Limitations:

- Weld for reinforcement is not checked by the program.
- One side reinforcement is not allowed per Equations 3-33 to 3-36 in DG2 3.7b5
- Material Take-Off does not consider the reduction in the steel beams self-weight due to the web openings in the steel beam.
- Beam stiffness reduction due to web opening is not considered. Therefore, the beam's deflection does not consider any effects due to the web opening.
- No design optimization is performed for web openings. Program performs beam design optimization solely based on the beam's strength and deflection checks independent of web openings.

AISC Design Codes - Member 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. For column members, each segment of a physical column will receive its own calculated value. However, when in RISA-3D via RISAFloor, the *highest* Cm value calculated for a column segment in RISAFloor will be used for the entire physical column in RISA-3D.

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 value is currently not used by RISAFloor for beam members and is in place for use with the future implementation of sloping floor and roof systems. However, Cm values for beam members are "passed" to RISA-3D when accessed via RISAFloor.
- In RISAFloor, beam members are assumed to be part of a non sway frame and subsequently braced against sway.

Cb - Lateral-Torsional Buckling Modification Factor

Cb Factors are described in Chapter F of the AISC code and are used in the calculation of the nominal flexural strength, Mn. If this entry is left blank, it will be calculated automatically for AISC code checks. For column members, each segment of a physical column will receive its own calculated value. However, when working in RISA-3D via RISAFloor, the *lowest* Cb value calculated for a column segment in RISAFloor will be used for the entire physical column in RISA-3D.

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.

Notes:

- 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.

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 13 code, maintain their original stiffness characteristics.

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 may enter an **ASIF** factor on the **Load Combinations Spreadsheet** to allow the increase for a specific load combination. The ASIF factor is then applied to the allowable stresses in accordance with section A5. The ASIF factor also is applied to the Euler values (F'e).

Note

- If the allowable stress increase is being used, the final code check value should still be compared to '1.0'.

All Other US Codes

Setting the **ASIF** factor to a value greater than '1.0' will not cause the capacities to be increased by that factor. However, setting the **ASIF** factor to a value greater than '1.0' is used as a flag to use of the seismic compactness criteria of Table I-8-1 of AISC 341-05 Seismic Provisions of Steel Buildings. Specifically we will use the limiting width-thickness ratios from this table for capacity calculations (for compression flange local buckling for example). In these cases, we use Table I-8-1 of AISC 341-05 as opposed to Table B4.1 of AISC 360-05.

AISC Design Codes - Limitations

AISC 360-16 (15th Ed.)

Wide Flanges:

- Flexural torsional buckling per Chapter E is included when Ltorque is greater than either Lbyy or Lbzz.
- 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 Ltorque is greater than either Lbyy or Lbzz.
- For AISC 360-05, flexural torsional buckling per Chapter E is only included when Ltorque > Lbyy and Ltorque > Lbzz.
- 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 Qa 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*Fy). For a complete and thorough treatment of channel code checks, refer to the LRFD specification.

RE Shapes - Rectangular bar members (on-line shapes) are assigned allowable bending stresses for the yielding limit state only. Lateral torsional buckling is not considered because the ASD code doesn't directly address this for rectangular shapes. The strong axis bending stress is assigned as 0.66*Fy and the weak axis bending stress is assigned as 0.75*Fy. If you have a case where lateral torsional buckling may govern, you should use the LRFD code, since the LRFD code does address this limit state.

LRFD 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.

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.

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"

AISC Design Codes - 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. 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/tw exceeds the limiting criteria listed in Table B5.1. This means Chapter G (plate girders) governs.

"Compressive stress fa exceeds F'e (Euler bucking limit)"

The axial compressive stress for the member is greater than the Euler buckling stress (per ASD criteria), meaning the member is unstable. A code check can not be calculated.

*"Tube depth>6*width (Sec F3-1) where width=bf-3*tw, Sec B5-1"*

A tube is failing to meet the depth/width requirements laid out in Section F3-1 of the ASD code. The depth of the tube is the full nominal depth, which the width is taken as the full width minus 3 times the thickness. Section B5-1 specifies this calculation for the width when the fillet radius is not known.

"Tee or Channel fails on Table A-B5.1 (Appendix B)"

This message appears for ASD code checking when Appendix B calculations are being done for a Tee or Channel shape and the shape fails to meet the requirements of Table A-B5.1, Limiting Proportions for Channels and Tees.

"Pipe diameter/thickness ratio exceeds 13,000/Fy (App. B)"

This message appears when Appendix B calculations are being done for a pipe shape and the diameter/thickness ratio exceeds the limit of 13000/Fy specified in Section B5-b for ASD, Section B5-3b for LRFD.

"KL/r > 200 for compression member"

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 may override this in the **Application Settings** on the **Tools** menu.

"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.

Canadian Design Codes - Member 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 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, RISAFloor 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 RISAFloor calculate it.

w2 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, RISAFloor 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 RISAFloor calculate it.

Canadian Design Codes - 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.

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_{el}/t exceeds $1014/(F_y)^{1/2}$.

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

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 Design Codes - Special Messages

When a code check is not performed for a particular member a message explaining why a code check is not possible will be listed instead of the code check value. You may click the cell that contains the message and look to the status bar to view the full message. Following are the messages that may be listed specifically for the Canadian Code:

"Tension member with L/r ratio > 300 (CAN Sect. 10)"

The maximum L/r ratio for this tension member exceeds the limit shown in Section 10.2.2 of the CAN/CSA S16.1-94 code.

"KL/r > 200 for compression member (CAN Sect. 10)"

The maximum L/r ratio for this compression member exceeds the limit shown in Section 10.2.1 of the CAN/CSA S16.1-94 code. You may override this check in the **Application Settings** on the **Tools** menu.

"Can't do unity check for slender compression member"

Compression strengths are not calculated for shapes that contain elements where the width to thickness ratios are classified as “slender”.

"Can't do unity check for flexural member with slender web"

Flexural strengths are not calculated for shapes which have a web depth to thickness ratio classified as “slender”.

"Currently can't do unity check for member with slender flanges"

Flexural strengths are not calculated for shapes which have a flange width to thickness ratio classified as “slender”.

"Can't do unity check for single angles in compression"

Compression strengths are not calculated for single angle members.

British Design Code - Member 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 Coefficients (Interactive Bending Coefficients)

Section 4.8.3.3.4 of the British code describes the m coefficient. If these entries are left blank, RISA will calculate them. The m value is influenced by the sway condition of the member and is dependent on the member's end moments, which will change from one Load Comb to the next. It's a good idea to leave these entries blank and let RISA calculate them.

m-LT Coefficients (Bending Coefficients)

This coefficient is discussed in Section 4.3.6.6 of the BS5950-1:2000 code and is used in the calculation of the flexural strength. If this entry is left blank it will be calculated automatically. This value also is impacted by the member's sway condition and is dependent on the member's end moments so it may be a good idea to let it be calculated internally. An exception to this would be for cantilever members in sway frames, this value should be 1.75, and it will be automatically calculated as 1.0. This will be addressed in a future program version.

British Design Code - 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.

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 Codes - Member 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 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. 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 CmLT 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 Coefficients (Interactive Bending Coefficients per the 1992 EuroCode)

Section 5.5.4 (7) of the EuroCode describes the Bm coefficients. If these entries are left blank, RISA will calculate them. The Bm value is influenced by the sway condition of the member and is dependent on the member's end moments, which will change from one Load Comb to the next. It's a good idea to leave these entries blank and let RISA calculate them.

B_m-LT 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.

$$C_1 := \frac{\sqrt{\sqrt{k_1} \cdot A_1 + \left(\frac{\left(1-\sqrt{k}\right)}{2} \cdot A_2\right)^2} + \frac{\left(1-\sqrt{k}\right)}{2} \cdot A_2}{A_1}$$

López, Yong, and Serna Eqn. (8)

$$C_2 := 0$$

$$C_3 := 0$$

Where:

$$A_1 := \frac{M_{max}^2 + \alpha_1 \cdot M_1^2 + \alpha_2 \cdot M_2^2 + \alpha_3 \cdot M_3^2 + \alpha_4 \cdot M_4^2 + \alpha_5 \cdot M_5^2}{\left(1+\alpha_1+\alpha_2+\alpha_3+\alpha_4+\alpha_5\right) \cdot M_{max}^2}$$

López, Yong, and Serna Eqn. (10)

$$A_2 := \left| \frac{M_1 + 2 \cdot M_2 + 3 \cdot M_3 + 2 \cdot M_4 + M_5}{9 \cdot M_{max}} \right|$$

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 that 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 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 ≤ 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 Design Codes - Limitations

Tapered Members - Tapered member design per the Eurocode is not supported at this time.

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 that 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. No bending is considered for Eurocodes.

EuroCode Design Codes - 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 Codes - Member 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 Coefficients (Interactive Bending Coefficients)

If these entries are left blank, RISA will calculate them. The m value is influenced by the sway condition of the member and is dependent on the member's end moments, which will change from one Load Comb to the next. It's a good idea to leave these entries blank and let RISA calculate them.

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 <= 40mm	0.34	0.21
	40mm < tf <= 100mm	0.49	0.34
	tf > 100 mm	0.76*	0.76*
Wide Flange h/b <= 1.2	tf <= 100mm	0.49	0.34
	100mm < 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 <= 40mm (Hot Finished)	0.49	0.34
	40mm < 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 Design Codes - 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.

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 Codes - Member 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)

1. If this entry is left blank, RISA will calculate it based on the full length of the member.

2. 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 will assume the alpha_ m used for this particular member to be 1.0.

3. 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 will be 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 will change from one Load Combination to the next.

Australian & New Zealand Design Codes - 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 will be used in a later release.

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

Welded Sections - There is a basic assumption in the program that the steel sections are hot rolled not welded. Therefore, any code provisions that assume additional stresses due to welding a built-up cross section are not specifically accounted for. For the AISC-LRFD codes this generally means that the flange residual stress (Fr) is always taken as 10ksi, as for a rolled shape. The only exception is for Tapered WF shapes where it is always taken as 16.5 ksi, as for a welded shape.

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

Single Angles - In all codes other than AISC 360-05 and 360-10, single angles are only checked for axial loading. Flexural effects are not considered in the code check calculation. Under AISC 360-05 and 360-10 single angles are checked for combined bending and axial about either their principal or geometric axes depending on how their unbraced length has been defined.

- Lateral-Torsional buckling does not apply to Geometric bending angles, and is therefore not checked for them.
- Lateral-Torsional buckling does not apply to minor-axis bending on Principal bending angles, and is therefore not checked for them.
- The provisions of Section E5 of the specification are not considered. The angle is considered to be loaded concentrically, and on both legs.
- Interaction equation H2-1 is used for all the code check on all angles (equal and unequal leg).

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.

Unsupported Member Type in RISAFloor - at this time the pipes will not be designed in RISAFloor. We do hope to add their design to a future version of the program.

Hot Rolled Steel - Composite Beam Design

RISAFloor is capable of providing analysis, code checks, and member optimization for Composite Beams. In RISAFloor a Composite Beam is defined as a hot rolled 'I' section beam with Composite Deck attached to its top flange. Composite behavior is achieved by steel studs which are welded to the top flange of the beam, and which project/embed into the concrete deck. The design of these beams may be performed 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

Note:

- Unless otherwise specified, specific code references below are to AISC 360-10. The general concepts are the same for all codes.

Overview

The construction process of composite beam structures consists of the following steps:

1. Steel frame (including beams) is erected.
2. (Optional) Shoring is placed under composite steel beams, providing temporary full support of the beams along their entire span.
3. Metal Deck or Concrete Formwork is placed on the top flange of the steel beams.
4. Steel Studs are welded to the top of the steel beams (through metal deck, if present)
5. Wet concrete is placed atop the metal deck or concrete formwork, to a depth that completely covers the studs which protrude upwards from the beam's top flange
6. The concrete hardens and reaches its design strength
7. Formwork and Shoring is removed, if present

Pre-Composite Design

If no shoring is provided under the beam during construction then the steel beam itself (without composite action) is required to support all of the following loads simultaneously, prior to the concrete hardening and reaching its design strength:

- Beam Self Weight (SW)
- Deck Self Weight, including wet weight of concrete(Deck SW)
- Construction Dead Load present only prior to the concrete hardening (Const DL)
- Superimposed Dead Load present prior to the concrete hardening, and remaining afterwards (Super DL)
- Other Dead Load present prior to the concrete hardening, and remaining afterwards (PreDL)
- Construction Live Load present only prior to the concrete hardening (Const LL)

During this Pre-Composite phase the bending moment due to the loads listed above is compared against the beam's bending moment capacity according to AISC 360 Chapter F.

If shoring will be provided during construction this must be specified as a property of the deck. Beams which are placed under Shored Deck are not checked for Pre-Composite design, as it is assumed that the shoring will resist the Pre-Composite loads.

The results for pre-composite design are reported in the Member Detail Report under the Non Composite Bending section:

Non Composite Bending: 13.2% Capacity at 13.1667ft for LC 2 (IBC 16-2 Pre)
M = 29.005k-ft Cb = 1.13636 LbTop = 26.3333ft LbBot = 26.3333ft fb = 23.374ksi phi*Mn = 219.295k-ft L-torque = 26.3333ft

Note:

- Load Combinations containing Pre-Composite Load Categories must solved in order to get Pre-Composite Design.

Post-Composite Design

Regardless of whether shoring was used during construction, the composite section resists the post-construction design loads, as defined in the Load Combinations. These include all load categories which can be defined in RISAFloor except Const DL and Const LL.

During the model solution the program determines the required number and placement of studs, such that the beam's bending capacity as calculated according to AISC 360 Chapter I is sufficient to handle the bending moment due to Post-Composite loads.

The results for composite design are reported in the Member Detail Report under the Composite Bending section:

Composite Bending: 56.9% Capacity at 15.5ft for LC 6 (1.2D+1.6L)

Deck:

	Deck	Height (in)	Wr_top (in)	Wr_bot (in)	Pitch (in)	Direction
Left:	Composite	2	7	5	12	Perpendicular
	Verco PLW2-W2 Formlok 4"NW 22ga W2 Formlok, Conc3000LW, 1.5in, 0.75in, 65ksi					
Right:	Composite	2	7	5	12	Perpendicular
	Verco PLW2-W2 Formlok 4"NW 22ga W2 Formlok, Conc3000LW, 1.5in, 0.75in, 65ksi					

Concrete:

	Thickness (in)	f'c (ksi)	wt (k/ft^3)	E (ksi)	b_eff (in)
Left:	4	3	0.11	2085	45.375
Right:	4	3	0.11	2085	45.375

Studs:

Diameter (in)	Height (in)	Rows	Qn (k)	Quantity
0.75	1.5	1	17.2297	20

V' (k)	Total Q* (k)	% Composite
456.5	172.2967	37.74

*Total Q is based on the number of studs between the point of maximum moment and the point of zero moment per AISC 360 Section I3.2d(1).

Composite Section:

Y Top (in)	Cf (k)	Cf' (k)	Itr (in^4)	I-eff (in^4)	I-Ib (in^4)
5.5075	456.5	172.2967	1023.4261	734.6945	710.3391

Mu (k-ft)	Phi	Mn (k-ft)	T max (k)	C max (k)	PNA Y1 (in)	conc a (in)
177.4365	0.9	346.6719	456.5	462.825	1.6847	0.7445

Deck

This section displays the deck type that controlled on each side of the beam, along with the deck properties as defined in the Deck Definitions.

Concrete

This section displays concrete properties of the controlling deck as well as the calculated effective width for each side of the beam.

Studs

This section displays the stud properties of the controlling deck, along with following calculated values:

- Shear Capacity of each stud, **Qn**
- The total number of studs on the beam, **Quantity**
- The total required shear force, **V'**
- The **Total Q** shear force transferred by the studs between the beam and the concrete. This is calculated by multiplying the shear capacity of each stud by the number of studs between the point of maximum moment and the point of zero moment per AISC 360 Section I3.2d(1).
- The utilization of composite action **% Composite**, which is a measurement of how much of the concrete is being utilized.

Note:

- The user can choose "strong" or "weak" position of the studs. Strong position requires notes on the construction drawings to ensure contractor will properly position the studs within the ribs.
- Strong and weak position is not currently considered for the Canadian codes.

Composite Section

This section displays the information used to calculate the composite properties and is very useful for comparison to hand calculations. Items that are reported include:

The distance from the Elastic Neutral Axis to the extreme top concrete fiber, **YTop.**

The Compressive force in the concrete flange required to achieve full composite behavior, **Cf**, along with the actual flange compression force, **Cf'.** For ASD 9th Edition, these values are reported as **Vh** and **Vh'**.

The full transformed moment of inertia, **Itr**, as if the beam were fully composite, along with the effected moment of inertia based on the fact that the beam is only partially composite, **Ieff**.

The Distance from the Plastic Neutral Axis to the top of the steel section, **PNA Y1.**

The moment demand, **Mu** and the nominal moment capacity, **Mn**.

The depth of the effective compression block, **a** in the concrete versus the compression capacity of the full concrete slab, **Cmax**.

The tension capacity of the full steel section in yield, **Tmax**.

Studs and Stud Distribution

You can specify composite beams with uniform or segmented stud distributions using the Model Settings. Both methods of stud distribution will satisfy both the minimum stud spacing and the required composite action due to the moments in the beam. These moments include both the maximum moments in the beam for each load combination and the moments at concentrated load locations.

Note

- The maximum moment will not always result in the maximum required stud density. For this reason, the program calculates the required stud distribution for **each** load combination and retains the maximum requirement for its design values.

Uniform Studs – When a uniform stud distribution is used, the program will calculate the maximum stud density required in the beam and apply that stud density to the entire beam.

Segmented Studs – When a segmented stud distribution is allowed, the program first fills the segments at the ends of the beam. The studs in these segments see a greater percentage of the total shear than the studs towards the mid-span of the beam.

The program will use the fewest number of stud rows possible. Once the number of studs in a segment reaches the number that can be fit into one row, the program will begin to add studs to the next beam segment. This continues until all the beam segments resisting the applied moment are filled. If necessary another row of studs will be added, starting again with the outer segments.

Note

- The maximum number of segments used for stud distribution is five. If a composite beam has more than five segments it will be designed using a uniform stud distribution.

Composite Decks

RISAFloor allows different composite decks on either side of a beam. The slab properties are "left" and "right" where the beam is oriented such that its start point is at the bottom and its end point is at the top. When there are multiple decks on one side of a beam span, the program finds the deck with the most conservative design properties for each side of the beam. This will be based on the deck orientation (parallel or perpendicular), the thickness of the slab, the elastic modulus of the concrete, and the distance from beam flange to slab centroid. Since perpendicular decks receive no additional stiffness from the concrete filled ribs and control stud placement, they are always assumed to control over parallel ones.

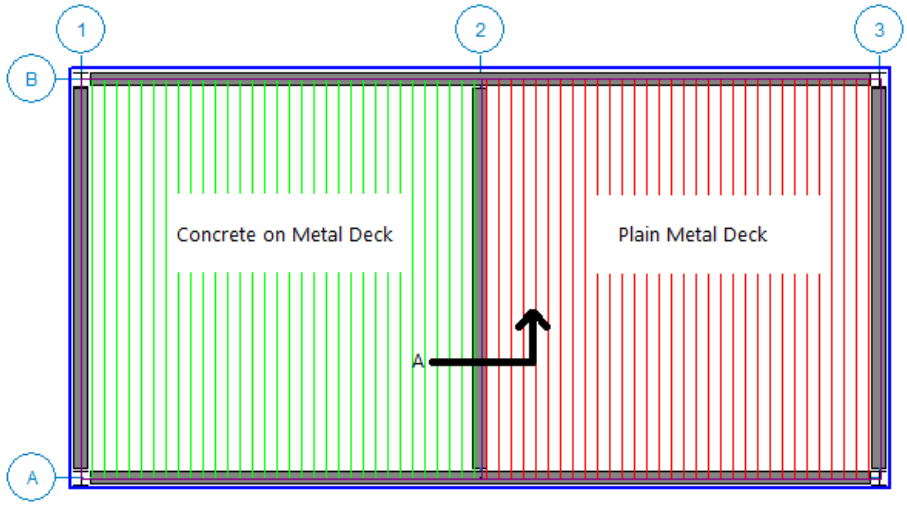
Note:

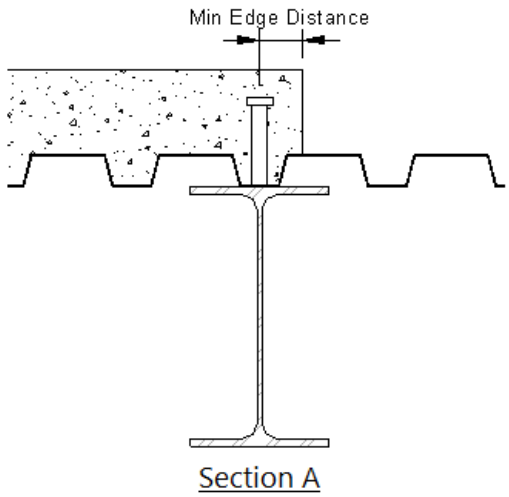
- By default a deck that makes an angle to the beam of 10 degrees or less is considered parallel to the beam. This default may be changed on the **Model Settings** screen.

Beams at Composite Deck Edges

If the deck on one side of the beam is not a Composite Deck, then the effective deck width is taken as zero for that side of the beam. If the overhang distance of a diaphragm edge or opening edge is zero with respect to the beam centerline then the effective deck width is taken as zero in the direction of the opening or diaphragm edge. Therefore the program is assuming that the actual constructed detail provides sufficient side cover for the studs to allow composite action as an "L" beam.

In the Detail below, the Min Edge Distance shown must meet the requirements of AISC 360-10, Section I8.2d. It is the engineer's responsibility to ensure this edge distance is met when actually constructed.





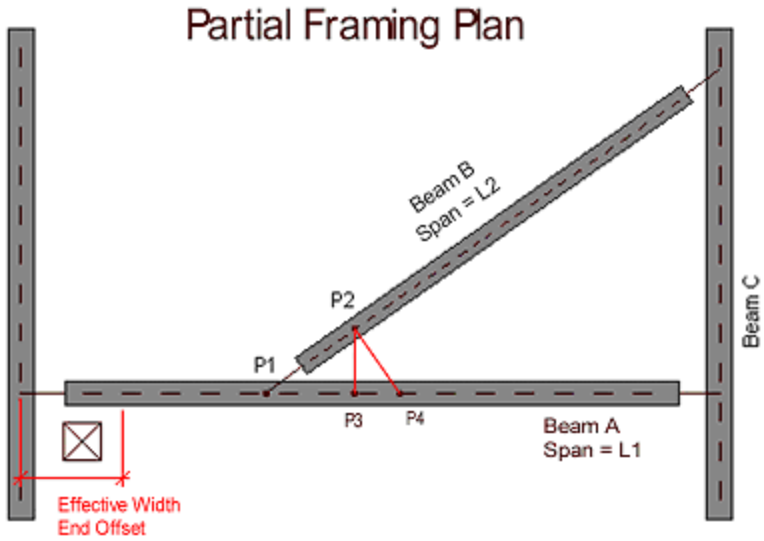
Effective Width Calculation

The effective width is calculated for each side of the composite beam, and will be the lesser of one-eighth of the beam span, one-half of the distance to an adjacent support (a parallel beam or wall), or the distance to the diaphragm edge. In addition, there are two **Model Settings** settings that help determine the way that the effective width is determined.

The first setting is the **Effective Width End Offset** which allows you to ignore elements at the end of the beam.

End Off Set Example 1 - In looking at Beam A in the following figure, the opening beneath the beam does not limit its effective width if it falls within this end offset distance.

End Off Set Example 2 - In looking at Beam B, the effective width beneath Beam B is half the distance between points P2 and P4. This is because point P2 is the offset distance from the end of Beam B, and point P4 is the point on beam A normal to beam B at the offset distance.



The second parameter used to control effective width calculations is the **Orthogonal Beam Angle** which lets you define what angle constitutes orthogonal framing and therefore does not encroach on the effective width. In the figure above, Beam A and Beam C truly are orthogonal and therefore have no affect on each other's effective widths. Beam B, however, is skewed to its supports and may or may not be considered orthogonal. For example, if the **Orthogonal Beam Angle** is set to 45 degrees then the Beam B – Beam C connection, which forms a 60 degree angle, is considered orthogonal. However, the Beam B – Beam A connection forms a 30 degree angle and therefore is not considered orthogonal and effective width calculations will not be performed there (i.e. the effective width above Beam A is zero). If you would like to override this effective width calculation you may do so by entering a value in the **B-eff Left** or **B-eff Right** columns on the **Hot Rolled** tab of the **Beams** spreadsheet.

Moment of Inertia and Deflection

Equivalent / Elastic Moment of Inertia

Depending on the design code chosen, composite beams are defined having an equivalent elastic moment of inertia of the form:

$$I_{equiv} = I_{steel} + \left(\sqrt{\Sigma Q_n / C_f}\right) * (I_{transformed} - I_{steel})$$

Where Qn and Cf represent the total strength of the anchors vs the strength required to achieve 100% composite behavior. Unfortunately, tests have shown that composite beams typically exhibit an effective short term stiffness that is 15% to 30% lower than this value. For this reason RISAFloor has introduced a user defined factor on the equivalent elastic moment of inertia (entered on the Composite tab of the Global Model Settings) to reduce this value. This reduction is used on all composite beams regardless of which code was used to design them. Therefore, if this reduction is not desired, it should be set to 100%.

For the AISC codes (other than the 9th Edition ASD) , the program will use the LARGER value of either this reduced I-Effective, or I-Lower Bound (described below). For the 9th edition ASD the program will use the reduced effective moment of inertia directly and will not consider any potential I-Lower Bound increase.

For the Canadian Codes the program uses the reduction factor times the equivalent I given in the CSA code (which is slightly different than the formula given above) and does not consider I-Lower Bound.

No direct consideration is given to long term effects such as creep.

Lower Bound Moment of Inertia

A lower bound moment of inertia is defined in the AISC commentary that may be used as an alternate to the reduced effective moment of inertia above. Refer to the AISC Specification and Commentary for more information on this formula.

$$I_{lower\ bound} = I_{steel} + A_{steel}(Y_{ENA} - d_3)^2 + \left(\frac{\Sigma Q_n}{F_y}\right) * (2d_3 + d_1 - Y_{ENA})^2$$

This lower bound moment of inertia will be used if it is larger than the reduced effective moment of inertia. But, it will NOT be used if the design code is Canadian or AISC ASD 9th edition.

Total Deflection Design and Beam Camber

The total beam deflection is a summation of the pre-composite deflection (minus beam camber) and the post composite deflection. The pre-composite deflection is based on the unfactored pre-composite loads and the **non-composite** beam properties. The post composite deflection is based on the additional post composite loads and the **composite** beam properties.

Beams are cambered to reduce a portion of the **pre-composite dead load deflection** of the beam. This camber value may be entered directly on the Beams spreadsheet, or can be designed by the program based on the Camber rules in the Member Design Rules spreadsheet.

General Limitations

Negative Moments - All beams with negative moments will be designed as non-composite beams. Typically these will be continuous beams and cantilevers.

Beams with Slender Webs – Beams that have slender webs will be designed as non-composite beams.

Long Term Deflections – No calculations are performed to account for additional deflections due to the creep or shrinkage of the concrete slab.

Neutral Axis within Slab - RISAFloor assumes that the entire concrete slab will be on the compression side of the neutral axis. When the neutral axis is within the slab, the program does NOT adjust the composite section properties to neglect the portion of the concrete slab that is in tension.

Limitations / Assumptions for Canadian Composite Design

Longitudinal Stud Spacing

For the calculation of the shear capacity of a single stud, longitudinal stud spacing (s) is assumed to be the greater of the 6 times the stud diameter or the value of min spacing given in the Model Settings. Refer to CSA S16-09, Clause 17.7.2.3(b).

Non-Standard Rib Heights for Metal Deck

No provision is given in the code for when deck rib height (hd) is not equal to 75 mm or 38 mm. For cases where hd is not equal to one of those values, the qrr value will be interpolated between the two formulas. Refer to CSA S16-09, Clause 17.7.2.4 for more information.

Concrete Pull Out Area for Shear Studs

Refer to CSA S16-09, Clause 17.7.2.4 for more information.

- No subtraction for the pull-out area is being done for beams adjacent to diaphragm edges (i.e. spandrel beams).
- No provisions is given for pull-out area in cases with more than two rows of studs. Therefore, the maximum number of stud rows is limited to two for beams with perpendicular decks.
- The c/c spacing between two rows of studs is taken as 4 times the diameter of the studs for calculating the additional pull-out area in case of two rows.
- When Off-Center studs are chosen, the studs are assumed to be 25% of the average rib depth from the mid point of the rib as shown in Table 5-7b of the CSA 11th edition manual.

Special Behavior (AISC ASD 9th Edition only)

There are number of different code checks that are required for composite ASD beams in the 9th Edition Specification. Unfortunately, that specification is not clear on what they are. An excellent explanation of this procedure is given in **Steel Structures: Design and Behavior** by Salmon & Johnson (1990, Harper Collins Publishers).

Note

- For 9th Edition ASD design, the transformed moment of inertia used to calculate post-composite stresses is based on normal weight concrete per the requirements in the 9th Edition ASD specification. However, the post composite deflection will be based on the weight of concrete specified by the user or the Elastic modulus entered on the Deck Properties Spreadsheet.

Summation of Pre and Post-Composite Steel Stresses vs. Yield

The first check looks at the sum of pre and post-composite stresses. The pre-composite stresses use the section modulus, S, of the steel beam by itself and the pre-composite loading. The post -composite stresses use the effective section modulus, Seff, of the composite section and the post-composite loading. These stresses are listed as fby (sum of actual stresses) and Fby (allowable stress) on the detail report. Generally, Fby is 0.90*Fy.

fby = M(pre-composite) / S + {M(post-composte)} / Seff

This check is based on the following sentences from the ASD specification: "For composite beams constructed without temporary shoring, stresses in the steel section shall not exceed 0.9Fy. Stresses shall be computed assuming the steel section alone resists all loads applied before the concrete has reached 75% of its required strength and the effective composite section resists all loads applied after that time."

All Loads applied to the Composite Section

The second check looks at the sum of pre and post-composite loads against the full composite section. These stresses are listed as Fb and fb on the detail report. Generally, Fb is 0.66*Fy.

fb = Total Moment / Seff

This check is based on the following sentences from the ASD specification: "When shear connectors are used in accordance with Section I4, the composite section shall be proportioned to support all of the loads without exceeding the allowable stress prescribed in Section F1.1, even when the steel section is not shored during construction."

Post Composite Loads vs. Allowable Concrete Stress

The third check looks at the concrete stresses for the post-composite loads ONLY. These are the fc and Fc values shown on the detail report. Generally, Fc is 0.45*f'c.

fc = M(post-composite) / (Itr / ytop * Es/Ec)

This check is based on the following sentences from the ASD specification: "The actual section modulus of the transformed composite section shall be used in calculating the concrete flexural compression stress and, for

construction without temporary shores, this stress shall be based upon loading applied AFTER the concrete has reached 75% of its required strength. The stress in the concrete shall not exceed 0.45f`c."

Troubleshooting Composite Design

Remember, it is possible for a composite beam to be designed as non-composite. This can occur for a number of reasons as described below.

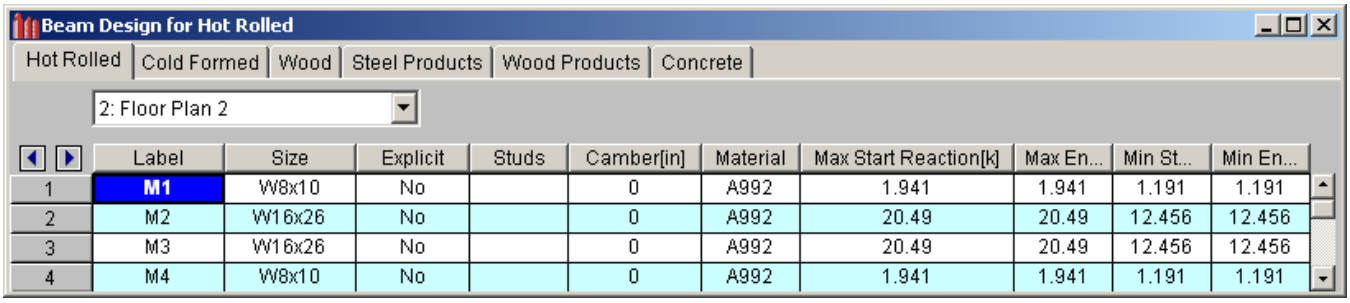
Frequently, a beam member may be designated as composite, but the final design will end up being non-composite. There are still a number of reasons why this may happen.

1. The Deck properties may not be sufficient for composite behavior. Some of these requirements are:
    a. Thickness of Slab must be at least 2.0 in (per ASD I5.1-5 or LRFD I3.5a)
    b. Height of Studs must extend at least 1.5 in into slab above deck (per I5.1-4 or LRFD I3.5a)
    c. Diameter Of Studs should not exceed 3/4 of an inch (per I5.1-3 or LRFD I3.5a)
    d. Rib Width must be greater than 2 in (per I5.1-2 or LRFD I3.5a)
    e. Weight of Concrete must be greater than 90 pcf (per ASD table I4.2 or LRFD I5.1)
    f. Strength of Concrete must be greater than 3 ksi (per ASD table I4.1)
2. There may not be enough room to add the required number of studs to achieve composite behavior. This may happen when a deck forms a relatively steep angle with the beam and there are ribs available for stud placement. The minimum composite behavior may designated on the Model Settings.
3. Load combinations that use a "DLpre" load are considered to be **pre-composite** load combinations. If you do not have a "shored" deck, then you should have at least one pre-composite load combination.
4. Load Combinations that do NOT use a DLpre load, are considered to be **post-composite** load combinations. You MUST have at least one post-composite load combination to get composite behavior.
5. Beams with negative bending are not currently designed for composite behavior.
6. The beam alone may satisfy the loading and deflection requirements without needing any help from composite behavior. In this case, you may force a composite design by clearing the **Use Non-Composite if Optimum** option in the Model Settings.

Hot Rolled Steel - Design Results

Design Results

The **Design Results Spreadsheet** displays the optimized design results for the beam elements and may be accessed by selecting **Designs** on the **Results** menu. The spreadsheet has six tabs: Hot Rolled, Cold Formed, Wood, Steel Products, Wood Products, and Concrete. The pull down list at the top of the spreadsheet allows you to toggle between floors.



	Label	Size	Explicit	Studs	Camber[in]	Material	Max Start Reaction[k]	Max En...	Min St...	Min En...
1	M1	W8x10	No		0	A992	1.941	1.941	1.191	1.191
2	M2	W16x26	No		0	A992	20.49	20.49	12.456	12.456
3	M3	W16x26	No		0	A992	20.49	20.49	12.456	12.456
4	M4	W8x10	No		0	A992	1.941	1.941	1.191	1.191

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

The **Size** 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 or deflection requirements (see Design Optimization), or adding more shapes to the available Redesign List (see Appendix A – Redesign Lists).

The **Explicit** column displays "Yes" if the beam has been locked to an explicit beam size by the user. When you have chosen a specific shape to override the programs automatic redesign, that beam becomes "locked" and will not be automatically redesigned by the program.

Note:

- To "unlock" a beam, you can use the beam – modify tool to assign a shape group. If the model has already been solved, you may optimize a beam by using the Member Redesign dialog. See Member Redesign for more details.
- Live load reduction effects are considered for the applicable values in this spreadsheet.

The **Material** Column displays the material label assigned to the beam.

Studs

The **Studs** column displays number of required studs for composite beams. If only one number is shown, this signifies uniform stud spacing. When segmented spacing is chosen, the number of studs required for each segment will be separated by a comma. Note, the program will specify a maximum of five segments before switching to a uniform stud spacing. The entry will be blank if the beam is not composite or if the program has determined that a non-composite beam will be optimum (see Model Settings - Composite).

Beam Camber

The **Camber** column displays the design camber. This is either the user-entered camber value, or the calculated design camber required per the input Camber Member Design Rules.

End Reactions

The **Max Start & End Reactions** column displays the maximum start and end reactions of the beam for ALL load combinations. If "Show Factored End reactions" in **Model Settings** is left unchecked, these displayed forces are not factored. If it is checked, then the displayed forces will have been multiplied by the factors in the load combinations.

The sign convention assigns positive reactions to downward forces. Negative reactions, if they occur, would indicate uplift.

The **Min Start & End Reactions** column displays the minimum start and end reaction of the beam.

Code Checks

The **Code Checks Spreadsheet** summarizes the code check results for the beams and may be accessed by selecting **Code Checks** on the **Results** menu. The spreadsheet has five tabs: Hot Rolled, Cold Formed, Wood, Concrete, and Wood Products. The pull down list at the top of the spreadsheet allows you to toggle between floors. For more information on Hot Rolled steel design, see Hot Rolled Steel Design or Steel Design – Composite.

Beam Code Summary for Hot Rolled

Hot Rolled | Cold Formed | Wood | Concrete | Wood Products | Web Opening

1: Floor Plan 1

	Label	Size	Explicit	Studs	Cam...	Mate...	Bendin...	Loc[ft]	LC	Defl C...	Loc[ft]	Cat	Shear ...	Lo...	LC	Web Opening Unity Check	Web Opening Misc. Check
1	M1	W12x14	No		0	A992	0.387	15	2	0.693	15	DL+LL	0.051	30	2	0.253	Pass
2	M2	W18x40	No		0	A992	0.877	12.838	2	0.898	12.5	DL+LL	0.241	25	2		
3	M3	W12x14	No		0	A992	0.387	15	2	0.693	15	DL+LL	0.051	30	2	0.418	Pass
4	M4	W18x40	No		0	A992	0.877	12.838	2	0.898	12.5	DL+LL	0.241	25	2		
5	PUR5	W18X60	Yes		0	A992	0.018	15	1	0.036	0	LL	0.005	30	1	0.019	Fail
6	PUR6	W14X211	Yes		0	A992	0.024	15	1	0.036	0	LL	0.01	30	1		

The **Label** column displays the beam label.

The **Size** column displays the beam 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 **Explicit** column displays "Yes" if the beam has been locked to an explicit beam size by the user. When you have chosen a specific shape to override the programs automatic redesign, that beam becomes "locked" and will not be automatically redesigned by the program.

Note:

- To "unlock" a beam, you can use the beam – modify tool to assign a shape group. If the model has already been solved, you may optimize a beam by using the Member Redesign dialog. See Member Redesign for more details.

The **Material** Column displays the material label assigned to the beam.

Studs

The **Studs** column displays number of required studs for composite beams. If only one number is shown, this signifies uniform stud spacing. When segmented spacing is chosen, the number of studs required for each segment will be separated by a comma. Note, the program will specify a maximum of five segments before switching to a uniform stud spacing. The entry will be blank if the beam is not composite or if the program has determined that a non-composite beam will be optimum (see Model Settings - Composite).

Beam Camber

The **Camber** column displays the design camber. This is either the user-entered camber value, or the calculated design camber required per the input Camber Member Design Rules.

Bending, Shear, and Deflection Checks

The **Bending Check** and **Shear Check** columns display the maximum bending check and shear check calculated by the program. This value is equal to the actual bending or shear demand (stress or force) divided by the actually beam resistance (allowable stress or ultimate capacity). You can see the details of these values in the **Bending**

Results or **Shear Results** spreadsheet. This check is calculated at 100 stations along each beam for each load combination and the maximum check is reported. See Results Spreadsheets for more information.

The **Deflection Check** displays the maximum deflection check. This value is equal to the ratio of actual deflection to allowable deflection. You can see the details of these values in the **Deflection Results** spreadsheet. This check is calculated at 100 stations along each beam and the maximum check is reported. See Beam Results - Deflection for more information.

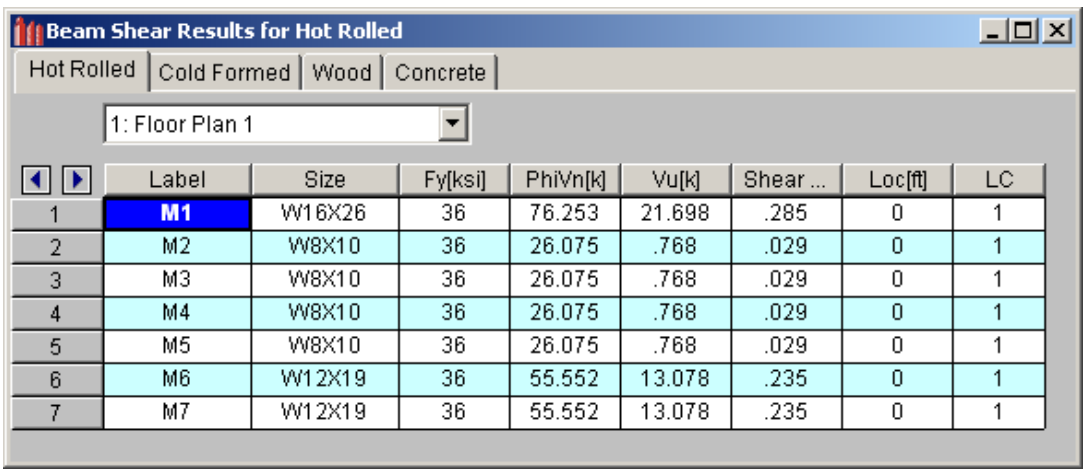
The **Location** columns display the location along the member where the maximum bending, shear, or deflection check occurs.

The **LC** column displays the controlling load combination which resulted in the maximum bending or shear check.

Deflection checks are based on Load Categories (Dead, Live or DL+LL), not Load Combinations. Therefore, the controlling Load Category for deflections is reported in the **Cat** column.

Shear Results

The **Shear Results Spreadsheet** records the shear results for the beam elements and may be accessed by selecting **Shear** on the **Results** menu. The spreadsheet has four tabs: Hot Rolled, Cold Formed, Wood, and Concrete. The pull down list at the top of the spreadsheet allows you to toggle between floors.



	Label	Size	Fy[ksi]	PhiVn[k]	Vu[k]	Shear ...	Loc[ft]	LC
1	M1	W16X26	36	76.253	21.698	.285	0	1
2	M2	W8X10	36	26.075	.768	.029	0	1
3	M3	W8X10	36	26.075	.768	.029	0	1
4	M4	W8X10	36	26.075	.768	.029	0	1
5	M5	W8X10	36	26.075	.768	.029	0	1
6	M6	W12X19	36	55.552	13.078	.235	0	1
7	M7	W12X19	36	55.552	13.078	.235	0	1

The **Label** column displays the beam label.

The **Size** column displays the beam size. When no adequate member could be found from the available shapes, this field will display the text “not designed”. When this occurs, 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 **Fy** column displays the yield strength of the material.

Note:

- Live load reduction effects are considered for the applicable values in this spreadsheet.

Shear Stress and Capacity

The **PhiVn** column displays the calculated allowable shear stress or capacity based on Chapter F of the applicable AISC code. The **Vu** column displays the maximum actual shear stress or shear demand that the member experiences.

The **Shear Check** column displays the maximum shear check. This value is equal to the actual shear demand (stress or force) divided by the actual beam resistance (allowable stress or ultimate capacity). This shear check is calculated at 100 locations along each beam for each load combination. The maximum shear stress, its location, and the controlling load combination are reported. See Results Spreadsheets for more information.

The **Location** column displays the location along the member where the maximum shear check occurs.

The **LC** column displays the controlling load combination which resulted in the maximum shear check.

Bending Results

The **Bending Results Spreadsheet** records the bending results for the beams and may be accessed by selecting **Bending** on the **Results** menu. The spreadsheet has four tabs: Hot Rolled, Cold Formed, Wood, and Concrete. The pull down list at the top of the spreadsheet allows you to toggle between floors.

Beam Bending Results for Hot Rolled

Hot Rolled | Cold Formed | Wood | Concrete

1: Floor Plan 1

	Label	Size	Composite	Studs	Fy[ksi]	Lb Top[ft]	Lb Bot[ft]	Cb	Cm	PhiMn[k-ft]	Mu[k-ft]	Fc[ksi]	fc[ksi]	Bending Check	Loc[ft]	LC	Equation
1	M1	W16X26	Yes	36	36					234.864	162.738	276.48	290.165	.693	15	1	I3.2.a
2	M2	W8X10	No	4	36	1	10	1.136	0	23.949	1.92			.08	5	1	H1-1b
3	M3	W8X10	No	4	36	1	10	1.136	0	23.949	1.92			.08	5	1	H1-1b
4	M4	W8X10	No	4	36	1	10	1.136	0	23.949	1.92			.08	5	1	H1-1b
5	M5	W8X10	No	4	36	1	10	1.136	0	23.949	1.92			.08	5	1	H1-1b
6	M6	W12X19	Yes	26	36					144.027	98.085	200.52	209.564	.681	15	1	I3.2.a
7	M7	W12X19	Yes	26	36					144.027	98.085	200.52	209.564	.681	15	1	I3.2.a

The **Label** column displays the beam label.

The **Size** column displays the beam size. When no adequate member could be found from the available shapes, this field will display the text “not designed”. When this occurs, 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 **Composite** column displays a 'Yes' when the beam is composite and 'No' when the bean is not composite.

The **Studs** column displays number of required studs for composite beams. If only one number is shown, this signifies uniform stud spacing. When segmented spacing is chosen, the number of studs required for each segment will be separated by a comma. Note, the program will specify a maximum of five segments before switching to a uniform stud spacing. The entry will be blank if the beam is not composite or if the program has determined that a non-composite beam will be optimum (see Model Settings - Composite).

The **Fy** column displays the yield strength of the material.

Note:

- Live load reduction effects are considered for the applicable values in this spreadsheet.

Unbraced Lengths

The **Lb top** and **Lb bottom** columns display the unbraced lengths associated with the controlling bending check. The entries will be blank if the beam is composite. See Unbraced Lengths for more information on how these values are calculated.

Steel Design Coefficients

The **Cb** and **Cm** columns display the bending coefficients associated with the controlling bending check. See Hot Rolled Steel Design for more information on how these values are calculated.

Bending Stress and Capacity

The **PhiMn, or Mn/Omega** column displays the calculated allowable bending stress or moment capacity based on Chapter F of the applicable AISC code. The **Mu** column displays the maximum actual bending stress or moment demand that the member experiences. The sign convention is defined so that positive bending will result in tension in the bottom fiber.

The **Fc** column displays the calculated allowable concrete stress for composite beams. The **fc** column displays the maximum actual concrete stress that the composite beam experiences. If the beam is not composite, these entries will be blank.

The **Bending Check** column displays the maximum bending check calculated by the program. This check is equal to the actual demand (bending stress or moment) divided by the beam resistance (allowable stress or moment capacity). This bending check is calculated at 100 locations along each beam for each load combination and the maximum demand (stress or moment), the location, and the controlling load combination are reported. See Results Spreadsheets for more information.

The **Location** column displays the location along the member where the maximum bending check occurs.

The **LC** column displays the controlling load combination which resulted in the maximum bending check.

The **Equation** column displays the code equation that controlled in the calculation of the bending check.

Column Results

The **Column Results Spreadsheet** summarizes the code check results and records the design results for columns and may be accessed by selecting **Column Results** on the **Results** menu. The spreadsheet has four tabs: Hot Rolled, Cold Formed, Wood, and Concrete.

Hot Rolled Steel Column Code Checks

Hot Rolled | Cold Formed | Wood | Concrete

	Stack	Lift	Shape	Code Check	Elev[ft]	LC	Shear Check	Elev[ft]	Dir	LC	phi*Pnc [k]	phi*Pnt [k]	phi*Mn y-y [k-ft]	phi*Mn z-z [k-ft]	Cb	Eqn
1	C1 (C-2)	1	PIPE_3.5	.786	0	1	.000	0		2	29.765	81.324	8.181	8.181	1	HSS 7.1-1
2	C2 (A-2)	1	PIPE_3.5	.786	0	1	.000	0		2	29.765	81.324	8.181	8.181	1	HSS 7.1-1
3	C3 (C-1)	1	PIPE_3.0	.745	0	1	.000	0		2	18.768	67.392	5.913	5.913	1	HSS 7.1-1
4	C4 (A-1)	1	PIPE_3.0	.745	0	1	.000	0		2	18.768	67.392	5.913	5.913	1	HSS 7.1-1
5	C5 (A-3)	1	PIPE_3.0	.745	0	1	.000	0		2	18.768	67.392	5.913	5.913	1	HSS 7.1-1
6	C6 (C-3)	1	PIPE_3.0	.745	0	1	.000	0		2	18.768	67.392	5.913	5.913	1	HSS 7.1-1

The **Stack** column displays the column stack label.

The **Lift** column displays the lift number for the physical column. Lift No. 1 is the lowermost physical column in a stack and the lifts are numbered sequentially moving up the column stack.

The **Shape** column displays the physical column 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).

Code Checks

The **Code Check** column displays the maximum combined axial and bending check calculated by the program. This value is equal to the actual combined axial and bending demand (stress or force) divided by the actually column resistance (allowable stress or ultimate capacity). You can see the details of this value in the subsequent **Axial Resistance (phi*Pnc** and **phi*Pnt)** and **Flexural Resistance (phi*Mn y-y & phi*Mn z-z)** columns of this spreadsheet. This check is calculated at 100 stations along each physical column for each load combination and the maximum check is reported. See Results Spreadsheets for more information.

The**Shear Check** column displays the maximum shear check calculated by the program. This value is equal to the actual shear demand (stress or force) divided by the actual column resistance (allowable stress or ultimate capacity). This check is calculated at 100 stations along each column for each load combination and the maximum check is reported. See Results Spreadsheets for more information.

The **Elev** columns displays the absolute elevation along the column stack where the maximum code check occurs.

The **LC** column displays the controlling load combination which produced the maximum code check and/or shear check.

The **Dir** column displays the column local axis along which the maximum shear check occurs.

Note:

- Code checks for beams are only done on Wide Flanges, Channels and Tube shapes.
- Code checks for columns are only done on Wide Flanges, Tubes and Pipes.

Column Stress and Capacity

The **Fa** or **Pnc/Omega** (ASD) or **phi*Pnc** (LRFD) column displays the calculated allowable compressive capacity based on Chapter E of the applicable AISC code.

The **Ft** or **Pnt/Omega** (ASD) or **phi*Pnt** (LRFD) column displays the calculated allowable tensile capacity based on Chapter D of the applicable AISC code.

The **Fb y-y, Fb z-z** or **Mnyy/Omega, Myzz/Omega** (ASD) or **phi*Mn y-y** and **phi*Mn z-z** (LRFD) columns display the calculated allowable bending moment capacity based on Chapter F of the applicable AISC code.

The **ASD Eqn** or **LRFD Eqn** column displays the code equation used in the calculation of the controlling code check.

Steel Design Coefficients

The **Cb**, **Cmy**, and **Cmz** columns display the bending coefficients associated with the controlling bending check. See Hot Rolled Steel Design for more information on how these values are calculated. The Cm values are not used in LRFD design.

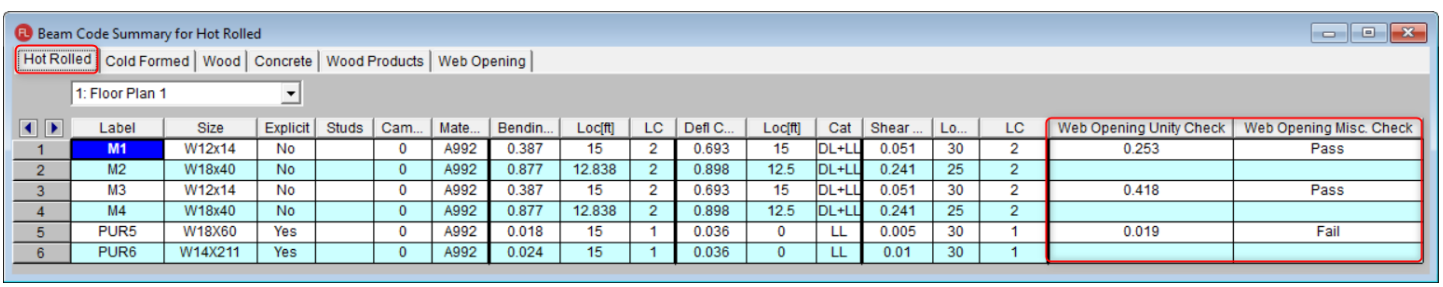
Web Opening Results

Web opening results can be found within the detail report of the steel beam in which the web opening occurs. The results are also provided in a tabular format in the results output spreadsheets.

Web Opening Results Spreadsheet

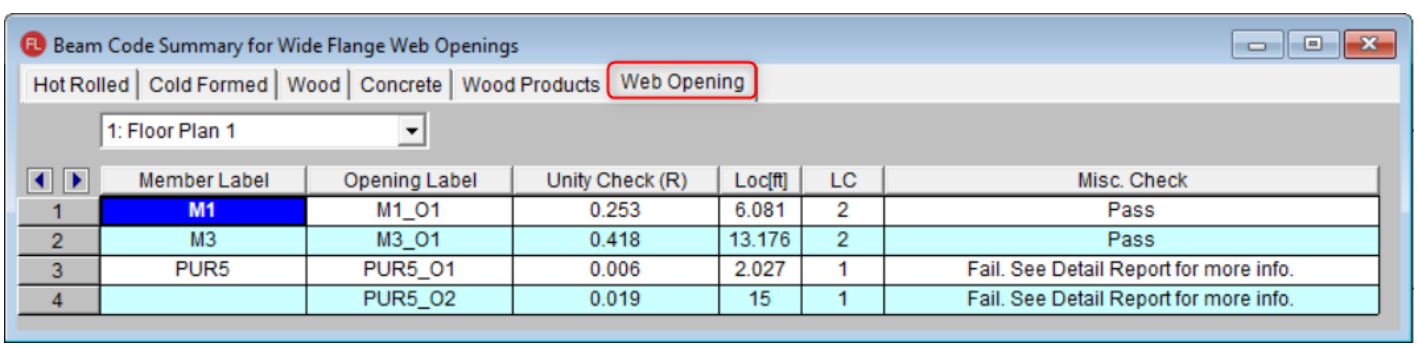
Code Checks Spreadsheet records the web opening results for the beam elements and may be accessed by selecting the **Code Checks** on the **Results** menu. The results are shown in both the **Hot Rolled** tab and the **Web Openings** tab.

The results in the **Hot Rolled** tab will only display the governing web opening unity check and Miscellaneous Check summary for that beam as shown in the image below.



	Label	Size	Explicit	Studs	Cam...	Mate...	Bendin...	Loc[ft]	LC	Defl C...	Loc[ft]	Cat	Shear ...	Lo...	LC	Web Opening Unity Check	Web Opening Misc. Check
1	M1	W12x14	No		0	A992	0.387	15	2	0.693	15	DL+LL	0.051	30	2	0.253	Pass
2	M2	W18x40	No		0	A992	0.877	12.838	2	0.898	12.5	DL+LL	0.241	25	2		
3	M3	W12x14	No		0	A992	0.387	15	2	0.693	15	DL+LL	0.051	30	2	0.418	Pass
4	M4	W18x40	No		0	A992	0.877	12.838	2	0.898	12.5	DL+LL	0.241	25	2		
5	PUR5	W18X60	Yes		0	A992	0.018	15	1	0.036	0	LL	0.005	30	1	0.019	Fail
6	PUR6	W14X211	Yes		0	A992	0.024	15	1	0.036	0	LL	0.01	30	1		

The results in the **Web Opening** tab will show the unity check for all of the web openings in each beam as shown in the image below.



	Member Label	Opening Label	Unity Check (R)	Loc[ft]	LC	Misc. Check
1	M1	M1_O1	0.253	6.081	2	Pass
2	M3	M3_O1	0.418	13.176	2	Pass
3	PUR5	PUR5_O1	0.006	2.027	1	Fail. See Detail Report for more info.
4		PUR5_O2	0.019	15	1	Fail. See Detail Report for more info.

The **Member Label** column identifies the beam associated with the web opening.

The **Opening Label** column displays the label for the web opening.

The **Unity Check** column identifies the unity check calculated by the program.

The **Loc** column displays the governing location of the unity check within the web opening length.

The **LC** column displays the governing load combination which resulted in the maximum unity code check.

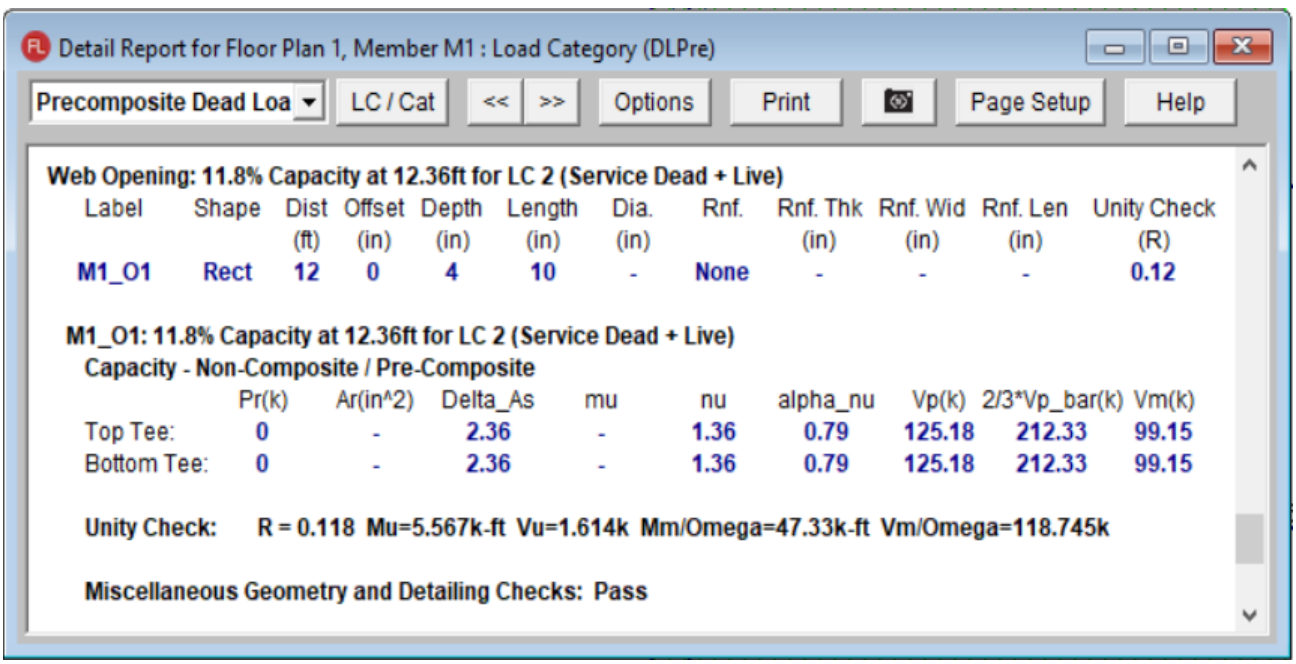
The **Misc. Check** column displays if the web opening passes or fails any miscellaneous checks. If the opening fails a miscellaneous check, the detail report will specifically state which check it fails.

Note:

- The following miscellaneous checks are performed:
  - Compression flange width to thickness ratio check per DG2 3.7a1
  - Opening geometry ao/ho & po check per DG2 3.7a2
  - Opening geometry ho/d, s/d, and nu check per DG2 3.7b1
  - Lateral Buckling check per DG2 3.7a4
  - Check whether tee needs to be further investigated per DG2 3.7a3
  - Edge of opening check and concentrated load check above opening per DG2 3.7b3
  - Minimum spacing check per DG2 3.7b6
  - Reinforcement length requirements per DG2 3.7b5

Web Opening Detail Report

The image below is the results of the web opening calculations within the detail report of a steel beam.



The web opening details can be found towards the bottom of the detail report as shown above. The first row will show the governing unity check at the location is occurs along the length of the beam as well as the governing load combination.

The following rows will list out all of the web openings on that beam along with the web opening's user specified dimensions and reinforcements.

The next section of results goes into greater detail for each web opening and lists critical factors and capacities for the Tee above and below the opening. More information on these factors and calculations can be found in the *AISC Design Guide #2 - Steel and Composite Beams with Web Openings*.

The final line in the web opening check specifies if the miscellaneous geometry and detailing checks meet design guide requirements. If not, the detail report will specify which design guide requirements checks it fails. The miscellaneous checks that the program performs are listed in the Web Openings Results Spreadsheet section in the Help Documentation.

Cardinal Points

Overview

A detailing layer is available in RISAFloor that lets the user set the "true" elevations and locations start/end, top of steel, etc...) for all members. For each member (both columns and beams) you can describe the connection point at member ends.

The areas where member detailing parameters can be added/viewed are:

- RISAFloor
  - Beams Data Entry Spreadsheet>Detailing tab
  - Columns Data Entry Spreadsheet>Detailing tab
  - Model Display Options>Beams/Columns/Walls tab

Note:
For each end of the member, both cardinal point positions and decimal local offsets are used to described the connection location. The location of the cardinal point is plotted as follows:

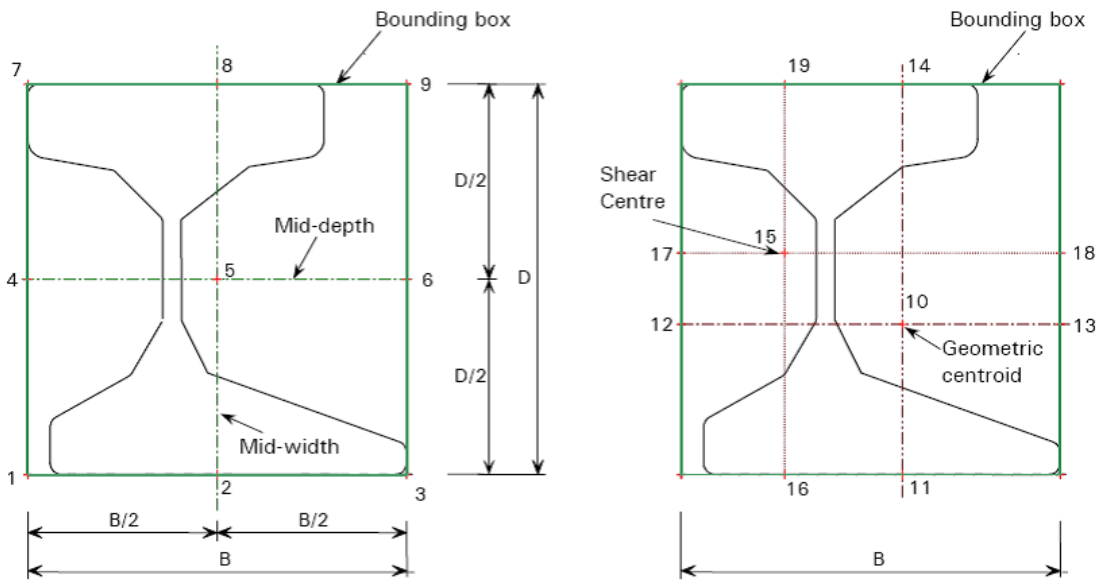


Figure 1: Cardinal point Location and Local Axis

Cardinal point 1:lower left corner of the member section bounding box
Cardinal point 2:lower center point in the member section bounding box
Cardinal point 3:lower right corner of the member section bounding box
Cardinal point 4:mid-depth left point in the member section bounding box
Cardinal point 5:mid-depth center point in the member section bounding box
Cardinal point 6:mid-depth right point in the member section bounding box
Cardinal point 7:upper left corner of the member section bounding box
Cardinal point 8:upper center point in the member section bounding box
Cardinal point 9:upper right corner of the member section bounding box
Cardinal point 10:Geometric centroid of the member section

Note:

- Only cardinal point 1-10 are supported by RISA. You may instead use an offset.

The x, y, and z offsets are based on the local axis of the member. The x local axis is defined along the member from I to J. It coincides with the geometric centroid of the member section. The standard cardinal point positions (1-10), as well as the decimal local offsets are both supported in current detailing definition. If the cardinal point is set, the y and z offset values will be automatically calculated and filled in the data structure. If there is a situation that doesn't match a cardinal point (angle brace as an example) you can also set the y and z offsets directly.

Note:

- The 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 will NOT be automatically updated to cardinal point 8 accordingly.

Detailing Input and Modification:

There are three ways for setting and modifying the detailing layer information. You can use the graphical member modify, the double-click dialog and the member detailing spreadsheet to set the detailing information.

Default

When a member is drawn, by default, x offsets are set to be 0 on both ends. If the member type is beam , by default the y and z detailing offsets will be on cardinal point 8 (top center) for both ends. If the member type is column, by default the detailing offset will be on cardinal point 10 for both ends.

Graphical member modify

A **Member Detailing** tab has been added to the **Graphical member draw/modify** window. The user can choose to “set y, z offset using cardinal point” or “set y, z offsets directly”. When a cardinal point is picked, the value in the “y Offset” and “z Offset” text box will automatically be updated.

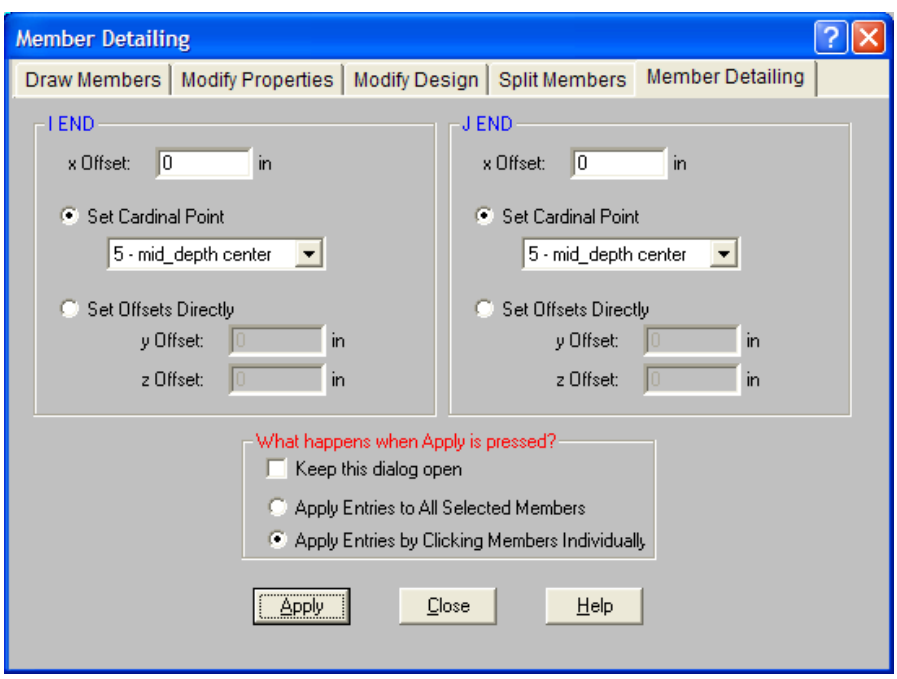


Figure 2.A: Member Detailing Tab in Graphical Member Modify (3D)

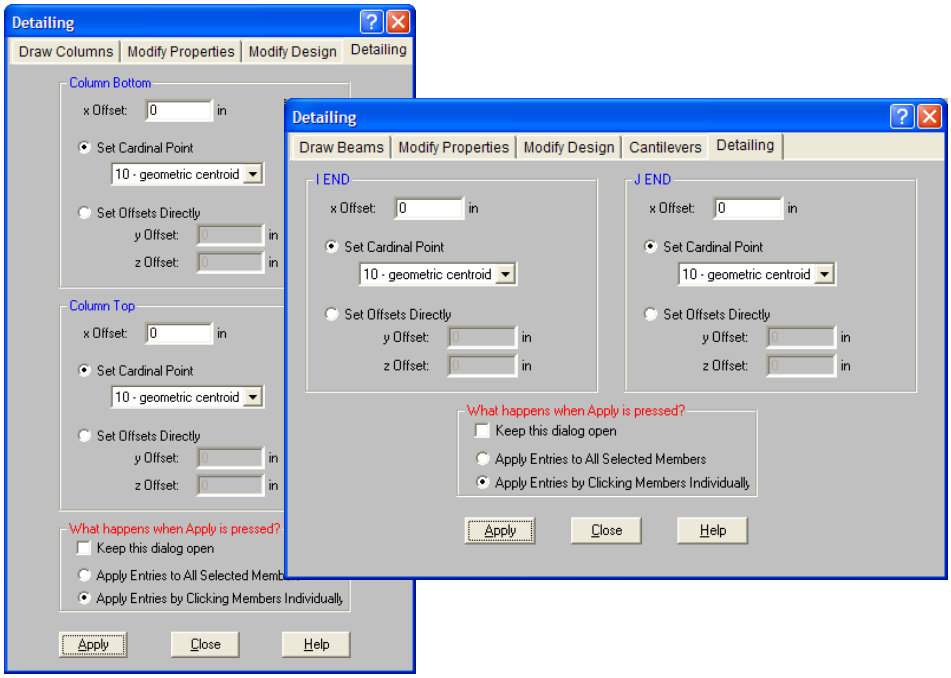


Figure 2.B: Column/Beam Detailing Tab in Graphical Member Modify (FLOOR)

Double click dialog

A **Detailing** tab has been added to the double click dialog for the displaying and modification of the detailing info. The user can choose to **Set y, z Offsets Using Cardinal Point** or **Set y, z Offsets Directly**. When a cardinal point is picked, the value in the **y Offset** and **z Offset** text box will automatically be updated.

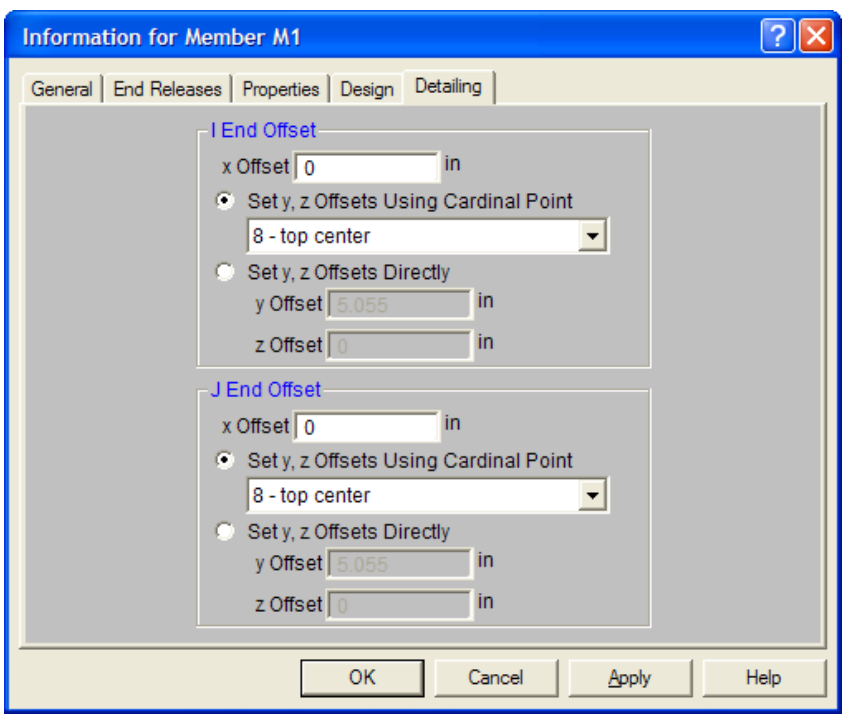


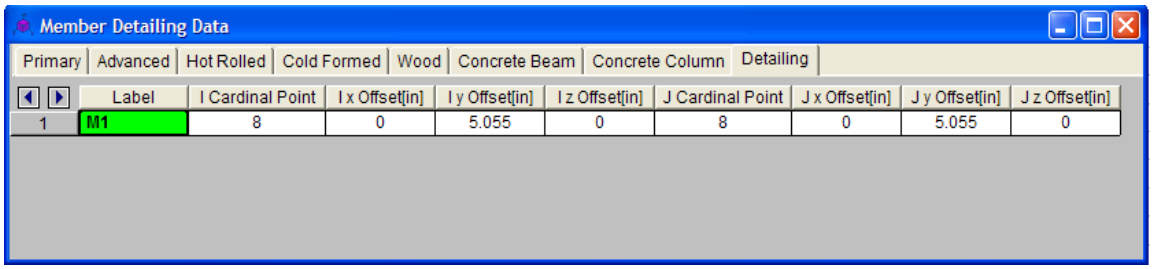
Figure 3: Detailing Tab in double click dialog

Spreadsheets

A detailing tab is available on the **Columns and Beams** 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 a user picks a cardinal point, the y and z offset values will update automatically based on the current shape. When a new y and z offset are input by the user and it doesn't fall on any cardinal point, the "Cardinal Point" will automatically change to "None". When the "Member Detailing" grid is active, a toolbar button is added to recalculate values based on new shapes (Figure 5).

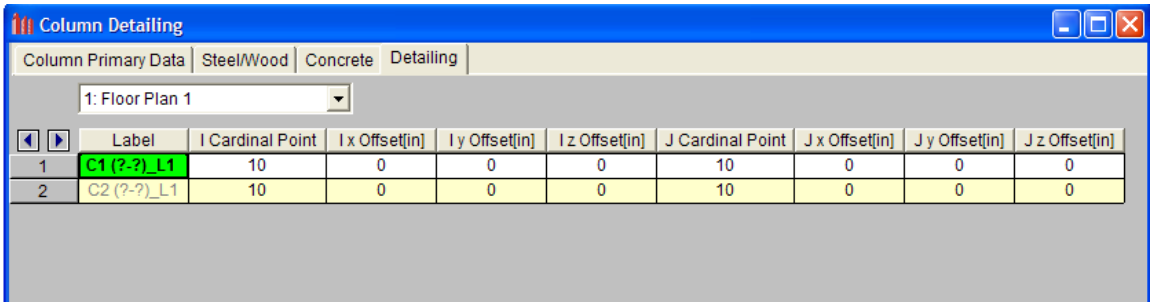
Note:

- When the design shape of the member is changed, the detailing offsets of the member is not automatically updated. The Recalculate tool bar button needs to be clicked to get new y, z offset with respect to the cardinal point picked.

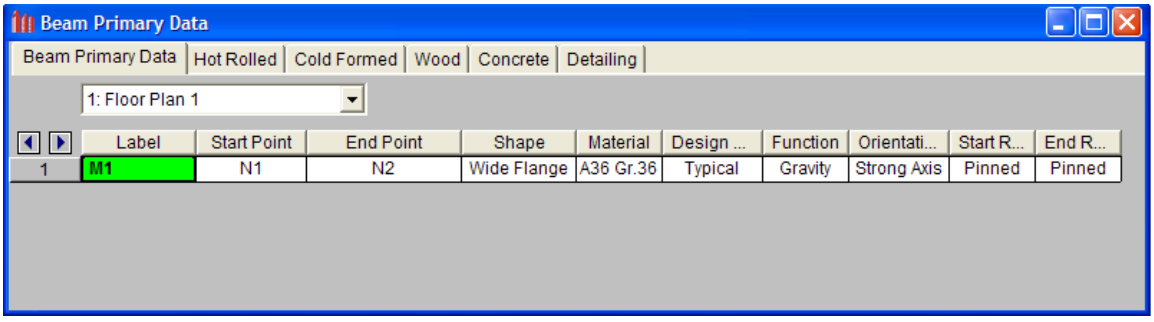


	Label	I Cardinal Point	I x Offset[in]	I y Offset[in]	I z Offset[in]	J Cardinal Point	J x Offset[in]	J y Offset[in]	J z Offset[in]
1	M1	8	0	5.055	0	8	0	5.055	0

Figure 4.A: Detailing Tab in Member Spreadsheet(3D)



	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	C1 (?-?)_L1	10	0	0	0	10	0	0	0
2	C2 (?-?)_L1	10	0	0	0	10	0	0	0



	Label	Start Point	End Point	Shape	Material	Design ...	Function	Orientati...	Start R...	End R...
1	M1	N1	N2	Wide Flange	A36 Gr.36	Typical	Gravity	Strong Axis	Pinned	Pinned

Figure 4.B: Beam/Column Detailing Tab in Member Spreadsheet(FLOOR)

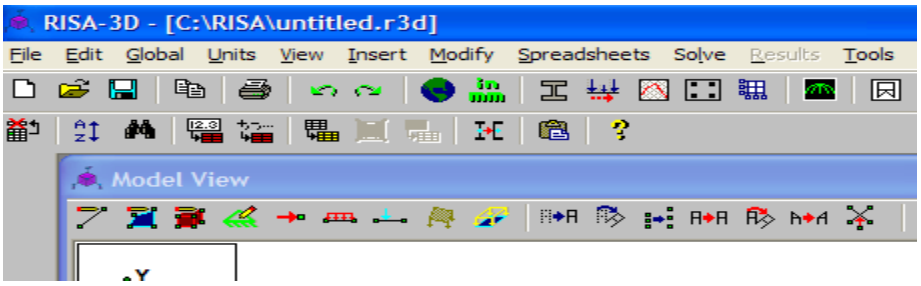


Figure 5: Recalculate Detailing Offset button when Member Detailing Spreadsheet in Active

Visualization of the Detailing Layer

To visualize the model with the detailing layer, go to the plot option dialog box and member tab. Check the "Detailing Info" check box (Figure 6).

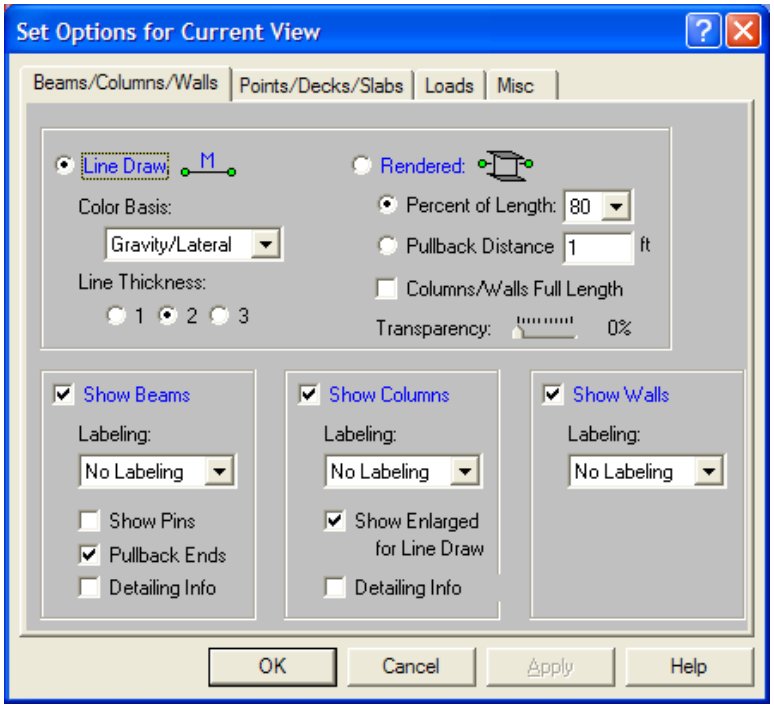


Figure 6: Detailing Info Checkbox in the Model Display Options Window

Note:

- When the **Detailing Info** box is checked, the members will be plotted solely using the x, y and z offsets. The "Analysis Offset" and "Rigid End Offset" will not be 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.

Loads

Loads can be defined and applied to the model graphically or from within the spreadsheets. There are a number of different **Load Types** that may used when loading the model. For more information about the application of each Load Type (Deck Load, Area Load, Line Load, and Point Load), see Loads - Load Types.

As you define the loads in the model, you will notice that each Load Type (deck load, area load, line load, and point load) is broken down into several pre-defined **Load Cases** such as pre-composite dead load, post-composite dead load, non-reducible live load, reducible live load, and such. These 'Load Cases' are the building blocks upon which the more general **Load Categories** are based. Please note that these 'Load Cases' are *automatically* assigned to 'Load Categories'. You will then use these **Load Categories** to define the various **Load Combinations** that you wish to solve. See Loads - Load Combinations for more information.

You may view the loads on your model for each load category or load combination. This is an excellent way to verify the loads applied to the model. See Model Display Options – Loads to learn more about displaying loads in model views.

Self Weight (Gravity Load)

The **Self Weight** of beams, columns, and walls is automatically calculated and included as part of the Pre-Composite Dead Load Category (DLPre). The **Deck/Slab Self Weight**, is likewise calculated. However, to give users more control over this value, the calculation is reported on the Loads tab of the **Deck Definitions Spreadsheet** and may be adjusted using the SuperDL load entry given on that same spreadsheet. See Deck Properties - Loads for more information.

The 'Self Weight' is applied as a uniform distributed load along each beam, column, or wall and as an area load for the deck/slab. The magnitude of the beam and column loads is the cross sectional area times the material weight density. The magnitude of the wall load is wall thickness times the height of the wall times material density.

Note

- If you do not want the program to apply a member's self weight, create that member using a weightless material. See Material Properties for more information.

Load Cases

As you apply your loads, you will notice that each load is broken down into several pre-defined **Load Cases** such as pre-composite dead load, post-composite dead load, non-reducible live load, and reducible live load. These loads are the building blocks of the loads applied to the structure upon which the more general **Load Categories** and **Load Combinations** will be based.

Point Loads, Line Loads, and Area Loads are applied using the following possible **Load Cases**:

Load Case	Abbreviation	Description
Self Weight	SW	Weight of Columns, Beams, Walls. Calculated based on Density defined in Materials spreadsheet.
Pre-composite Dead Load	PreDL	Superimposed Dead Load existing prior to concrete deck hardening.
Post-composite Dead Load	PostDL	Superimposed Dead Load existing after concrete deck hardening.
Live Loads	LL-Non	Non-Reducible Live Load
	LL-Reduce	Reducible Live Load
	LLS-Non	Non-Reducible Live Load for garages,

Load Case	Abbreviation	Description
		public assembly areas, live load magnitudes > 100psf
	LLS-Reduce	Reducible Live Load for garages, public assembly areas, live load magnitudes > 100psf
	RLL-Non	Non-Reducible Roof Live Load
	RLL-Reduce	Reducible Roof Live Load
	SL	Snow Load
	SLN	Non-Shedding Snow Load
	RL	Rain Load
Other Loads	OL1	Other Load 1
	OL2	Other Load 2
	OL3	Other Load 3
	OL4	Other Load 4
Dynamic	Dyn Mass	Seismically Participating Load (for RISA-3D seismic weight calculations). This number should typically be greater than or equal to PostDL.
Vibration Live Load	VL	Realistic Live Load (for floor vibration checks). Often much less than live load for strength design (LL).

The 'Loads Cases' that are specific to decks or slabs are defined within the **Deck Definitions Spreadsheet**. These load cases are described in the table below:

Load Case	Abbreviation	Description
Deck / Slab	Deck SW	Deck or Slab Self Weight (auto-calculated based on Deck Material Type)
Construction Dead Load	Const DL	Comparable to PreDL, but a property of the deck instead of an applied load.
Construction Live Load	Const LL	Live load present prior to concrete deck hardening.
Superimposed Dead Load	Super DL	Comparable to PostDL, but a property of the deck instead of an applied load. Often used to account for flooring materials, fire-proofing.

Load Categories

The 'Load Cases' listed above are automatically assigned to one or more of the **Load Categories** described below. At solution time, the model will be solved against **Load Combinations** that are built from the following **Load Categories**. The load categories and descriptions below are based on the definitions and load combinations listed in various building codes.

Load Category	Includes Addition of Load Cases	Sloped Members Load Direction
DLPre	SW + Deck SW + Const DL + PreDL+ Super DL	Y

Load Category	Includes Addition of Load Cases	Sloped Members Load Direction
LLConst	Const LL	PY
DLConst	Const DL	Y
DL	SW + Deck SW + PreDL + PostDL+ Super DL	Y
LL	LL-Non + LL-Reduce	PY
LLS	LLS-Non + LLS-Reduce	PY
RLL	RLL-Non + RLL-Reduce	PY
SL	SL	PY
SLN	SLN	PY
RL	RL	PY
OL1	OL1	Y,PY,y
OL2	OL2	Y,PY,y
OL3	OL3	Y,PY,y
OL4	OL4	Y,PY,y

Note:

- To investigate what area Load Cases are applied at a particular location, for a give Load Combination you may use the Area Load Query utility.

Modifying Loads

Modifying point loads, line loads, and tapered area loads can be done from within their respective spreadsheets. Typically, the modification of load geometry and magnitude may be performed in this way. Select **Loads** from the **Spreadsheets Menu** and then choose the spreadsheet that you wish to modify. You may then move through different Floors using the drop down list in the spreadsheet.You may use spreadsheet operations to help you modify the loads. See Spreadsheet Operations for more information.

Modifying area loads (non-tapered) can be done graphically. Click the **Draw Area Loads** tool from the **Drawing Toolbar** and select the **Modify Area Loads** tab.

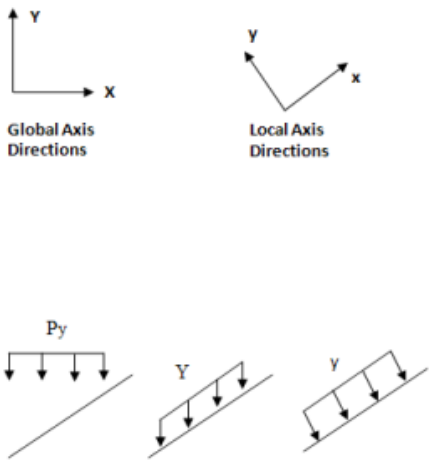
Deleting Loads

Deleting loads may be done manually within the spreadsheets or graphically using the **Delete** button on the **Drawing Toolbar**. Select **Loads** from the **Spreadsheets Menu** and then choose the spreadsheet that you wish to modify or delete. You may then move through different Floors using the drop down list on the in the spreadsheet. You may use spreadsheet operations to help you modify or delete the loads. See Spreadsheet Operations for more information.

Load Direction

The load direction indicates how the distributed loads are applied to the sloped members.

Y- Loads applied in the global Y-axis
PY- Projected load in the global Y-axis
y- Load applied in the local y-axis



Positive loads are in the downward direction.

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 user entered magnitudes reduced by the ratio L/Projected length.

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.

Loads - Load Types

This section discusses the various **Load Types** that are available for applying loads to a model.

Area Loads

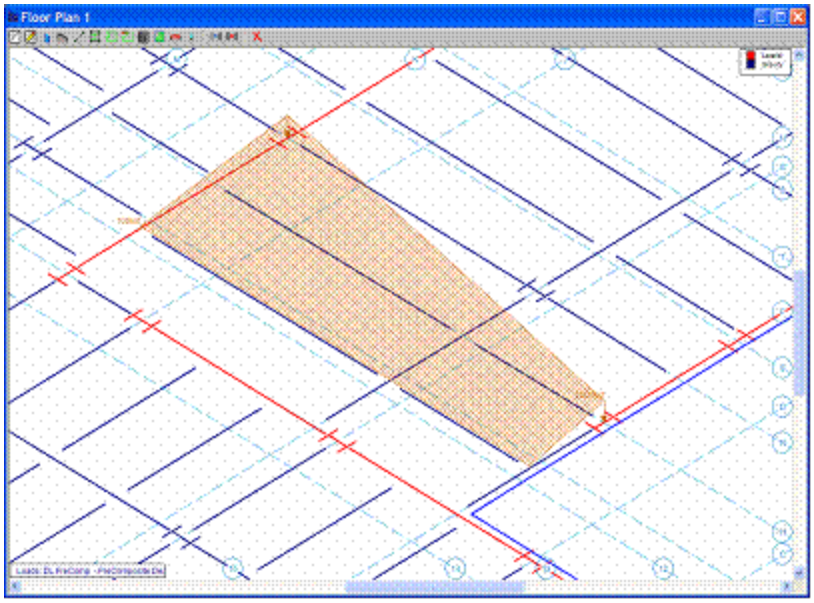
Area Loads that are applied to a floor are automatically attributed to the nearest members that support that area. See Loads – Attribution for more information. This attribution gives you the ability to model all the loading effects on a floor without having to calculate all the loads yourself. The area loads can be uniform or tapered and the loads may be distributed based on one-way or two-way action depending on the deck or slab that supports them.

Note:

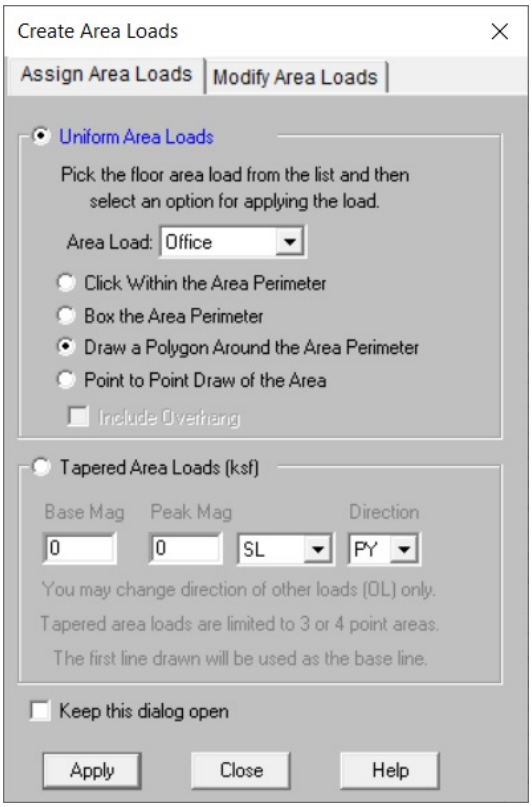
- To investigate what area loads are applied at a particular location, you may use the Area Load Query utility.

Drawing Area Loads

To apply area loads click on the **Drawing Toolbar** to open the **Draw Area Loads Dialog** and select a uniform or tapered area load. For a **Uniform Area Load**, select the Area Load you would like to assign from the drop down list. The area loads that are listed are those that are defined in the **Area Load Definitions Spreadsheet**. Apply the area load by clicking within or boxing the area load perimeter, by drawing a polygon around the area load perimeter, or by clicking from point to point to define the perimeter of the area load.



For a **Tapered Area Load**, enter in the base magnitude, peak magnitude, and load category of the load. Apply the tapered area load by clicking on the grid or existing point locations. The **Base Magnitude** of the tapered load is magnitude of the load between the first two points drawn when defining the load. The **Peak Magnitude** of the tapered load is the magnitude of the load at the third or fourth point entered.



To Apply Area Loads

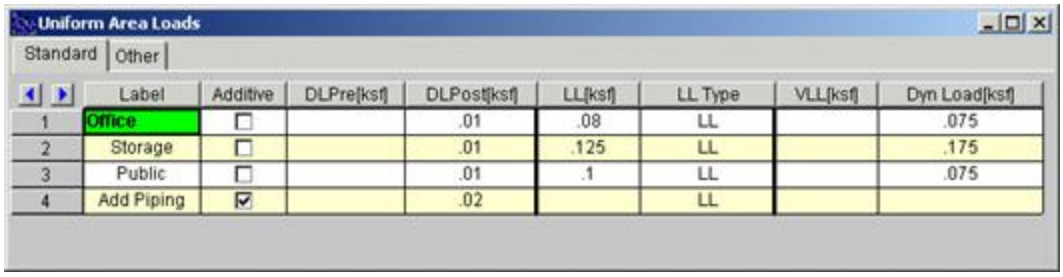
1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Click the **Area Load** button on the **Drawing Toolbar** and define the load in the **Draw Area Loads Dialog**.
3. Click **Apply**.
4. For Uniform Area Loads, apply the load by clicking within or boxing the area load perimeter, by drawing a polygon around the area load perimeter, or by clicking point to point to define the perimeter of the area load.
5. For Tapered Area Loads, click on four grid or point locations to define the area. For three sided areas click on the last point twice.
6. To stop drawing altogether, click the right mouse button or press the Esc key.

Note:

- To apply more loads with different parameters, press CTRL-D to recall the **Draw Area Loads Dialog** .
- You may undo any mistakes by clicking the Undo button.
- The listed area loads are presented in the same order in which they appear in the **Area Load Definitions Spreadsheet**. Therefore, it may be beneficial to re-order this spreadsheet so that the most commonly used items are at the top of the list.
- You may change directions of Tapered Area Loads only.
- Include Overhang box is not available for elevated slab floors.
- Include Overhang box only works for 'Box the Area Perimeter' and 'Draw Polygon Around the Area Perimeter' options on beam supported floors.
- 'Point to Point Draw of the Area' is the only option to manually assign area loads to elevated slab floors.

Area Load Definitions Spreadsheet

The **Area Load Definitions Spreadsheet** defines the uniform area load magnitudes and may be accessed by selecting **Loads ▸ Area Load Definitions** from the **Spreadsheets Menu**. While the load magnitudes are defined in this spreadsheet, they can only be assigned graphically.



Uniform Area Loads

Standard | Other

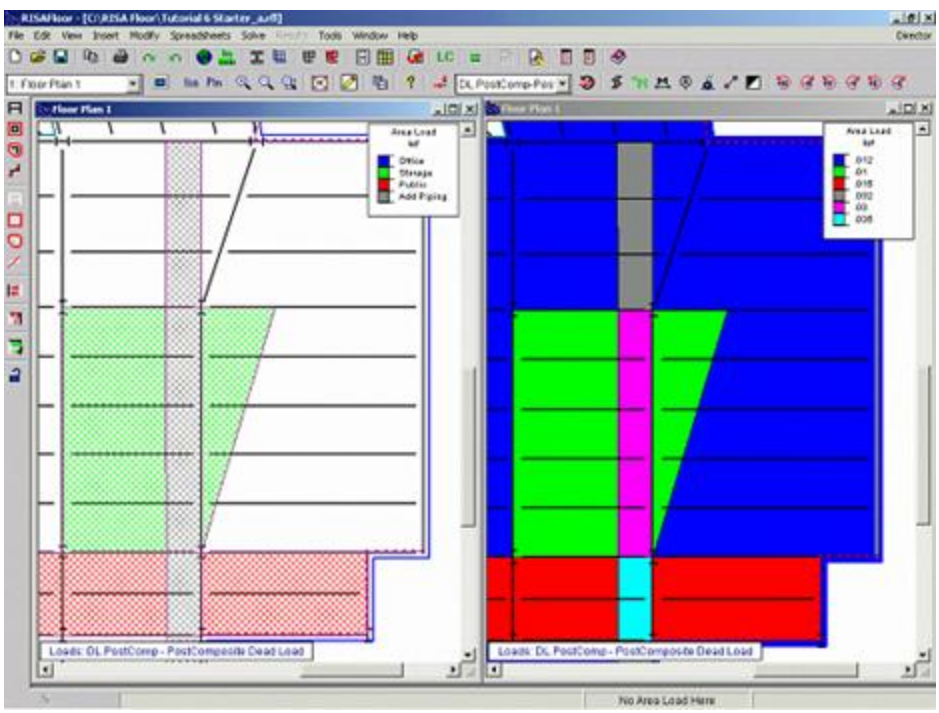
	Label	Additive	DLPre[ksf]	DLPost[ksf]	LL[ksf]	LL Type	VLL[ksf]	Dyn Load[ksf]
1	Office	☐		.01	.08	LL		.075
2	Storage	☐		.01	.125	LL		.175
3	Public	☐		.01	.1	LL		.075
4	Add Piping	☑		.02		LL		

When you open this spreadsheet you are presented with two tabs, Standard and Other.

The first column is the area load **Label**. This label is a description for your own use.

A check in the second column indicates that the load is additive. An additive load gets added to the area load defined directly beneath it. For example, loads such as piping or HVAC are sometimes applied in addition to the office or storage loads defined for that area. In the following diagram the additive piping load (Add Piping) is added directly to the office, storage, and public loads that were drawn beneath it.

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



The remaining columns define the magnitude of the load for each load case. The last two columns are the Vibration Live Load and the Dynamic Load. The Vibration Live Load will used when checking floor vibration per AISC Design Guide 11. See Vibrations for more information. The dynamic mass will be included as part of the defined mass when interfacing with RISA-3D. See RISA-3D Integration for more information.

Other loads can also be applied graphically, see Drawing Area Loads. Other loads can be defined in any direction, see Load Direction. On the **Other** tab of the **Area Load Definitions** Spreadsheet the first column contains the load magnitude while the second contains the load direction.

Other Uniform Area Loads

Standard | Other

	Label	OL1[ksf]	Direction	OL2[ksf]	Direction	OL3[ksf]	Direction	OL4[ksf]	Direction
1	Roof		Y		Y		Y		Y

Tapered Area Loads

The **Tapered Area Load Spreadsheet** records the uniform area loads to be attributed to the members and may be accessed by selecting **Loads** ▸ **Tapered Area Loads** on the **Spreadsheets Menu**. You may define Tapered Area Loads graphically or by using the spreadsheets. See Drawing Area Loads to learn how to draw these loads graphically.

Tapered Area Loads

1: Floor Plan 1

	A Point	B Point	C Point	D Point	Base Mag[ksf]	Peak Mag[ksf]	Category	Direction
1	N1	N3	N4	N2	0	10	DL PreComp	Y
2	N10	N4	N3	N9	1	3	LL-Non	PY
3	N10	N2	N1	N9	1	3	LL-Non	PY

The spreadsheet contains the points that define the area load, the base magnitude and the peak magnitude. The drop down list at the top of the spreadsheet allows you to toggle between floors.

Note:

- Tapered area loads are always additive to area load definitions defined below them. They never negate or supersede other area loads.
- To investigate the value of an tapered area load at a specific location, you may use the Area Load Query utility.

Deck Loads

Deck Loads consist of the deck self weight, construction dead load, and construction live loads and are all specified from within the **Deck Definitions Spreadsheet**. Note that these are different from the area loads defined in the **Area Load Definitions Spreadsheet** as these loads are specifically linked to the type of deck construction. For more information on this topic refer to Deck Properties - Loads.

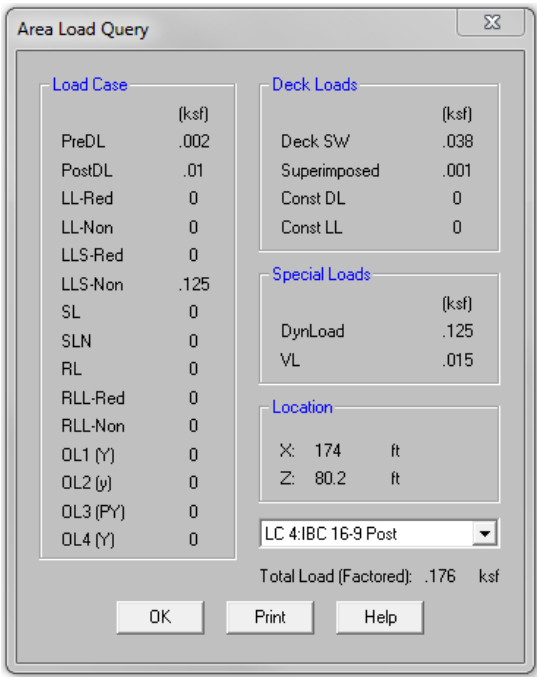
Note:

- To investigate what area loads are applied at a particular location, you may use the Area Load Query utility.

Area Load Query Dialog

The **Area Load Query** dialog is activated by clicking on the icon shown above and then clicking on any location on the floor. The purpose of this dialog is to allow the user to investigate the loading at any point on their floor.

This dialog will display the loading associated with a particular Load Combination. It lets you see how the various default loading interacts with defined area loads, tapered area loads, and deck loading.



The **Load Combination** drop down menu allows the users to select whichever load combination they are interested in.

The **Location** reports the point on the floor for which the loading is being reported.

The **Load Case** section of the dialog will display loading any Area Load Definitions that will apply at the selected location for that load combination. This also includes and Tapered Area Loads that are applied in that location.

Note:

- An asterisk (*) in this section means that there were area loads combined together with different directions (y, PY, and Y) to sum to this magnitude. These different directions will only yield differing directions if on a sloping level.

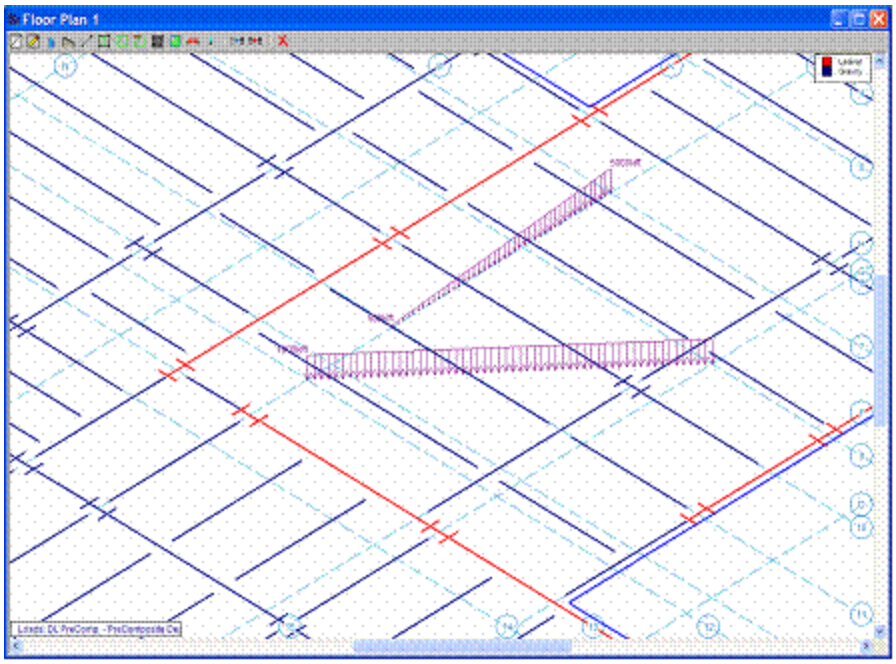
The **Deck Loads** section refers to loading associated with the deck defined at that location.

The **Special Loads** section is NOT associated with the particular load combination because these "special" loads are used for vibration or seismic load calculations and are never themselves included in a load combination.

The **Total Load (Factored)** adds up all the forms of area loading applied in the specified load combination. In includes any load factors applied in that load combination. This, obviously, does not include other forms of loading such as point loads, line load, column or wall reactions from above. Nor does it include any reduction of Live Loading based on tributary area.

Line Loads

Line Loads are linear loads that can be placed anywhere on a floor and can be of uniform, stepped, or varying magnitude such as triangular or trapezoidal. Line loads may be assigned manually in the **Line Loads Spreadsheet** or drawn graphically in a model view by clicking the **Draw Line Loads** button on the **Drawing Toolbar**.



Drawing Line Loads

To draw line loads graphically, click the button on the **Drawing Toolbar** to open the **Draw Line Loads Dialog**. Enter the start magnitude and the end magnitude in the categories you wish to use. Apply the line load by clicking on grid or point locations in the model view. You may apply a line load anywhere on a floor.

To Apply Line Loads

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Click the **Line Loads** button on the **Drawing Toolbar** and define the load in the **Draw Line Loads Dialog**.

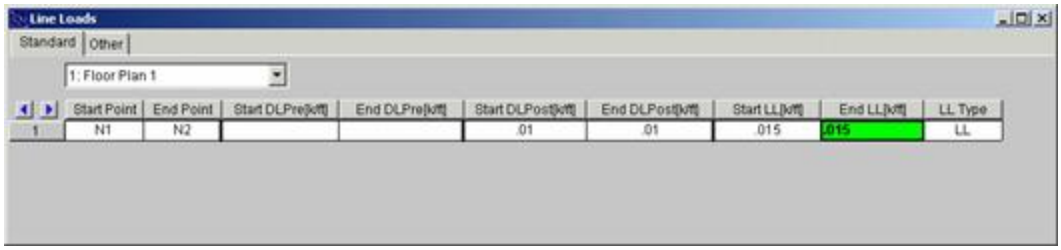
3. Click **Apply** and draw the line load by clicking on consecutive drawing grid or point locations.
4. To stop drawing altogether, click the right mouse button, or press the Esc key.

Note

- To apply more loads with different parameters, press CTRL-D to recall the **Draw Line Loads Dialog**.
- You may also specify or edit line loads in the **Line Loads Spreadsheet.**
- You may undo any mistakes by clicking the **Undo** button.
- You may only change the direction of Other Loads.

Line Loads Spreadsheet

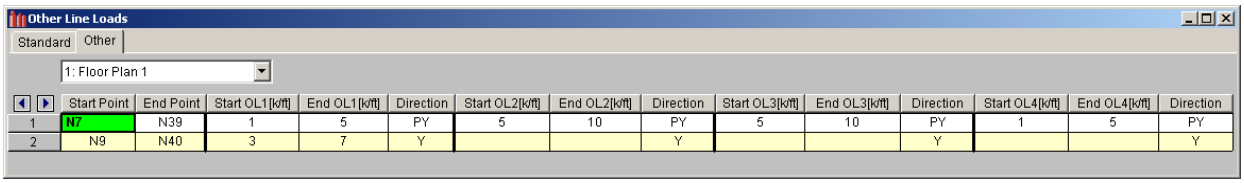
The **Line Loads Spreadsheet** records the line loads on each floor and may be accessed by selecting **Loads ▸ Line Loads** on the **Spreadsheets Menu**.



On the **Standard** tab, the first column contains the **Start Point** of the load and the second column contains the **End Point** of the load. The pull down list at the top of the spreadsheet allows you to toggle between floors.

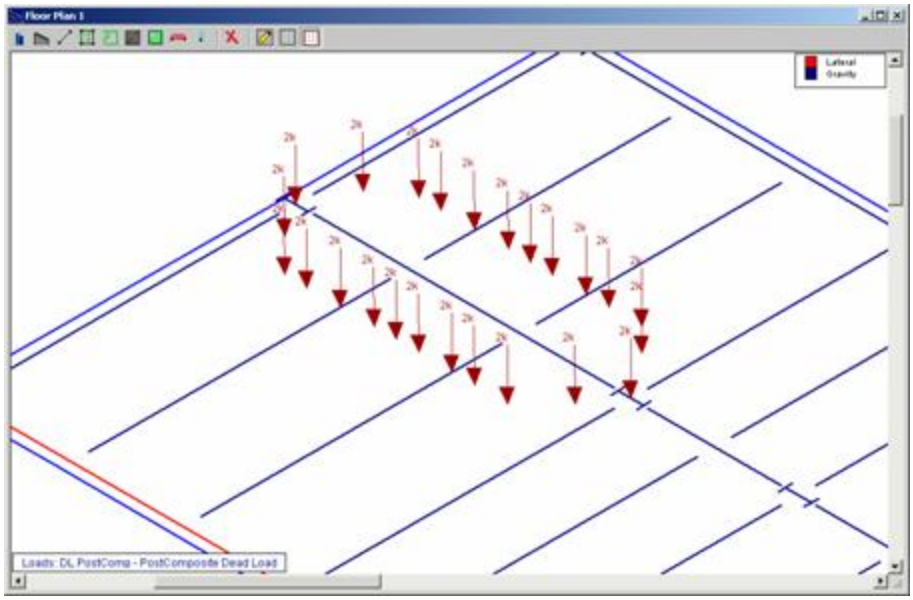
The remaining columns contain the **Start Magnitude** and **End Magnitude** of each line load for the **Standard Load** cases.

On the **Other** tab, the columns contain the **Start and End Points** and the **Start and End Magnitudes** of each line load for the **Other Load** cases as well as the load direction



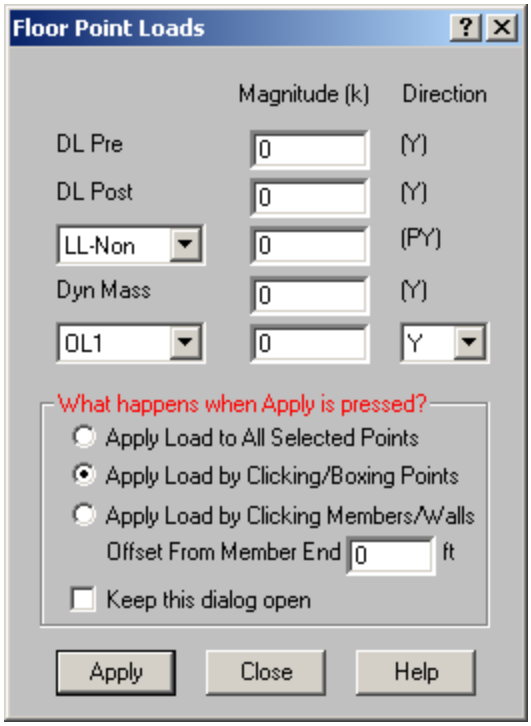
Point Loads

Point Loads are concentrated loads and may be placed anywhere on a floor. Point loads may be assigned manually in the **Point Loads Spreadsheet** or drawn graphically in a model view by clicking the **Draw Point Loads** button on the **Drawing Toolbar**.



Drawing Point Loads

To draw point loads graphically, click the button on the **Drawing Toolbar** to open the **Draw Point Loads Dialog**. Enter the magnitude of the point load in the categories you wish to use. Apply the point load to all selected points or by clicking or boxing grid or point locations in the model view. You may apply a point load anywhere on a floor.



To Apply Point Loads

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.

2. Click the **Point Loads** button on the **Drawing Toolbar** and define the load in the **Draw Point Loads Dialog**.
3. You can apply the point load to all selected points and click **Apply**.

   You can apply the point load by clicking **Apply** and clicking or boxing grid or point locations in the model view.

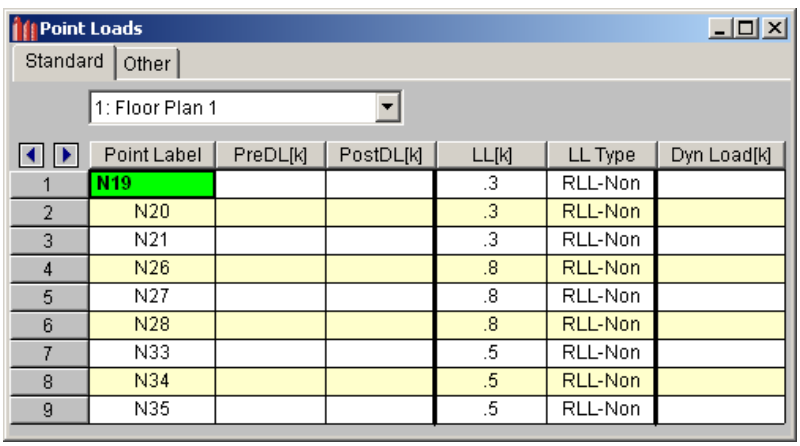
4. To stop drawing altogether click the right mouse button, or press the Esc key.

Note

- To apply more loads with different parameters, press CTRL-D to recall the **Draw Point Loads Dialog**.
- You may also specify or edit point loads in the **Point Loads Spreadsheet.**
- You may undo any mistakes by clicking the **Undo** button.
- You may only change the direction of Other Loads.

Point Loads Spreadsheet

The **Point Loads Spreadsheet** records the point loads on each floor and may be accessed by selecting **Loads** ▸ **Point Loads** on the **Spreadsheets Menu**.



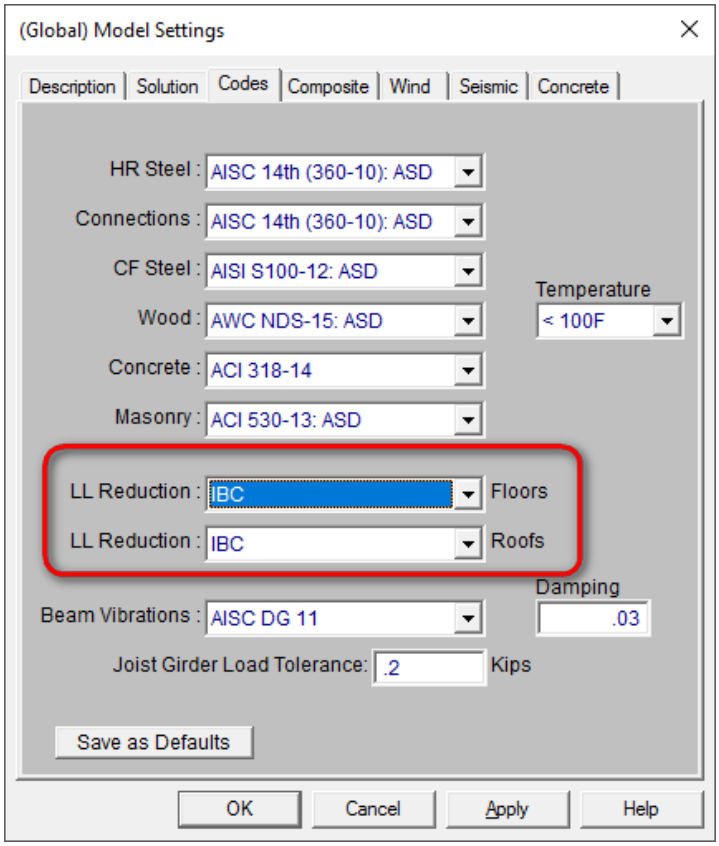
	Point Label	PreDL[k]	PostDL[k]	LL[k]	LL Type	Dyn Load[k]
1	N19			.3	RLL-Non	
2	N20			.3	RLL-Non	
3	N21			.3	RLL-Non	
4	N26			.8	RLL-Non	
5	N27			.8	RLL-Non	
6	N28			.8	RLL-Non	
7	N33			.5	RLL-Non	
8	N34			.5	RLL-Non	
9	N35			.5	RLL-Non	

In either the Standard or Other tabs, the first column contains the point that receives the load. The next columns contain the magnitudes of the load respectively for each load case. The pull down list at the top of the spreadsheet allows you to toggle between floors.

Loads – Live Load Reduction

Overview

Live load reduction is automatically applied to members based upon their designation (column or beam), loading, tributary area and upon the chosen code in the Model Settings - Codes settings.



The supported codes for live load reduction on **Floors** are:

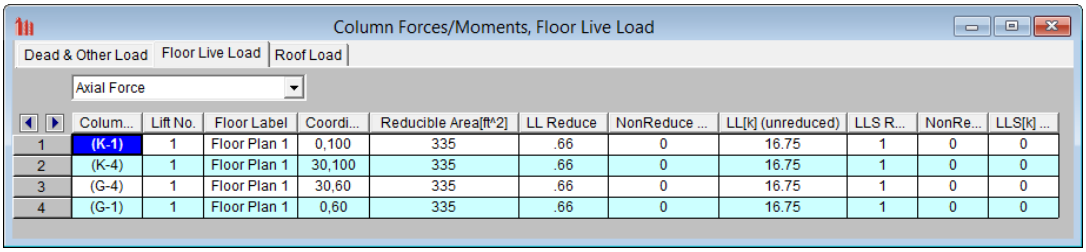
- IBC (2000 - 2015)
- IBC 2000/03 Alternate
- IBC 2006/09/12 Alternate
- IBC 2015 Alternate
- NBC 2005/10
- SBC 1999
- UBC 97/97 Alternate

The supported codes for live load reduction on **Roofs** are:

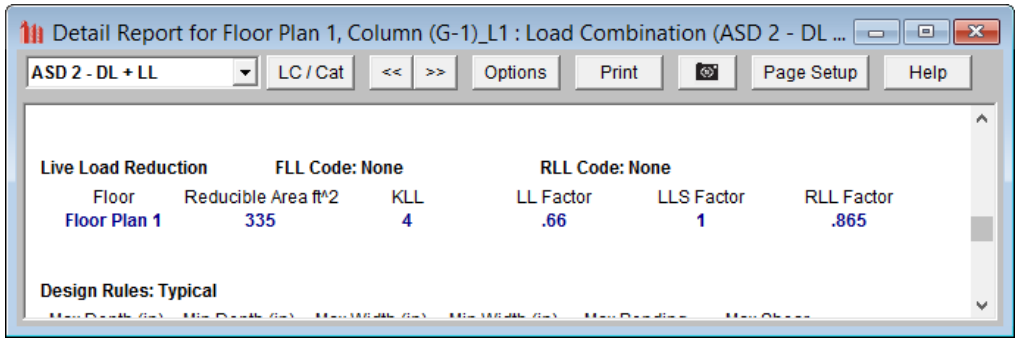
- ASCE 7 (1998 - 2010)
- BOCA 1999
- IBC (2000 - 2015)
- NBC 2005/10
- SBC 1999
- UBC 97 Method 1/Method 2

Columns

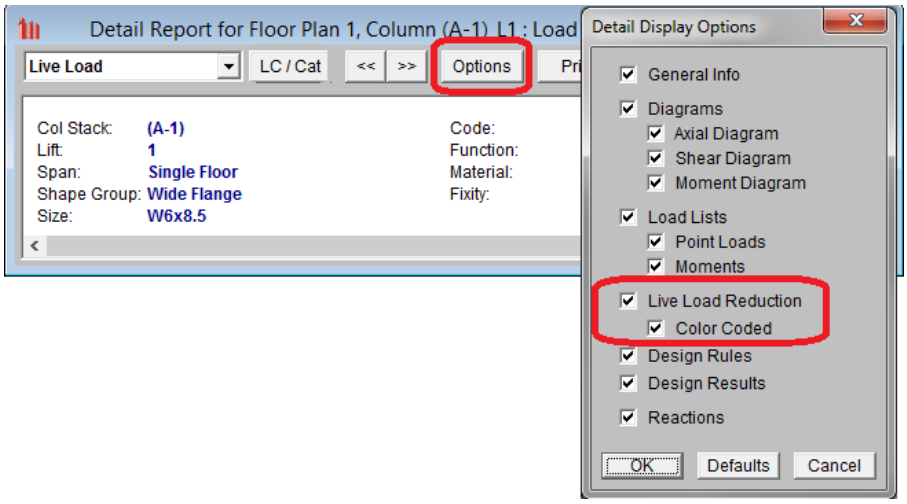
For columns, the calculated reduction factors are displayed both in the **Column Forces** spreadsheet on the **Floor Live Load** tab and the **Column Detail Report**.



	Colum...	Lift No.	Floor Label	Coordi...	Reducible Area[ft^2]	LL Reduce	NonReduce ...	LL[k] (unreduced)	LLS R...	NonRe...	LLS[k] ...
1	(K-1)	1	Floor Plan 1	0,100	335	.66	0	16.75	1	0	0
2	(K-4)	1	Floor Plan 1	30,100	335	.66	0	16.75	1	0	0
3	(G-4)	1	Floor Plan 1	30,60	335	.66	0	16.75	1	0	0
4	(G-1)	1	Floor Plan 1	0,60	335	.66	0	16.75	1	0	0



The axial, shear, and moment displayed in the diagrams of the **Member Detail Report** include effects of live load reduction in the results presentation based on the Detail Display Options. Lighter colors in the diagram indicate the portion of the display load that got reduced.



The effects of live load reduction are applied to the design axial force in the **Bending** code checks section of the report. However, the moment and shear design forces do NOT include the effects of live load reduction for columns. The diagrams will color code the shear and moments based on Live Load Reduction, but the values are NOT reduced in the column code checks.

Design Results

Shear (y) : 1% Capacity at 0 ft for LC 1 (ASD 1 - DL)

Vy = -.204k	fvy = .147ksi	Vny/om = 19.9714k
Vz = .114k	fvz = .051ksi	Vnz/om = 28.9725k

Bending: 80.2% Capacity at 12 ft for LC 1 (ASD 1 - DL)

P = 11.051k	fa = 3.113ksi	Pnc/om = 21.6734k
Mz = -1.636k-ft	fbz = 2.678ksi	Mnz/om = 14.9102k-ft
My = -.914k-ft	fby = 7.337ksi	Mny/om = 4.16766k-ft
	y-y	z-z
Lb	12 ft	12 ft
KL/r	156.907	57.714
Cb = 2.174	Lcomp Top = 12 ft	Lcomp Bot = 12 ft
L-torque = 12 ft		

Walls

For walls, the calculated reduction factors are displayed both in the **Wall Forces** spreadsheet on the **Floor Live Load** tab. Wall will transfer 50% of their total tributary area into each element that supports its starting and ending joint at the floor below. So, in cases where they are supported by a beam along their length and a post at the ends, all the tributary area is assumed is transferred to the posts directly.

Wall Forces, Dead and Other

Dead & Other | Floor Live Load | Roof Load

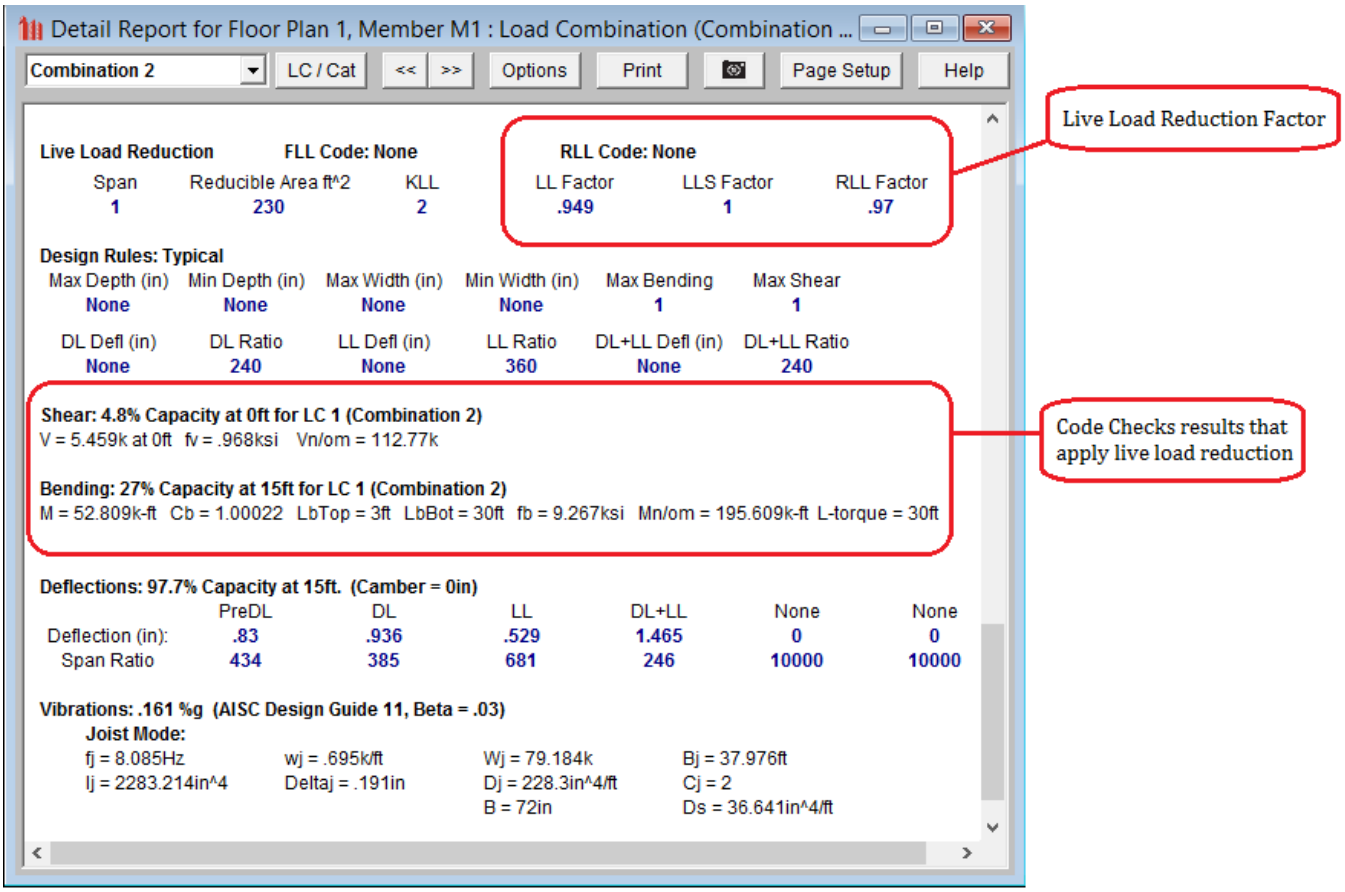
1: Floor Plan 1

	Wall	St Loc[ft]	End Loc[ft]	Length[ft]	Max Base Rea...	Max Base Rea...	DL[k]	Pre DL[k]	OL1[k]	OL2[k]	OL3[k]	OL4[k]
1	WP1	45,214	119,214	74	472.571	11	283.693	191.226	0	0	0	0
2	WP2	45,119	89,119	44	507.788	11	287.087	171.495	0	0	0	0
3	WP3	119,89	119,45	44	444.686	11	229.369	170.529	0	0	0	0
4	WP4	214,119	214,45	74	527.055	11	321.832	229.111	0	0	0	0

Beams

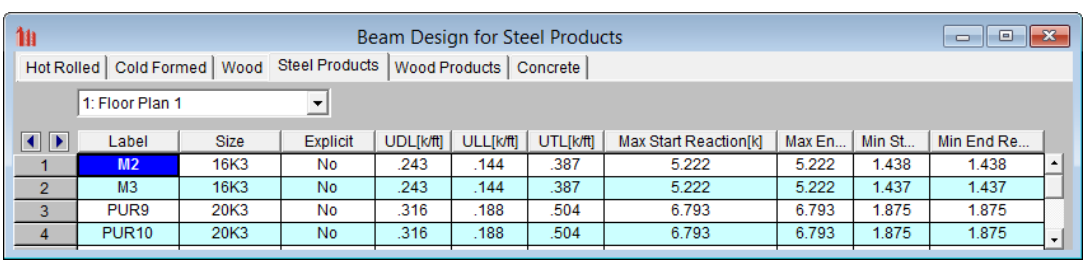
For beam members, the calculated reduction factors are displayed in the **Member Detail Report**. The effects of live load reduction, however, will be considered in the**End Reactions**, **Shear Results**, and **Bending Results** spreadsheets, as they do reduce the load when calculating maximum end reactions, shear, bending stresses/forces and deflection calculations.

The force diagrams in the beam's detail report consider the effects of live load reduction based on the Detail Display Options. Lighter colors in the diagram indicate the portion of the display load that got reduced.



Steel Products

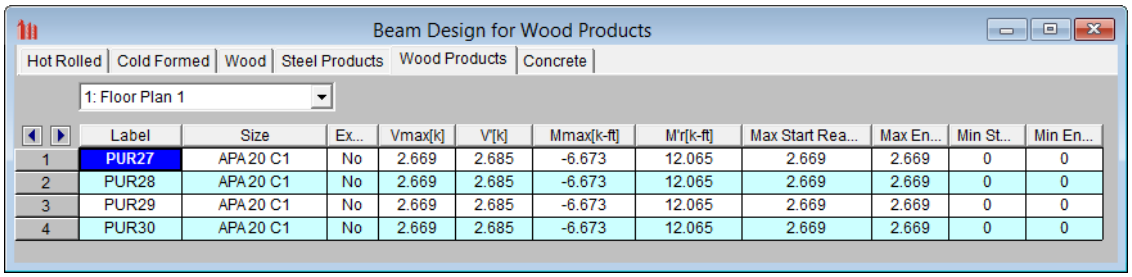
For steel products, live load reduction factors are calculated for shear and bending in the code checks section of the **Member Detail Report**. The force diagrams in the beam's detail report consider the effects of live load reduction based on the Detail Display Options. Lighter colors in the diagram indicate the portion of the display load that got reduced. The **Design Results** spreadsheet includes live load reduction effects The effects of live load reduction are NOT considered in deflection calculations.



	Label	Size	Explicit	UDL[k/ft]	ULL[k/ft]	UTL[k/ft]	Max Start Reaction[k]	Max En...	Min St...	Min End Re...
1	M2	16K3	No	.243	.144	.387	5.222	5.222	1.438	1.438
2	M3	16K3	No	.243	.144	.387	5.222	5.222	1.437	1.437
3	PUR9	20K3	No	.316	.188	.504	6.793	6.793	1.875	1.875
4	PUR10	20K3	No	.316	.188	.504	6.793	6.793	1.875	1.875

Wood Products

For wood products, live load reduction factors are calculated for shear and bending in the code checks section of the **Member Detail Report**. The force diagrams in the beam's detail report consider the effects of live load reduction based on the Detail Display Options. Lighter colors in the diagram indicate the portion of the display load that got reduced. The effects of live load reduction are NOT considered in deflection calculations.



	Label	Size	Ex...	Vmax[k]	V'[k]	Mmax[k-ft]	M'r[k-ft]	Max Start Rea...	Max En...	Min St...	Min En...
1	PUR27	APA 20 C1	No	2.669	2.685	-6.673	12.065	2.669	2.669	0	0
2	PUR28	APA 20 C1	No	2.669	2.685	-6.673	12.065	2.669	2.669	0	0
3	PUR29	APA 20 C1	No	2.669	2.685	-6.673	12.065	2.669	2.669	0	0
4	PUR30	APA 20 C1	No	2.669	2.685	-6.673	12.065	2.669	2.669	0	0

Live Load Considerations

Reducible Load Types

Live loads are only reduced for **Load Cases** specifically designated as Reducible. This includes LL-Reduce, LLS-Reduce and RLL-Reduce. No load reduction is applied for the other Load Cases.

Special Live Loads

Special Live Loads use the same reduction procedures as regular live loads. However, they will only be reduced for members that support more than one floor, and will not be reduced more than 20%. If the UBC or IBC provisions are specified this reduction will only be allowed for columns.

Area Calculations

When a member supports multiple load types, only the area that is subjected to reducible loads is considered in the **Tributary Area** calculation.

Tributary Area for Columns

The reducible area for live load reduction for columns is taken as the cumulative area tributary to the column lift. The area used at each floor level includes the area accumulated in the floor levels above.

Tributary Area for Beam Supported Floors

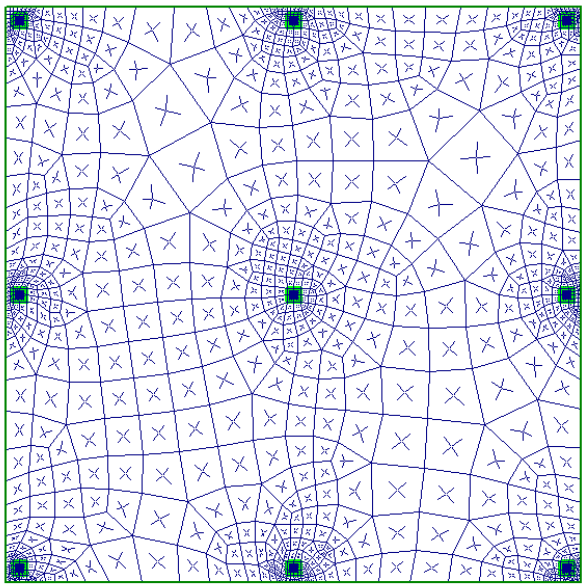
This is a very basic tributary area calculation based on the geometry of the floor level.

Tributary Area for Concrete Slab Floors

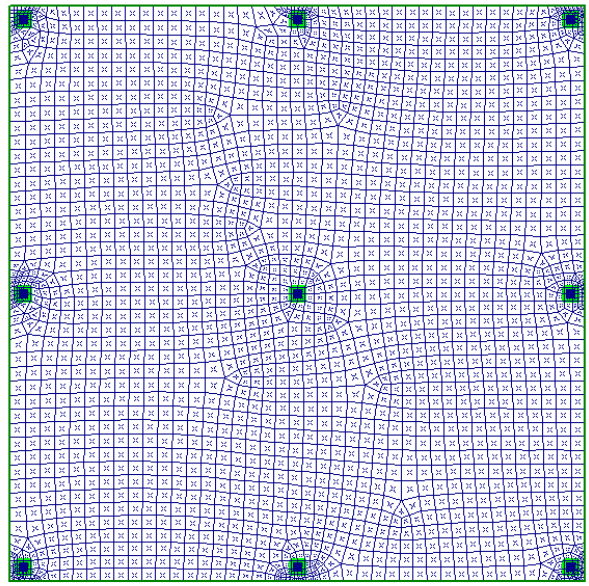
Because of the nature of the FEA solution, the tributary area for concrete slab floors will not always exactly match your hand calculations. The load is attributed to supporting elements based on relative stiffness; RISA then back calculates how much tributary area contributed to each supporting element based on how much load it took.

To determine this value, the program will run a test solution where it applies a 1 ksf load to the slab mesh. It then solves to find out how much load goes to each supporting element. The tributary area is then back-calculated from the load that is found in each supporting element per the finite element solution.

You can refine this calculation (and therefore your solution) to be more accurate by reducing your Slab Mesh in Model Settings.



Coarse Slab Mesh



Fine Slab Mesh

------Better Tributary Area Results---->

Integration with RISA-3D and RISAFoundation

Methodology for Live Load Reduction in RISA-3D

One of the reasons why it is easier for a program like RISAFloor to perform Live Load Reduction is because the analysis results are based on category solutions. At any point in a column or beam, it is known exactly how much of the result is due to Live Load, or Roof Live Load or such. The Live Load results are specifically known at all locations.

However, programs like RISA-3D need to be based on Load Combination solutions if they are to properly account for non-linear effects like P-Delta , compression only springs, tension only beams, Direct Analysis Stiffness adjustment factors or such. Therefore, it is not possible for RISA-3D to know precisely how much of a given result is due to Live Load.

When the two programs are used together, RISA-3D can estimate the amount of LL Reduction in a member based on the category results and LL reduction factors calculated in RISAFloor. That way, the analysis results in RISA-3D are still accounting for the non-linear effects of the overall load combination, but the design results / code checks can reduce those values based on the RISAFloor results.

This is NOT an exact solution. If the axial force calculated for a column is dramatically different in the RISA-3D solution than it is in the RISAFloor solution, then this could be problematic. For this reason the user is given the opportunity to turn off LL reduction on the Codes tab of the (Global) Model Settings in RISA-3D.

Integration from RISAFloor to RISA-3D

Lateral columns that are brought into RISA-3D will only have their axial load reduced. Shear, and moment will not be reduced.

Lateral beams that are brought into RISA-3D will only have their shear and moment values reduced. Axial loads will not be reduced.

Live Load reduction is not considered in the seismic detailing calculations. This specifically refers to the calculation of Vg and Column / Beam moment ratio used in Column -Beam moment connections.

Integration from RISAFloor to RISAFoundation

Only column reactions will be reduced in RISAFoundation.

Wall reactions will NOT be reduced in RISAFoundation.

Limitations

Ratio of Dead and Live Loads

The maximum live load reduction is not restricted based on the ratio of Dead and Live Loads given in the UBC-97, SBC-99, and IBC-00/03 alternate. In general, this could affect beam members where the Dead Load is less than 40% of the total gravity load and column members where dead load is less than 60% of the total gravity load.

K_{LL} Factor

Currently the program will use a K_{LL} factor equal to 4 for all columns, and a K_{LL}factor equal to 2 for all beams. RISA uses this approach because there is currently no input to define the different categories that would be necessary to give the exact value.

Two Way Deck

Live Load Reduction is not currently performed for two way deck on **Beam Supported Floors**.

Integration from RISAFloor to RISAConnection

The effects of live load reduction on **End Reactions** from RISAFloor WILL carry over to RISAConnection.

Loads – Load Attribution

The type and direction of the deck (for beam supported floors) or slab (for concrete slab floors) determines which members support an area and thus the loads that are attributed to the members. Unless the deck/slab is defined as "Two Way", the distribution direction is indicated by the Deck Angle (for beam supported floors) or the Direction of the slab definition (for concrete slab floors).

One Way Distribution

You may place area loads, line loads and point loads directly on a one way deck or slab and it will be automatically attributed to the supporting beams, columns and walls. For one-way distribution, loads are attributed to supporting members based on simple beam theory. The deck or slab is viewed as a simple beam that spans between supporting members. Therefore, the support closest to the applied load receives a higher percentage of the load based on the span length and load location.

In this way, **Area Loads** and **Tapered Area Loads** are attributed to beams and columns as distributed loads. Although at times it is beneficial to draw area loads beyond the diaphragm edges, only loading within the edges is attributed to the model. Area loads that are defined outside the diaphragm edges will not be applied to the solved model.

Note

- **Line Loads** are attributed in the same way as area loads except that when the are parallel to the distribution span they are applied concentrated loads on the supporting beams or columns. If a line load is applied outside of a diaphragm edge, only the portion of the line load that is directly supported by a beam or wall will be attributed. A warning is entered in the warning log for each line load that is not attributed.
- Just like area and line loads, **Point Loads** are applied using simple beam theory. Therefore, a point load defined on the deck or slab will be attributed to the supporting members based on the distribution span and the distance to the applied load. Point loads that are applied outside of a diaphragm edge, will only be attributed if they are directly supported by a beam, column or wall. A warning is entered in the message log for each point load that is not attributed.

Ranking of Area Loads

If two area loads are drawn over the same location then the load that was drawn in *last* (i.e. the top-most load) will govern for that region. In this way, the area load polygons can be seen as a "stack" of loads where the one on top of the stack has the highest priority or ranking and will supersede any of the area loads that are lower down in the stack.

The one exception to the stack analogy are the"**Additive**" area loads. Any loads that are defined as additive will be added together first. The,n the total will be added to the normal (non-additive) area load that is highest in the stack to determine the total area loading in that region.

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

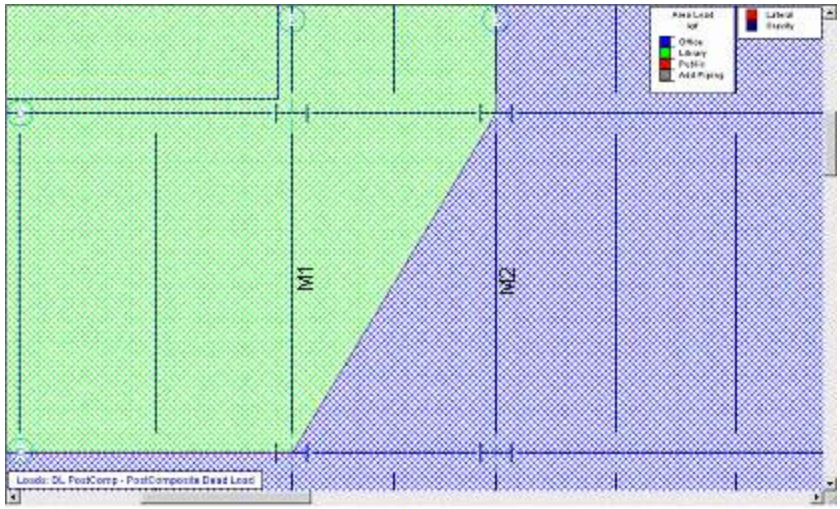
Note:

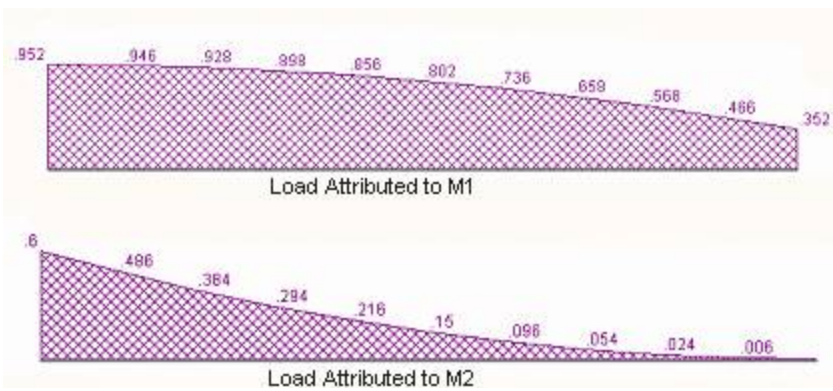
- To investigate what area loads actually get applied at a particular location, you may use the Area Load Query utility.

Parabolic Attribution

For both area loads and line loads there are cases where the resulting distributed loads are actually parabolic or cubic in nature. The partial deck loading shown in the following image is a good example of a loading condition

that produces parabolic loads. RISAFloor approximates the parabolic load as a series of trapezoidal distributed loads as shown in the following diagram.





Two Way Distribution

Distribution on Beam Supported Floors

For two way decks, **Area Loads** are each divided into meshes of smaller areas which are assigned an equivalent point load. Each point load is attributed to the surrounding members based on their relative distance to the point load. The size of the mesh may be set on the **Global** parameters with smaller mesh sizes giving more accuracy. Beams that receive area loads are divided into ten segments and the area loads that fall on these segments are "smoothed" into equivalent distributed loads. Although at times it is beneficial to draw area loads beyond the diaphragm edges, only loading within the edges is attributed to the model. Area loads that are defined outside the diaphragm edges will not be applied to the solved model.

Similarly, **Line Loads** on two way decks are divided into smaller loads based on this area load mesh size. These loads are then attributed to their supporting member and converted into equivalent distributed loads. If a line load is applied outside of a diaphragm edge, only the portion of the line load that is directly supported by a beam or wall will be attributed.

On two way decks, **Point Loads** are attributed to the surrounding members based on their relative distances to the point load. Point loads that are applied outside of a diaphragm edge, will only be attributed if they are directly supported by a beam, column or wall.

Distribution on Concrete Slab Supported Floors

Because of the nature of the FEA solution, the two way load distribution for concrete slab floors will not always exactly match your hand calculations. This is because the program analysis is based on a load distribution due to element stiffness not tributary area.

The load is attributed to supporting elements based on relative stiffness which is increased by the slab (plate) floor. Because of this, supporting columns at the interior of the slab will be stiffer (and therefore attract more load) than those along the edges.

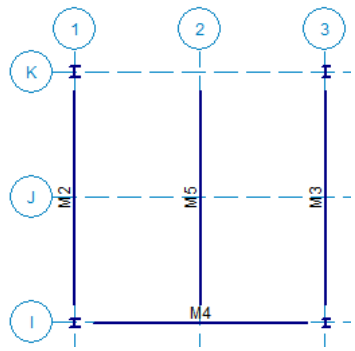
Loads – Load Takedown

Load takedown is an important concept in RISAFloor to make sure applied loads and self-weight make their way through the model in a straightforward fashion. For the most part there is a common sense load path with load flowing through joists, into girders, to columns and then down to the foundation. However, there are modeling/framing situations that are not quite as obvious. Here we will highlight some of these conditions and explain how the program addresses them.

Beams

In RISAFloor it is only possible to draw beam members if there is a supporting member at the member end to support it. Simply supported beams will always transfer their loads at member ends. Continuous and cantilevers will always transfer their loads into the supports. It is possible to move or delete framing after the initial drawing. This can cause problems with the solution, as the program requires all beams to be supported for stability sake. If a framing modification creates unsupported beam members the program will give an error at solution time (Error 2049) and point you directly to the beams that are unsupported in the warning log (Warning 41518).

Beam M5 in the framing below will give an error at solution, as it is unsupported along grid line K.



Columns and Walls

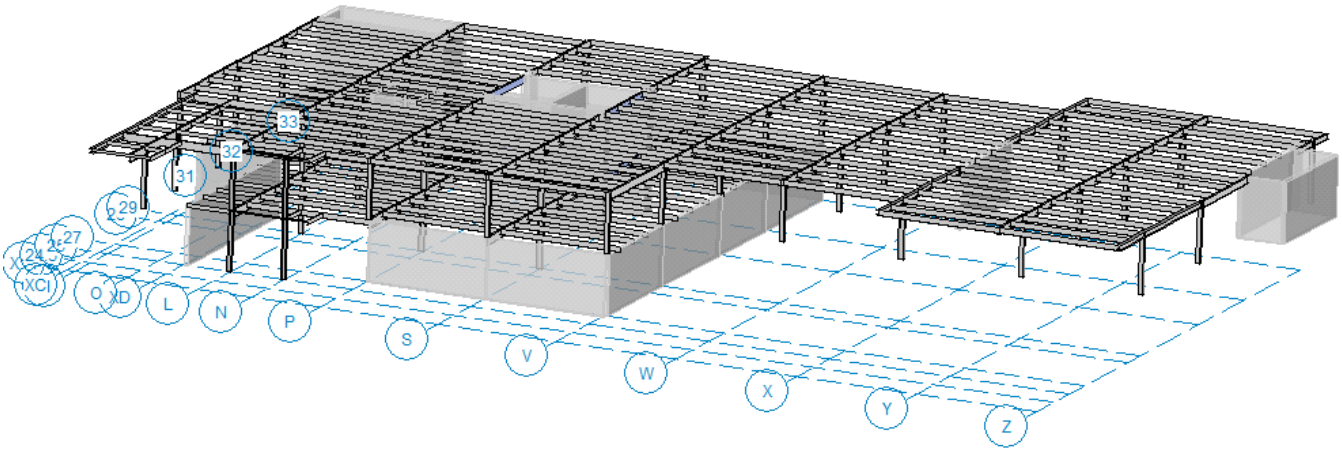
Column and wall load takedown is also relatively straightforward in most cases. For a single story structure it is assumed that all columns and walls land on boundary conditions at the base of the model. For multi-story structures, if columns/walls land on beams, columns and walls below, the load is transferred directly into those elements and then down the rest of the structure.

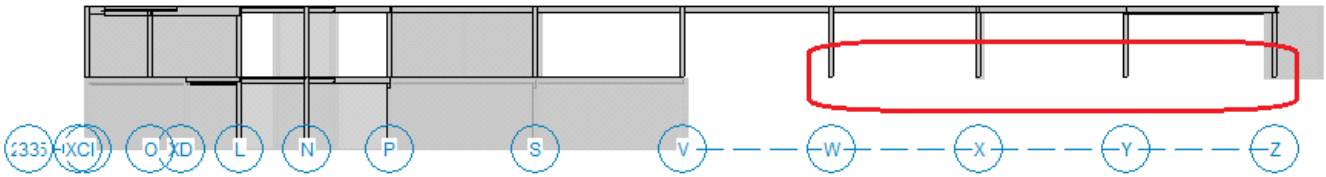
Here we will talk about cases where columns and walls do NOT land on framing elements on the floors below.

Columns and Walls That Land Outside of a Deck or Slab on the Floor Below

For columns and walls that land outside of a deck the program places a boundary condition at this location and gives a warning (Warning XXXXX) letting the user know that this happened.

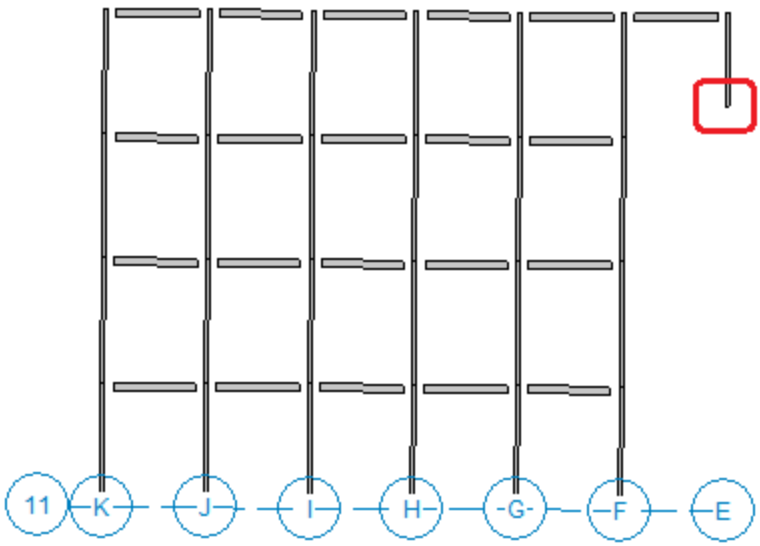
For stepped buildings that may be built on a slope, or for buildings with partial basement floors, it is common to have columns on an upper level that are unsupported on the floors below.





In these cases this is how the structure is actually supported and is understandable.

The other possibility is that a column framing problem was created from a modeling error or a framing situation was not completely understood. Here is an example of that situation:



The program will still apply a boundary condition here. You must use the warning (Warning XXXXX) to note this and modify the framing to fix the situation.

Note:

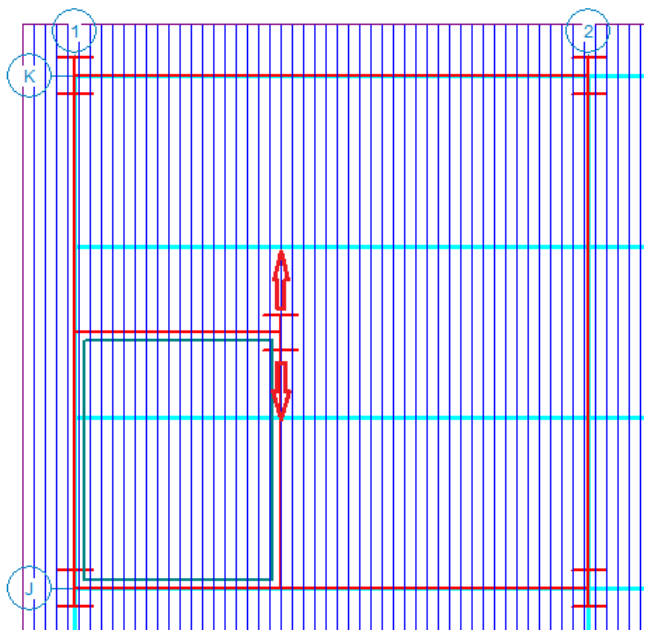
- For walls or columns who's base does not land exactly on a floor elevation below the program does not give a warning.
- For walls that are partially supported by columns, beams and walls, the portion of wall that lands on these entities will be transferred to the floor below. For the portion of wall that lands outside of a deck and not on a column, beam, or wall, a boundary condition and a warning log will be given.

Columns and Walls That Land Inside of a Deck or Slab on the Floor Below But NOT on Framing

Beam Supported Floors

If a column lands on a beam-supported floor but not on a framing element (beam, column or wall) then the load will still be transferred according to the deck direction and tributary length. A warning will be generated (Warning XXXXX) letting you know that this. column reaction was transferred solely by the deck, as this is typically an unusual situation. Typically a column reaction at a floor level requires framing to get that reaction out to the girders, walls and columns at that floor level.

In the image below the framing in RED is on the current level and the framing in BLUE is on the floor below. The vertical blue lines show the direction of the decking below. In this case the column does not have framing directly below it, but the deck spans the deck taking the load to the tributary beams.



Note:

- For walls that are partially supported by columns, beams and walls, the portion of wall that lands on these entities will be transferred to the floor below. For the portion of wall that lands on the deck the same principles that apply to columns also apply to walls.

Concrete Slab Supported Floors

For concrete slab floors the slab will create an FEA mesh and work as the load transfer mechanism to get the load down the structure. There are no problems in this case except if this is a lateral column or wall on a rigid diaphragm level.

Gravity columns/walls are only considered in RISAFloor, so there are no implications for these elements when going to RISA-3D.

Lateral columns/walls that land on a Semi-Rigid diaphragm are also OK, as in RISA-3D the slab is modeled as an FEA mesh.

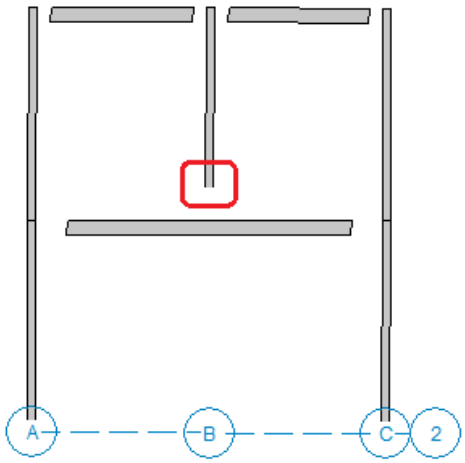
Lateral columns/walls that land on a Rigid diaphragm will be a problem in RISA-3D, as the slab is modeled using a rigid diaphragm in RISA-3D and there is no out-of-plane rigidity defined on these slab elements.

Note:

- RISAFloor will give a warning (Warning XXXXX) if there are lateral columns/walls landing on a slab without framing elements below, letting you know that you may have problems when solving the model in RISA-3D.

"Floating" Columns and Walls

It is also possible to create situations where columns and walls are drawn inside of the plan view of the floor below, but that do not land on the floor below because of the **Splice Distance Below** field. This applies to both beam and slab supported floors.



For these columns and walls there is no logical place to transfer the loads, so a boundary condition is created. In general the program makes it difficult to create this illogical framing situation, so this is a rare situation.

Loads - Load Combinations

During solution the model is loaded with a combination of factored Load Categories. These combinations, load factors, and other parameters are defined on the **Load Combinations Spreadsheet**. Most standard load combinations are included in the program. See Solution to learn how to solve load combinations.

To Add Load Combinations Manually

1. From the **Spreadsheets Menu** select **Load Combinations**.
2. Enter load combinations by pairing loads in the **Category** fields with factors in the **Factor** fields.

To Add Auto-Generated Building Code Combinations

1. From the **Spreadsheets Menu** select **Load Combinations**.
2. Select the LC Generator button from the **Window Toolbar**.
3. Select the Load Combination **Region** and **Code** from the drop down lists provided in the Gravity tab.
4. Modify the generated combinations and options in the spreadsheet as necessary.

Note

- The generated building code combinations are made up of Load Categories and Factors. Loads that are not assigned to these categories will not be included in the combinations upon solution.
- All generated combinations are added and marked for the envelope solution. You may remove combinations from the envelope after adding them.
- 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 may save any preferred load combinations as the default by clicking the **Save As Defaults** button on the **Window Toolbar**.

To Solve Load Combinations

- To solve load combinations, first mark the combinations to be solved by checking the **Solve** box on the **Load Combinations** spreadsheet. Click the **Solve** button on the **RISA Toolbar**.

Load Combinations Spreadsheet

The **Load Combinations Spreadsheet** records the combinations of loads for solution and may be accessed by selecting **Load Combinations** on the **Spreadsheets Menu**.

Combinations

Combinations | Design

	Label	Solve	Category	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor	Cate
1	ASCE Strength 1 Pre	☑	DLPre	1.4									
2	ASCE Strength 2 Pre	☑	DLPre	1.2	LLConst	1.6							
3	ASCE Strength 1 Post	☑	DL	1.4									
4	ASCE Strength 2 (a) Post	☑	DL	1.2	LL	1.6	LLS	1.6	RLL	.5			
5	ASCE Strength 2 (b) Post	☑	DL	1.2	LL	1.6	LLS	1.6	SL	.5	SLN	.5	
6	ASCE Strength 2 (c) Post	☑	DL	1.2	LL	1.6	LLS	1.6	RL	.5			
7	ASCE Strength 3 (a) Post	☑	DL	1.2	RLL	1.6	LL	.5	LLS	1			
8	ASCE Strength 3 (c) Post	☑	DL	1.2	SL	1.6	SLN	1.6	LL	.5	LLS	1	
9	ASCE Strength 3 (e) Post	☑	DL	1.2	RL	1.6	LL	.5	LLS	1			

Combinations Tab

The first field, the **Label**, is strictly for the user's reference. Enter any descriptive label you wish and it may be displayed with the results when the combination is solved.

The **Solve** box is used to indicate which combinations should be included the solution. See Solution for more information.

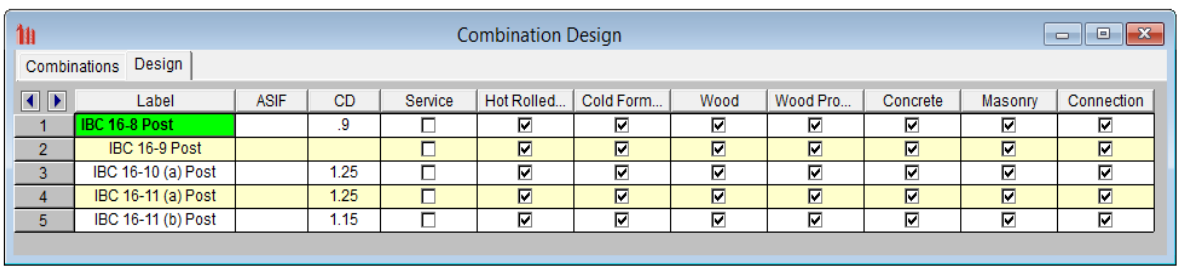
The next eight pairs of fields (**Category**, **Factor**) are for defining what load categories are to be part of the combination, along with factors for each. Select load categories from the drop down lists in the **Category** columns of the spreadsheet. See Load Categories for a list of the categories available for use.

Enter in the **Factor** field a multiplier to be applied to the load category being included. If you are designing an unshored composite beam system, you should define at least one pre-composite load combination. If you fail to do so, the beams will not be designed to resist the effects of the pre-composite loads.

Note

- Any time a load combination contains the load category DLPre, that Load Combination will be considered to act solely on the pre-composite floor system.

Design Tab



	Label	ASIF	CD	Service	Hot Rolled...	Cold Form...	Wood	Wood Pro...	Concrete	Masonry	Connection
1	IBC 16-8 Post		.9	☐	☑	☑	☑	☑	☑	☑	☑
2	IBC 16-9 Post			☐	☑	☑	☑	☑	☑	☑	☑
3	IBC 16-10 (a) Post		1.25	☐	☑	☑	☑	☑	☑	☑	☑
4	IBC 16-11 (a) Post		1.25	☐	☑	☑	☑	☑	☑	☑	☑
5	IBC 16-11 (b) Post		1.15	☐	☑	☑	☑	☑	☑	☑	☑

The first field, the **Label**, is strictly for the user's reference. Enter any descriptive label you wish and it may be displayed with the results when the combination is solved.

The **ASIF** column is used to set the allowable stress increase factor used 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 only applies to RISAFloor ES concrete slab floors. Load Combinations with the Service check box turned on will be checked for the Slab Point Deflection results and will also have the plate stiffness factored. See the Elevated Slabs - Analysis topic for more information.

The next six check boxes designate which load combinations should be used for the code checking of each material type. For example when using steel and wood in your model, you may design your steel for one set of load combinations and your wood for another.

The last check box for **Connection** defines two things:

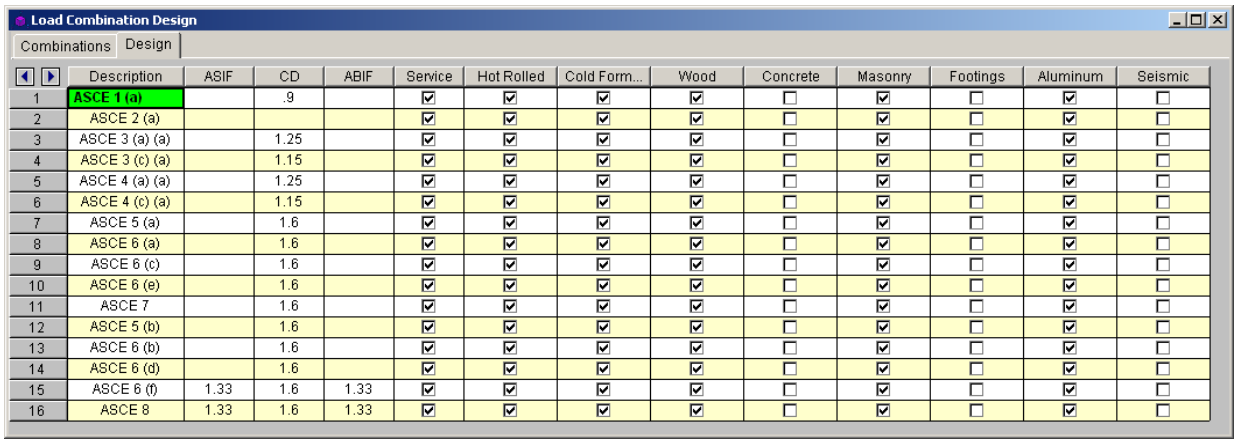
- Whether this load combination should be used to calculate the end reactions or not for hot-rolled steel members. Only load combinations with this check box will come into play when calculating the end reactions for the End Reactions spreadsheet and the graphic viewing of the reactions for hot-rolled steel members. All other member types will be based on their own material check boxes.
- If this LC should be used in designing hot-rolled steel connections with RISAConnection.

Note

- Steel products are always designed to the UDL, ULL and UTL loads described in the Beam Results section of this manual. See Uniform Loads on Steel Products for more information.

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.



Load Combination Design

Combinations | Design

	Description	ASIF	CD	ABIF	Service	Hot Rolled	Cold Form...	Wood	Concrete	Masonry	Footings	Aluminum	Seismic
1	ASCE 1 (a)		.9		☑	☑	☑	☑	☐	☑	☐	☑	☐
2	ASCE 2 (a)				☑	☑	☑	☑	☐	☑	☐	☑	☐
3	ASCE 3 (a) (a)		1.25		☑	☑	☑	☑	☐	☑	☐	☑	☐
4	ASCE 3 (c) (a)		1.15		☑	☑	☑	☑	☐	☑	☐	☑	☐
5	ASCE 4 (a) (a)		1.25		☑	☑	☑	☑	☐	☑	☐	☑	☐
6	ASCE 4 (c) (a)		1.15		☑	☑	☑	☑	☐	☑	☐	☑	☐
7	ASCE 5 (a)		1.6		☑	☑	☑	☑	☐	☑	☐	☑	☐
8	ASCE 6 (a)		1.6		☑	☑	☑	☑	☐	☑	☐	☑	☐
9	ASCE 6 (c)		1.6		☑	☑	☑	☑	☐	☑	☐	☑	☐
10	ASCE 6 (e)		1.6		☑	☑	☑	☑	☐	☑	☐	☑	☐
11	ASCE 7		1.6		☑	☑	☑	☑	☐	☑	☐	☑	☐
12	ASCE 5 (b)		1.6		☑	☑	☑	☑	☐	☑	☐	☑	☐
13	ASCE 6 (b)		1.6		☑	☑	☑	☑	☐	☑	☐	☑	☐
14	ASCE 6 (d)		1.6		☑	☑	☑	☑	☐	☑	☐	☑	☐
15	ASCE 6 (f)	1.33	1.6	1.33	☑	☑	☑	☑	☐	☑	☐	☑	☐
16	ASCE 8	1.33	1.6	1.33	☑	☑	☑	☑	☐	☑	☐	☑	☐

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

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**.

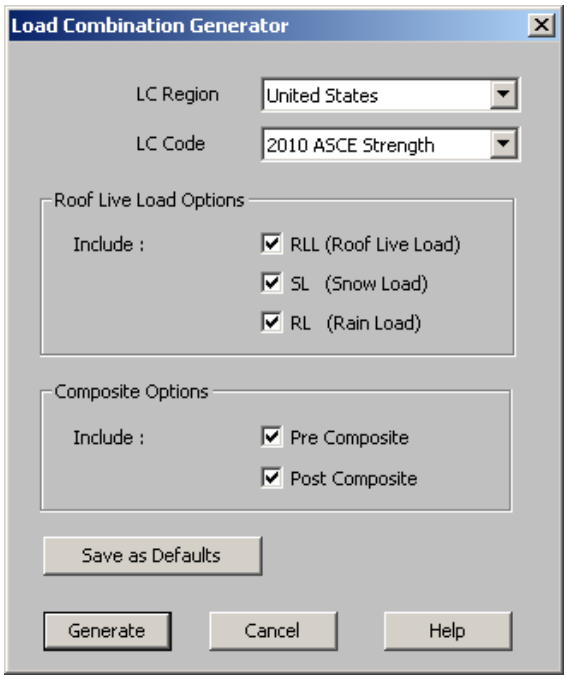
Note

- 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.

Generating Building Code Combinations

Major portions of the load combinations that are specified by building codes are included and may be applied to the model for solution. These combinations may be inserted by selecting the LC Generator button from the **Window**

Toolbar once the **Load Combinations Spreadsheet** is open and active. This will activate the **Load Combination Generator Dialog** shown below:



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.

Roof Live Load Options

The **Roof Live Load Options** allow the user to specify how complex the load combinations including roof live loads should be.

Placing a check in the box next to **RLL (Roof Live Load)**, **SL (Snow Load)**, and/or **RL (Rain Load)** will indicate that the selected load categories are to be included in the generated load combinations. If the box next to SL (Snow Load) is checked, the load combinations will include the Non-Shedding Snow Load category, **SLN**, as well as the SL load category.

The **Composite Options** allow the user to specify whether **Pre-Composite** and/or **Post-Composite** loads are to be included in the generated load combinations.

If the **Pre-Composite** checkbox is checked, load combinations containing the **DLPre** and **LLConst** load categories will be generated based on the selected code.

If the **Post-Composite** checkbox is checked, 'traditional' load combinations containing the **DL**, **LL**, **LLS**, and any of the categories included in the 'Roof Live Load Options' section above will be generated based on the selected code.

For more information on any of the load categories mentioned in this section, see Loads - Vertical Loads.

The **Save as Defaults** button may be used to establish the current load combination generator settings as the defaults for future use. Clicking the **Generate** button will generate load combinations in the **Load Combinations Spreadsheet** based on the selected options.

General Notes on the Load Combination Generator

The following loads are not generally included in the standard combinations but may be added by editing the combinations in the spreadsheet or by modifying the source document itself. For more information, see Customizing the Load Combination Generator below:

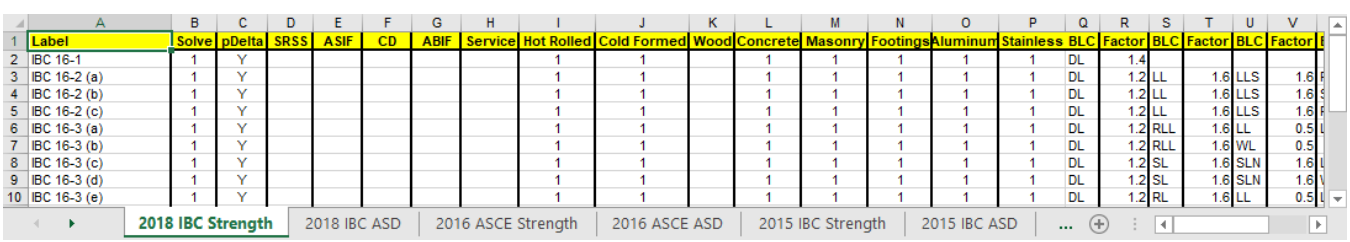
Category Code	Description
TL	Long Term Load
HL	Hydrostatic Load
FL	Fluid Pressure Load
PL	Ponding Load Category
EPL	Earth Pressure Load
DLConst	Construction Dead Load
OL#	Other Load Categories

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.
- 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.
- See Editing Spreadsheets - Inserting, Deleting and Clearing Cells for quick ways to edit or add combinations.
- When a new installation is performed the program will back-up any existing XML files that contain load combination data rather than overwriting them. The back up copies will be given the extension BAK.

Customizing the Load Combination Generator

The Load Combinations for each region are contained in an XML file which can be opened and edited using a standard spreadsheet program. An example of on of these XML files is shown below:



	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R	S	T	U	V
1	Label	Solve	pDelta	SRSS	ASIF	CD	ABIF	Service	Hot Rolled	Cold Formed	Wood	Concrete	Masonry	Footings	Aluminum	Stainless	BLC	Factor	BLC	Factor	BLC	Factor
2	IBC 16-1	1	Y						1	1		1	1	1	1	1	DL	1.4				
3	IBC 16-2 (a)	1	Y						1	1		1	1	1	1	1	DL	1.2	LL	1.6	LLS	1.6
4	IBC 16-2 (b)	1	Y						1	1		1	1	1	1	1	DL	1.2	LL	1.6	LLS	1.6
5	IBC 16-2 (c)	1	Y						1	1		1	1	1	1	1	DL	1.2	LL	1.6	LLS	1.6
6	IBC 16-3 (a)	1	Y						1	1		1	1	1	1	1	DL	1.2	RLL	1.6	LL	0.5
7	IBC 16-3 (b)	1	Y						1	1		1	1	1	1	1	DL	1.2	RLL	1.6	WL	0.5
8	IBC 16-3 (c)	1	Y						1	1		1	1	1	1	1	DL	1.2	SL	1.6	SLN	1.6
9	IBC 16-3 (d)	1	Y						1	1		1	1	1	1	1	DL	1.2	SL	1.6	SLN	1.6
10	IBC 16-3 (e)	1	Y						1	1		1	1	1	1	1	DL	1.2	RL	1.6	LL	0.5

2018 IBC Strength | 2018 IBC ASD | 2016 ASCE Strength | 2016 ASCE ASD | 2015 IBC Strength | 2015 IBC ASD | ...

The name of the file itself becomes the name of the Load Combination Region that appears in the Load Combination Generator. Each XML file has a series of "worksheets" and the name of each worksheet will be the name of the Load Combination Code that appears in the Load Combination Generator. The user may add, edit or modify these documents to completely customize the available load combinations.

Note

- When the program is updated for new versions, the existing XML files will be "backed up" and saved with a '.bak' extension while the new XML files in the update will replace them. If any customizations were made to the XML files, they can be retrieved from the back ups.

The first row of each sheet is reserved for the column headers. The recognized column headers are as follows: Label, Solve, ASIF, CD, Hot Rolled, Cold Formed, Wood, Wood Products, and Concrete, Masonry, Footings, Aluminum, and Stainless,. In addition, there will be multiple pairs of Category & Factor headers.

Other than Label and the Category / Factor pairs, the user may omit columns. If a column is omitted, the default values will be used for those entries. The order of the column labels is optional except for pairs of Category and Factor. For the program to correspond the Factor with correct Category, the user should always provide Category label PRIOR to the corresponding Factor. The user may insert blank columns or columns with other labels than described above. In those cases, the program omits those undefined columns.

Note

- The program will read these files from the directory specified in Tool - Application Settings - File Locations. See Customizing RISA for more information.
- 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.

Load Generation - Notional Loads

Notional Loads are used by some building codes for the stability design of a structure. They serve as a minimum lateral load, or as an alternative to modeling the actual out-of-plumbness or out-of-straightness of the structure. Instead of changing the geometry of the structure, an equivalent de-stabilizing load is added to the structure. There are numerical benefits to handling this out-of-plumbness issue with loads rather than geometry. Essentially, it is quicker and easier to adjust the loading on a structure than it is to modify the stiffness matrix of the structure.

The implementation of these notional loads is not based on a single code, but on the concept of using lateral forces equal to a percentage of the applied vertical load at each floor level. Codes that may require the use of notional loads include the following:

- ASCE 7: A minimum lateral load of 1% of the Dead Load of the structure should be applied at each floor as a notional load.
- AISC 360: A notional load to account for out-of-plumbness of the structure of 0.2% to 0.3% of the total gravity load (DL + LL) 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.

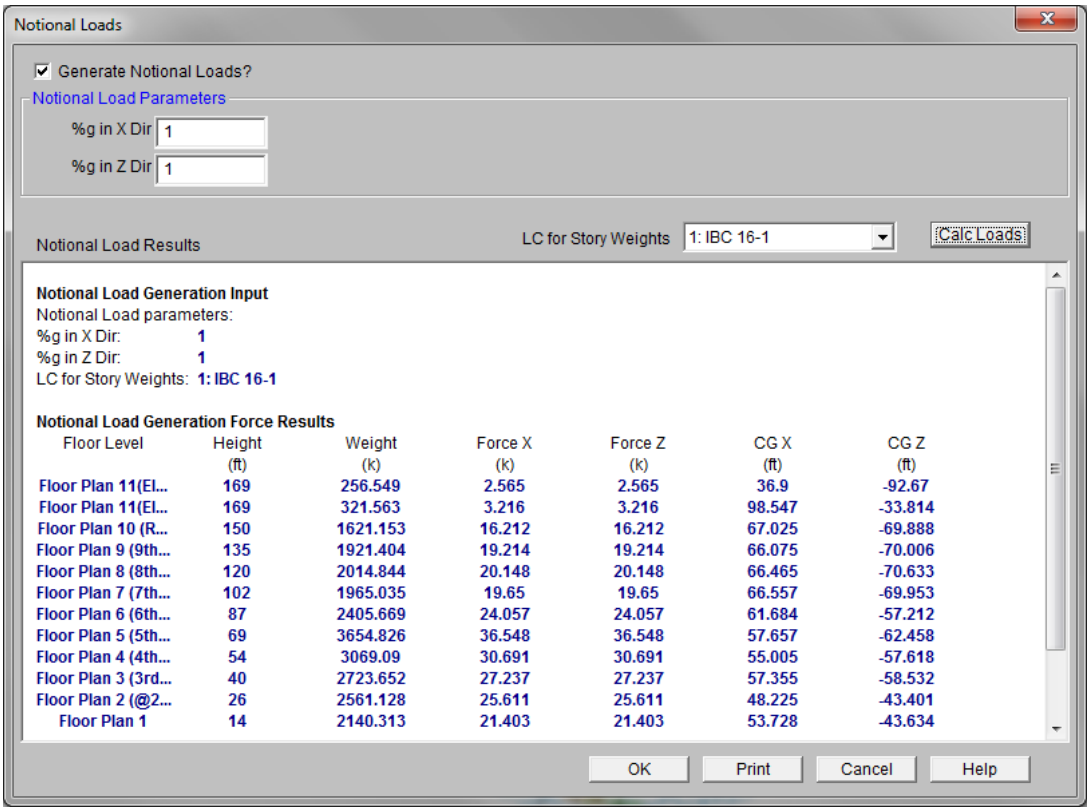
For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Notional Loads**.

Vertical Load used for Notional Force Calculations

The **Weight** of each floor is the total tributary weight associated with it depending on the Load Combination chosen for calculating these forces on the Notional Loads dialog box.

Notional Load Generation Dialog

The notional load dialog only asks for a load combination and the % of gravity loads that should be applied in the lateral direction. Notional loads are only generated for a single load combination. Therefore, the user is encouraged to choose a load combination which includes the most severe loading.



Floor Level	Height (ft)	Weight (k)	Force X (k)	Force Z (k)	CG X (ft)	CG Z (ft)
Floor Plan 11(El...	169	256.549	2.565	2.565	36.9	-92.67
Floor Plan 11(El...	169	321.563	3.216	3.216	98.547	-33.814
Floor Plan 10 (R...	150	1621.153	16.212	16.212	67.025	-69.888
Floor Plan 9 (9th...	135	1921.404	19.214	19.214	66.075	-70.006
Floor Plan 8 (8th...	120	2014.844	20.148	20.148	66.465	-70.633
Floor Plan 7 (7th...	102	1965.035	19.65	19.65	66.557	-69.953
Floor Plan 6 (6th...	87	2405.669	24.057	24.057	61.684	-57.212
Floor Plan 5 (5th...	69	3654.826	36.548	36.548	57.657	-62.458
Floor Plan 4 (4th...	54	3069.09	30.691	30.691	55.005	-57.618
Floor Plan 3 (3rd...	40	2723.652	27.237	27.237	57.355	-58.532
Floor Plan 2 (@2...	26	2561.128	25.611	25.611	48.225	-43.401
Floor Plan 1	14	2140.313	21.403	21.403	53.728	-43.634

Notional Load Results

The program does not automatically re-calculate them whenever changes are made to the RISAFloor loading in the same way that Wind and Seismic Loads are calculated.

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.

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 will automatically calculate the center of mass and the 5% accidental eccentricity for the various seismic load cases. If a greater eccentricity is desired, it may be specified in the Diaphragms spreadsheet in RISA-3D.

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 floor is the self weight of all members associated with that particular floor diaphragm as well as any applied loads utilizing the Dyn Mass load category.

Self Weight of Structure

The self weight of the structure includes the deck, all gravity and lateral members, and all walls. While computing the seismic weight of each floor diaphragm, the column and wall weights in each story are equally distributed to the floors above and below the story. These items can be manually excluded from the seismic load by un-checking the desired elements on the **Seismic** tab of the **Model Settings Dialog**. For more information on these exclusions refer to the Model Settings - Seismic.

Note:

- Loads assigned to the Super DL category in Deck Definitions are treated as an additional deck self weight, not as an Applied Load.

Applied Loads

The program does not include all applied loads as part of the seismic weight. The only applied loads that are included are the ones assigned to the Load Category called "Dyn Mass". The super-imposed dead loads (PreDL or PostDL) are not included. The reason for this exclusion is due to the fact that the super-imposed dead load is often a

very conservative number (accounting for the inconsistencies and unknowns in the construction process). That load may be overly conservative for the calculation of seismic weight.

Therefore, the user must specify a "Dyn Load" or "Dyn Mass" for each Area Load, Line Load, and Point load that they apply to their structure. This "Dyn Mass" represents the additional load (above and beyond self weight) that must be included in their calculation of Seismic Weight.

Note:

- For DynLoad line loads which fall partially within a given diaphragm, only the portion of the line load within the diaphragm is included in the seismic weight for that diaphragm.
- Loads assigned to the Super DL category in Deck Definitions are treated as an additional deck self weight, not as an Applied Load.

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

Total Seismic Weight

The total seismic weight of the whole structure is the sum of the seismic weights associated with all floors 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 in RISA-3D.

The seismic force is calculated for each diaphragm only. Therefore, any seismic weight that is not connected to diaphragms at all (e.g. walls located outside of a slab edge) will not be included in the seismic force distribution.

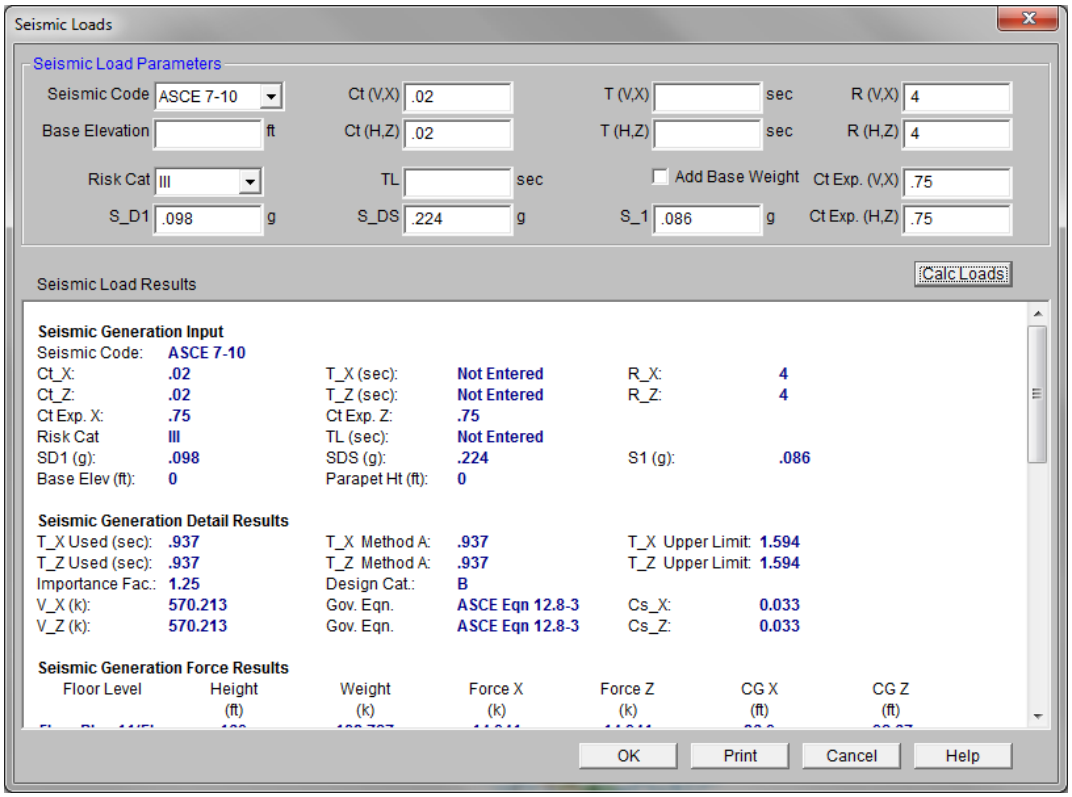
Note:

- For walls which fall partially within a given diaphragm the self weight of only the portion of the wall which is within the diaphragm is included in the seismic weight for that diaphragm.

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 on the **Seismic** tab of the **Model Settings**. These settings may also be changed or manipulated when changing from the RISAFloor interface to the RISA-3D interface.



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

Seismic Code currently allows you to choose the code which will be used for seismic load generation. For reference, sections of the 2016 edition of ASCE 7 will be 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 will calculate 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.

Ct is the building period coefficient as defined in 12.8.2.1 of ASCE 7-16. It is used in conjunction with the Ct 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 Ct and the Ct exponent "x" and the program will use 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 will be assumed to be the base elevation.

Risk Categoryis used to determine the importance factor assigned to the structure per table 1.5-2 of ASCE 7-16.

S_D1 represents the 5% damped spectral response Design acceleration for a 1.0 second period.

S_DS represents the 5% damped spectral response Design acceleration for short period response.

S_1 represents the 5% damped spectral response Mapped acceleration for a 1.0 second period.

TL represents the point at which the structural response is assumed to transition from a velocity controlled response to a displacement controlled response. These values are shown on Figures 22-14 through 22-17 in the ASCE 7-16.

Seismic Load Results

When you activate RISA-3D via the Director Menu, the program will calculate the appropriate seismic loads and present the calculations in a printable report. You may open the seismic load generator at any time to view, print, or recalculate the seismic loads. When you change something in the RISAFloor model that changes the seismic loads they are automatically updated the next time you enter RISA-3D.

Seismic Generation Input

This section displays the user 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 Usedand are the periods which was actually used to determine an upper limit for Cs 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 Method A is the approximate period calculated per equation 12.8-7 of ASCE 7-16.

T Upper Limit is the maximum allowable period to be used in equations 12.8-3 and 12.8-4, calculated per Table 12.8-1 of ASCE 7-16.

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.

Cs 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.

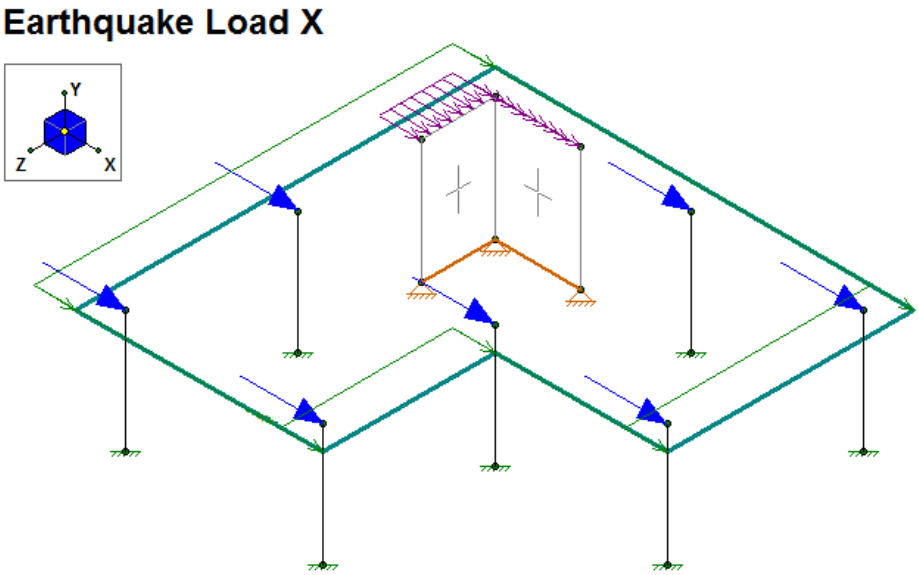
Note

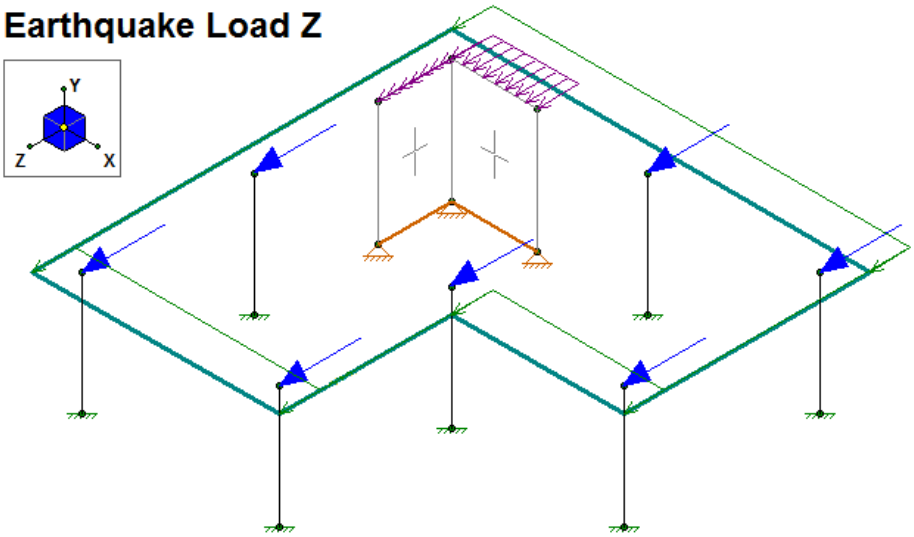
- The magnitudes of the seismic loads are based on the settings under the Seismic tab of the Model Settings dialog.
- The Joints / Nodes used to apply seismic load may appear as a "diamond" shape pattern of unattached nodes at that diaphragm level. While these nodes may not be attached to any beams or 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.
- To account for self supporting cladding (which needs to be included in the seismic weight, but not the regular dead load) create a line load that ONLY has a "Dyn Mass" load case specified for it.

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 will apply a Diaphragm Surface load which represents the seismic contribution of the Slab weight and any additional Dyn Load. There will be 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" will also be 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.





Note

- 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.
- 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)

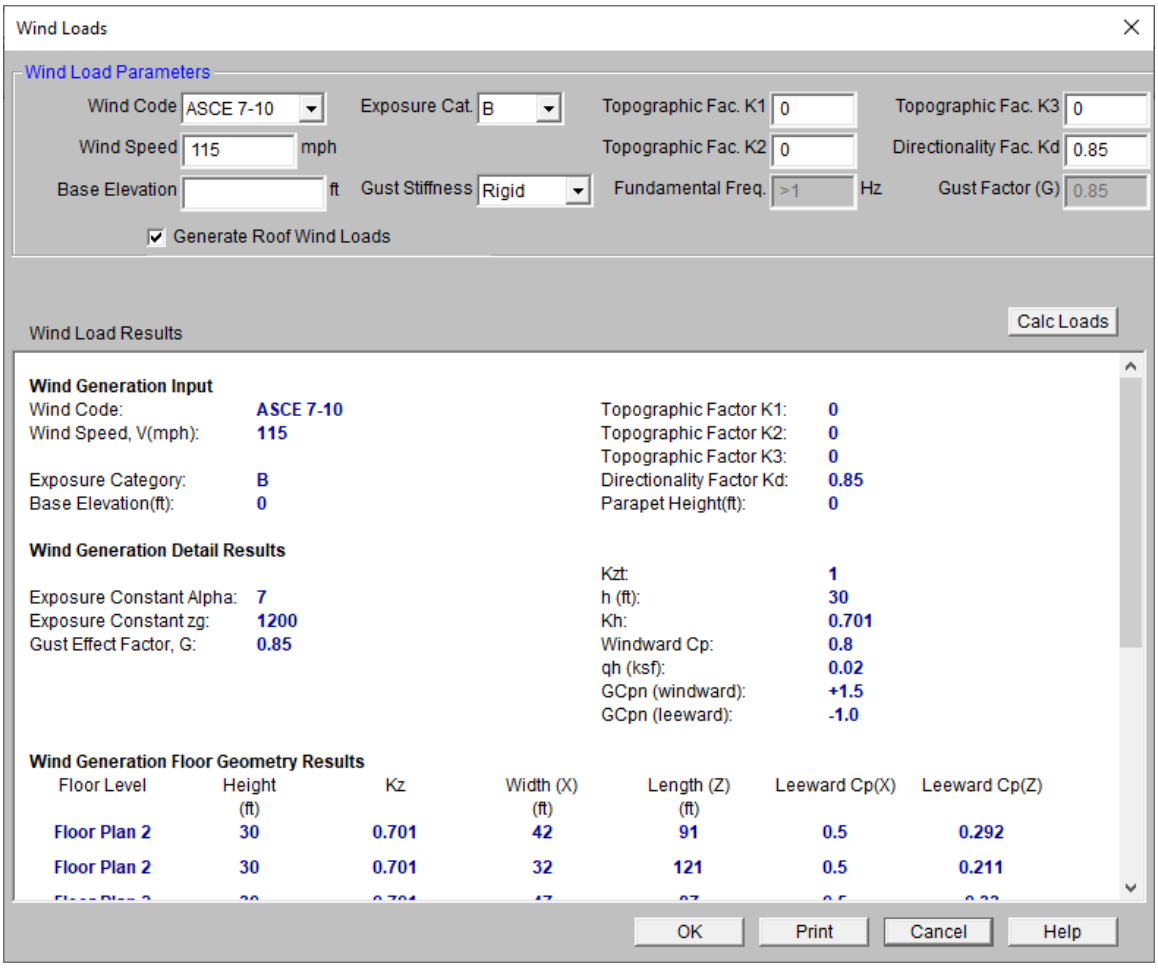
Note:

- 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 Load Parameters

The parameters used for automated wind load generation may be viewed or changed on the **Wind** tab of the Model Settings. These settings may also be changed when the wind load dialog pops up when transferring between RISAFloor and RISA-3D.

The parameters and results shown below are specific to ASCE 7-16, however the concepts apply to all wind codes. For questions specific to other wind codes contact RISA Technical Support.



Wind Code specifies which code will be used to generate the loads.

Wind Speed (V) is used to calculate wind pressures. See ASCE 7-16, 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 will appear and can 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.

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, Ke is permitted to equal 1 for all elevations). This option will appear and can 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 may open the wind load generator at any time to view, print or recalculate the wind loads.

Changes made to RISAFloor models which are integrated with RISA-3D models will be reflected in automatically updated wind loads when transferring between the programs.

Wind Generation Input

This section reports values which were directly or indirectly input from the Wind Load Parameters.

Wind Generation Detail Results

This section reports important values that were calculated using the Wind Load Parameters, as well as the height and base elevation of the structure.

Importance Factor is calculated using the input Occupancy Category.

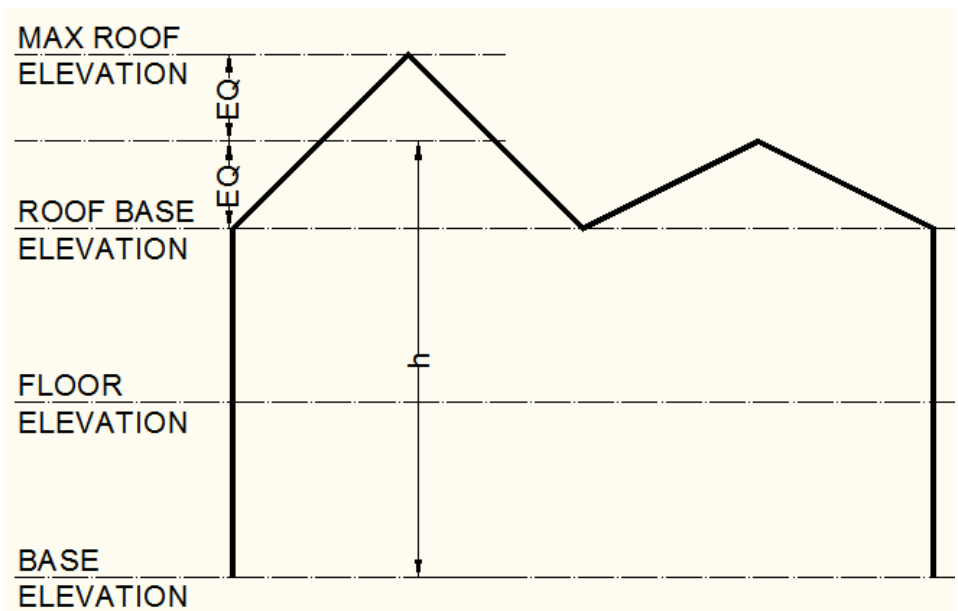
Exposure Constant Alpha (α) is determined from ASCE 7-16, Table 26.11-1 based on the input Exposure Category.

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 (K1, K2, K3)

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 (Kh) 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.

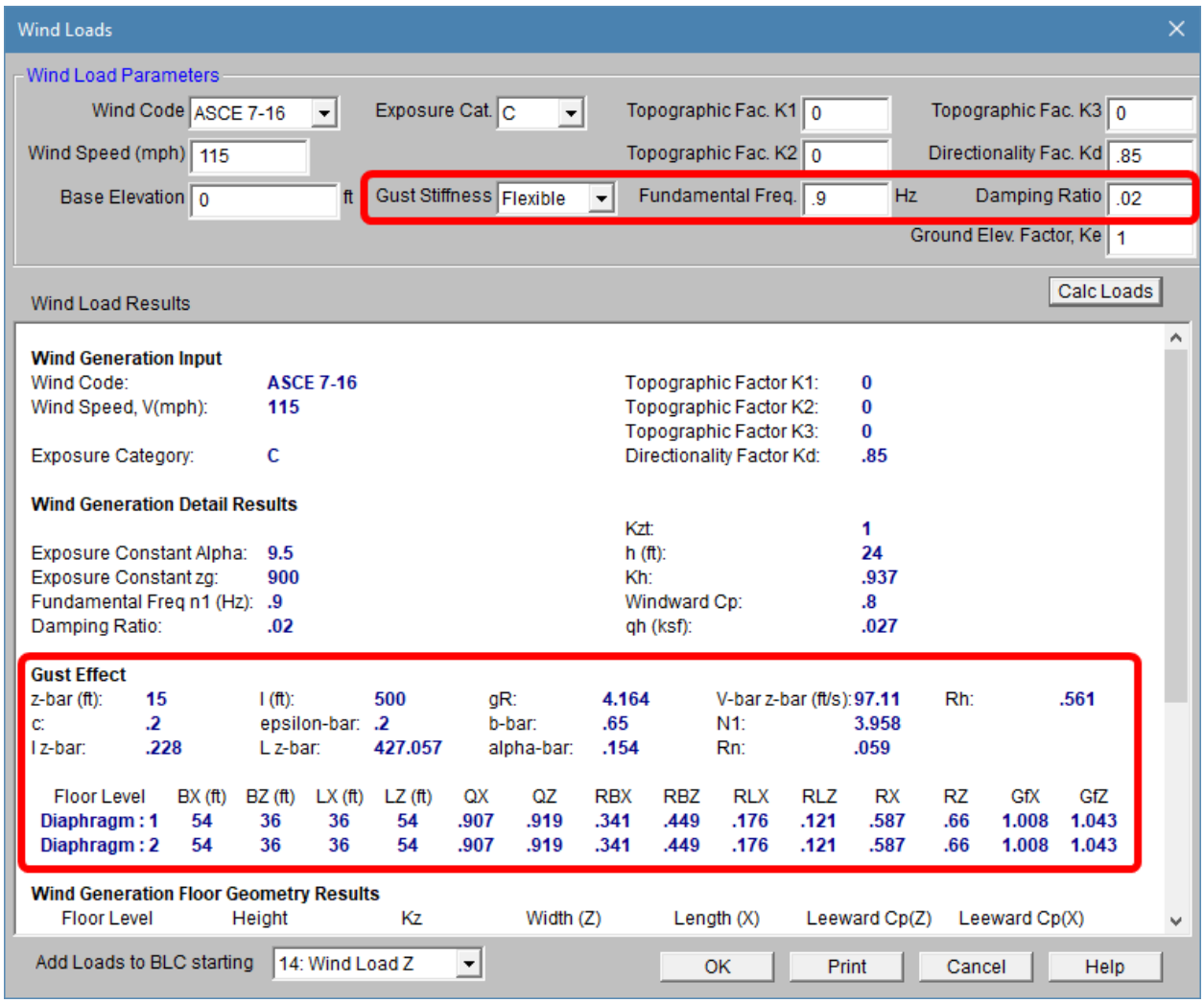
Mean Roof Velocity Pressure (qh) is calculated per ASCE 7-16, Section 26.10.2. The input Directionality Factor and Wind Speeds are used.

Parapet Wind Pressure Coefficient (GCpn) is calculated per ASCE 7-16 Section. 27.3.4 for both windward and leeward directions.

Gust Effect

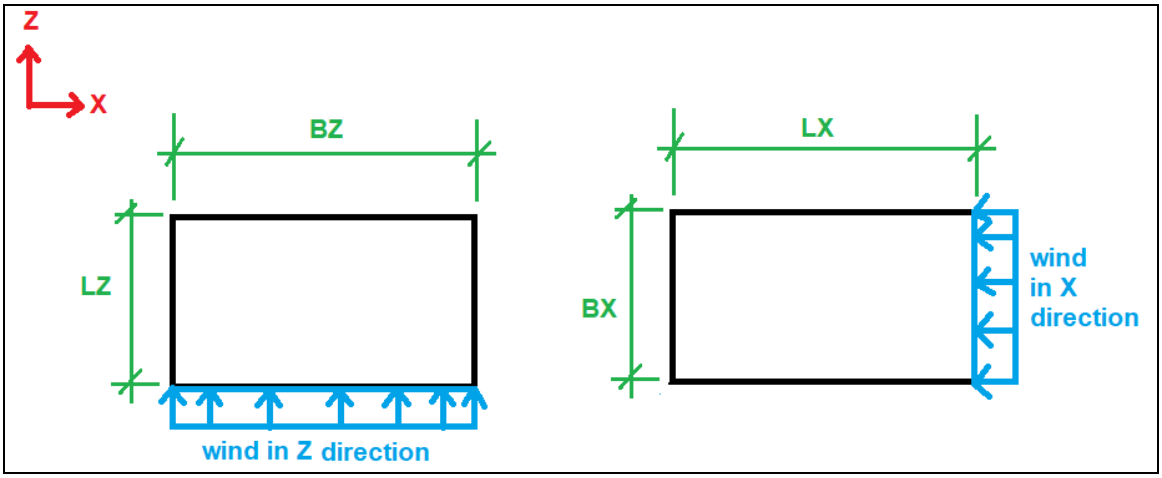
When the Gust Stiffness is set to Flexible, this section reports the values used in the calculation of the gust effect factor.

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*).



- 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 $\mathbf{z_{bar}}$, **c, l, epsilon**$\mathbf{_{bar}}$, $\mathbf{b_{bar}}$, **alpha**$\mathbf{_{bar}}$ are determined from *ASCE 7-16 Table 26.11-1*.
  - $\mathbf{I_{z\text{-}bar}}$ is calculated per *ASCE 7-16 Eqn (26.11-7)*.
  - $\mathbf{L_{z\text{-}bar}}$ is calculated per *ASCE 7-16 Eqn (26.11-9)*.
  - $\mathbf{V_{z\text{-}bar}}$ is calculated per *ASCE 7-16 Eqn (26.11-16)*.
  - Variables $\mathbf{g_Q}$ and $\mathbf{g_v}$ are taken as 3.4 per *ASCE 7-16 section 26.11.5*.
  - $\mathbf{g_R}$ 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

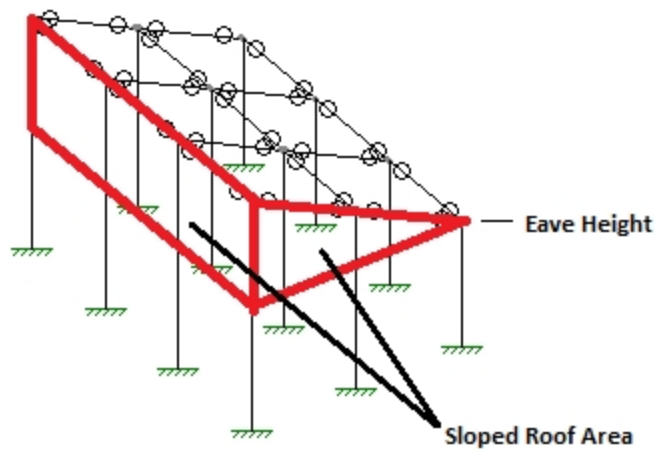
Sloped Roof Area is used to account for the wind load on walls which project above the base roof elevation due to a sloping roof. This only occurs in models which are integrated with RISAFloor. The program determines area by

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		
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.

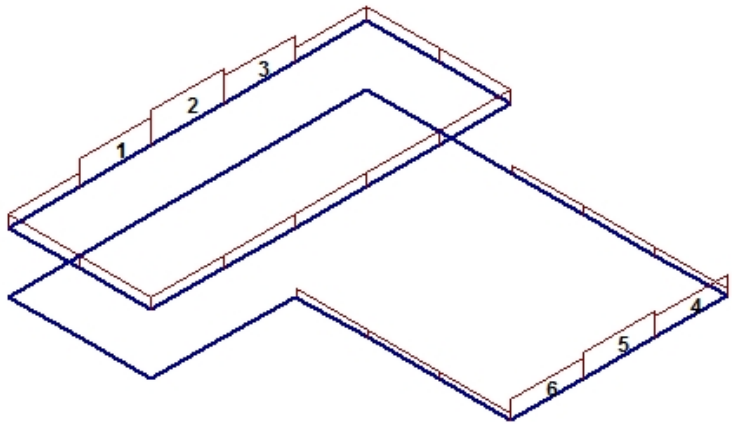
Wind Generation Floor Geometry Results

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)



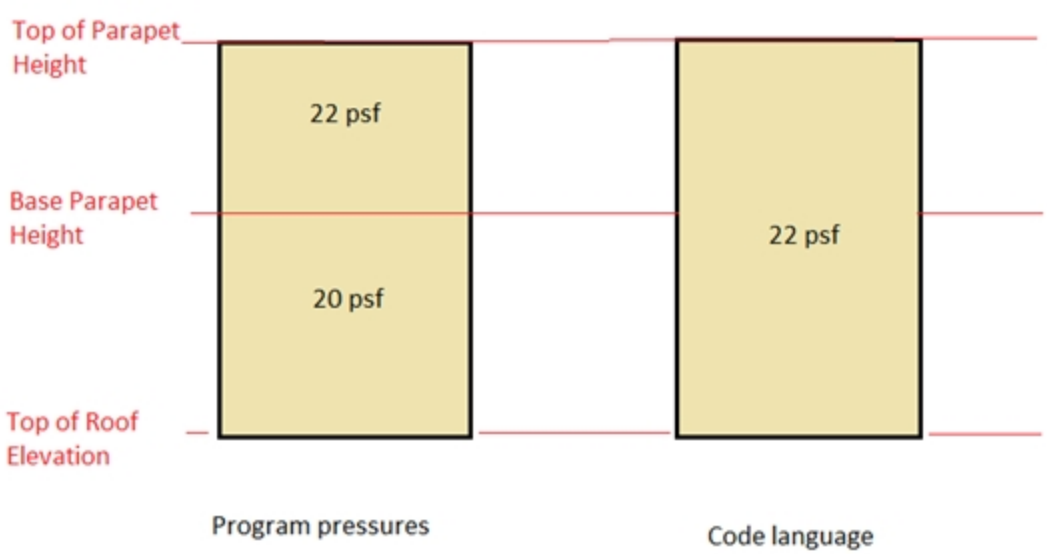
Parapet	Height Above	X Length	Z Length	qp (ksf)	Force X (k)	Force Z (k)
1	5	0	10	.02	1.528	0
2	8	0	10	.021	2.494	0
3	5	0	10	.02	1.528	0
4	5	0	10	.018	1.372	0
5	7	0	10	.019	1.964	0
6	5	0	10	.018	1.372	0

Note: GCpn = 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**.

Floors

	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	☐	10	Shear	☐	3
2	Roof	20	Roof	Floor Beam	Wood	90	None	☐	7.5	Moment	☐	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.

Columns

Column Primary Data | Steel/Wood | Concrete | Shear Cap | Detailing

1: Second Floor

	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	☑	Default (10)	Default (Shear)	None	NA	Fixe
11	CS11_L1	N10	Wide Flange	A36 Gr.36	Lateral			Typical	☑	Default (10)	Default (Shear)	None	(Default) 3	Fixe
12	CS12_L1	N11	Wide Flange	A36 Gr.36	Lateral			Typical	☑	Default (10)	Default (Shear)	None	(Default) 3	Fixe
13	CS13_L1	N12	Wide Flange	A36 Gr.36	Lateral			Typical	☑	Default (10)	Default (Shear)	None	NA	Fixe
14	CS14_L1	N9	Wide Flange	A36 Gr.36	Lateral			Typical	☑	Default (10)	Default (Shear)	None	(Default) 3	Fixe

Wall Panel Parameters

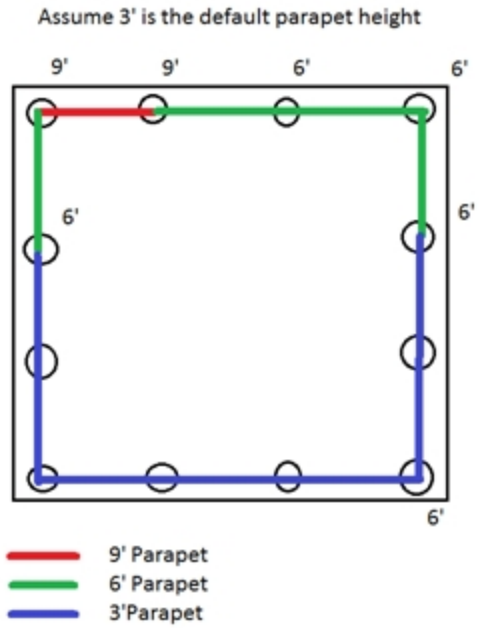
Primary | Advanced

	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:

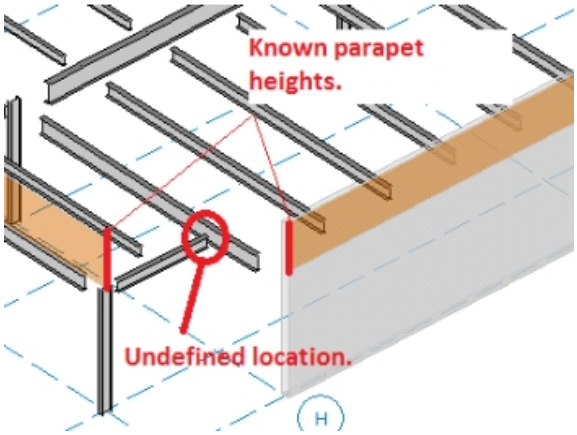
- Limitations for slab-supported floors:
  - Only the highest elevation floor in the structure can have a parapet.
  - Parapets must be a uniform height around the entire slab edge and the height is defined in the Floors spreadsheet.

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.

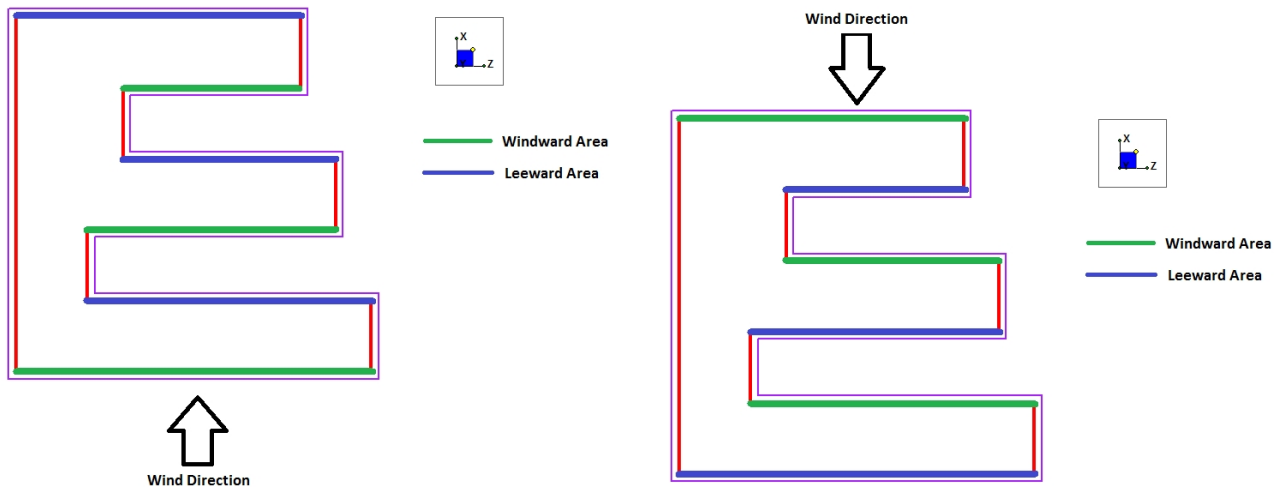
Note:

- 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.

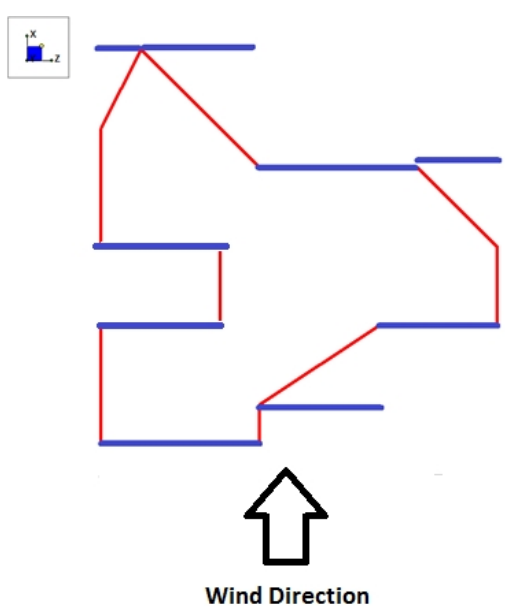
Windward/Leeward Pressure Considerations

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:

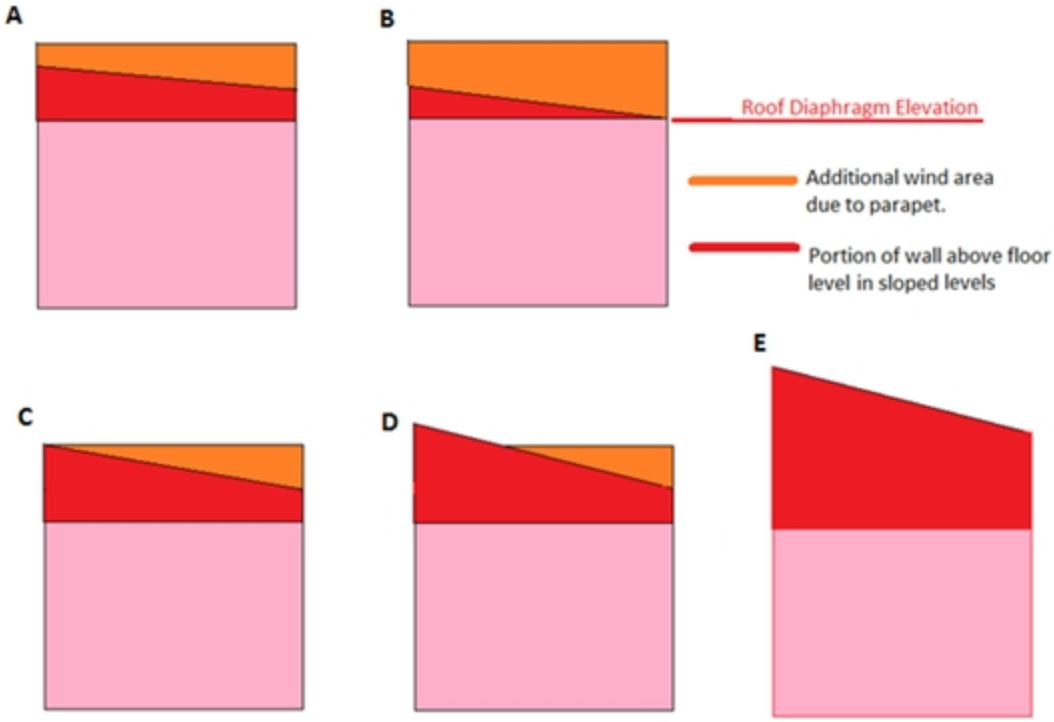
- For parapets that are non-orthogonal to the global axes, the program properly takes the projected area when calculating wind areas.



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.

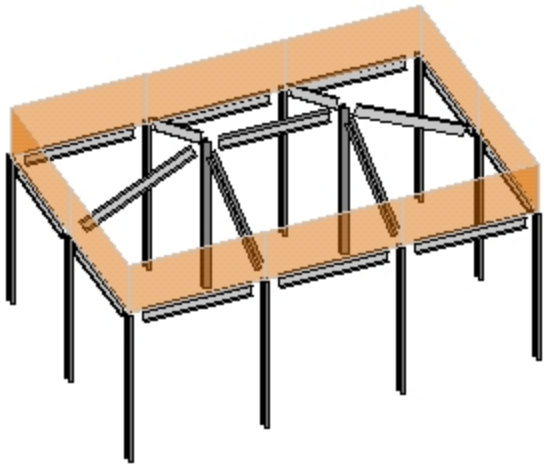


Note:

- 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.

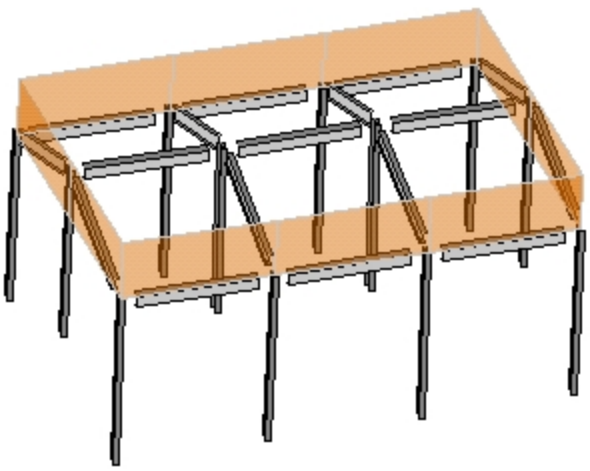
Here are some common conditions:

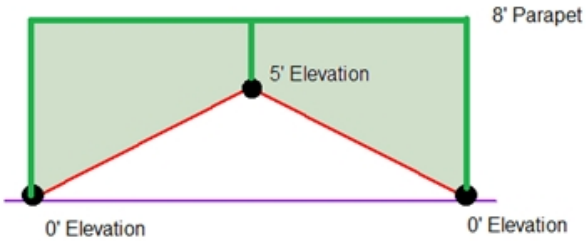
Gabled Roof with No Vertical Wind Faces and Parapets



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



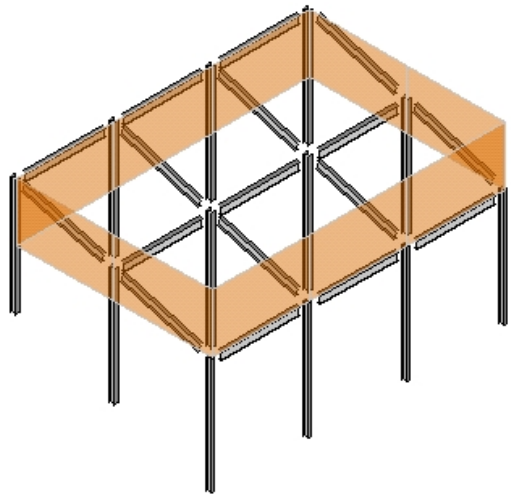


In this case the triangular face is taken as a regular wind load and the additional triangular/polygon loads that are due to the parapet (similar to Case A,B, or C) have their areas calculated and included as parapet loads.

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

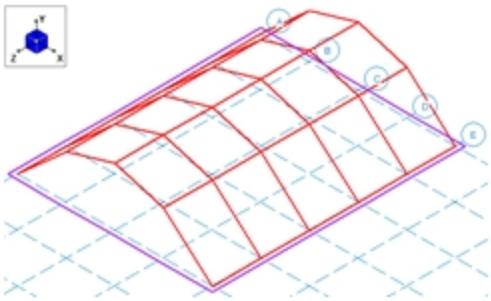
Monoslope Roof With Parapet

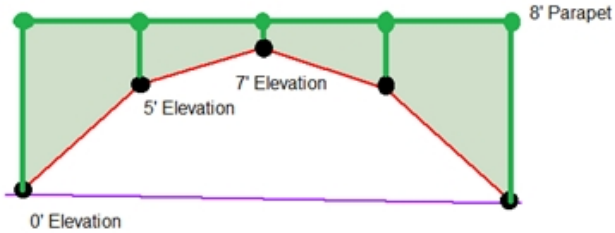


In this case there are triangular parapet wind areas on two sides (similar to Case C above) , a full parapet area on one side, and no parapet on the backside. In the triangular parapet direction the wind areas are equal in each direction thus the same value will occur in both directions. In the perpendicular 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

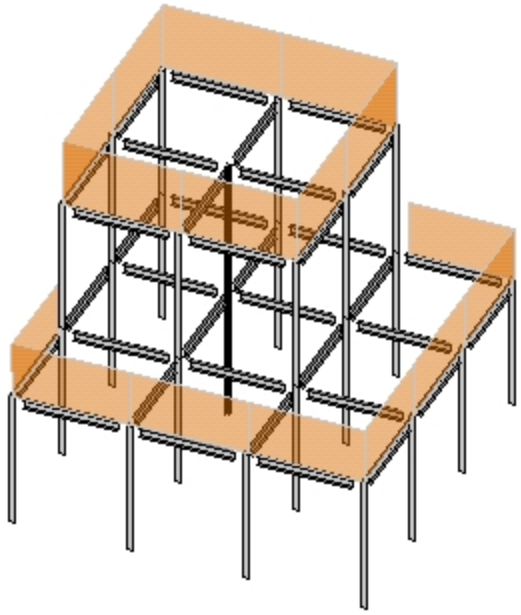
This is similar to the gabled with a slightly different geometry.





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 may designate any floor level as a mezzanine or "No Wind" level in the Floors spreadsheet.

Floors

	Label	Elevation[ft]	Area Load ...	Floor Type	Deck Default	Deck Angl...	Parent	Inacti...	Splice ...	Splice ...	No Wind/Drift	Parap...
1	Floor Plan 1	10	Roof	Floor Beam	Wood	0	None	☐	10	Moment	☐	0
2	Floor Plan 2	20	Roof	Floor Beam	Wood	0	None	☐	7.5	Moment	☑	0

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 of 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.

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

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

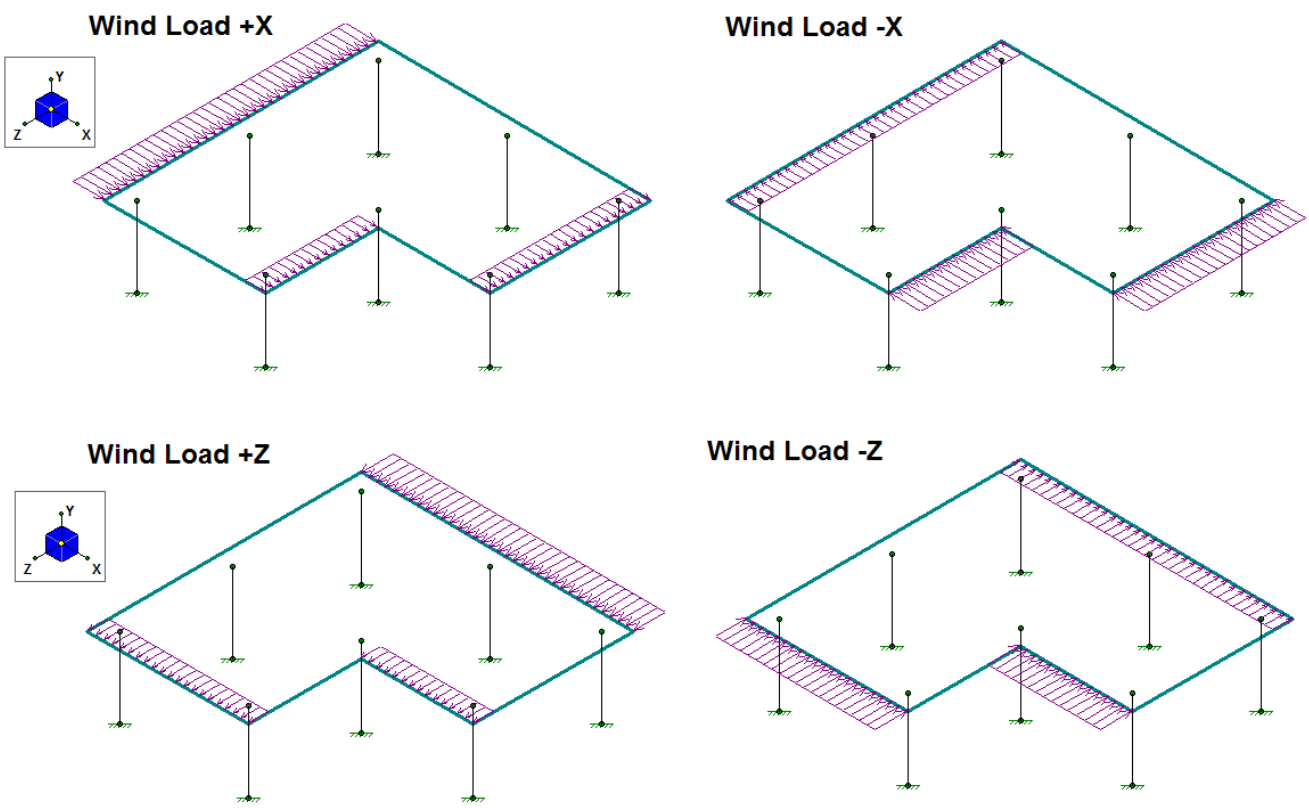
	BLC Description	Category	X Gravity	Y Gravity	Z Gravity	Joint	Point	Distributed
28	Semi-Rigid Wind Load +X	WL+X						3
29	Semi-Rigid Wind Load -X	WL-X						3
30	Semi-Rigid Wind Load +Z	WL+Z						3
31	Semi-Rigid Wind Load -Z	WL-Z						3

Diaphragm Distributed Loads

Member | Wall Panel | Diaphragm

BLC 28:Semi-Rigid Wind Load + | Next BLC | Prev BLC

	Diaphragm La...	Directi...	Magnitude...
1	S1	PX	.0351
2	S1	PX	.0351
3	S1	PX	.0562



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

Note:

- Both the static and dynamic wind procedures are considered.
- Cp is calculated from Figure I-15 of the User's Guide - NBC 2015 Structural Commentary.
- Ce is calculated from section 4.1.7.3.5 of the 2015 Building Code. Interpolation between low and rough terrain is not supported.
- Cg is calculated from section 4.1.7.3.8 of the 2015 Building Code. Note that for the static procedure Cg = 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

Note:

- Both the static and dynamic wind procedures are considered.
- Cp is calculated from Figure I-15 of the User's Guide - NBC 2010/05 Structural Commentaries.
- Ce is calculated from section 4.1.7.1.5 of the 2010 Building Code. Interpolation between low and rough terrain is not supported.
- Cg is calculated from section 4.1.7.1.6 of the 2010 Building Code. Note that for the static procedure Cg = 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 beams, columns, and walls. You may perform analysis using any type of material; simply define the properties for the material here. You may use up to 500 materials in a single model although most models will only have one or two. For example, your model might be made up of members of various grades of steel along with different concrete materials, timber or aluminum.

Material Properties Spreadsheet

The **Material Properties Spreadsheet** records the material properties to be used in the model and may be accessed by selecting **Materials** on the **Spreadsheets Menu**. The entries are explained below.

Hot Rolled Steel Properties

Hot Rolled | Cold Formed | Wood | Concrete | Masonry | General

	Label	E [ksi]	G [ksi]	Nu	Therm ...	Densit...	Yield[k...	Ry	Fu[ksi]	Rt
1	A992	29000	11154	.3	.65	.49	50	1.1	65	1.1
2	A36 Gr.36	29000	11154	.3	.65	.49	36	1.5	58	1.2
3	A572 Gr.50	29000	11154	.3	.65	.49	50	1.1	65	1.1
4	A500 Gr.B RND	29000	11154	.3	.65	.527	42	1.4	58	1.3
5	A500 Gr.B Rect	29000	11154	.3	.65	.527	46	1.4	58	1.3
6	A53 Gr.B	29000	11154	.3	.65	.49	35	1.6	60	1.2
7	A1085	29000	11154	.3	.65	.49	50	1.4	65	1.3

The values for A36 steel are the default material set. You of course don't have to use the A36 properties; you can change these and also add as many other materials as you need. You may then save your preferred materials as the default materials by clicking the **Save as Defaults** button on the **Window Toolbar**.

Label is the material label you wish to use to describe the entered material properties. This label is how you will reference the set of properties later when defining beams, columns, and walls.

E is Young's modulus that describes the material stiffness.

G is the shear modulus and may be left blank if you would like it calculated automatically. The equation for “G” is:

$$G = \frac{E}{2.0 * (1.0 + Poisson's\ Ratio)}$$

Note

- If you enter a value for Shear Modulus that does not match the value calculated using the above equation you will be given a warning.
- 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 is Poisson's ratio. Besides being used for the “G” calculation, this value is also used for shear deformation calculations. The value of Poisson's ratio may not exceed '0.5'.

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

Density is the material density and is used in the calculation of the member and plate self weight. This density times the area gives the self weight per unit length for the members; density times plate volume gives the total weight of a given plate.

Yield is the yield stress and is used only for Hot Rolled and Cold Formed steel design.

Hot Rolled Specific Material Data

The **Hot Rolled** tab records a number of hot rolled steel specific material properties that do not exist for the other materials. These entries are described below:

Hot Rolled Steel Properties

Hot Rolled | Cold Formed | Wood | Concrete | Masonry | Aluminum | Stainless | General

	Label	E [ksi]	G [ksi]	Nu	Therm ...	Densit...	Yield[k...	Ry	Fu[ksi]	Rt
1	A992	29000	11154	.3	.65	.49	50	1.1	65	1.1
2	A36 Gr.36	29000	11154	.3	.65	.49	36	1.5	58	1.2
3	A572 Gr.50	29000	11154	.3	.65	.49	50	1.1	65	1.1
4	A500 Gr.B RND	29000	11154	.3	.65	.527	42	1.4	58	1.3
5	A500 Gr.B Rect	29000	11154	.3	.65	.527	46	1.4	58	1.3
6	A53 Gr.B	29000	11154	.3	.65	.49	35	1.6	60	1.2
7	A1085	29000	11154	.3	.65	.49	50	1.4	65	1.3

Ry is the ratio of the expected yield stress to the specified minimum yield stress. This value is used in determining the required strength of an element for the seismic detailing checks.

Fu is the specified minimum tensile strength. This value is used per *AISC 358* to calculate C_{pr} which is in turn used to calculate the probable maximum moment at a plastic hinge in the seismic detailing calculations.

Rt is currently not used by the program. In a future version of the program this value will be used to calculate the expected tensile strength for the seismic detailing checks.

Cold Formed Specific Material Data

The **Cold Formed** tab records a cold formed material property that may not exist for the other materials. This entry is described below:

Cold Formed Steel Properties

Hot Rolled | Cold Formed | Wood | Concrete | Masonry | Aluminum | Stainless | General

	Label	E [ksi]	G [ksi]	Nu	Therm ...	Densit...	Yield[k...	Fu[ksi]
1	A653 SS Gr33	29500	11346	.3	.65	.49	33	45
2	A653 SS Gr50/1	29500	11346	.3	.65	.49	50	65

Fu (Ultimate) is the ultimate tensile stress.

Wood Specific Material Data

The **Wood** tab records a number of wood specific material properties that do not exist for the other materials. These entries are described below:

Wood Material Properties

Hot Rolled | Cold Formed | Wood | Concrete | Masonry | Aluminum | Stainless | General

	Label	Type	Database	Species	Grade	Cm	Emod	Nu	Ther...	Dens[k...
1	DF	Solid Sawn	Visually Graded	Douglas Fir-Larch	No.1	☐	1	.3	.3	.035
2	SP	Solid Sawn	Visually Graded	Southern Pine	No.1	☐	1	.3	.3	.035
3	HF	Solid Sawn	Visually Graded	Hem-Fir	No.1	☐	1	.3	.3	.035
4	SPF	Solid Sawn	Visually Graded	Spruce-Pine-fir	No.1	☐	1	.3	.3	.035
5	24F-1.8E DF Balanced	Glulam	Table 5A	24F-1.8E_DF_BAL	na	☐	1	.3	.3	.035
6	24F-1.8E DF Unbalanced	Glulam	Table 5A	24F-1.8E_DF_UNBAL	na	☐	1	.3	.3	.035
7	24F-1.8E SP Balanced	Glulam	Table 5A	24F-1.8E_SP_BAL	na	☐	1	.3	.3	.035
8	24F-1.8E SP Unbalanced	Glulam	Table 5A	24F-1.8E_SP_UNBAL	na	☐	1	.3	.3	.035

Type - Currently there are four wood types available: Solid Sawn, Glulam, SCL (this includes LVLs), and Custom.

Database - Based on your **Type** selection, the **Database** menu will filter to the appropriate databases.

Species - Based on your **Database** selection, the **Species** menu will filter to the appropriate databases. This is the wood species designation from your selected design code.

Grade is the wood grade designation from your selected design code. This has a drop down list where you can select the appropriate grade.

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).

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 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.

Note:

- Refer to Custom Wood to specify a custom wood material.

Commercial Species Groups:

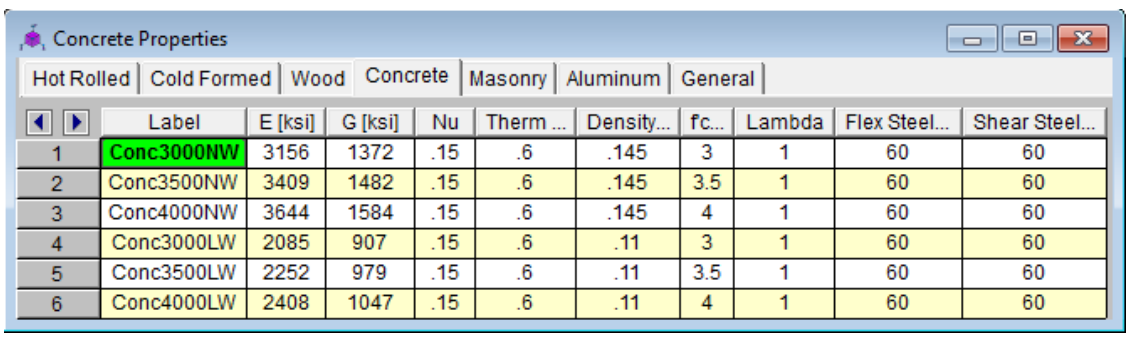
When either the NDS 05/08, 12, 15, or 2018 wood design code is selected, the Solid Sawn Species menu lists two new species listed that are not specifically shown in the NDS code. These are the *Commercial Species Group I - DF/SP* and *Commercial Species Group II - HF/SPFe Fir*. These are meant to be 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.

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.

Concrete Specific Material Data

The **Concrete** tab records a few concrete specific material properties that do not exist for the other materials. These entries are described below:



	Label	E [ksi]	G [ksi]	Nu	Therm ...	Density...	f'c...	Lambda	Flex Steel...	Shear Steel...
1	Conc3000NW	3156	1372	.15	.6	.145	3	1	60	60
2	Conc3500NW	3409	1482	.15	.6	.145	3.5	1	60	60
3	Conc4000NW	3644	1584	.15	.6	.145	4	1	60	60
4	Conc3000LW	2085	907	.15	.6	.11	3	1	60	60
5	Conc3500LW	2252	979	.15	.6	.11	3.5	1	60	60
6	Conc4000LW	2408	1047	.15	.6	.11	4	1	60	60

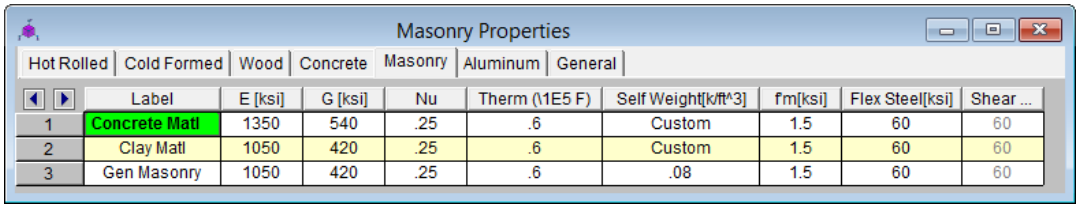
f'c is the concrete compressive strength used for concrete design.

Lambda, **λ** is the lightweight concrete modification factor. This factor only applies to the *ACI 318-14*, *ACI 318-11*, *ACI 318-08* and *CSA A23.3-04* codes. For all other codes the Density value of the material determines any strength reduction. The program will automatically calculate the correct value if it is left blank. Only values between 0.75 and 1.0 will be considered.

Flex Steel is the reinforcement yield strength for flexural bars in members and vertical bars in walls. **Shear Steel** is the reinforcement yield strength for shear bars in members and horizontal bars in walls.

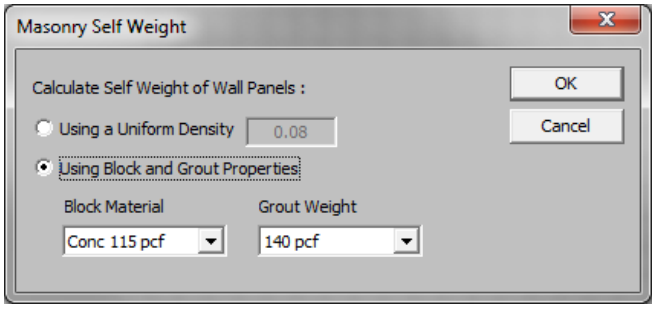
Masonry Specific Material Data

The **Masonry** tab records masonry specific material properties. Here we will describe items that do not exist for other materials.



	Label	E [ksi]	G [ksi]	Nu	Therm (\1E5 F)	Self Weight[k/ft^3]	f'm[ksi]	Flex Steel[ksi]	Shear ...
1	Concrete Matl	1350	540	.25	.6	Custom	1.5	60	60
2	Clay Matl	1050	420	.25	.6	Custom	1.5	60	60
3	Gen Masonry	1050	420	.25	.6	.08	1.5	60	60

Masonry **Self Weight** can be accounted for by two different methods. Entering a number for the density will result in masonry walls which have a self weight equal to that density multiplied by the wall cross-sectional area. Otherwise, click within the cell to launch the masonry self weight dialog:



Set the Using **Block and Grout Properties** option to have the program automatically calculate the self weight of the wall using the weights from tableB3 of the Reinforced Masonry Engineering Handbook (RMEH). The self weight will be listed as **Custom**.

f'm is the masonry compressive strength used for masonry design.

Flex and **Shear Steel** are the rebar yield strengths for flexural and shear bars used to reinforce the masonry.

Note:

- Masonry Shear Steel is automatically set to the Flex steel strength.
- Masonry materials are used only for walls in wall footings. See the Wall Footing Definitions spreadsheet for more information.

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	Concrete Members			Volume (yds^3)	
2	Conc3000NW	CRECT10X10	10	3.1	12.1
3	Conc3000NW	CRECT10X8	6	1.5	5.8
4	Conc3000NW	CRECT12X10	4	1.5	5.8
5	Conc3000NW	CRECT12X12	1	.4	1.7
6	Conc3000NW	CRECT12X8	9	2.7	10.4
7	Conc3000NW	CRECT14X10	8	3.5	13.5
8	Conc3000NW	CRECT16X10	1	.5	1.9
9	Conc3000NW	CRECT8X8	12	2.4	9.3
10	Concrete Slabs				
11	Conc4000NW		1	967.8	3788.8
12	Total Concrete		52	983.2	3849.4

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 Area * Density * Length for each member, again with offset distances deducted from the length.

For Concrete members, 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.
- Concrete slabs floors are included in the Material Take Off. But, walls and deck are not.
- RISA-3D only members will be included in the Floor Material Take-Off.

Modeling Tips

Dummy Members

Dummy Members are used to simplify the framing in order to properly attribute load in complex regions. They must be weightless members and they must run parallel to the deck. Doing those two thing assures that these members will not receive any load of their own and will only serve to break a complex region into two smaller regions.

To Change a Beam into a Dummy Member:

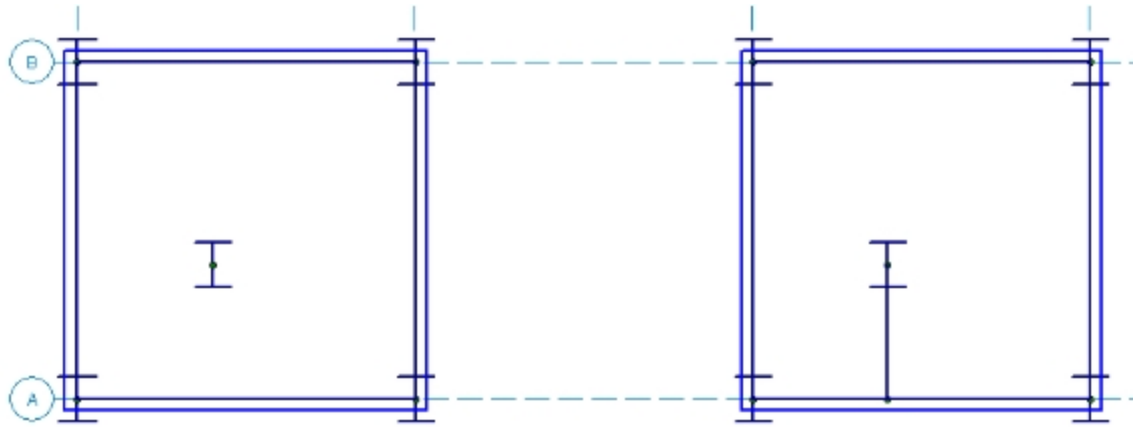
1. Make sure that the member you are changing has been drawn in parallel to the deck. Frequently, this will require the use of the “Beam to Beam” draw options shown below. Also, make sure to set the end releases to Pinned at both ends.
2. On the General tab of the Materials spreadsheet create a material Label called "Dummy". Use 29000 ksi for the value of E. Blank out the value for G by going to that field and pressing the space bar. Set the Density to zero. Leave all the other values as their defaults.
3. Use the Modify Beam Properties dialog to change the beam into a General Member with an the material type of "Dummy" and an explicit shape of “BAR1”.
    - Any shape will work. However, on-line shapes like BAR1 will be available for any database or region.

Note:

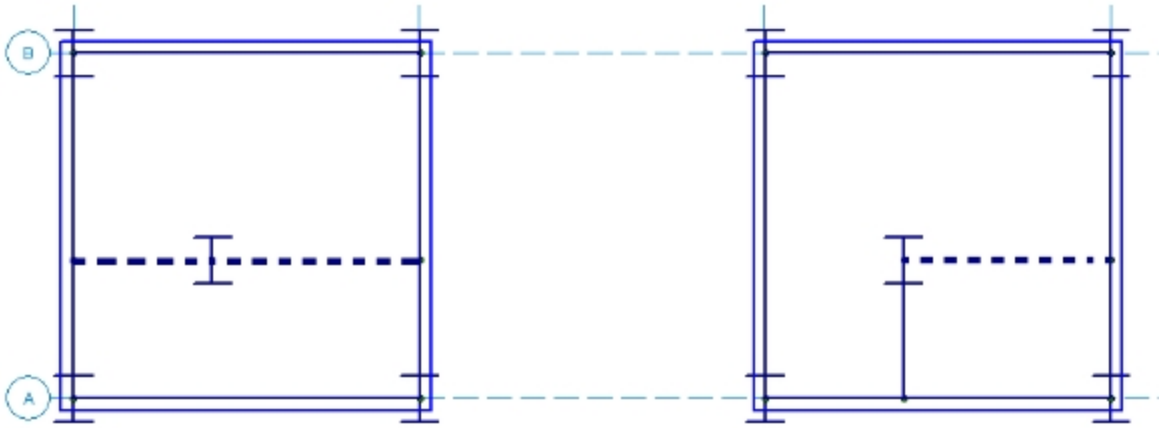
- The “Beam to Beam” options can also be used to draw from a wall to a beam or wall to another wall.

Stranded Columns

The image below shows two types of typical "stranded" columns. By calling it "stranded", we mean to point out that the framing around this column is incomplete. The load attribution code will tend to have problems with this type of situation. This sort of framing configuration will halt the solution. If properly detected by the program, it will result in a "framing error" which will identify the exact location of the framing issue. If not properly detected by the program, it could result in a more ambiguous error such as a Load Attribution or Polygon Error.

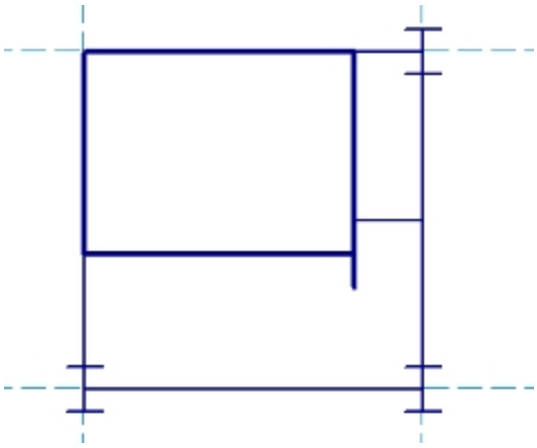


To correct this type of problem, you would typically add in a dummy member parallel to the deck that would frame out the column but would not receive any loading itself. An example of this type of correction is shown using the dashed lines in the image below. In order to avoid these types of framing problems a column must be connected on at least two sides:

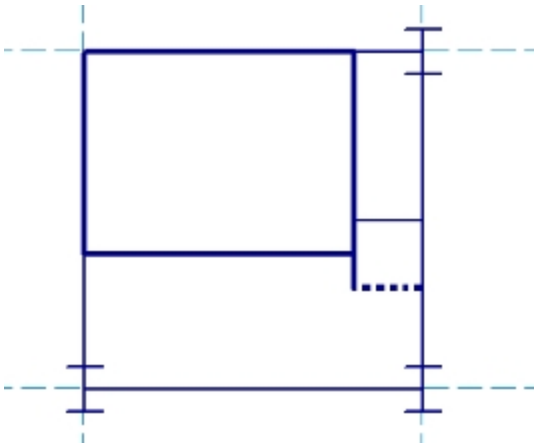


Stranded Walls

The image below shows a typical "stranded" wall. By calling it "stranded", we mean to point out that the framing around this wall is incomplete. The load attribution code will tend to have problems with this type of situation. This sort of framing configuration will halt the solution. If properly detected by the program, it will result in a "framing error" which will identify the exact location of the framing issue. If not properly detected by the program, it could result in a more ambiguous error such as a Load Attribution or Polygon Error.



To correct this type of problem, you would typically add in a dummy member parallel to the deck that would frame out the wall, but would not receive any loading itself. An example of this type of correction is shown using the dashed lines in the image below. In order to avoid these types of framing problems a wall must have at least one beam (or wall) framing into each end joint of the wall.



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 Divergence

Leaning Column Effect

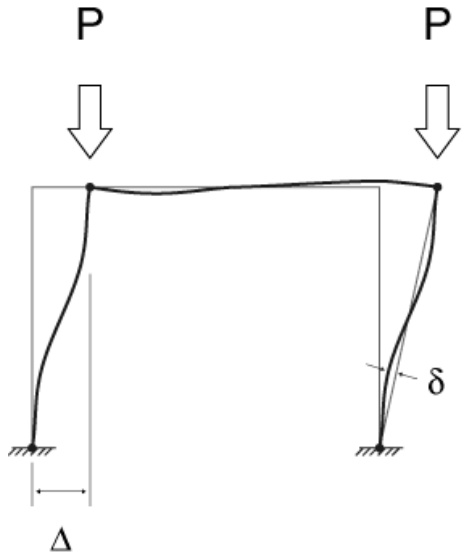
Gravity columns and walls that are modeled in RISAFloor, but which are NOT included in the Lateral analysis will still have an effect on the P-Delta analysis in RISA-3D. This is because RISA-3D automatically includes this leaning column / leaning wall effect. This is done only for columns or walls that 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.

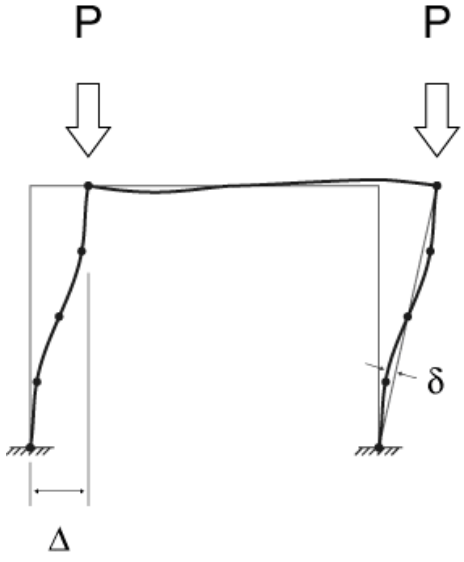
P-Little Delta Analysis

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 RISAFloor

Because RISAFloor does not perform a lateral analysis, the effects of P-Big-Delta are ignored for gravity design. The effects of P-Little-Delta are accounted for using the Non-Sway Moment Magnification Procedure of ACI 318-11 Section 10.10.6 or ACI 318-14 Section 6.6.4.5.

Note

- Per the commentary to Section 10.10.6 (ACI 318-14 Section 6.6.4.4.4), the EI value is assumed to be equal to $0.25*EI_{gross}$

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.

Points

Points are used to define the basic geometry of the structure or model. This includes columns, beams, walls, and slab edges. Points are also used to define load locations such as point loads, line loads, and area loads. Each Point is a point in space defined by coordinates in the horizontal and vertical directions or axes. Each floor level will have its own unique points.

Points are created automatically as you draw, but may be created manually from within the **Point Locations Spreadsheet**. Once created, either automatically or manually, the points may be edited within this spreadsheet.

To Define Points

- Select the **Point Locations Spreadsheet** from the **Spreadsheets Menu** and define the point coordinates.

Note

- You may use cut and paste, block fill, and block math to enter and edit points.
- You may choose the prefix that is used to label the points.

Rounding Off Joint Coordinates

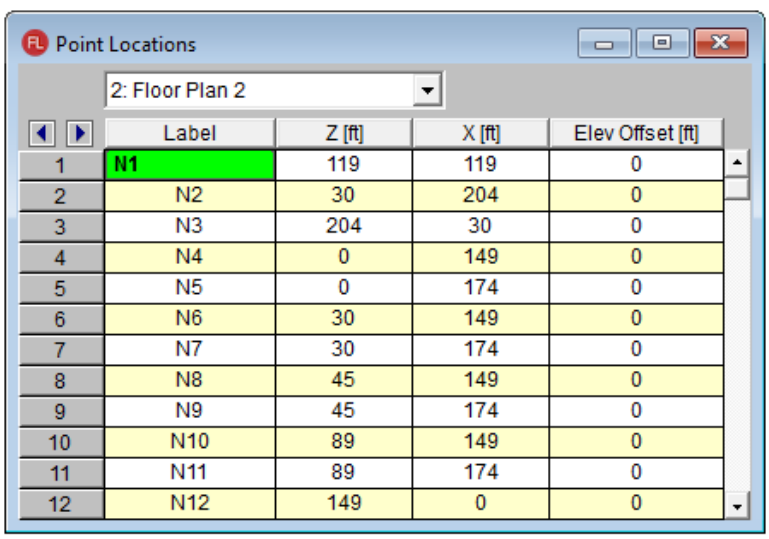
- Select the **Tools Menu** and choose **Round Off Joint Coordinates**.

Note

- This utility will round off all the joint coordinates to 1, 2, or 3 decimal places (user selected). This sometimes is useful for models that have been generated or created via DXF import and have joint coordinates with a large number of decimal places that aren't significant. Rounding off those coordinates makes the data more consistent and can help avoid problems such as non-coplanar plate/wall joints. Also, the default member orientations can change based on whether the member is exactly parallel to a global axis and slight unintended coordinate differences can have an impact here as well.

Point Locations Spreadsheet

The **Point Locations Spreadsheet** records the labels and coordinates for the points and may be accessed by selecting **Point Locations** on the **Spreadsheets Menu**.



	Label	Z [ft]	X [ft]	Elev Offset [ft]
1	N1	119	119	0
2	N2	30	204	0
3	N3	204	30	0
4	N4	0	149	0
5	N5	0	174	0
6	N6	30	149	0
7	N7	30	174	0
8	N8	45	149	0
9	N9	45	174	0
10	N10	89	149	0
11	N11	89	174	0
12	N12	149	0	0

The pull down list at the top of the spreadsheet allows you to toggle between floors.

Use the first column to assign a **Label** for each point. You can then refer to each point by its label. Each label has to be unique for that floor level. If you try to enter the same label more than once you will receive an error message. As you create new lines, the program will automatically create a new, unique label for each point.

The next two columns contain the coordinates of the point in the**Z** direction, or horizontal and the **X** direction or vertical directions. These represent the offsets of the points from the origin of the coordinate system (0, 0). The appropriate units are listed at the top of each column.

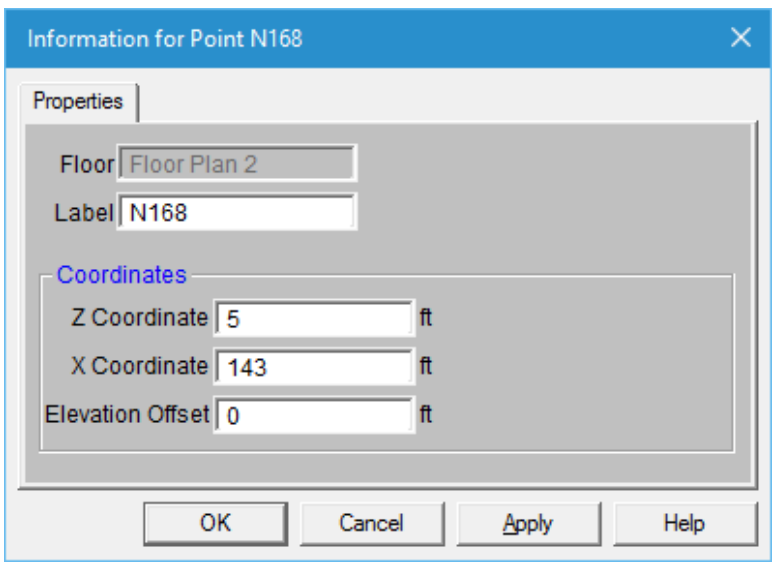
The last column contains **Elev Offset**. The elevation of the point is referenced for the floor level. If you enter a number larger than zero, the floor will be sloped.

Note:

- Only the points located at supports can have an Elevation Offset larger than zero
- The Elevation Offset value can not be entered into the spreadsheet negative. If a negative value is calculated for a cantilever, the value cannot be manually adjusted.
- Only the top floor level in the model is allowed to have an Elevation Offset larger than zero. This feature is meant for the use of sloping roofs.

PointJoint Information Dialog

Just as with the members and plates you You may double-click any pointjoint to view it's properties. All of the same information that is stored in the **Joints Points** spreadsheet is displayed for the pointjoint you choose, and may be edited. This is a quick way to view and change pointjoint properties. For large selections of pointsjoints however the spreadsheet and graphic editing tools may be the faster solution.



Floor - This reports which floor level the point is located on. It is not editable in this dialog.

Label - You can view and edit the pointjoint's label.

Coordinates - You can view and edit the pointjoint coordinates.

Elevation Offset - You can view and edit the vertical elevation offset (for points on sloping floors). This is only available on the top-most floor level.

Printing

You may print graphics as they appear on the screen or print all (or part) of the tabulated model information. If a spreadsheet is currently active and you click the print button the following options are presented. If a model view is currently active, and you select the print button, Graphic Print Options are presented.

If you have lines selected in the spreadsheet then you may print only those lines by selecting the first option. If you wish to print the entire spreadsheet you may choose the second option. The last option allows you to print multiple spreadsheets in a custom report that is discussed in the next section. You may combine the last two options with the Exclude feature to hide some data and print only the data you want.

To Print a Report or a Spreadsheet

- While in a spreadsheet click on the **Print** button and choose to print the current spreadsheet, a selected portion of the spreadsheet, or multiple sections by printing a report. If you are in a model view the Graphic Print Options are opened so click 'Print a Report Instead'.

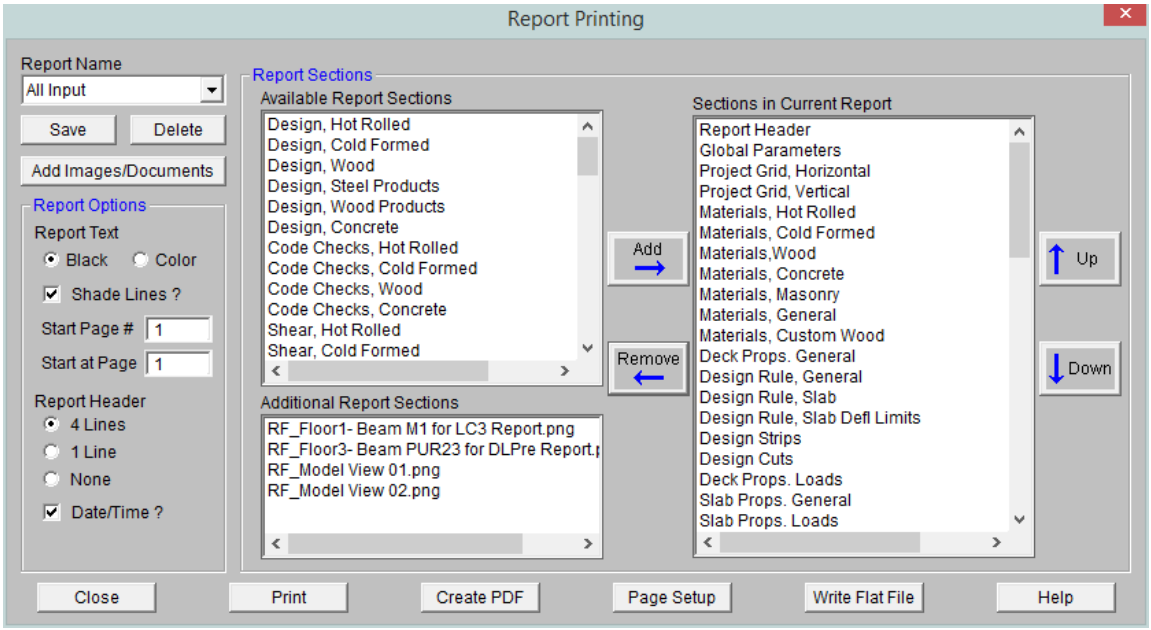
To Print Graphics

- While in a model view click on the **Print** button and choose from the options.

Note

- If you are not in a model view the Data Printing settings are opened instead. Close this and click on the model view that you want to print before clicking the **Print** button.

Printing Reports



The **Report Printing Dialog** options help you build your reports. There are standard reports for you to choose from and you may also name and save any report format you custom build. To choose a standard report, simply pick it from the **Report Name** drop down list.

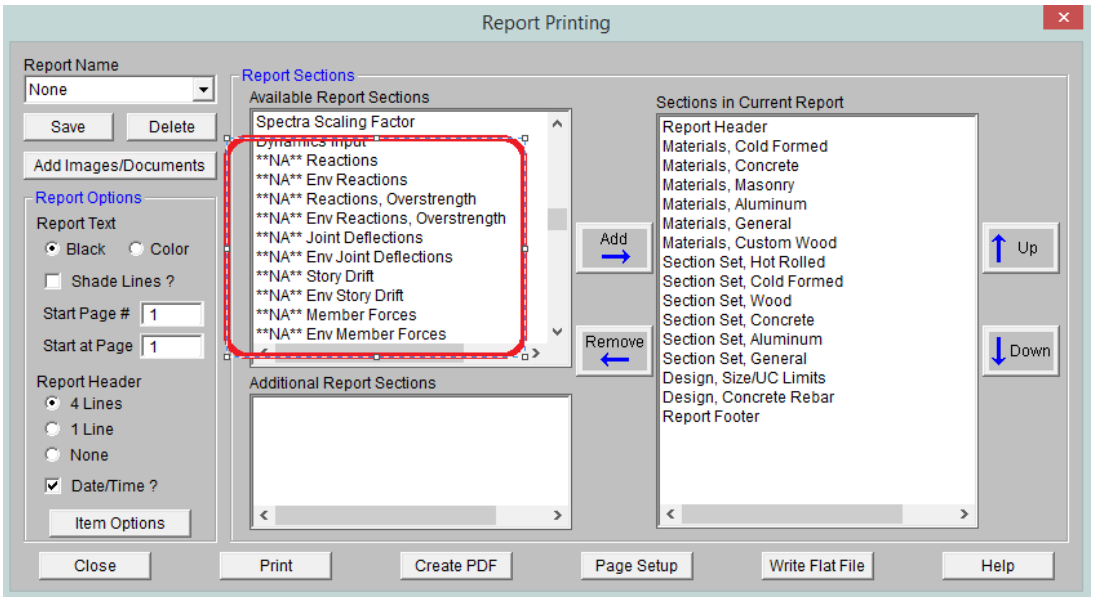
Building a Report

To build your own custom report you may double-click on report sections in order to move them from the available sections on the left, to the report defined on the right. You may use the mouse and the SHIFT or CTRL keys to pick multiple sections and then move them with the **Add** button. You can re-order the report by dragging and dropping the sections in your report you would like to movewith the **Up** and **Down** buttons and remove sections with the **Remove** button.

The **Sections in Current Report** portion of the dialog shows which sections and the order the printed report will be built. Report sections can come from either the **Available Report Sections** or the **Additional Report Sections**. The **Available Report Sections** contains every input and output spreadsheet in the program. The **Additional Report Sections** contains all images that were added to the project. Images can be added either using the "Snapshot" option or by adding external images using the **Add Images/Documents** button.

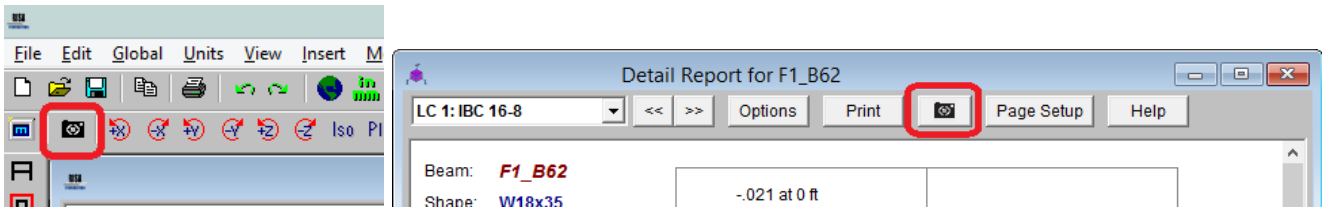
Note:

- Any saved reports from the Report Names drop-down will currently only save spreadsheet information. Any snapshots or images 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 Images

A snapshot image button is on the main graphic toolbar and most of the detail report windows.

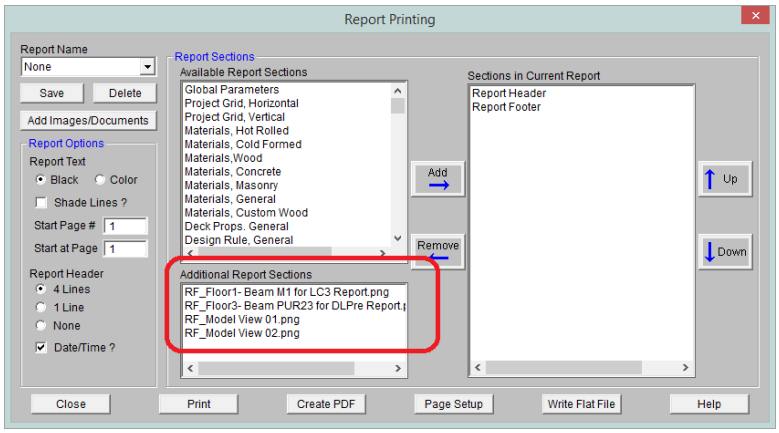


When this button is pressed a folder will be automatically created in the same location where the model file is located. This button will then give some options regarding the print location and file name. Once you define this information press Save and the program will create this image.

Note:

- If the model has yet to be solved the image will be created in the **Model Data Files** location from **Tools - 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 **Additional Report Sections** in the **Report Printing** dialog and can then be added to the **Sections in CurrentReport**.



Adding Images/Documents

Any additional images, documents that you wish to add to your report that are not from RISA, can be added using the **Add Images/Documents** button. Press this button and navigate to any PNG, PDF, or JPEG file and you can select it. Once that happens it will then show up in the **Additional Report Sections** and can then be added to the **Sections in CurrentReport**.

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 Options

Formatting options allow you to specify many options:

Report Text - Allows you to define the text color as black or blue.

Shade Lines? - Allows you to shade every other line to enhance readability.

Start Page # - Allows you select the starting page number. The number shown will be the next page number in the current sequence, but you can override this for occasions where you need to insert your calculation pages into an existing report and you need the page numbering to match.

Start at Page - Allows you to start your numbering at some page other than page one. This can be helpful if you have a title page or table of contents that you don't want to number.

Report Header - All reports have a footer with version information, the file name and path, and the page number. The single line header option will include the Model Title specified in the **Model Settings** along with the date. The triple line header adds company, designer, job number, and a place to initial any checking to the header.

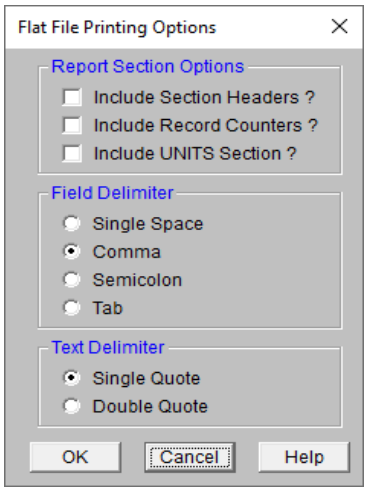
Date/Time? - Allows you to turn off the Date/Time stamp so that this will not appear on your printed reports.

Note:

- See the Customizing RISAFloor topic for more information about printing, including the ability to print out a RISA or custom company logo.

Printing to a File

A flat file is a file without column headings, print formatting, or graphical elements and is useful for importing and parsing into spreadsheets, database tables, or as post processor input data.



There are several options available to make the flat file easier to parse. Note that printing and then looking at a sample output file with all the options selected will make it easier to understand what the options do.

You have an option to include the 'Section Headers' which will print a text description of each block of data. For example, the Point Locations data would be preceded by a [POINT LOCATIONS] header on it's own line before the data. You may also include a 'Record Count' which is useful if you're writing looping code to read in the number of records within each data block. The number of records prints on it's own line. You can also have a special 'UNITS Section' be printed out which will give you the units used for all the values in the program.

The field delimiter options let you choose what character will be used to separate each field of data within record. For example, each coordinate value in the Point Locations data would be separated by the character selected. Programs that you might be importing the file into like MS Excel or MS Access often have options to select what the field delimiter will be for records of data.

The text delimiter works like the field delimiter, except that it's used to set apart text labels. All text in the flat file will be enclosed at the beginning and the end by the selected text delimiter character. This is very useful when trying to read in label strings that contain embedded spaces. As an example, the Point labels in the Point Locations data would each be enclosed by a single or double quote.

The current flat file options are saved each time the program is closed.

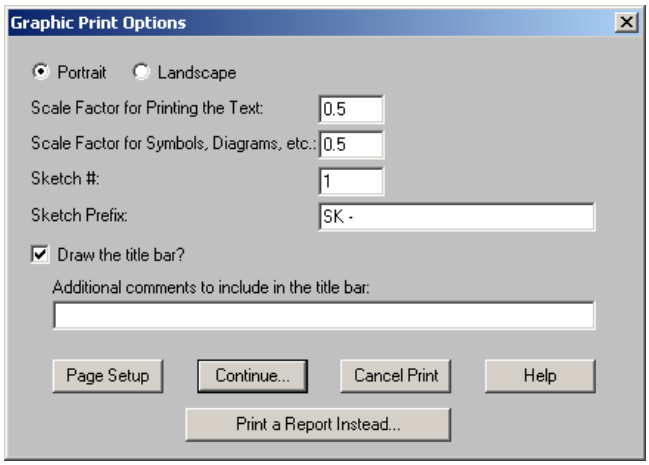
To Print to a Flat File

- While in the a spreadsheet click on the **Print** button, Choose **Use Report Printing Options** and click **OK**, then choose **Write Flat File**.

Graphics Printing

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**.

These options control the graphics printing. You may choose to print in **Portrait** or **Landscape** mode.



Next, two **Scale Factors** are defined. These scale factors are used to make the text and symbols displayed as part of the graphic, larger or smaller. A higher scale factor makes the text or symbol bigger. Since the resolution of the printer is probably much greater than the resolution of your screen, you can probably make the text and symbols smaller (by using a scale factor of less than 1.) than they appear on screen and they will still be easily readable. This makes for a cleaner looking graphic print. As far as what scale factors you should use, the only way to be sure is to experiment a little.

You may include the **Title Bar** 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 Bar. **Page Setup** allows you to set the margins.

You can specify a **Sketch #** and a **Sketch Prefix** to label your graphic prints as well.

Once you have everything the way you want it choose **Continue**. This will bring you to the print settings that are specific to your printer. The choices are different for each printer but generally allow you to choose the printer, orientation, quality and quantity.

The bottom button **Print a Report Instead** is provided so you can get to the report printing dialog directly from a graphic view.

Results

You may work with the results of a solution by viewing and sorting data in the spreadsheets, graphically plotting them with the model or by viewing detailed beam reports. You may also print the results in any of these forms. To learn about printing results, see Printing.

Upon the completion of a static solution, RISA opens the **Results Toolbar** and the **Design Results Spreadsheet**, and plots all the design shapes to the screen. If any warnings have been written to the **Warning Log**, it is displayed as well. You may then proceed to view any results and make any changes for further analysis.

If you make any changes to the model that would void the results, such as moving points or adding beams, 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

- Hot Rolled Design Results
- Cold Formed Design Results
- Wood Design Results
- Concrete Beam Design Results
- Steel Products Design Results
- Wood Products Design Results
- Deflection Results
- Vibration Results
- Beam End Reaction Results
- Wall Results
- Material Take-Off

Detail Reports

- Concrete Beam Detail Reports
- Concrete Column Detail Reports
- Beam Detail Reports

Saving Results

When you save a file that has been solved you may also save the results. The next time that the file is opened the saved results will be opened as well. You may use the **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 may access the result spreadsheets by selecting them from the **Results Menu**.

Choosing a Floor for Results

- The Floor of the results you are viewing is listed in the title of the spreadsheet. To change to another Floor select it from the drop down list on the toolbar.

Finding Results

- To go to a certain item while in a spreadsheet click the **Find** button, and type in the desired beam or point label.

Excluding Results

There are two ways to exclude results so that you can work with the results that are important to you.

Excluding Results After the Solution

You may run the solution and then graphically select the nodes, members, wall panels, slabs, and design strips that are of interest. By clicking the Exclude button, the spreadsheets and printed reports will be updated so that they will only have results for the selected items. All of the spreadsheets and reports will be controlled by this selection and you may adjust this selection at any time.

Excluding Results While in Spreadsheets

You may also select the results of interest on each spreadsheet. While viewing spreadsheet results you may select the last line of interest and exclude the rest. Exclusions are applied independently for each results spreadsheet. Any exclusion applied to the Member Forces spreadsheet will not affect the Member Stresses spreadsheet, etc. Excluding items graphically will reset **all** of the spreadsheets to match the graphic selection. If you wish to combine these two features to fine tune the results perform the graphic selection first.

To exclude results click on the last line of results you wish to keep and then right click and select click the **Exclude After**. You can bring the results back by right clicking in the spreadsheet and selecting **Unexclude**.

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 beams you may plot and color-code the members by bending, shear or deflection checks. See Model Display Options for more information.

To Plot Results Graphically

1. If you do not already have the model view open click the **New View** button on the **RISA Toolbar.**
2. Click the **Model Display Options** button on the **Window Toolbar**.
3. Select the options you would like to view and click **Apply**.

Clearing Results

You will be provided with a warning if changes are made to the model that might invalidate the current results. Should you decide to proceed, the results will automatically be cleared and you will have to re-solve the model to get results once you are finished making changes. This warning may be disabled in the **Application Settings** on the **Tools Menu**.

To manually clear results, click the **Clear Results** button.

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.

Member Detail Report

After you have performed a solution you may view a detailed report of the beam design results. This report contains detailed loading, force, and deflection diagrams for the beam members. Use the LC / Cat button to switch the diagram plots from Load Categories to Load Combinations, and the drop down menu to select the desired category or combination. The report also contains detailed design information on the shear and moment capacities, member deflection, end reactions, composite properties, and floor vibration. Use the << >> button to see the detail report for the next or previous member.

To View a Detail Report

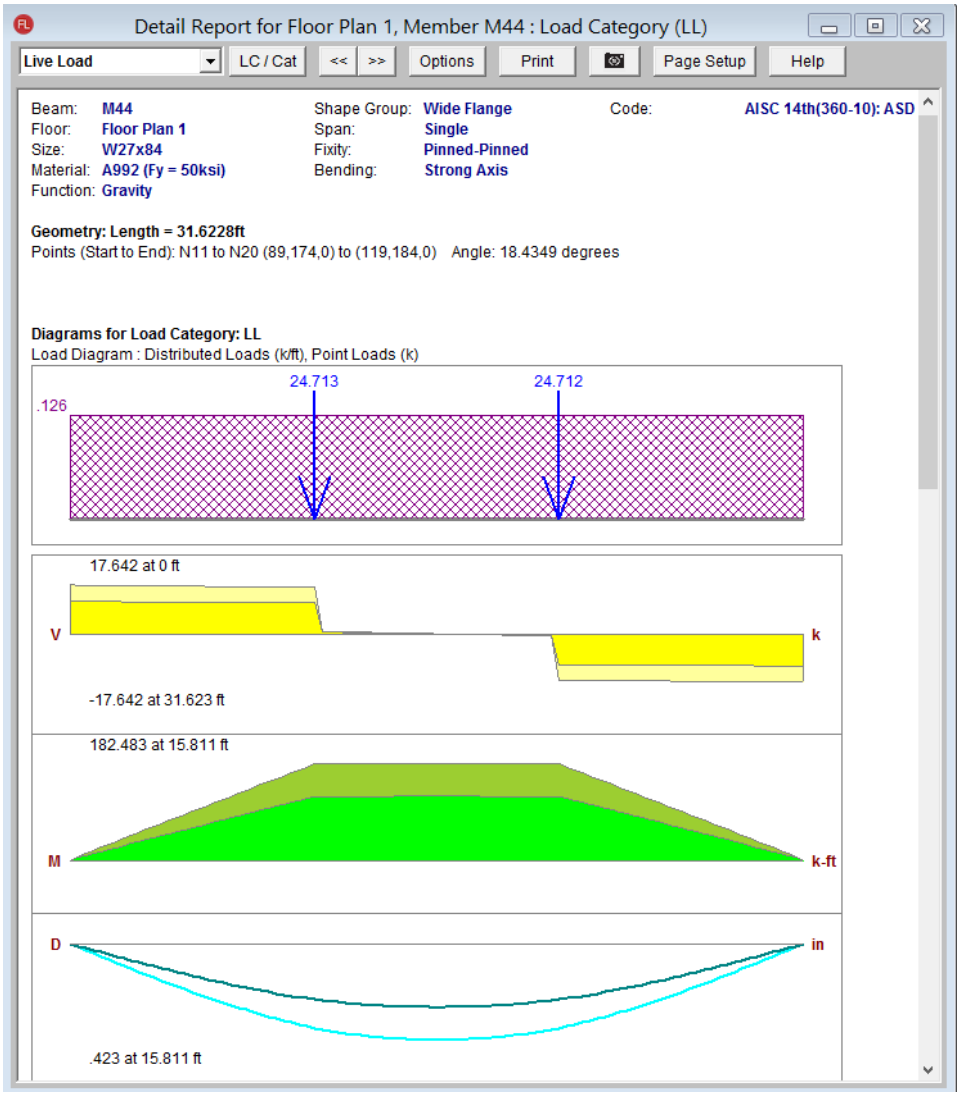
- To view a member detail report after running a solution, click on the Detail button on the Selection toolbar and then click on that member. If you do not already have the model view open click the **New View** button on the RISA toolbar.
- You may also access the detail report while viewing a member results spreadsheet such as **Design Results** by clicking the Detail Report for Current Item button on the Window toolbar.
- You may also use the **Results** ▸ **Members** ▸ **Detail Report** menu option to open the report for any member.
  - Once the detail report is open, select a Load Combination or Load Category Precomposite Dead Loa from the pull down list.
  - The detail report diagrams the load, force, and deflected state of the member. You can use the **Options** button to pick the details or diagrams that you wish to display or print.

Note:

- For sloped members, the load diagram will display the load applied in the local y-axis and the user applied loads will be in parenthesis.
- - This 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.

The available diagrams provided are:

Plot Designation	Plotted Value
Load Diagram	Applied Loads
V	Shear Force
M	Bending Moment
D	Deflection
A	Axial Force



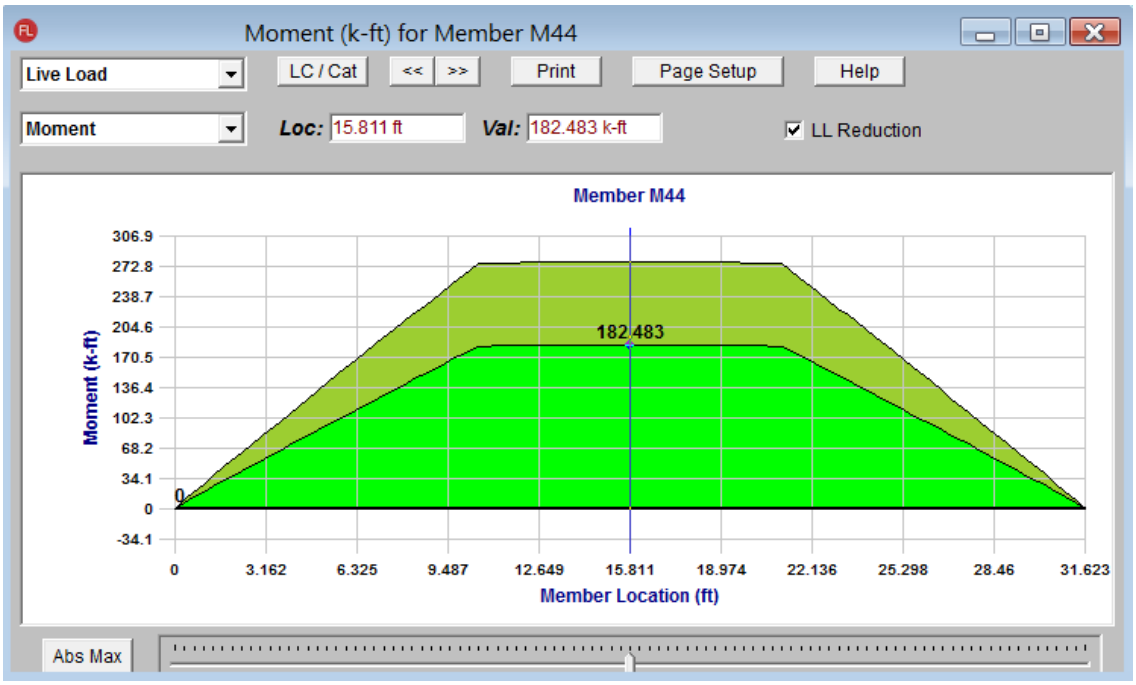
The load categories and load combinations that may be selected are outlined in the *Loads* sections of this manual. Refer to Loads - Load Types and Loads - Load Combinations for more information.

Note:

- The multiple colors shown in the diagrams represent live load reduction.
- To control whether the displayed diagrams and values consider LL Reduction, click on the Options button.

Interactive Diagrams

Once a beam detail report is open, you can also click on any of the shear, moment, deflection, or axial diagrams to open an interactive diagram. Use the cursor at the bottom to drag along the length of the member to see how the value changes.



Note:

- The multiple colors shown in the diagrams represent live load reduction. When the LL Reduction box is checked, the value displayed on the diagram will represent the reduced value. When the LL Reduction box is unchecked, the value on the diagram represents the unreduced value.

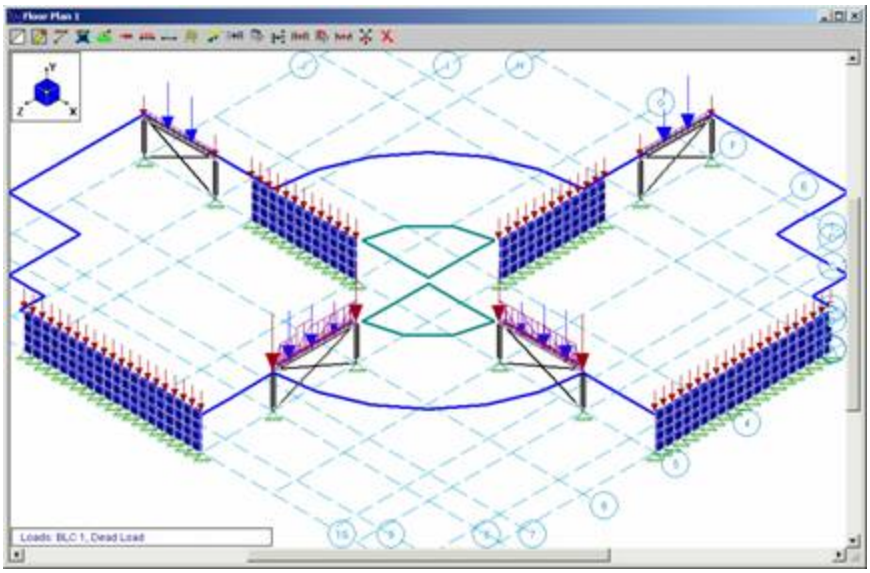
Concrete Member Detail Reports

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 and 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 of the manual 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 Misc Tab of the Model Display Options 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

	Elevati...	Mass[k]	Mass ...	Center of Mas...	Plus X Ecc...	Minus X E...	Plus Z Ecc...	Minus Z E...	Inactive	Diaphragm	Type	Region	Design Rule
1	12	62.399	4199.72	10, 10	%5	%5	%5	%5	☐	D2	Flexible	DR1	Lower Level
2									☐			DR2	Lower Level
3									☐			DR3	Roof Level
4									☐			DR4	Typical
5	12	247.759	116858	89.997, 34.453	%5	%5	%5	%5	☐	D3	Flexible	DR1	Typical
6									☐			DR2	Lower Level
7									☐			DR3	Roof Level
8	12	46.582	3964.757	45, 15	%5	%5	%5	%5	☐	D4	Rigid		

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.

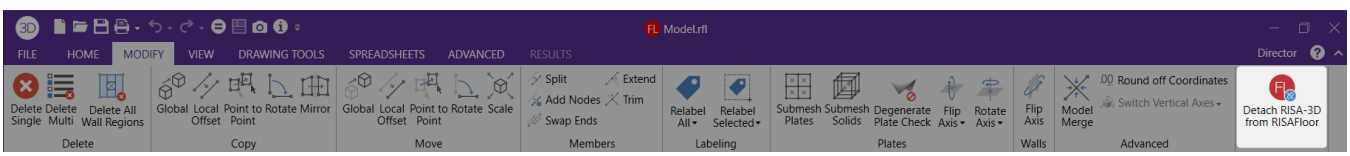
See Load Generation - Seismic Loads for more information.

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 tab, the **Detach RISA-3D from RISAFloor** option will detach the RISA-3D model from the RISAFloor model. This will delete any diaphragm information and generated loads.

RISAFoundation Interaction

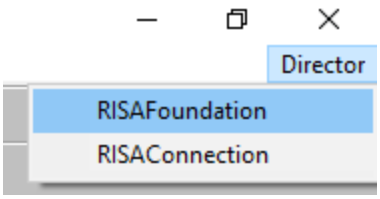
RISAFoundation has the ability to transfer load information from RISA-3D and RISAFloor and use this information to design foundations. The following entry will describe how these loads come across the design modules so you can accurately use RISAFoundation for your foundation design.

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 besplit into specific load categories (DL, LL, etc.) under the Basic Load Cases spreadsheet.
2. The model must be solved.
3. The **Director** button is used to then transfer to RISAFoundation.

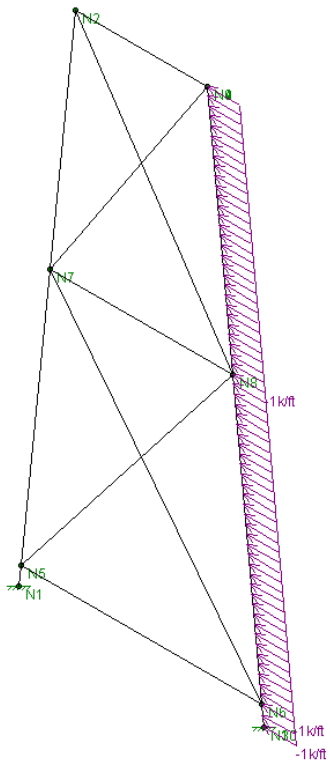


Note:

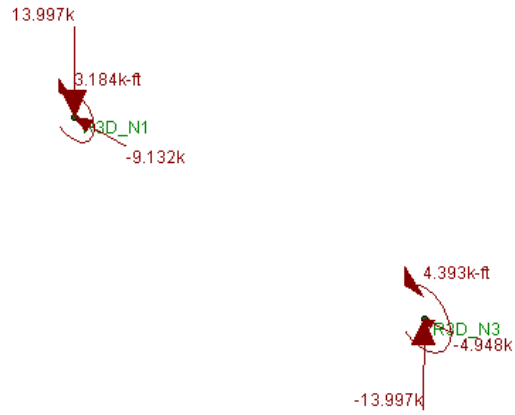
- If you do not define a load category for a basic load case, then the loads from that case will not be transferred into RISAFoundation.
- You must run at least a single load combination.
- When you are working between RISA-3D and RISAFoundation Model Settings will only be brought over to RISAFoundation 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 Interaction

A simple example would be the braced frame structure seen below on the left. This has been created in RISA-3D. The load applied to this structure came from Load Category WL (wind load). Note that this structure had other loads applied as well. But the wind loads are used as an example.



RISA-3D Wind Loads Graphical Representation



RISAFoundation Wind Loads Graphical Representation

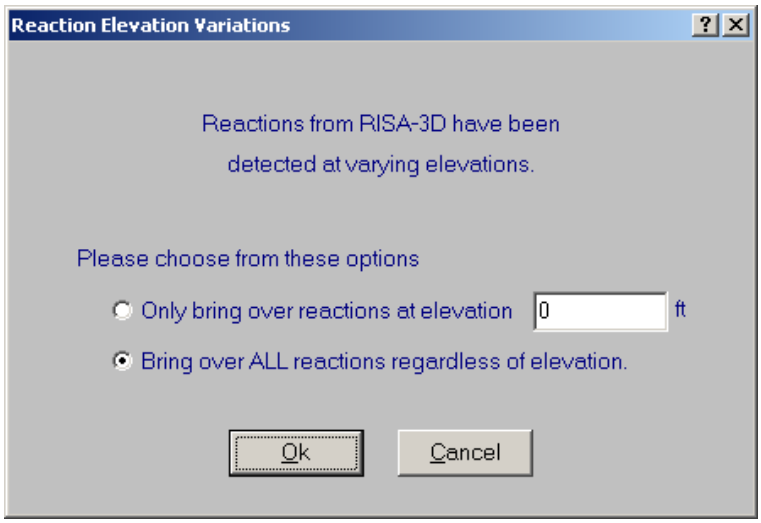
First a load combination is run (note that this does not have to have wind in the load combination), then RISAFoundation is chosen from the Director in RISA-3D. The Director is the means by which navigation to different modules in the program is possible. When RISAFoundation is first entered dead loads applied to points in a plan view of your structure are displayed. Clicking the Iso button will then get an orientation similar to the one shown above. Next, choose WL-Wind Load from the drop down list. WL-Wind Load The loads shown above on the right will be displayed. Note that these loads appear consistent for the type of loading present.

Note:

- The points are relabeled in RISAFoundation, though labeled in a way that can be easily compared to the original RISA-3D model joints. You can scroll through the different types of loads and compare them to RISA-3D for accuracy verification.
- Load combinations do not transfer over from RISA-3D, thus they will need to be re-created in RISAFoundation. An easy way to do this would be to just copy the spreadsheet over from RISA-3D.

Foundations at Different Elevations

If, when moving into the RISAFoundation from RISA-3D, there are foundations that are at different elevations, a dialog box will pop-up that looks like this:



You may choose to bring over reactions at a single elevation of your choice. This may be helpful if there are some reactions that are due to an adjacent structure or support that is not a foundation.

You may choose to bring over all reactions regardless of elevation. This is a good option to have if there is a partial basement level or any condition where there truly are foundations at different levels.

Note:

- All of the differing elevations will be brought in at a single elevation.

RISAFoundation Interaction with RISAFloor

This interaction is a little more involved because it involves three programs (RISA-3D, RISAFloor and RISAFoundation). For RISAFoundation, the bases of columns and the bottom of walls will be considered boundary conditions. Thus, loads are applied at the bases of these elements for foundation design in RISAFoundation.

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 force*distance about one end of the wall. Once P and M are calculated, the magnitudes at either end of the wall are calculated as:

$$I_{end} := \frac{4 \cdot P}{L_{wall}} - \frac{6 \cdot M}{L_{wall}^{2}} \qquad J_{end} := \frac{6 \cdot M}{L_{wall}^{2}} - \frac{2 \cdot P}{L_{wall}^{2}}$$

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

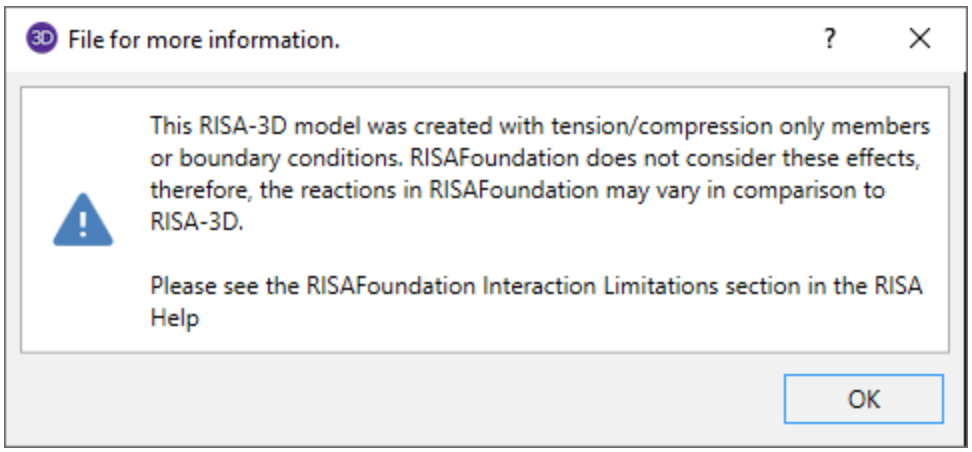
Moving Loads, not described as a load category, and their reactions can't be read as loading within RISAFoundation, thus loads will need to be applied manually to the foundations to account for these.

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.

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 Steel Joists are available?
- What Wood I-Joists are available?

Database Shape Types

There are different types of shapes for each material type including General shapes. Names for each shape type follow a syntax so that they may be typed directly into the **Shape** field on the **Primary** tab of the **s** spreadsheet. Alternately you may click the button to look up a shape and select it.

Hot Rolled Shapes

AISC, Canadian, Trade Arbed and custom Hot Rolled shapes are accessed by clicking the **Edit Shape Database** button from the **RISA Toolbar**, and then clicking the **Hot Rolled** tab from within the **Shape Database Editor** dialog. 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/Manufacturer** 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 **Edit Shape Database** button from the **RISA Toolbar**, and then clicking the **Cold Formed** tab from within the **Shape Database Editor** dialog. 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.

General Shapes

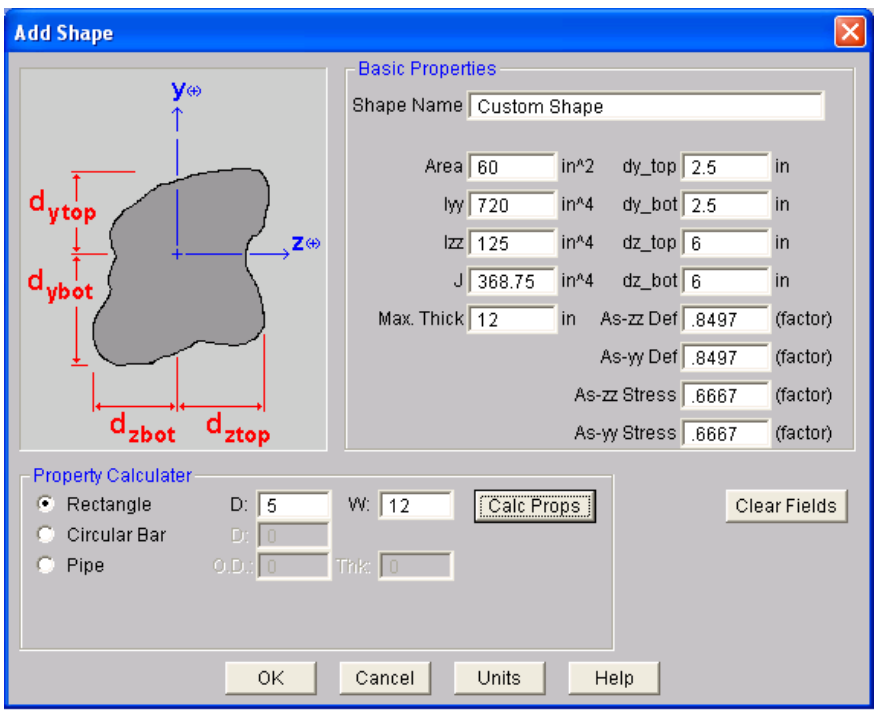
Arbitrary Shapes

Arbitrary Shapes are a special, catch-all shape. This arbitrary shape type is provided so that any shape can be added to the shape database.

AISC code checks are not calculated for arbitrary shapes since their place in the specification is unknown. Everything else will be calculated for them (forces, deflections, stresses). The max thickness (Max thick) value for the cross section is used to determine the pure torsional shear stress for the shape. "J" is the torsional constant. The "d" values (the distances to the extreme fibers) allow the program to calculate stresses at the extreme fibers.

Note:

- These shapes will generally be rendered using a greenish cruciform shape. The center of the cruciform will reflect the centroid of the section with the tips of the cruciform representing the distances from the neutral axis to the extreme fiber.
- Refer to Member Shear Deformations and Member Shear Stresses for more information on the shear area factors (As-zz Def, As-yy Def, .As-zz Stress, & As-yy Stress) shown in the figure below.



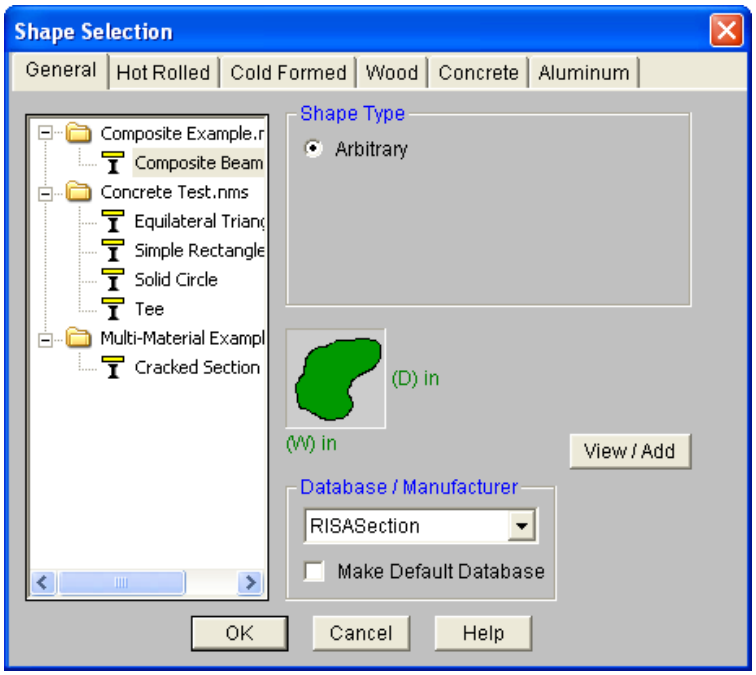
To Add an Arbitrary Shape to the Database

- To enter an arbitrary shape in the database, click the **Edit Shape Database** button from the **RISA Toolbar**, and then clicking the **General** tab from within the Shape Database Editor dialog. Set the shape type to Arbitrary, click **Add** and then enter the shape name and properties.

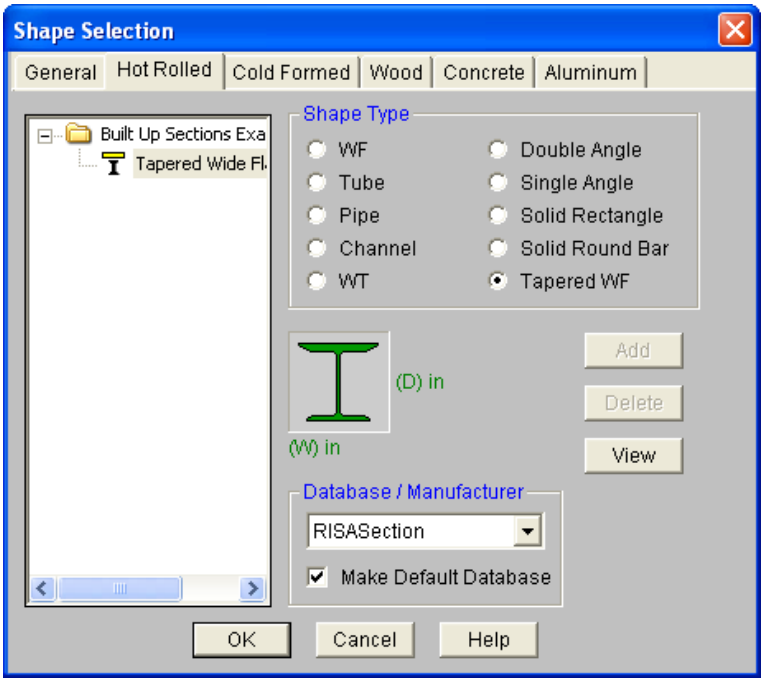
RISASection Files

You may create simple or complex sections in RISASection and then import those sections for use in your model. Sections that exist in RISASection (files located in the "RISASection" file specified in **Tools - 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/Manufacturer**.



The shapes that are designated as Hot Rolled Steel Material in RISASection will show up in the **Hot Rolled** tab under the appropriate Shape Type (Channel, Wide Flange, etc.) when "**RISASection**" is selected as the **Database/Manufacturer**.



Note: Currently, RISASection can only import General and Hot Rolled Steel shapes. More material choices will be available in a future version.

Local axis nomenclature differs between RISASection and RISA-3D. The local y-axis is the same in both programs, but the x-axis in RISASection is the local z-axis in RISA-3D. Section properties are labeled accordingly, for example Ixx in RISASection will be Izz in RISA-3D. You can display local axes in the 3D view to verify the orientation, and modify using the Rotate property as needed.

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 / RISAFloor Interaction

There are a few common issues that arise when attempting to access RISASection files from within RISAFloor.

- When RISAFloor launches, it will only import RISASection files that exist in the directory specified in **Tools - Application Settings - File Locations**.
- RISAFloor only reads in database files when it is first launched. Therefore, if a RISASection file is created or modified, then RISAFloor 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 RISAFloor.

- The name of the RISASection "File" is actually different from the name of the section itself. Both names will appear in the RISAFloor 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.

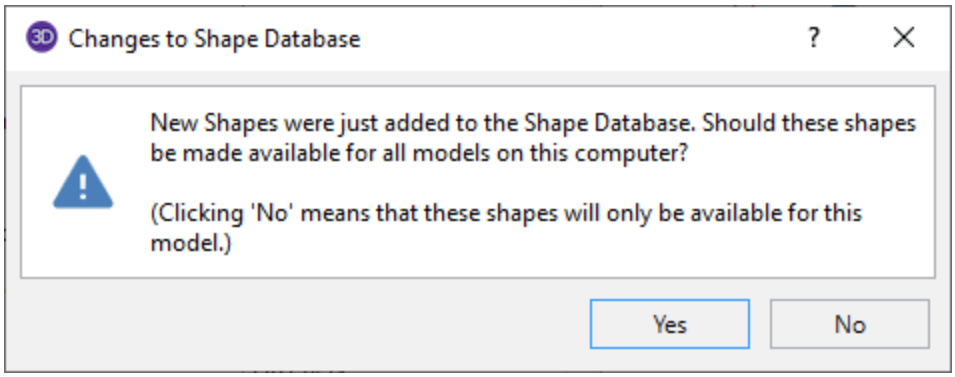
Solid Circular Shapes

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.

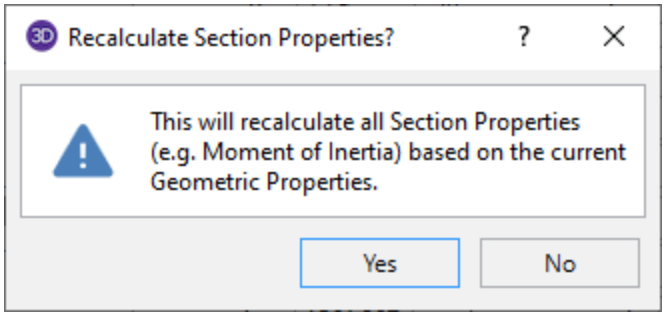
Database Files

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 **Tools** menu.

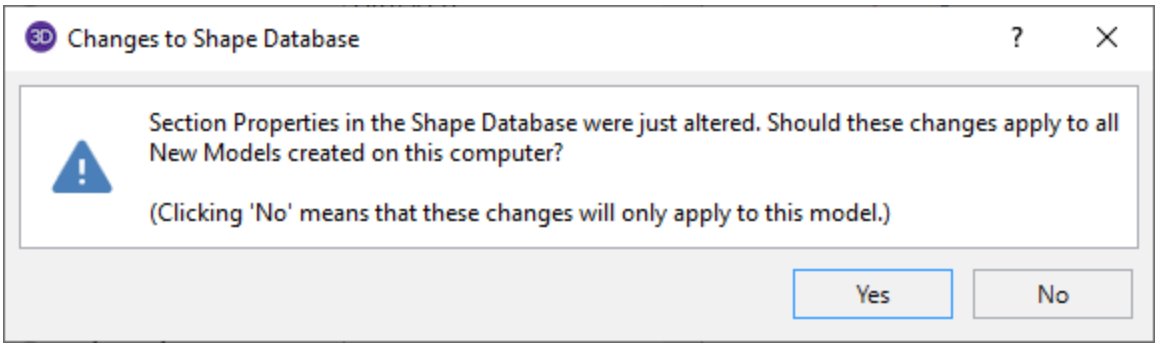
After adding a new shape in your model, you will be prompted with this Changes to Shape Database dialog. Click 'Yes' if you want to add the shape to your Shape Database to be available for all future models. Click 'No' if you do not want the new shape to be available for other models, this will not save 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.
- To delete a shape specify the database and shape type you wish to delete and then click the **Delete** button.
- To edit a shape click the **Edit** button and edit the shape properties. Values can only be manually edited here, nothing will be recalculated. If you wish to have all the values for a shape recalculated, you will need to delete the shape and then add it again with the new properties. For Tapered WF shapes only, you can click **Calc Props** to recalculate the shape properties based on the basic dimensions. Tapered WF shapes are stored parametrically, so there are no database values to edit since they are calculated on the fly whenever an analysis is performed.

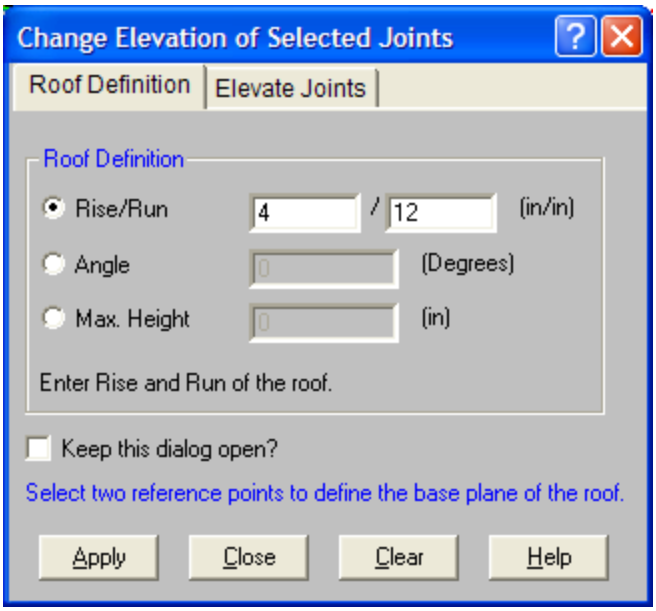
Sloped Members

You may elevate or slope the top floor of your model directly in the model view. Beams can be sloped based on the Point locations of supports. You can draw beams or generate infill beams on an existing sloped area. Using the change elevated points button to create a sloped beam or section, you can define the sloped beams by lifting their supports.

The slope beams tool will lift the area based on the Selected points or beams. See Graphic Selection to learn how to make selections.

Notes:

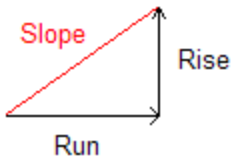
- Only the top floor elevation can be sloped.
- The slope can only be positive.
- Any Points on a cantilever (overhang) with a positive slope will automatically be placed at negative elevations below the floor. These elevations cannot be manually changed in the Point Location spreadsheet.
- Once a slope is defined, additional beams may only be drawn in the plan view.



To Slope the Beams By Roof Definition Method

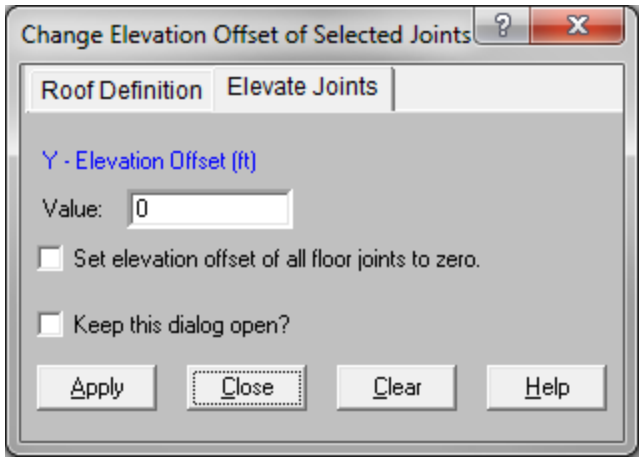
1. If there is not a model view already open then click on the RISA Toolbar to open a new view and click on the Graphic Editing Toolbar Icon to turn on the Drawing Toolbar if it is not already displayed.
2. Using the selection tools, select the supports/Points that will have the elevation change.
3. Select the icon Change the elevation of selected joints in the model button.
4. Select the Roof Definition tab, click in one of the radio boxes to select the method you would like to slope the members :

- Rise/Run- Enter two values that describe the slope.



- Angle- Enter an angle between 0 and 90 degrees from the horizontal plane.
- Maximum Height- Enter a height above the floor level that will become the maximum elevation of the slope.

5. Press Apply and the slope members icon will appear, prompting you to select the two reference points from which the base plane of the slope will be calculated from. You can select any two Points in the model to create the base plane and everything else in the model will be sloped from this line.
6. This option creates a mono-sloped roof out of the existing selected columns and walls. Therefore, if a roof configuration other than a mono-sloped is required, then it is necessary to use the feature multiple times with only portions of the model selected.



To Slope the Members By Elevate Joints Method

1. If there is not a model view already open then click on the RISA Toolbar to open a new view and click on the Graphic Editing Toolbar Icon to turn on the Drawing Toolbar if it is not already displayed.
2. Using the selection tools, select the supports/Points that will have the elevation change.
3. Select the icon Change the elevation of selected joints in the model button.
4. Select the Elevate Joints tab, and enter a value in the dialog box. This value is the absolute offset from the floor elevation.
5. Press Apply. All of the selected joints in the model will move up to the input elevation.

The "Set elevation offset..." option allows you to flatten an entire floor in one click. This bypasses the sloping validation routines and is primarily useful for models that have been imported from other programs (such as Revit).

Note:

- To view the sloped members you can view the model in isometric view by pressing on the Iso button. Wireframe views will not show columns or walls at their elevated locations. To see this, click on the button Toggle between Wireframe and Rendering of Beams to enter a rendered view.

Loading for Sloped Floors

Per section 1607.11 of the 2006 IBC, live loads that act on a sloping surface are assumed to act vertically on the horizontal projection of that surface. This applies by default to LL, LLS, SL, SLN, RL, and RLL.

The dead load cases will always apply the load on the full inclined surface rather than using the reduction for projected loads.

Loads that are defined using the OL categories (OL1 through OL4) may be applied based on either the projected or inclined surface area, depending on how they are defined.

Load that are defined as Tapered Area loads may also be applied based on either the projected or inclined surface areas.

Tips for Modeling on Sloped Floors

The graphical user interface (GUI) for RISAFloor was built assuming that most input would be done while viewing the floor in a typical plan view. This is true even for roof floors that will be sloped. These floors can be viewed in an Isometric view in order to visually verify the geometry. However, there are limitations to the ability of the program to display sloped floors in an isometric view.

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

Display of Loads for Sloped Floors

The loading on sloped floors is always shown at the base elevation of the floor. For this reason the displayed location of Point Loads, Line Loads and Tapered are loads can be misleading when viewed from an isometric model view.

Display of columns

When shown in the Wireframe option for a single floor view, the columns will display at the base elevation of the floor and will not extend all the way up to their top elevation. Rendered views will show the column extended all the way up to the top of the defined level for that column. Any parapet extensions for the columns can only be viewed from the Full Model view.

Splice Definitions

Column splices are typically defined as the distance from the upper floor DOWN to the splice location below that floor. This can be problematic for sloped floors because each column can have a different elevation. For this reason, the splices at sloped floor must all be defined from the same reference point. This reference point is taken the Elevation for that floor that is listed in the Floors spreadsheet.

Refer to the section on the Columns Spreadsheet for more information about splices.

Adding framing to an existing sloped roof

Any Points that are added to an existing floor will be created at the base elevation of that floor. During a solution the program searches for elevation discrepancies and moves those Points up to a location that matches the existing sloped beams. This is dictated based on the Global Parameter setting Move Invalid Joints on Sloped Members.

Limitations

Roof Levels Only

Currently you may only define sloped floors at the top floor of your building (i.e. the roof).

All Framing Circuits must be co-planar or load attribution will be simplified.

For the load attribution routine to be valid, every closed framing circuit must be defined by a single planar surface. If a non-planar surface is detected then the load attribution will be simplified. For these circuits the program will assume a constant elevation for load attribution. Thus, the dead load and any other loads that are not defined as projected will be applied unconservatively. A warning log message will come up at solution time for these non-coplanar circuits.

Saw Tooth Roofs

Any Point on the roof level can only exist at a single elevation. Therefore, saw-tooth roof systems cannot be modeled since they require the roof to exist at two elevations at the same column line.

Axial Force in the Design of Concrete Beams

The Member detail reports for sloped members will report the axial force diagrams for steel or wood members. These forces will be used in the member code check process. However, concrete beam members will neither show the axial forces nor use them in the code check.

Weak Axis Moment and Shear Diagrams

Any weak axis shear force or bending moment that develops in members which resist the lateral thrust of a sloped member are used in the member design and code check calculations (for steel and wood). However, these forces and moments are not reported on the Member Detail Report force diagrams.

Concrete beam members are not designed for weak axis shear forces or bending moments.

Elevated Concrete Slabs

Sloped floors are not allowed for Elevated Concrete Slab floor levels.

Solution

To solve the model select **Solve** on the **Main Menu** or click the button on the **RISA Toolbar**. A progress bar displays information as the solution proceeds. In general, loads are attributed to the members, members are optimized, vibration analyses are performed and the loads are transferred to the next level. Any warnings are listed in the **Warning Log** which can be opened from the **Spreadsheets Menu**. See Design Optimization for more information on how members are optimized.

Solution Method

The solution is based on load categories and load combinations using 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 point deflections. These point 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).

RISAFloor performs and analysis and design for all members defined within the program. This includes Gravity only members as well as Lateral members. The main difference between the two member functions is that the final design of Lateral members is assumed to occur in RISA-3D. Refer to the section on RISA-3D Integration for more details.

When a solution has been performed and the results are available the “S” icon on the status bar in the lower left corner will change from (gray) to (blue). You may then modify the design, while maintaining the results, using the Member Redesign Dialog. See Member Redesign Dialog for more information.

After a solution has been performed and modifications are made to the model, the "S" icon changes to (yellow). This indicates that the model has been solved but then modified. These changes may affect the results and thus another solution may be required. The reason for this is that the stiffness and/or weight of the modified members may have changed. While these differences are often negligible it is a good idea to resolve the model when you are finished modifying the designs so that the results reflect the actual stiffness and weight.

Other changes made outside the Member Redesign Dialog, such as changes to the geometry or within the spreadsheets, cause the results to be cleared and another solution is then necessary.

When you save a file that has been solved, you may also save your results. The next time that file is opened the saved results will be opened as well. You may use the Application Settings on the Tools Menu to change the way that you are prompted to save results. See Application Settings for more information.

Differences Between RISA-3D and RISAFloor

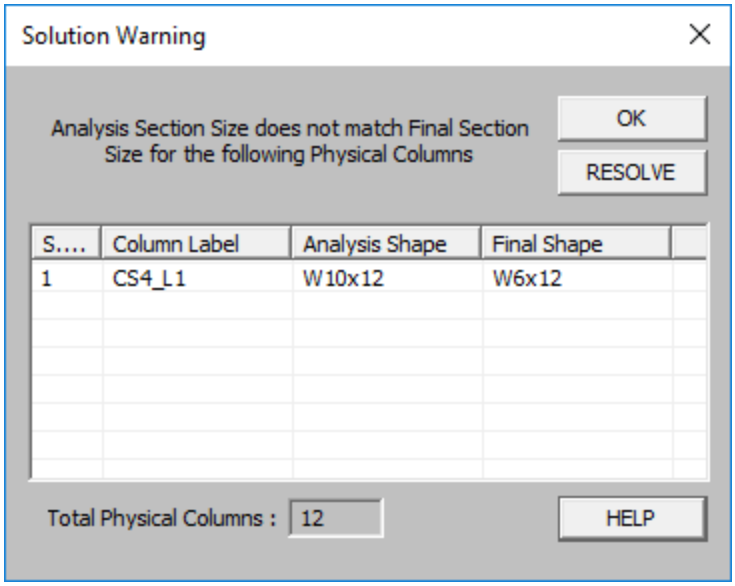
RISAFloor performs an independent solution at each floor. The assumptions in the Floor solution are essentially the following:

- 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.

Difference Between Analysis and Design Sizes

When you include the column rotational stiffness in the RISAFloor analysis (see the setting on Model Settings), changes to the column size will affect the stiffness of the floor and the distribution of forces on that floor. Because the program performs solution for upper floors first, it is common for the original "required" size of a column at an upper level to be insufficient when the forces from the lower level are added in. The column will then be resized and the "Final Shape"doesn't match the analysis size. For this reason, the program may present the user with the following dialog.



The purpose of this dialog is to ask the user if he/she wants to re-solve the model using the final shapes on those upper floor levels to see if these are going to significantly affect the beam design (or slab forces for RISAFloor ES models only).

One may choose to ignore those differences because the columns are expected to change again when the lateral analysis is performed in RISA-3D. Therefore, the column sizes are only approximations and may be perfectly acceptable without a re-analysis.

Static Solutions / Stiffness Matrix

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 "S" icon on the status bar in the lower left corner will change from to . If any changes are made to the model that invalidate the results then the results will be cleared and another solution will be necessary.

The solution is based on the widely accepted linear elastic stiffness method for solution of the model. The stiffness of each element of the structure is calculated independently. These element stiffnesses are then combined to produce the model's overall (global) stiffness matrix. This global matrix is then solved versus the applied loads to calculate joint deflections. These joint deflections are then used to calculate the individual element stresses. The primary reference for the procedures used is **Finite Element Procedures**, by K. J. Bathe (Prentice-Hall, 1996).

Skyline Solver

This solution method is also sometimes referred to as an Active Column solution method. In finite element analysis, the nonzero terms of the stiffness matrix are always clustered around the main diagonal of the stiffness matrix. Therefore, the Skyline or Active Column solutions take advantage of this by condensing the stiffness matrix to exclude any zero stiffness terms that exist beyond the last non-zero term in that column of the matrix.

Since the majority of terms in a stiffness matrix are zero stiffness terms, this method greatly reduces the storage requirements needed to store the full stiffness matrix. However, for large models (+10,000 joints) , the memory requirements even for a skyline solution can be problematic.

This solution method has been used successfully in RISA for more than 20 years, and has proven its accuracy continuously during that time.

Sparse Solver

The skyline solver described above is moderately efficient because it only stores and performs operations on the terms within the "skyline" of the stiffness matrix. However, that solution still contains a great number of zero stiffness terms within the skyline of the matrix. A Sparse Solver will reduce the matrix size to an absolute minimum by eliminating the storage of ALL zero stiffness terms.

A sparse solver is the most efficient solution methodology possible because it stores and performs operation only on the non-zero terms of the stiffness matrix. For this reason the sparse solution is preferred from both a solution speed and memory requirement standpoint.

Note

- The skyline solver is retained mostly for comparison and verification purposes.

◦ Spreadsheet Operations

Powerful spreadsheets may be used to view, sort and edit the input data and results. The spreadsheets and model views are synchronized, unless you turn this feature off in the **Application Settings** on the **Tools Menu**. As you edit a model graphically the spreadsheets are automatically updated and as you make changes in the spreadsheets the model views reflect these changes immediately.

The input data may be accessed from the**Spreadsheets Menu**. You may edit the data or you can add new data. You can also paste data from another application directly into the spreadsheet via the Windows clipboard. Any changes made to the input spreadsheets may also be viewed graphically.

After solving the model, results are recorded in spreadsheets for browsing. These spreadsheets may be accessed from the **Results Menu**. You may sort the results in order to find maximums and exclude data that is not important. You may also copy this data to the Windows clipboard and use it in another application.

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

Moving and Scrolling

To move between cells in a spreadsheet, click any cell or use the keyboard arrow keys. When you move to a cell it becomes the active cell. To see a different area of the spreadsheet use the vertical scroll bars on the right side of the spreadsheet.

To Scroll	Do This
One row up or down	Click the arrows on the vertical scroll bar.
One column left or right	Click the arrows on the horizontal scroll bar.
One page up or down	Click above or below the scroll box in the vertical scroll bar.
One page left or right	Click to the left or right of the scroll box in the horizontal scroll bar.
A large distance	Drag the scroll box to the approximate relative position.

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

Key	Function
TAB	Move right one cell
ENTER	Move to the first column of the next line. Adds new line if necessary.
PAGE UP	Move the active cell one full page up.
PAGE DOWN	Move the active cell one full page down.
HOME	Move to the first line of the spreadsheet.
END	Move to the last line of the spreadsheet.
F3	Insert new line below current line.
F4	Delete current line.
F8	Insert new line below current line and repeat the current values in the new line.

Selecting Spreadsheet Cells

Before you can carry out commands or tasks in a spreadsheet, you must select the cells that you want to work with.

To Select	Do This
A single cell	Click the cell, or press the arrow keys to move to the cell.
A range of cells	Click the first cell of the range, and then drag to the last cell.
An entire row	Click the row heading.
An entire column	Click the column heading.
Adjacent rows or columns	Drag across the row or column headings

Note:

- To cancel a selection of cells, click any single cell in the spreadsheet.

Undoing Operations

RISAFloor provides you with virtually unlimited **Undo** capability so that you may easily correct mistakes or just back up to try different possibilities. Simply click the Undo button as many times as you wish to go back a step. The model view and the spreadsheets will visually display the "undoing". Remember that graphic edits are undone as well.

Redoing Operations

RISAFloor 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. The model view and the spreadsheets will visually display the "redoing". Remember that graphic edits are redone as well.

Editing Spreadsheets

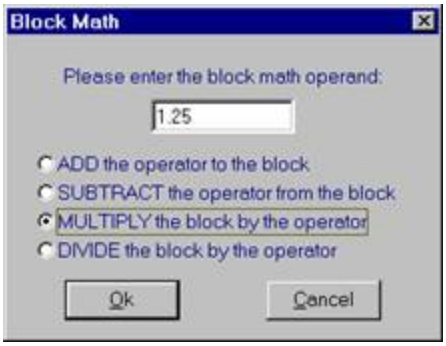
The spreadsheets have been specifically developed for the input and editing of structural models. There are many ways to edit the spreadsheets allowing you to quickly build your model. You may copy and move data from other locations or other files. You may also fill large blocks of cells automatically and perform math on these cells.

To Fill Selected Cells

1. Select the cells you wish to fill.
2. Click the **Block Fill** button.
3. Type the value you wish to apply.
4. Click **OK**.

To Perform Math on Selected Cells

1. Select the desired cells.
2. Click the **Block Math** button.
3. Specify the value and the operation you wish to apply.
4. Click **OK**.



To Cancel or Undo Editing

- To cancel an entry before you press ENTER, press **ESC**.
- To undo a completed entry, click the **Undo** button.

To Edit Cell Contents

1. Click the cell that contains the data you want to edit.
2. Make any changes to the cell contents.
3. To enter your changes, press **ENTER**.
4. To cancel your changes, press **ESC**.

Note

- To replace the contents of a cell, click on the cell and type the new entry.
- To edit the contents of a cell, double-click on the cell and use the arrow keys to locate the cursor and perform the edit.

Inserting, Deleting and Clearing Cells

To insert cells you must insert an entire row. When you delete cells you must delete an entire row. The row is removed from the worksheet and shifts the lower cells to fill the space. When you clear cells, you remove the cell contents but leave the blank cells on the spreadsheet.

To Insert Rows

1. Click a cell in the row immediately above where you want the new row.
2. To insert a blank line, click the **Insert New Line** button.
3. To have the values of the current line copied in the new line, click the **Repeat Current Line** button.

To Clear Cell Contents

1. Select the cells, rows, or columns you want to clear and press the **DELETE** key.

To Delete Rows

1. Select the rows you want to delete.
2. Click the **Delete Marked Lines** button and 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 may not delete them.

Moving and Copying Cell Contents

Standard Windows cut, copy and paste functions are fully supported. By using the clipboard you may move or copy any spreadsheet data to another location in the same spreadsheet, or to another spreadsheet (provided it fits). You may also copy data to or from another model or any other application that supports copy and paste such as a word processor or spreadsheet.

To Move or Copy Cells

1. Select the cells you want to move or copy.
2. To copy the cells select **Copy** on the **Edit Menu**.

   To move cells select **Cut** on the **Edit Menu**.

3. Click on the cell you wish to place the data in and click the **Paste** button.

To Move or Copy Cells between Existing Cells

1. Select the cells you want to move or copy.
2. To copy the cells select**Copy** on the **Edit Menu**.

To move cells select **Cut** on the **Edit Menu**.

3. Select a cell on the row above where you want to place the data.
4. Click the **Insert New Line** button for each new line that is needed.
5. Click on the upper left cell you wish to place the data in and click the **Paste** button.

To Move or Copy Cells to Another File

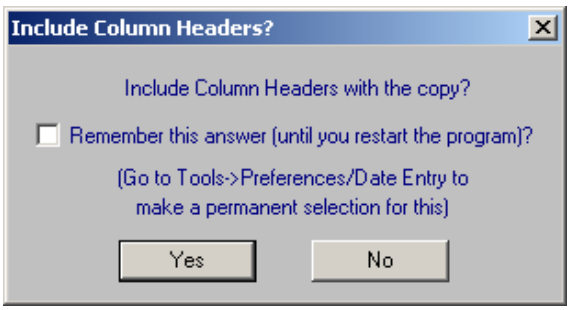
1. Select the cells you want to move or copy.
2. To copy the cells select**Copy** on the **Edit Menu**.

   To move cells select **Cut** on the **Edit Menu**.

3. Open the file you wish to copy the data to.
4. Click on the cell you wish to place the data in and click the **Paste** button.

Note:

- Your data stays in the clipboard until you cut or copy new data. You may repeat step 4 to move or copy data to multiple locations.
- When you copy information a dialog will come up asking if you want to copy the spreadsheet headers as well. This is helpful if you are copying information to a spreadsheet-type program. See the Customizing RISA topic for more information on this.



Finding in Spreadsheets

To locate or find a specific point or member while in a spreadsheet, click the **Find** button on the **Window Toolbar**.

Sorting is not yet available in RISAFloor spreadsheets.

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**,**Deck Definitions**, **Area Load Definitions**, **Footings**, and **Load Combinations**.

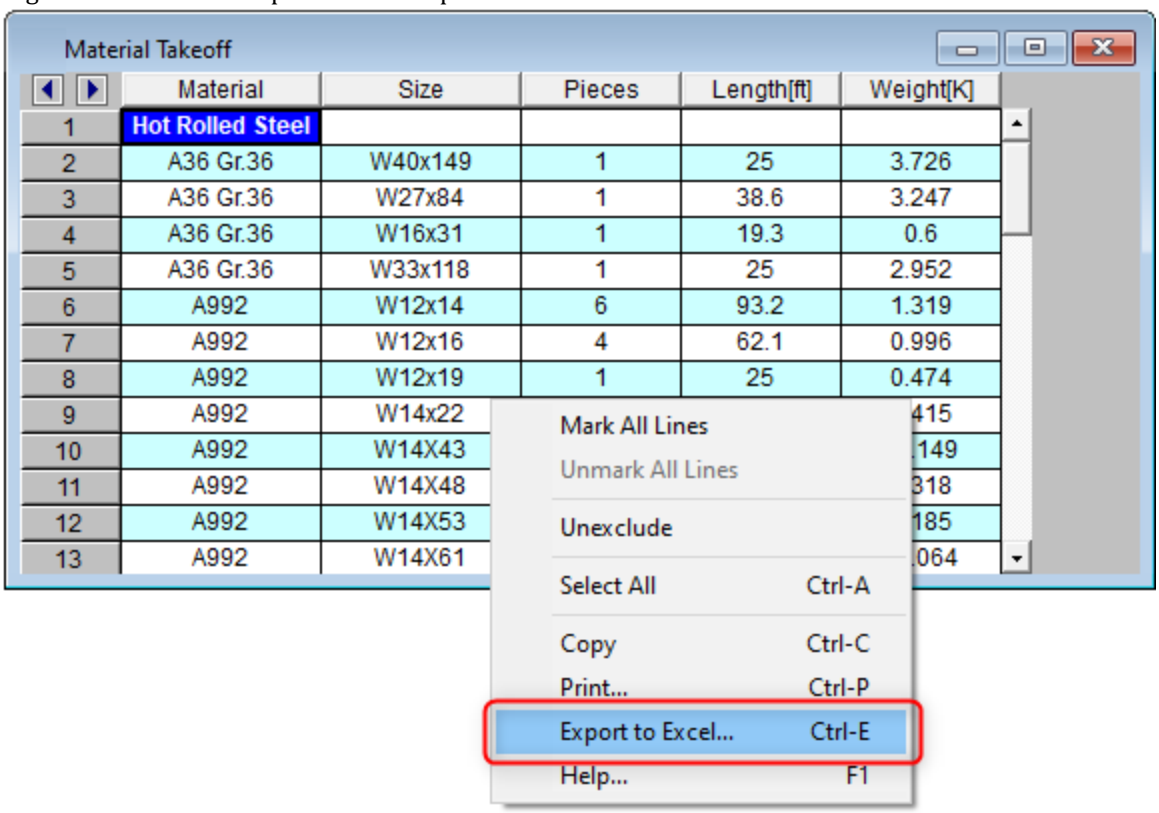
To save default data simply click the **Save as Defaults** button when you are ready and the current data in the spreadsheet will be used in each new file that you create. The data is saved for the current, active, spreadsheet only and affects no other open or closed spreadsheets.

Export to Excel Spreadsheet

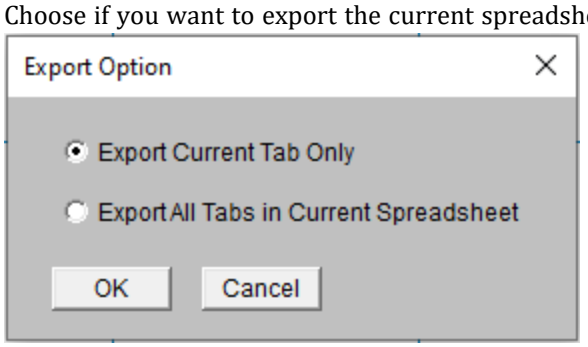
For all Results spreadsheet, the program is capable of exporting them into Excel Spreadsheet.

The process is as follows:

1. Right Click and click Export to Excel option.



2. Choose if you want to export the current spreadsheet tab or all tabs.



3. 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 Export to Excel.

Special Spreadsheet Functions

There are special functions that help you with a particular spreadsheet.

Load Combinations Spreadsheet

You may 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.

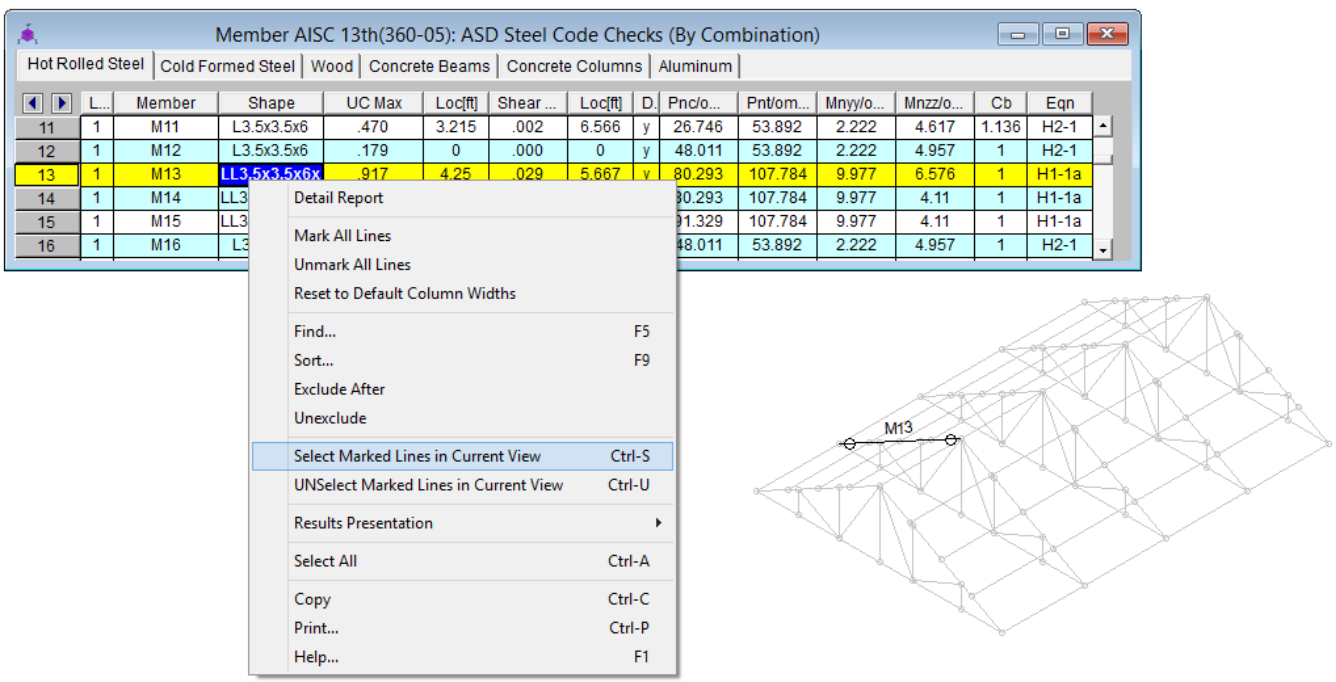
Selecting Items in the Model View

In certain results spreadsheets there is the capability to have elements selected graphically.

The process is as follows:

1. Unselect the entire model.
2. Open the desired results spreadsheet.
3. Highlight in Yellow the lines you wish to view graphically, by clicking on the number of those lines on the far left.
4. Click the **Select Marked Lines in Current View** icon .
5. Only those elements highlighted will be selected graphically.

Note: The inverse process can be used to Unselect Marked Lines using the .



Steel Products (Joists and Joist Girders)

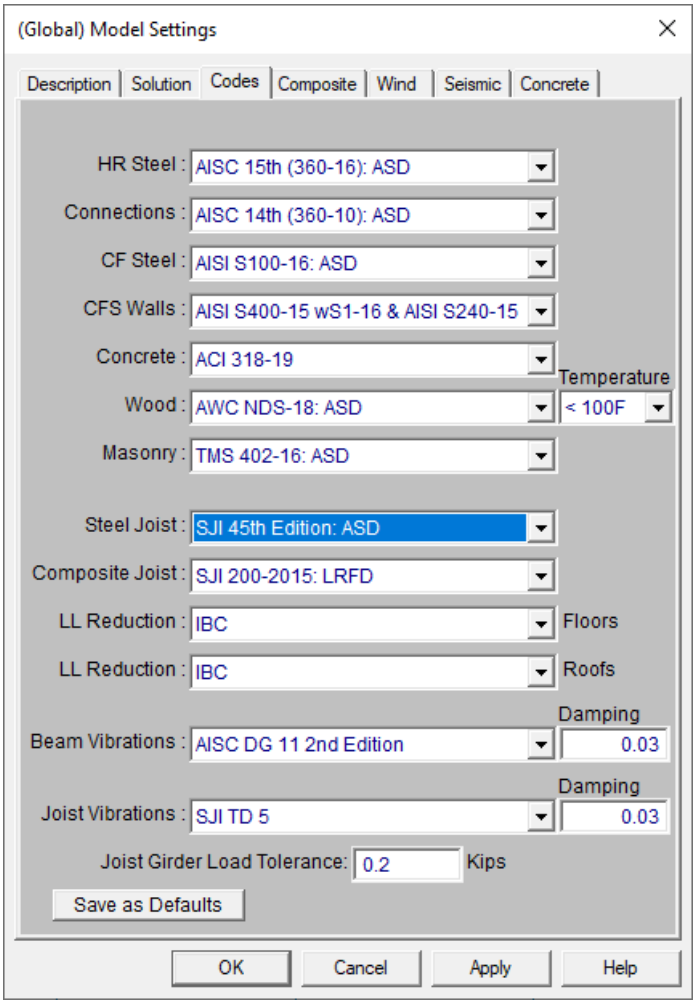
Steel Product design can be performed on standard steel joists. The steel products are automatically optimized based on minimum weight and based on the criteria specified in the **Design Rules** spreadsheet. There are options for Steel Joist Databases which have different capacity values. The Steel Joist Institute (SJI) Edition from which the values are determined are selected based on the Code specified in the Model Settings.

The following editions are supported:

- SJI 45th Edition: ASD
- SJI 43rd/44th Edition: ASD
- SJI 42nd Edition: ASD

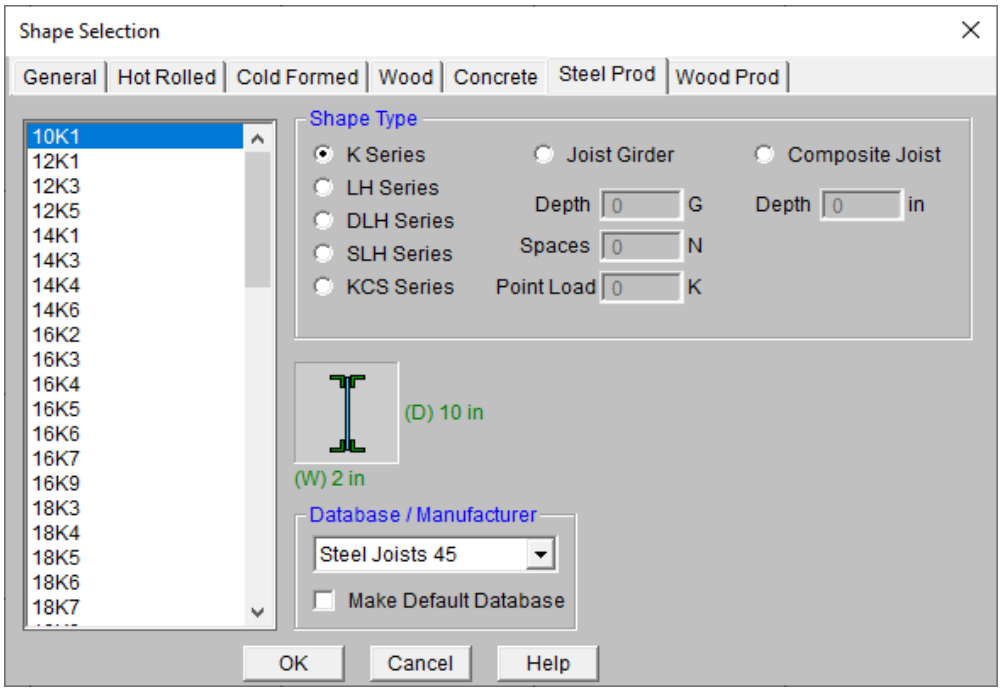
How To Apply a Steel Product Design Code

1. On the **Model Settings** - **Codes** tab, select the steel joist code from the drop down list.
2. For composite joist, select the code from the composite joist drop down list.



Steel Products Database

The various Steel Product databases may be accessed from the **Beams** spreadsheet by clicking in the **Shape** field and then clicking .



Steel Products Design

Allowable loads for Steel Joists are based on the standard load and span tables for K, LH, DLH, and SLH Steel Joists. This also includes the short span K2.5 series Joists. These tables list a maximum allowable uniform load and a uniform load that will produces a deflection of L/360. The maximum allowable load (black number) is used directly by the program and cannot be adjusted. The L/360 load (red number), on the other hand, is used to obtain the joist stiffness for the given span. This load can be exceeded if a more relaxed deflection criteria (such as L/240) is specified in the Design Rules.

Steel products are designed to resist a uniform dead load (UDL), a uniform live load (ULL) and a uniform total load (UTL). The uniform dead load (UDL) on the beam includes the sum of the beam self weight, the deck self weight, and any applied / superimposed dead load.

The uniform live load (ULL) on the beam includes the sum of ALL the floor live load cases including the reducible and non-reducible versions of LL (Live Load), and LLS (Live Load - Special)as well as the controlling roof load (RLL, SL +SLN, or RL). Each of the roof loads are treated separately and are NOT added together for the calculation of the ULL or UTL. The uniform total load is the sum of UDL and ULL.

Per the SJI 100 - 2015, the design length shall equal the span minus 4 inches.

Note:

- The load combinations defined within the Load Combinations spreadsheet are NOT used for the design of steel products. Rather, steel products are designed to the UDL, ULL and UTL loads described above regardless of what load combinations have been built by the user.
- The steel-product self weight is considered in the total dead load (UDL) and included in the Detail Report diagrams available after analysis.
- Other Loads (OL1, OL2, et cetera) are not considered in joist code checks.
- If a concentrated point load (even one with a small magnitude) is detected on a standard joist, the joist will be flagged as a Special (SP) Joist and will be designed as described in the next section.
- The distributed load on the joist cannot vary by more than 5% along the length of the joist. If it does, then the joist will be flagged as a Special (SP) Joist and will be designed as described in the section below. The only exception to this is some additional load variation is allowed within 10% of the end of the joist.

- Sloping steel joist design considers the FULL length of the joist and the load applied perpendicular to the member. Any axial component of loading is given but is not designed for.
- Live load reduction is considered for steel joist design and is included in the **Member Detail Report** under the Special Joist Load Diagram and Design sections.

Design for Joists with Non-Uniform Loads

When considering non-uniform or concentrated loads, the joists must be designed for these special loading conditions. This may be done by specifying a KCS joist. If a KCS joist is not assigned by the user then a standard joist will be chosen from the design list, but will have an (SP) extension to indicate that the joist must be designed to resist the special applied loads.

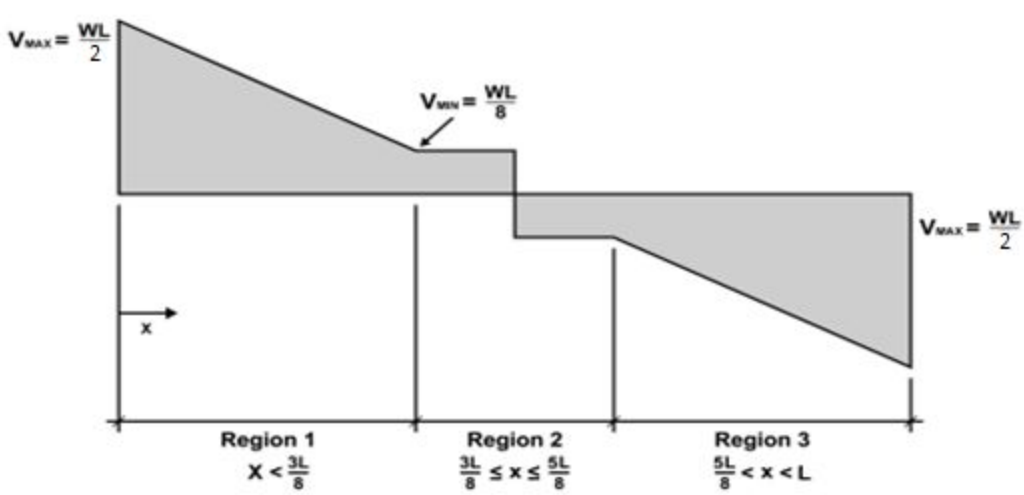
KCS Joists

KCS joists are joists that are designed to have a constant shear and moment capacity along the entire length of the joist. In RISAFloor, KCS joists are checked versus the constant moment capacity given in the KCS joist tables. They also check that the largest applied point loads is less than the shear capacity and that the algebraic sum of the distributed loads at a locations are less than the 550 plf maximum for K series joists. In these last 2 cases, the code check for a KCS joist may be the largest applied point load / shear capacity or the largest distributed load magnitude / 550. No special message is generated if either of these 2 checks are violated as the code check will come back > 1.0. KCS joists are also checked against the deflection requirements. Currently, KCS joists never get the “SP” suffix added.

SP Joists

When a joist has special loading that requires extra consideration, the most economical solution may be to use a standard joist that is capable of resisting the non-standard or *special* loading. When this type of design is given, the program will add an (SP) extension to that joist designation to indicate that it is resisting a non-standard load. Because this loading falls outside the normal design criteria for the joist tables, these (SP) joists may warrant further detailing by the engineer or joist manufacturer to insure adequate performance. The RISAFloor detail report provides a *Special Joist Load Diagram* which shows the Point Loads, Distributed Dead, and Distributed Live loads with their magnitudes and dimensions.

RISAFloor will choose a joist such that the capacity shear envelope of the joist under a standard uniform load is still greater than the actual shear at all locations along the length of the joist. Because the joist will have a shear strength greater than zero at the midspan of the beam, the shear envelope for a joist with a maximum uniform load capacity of W, would be given by the figure shown below:



In addition to checking the shear envelope, the program will make sure that the maximum allowable moment is not exceeded at one point along the length of the joist. The maximum moment is defined as $wL^2/8$, where w is the maximum allowable uniform load for that joist and span that is listed in the joist tables.

More information on this procedure can be found in the book, **Designing with Vulcraft: Steel Joists, Joist Girders and Steel Deck**, published by Nucor Corporation.

Joist Girders

The program will perform Joist Girder design according to the basic Joist Girder tables given in the Steel Joist Institute tables and / or the Vulcraft catalog. The intended use is that the user will designate the depth of the Joist Girder before the model is solved. If a generic "Joist Girder " is chosen, then the program will optimize the beam by finding the Girder with the least DEPTH. This will often be a heavier member than is necessary for design. For this reason it is strongly suggested that the user provide a Girder depth whenever they specify a joist Girder.

RISAFloor will call out a Joist Girder in the design results using the following terminology:

16G4N2.4

Where:

- 16G refers to the DEPTH of the Joist Girder (16 inches for this example)
- 4N corresponds to the NUMBER of equally spaced concentrated loads (4 for this example)
- 2.4 refers to the MAGNITUDE of the concentrated loads (2.4 kips for this example)

The Joist Girder will be flagged with an (SP) if non-standard loading is detected. This includes situations where the magnitude or spacing of the concentrated loads vary by more than the **Joist Girder Load Tolerance** set on the Codes Tab of the Model Settings Dialog. Whenever the loading falls outside the normal design criteria for the joist tables, these (SP) joists may warrant further detailing by the engineer or joist manufacturer to insure adequate performance. RISAFloor provides a loading diagram in the detail report that can be directly supplied to the manufacturer for the design of the Joist Girder.

Note:

- If all point load magnitudes are within the **Joist Girder Load Tolerance**, then this point load will be used in the call-out and SP will not be used.
- If more than 50% of point loads are within tolerance of each other, use this point load value as the call-out and flag the joist as SP.
- If less than 50% of point loads are within tolerance of each other, use the maximum point load value as the call-out and flag the joist as SP.

Estimating the Moment of Inertia

When only the depth of the girder has been specified by the user then the moment of inertia is approximated as:

Izz = .027*2.*4*(bm_len/12.)*JG_depth

When the user gives the number and magnitude of the point loads and the magnitude then the moment of inertia is approximated as:

Izz = .027*joist_spaces*point_load*(bm_len/12.)*JG_depth

Note:

- If Point Loads are very closely spaced (within 3 inches of each other) they will be interpreted as occurring at the same location.

Composite Joist

The program will perform the Composite Joist (CJ) design according to the design approaches provided by SJI. The detailed design steps and assumptions are discussed in this section below.

The intended use is that the user will designate the depth of the Composite Joist before the model is solved. If a generic "Composite Joist" is chosen, then the program will optimize the beam by using the rule of thumb of L/18 provided by SJI for the design. L is taken as the span of the joist. For user input Composite Joist Depth, it needs to fall in the range of L/12 and L/30 as required by SJI. Currently it is prohibited to input joist depth that is out of this range in the program.

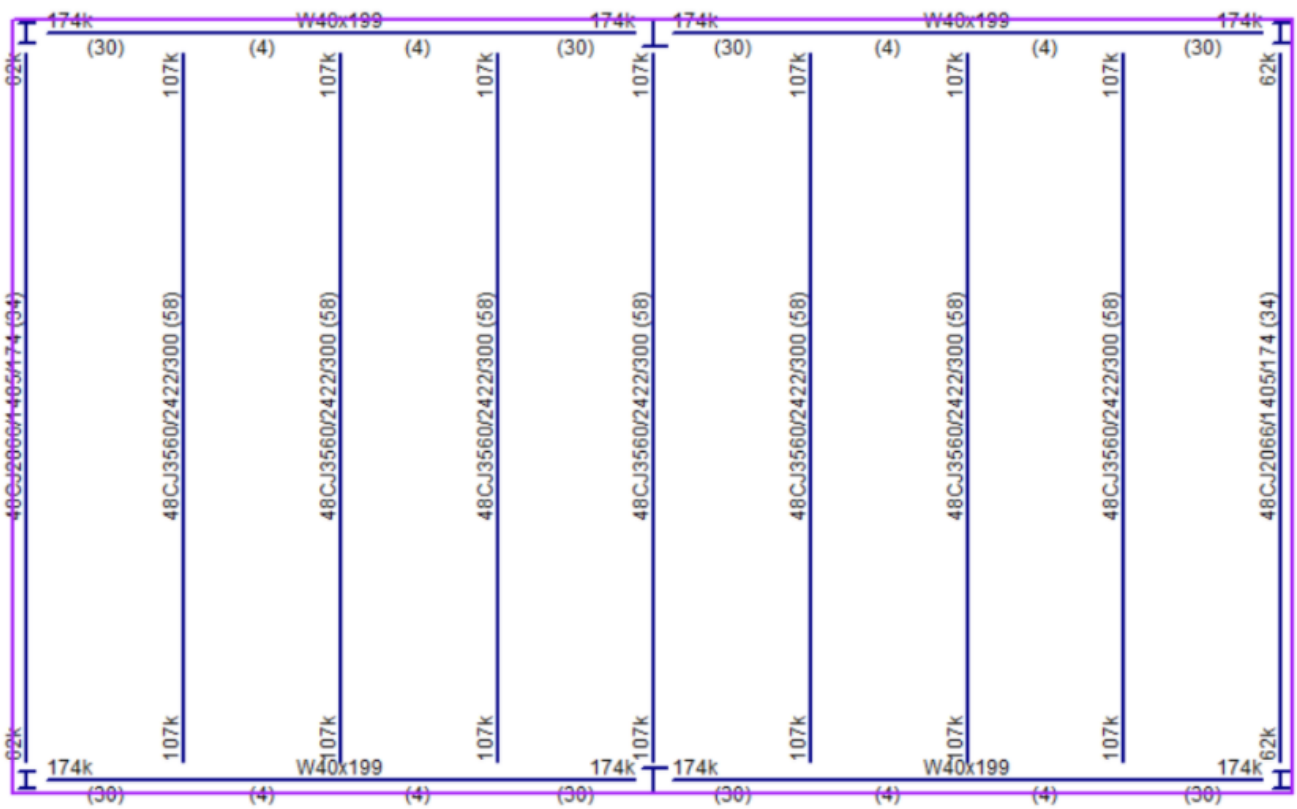
RISAFloor will call out a Joist Girder in the design results using the following terminology:

48CJ3540/2423/300 (58)

Where:

- 48CJ refers to the DEPTH of the Composite Joist Girder (48 inches for this example)
- 3540 refers to the equivalent uniform total factored load (3540 plf for this example)
- 2423 refers to the equivalent uniform total factored LL load (2423 plf for this example)
- 300 refers to the equivalent uniform total factored composite DL load (300 plf for this example)

If there is non-standard loading on the joist, the program will calculate the maximum moment on the joist and convert that to an equivalent uniform load for the joist callout.



Design Methods (per procedures provided by SJI):

1. The program calculates the maximum factored moment demand on the joist based on the load input.
    a. As a starting point of the design iteration, joist self-weight is estimated as 45 plf. This will be updated through the design iteration once the joist size is finalized.
    b. Bridging self-weight is estimated as 0.25*Wtrib in plf, where Wtrib is the joist tributary width.
2. The design span is taken as Ldes = L - 0.33 ft, where L is the centerline to centerline span of the member.
3. Bottom chord sizes are first estimated based on the applied loading and an estimated joist geometry. The joist geometry will be updated once a chord size is selected. The design process will iterate until the final size is reached. The design iteration starts by using an estimated concrete stress block depth a = 1.0 in and an estimated distance from bottom of joists to centroid of bottom chords ybcr = 1.0 in.
    a. The estimated joist effective depth for moment arm is calculated as de = dj + hdeck + tc - a/2 - ybcr, where dj is the depth of the joist, hdeck is height of the metal deck (rib height), tc is the height of concrete over the metal deck.
    b. The factored moment demand Mu is calculated based on the applied loads and the design span Ldes.
    c. The estimated required axial force in bottom chords is calculated as Fbc = Mu/de.
    d. The estimated required bottom chord area is calculated as Abcr = Fbc/(phi*Fy).
    e. The bottom chords (double-angles) are picked based on estimated bottom chord area. Abc and ybc are updated based on the selected chord size.
4. The area of top chords is estimated as Atc = 0.60 Abc.
5. The final joist self-weight is estimated based on the total weight of top and bottom chords with a 30% increase: SWj = (SWtc + SWbc) * 1.3. The factor 1.3 is to include the contribution of web members.
6. The design is reiterated with the updated top and bottom chord sizes and the self-weight of the joist. The final moment capacity of the joist is calculated as: Mn = min(phi_t*Abc*Fy*de, phi_cc*0.85*f'c*be*te*de).
7. Bridging design: only the number of bridging lines is provided in RISAFloor for budgetary purposes. The final quantity and location of bridging will be designed by SJI certified manufacturers.
    a. The maximum spacing between lines of bridging (Lbrmax) is determined per SJI specifications (SJI 200-2015 Section 5.5.3).
    b. The number of bridging lines is calculated as: n = L/Lbrmax - 1.

$$\ell_{brmax} = \left(100 + 0.67\,d_j + 40\frac{d_j}{L}\right) r_y \text{, in.} \qquad (5.5\text{-}2a)$$

$$\ell_{brmax} = \left(100 + 0.026\,d_j + 0.48\frac{d_j}{L}\right) r_y \text{, mm} \qquad (5.5\text{-}2b)$$

or,

$$\ell_{brmax} = 170\ r_y \qquad (5.5\text{-}3)$$

Where:

d_j = **CJ**-Series joist depth, in. (mm)
L = **CJ**-Series joist span length, ft. (m)
r_y = radius of gyration of the top chord about the vertical axis of the joist cross section, in. (mm)

8. Seat Depth is calculated based on the top chord leg width per SJI Table 5.4-1.
9. Stud Design:
    a. The number of required shear connections in half-span by design is calculated as.

$$N_{Design} := \max\left(\frac{0.5 \cdot A_{sbc} \cdot F_y}{Q_n}, \frac{M_{u_total}}{\phi_{stud} \cdot Q_n \cdot d_e}\right)$$

b. The shear capacity Qn for each stud is calculated based on the Composite Joist Specification (SJI 200-2015, Section 4.5.4). The equations are listed below:

For studs in 1.5 in. (38 mm), 2 in. (51 mm), or 3 in. (76 mm) deep decks with $d_{stud}/t_{topchord} \le 2.7$:

$$Q_n = \text{Min}\left[0.5A_{stud}\sqrt{f'_c E_c}\ ,\ R_p R_g A_{stud} F_{u\,stud}\right] \text{ (kips)} \quad (4.5\text{-}1a)$$

$$Q_n = \text{Min}\left[0.5A_{stud}\sqrt{f'_c E_c}\ ,\ \left(R_p R_g A_{stud} F_{u\,stud}/1000\right)\right] \text{ (kN)} \quad (4.5\text{-}1b)$$

For studs in 1.5 in. (38 mm), 2 in. (51 mm), or 3 in. (76 mm) deep decks with $2.7 < d_{stud}/t_{top\ chord} \le 3.0$:

$$Q_n = \text{Min}\left[0.5A_{stud}\sqrt{f'_c E_c}\ ,\ R_p R_g A_{stud} F_{u\,stud} - 1.5\left(\frac{d_{stud}}{t_{top\ chord}} - 2.7\right)\right] \text{(kips)} \quad (4.5\text{-}2a)$$

$$Q_n = \text{Min}\left[0.5A_{stud}\sqrt{f'_c E_c}\ ,\ \left(R_p R_g A_{stud} F_{u\,stud}/1000\right) - 6.67\left(\frac{d_{stud}}{t_{top\ chord}} - 2.7\right)\right] \text{(kN)} \quad (4.5\text{-}2b)$$

Where:

A_{stud} = cross-sectional area of shear stud, in.^{2} (mm^2)
d_{stud} = diameter of shear stud, in. (mm)
E_c = modulus of elasticity of the concrete, ksi (MPa)
f'_c = specified compressive strength of concrete, ksi (MPa)
$F_{u\,stud}$ = minimum tensile strength of stud, 65 ksi (450 MPa)
Q_n = shear capacity of a single shear stud, kips (kN)
R_p = shear stud coefficient from Table 4.5-1
R_g = 1.00 for one stud per rib or staggered position studs
= 0.85 for two studs per rib side-by-side
= 0.70 for three studs per rib side-by-side
$t_{top\ chord}$ = thickness of top chord horizontal leg or flange, in. (mm)

10. Composite Deflection Calculation:
    a. The post-composite deflection is calculated based on the effective composite moment of inertia Ieff per the following equation:

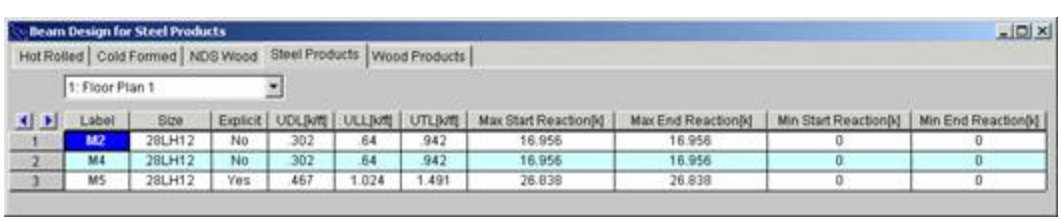
$$\Delta = \frac{5 \cdot \omega \cdot L_{des}^4}{384 \cdot E \cdot I_{eff}}$$

    b. The effective composite moment of inertia Ieff is determined following the AISC Steel Design Guide 11 "Floor Vibrations due to Human Activity".

Note:

- The design results for composite joists in RISAFloor are for budgetary purposes. The final design and detailing of composite joists will be determined by an SJI certified manufacturer.

Design Results - Steel Products



	Label	Size	Explicit	UDL[k/ft]	ULL[k/ft]	UTL[k/ft]	Max Start Reaction[k]	Max End Reaction[k]	Min Start Reaction[k]	Min End Reaction[k]
1	M2	28LH12	No	.302	.64	.942	16.956	16.956	0	0
2	M4	28LH12	No	.302	.64	.942	16.956	16.956	0	0
3	M5	28LH12	Yes	.467	1.024	1.491	26.838	26.838	0	0

The results for the **Steel and Wood Products** tab are explained below. The steel product results give the uniform loads applied to the joists. The results for the wood products give the maximum bending moments and shears as well as the allowable moments and shears for the joist. The pull down list at the top of the spreadsheet allows you to toggle between floors.

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

The **Size** column displays the designation of the steel or wood product. When no adequate member could be found from the redesign list, this field will display the text "not designed". Consider reframing, relaxing the design or deflection requirements (see Design Optimization), or adding more shapes to the available Redesign List (see Appendix A – Redesign Lists).

The **Explicit** column displays "Yes" if the beam has been locked to an explicit beam size by the user. When you have chosen a specific shape to override the programs automatic redesign, that beam becomes "locked" and will not be automatically redesigned by the program.

Note:

- To "unlock" a beam, you can use the beam – modify tool to assign a shape group. If the model has already been solved, you may optimize a beam by using the Member Redesign dialog. See Member Redesign for more details.

Uniform Loads on Steel Products

The **UDL** column for Steel Products displays the total uniform dead load on the beam. This includes the sum of the beam self weight, the deck self weight, and the applied dead load.

The **ULL** column for Steel Products displays the total uniform live load on the beam as described above in the Steel Products Design section. .

The **UTL** column for Steel Products represents the total of Dead plus Live loads and displays the sum of UDL and ULL.

Note:

- The load combinations defined within the Load Combinations spreadsheet are not used for the design of steel products. Rather, steel products are designed to the UDL, ULL and UTL loads described above.

End Reactions

The **Max Start & End Reactions** column displays the maximum start and end reactions of the beam for ALL load combinations. If "Show Factored End Reactions" in Model Settings is left unchecked, these displayed loads are not factored. If it is checked, then the displayed loads will have been multiplied by the factors in the load combinations. The sign convention assigns positive reactions to downward forces. Negative reactions, if they occur, would indicate uplift.

The **Min Start & End Reactions** column displays the minimum start and end reaction of the beam.

Limitations

Top Chord Extensions

The program does not currently allow for a cantilevered extension of the top chord of a joist. If this is done, the program will NOT design the joist.

Continuous Beams or Beams with Fixed Ends

The program does not design continuous or fixed end Joists or Joist Girders.

Uplift Forces and Bridging

No attempt is made to design joists for uplift forces or to call out bridging requirements that may be required during construction.

Design for Diaphragm Chord Forces

No attempt is made to calculate or design for the axial force imparted onto the members of the structure that will serve as the main chords of roof or floor diaphragms.

No Design for Spans Less then 10 Feet

The program does not design joists for less than 10 feet because the allowable capacity joist tables do not give explicit capacities for shorter span joists.

Composite Joist Design Limitations

Pre-composite design, camber design, long-term deflection, shear check are not provided in the program. These design checks need to be provided by SJI certified joist manufacturers. Vibration analysis for composite joists is also not provided currently.

Only 0.5 in, 0.625 in, 0.75 in stud sizes and 65 ksi stud Fu are currently supported for composite joist design.

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.
- Columns in RISAFloor are assumed to be fully braced at each floor level when their unbraced length field is left empty/blank.

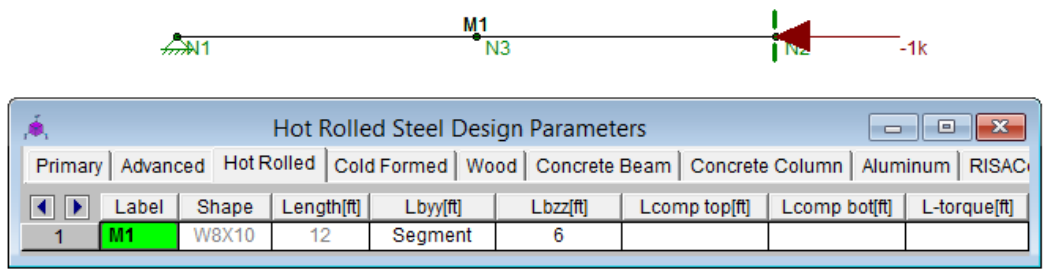
You may specify unbraced lengths as a fixed distance or by using RISA's Unbraced Length Commands. The unbraced lengths are:

- **Lbyy**
- **Lbzz**
- **Lcomp-top**
- **Lcomp-bot**
- **L-torque**

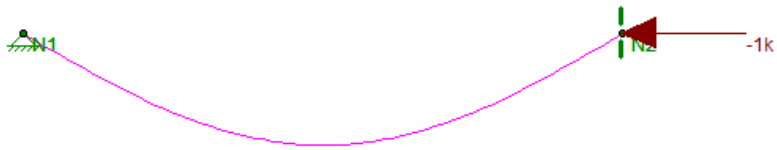
The unbraced lengths that are used for member capacity calculations are listed in the **Member Detail** report after solution.

Note:

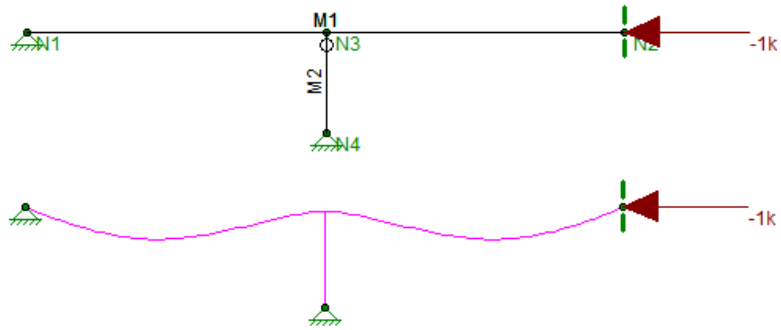
- Specifying bracing via unbraced lengths does not provide physical bracing to the model for analysis. It only affects the member capacity calculated for code checks. See the example below.



For compression capacity, M1 (above) is braced at N3 based on specified Lbyy and Lbzz



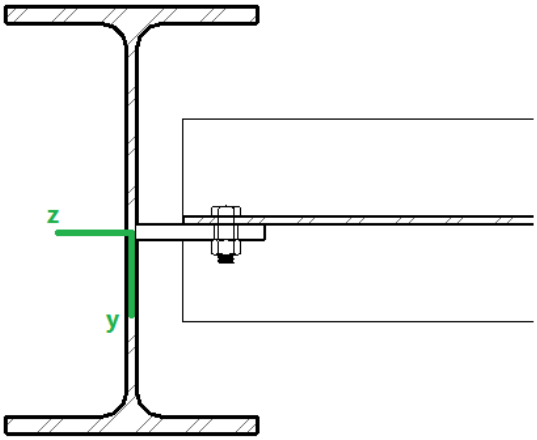
However, for analysis this member is not considered braced (see above)



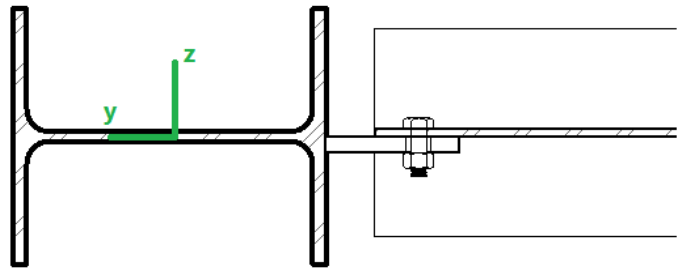
A bracing member (M2) must be modeled to provide bracing for analysis

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



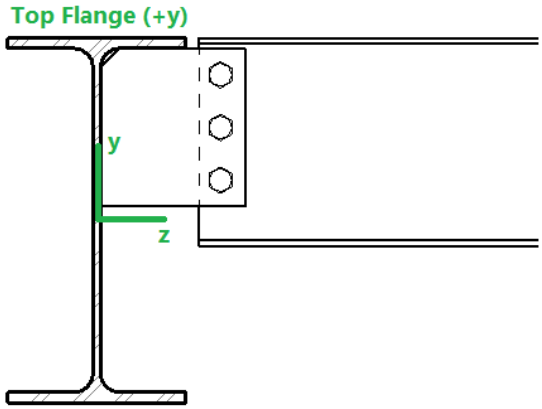
Lb-zz Bracing prevents column from buckling about z-z axis

Lcomp Values (Lcomp, Le-bend)

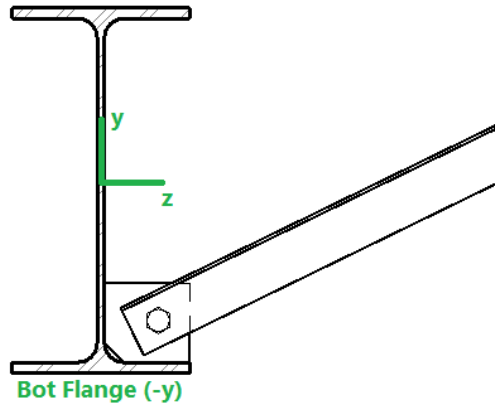
The **Lcomp** values, **Lcomp-top** and **Lcomp-bot**, represent the distance between points which brace the top or bottom flange of the member against Lateral-Torsional (beam-type) Buckling. These **Lcomp** values are used to calculate the member's flexural (bending) capacity. Where the top flange of 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 if rotated 180 degrees about its own axis (flipped upside down) the "top" flange will actually be facing downwards.



Lcomp-top Bracing prevents lateral movement of beam's top flange



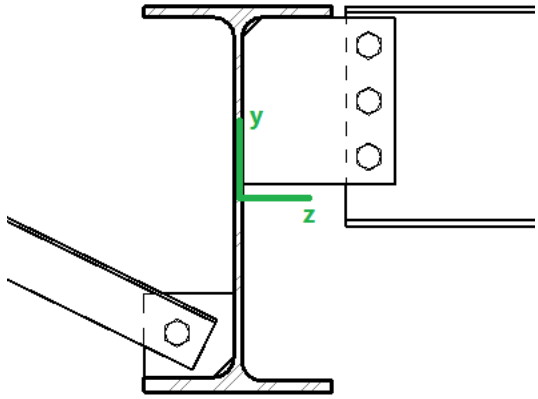
Lcomp-bot Bracing prevents lateral movement of beam's bottom flange

Note:

- Concrete members are not checked for lateral-torsional buckling, so **Lcomp** 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.



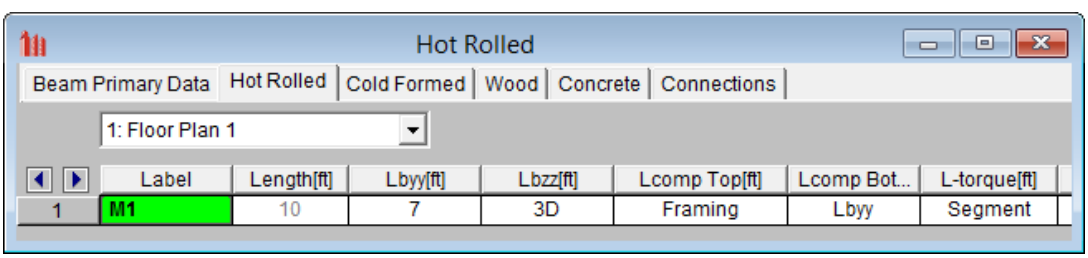
L-torque Bracing prevents beam from twisting about its own axis

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** > **Lbzz***and* **Lbyy**.
- 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.



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

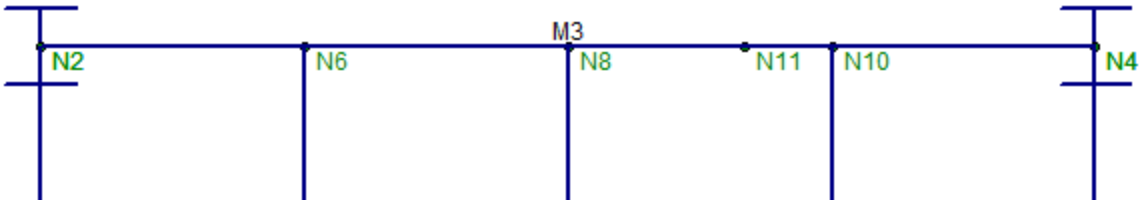
When this command is used, all Points which fall along the length of the physical member are assumed to provide bracing. If Points are not evenly spaced along the member then an unbraced length is calculated for each "segment". A segment is defined as the distance between adjacent Points. In the example 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 Lb_1, and the forces occuring in Segment 2 will be compared against a capacity calculated using Lb_2.

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.

Framing

This command is only available for beam members in RISAFloor. This command works like the Segment command, except that only nodes which have another beam framing into them are considered brace points. In the example below, beam M3 is considered braced at N6, N8, and N10. N11 is ignored as a brace point since no other beam frames into M3 at N11. If the Segment command were used instead of Framing then N11 would be considered a brace point.

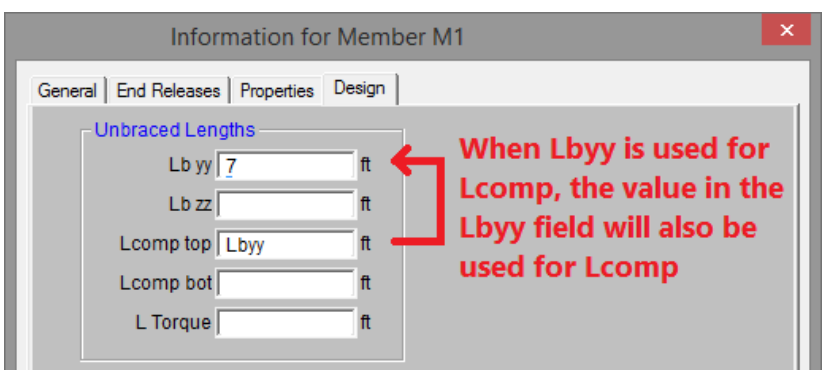


For Lcomp-top only, the unbraced length of the top flange is taken as the lesser of the Framing length and the Deck Unbraced Length.

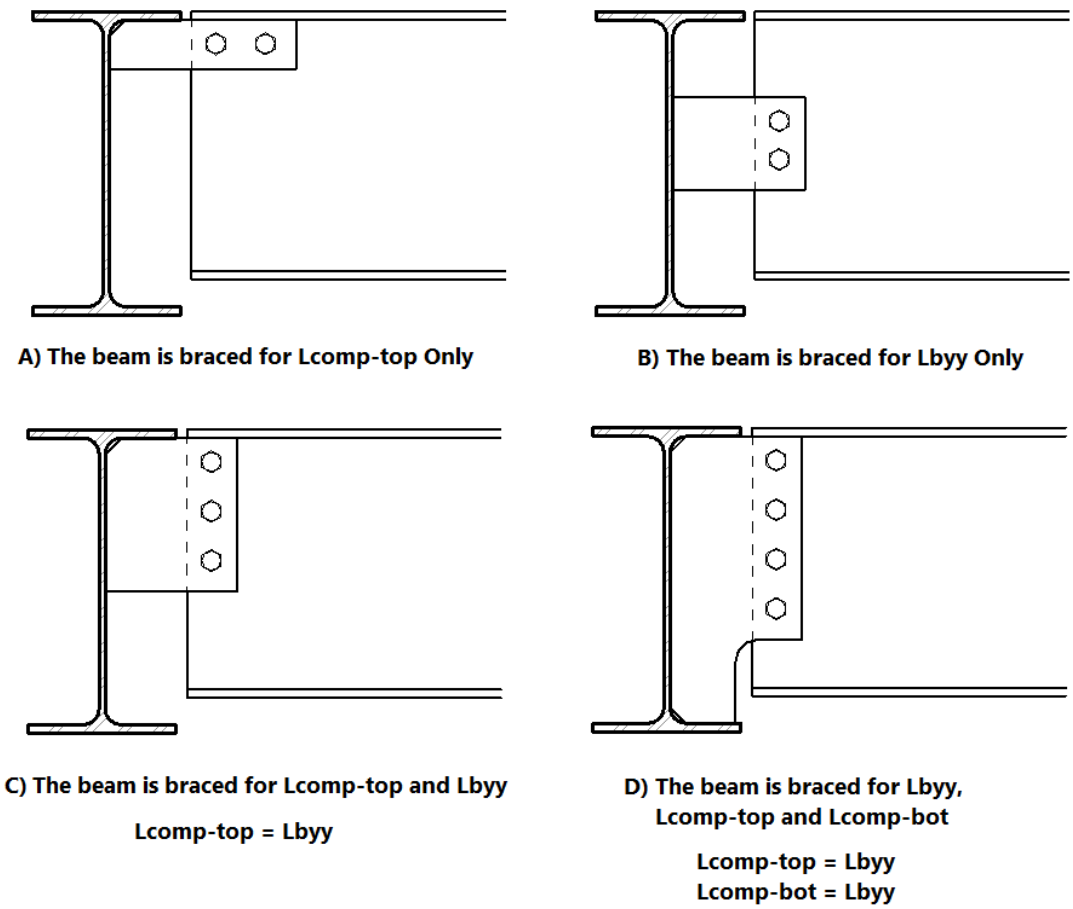
When using the Framing command it is the user's responsibility to ensure that adequate bracing is provided by the framing and/or deck.

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.



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.



3D

For RISAFloor models which are linked to RISA-3D models, if an unbraced length is set in RISA-3D, it is listed as **3D** in the RISAFloor spreadsheets. This indicates that the unbraced length was set in RISA-3D, and that same value is being used in the RISAFloor model.

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)
Lt = Lcomp
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)
Lt= smaller of Lbyy and Lcomp

When Lateral Torsional Buckling governs over local buckling, the display of the Seff in the Detail Report will report the Sc which is the elastic section modulus of effective section calculate relative to the extreme compression fiber at Fc.

Distortional Buckling (AISI C3.1.4)
Lm=Lcomp
Lcomp is the unbraced length for the flange that's in bending compression at that section (either Lcomp_top or Lcomp_bot).

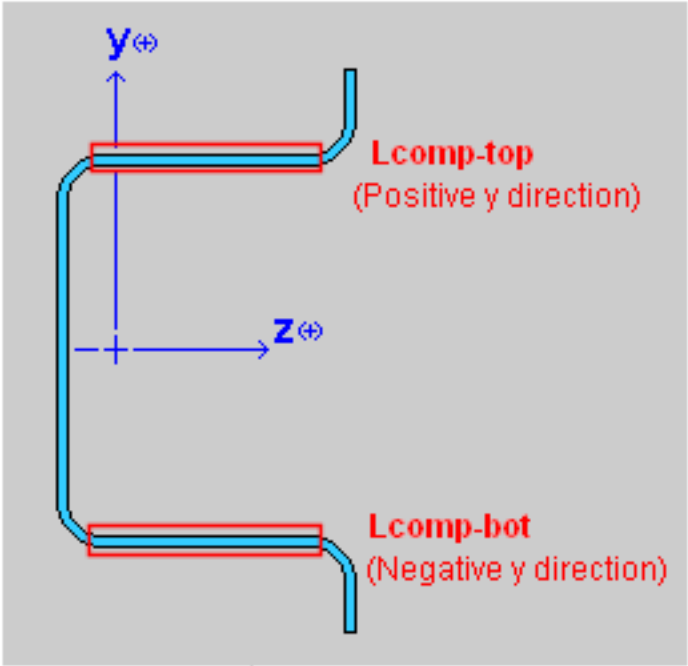
Axial Strength (AISI C4.1)
L= Could be Lbyy or Lbzz, whichever gives the larger of (KL/r)yy or (KL/r)zz

Flexural Torsional Buckling or Torsional Buckling (AISI C4.1.2)
Lt= L-Torque

Note: Older Versions of RISA-3D (v12 and older)
Lt= smaller of Lbyy and Lcomp

Distortional Buckling (AISI C4.2)
Lm= L-Torque

- The Torsional Warping constant for Back-to-Back Channels and Tracks, Cw 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), Cw is twice the value of each individual section.
- For Hat Channel (HU) shape types, the **Lcomp-top** and **Lcomp-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.

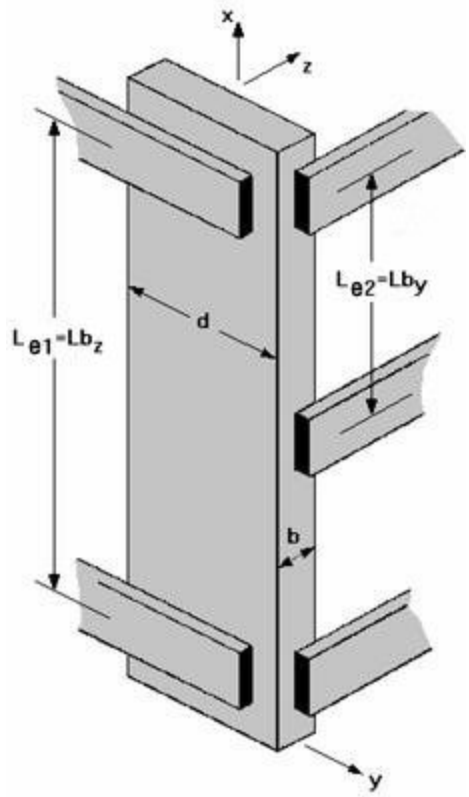


Concrete

- The unbraced lengths for Flexural Buckling (Lbyy, Lbzz) are called Lu-yy and Lu-zz for concrete. Their behavior is the same as the behavior for Lbyy and Lbzz.
- The unbraced lengths are used for the Moment Magnification procedure in older versions of the ACI code.

Wood

- The unbraced lengths for Flexural Buckling (Lbzz, Lbyy) are called Le1 and Le2 for wood. Their behavior is the same as the behavior for Lbyy and Lbzz.



- The unbraced lengths for Lateral-Torsional buckling (Lcomp-top, Lcomp-bot) are called Le-bend Top and Le-bend Bot for wood. Their behavior is the same as the behavior for Lcomp-top and Lcomp-bot.
- See AF&PA NDS-2012, Table 3.3.3 for information on how to determine Le-bend based on the unbraced length. This procedure is not followed automatically in RISA.

Aluminum

- 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.

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)

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 segment of the physical column between the current floor level or splice above and/or the floor level or splice below. When working in RISA-3D via RISAFloor, the largest K factor entered for any segment of a physical column will be used for the entire physical column. If an entry is not made (left blank), the value will internally default to '1' for that column segment.

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.

Sway Flags

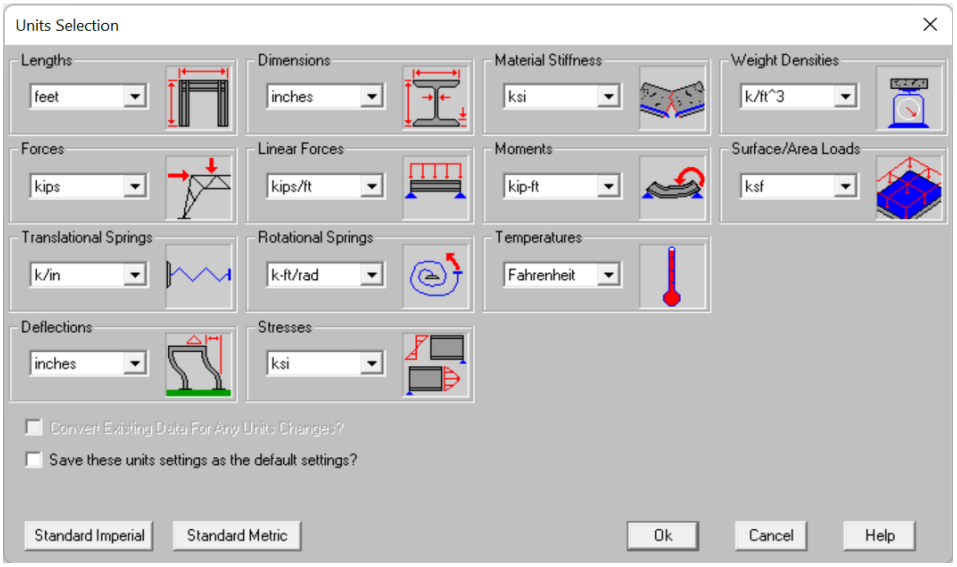
The **Sway Flags** indicate whether the member is to be considered subject to sidesway for bending about its local y and z axes. The **y sway** field is for y-y axis bending and the **z sway** field is for z-z axis bending. Click on the field to check the box and indicate that the member is subject to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

Sway flags are reserved for column members in RISAFloor and may be applied to any column segment at any floor level. However, when working in RISA-3D via RISAFloor, if a sway flag is checked for any segment of a physical column, the entire physical column will be assumed subject to sway. Beam members in RISAFloor are assumed to be braced against sway.

Units

You can work with imperial (kips, inches, etc.) or metric (KN, meters, etc.) units, or any combination of the two. The current units appropriate for each data item are shown at the tops of the data columns in the spreadsheets and with the plot of values in the model view.

You may save any of the units as the default setting so that when you start a new model that information is already there. To do this, simply enter the information that you want to save and check the **Save as Defaults** box and click **OK**.



To Change Units

Standard units systems are preset and may be specified by clicking the **Standard Imperial** and **Standard Metric** buttons.

3. If you do not wish to convert values already entered then clear the check box for **Converting Existing Data**.

Standard Imperial Units

This is the units system currently prevalent in the United States. 'Feet' are used for location entries such as point coordinates and load locations, and 'Inches' are used for section property entries such as area and moment of inertia. Force and weight units are in 'kips', where 1 kip = 1000 pounds. Stress units are in 'ksi' (kips per square inch).

Standard Metric Units

This units system uses 'Meters' for location entries and 'Centimeters' for section property entries. Force units are in 'kN' (kiloNewtons), where 1 KN = 1000 Newtons. Stress units are in 'MegaPascals' (MPa), where a MegaPascal is 1,000,000 Newtons per square meter. Weight units are in 'Kilograms' and thermal units are in 'Degrees centigrade.

Units Specifications

The following are the unit specifications and their applications:

Measurement	Usage
Lengths	Coordinates, Unbraced Lengths, Load Locations

Measurement	Usage
Dimensions	Shape Properties, Wall Thickness, Member Offsets
Material Stiffness	E, G
Weight Densities	Material Density
Forces	Loads, Forces
Linear Forces	Line Loads
Moments	Loads, Forces
Area Loads	Area Loads, Deck Self Weight
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

Vibrations - Floor Framing

RISAFloor implements AISC's vibration calculations and automatically calculates floor framing accelerations and frequencies as a standard part of every design. This method is based on walking resonance and is described in detail in the AISC publication, Design Guide 11: Floor Vibrations Due to Human Activity. For floor with steel joist framing, RISAFloor follows SJI Technical Digest 5: Vibration of Steel Joist - Concrete Slab Floors guidelines on calculating floor accelerations and frequencies.

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

Vibrations Spreadsheet

The **Vibration Results Spreadsheet** displays the results for each beam that has been checked. The pull down list at the top of the spreadsheet allows you to toggle between floors.

The Vibration code and the damping ratio are set on the Codes tab of the Model Settings. If you do not want any vibration checks to be performed you may specify “none” as the vibration code.

Beam Vibrations

1: Floor Plan 1

	Beam	Ij[in]	Delta j[...	fj[Hz]	Wj[k]	Ig[in^4]	Delta g...	fg[Hz]	Wg[k]	fn[Hz]	Wj[k]	%g
36	B38	2601.294	.15	9.135	.62					9.135	130.911	0
37	B39	2638.2	.758	4.063	1.006					4.063	341.704	.002
38	B40	455.641	.043	17.092	.497	2638.2	.965	3.601	103.066	3.523	99.379	.635
39	B42	662.586	.309	6.36	.326					6.36	81.278	.003
40	B51	665.945	.308	6.376	.326					6.376	81.175	.003
41	B53	2936.187	.686	4.27	1.014					4.27	335.15	.001
42	B54	453.641	.044	16.903	.506	2936.187	.873	3.785	100.978	3.693	96.95	.614
43	B55	1489.769	.376	5.764	.892					5.764	153.664	.002
44	B56	1265.959	.728	4.146	1.466					4.146	262.864	.002
45	B60	1265.959	.728	4.146	1.466					4.146	262.864	.002
46	B61	2862.273	.284	6.639	1.292					6.639	189.018	.001
47	M71	921.984	.271	6.798	.397	455.641	.027	15.147	9.509	6.479	70.916	.316
48	M72	2963.639	.552	4.762	.823	14958.858	.297	6.488	123.135	3.839	165.424	.342
49	M73	2963.639	.689	4.259	1.029	14958.858	.297	6.488	123.135	3.56	201.479	.309
50	M74	2963.639	.811	3.927	1.21	14958.858	.297	6.488	123.135	3.359	235.562	.284
51	M75	2963.639	.917	3.694	1.368	14958.858	.297	6.488	123.135	3.21	266.303	.265

Beam Label

The **Beam Label** column displays the beam label.

Moment of Inertia- Ij, Ig

Two Moments of Inertia are reported (**Ij** and **Ig**). Ij is the moment of the inertia for the beam being considered. Ig is the moment of inertia of the supporting beam if there is one. If there are two supporting beams, Ig will be based on the more flexible of the two supporting beams per section 4.3 of the Design Guide.

Note

- The combined Girder / Joist Panel mode results are only provided for beam members that are directly *supported* by a girder. Column supported girders themselves do not have a combined Girder / Joist Panel mode of their own. Those combined modes are instead associated with the *joists* that are supported by the girder. As such, column supported girders will not have an Ig, Deltag, or Wg associated with their vibration calculations. Instead, they behave as a simple Joist Panel mode with the normal Ij, Deltaj and Wj used in their vibration calculations.

Deflection- Delta j, Delta g

The midspan deflections (**Deltaj** and **Deltag**) are calculated from the supported vibration loads. Where Deltaj is the deflection for the beam being considered and Deltag is the deflection of the most flexible supporting girder if there is one.

Frequency-fj, fg, fn

The natural frequencies of the beam and girder (**fj** and **fg**) are based on their midspan deflections. They are used to calculate the panel frequency (**fn**). Refer to the AISC Design Guide for more information.

Panel Weights-Wj, Wg, W

The Equivalent Panel Zone Weight (**W**) is calculated from the effective beam panel weights (**Wj** and **Wg**). The load used to calculate W, Wj and Wg is a combination of member self weight, deck self weight and the VL load category.

Note

- The w (lower case) values shown in the detail reports are weight per length of the joist or girder as opposed to weight per area as they are shown in the design guide. This doesn't affect the calculation, but is mentioned to clear up any confusion that may occur when comparing the detail report output to the design guide.
- Because of this difference, the weight per length values are affected by the B value (effective width) of the joist or girder.

Steel Joists

Vibration calculations are performed for steel joist framing floor per SJI Technical Digest 5: Vibration of Steel Joist - Concrete Slab Floors.

Note

- Web members are assumed to be single or double angles in determining Cr values for effective moment of inertia of joists (SJI Technical Digest 5, Eqn. 2.7).
- Joist seat depth is assumed to be 5 inch standard depth for joist girder vibration calculations.
- No vibration calculation is currently available for joist girder if explicit girder shapes are used.

Calculation of Vibration Loads and Deflections

The program will automatically include the self weight of all members and deck when calculating the vibration loads. However, the portion of the *Super Imposed* Dead and Live Loads that should be included in the vibration loads must be specified by the user. This is accomplished by assigning the load to the "VL" Load Case. Therefore, the total vibration load will be the summation of the self weight loads and the VL loads.

The total vibration load is converted into an equivalent uniformly distributed load over the length of the member. The total load is assumed to be distributed equally over the length of the member as a uniform load as describe in section 3.3 of the AISC Design Guide.

The loads are then used to calculate the beam deflections (Delta j and Delta g). The only exception to this is when a girder supports a single point load at the mid-span of the member. When this occurs, the program uses a deflection equal to 4/pi times the regular equivalent load deflection as described in section 3.3 of the AISC Design Guide.

Note

- The VL load case is only available in the Area Load Definitions. Therefore, you cannot apply vibration loads as Point Loads, Line Loads, or Tapered Area Loads.

Limitations

Deck Type

AISC Design Guide #11 and SJI Technical Digest 5 only addresses vibrations on floors with composite deck or concrete slab. Therefore the program only provides vibration check when the Deck Type is selected as Composite Deck. Under Composite Deck type, one can select a specific composite deck system or concrete deck from the Database / Manufacturer drop-down list.

Floor Stiffness Requirement

The floor stiffness requirements recommended for panels that have a natural frequency greater than 9 or 10 Hz are not checked within RISAFloor.

Excitation Force

The excitation force Po that is used to calculate the walking excitation is always assumed to be 65 lbs. Per Table 4.1 of the design guide, this is an appropriate value for offices, residences, churches, and shopping malls. However, it may not be appropriate for footbridges.

Column Stiffness

Equation 3.5 of the design guide includes a deflection contribution for columns that applies for tall buildings. This is not currently accounted for by the program.

Continuous Beams

Section 3.4 of the design guide is not taken into account for continuous beams.

Cantilevered Beams/Outriggers

AISC Design Guide #11 does not have specific guidelines for cantilevered beams and outriggers, so currently there is no special consideration for these members by the program.

Effective Width Limitation

The program does not limit the Bg (effective width) calculation of equation 4.3b to 2/3 of the floor length.

Wall Panels

The wall panel element allows you to easily model walls for in plane and out of plane loads. Wall panel data may be viewed and edited in two ways: graphically in the Wall Panel Editor or in the Wall Panels Spreadsheet.

Drawing Wall Panels

There are several graphic-editing features that make the creation and modification of models quite easy. Use the Insert and Modify menus or the Drawing Toolbar to use these features in the model view. To create new wall panels, you can draw them using a drawing grid or draw "dot to dot" existing joints. Once you have created these items you may use other graphic features to apply loads and set boundary conditions.

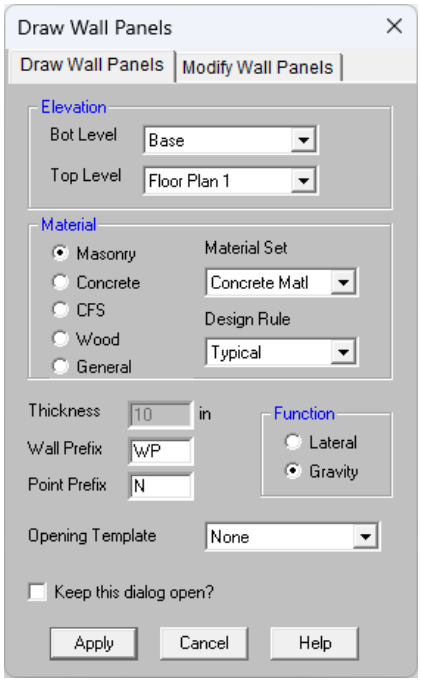
You can set many of the wall panel properties up front or you can modify these properties after you draw them. Modifying properties is discussed in the next sections. See the Wall Panels topic for information on wall panels and their properties.

The **Draw Wall Panels** button lets you graphically draw wall panels in your model. Enter the appropriate wall panel parameters, click **OK** and draw wall panels between existing joints or on the drawing grid. You will also notice that the coordinates of the joint or grid point that is closest to your cursor are displayed in the lower right hand corner of the model view. The new wall panels will be shown on screen and will be recorded in the **Wall Panels Spreadsheet**.

To actually draw a wall panel, you have two options. One way is to modify your Drawing Grid according to how you wish to lay out your wall panels and use the **Create Wall Panels by Clicking on Grid Areas** option. Wall panels can then be created by clicking in the grid areas formed by the intersecting grid lines. As you click on an area, a wall panel will automatically be created in that area. The second option is to create wall panels by drawing them one joint at a time. First click on the grid point or joint that you want to be the "A" joint for the plate, then the "B" joint, "C" joint, and "D" joint in either clockwise or counter-clockwise order. The wall panel will "stretch" like a rubber band as you draw from joint to joint.

Note:

- You must draw wall panels rectangular.



The parameters shown are the same parameters that you would enter on the Wall Panels Spreadsheet.

To Draw Wall Panels

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. If you are not drawing between existing joints, you will need to create a drawing grid or define joints on the **Joint Coordinates** spreadsheet.
3. Click the **Draw / Modify Wall Panels** button and select the **Draw Wall Panels** tab. Then set the wall panel properties.
4. Click **Apply** to start drawing wall panels by connecting grid points or by clicking on individual grids.
5. To stop drawing altogether right click or press the **Esc** key.

Note

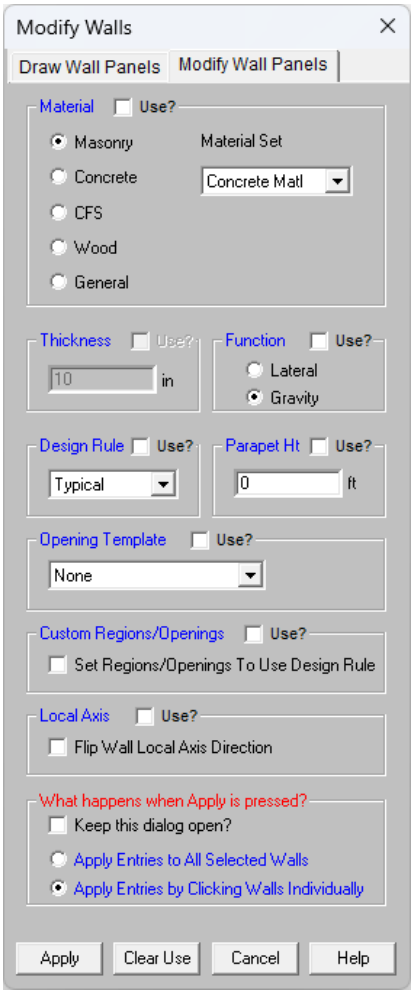
- To draw more wall panels with different properties, press CTRL-D to recall the **Wall Panel Properties** settings.
- You may also specify or edit wall panels in the Wall Panels Spreadsheet.
- You may also view and edit wall panel properties by double-clicking on a wall panel.
- You may undo any mistakes by clicking the Undo button.
- Drawing clockwise or counter-clockwise determines the panel's local axes directions. See Wall Panel Local Axis for more information.
- You can choose elevations where the wall panels begin and end if you have multiple floor levels. This allows you to have continuous wall panels along a given height.
- When graphically drawing wall panels, it is irrelevant which floor you are on when you are drawing. You can draw wall panels on floors that are not in the current view. Thus, you need to be careful when drawing wall panels that you have the right floor to floor levels for your wall panels.
- In the **Draw Wall Panels** tab, you can select an Opening Template to be applied to the wall panel you are drawing. See Wall Opening Template for more information.

Modifying Wall Panels

There are a number of ways to modify wall panels. You may view and edit the member data in the **Wall Panel Spreadsheet**, you may double-click a wall panel to view and edit its properties, or you can use the **Modify Wall Panels** tool to graphically modify panels.

The graphical **Wall Panel Modify** tool modifies the properties of wall panels that already exist in the model. To use this tool, specify the properties you want to change and then select the wall panels that you want to apply these changes to. Wall panels can be modified one-at-a-time by selecting the **Apply Entries by Clicking** option. This will change the mouse cursor to a modify tool which applies to any wall panels which you click on. A group of selected wall panels can be modified all at once by using the **Apply Entries to All Selected** option. See the Graphic Selection topic for more on selecting.

The parameters shown are the same as those used to define new wall panels.



The **Use?** check boxes next to the data fields indicate whether the particular parameter will be used or not when the modification is applied. If the box next to a field is checked, that parameter will be applied to any selected wall panels. If the box is NOT checked, the parameter will NOT be applied, even if a value is entered in the field. This allows you to easily change one or two properties on wall panels without affecting all the rest of the properties. Note that if a no value is entered in a field (i.e. the field is blank) and the corresponding check box is checked, clicking "Apply" will have the effect of clearing the data for that field.

To Modify Wall Panels

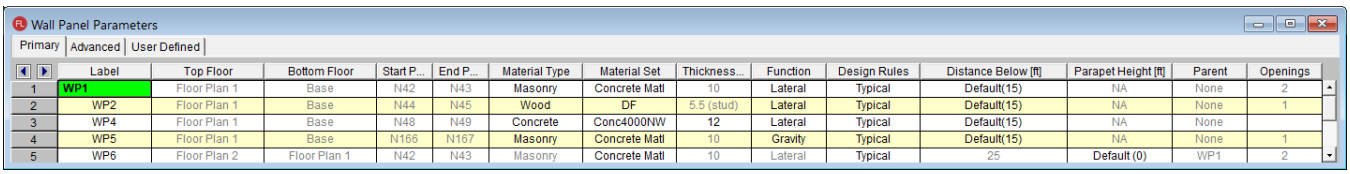
1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Click the **Draw / Modify Wall Panels** button and select the **Modify Wall Panels** tab. Then set the parameters for the new wall panels. Check the **Use?** Box for the items to apply.
3. You may choose to modify a single wall panel at a time or to an entire selection of wall panels.
    1. To modify a few wall panels choose **Apply Entry by Clicking Items Individually** and click **Apply**. Click on the wall panels with the left mouse button.
    2. To modify a selection of wall panels, choose **Apply Entries to All Selected Items** and click **Apply.**

Note

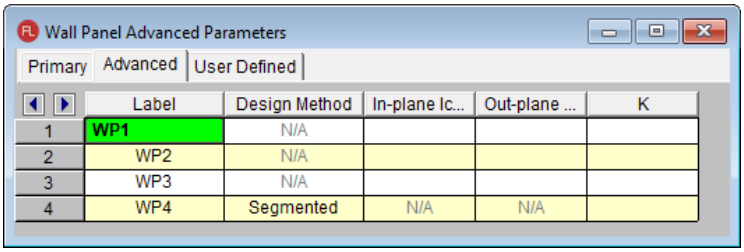
- To modify more wall panels with different parameters, press CTRL-D to recall the **Modify Wall Panels** settings.
- You may also modify wall panels in the Wall Panels Spreadsheet.
- You may undo any mistakes by clicking the **Undo** button.
- The thickness option is only available if you are choosing a General or Concrete material. Wood and Masonry require you to change their thickness in the Design Rules spreadsheet.
- Only walls that exist on the currently viewed floor will be modified if "Apply to All Selected.." is chosen.
- For existing models of version 5.1.1 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.
- In the **Modify Wall Panels** tab, you can select an Opening Template to apply to selected walls. See Wall Opening Template for more information.

Wall Panel Spreadsheets

Another way of editing wall panels is through the **Wall Panel Spreadsheet**. This spreadsheet is accessible through the **Data Entry Toolbar** and includes data on two tabs: Primary and Advanced.



	Label	Top Floor	Bottom Floor	Start P...	End P...	Material Type	Material Set	Thickness...	Function	Design Rules	Distance Below [ft]	Parapet Height [ft]	Parent	Openings
1	WP1	Floor Plan 1	Base	N42	N43	Masonry	Concrete Matl	10	Lateral	Typical	Default(15)	NA	None	2
2	WP2	Floor Plan 1	Base	N44	N45	Wood	DF	5.5 (stud)	Lateral	Typical	Default(15)	NA	None	1
3	WP4	Floor Plan 1	Base	N48	N49	Concrete	Conc4000NW	12	Lateral	Typical	Default(15)	NA	None	
4	WP5	Floor Plan 1	Base	N166	N167	Masonry	Concrete Matl	10	Gravity	Typical	Default(15)	NA	None	1
5	WP6	Floor Plan 2	Floor Plan 1	N42	N43	Masonry	Concrete Matl	10	Lateral	Typical	25	Default (0)	WP1	2



	Label	Design Method	In-plane Ic...	Out-plane ...	K
1	WP1	N/A			
2	WP2	N/A			
3	WP3	N/A			
4	WP4	Segmented	N/A	N/A	

The following data columns hold the primary data for the wall panels:

Wall Panel Labels

You may assign a unique label to any or all of the wall panels. You can then refer to the wall panel by its label. Each label has to be unique, so if you try to enter the same label more than once you will get an error message. You may relabel wall panels at any time with the Relabel Wall Panels option on the Tools menu.

Top/Bottom Floor

This defines what level in the structure the wall panel starts and stops. This can be edited in this window.

Start/End Point

This shows the start and end joint that defines the extreme ends of the wall. This is for display only. You must delete and redraw wall panels to modify these values.

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.

Wall Panel Function

If the function is defined as Lateral, then this element will be brought into RISA-3D for the lateral design. If the wall panel is defined as Gravity, then it will only exist in RISAFloor.

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.

Distance Below

This value allows you to define the height of the lowest level walls in the structure. This allows you to have control over bottom elevation of each individual wall panel. By default we will use the **Splice Distance Below** value from the Floors spreadsheet. If you change this value and want to go back to the default, simply delete the value.

For wall panels that do not sit at the base, this field is not used and will be grayed out.

Parapet Height

This value allows you to define the height of a wall parapet. The Parapet Height entry will be used to calculate the self-weight of the parapet for loading and seismic weight calculations. It will also physically bring the parapet into RISA-3D on beam-supported floors (not for slab floors).

The Floors spreadsheet defines a default parapet height that will update all walls simultaneously, thus that is the primary place to update your parapet heights. You would only update them in the Wall Panels spreadsheet if you had differing height parapets around your roof level.

Note

- Wall parapets can only be defined at the top of a wall stack and must be collinear with a deck edge.
- Wall parapets can not be added above slab floors. The deck edge defines the parapet perimeter in that case.
- Edits to the Parapet Height must be input greater than the Parapet Height Default from the Floors spreadsheet.
- See the Load Generation - Wind Loads topic for the effects on wind load calculations.

Parent

This column will show if there is a parent wall panel to the current wall panel. See Parent/Child for more information.

Openings

This column shows how many openings are present in a wall panel. The cell will be blank if a wall panel does not have any openings. Clicking in the cell will open the Wall Panel Editor where you can review and make modifications to your openings.

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.

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 Igross 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 Icr 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 Icr Factor and 0.35 for Out-plane Icr 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*Icr) 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:

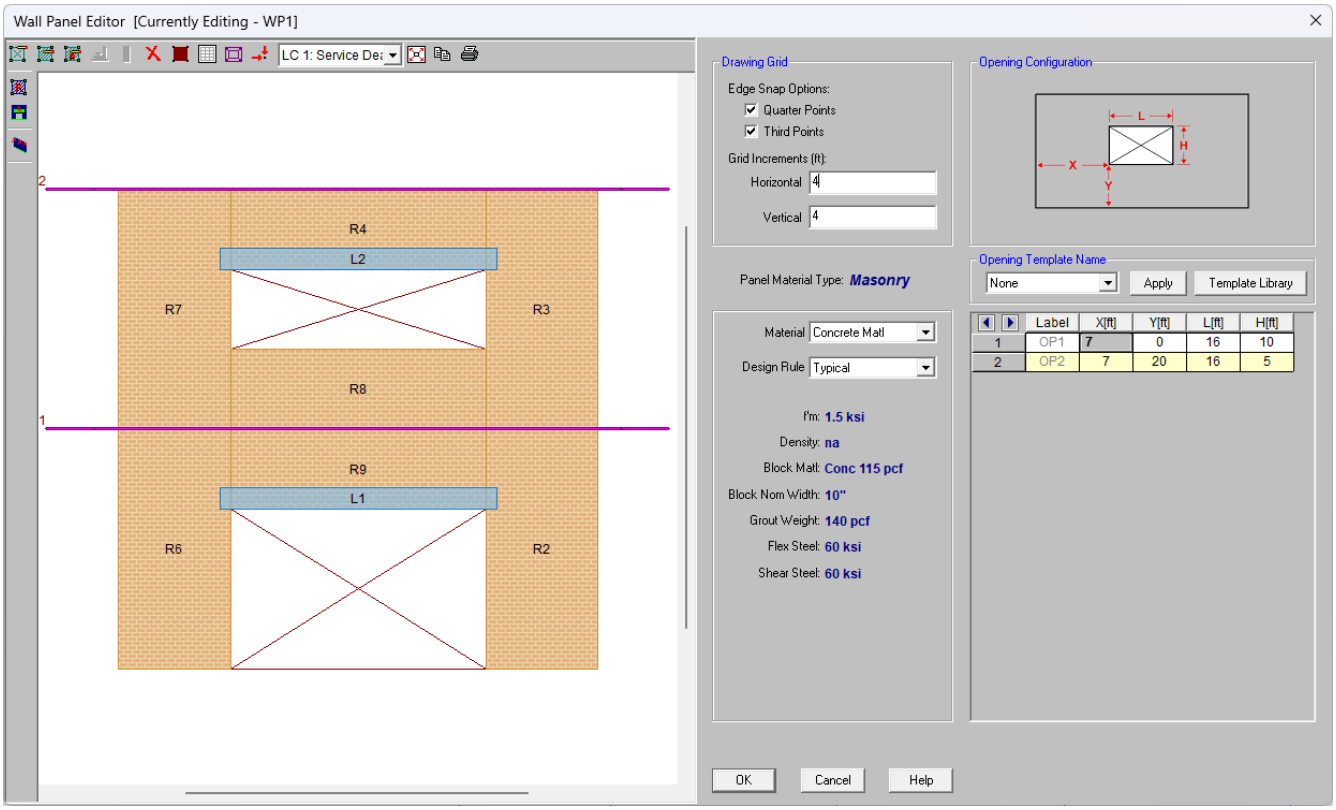
- For masonry this factor affects ACI 530-11 equations 2-16, 2-17, 2-19, 2-21, 2-22, 3-11, 3-12, 3-18, 3-19, and Section 3.3.5.3.
- For wood this affects the C_p factor calculation (specifically F_{cE}) for wood compression capacity for both studs and chord design.

Inactive

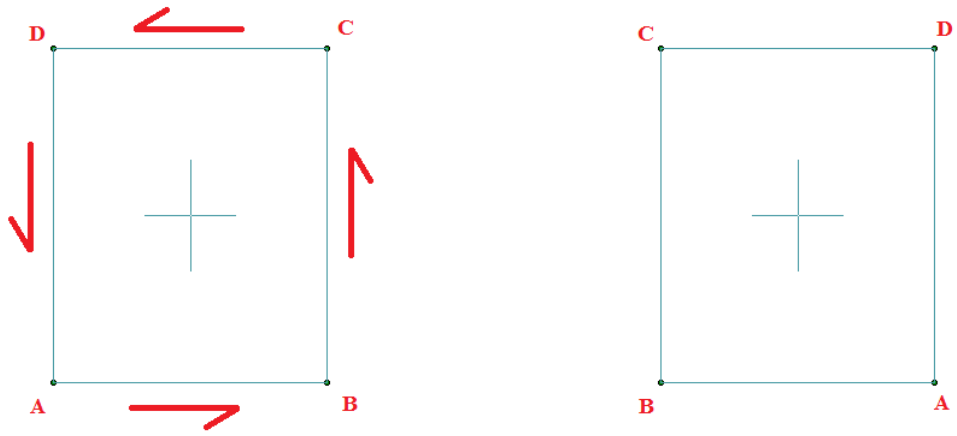
This column allows you define a wall panel as inactive or excluded. Please see the Inactive and Excluded Elements for more information.

Wall Panel Editor

The **Wall Panel Editor** allows the user to edit the detailed properties of a wall panel including openings and regions. This dialog also gives 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 clockwise it will be drawn with the panel's local x-axis going from left to right. If drawn counter-clockwise 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.

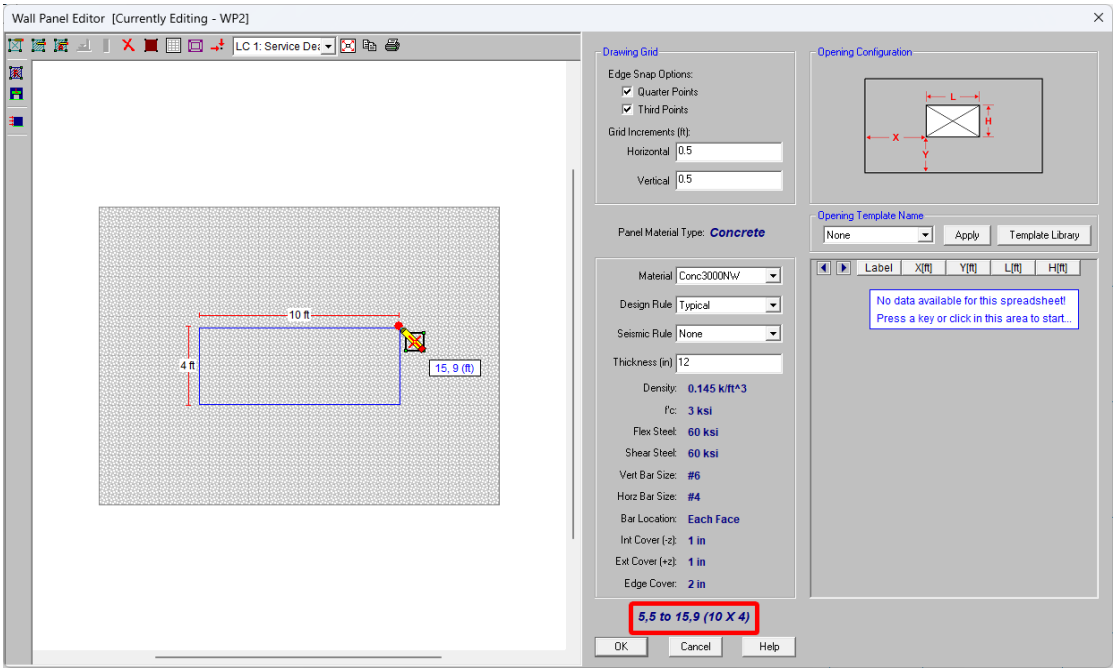


Note:

- There are many icons, drop-down 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.

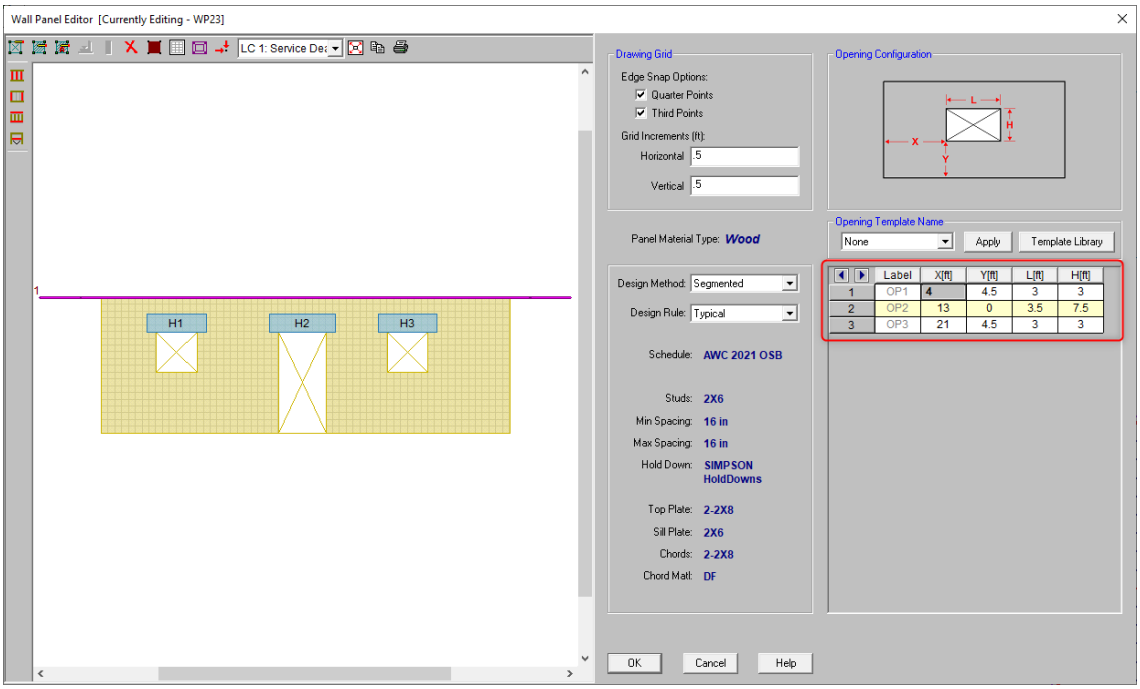
Creating Openings

Within the **Wall Panel Editor**, you have the option of adding rectangular openings to the wall panel. To draw an opening, select the **Create New Openings** button and then select two grid intersections which make up the two diagonal corners of your opening. Notice that you can view the relative coordinates at the cursor location by hovering over a snap point. The opening dimensions are displayed based on the first click and the current location of the cursor. The relative coordinates and opening dimensions are also displayed in the lower right portion the Wall Panel Editor window.



To exit this tool right-click your mouse.

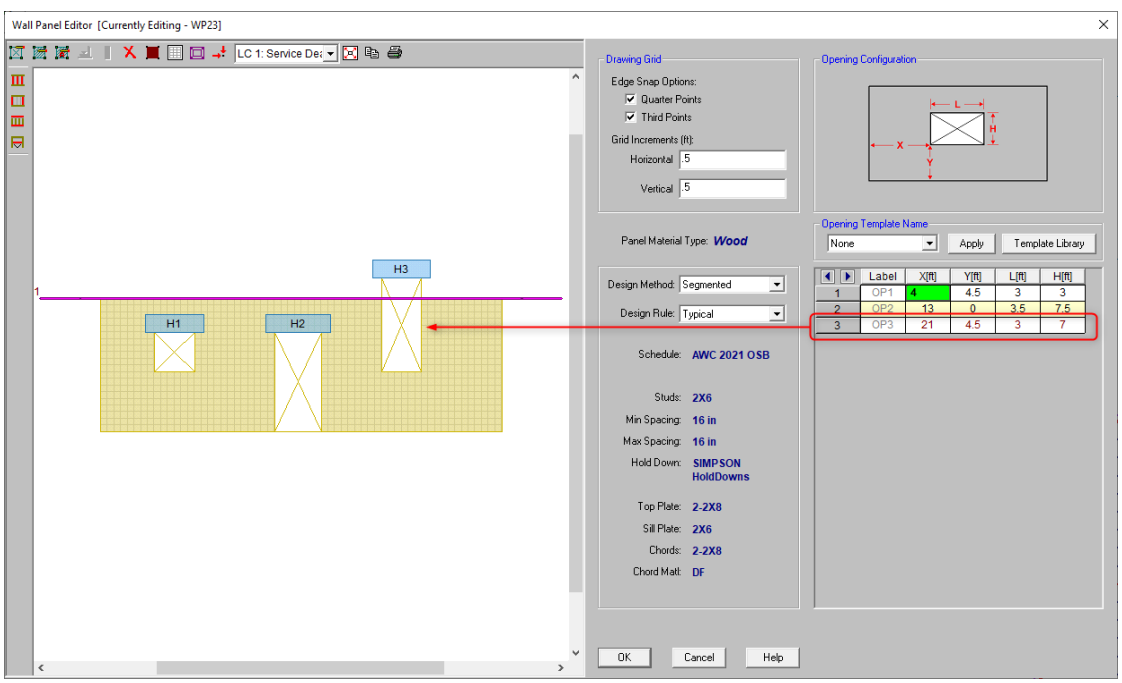
You can also create openings using the spreadsheet provided in the Wall Panel Editor. Similar to other spreadsheets, you can add, delete, and modify the geometries of your openings. The variables used for defining openings are shown in the figure in the upper right portion of the window. The location of the opening (X, Y) is defined from the lower left corner of the opening to the lower left corner of the wall panel. Spreadsheet operations such as Fill Block and Math on Block can also be used within the spreadsheet to quickly modify multiple cells. These spreadsheet tools can be accessed by right-clicking in the spreadsheet. See Spreadsheet Operations for more information.



If you have invalid openings, which include:

- Openings beyond the wall extents
- Openings overlapping with another opening
- Openings overlapping with an existing wall region
- Openings spanning over a diaphragm
- Openings created on an inactive floor

The program will display red text in the spreadsheet for the entries associated with the invalid openings.



When you click OK, the program will display an error message asking you to fix the invalid wall openings before they can be applied to the wall panel.

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 must be drawn greater than 3 inches away from wall edges.

Creating Regions

Within the Wall Panel Editor, you also have the option of creating different rectangular regions within your wall panel. Regions are used to further define areas of your wall panel for use in analysis/design. If you do not specify a region in a wall panel without openings, then the entire wall panel will be considered a region.

To automatically draw regions you must first have your openings input. Once you have that you can click the **Generate Wall Regions Automatically** button and the program will define regions as we would expect a user to want them.

Note:

- If the regions defined are not located correctly by the generator, you can delete the generated regions with the Delete button and redraw them manually. See below for more information on this.

To manually draw a region, select the **Create New Regions** button and use your cursor to select two grid intersections which make up the diagonal corners of the region. Similar to wall openings, local coordinates and region dimensions will be displayed in the lower left corner in the Wall Panel Editor. To exit this tool right-click your mouse.

Note:

- Results are only presented for full-height regions. For a gravity analysis this is generally sufficient. If a gravity analysis is required for a non full-height region you must transfer this wall to RISA-3D.
- 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.

Drawing 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:

- A single increment: If you put a single value here (0.5 ft for example), then 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 following view controls:

Delete allows you to delete openings, regions or boundary conditions from the wall panel.

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

Nodes will show any nodes that fall in the plane of the wall panel and allow you to snap to them when drawing regions or openings.

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

Diaphragm Display will turn the display of the Diaphragms on or off, depending on the current setting.

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

Redraw Full Wall View redraws the wall panel to fit within the **Wall Panel Editor** window.

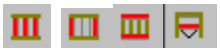
Copy Current View makes a copy of the current view and saves it to the clipboard.

Print Current View prints your current wall panel view.

Note:

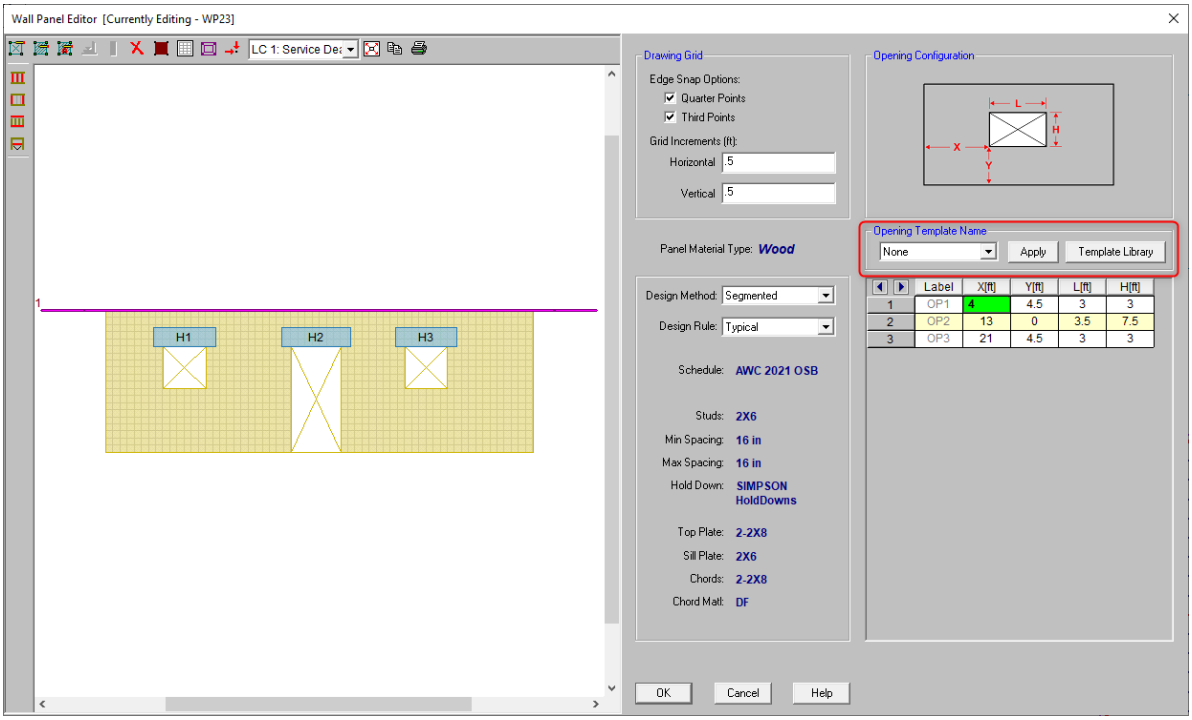
- There are also view controls specific to concrete and masonry. For more information see the Concrete Wall - Design and Masonry Wall - Design topic.

- There are also view controls specific to wood. For more information see the Wood Wall - Design topic.

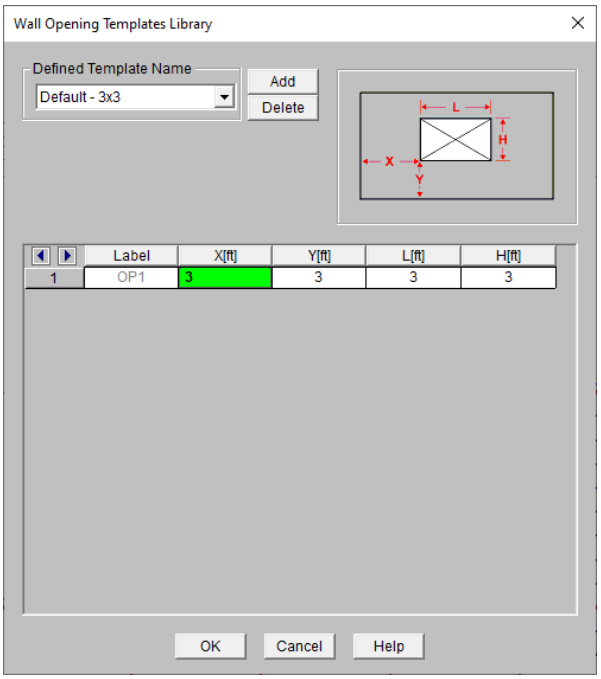


Wall Opening Template

In the Wall Panel Editor, you have the option to save opening configurations and reuse them for other wall panels in the current model as well as others.



Click the **Template Library** button above the wall opening spreadsheet to add, delete, and modify wall opening templates. You can also access the Wall Opening Template Library dialog by clicking the button on the toolbar.



After creating new templates, you can select the template from the drop-down menu in the Wall Panel Editor and click the Apply button to import a template into the current wall panel.

Note:

- If you have existing openings or regions when you apply a template, the program will erase the existing openings and regions and apply the openings defined in the template. A warning message will be displayed when you click Apply indicating this.
- Similar to the Wall Panel Editor spreadsheet, the program will display red text in the spreadsheet if there are openings that overlap. The invalid openings must be fixed in order to save the template.

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.

Wall Panel Load Attribution and Load Transfer

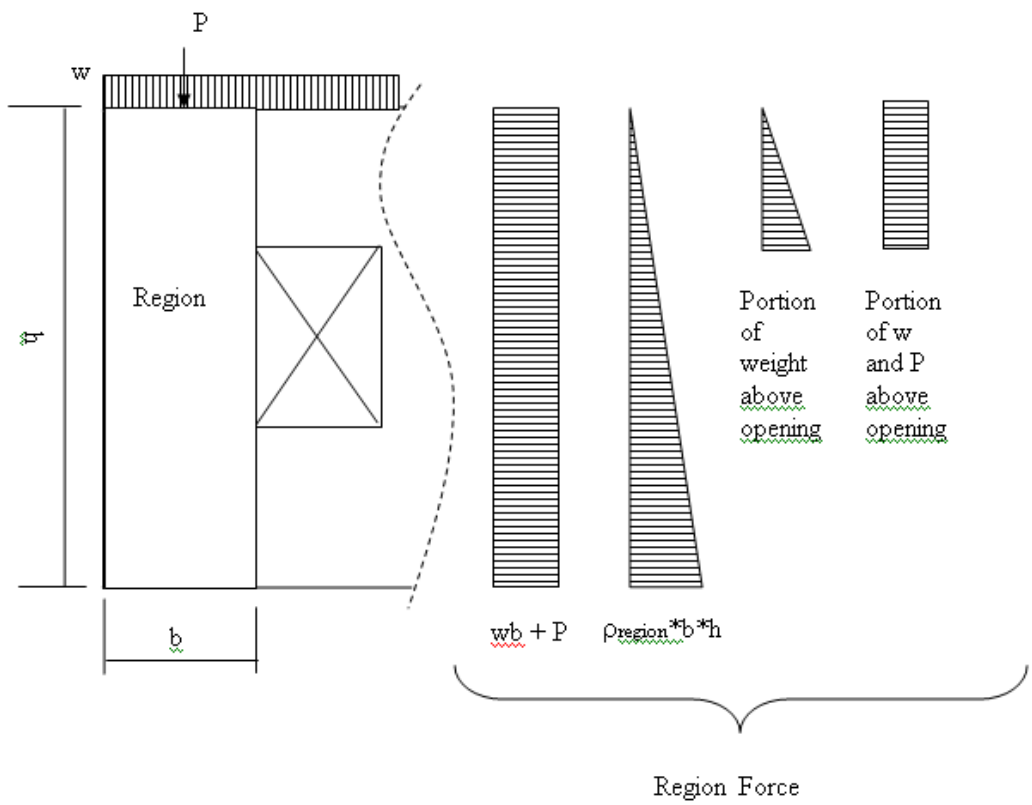
Here is a discussion on how load attribution works for different types of walls.

Wall Panel Load Attribution

Region Force Calculation

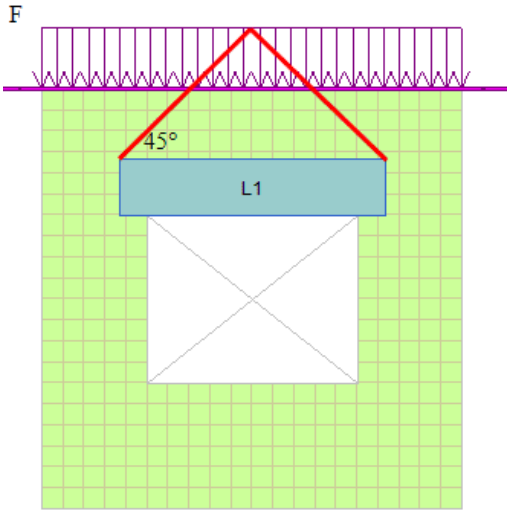
The region force will only be calculated when a region is defined. The region forces are calculated from three types of loads:

- The line load and point load at the floor level directly above the region.
- The transferred load from the lintel which includes part of the self weight above the lintel and part of the load above the opening.
- The weight of the region itself.

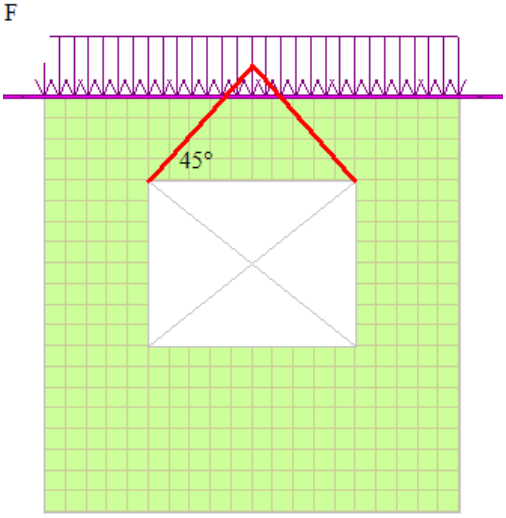


Lintel Force Calculation

If the top of the 45° triangle crosses the floor level, arching effects will not be considered. Line loads above the wall will be applied to the lintel with the same magnitude. The self weight of the rectangular area between the top of the lintel and floor level will also be applied to the lintel.

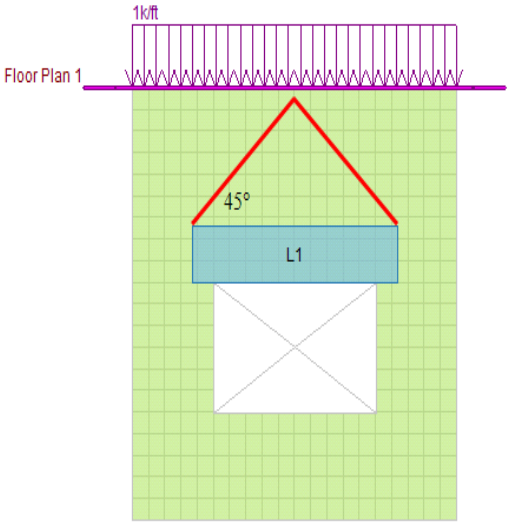


Masonry Wall

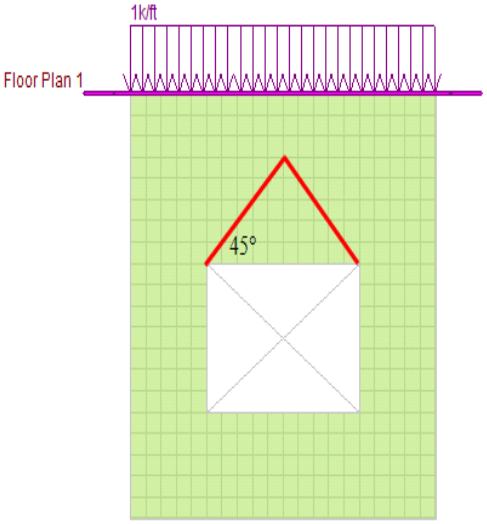


General/Concrete Wall

If the top of the 45° triangle doesn't cross the floor level, arching effects will be considered. Line loads on the floor level will not be contributing to the lintel force. Only the self weight of the 45° triangle will be applied to the lintel.

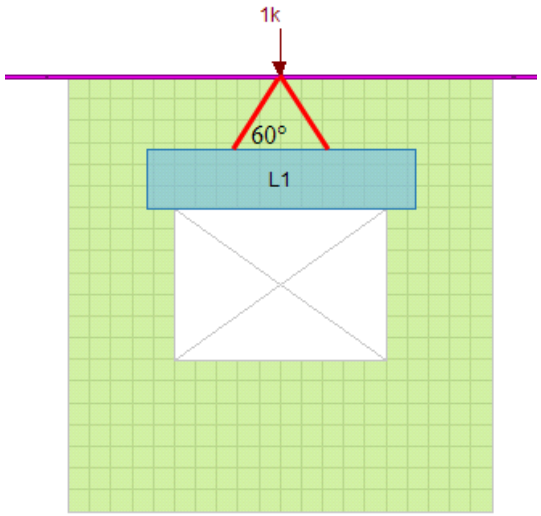


Masonry Wall

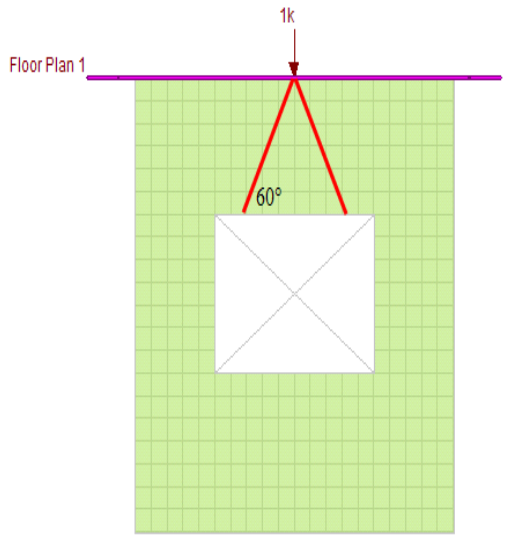


General/Concrete Wall

A point load is always assumed to be dispersed at 60° with the horizontal on each side of the load point.



Masonry Wall

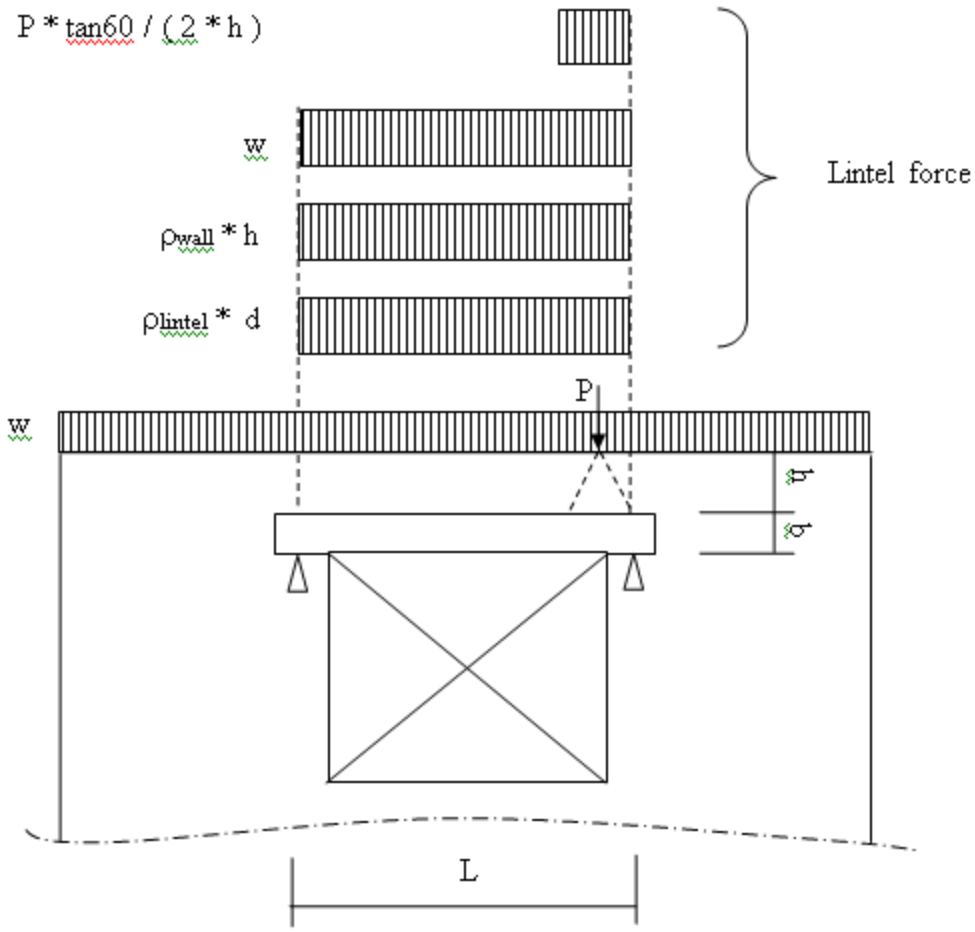


General/Concrete Wall

In summary, if the top of the 45° triangle crosses the top floor, arching effect will not be considered in the lintel force calculation. The load carried by lintel is calculated as:

Lintel Force = Self weight of lintel (masonry material only) + distributed load along the effective span of the lintel + 60° disperse of the point load + Self weight of rectangular area above lintel

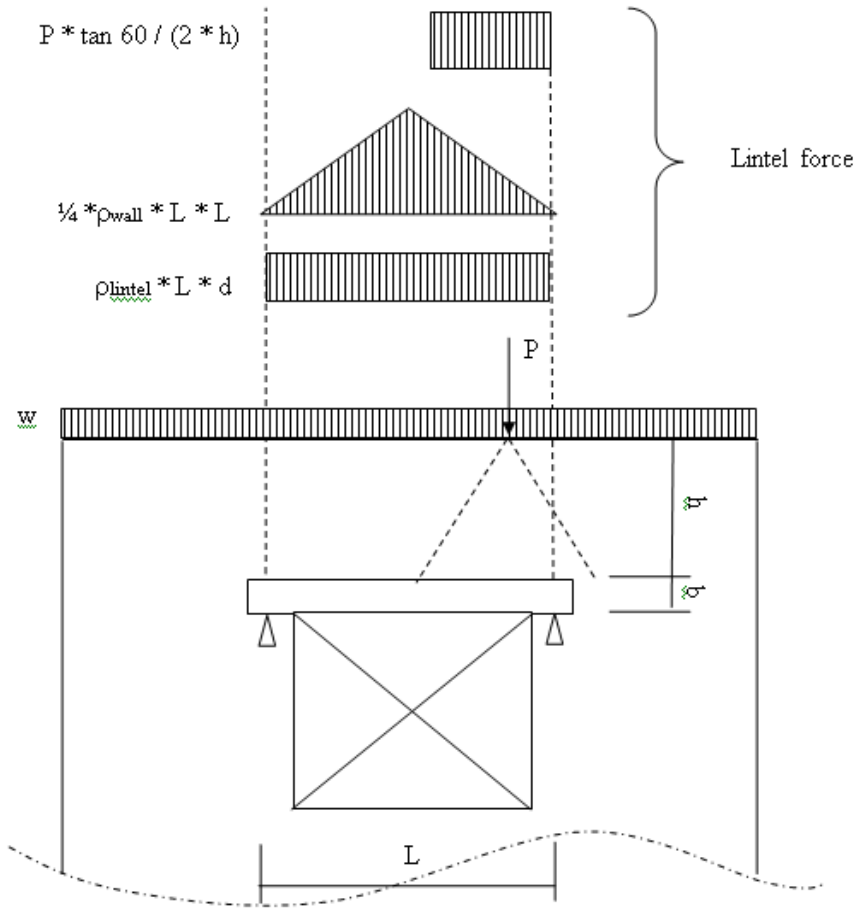
For masonry material, self weights were calculated using the effective span of the lintel. For general and concrete materials, opening width is used in the calculation.



If the top of the 45° triangle cross the top floor, arching effect will not be considered in the lintel force calculation. The load carried by lintel is calculated as

Lintel Force = Self weight of lintel (masonry material) + 60° disperse of the point load + Self weight of the 45° triangle portion of the wall

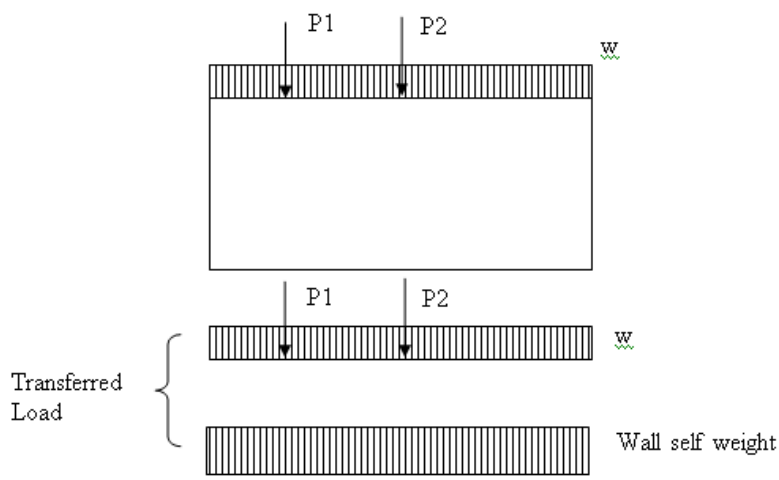
The distributed load on the top of the wall will not be contributing to the lintel force in this case. For masonry material, self weights were calculated using the effective span of the lintel. For general and concrete materials, opening width is used in the calculation.



Wall Panel Load Transfer

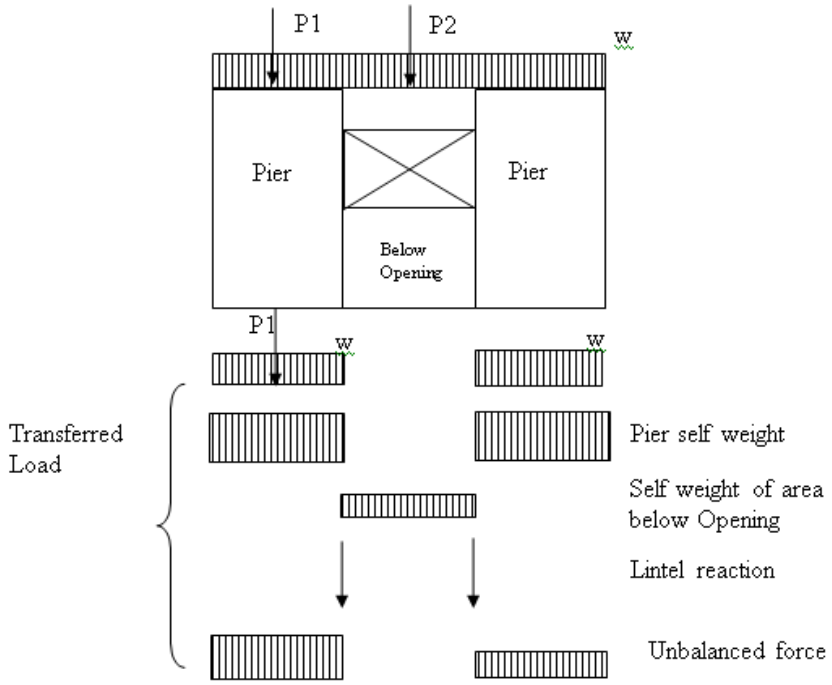
No opening

If there is no opening on the wall, the load on the floor level will be transferred intact to the floor below. The wall self weight will also be added to the load.



With Opening

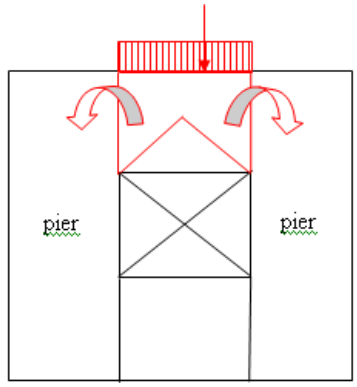
If there are openings on the wall, the load transfer to the floor below is calculated as in the following picture.



The load above the opening will be truncated. The weight above the opening will not be transferred. The lintel reactions will be calculated from the lintel forces attribution the opening/lintel and will be transferred to the floor below.

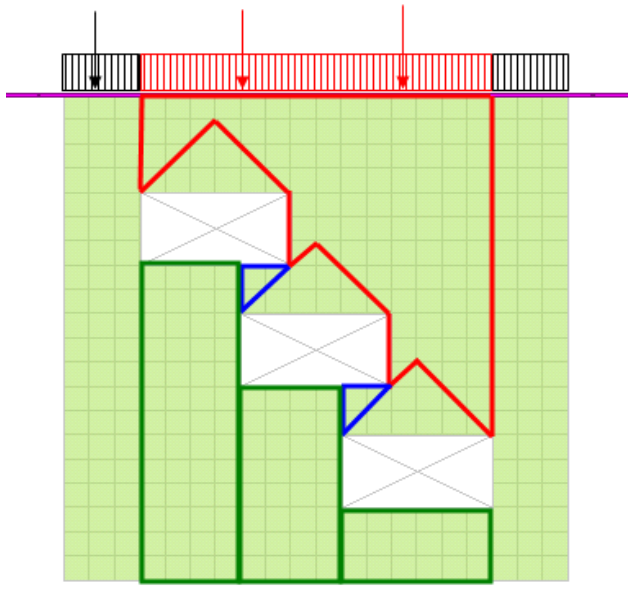
If the 45 degree triangle above the lintel crosses the floor level, there will be no arching effect, the load and self weight right above the opening will be equal to the lintel reaction. In this case the truncated loads and the self weight will be equal to the lintel reaction and there will be no unbalanced forces.

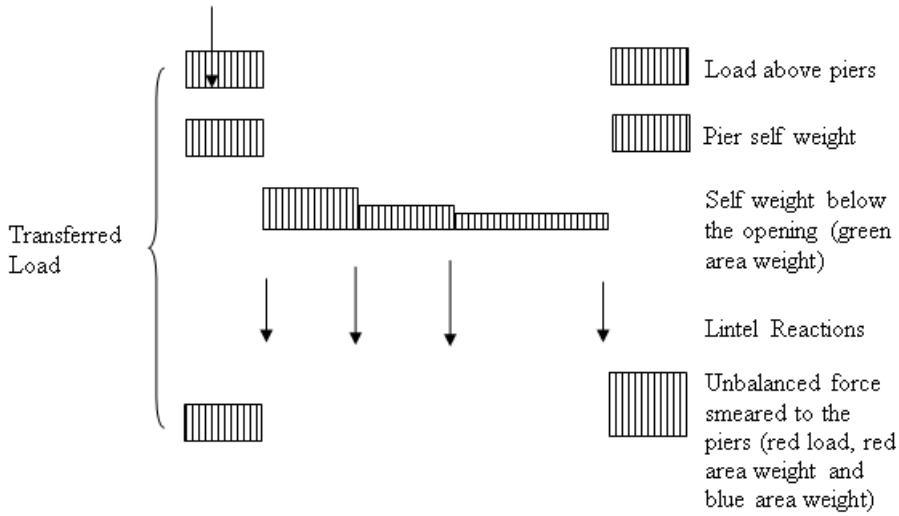
However, if the 45 triangle above the lintel does not cross floor level and arching effect is present, the lintel reaction will be smaller than the total wall self weight and loading. There will be an unbalanced force which equals to the weight of the red area plus the red loads in the following picture. The unbalanced force is smeared into the adjacent piers as additional uniformly distributed load.



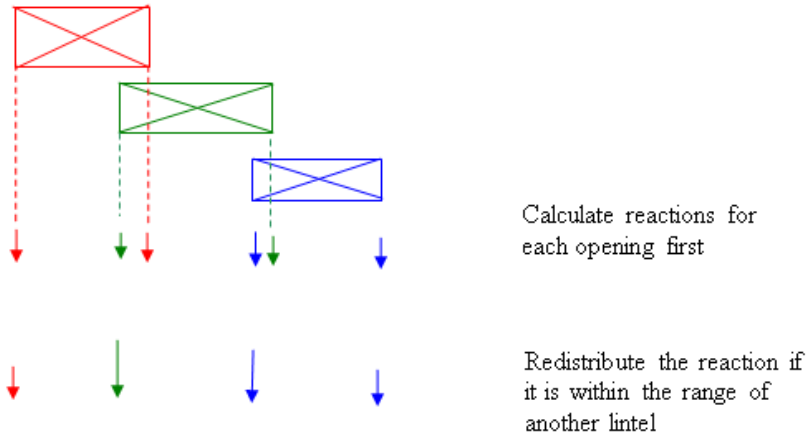
Cascading Opening

More complicated load transfer can happen when openings overlap each other in the horizontal direction as shown in the following picture. Like the wall with a single opening, the self weights of piers and the area below the opening (green box on the picture) will still be transferred as uniformly distributed load. The loads that are right above the pier (load displayed black in the following picture) will still be transferred without any change. If the 45° triangle does not cross the floor level, the wall self weight within the 45° triangle will be transferred as lintel reactions. The wall self weight above the 45° triangle and the load above (the load displayed in red and self weight enclosed in the red and blue lines) will be smeared into the bounding piers. If the 45° triangle crosses the floor level, the self weight above the opening and load above the opening will all be included as lintel reactions.





In the case of cascading opening, the interaction of the lintel reaction is also considered during the calculation. The individual lintel reactions for each opening were first calculated. Then, starting from the top opening, the lintel reactions are recalculated if the lintel support is within the range of other lintels.


Calculate reactions for each opening first
Redistribute the reaction if it is within the range of another lintel

Wall Panels - Results

When the model is solved there are results spreadsheets for Wall Forces and Wall Results. Each tab gives code checks based on the relevant code depending on the material type. These spreadsheets can be used as a summary of all of the panels in your model. To get detailed information about each panel, you can use the Wall Panel Detail Reports.

Concrete Wall Spreadsheet Results

- See below

Concrete Wall Detail Reports

- Wall Summary Detail Report
- In Plane Detail Report

Masonry Wall Spreadsheet Results

- See below

Masonry Detail Reports

- In-Plane / Shear Wall Detail Report
- Lintel Detail Report

Wood Wall Spreadsheet Results

- See below

Wood Wall Detail Reports

- In-Plane or Shear Wall Detail Report

General Wall Detail Report

The general wall detail report shows the material type, height, length, and envelope forces for general walls.

Note

- No detail report is generated for general walls with openings when there is no region defined.

Wall Panel Results

Wall Results Spreadsheet

The **Wall Results Spreadsheet** displays the calculated results for wall elements and may be accessed by selecting **Wall Results** on the **Results** menu. The spreadsheet has three tabs: Masonry Wall, Masonry Lintel, and Wood Wall.

Concrete Wall Tab

The results displayed on this tab are explained below.

Wall Panel ACI 318-05: Concrete Code Checks

Concrete Wall | Masonry Wall | Masonry Lintel | Wood Wall

	Wall Panel	Region	Axial UC	LC	Pn*phi[k/ft]
1	WP1	R2	.0267	2	4214.7713
2		R3	.0128	2	4214.7713
3	WP2	R1	.013	2	3001.1121
4	WP3	R1	.013	2	3001.1121
5	WP4	R2	.0267	2	4214.7713
6		R3	.0128	2	4214.7713
7	WP5	R1	.012	2	4214.7713

The **Wall Panel** column displays the wall panel label.

The **Region** column displays the region label.

The **Axial UC** column displays the axial force code check.

The **LC** column reports the governing Load Combination.

The **Pn*Phi** column reports the allowable axial force for the reported Region on a per foot basis.

Masonry Wall Tab

The results displayed on this tab are explained below.

Wall Panel Masonry Code Checks for Wall Regions (ACI 530-05/08: ASD)

Concrete Wall | Masonry Wall | Masonry Lintel | Wood Wall

	Wall Panel	Region	Axial UC	LC	Fa[ksi]
1	WP6	R1	.0319	2	.3503
2		R4	.0312	2	.3503
3	WP7	R1	.0263	2	.3503

The **Wall Panel** column displays the wall panel label.

The **Region** column displays the region label.

The **Axial UC** column displays the axial force code check.

The **LC** column reports the governing Load Combination.

The **Fa** column reports the allowable axial force for the reported Region.

Masonry Lintel Tab

The **Masonry Lintel** tab displays the various code checks and their associated load cases as well as the allowable forces. The results displayed on this tab are explained below.

Wall Panel Masonry Code Checks for Lintels (ACI 530-05/08: ASD)

Concrete Wall | Masonry Wall | Masonry Lintel | Wood Wall

	Wall Panel	Lintel	Flexure UC	LC	Shear UC	LC	Fm[ksi]	Fs[ksi]	Fvm[ksi]	Fvs[ksi]
1	WP6	L1	.0195	1	.052	1	.5	24	.039	.116

The **WallPanel** column displays the wall panel label.

The **Lintel** column displays the lintel label.

The **Flexure UC** column reports the flexural code check for the lintel.

The **Shear UC** column reports the shear code check for the lintel.

The **LC** columns following the Flexure LC and the Shear LC report the governing load combinations for each of those checks.

The **Fm** column displays the allowable compressive stress of the masonry.

The **Fs** column displays the allowable tensile or compressive stress of the reinforcing steel.

The **Fvm** column displays the allowable shear stress of the masonry.

The **Fvs** column displays the allowable shear stress of the reinforcing steel.

Wood Wall Tab

The **Wood Wall** tab displays the various code checks and their associated load cases as well for the studs and the chords. The results displayed on this tab are explained below.

Wall Results, Wood Wall Panel

Concrete Wall | Masonry Wall | Masonry Lintel | Wood Wall

	Wall Panel	Region	Stud Size	Stud Spacing[...	Axial Check	Gov LC
1	WP8	R1	2X6	16	.0753	2
2		R4	2X6	16	.1715	2
3	WP9	R1	2X6	16	.1034	2

The **Region** column reports which region the results are being shown for. See the notes below for more information.

The **Stud Size**reports the studs for the designed wall.

The **Stud Spacing** displays the spacing for the designed studs.

The **Axial Check** reports the code checks for the wood wall.

The **Gov LC** reports the governing load combinations for the code checks.

Note:

- For wood wall panels designed as Perforated or Force Transfer Around Opening there is no design and an NC (no calculation) will be shown. If you intend to transfer to RISA-3D this is not a problem, as the design will be done there.
- For wood wall panels designed as Segmented, the regions above and below openings will not be designed and an NC (no calculation) will be shown. The segmented method essentially disregards these portions of the wall in design.

Wood Wall Header Results

Wood Wall Panel Header Code Checks (AF&PA NDS-05/08: ASD)

Concrete In | Concrete Out | Masonry In | Masonry Out | Masonry Lintel | Wood Wall Axial | Wood Wall In-Plane | Wood Header

	Wall Panel	Header La...	Design Ru...	Size	Bend UC	Shear UC	Gov LC
1	WP1	H1	Typical	6x8	.626	.318	1 (bending)
2	WP2	H1	Typical	6x8	.417	.212	1 (bending)

The **Header Label** gives 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.

The **Design Rule** gives the rule that is defining the size and material.

The **Size** gives the size from the Design Rule.

The **Bend UC** value is a bending code check ratio. This is given for the load combination from the Gov LC column.

The **Shear UC** value is a shear code check ratio. This is given for the load combination from the Gov LC column.

The **Gov LC** gives 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:

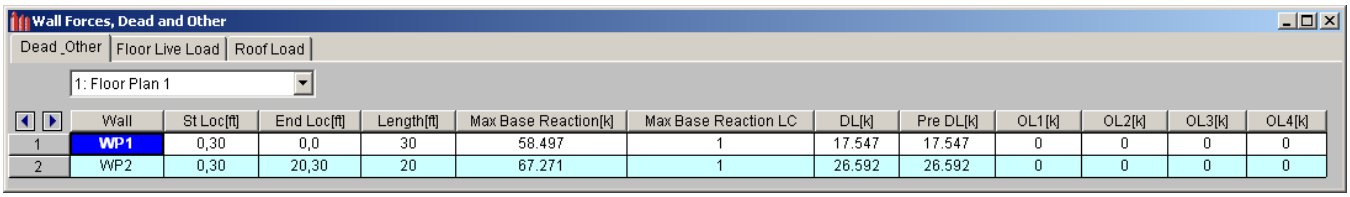
- The **Bend UC** and **Shear UC** are always given for the SAME load combination. Whichever load combination produced the highest code check for either bending or shear will have its results shown in these columns.

Wall Forces Spreadsheet

The **Wall Forces Spreadsheet** displays the calculated results for wall panel elements and may be accessed by selecting **Wall Results** on the **Results** menu. The spreadsheet has three tabs: **Dead & Other, Floor Live Load,** and **Roof Load**.

Dead & Other Tab

The **Dead & Other** tab displays the axial forces in the wall due to dead and other load categories. The results displayed on this tab are explained below.



	Wall	St Loc[ft]	End Loc[ft]	Length[ft]	Max Base Reaction[k]	Max Base Reaction LC	DL[k]	Pre DL[k]	OL1[k]	OL2[k]	OL3[k]	OL4[k]
1	WP1	0,30	0,0	30	58.497	1	17.547	17.547	0	0	0	0
2	WP2	0,30	20,30	20	67.271	1	26.592	26.592	0	0	0	0

The **Wall** column displays the wall label.

The **St Loc** and **End Loc** columns display the plan coordinates for the starting and ending points of each wall.

The **Length** column displays the length of the wall.

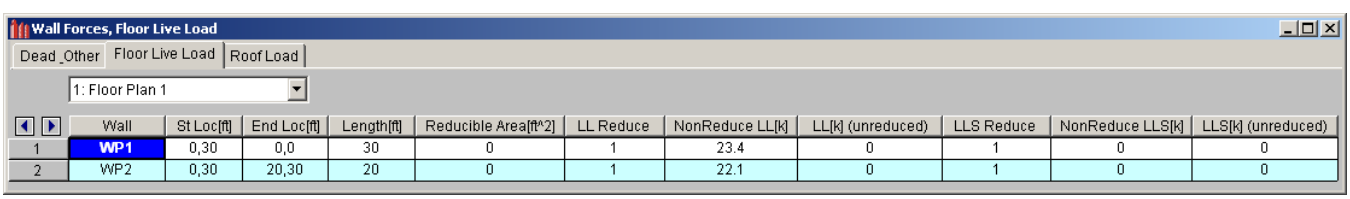
The **Max Base Reaction** and **Max Base Reaction LC** columns display the maximum base reaction and its governing load combination.

PreDL and **DL** give the *total* axial forces in the wall due to the dead load categories.

The **OL1**, **OL2**, **OL3**, and **OL4** columns give the total axial force in the wall due to the 'Other' load categories.

Floor Live Load Tab

The **Floor Live Load** tab displays the location and live load reduction information for each wall panel. The results displayed on this tab are explained below.



	Wall	St Loc[ft]	End Loc[ft]	Length[ft]	Reducible Area[ft^2]	LL Reduce	NonReduce LL[k]	LL[k] (unreduced)	LLS Reduce	NonReduce LLS[k]	LLS[k] (unreduced)
1	WP1	0,30	0,0	30	0	1	23.4	0	1	0	0
2	WP2	0,30	20,30	20	0	1	22.1	0	1	0	0

The **St Loc** and **End Loc** columns display the plan coordinates for the starting and ending points of each wall.

The **Length** column displays the length of the wall.

The **Reducible Area** column lists the total tributary area to be considered for live load reduction for each wall.

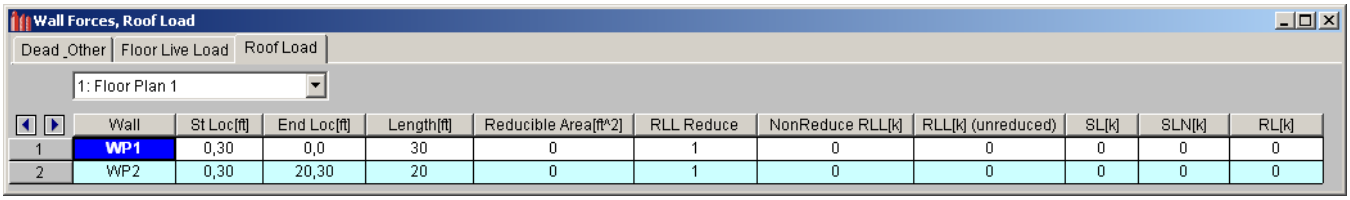
The **LL Reduce** and **LLS Reduce** columns display the live load reduction factors for the various live load categories.

The **NonReduce LL** columns give the total axial force in the wall due to the non-reducible live load cases. The included load cases are LL and LLS.

The **LL Unreduced** columns give the *total* *unreduced* axial forces in the wall due to the reducible live load cases. The included load cases are LL-Reduce and LLS-Reduce. It is important to note that the forces listed in this column have not been reduced by the load reduction factors.

Roof Load Tab

The **Roof Load** tab displays the location and roof live load reduction information for each wall panel. The results displayed on this tab are explained below.



	Wall	St Loc[ft]	End Loc[ft]	Length[ft]	Reducible Area[ft^2]	RLL Reduce	NonReduce RLL[k]	RLL[k] (unreduced)	SL[k]	SLN[k]	RL[k]
1	WP1	0,30	0,0	30	0	1	0	0	0	0	0
2	WP2	0,30	20,30	20	0	1	0	0	0	0	0

The **St Loc** and **End Loc** columns display the plan coordinates for the starting and ending points of each wall.

The **Length** column displays the length of the wall.

The **Reducible Area** column lists the total tributary area to be considered for live load reduction for each wall.

The **RLL Reduce** column displays the live load reduction factor.

The **NonReduce RLL** column gives the total axial force in the wall due to the non-reducible roof live load case.

The **RLL Unreduced** column gives the *total* *unreduced* axial forces in the wall due to the reducible roof live load cases. It is important to note that the forces listed in this column have not been reduced by the load reduction factors

The **SL, SLN,** and **RL** columns display the total axial force in the wall due to these load cases.

CFS Wall - 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.

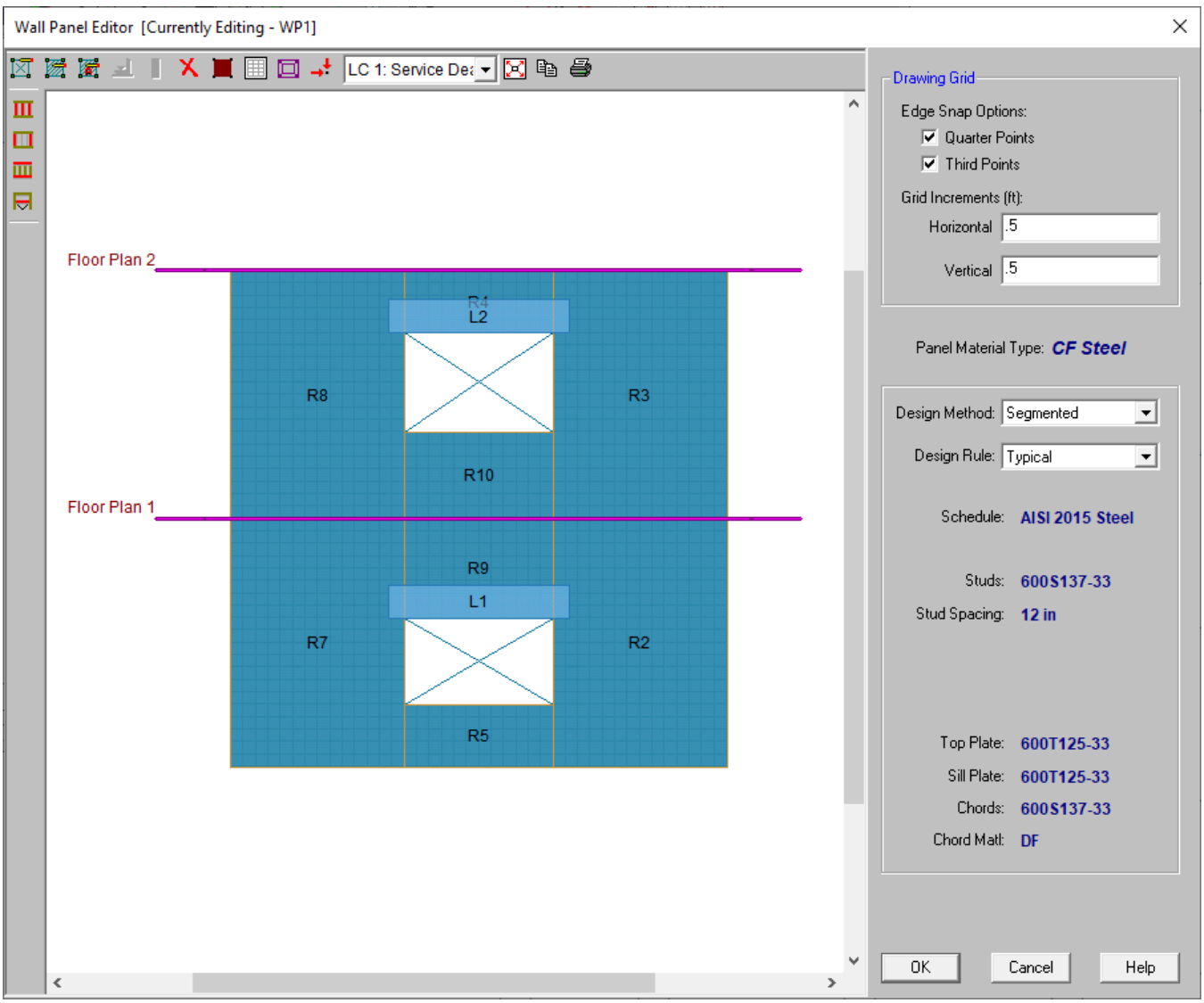
For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **CFS Walls**.

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
- The AISI S240-20 CFS Structural Framing & AISI S400-20 Seismic Design of CFS Structural Systems

CFS Wall Input

The **Wall Panel Editor** gives some specific information and options for modeling/analysis of CFS walls.



CFS View Controls

Toggle Wall Studs Display allows you to turn the display of the studs on and off.

Toggle Wall Chords Display allows you to turn the display of wall panel region chords on and off.

Toggle Top/Sill Plate Display allows you to turn the display of the top/sill plates on and off.

Toggle Opening Headers Display allows you to turn the display of headers on and off.

Design Rules

You must set up design rules for the stud/chord sizes, as well as make database selection for shear panels. 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.

Creating Openings in CFS Walls with Headers

Within the **Wall Panel Editor** tab, you have the option of adding rectangular openings to wood wall panels. To draw an opening, select the **Create New Openings** button and then select two nodes or grid intersections which make up the two diagonal corners of your opening. When an opening is drawn, a header beam is automatically created above the opening. Currently header design is not available for CFS walls

Label - This defines the name of this header and shows up in the results output for this header.

Wall Design Rule - This is the default setting here and is the default header information for all headers in the wall.

Custom - This allows you to define a header size and material that is different from the Wall Design Rule. You would only set a custom header if you have multiple headers in the same wall that are different sizes.

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 within the **Wall Panel Editor**.

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 **Add New Wood Hold Downs** button .

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

Straps represent the connection of the current wall panel to a wall panel below. To add straps to the base of your wall, first select the Add New Wood Straps button .

Straps also can only be added after regions are created and can only 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.
- Hold-downs and straps can only be added in RISA-3D.
- 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.

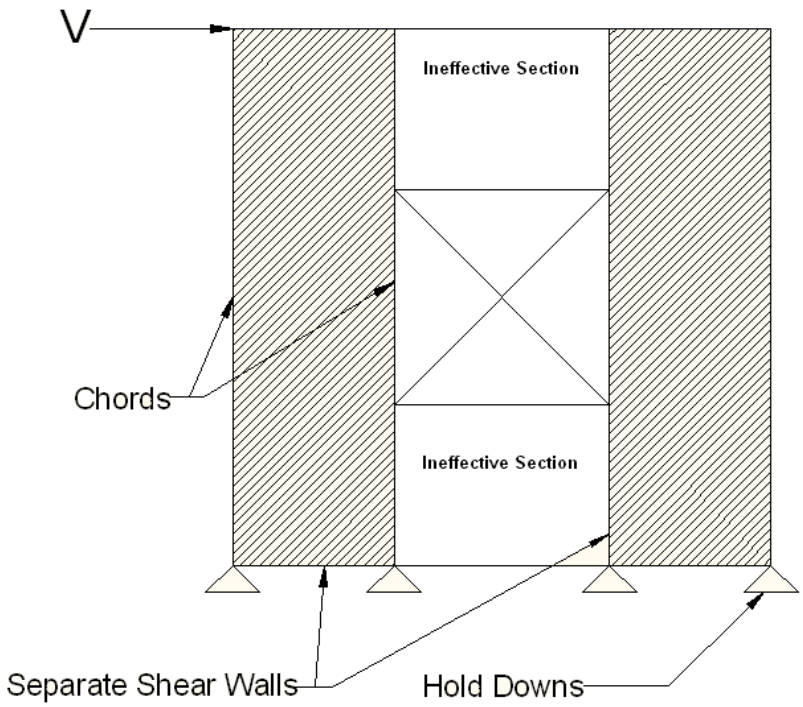
Type I - Segmented Method

The Type I - Segmented methodology is defined in Section B5.2 of the *AISI S240-15/20 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 S240-15/20 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)

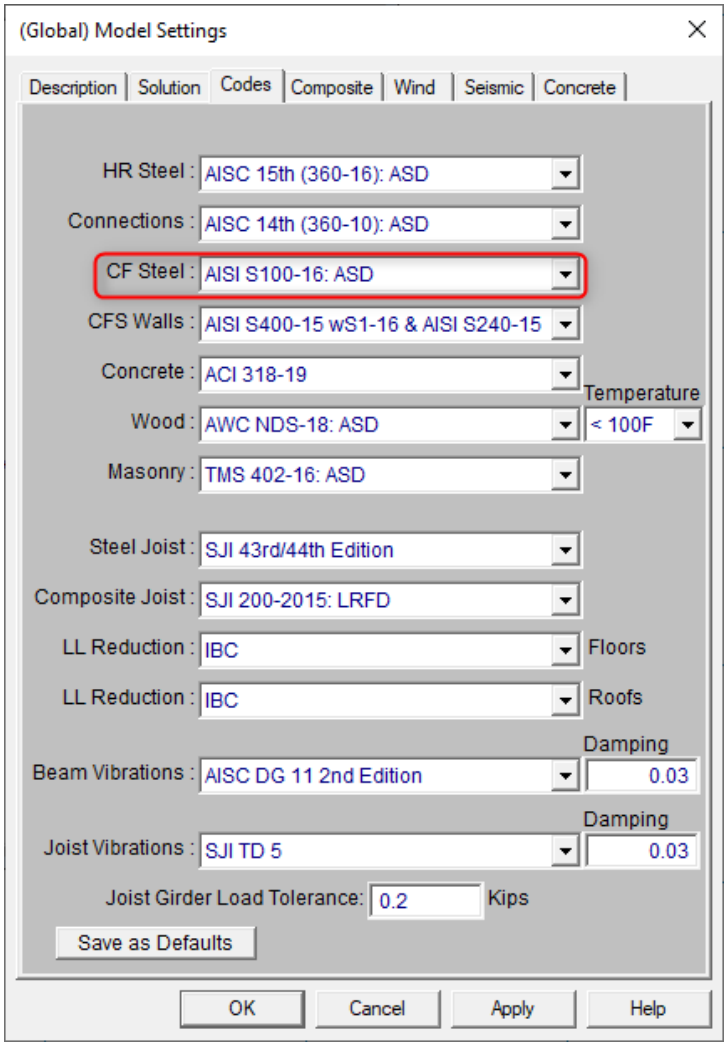
General Program Functionality and Limitations

Assigning Code and Design Methodology

CFS wall designs will be based on the *AISI S240-15/20 North American Standard for Cold-Formed Steel Structural Framing* and *S400-15/20 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 Global 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.

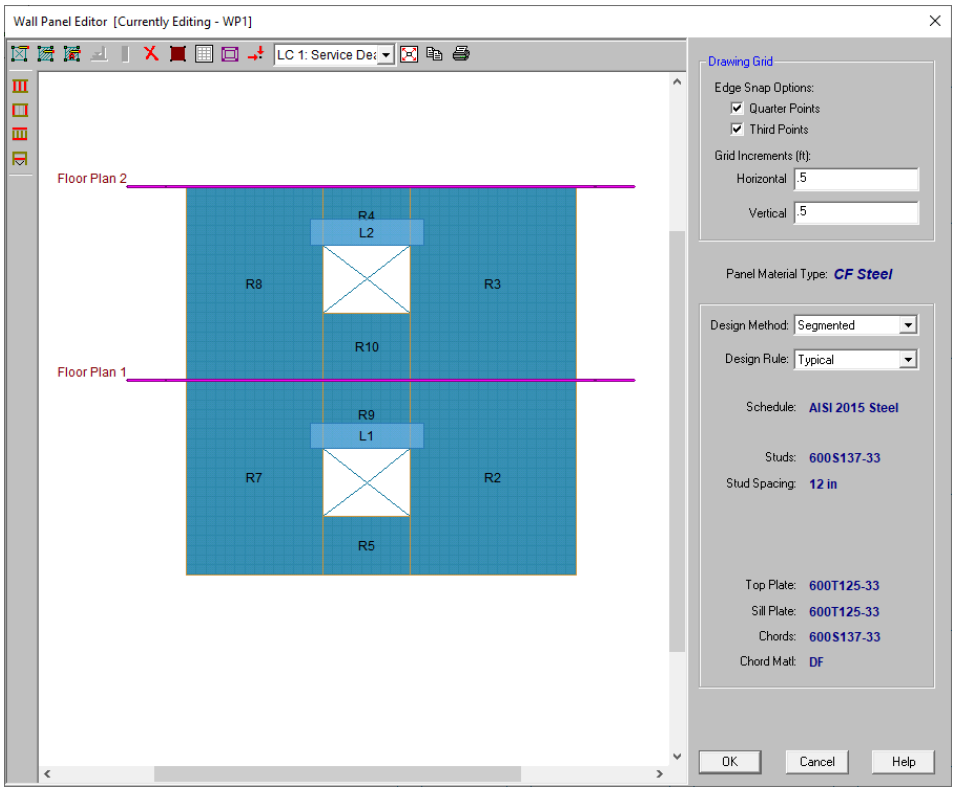


RISAFloor and RISA-3D Interaction

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.

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.

Panel Optimization

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 S240-15/20 and AISI S400-15/20.

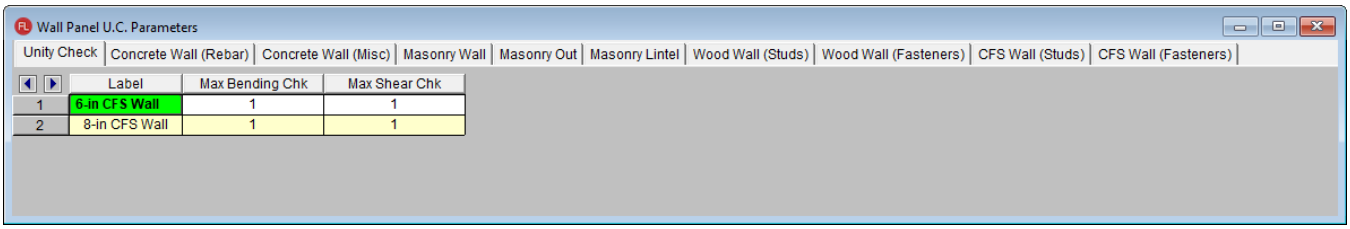
Note:

- 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 Wall - 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



	Label	Max Bending Chk	Max Shear Chk
1	6-in CFS Wall	1	1
2	8-in CFS Wall	1	1

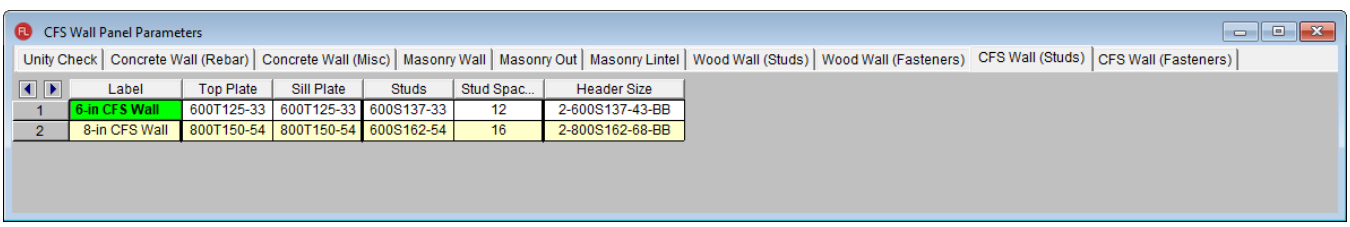
Max Bending Chk

This will define 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

This will define the shear panel selection maximum code check value. Typically this will always be 1.0, but this 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.

CFS Wall (Studs)



	Label	Top Plate	Sill Plate	Studs	Stud Spac...	Header Size
1	6-in CFS Wall	600T125-33	600T125-33	600S137-33	12	2-600S137-43-BB
2	8-in CFS Wall	800T150-54	800T150-54	600S162-54	16	2-800S162-68-BB

Top Plate

Use the **Top Plate** column to specify the member to be used as a top plate for your wall. A top plate is a member that runs continuously along the top of the wall studs.

Sill Plate

Use the **Sill Plate** column to specify the member to be used as a sill plate for your wall. A sill plate is a member that runs continuously along the bottom of the wall studs.

Studs

Use the **Studs** column to specify the member to be used for studs in your wall. Studs are vertical members in the wall, attached to the sill plate at the bottom and the top plate at the top. Note that you can use back-to-back or toe-to-toe options as well.

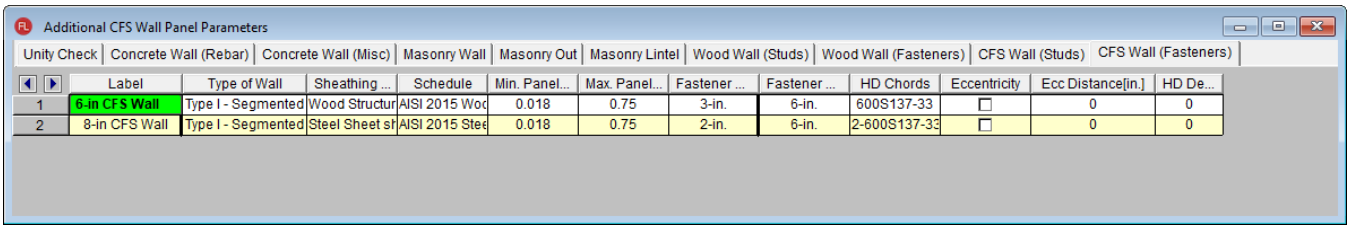
Stud Space

Use the **Stud Space** column to specify the spacing between the studs within the wall.

Header Size

This defines the default header size for all openings. Note that the program will not provide a design for CFS headers at this time.

Design Rules - CFS Wall (Fasteners)



	Label	Type of Wall	Sheathing ...	Schedule	Min. Panel...	Max. Panel...	Fastener ...	Fastener ...	HD Chords	Eccentricity	Ecc Distance[in.]	HD De...
1	6-in CFS Wall	Type I - Segmented	Wood Structur	AISI 2015 Woc	0.018	0.75	3-in.	6-in.	600S137-33	☐	0	0
2	8-in CFS Wall	Type I - Segmented	Steel Sheet st	AISI 2015 Stee	0.018	0.75	2-in.	6-in.	2-600S137-33	☐	0	0

Type of Wall

This option determines the design methodology of the CFS wall panel. Currently CFS walls can only be designed using the Type I - Segmented method.

Sheathing Material

You can select which sheathing material you would like to be used for lateral design optimization. The specific sheathing material that you select will automatically populate the corresponding sheathing database in the adjacent Schedule column.

Schedule

You can select the sheathing database and Panel Group you would like to use for design optimization. By selecting the **Select Individual Panel** option, an individual panel type may be assigned. 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

Design Method

This column allows you to choose the method in which the lateral capacity will be determined by the program. If you choose the Table option, the program will reference 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. For more information on this see the CFS Wall - Design topic.

Min/Max Panel Thick

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

Fastener Min Edge Space Max/Min

These values set minimums and maximums for the spacing of the nails that fasten the sheathing to the boundary members (top plate, sill plate, hold down chords). Note that a 12" spacing is assumed for all field nailing (nails fastening the sheathing to the internal studs).

HD Chords

You can choose what member size you would like to use for the Hold Down Chords (Posts) at both ends of the wall panel.

Eccentricity

This checkbox controls whether the eccentricity distance of the hold-down (input into the Ecc Dist column) will be 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

You can use this column to 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

You can use this column to input the manufacturer hold down deflection which will be used in the reported shear wall design deflection calculation.

CFS Wall 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

See Wall Panels - Results for this topic.

Wall Results, CFS Wall Panel

Concrete Wall | Masonry Wall | Masonry Lintel | Wood Wall | Wood Header | CFS Wall

	Wall Panel	Region	Stud Size	Stud Spacing[in]	Axial Check	Gov LC
1	WP1	R2	800S162-54	16	0.643	2
2		R3	800S162-54	16	0.32	2
3		R8	800S162-54	16	0.32	2
4		R7	800S162-54	16	0.643	2
5	WP2	R2	800S162-54	16	0.99	2
6		R3	800S162-54	16	0.505	2
7	WP3	R2	800S162-54	16	0.464	2
8		R3	800S162-54	16	0.232	2
9	WP4	R2	800S162-54	16	0.99	2
10		R3	800S162-54	16	0.505	2

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

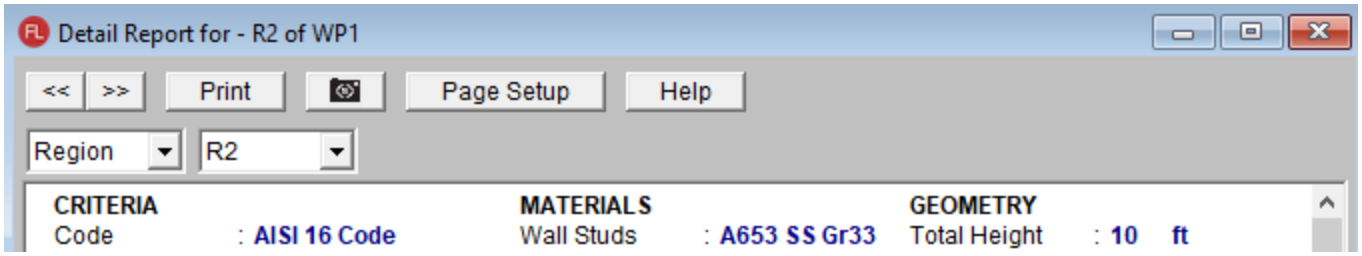
Once you have a solved model, the detail reports become available. They are accessible in two ways:

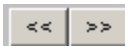
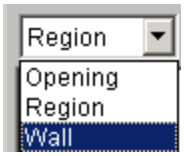
1. If you have the Wall Results spreadsheet open, there will be a button at the top of the screen: Detail Report for Current Item. This will open up the detail report window.
2. If you are in a graphic view of your model, there is a Detail button that will open up the detail report window.

Note:

- Detail report information is not available for an envelope solution.

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



-  - Allows you to click between the different wall panels in your model.
- - This 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.
- 

- This dropdown list allows you to select between the three different parts of the wood wall panel detail report. Below we will explain the importance of each of these sections.
-

- This dropdown list allows you to select between different regions or headers defined within the individual wall panel.

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

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

GENERAL

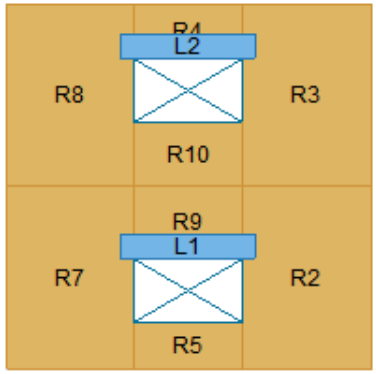
Code	:	AISI 16 Code
Design Method	:	Segmented
Wall Material	:	A653 SS Gr33
Function	:	Lateral

GEOMETRY

Total Height	:	20	ft
Total Length	:	20	ft
Parapet Height	:	0	ft
Wall H/W Ratio	:	1.00	
K	:	1.00	

MATERIALS

Description	Material	Size
Top Pl	A653 SS Gr33	800T150-54
Sill	A653 SS Gr33	800T150-54
Wall Stud	A653 SS Gr33	800S162-54
Chord	A653 SS Gr33	2-600S137-33-BB



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

Design Details

DESIGN DETAILS

REGION INFORMATION

Full-Height Region Label	Stud Required Cap	Stud Provided Cap	Stud UC	Stud LC
R2	0.637	0.991	0.643	2
R3	0.318	0.991	0.32	2
R8	0.318	0.991	0.32	2
R7	0.637	0.991	0.643	2

The **Region Information** gives the tabulated results of all of the full-height regions in your wall.

The **Opening Information** gives the label, size, shear and bending code checks and the governing LC for each header in the wall.

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

CRITERIA		MATERIALS		GEOMETRY		
Code	: AISI 16 Code	Wall Studs	: A653 SS Gr33	Total Height	: 10	ft
		Stud Size	: 800S162-54	Total Length	: 7	ft
Wall Material	: A653 SS Gr33	Chord Material	: A653 SS Gr33	Region H/W	: 1.43	
		Chord Size	: 2-600S137-33-BB	Cap. Adj. (2w/h)	: 1.00	
				Aspect Ratio	: 1.00	
				Gov. H/W Cap.	: 1.00	
				Stud Spacing	: 16	in
				K	: 1.00	

Below is the input echo portion of the detail report.

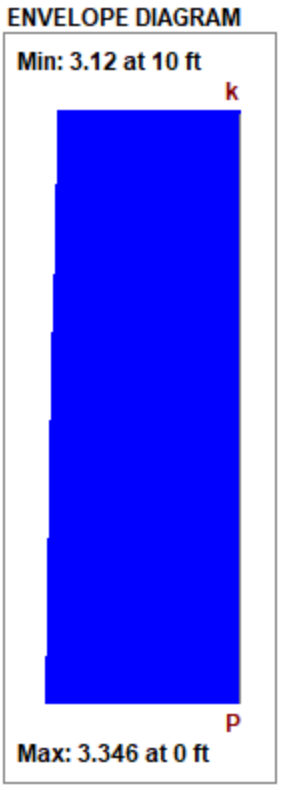
Criteria	Description
Code	Gives the code used to design your wall panel.
Wall Material	Specifies the CFS type assigned to the entire wall

Materials	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	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 S240-15/20 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.
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 the wall.



DESIGN SUMMARY

STUDS

Required Cap	: 0.637k	
Provided Cap	: 0.991	k
Ratio	: 0.643	
Governing LC	: 2	

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.

Studs

The provided capacities of these members are calculated using the standard provisions for tension/compression members. These members are assumed to be fully braced in the weak axis, and unbraced in the strong axis. For more information on the chord force calculations, see the CFS Wall - Design 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.

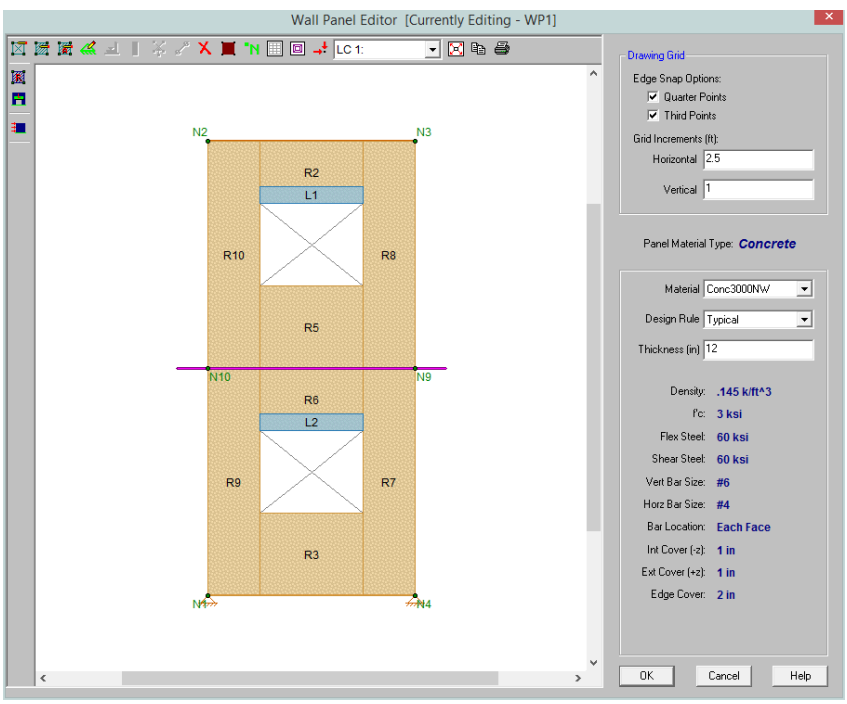
Note:

- 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

Double-click on the wall from the model view to open the **Wall Panel Editor.** This dialog displays input information such as Material and Design Rule, as well as gives viewing options for Region and reinforcement display (after solution). Within this dialog you also have control over the application of continuous boundary conditions.



Concrete Wall View Controls

Concrete wall panels have the following view controls:

- **Toggle Region** display allows you to turn the display of regions on or off.
- **Toggle Lintel** display allows you to turn the display of lintels on or off.

- **Toggle Reinforcement** display allows you to turn the display of reinforcement on or off after you have solved your model.
- The **Drawing Grid** allows you to modify your snap options, drawing grid in the wall panel editor.
- The **Material** and **Design Rule** drop down lists allow you to modify these parameters in the wall panel editor.

Concrete Wall Regions

Concrete walls depend on Regions for results presentation. The program automatically creates these regions at solution time. If you have a wall panel with no floors 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 floors 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 floors and around the openings, giving a different reinforcement design for each region.

Within each region, the program will optimize the spacing of bars for strength, spacing and minimum reinforcement considerations of the wall.

From within the **Wall Panel Editor**, you have the option of creating rectangular regions within the concrete wall panel. Regions are used to define reinforcement in different parts of the wall. Each region will be assigned a uniform reinforcement, which may be different than the reinforcement in other parts of the same wall (unless you are using Group Story option from Wall Design Rules).

If no regions have been drawn on a wall then they will be automatically generated when a solution is performed. To automatically generate regions prior to running a solution, click the **Generate Wall Regions Automatically** button .

To manually draw regions, select the **Create New Regions** button and use your cursor to select two nodes or grid intersections which make up the diagonal corners of the region. To exit this tool right-click your mouse.

Concrete Design Considerations

ACI 318-14/19 Reinforcement Design

The program will design the reinforcement spacing for you. For this design reinforcement spacing, rho, and strength requirements are considered for design. If specific reinforcement is defined in the Wall Design Rules spreadsheet then it may be possible for the reinforcement design to not meet code requirements.

Sections 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 Vu exceeds 0.5*ϕ*Vc.

ACI 318-11 Reinforcement Design

The program will design the reinforcement spacing for you. For this design reinforcement spacing, rho, and strength requirements are considered for design. If specific reinforcement is defined in the Wall Design Rules spreadsheet then it may be possible for the reinforcement design to not meet code requirements.

Sections 7.6, 14.3 and 11.9 all have provisions regarding min/max spacing, required reinforcement ratios, and proper proportioning of wall reinforcement. 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 Vu exceeds 0.5*ϕ*Vc (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, 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.

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 lw / 25 or hu / 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.

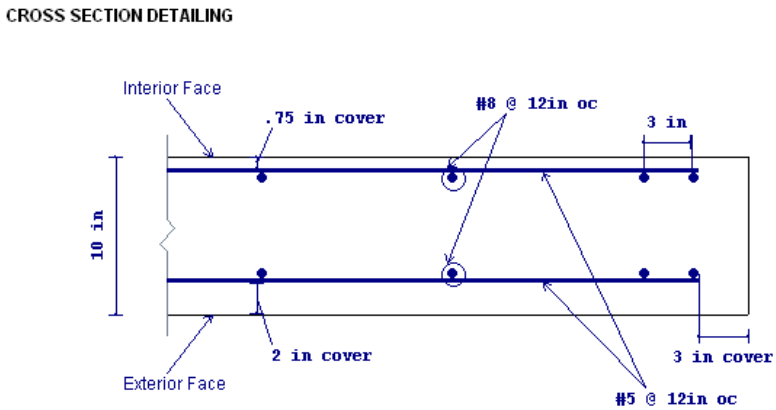
The reinforcement layout algorithm works as follows (picture looking down on a cross section of wall):

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.



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:

- In RISAFloor only axial forces are considered in walls. Because slenderness is considered in the specification by increasing the design moment, there are no slenderness considerations in RISAFloor. To consider these effects take the concrete wall into RISA-3D via the Director tool.

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 ρw 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 ρw 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.

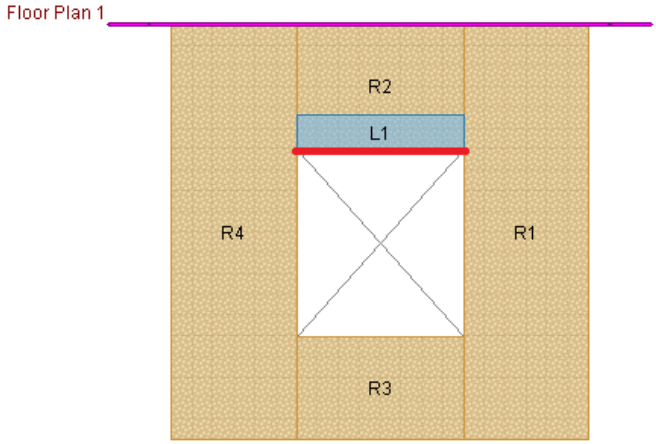
When CSA A23.3-14 are selected, V_c is calculated in the same way as In-Plane shear section.

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 RISAFloor the lintel forces are taken from a point that is directly at the top of opening location (see the red line below). These forces are then used to calculate the shears and moments along the length of the lintel. These analysis results are shown in the detail report.



Thus, every lintel (regardless of how regions are laid out) will give the shear and moment diagrams in the detail report.

Load Attribution

See the Wall Panel Load Attribution topic for more information.

Concrete Wall Modeling Considerations

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.

RISAFloor/RISA-3D Design Enveloping

If you are using RISAFloor and RISA-3D in tandem to do gravity and lateral design, then the program envelopes the results of both. Thus, RISAFloor will never increase the spacing of bars larger than what was required in RISA-3D and vice versa. Also, when moving between programs, the reinforcement spacing that controls will be carried on into the other program.

For example, let's assume RISAFloor required vertical bars that are #6 @ 12" oc. We then use the **Director** to take the model to RISA-3D. In RISA-3D, the required vertical bars are #6 @ 8" oc. Now, if we take the model back to RISAFloor and solve again, the #6 @ 8" oc spacing is brought over to RISAFloor and code checks are now based on this spacing.

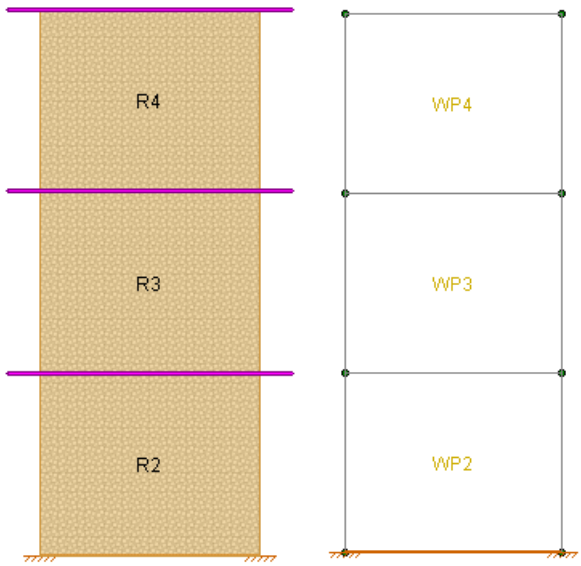
Note:

- When going back and forth between RISAFloor and RISA-3D, you must solve the model in each of the programs to capture this enveloping. If you do not re-solve, then you are then seeing the results from the previous time you solved in that program without updating for the enveloping.

Modeling Multi-Story Shear Walls

For multi-story walls that have floors intersecting, separate regions will be drawn above and below the floors (see Figure 1). The design of each of these regions can be different. This, however, is only true of the spacing of reinforcement. The bar size must be equal for the full-height of the wall, because the Design Rule is for the entire wall and all regions.

If you want to change bar size over the height of the wall, simply create separate wall panels and stack them on top of one another. In this way you can define different Design Rules and thus different reinforcement bar sizes up and down the wall (see Figure 2).



For the stacked regions model where the intention is to drop off bars as you work your way up the wall, lets give an example. Let's say that at the base of the wall you have bars at a 4" o.c. spacing. At some point you want to drop off bars to create an 8" o.c. spacing.

All you need to do here is set your Design Rules such that the Min Vert Bar Spacing is set to 4", the Max Vert Bar Spacing is set to 8" and the Spacing Increment is set to 4".

Wall Cover Dimensions and Local Axes

The program is able to consider different reinforcement cover dimensions for each face of the wall. Because of this the orientation of the wall is important. In the program the **Exterior Face of the wall is oriented in the +z local axis direction**. The **Interior Face of the walls is oriented in the -z local axis direction**.

Note:

- In RISAFloor the wall local axis is irrelevant as we are only doing axial checks.
- If the cover on both faces is identical then the local axis orientation is irrelevant.

A wall drawn in a clock-wise fashion will have its local axis pointed in the positive direction. A wall drawn in a counter-clockwise fashion will have its local axis point in the negative direction.

If the wall local axis is facing the wrong direction then use the Local Axis Flip from the Modify Walls dialog to correct it.

When bringing a model from RISAFloor to RISA-3D then you want to think about how you are drawing in RISAFloor so that the local axes come in properly in RISA-3D. You will want to draw the walls in in RISAFloor in a counter-clockwise manner to get the local z axes to point outward.

Wall Panels Drawn in RISAFloor in Counter-Clockwise Fashion

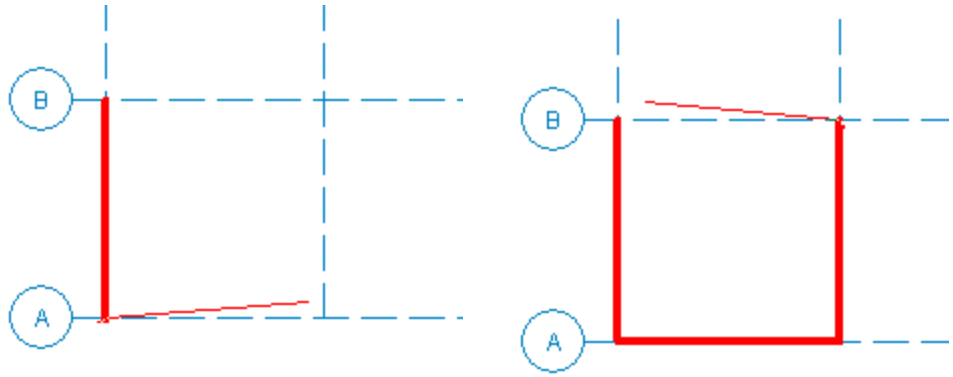
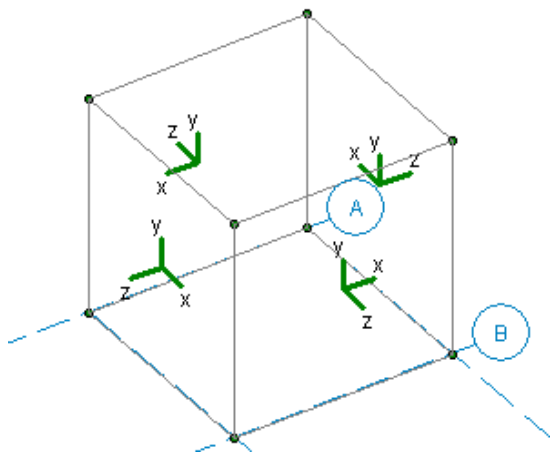


Image of Local Axes in RISA-3D



You can go to Model Display Options - Panels tab to view the local axes for the wall.

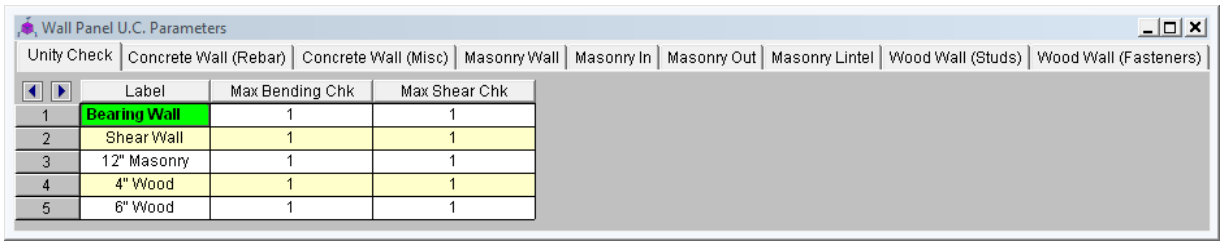
Limitations

- 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.
- 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



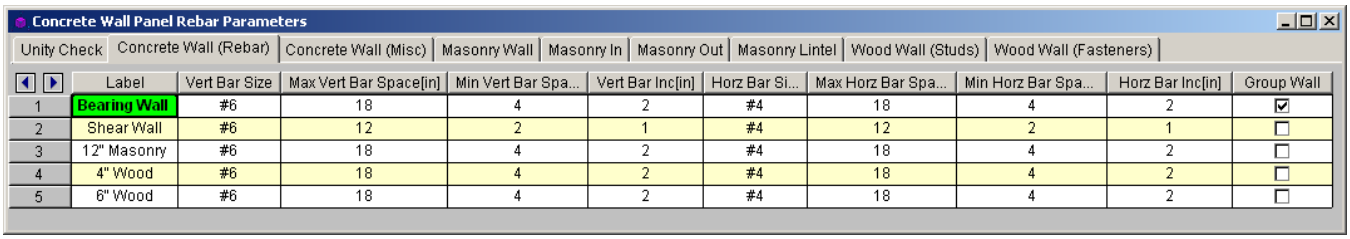
	Label	Max Bending Chk	Max Shear Chk
1	Bearing Wall	1	1
2	Shear Wall	1	1
3	12" Masonry	1	1
4	4" Wood	1	1
5	6" Wood	1	1

Setting a maximum Bending Check (Axial & Bending) or a maximum Shear Check controls the rebar which the program chooses for the wall design. A value of 0.9 denotes that the program may choose a rebar layout that is at 90% of capacity.

Note:

- The same unity check parameters are valid for masonry walls as well. However, these parameters are not considered in wood wall design. For wood walls these values are always assumed to equal 1.0.

Concrete Wall (Rebar) Rules



	Label	Vert Bar Size	Max Vert Bar Space[in]	Min Vert Bar Spa...	Vert Bar Inc[in]	Horz Bar Si...	Max Horz Bar Spa...	Min Horz Bar Spa...	Horz Bar Inc[in]	Group Wall
1	Bearing Wall	#6	18	4	2	#4	18	4	2	☑
2	Shear Wall	#6	12	2	1	#4	12	2	1	☐
3	12" Masonry	#6	18	4	2	#4	18	4	2	☐
4	4" Wood	#6	18	4	2	#4	18	4	2	☐
5	6" Wood	#6	18	4	2	#4	18	4	2	☐

Vert and Horz Bar Size

These are the vertical and horizontal bar sizes used for reinforcement of the wall.

Note:

- The bar size and spacing is assumed to be the same for each face of the wall. Currently reinforcement must be the same for both faces.

Max/Min Vert and Horz Bar Space

The program will design the reinforcement spacing based on these guidelines. If you want the reinforcement to be at an exact spacing, simply enter that spacing as both the min and max in order to force this spacing.

Horz and Vert Bar Increment

This is the spacing change increment that the program will use for design. If the maximum spacing does not work, the spacing will drop by this increment and be checked again. The program will work its way down until it reaches a spacing that meets all reinforcement requirements.

Group Wall

For walls that have multiple regions, this checkbox allows you to group the reinforcement for the regions in a wall. Thus, the worst case vertical and horizontal reinforcement spacing will be used for all regions in the wall.

Concrete Wall (Cover) Rules

Concrete Wall Panel Cover Parameters

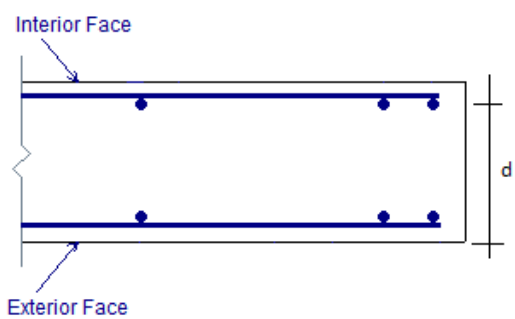
Unity Check | Concrete Wall (Rebar) | Concrete Wall (Misc) | Masonry Wall | Masonry In | Masonry Out | Masonry Lintel | Wood Wall (Studs) | Wood Wall (Fasteners)

	Label	Outer Bars	Location	Int Cover -z[in]	Ext Cover +z[...	Edge Cover[...	Transfer In	Transfer Out
1	Bearing Wall	Horizontal	Centered	1	1	2	☑	☑
2	Shear Wall	Vertical	Each Face	1.5	1.5	2.5	☐	☐
3	12" Masonry	Vertical	Each Face	1	1	2	☐	☐
4	4" Wood	Vertical	Each Face	1	1	2	☐	☐
5	6" Wood	Vertical	Each Face	1	1	2	☐	☐

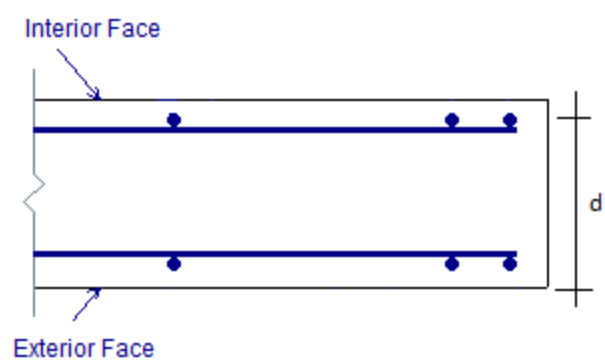
Outer Bars

This defines whether the reinforcement mesh has the Horizontal or Vertical bars closest to the face of concrete. This will affect the "d" calculation for the wall. If the location is Centered then this defines which bar is nearest the outside face of concrete.

Horizontal



Vertical



Location

This allows you to locate reinforcement at each face of wall or centered. If the reinforcement is defined as centered then the program places the vertical bar directly at the center of the wall. The horizontal bar is then placed to one

side or the other based on the "Outer Bars" designation.

Note:

- The ACI code requires two curtains of reinforcement if the wall is 10" thick or greater, thus the program will give a warning in the results if you configure your wall like this.

Int Cover (-z)

This is the 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)

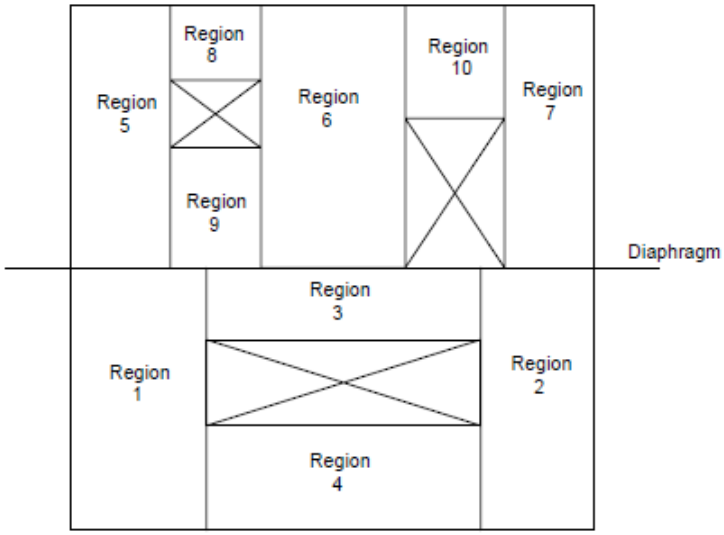
This is the clear cover distance from the exterior face of wall to the outer reinforcement. The exterior face of the wall is defined by the positive z local axis direction of the wall.

Edge Cover

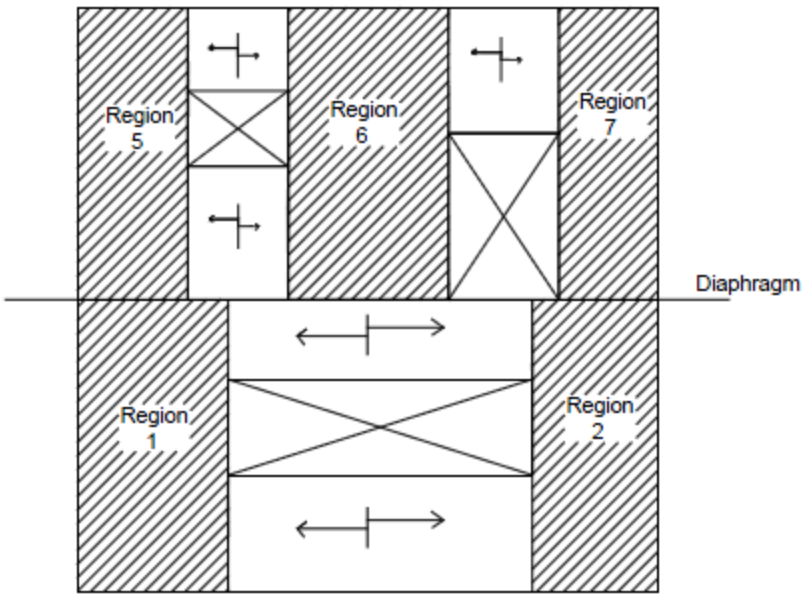
This is the "in plane" cover dimension for the outer edges of walls.

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

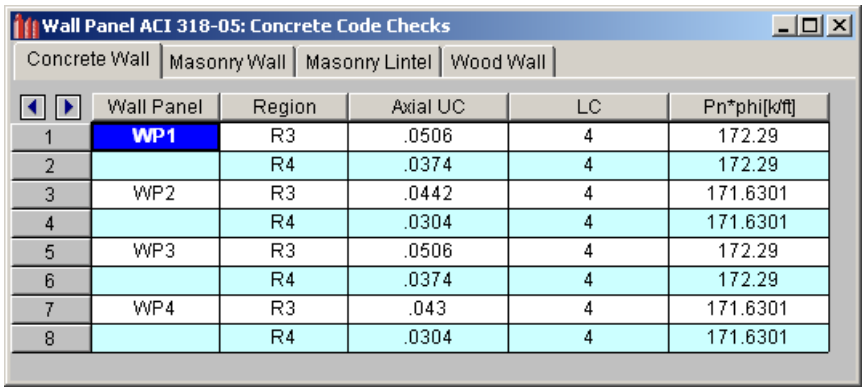
Concrete wall results are presented in the **Wall Panel Design** spreadsheet on the **Concrete** tabs, the **Concrete Reinforcing** spreadsheet on the **Concrete Wall** tab and the detail reports. Results are reported on a region by region basis.

Concrete Wall Spreadsheet Results

The **Wall Panel Design** spreadsheet contains one tab that involve concrete wall design: **Concrete Wall**, . 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.



	Wall Panel	Region	Axial UC	LC	Pn*phi[k/ft]
1	WP1	R3	.0506	4	172.29
2		R4	.0374	4	172.29
3	WP2	R3	.0442	4	171.6301
4		R4	.0304	4	171.6301
5	WP3	R3	.0506	4	172.29
6		R4	.0374	4	172.29
7	WP4	R3	.043	4	171.6301
8		R4	.0304	4	171.6301

The **Axial UC** entry gives the maximum code check due to axial force.

The **LC** columns report the load combination that produces each of the highest code check values.

The **Pn*Phi** reports the axial capacity of the wall.

Concrete Reinforcing Spreadsheet Results

The **Concrete Wall** tab contains reinforcement results for each region in a concrete wall panel.

Note:

- If a wall panel is set to Transfer forces, then regions above and below opening will not have results. See the Concrete Wall - Design Rules topic for more information.

Concrete Wall

Concrete Wall Reinforcement (By Combination)

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

	Wall	Region	Thickness[in]	Hor. Bar Size	Vert. Bar Size
1	WP1	R2	8	#4@18in oc (ef)	#6@18in oc (ef)
2		R3	8	#4@18in oc (ef)	#6@18in oc (ef)
3		R4	8	#4@18in oc (ef)	#6@18in oc (ef)
4		R5	8	#4@18in oc (ef)	#6@18in oc (ef)
5		R6	8	#4@18in oc (ef)	#6@18in oc (ef)
6	WP2	R2	8	#4@18in oc (ef)	#6@18in oc (ef)
7		R3	8	#4@18in oc (ef)	#6@18in oc (ef)

The **Concrete Wall** tab displays the thickness, horizontal and vertical reinforcement sizes, and spacing for each region in the concrete wall.

Concrete Wall Detail Reports

The detail reports show the overall geometry, analysis, and design for the individual regions/stories of the wall panel. The report also shows envelope diagrams for the forces and moments in the region.

Three basic types of detail reports are provided: Wall, Region and Opening.

Accessing the Detail Reports and the Specific Windows

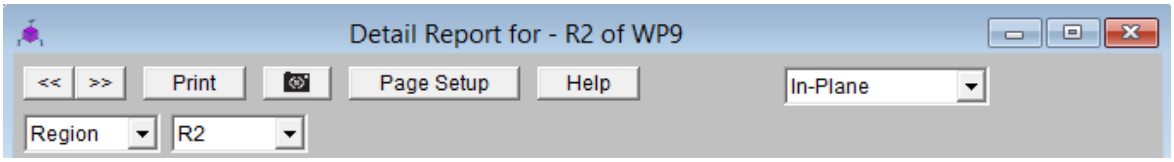
Once you have a solved model, the detail reports become available. They are accessible in two ways:

1. If you have the **Wall Results** spreadsheet open, there will be a button at the top of the screen: Detail Report for Current Item. This will open up the detail report window.
2. If you are in a graphic view of your model, there is a Detail button 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:

- If a wall panel is set to Transfer forces, then regions above and below opening will not have results. See the Concrete Wall - Design Rules topic for more information.

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

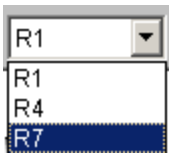


These options control the display of the Detail Report:

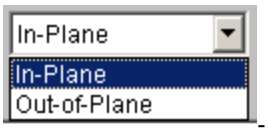
- << >> - The arrow buttons allow you to scroll quickly between the different wall panels in your model.
- - This button will allow you to take a snapshot of the current detail report you are viewing so that it can be added to a report. View the Printing topic for more information.

-

- The first drop down list allows you to select between individual Region and Opening (Lintel) results and a summary of the entire Wall. Below we will explain the importance of each of these sections.

- 

- The second drop down list allows you to select between different Regions or Openings within the individual wall panel.

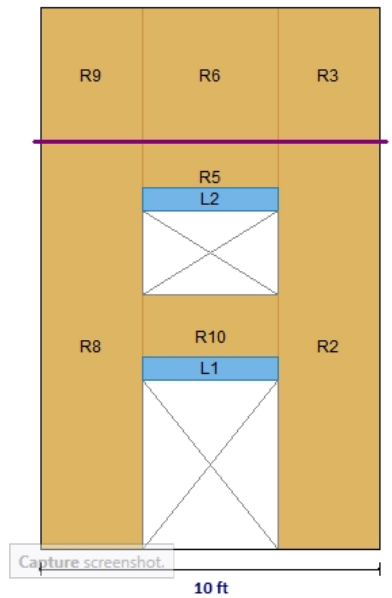
- 

- If you have selected a Region, then you have the option of whether to view the in plane or out of plane report.

Wall Report

This report gives an overview of the wall, a summary of the controlling code checks and deflection information. This report also displays information about the wall, similar to the Region Input Echo and also gives an image of the wall. The image shows region locations, wall length and story dimensions, and the nodes that define the corners of the wall panel.

CRITERIA		MATERIALS			GEOMETRY		
Code	: ACI 318-14	Material Set	: Conc3000NW		Total Height	: 16	ft
Design Rule	: Typical	Concrete fc	: 3	ksi	Total Length	: 10	ft
Loc of r/f	: Each Face	Concrete E	: 3156	ksi	Thickness	: 9.625	in
Outer Bars	: Vertical	Concrete G	: 1372	ksi			
		Conc Density	: .145	k/ft^3	Int Cover (-z)	: .984	in
Vert Bar Size	: #5	Lambda	: 1		Ext Cover (+z)	: .984	in
Horz Bar Size	: #4	Conc Str Blk	: Rectangular		Cover Open/Edge	1.968	in
					K	: 1	
Transfer In?	: No	Vert Bar Fy	: 60	ksi	Use Cracked?	: Yes	
Transfer Out?	: No	Horz Bar Fy	: 60	ksi	In Icr Factor	: .7	
Group Wall?	: No	Steel E	: 29000	ksi	Out Icr Factor	: .35	
					Parapet Height	: 4	ft



REGION RESULTS

Region	UC Max In Plane	LC
R2	.016	1
R3	.007	1
R7	.016	1
R8	.007	1

REINFORCEMENT RESULTS

Region	Vertical Reinforcement	Horizontal Reinforcement
R2	#6@18in oc e.f.	#4@18in oc e.f.
R3	#6@18in oc e.f.	#4@18in oc e.f.
R7	#6@18in oc e.f.	#4@18in oc e.f.
R8	#6@18in oc e.f.	#4@18in oc e.f.

The **Region Results** section gives the tabulated results of all regions in the wall for in plane 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 four portions: input echo, diagrams and design, wall section properties and cross section detailing.

Note:

- In RISAFloor, the detail reports are less detailed because RISAFloor does not consider lateral forces.

Input Echo

Below is the input echo portion of the detail report.

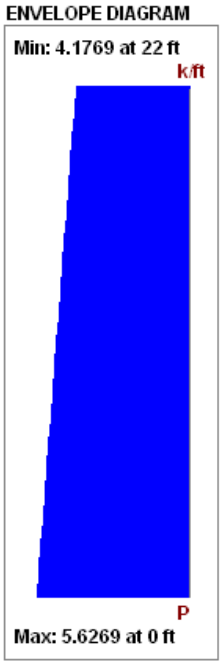
CRITERIA			MATERIALS			GEOMETRY		
Code	: ACI 318-14		Material Set	: Conc3000NW		Total Height	: 12	ft
Design Rule	: Typical		Concrete f'c	: 3	ksi	Total Length	: 3	ft
Loc of r/f	: Each Face		Concrete E	: 3156	ksi	Thickness	: 9.625	in
Outer Bars	: Vertical		Concrete G	: 1372	ksi			
			Conc Density	: .145	k/ft^3	Int Cover (-z)	: .984	in
Vert Bar Size	: #5		Lambda	: 1		Ext Cover (+z)	: .984	in
Horz Bar Size	: #4		Conc Str Blk	: Rectangular		Cover Open/Edge	1.968	in
						K	: 1	
Vert Bar Spac	: 16	in	Vert Bar Fy	: 60	ksi	Use Cracked?	: Yes	
Horz Bar Spac	: 16	in	Horz Bar Fy	: 60	ksi	Icr Factor	: .35	
Group Wall?	: No		Steel E	: 29000	ksi	Parapet Height	: 4	ft

Criteria	Description
Code	Gives the code used to design the wall.
Design Rule	Gives the design rule used to design reinforcement and cover for the wall.
Loc of r/f	States whether reinforcement is defined at each face of the wall or centered.
Outer Bars	States whether the outer bar in the wall is vertical or horizontal.
Bar Size	States the bar size for both horizontal and vertical reinforcement
Bar Spacing	States the bar spacing for both horizontal and vertical reinforcement
Group Wall?	States whether the regions in the wall are grouped or not. See the Concrete Wall - Design Rules topic for more information.

Materials	Description
Material Set	States the Material used for the design of the wall.
Concrete f'c	States the compressive strength of the concrete.
Concrete E	States the modulus of elasticity of the concrete.
Concrete G	States the shear modulus of the concrete.
Conc Density	States the unit density of concrete used for self-weight calculations.
Lambda	States the lightweight concrete factor for shear design.
Conc Str Blk	States whether a rectangular (Whitney's) or parabolic stress block was used.
Bar Fy	States the reinforcement strength for both vertical and horizontal bars
Steel E	States the modulus of elasticity for reinforcement.

Geometry	Description
Wall Dimensions	States the height, length, and thickness of the wall panel region.
Cover Dimensions	States the interior, exterior, and edge reinf. cover dimensions.
K	States the effective length factor which is used in determining slenderness of the wall.
Use Cracked?	States whether a wall is considered to be cracked or not. Defines whether or not to use cracking in the determination of the moment of inertia.
Icr Factor	States the factor that Igross is multiplied by to get the cracked moment of inertia. This defaults to 0.7 for in plane and 0.35 for out of plane.
Parapet Height	Additional height from the Wall Panels spreadsheet

Diagrams and Design



ACI 318-05 Code Check

AXIAL DETAILS

UC Max	: .0267	
Location	: 0	ft
Gov Pu	: 5.6269	k/ft
phi*Pn	: 210.7386	k/ft
phi eff.	: .65	
Gov LC	: 2	

Envelope Diagrams

These diagrams show the axial forces in the wall region. 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 Details

The demand and capacities here are presented on a per foot length of the wall region. Thus, for demand the program will take the axial tension/compression over the entire wall region and divide it by the length of the wall region. The axial capacity is also based on the capacity of the entire wall divided by the length. For more information on tension/compression capacity of concrete walls, see the Concrete Wall - Design topic.

Note:

- The program is not currently considering slenderness for gravity-only walls. The ACI code specifies a minimum moment demand and then factors that demand to consider slenderness. This

factored minimum moment will be considered in a future version of the program.

Wall Section Properties

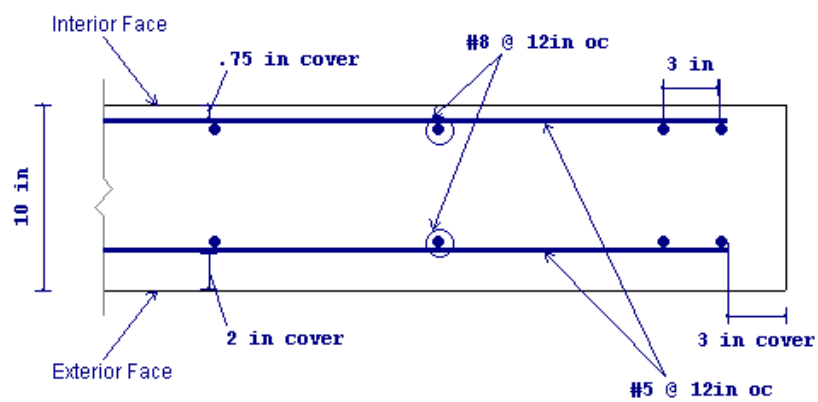
WALL SECTION PROPERTIES

Total Length	: 20	ft	r	: 69.282	in	As Provided (H)	: 14.7262	in^2
A	: 2400	in^2	KL/r	: 2.078		rho Provided (H)	: .0061	
Igross	: 1.152e+7	in^4				As min (H)	: 5.28	in^2
Icracked	: 8.064e+6	in^4				rho min (H)	: .0022	
Cracked Mom, Mcr	: 3286.3353	k-ft						
						As Provided (V)	: 34.5575	in^2
						rho Provided (V)	: .0144	
						As min (V)	: 3.6	in^2
						rho min (V)	: .0015	

This section reports the properties used to calculate the wall capacities. The reinforcement details (minimums and provided area) are reported.

Cross Section Detailing

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

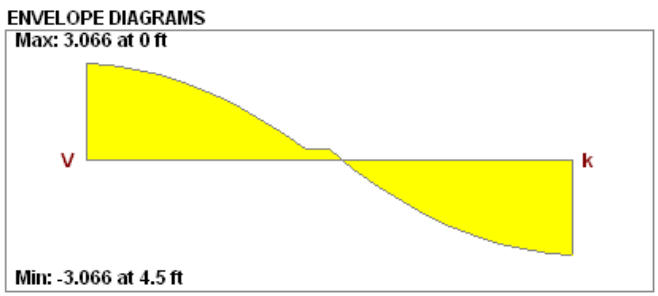
The firstsection of the detail report echoes back the basic input parameters (Criteria, Materials, Geometry) entered by the user. An example is shown below:

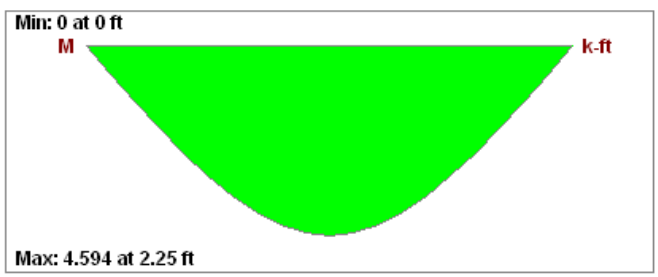
CRITERIA		MATERIALS			GEOMETRY		
Code	: ACI 318-08	Material Set	: Conc3000NW		Total Height	: 3.5	ft
Design Rule	: Typical	Concrete f'c	: 3	ksi	Total Length	: 6	ft
Loc of r/f	: Each Face	Concrete E	: 3156	ksi	Thickness	: 12	in
Outer Bars	: Vertical	Concrete G	: 1372	ksi			
		Conc Density	: .145	k/ft^3	Int Cover (-z)	: 1	in
Vert Bar Size	: #6	Lambda	: 1		Ext Cover (+z)	: 1	in
Horz Bar Size	: #4	Conc Str Blk	: Rectangular		Cover Open/Edge	2	in
Valid Region	: R2	Vert Bar Fy	: 60	ksi			
		Horz Bar Fy	: 60	ksi			
		Steel E	: 29000	ksi			

Since the lintel is not being designed, much of this information is not used but is displayed for reference.

Information	Description
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





Masonry Wall Panel - Modeling

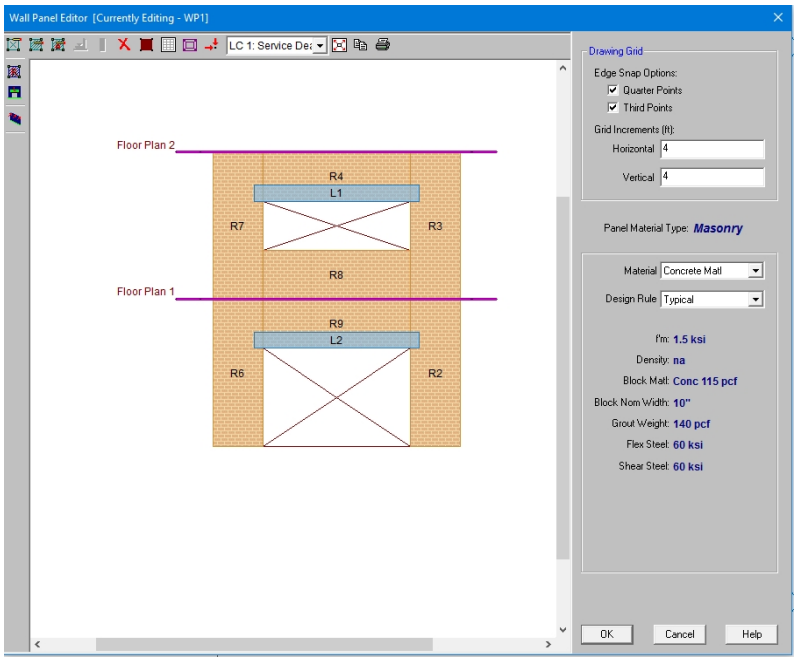
The masonry wall panel element allows you to easily model, analyze and design masonry walls for in plane and out of plane loads. Here we will explain the masonry specific inputs and procedures for modeling. For general wall panel information, see the Wall Panels topic. For information on masonry design 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 will have the following view controls:

Toggle Region Display allows you to turn the display of regions on or off.

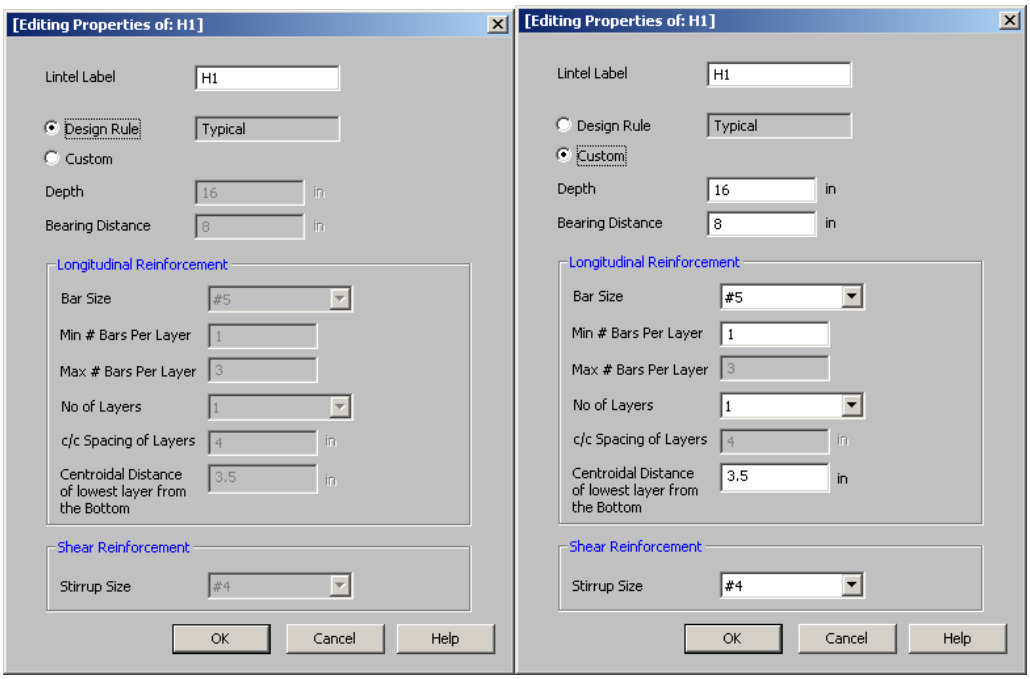
Toggle Lintel Display allows you to turn the display of lintels on or off.

Toggle Out of Plane Reinforcement allows you to turn the display of out of plane reinforcement on or off after you have solved your model.

Toggle In Plane Reinforcement allows you to turn the display of in plane reinforcement on or off after you have solved your model.

Creating Openings in Masonry with Lintels

Within the **Wall Panel Editor**, you have the option of adding rectangular openings to masonry wall panels. To draw an opening, select the **Openings** button and then select two nodes or grid intersections which make up the two diagonal corners of your opening. When an opening is drawn a lintel is automatically created above the opening. To view or edit the properties of a masonry lintel, double-click inside the boundary of the drawn opening. This will bring up the Editing Properties window for that particular lintel.



This window will show the design options set in the Wall Design Rules - Lintel 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, then you canchoose the **Custom** option. When using the custom option, the program will now use all of the information set in the Lintel Editor and will disregard any information given by the design rule.

Here we will walk through the different input options available for designing/analyzing lintels:

Depth - This is the depth of your lintel.

Bearing Distance - This is the bearing length at either end of the lintel. This is used to calculate the effective length of the lintel.

Bar Size - This is the reinforcement size you wish to use for your main reinforcing in the lintel.

Bars Per Layer - This is the number of bars you wish to have in a given layer of reinforcement. There is also an option to have the this value optimized based on geometry of the section and also the number of layers that you have defined.

Number of Rebar Layers - This is an option if you need multiple layers of reinforcement in the lintel.

c/c Spacing of Layers - This is the distance between layers (if there is more than one).

Centroidal Distance of lowest layer from the Bottom - This value is used to calculate the "d" value for the lintel.

Stirrup Size - This is the size of stirrup that will be added to the lintel if required.

Note:

- When inputting bar sizes for your lintels, the program will not allow you to place reinforcement that will not actually fit into the lintel because of width constraints. We use the actual dimensions of the block, the face-

shell thickness for the given block chosen and use a 1/2" clear cover between reinforcement and the block per section 6.1.3.5.

Masonry Regions

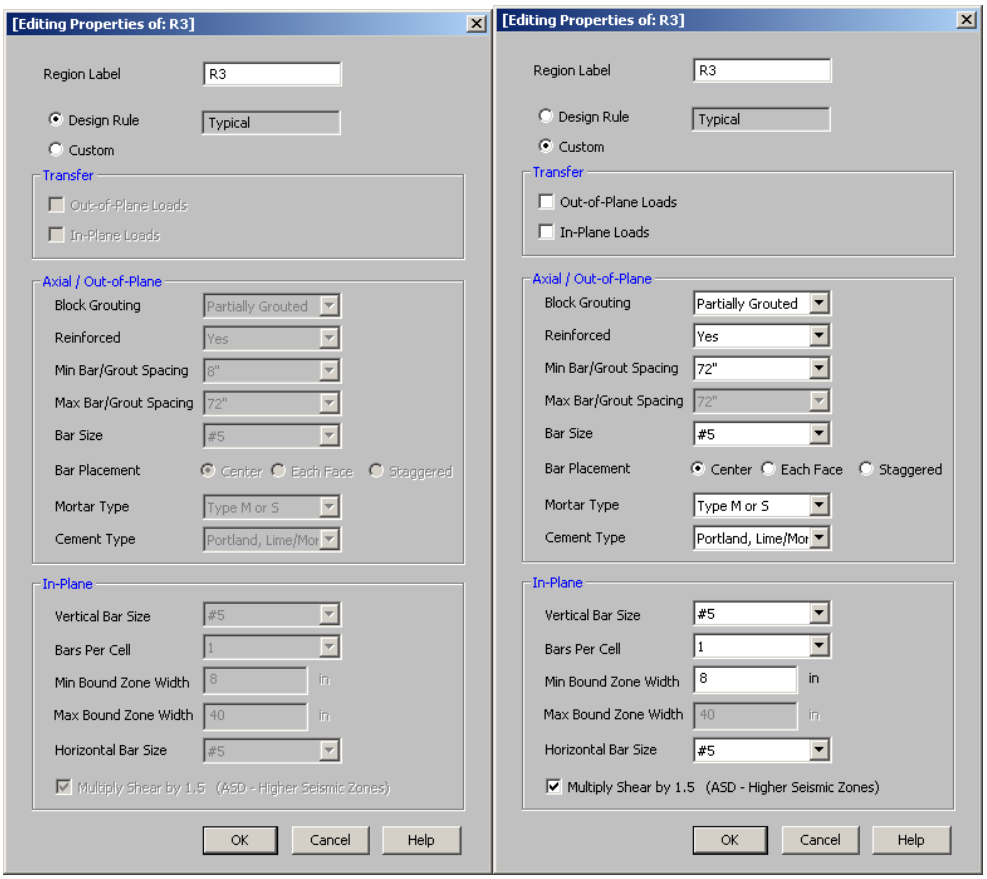
Within the Wall Panel Editor, you have the option of creating rectangular regions within the masonry wall panel. Regions are used to define reinforcement in different parts of the wall. Each region will be assigned a uniform reinforcement, which may be different than the reinforcement in other parts of the same wall.

If no regions have been drawn on a wall then they will be automatically generated when a solution is performed. To automatically generate regions prior to running a solution, click the **Generate Wall Regions Automatically** button

To manually draw regions, select the **Create New Regions** button and use your cursor to select two nodes or grid intersections which make up the diagonal corners of the region. To exit this tool right-click your mouse.

To view or edit the properties of a masonry region, double-click inside the boundary of the drawn region. This will bring up the Editing Properties window for that particular region.

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.



In most cases this Region Editor would only be used as a viewer. If, however, you want to change the reinforcement for a region within the wall to make it different than the Wall Design Rules, you can use the **Custom** option. When using the custom option the program will now use all of the information set in the Region Editor and will disregard any information given by the design rule.

Note:

- For models created with version 5.1.1 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 6 or later the regions will default to the wall design rule.

Within this dialog you can specify the properties which will be used for the design of the region. The program can optimize the bar spacing and the boundary zone width based on code checks. The block size, reinforcing strength and the method of self-weight calculation are defined in the Design Rules under the Masonry Wall tab.

Note:

The program designs regions separately for out-of-plane and in-plane forces, thus the Region Editor is divided into different parts.

Here we will walk through the different input options available for designing/analyzing regions:

Transfer - This is an option as to whether or not you want this region to transfer Out of Plane and In Plane Loads. If you check these transfer options, the program will dump 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/Out-of-Plane - Allows you to define properties of a region based on out-of-plane/axial forces.

- **Block Grouting** - Allows you to define how you want your wall grouted. If you choose "Partially Grouted" the program will optimize the grout spacing with the Bar/Grout Spacing
- **Reinforced** - Gives the option of designing the wall as reinforced or unreinforced.
- **Bar/Grout Spacing** - This allows you to define the bar/grout spacing. If you have the "Optimize" box checked, the program will optimize the reinforcement spacing based on code checks.
- **Bar Size** - This is the main vertical bar size that will be used in design.
- **Bar Placement** - This defines how you want to lay out the reinforcement in your region. Each Face will put reinforcement on both faces of a given cell. Staggered alternates the bars to either face of wall region.
- **Note:**
  - When using the staggered option you are selecting to space the bars at each face at double the bar/-grout spacing defined above. For example, if you have a staggered spacing at 24" oc you have a bar on the outside face at 48" oc and a bar on the inside face at 48" oc. These bars are staggered, thus you have grout filled cores at 24" oc.
- **Mortar Type** - Allows you to specify the type of mortar to design for.
- **Cement Type** - Allows you to specify the type of cement to design for.

In-Plane - Allows you to define properties of a region based on in-plane forces.

- **Vertical Bar Size** - Allows you to define vertical bar size for the boundary zones.
- **Bars Per Cell** - Allows you to define one or two bars per cell.
- **Boundary Zone Width** - The user must define a boundary zone width but RISA will optimize the width if the "Optimize" box is checked.
  - **Note:** If you have the optimize checkbox selected, the program will optimize the boundary zone width based on code checks.
- **Horizontal Bar Size** - Allows you to define horizontal bar size to be used if horizontal reinforcing is required.
- **Multiply Shear by 1.5** - This is an option that may be required in high seismic zones. This option is available per section 7.3.2.6.1.2.

Masonry Wall Optimization

The program will optimize masonry walls and lintels based on the required demand forces. The program can optimize:

- Vertical bar/grout spacing for out of plane design.
- Reinforcement bars for lintel flexural design.

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.

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 is one option for optimizing. For out of plane design, the program will optimize the spacing of bars for strength (not deflection) considerations of the wall .

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.
- You can only define regions that are full height from floor to floor. If you have a panel that is multiple floors high, you can span your regions to multiple floors or single floors.
- 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 Panel - Design

The masonry wall panel element allows you to easily model, analyze and design masonry walls for in plane and out of plane loads 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.

Axial Design - ASD

In Plane Axial Stress, fa

The axial stress in a wall due to axial forces, fa, is calculated as:

$$f_a := \frac{P}{A_n}$$

P = Total axial force in wall at the reported location

$$A_{nRMEH} := Eq_{solidthickness} \cdot L_{wall}$$

$$A_{nvNCMA} := A_{nvNCMAperfoot} \cdot L_{wall}$$

This is applicable for all out of plane masonry calculations.

Note:

- The program does not design masonry for net tension forces. fa will equal 0 ksi in these cases.

In Plane Axial Capacity, Fa

The calculation of Fa 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 Ast = 0. RISA conservatively uses only the masonry in calculating the compression capacity. The equations are as follows:

if h/r <= 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 = $K*h_{actual}$

Note:

- The program does not design masonry for net tension forces. If there is net tension, Fa will still equal the compressive capacity of masonry.

In Plane Design - Strength

In Plane Axial Strength, Pn

This is calculated from Equations 9-15 and 9-16:

if h/r <= 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 = $K*h_{actual}$

Note:

- The maximum usable masonry strain at the extreme compression fiber, $\mathbf{e_{mu}}$, 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.

Out of plane deflections are calculated based on the Wall's Finite Element Analysis using a modified thickness, tmod, for the wall plates. The modified thickness is calculated using the wall panel's out of plate cracked moment of inertia, Icr, which can be input in the Advanced tab of the Wall Panels spreadsheet.

$$t_{mod} := t_{original} \cdot \sqrt{I_{cr}}$$

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.

Unreinforced Masonry Design - ASD

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.
- Note that fb/Fb is not considered for RISAFloor

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, fv

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, Fv

The program calculates the capacity, Fv, from Equation 8-25, which is shown in the previous In Plane Shear Capacity Fv section.

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 **fv > Fvm**. If **fv < Fvm**, 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. 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.

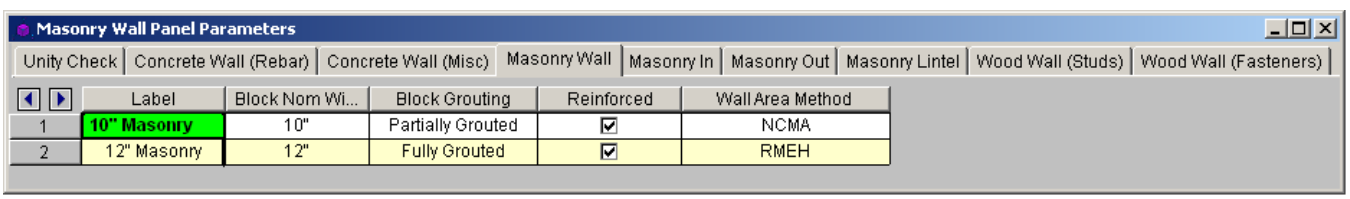
Unity Check

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



	Label	Block Nom Wi...	Block Grouting	Reinforced	Wall Area Method
1	10" Masonry	10"	Partially Grouted	☑	NCMA
2	12" Masonry	12"	Fully Grouted	☑	RMEH

Block Nominal Width

This 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, by Amrhein, Copyright 1998.

Block Grouting

This option defines how the wall is grouted. If "Partially Grouted" is chosen, then the spacing of grout will be based on the bar spacing defined on the **Masonry Out** tab.

Reinforced

This defines whether the wall is reinforced or not.

Wall Area Method

This option 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 *Reinforced Masonry Engineering Handbook*, James Amrhein, 5th edition copyright 1998.

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.

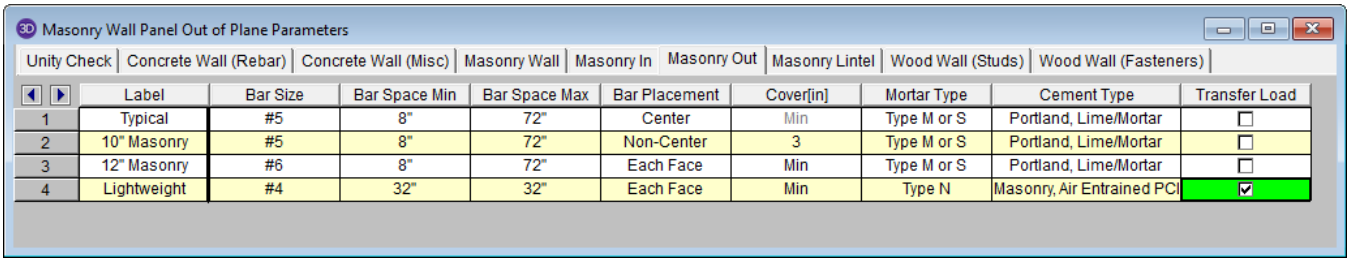
The **RMEH** values assume full mortar bedding (both face-shells and all webs). The Amrhein values also appear to average in the area of horizontal bond beams as well. This would make the area conservative for a self-weight calculation, but unconservative for stress calculations. With these considerations in mind we are defaulting the behavior to use **NCMA** by default.

See the Masonry Wall - Design topic for the specific calculations regarding NCMA or RMEH.

Note:

- In version RISAFloor V6.0.1 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 RISAFloor V6.0.1 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.

Masonry Wall Out of Plane Design



	Label	Bar Size	Bar Space Min	Bar Space Max	Bar Placement	Cover[in]	Mortar Type	Cement Type	Transfer Load
1	Typical	#5	8"	72"	Center	Min	Type M or S	Portland, Lime/Mortar	☐
2	10" Masonry	#5	8"	72"	Non-Center	3	Type M or S	Portland, Lime/Mortar	☐
3	12" Masonry	#6	8"	72"	Each Face	Min	Type M or S	Portland, Lime/Mortar	☐
4	Lightweight	#4	32"	32"	Each Face	Min	Type N	Masonry, Air Entrained PCI	☑

Bar Size

This is the main vertical bar size that will be used for axial/out of plane design.

Bar Space Min/Max

This allows you to give a maximum and minimum bar/grout spacing. If you give a range between the max and min, then the program will optimize the reinforcement spacing according to strength requirements (code checks).

Bar Placement

This defines how the reinforcement is placed in the wall region.

Center will put a single bar centered in a given cell.

Non-Center will put a single bar non-centered in a given cell.

Each Face will put reinforcement on both faces of a given cell.

Staggered alternates the bars on either face along the length of the region.

Note:

- When using the staggered option you are selecting to space the bars at each face at double the bar/grout spacing defined above. For example, if you have a staggered spacing at 24" oc you have a bar on the outside face at 48" oc and a bar on the inside face at 48" oc. These bars are staggered, thus you have grout filled cores at 24" oc.

Cover

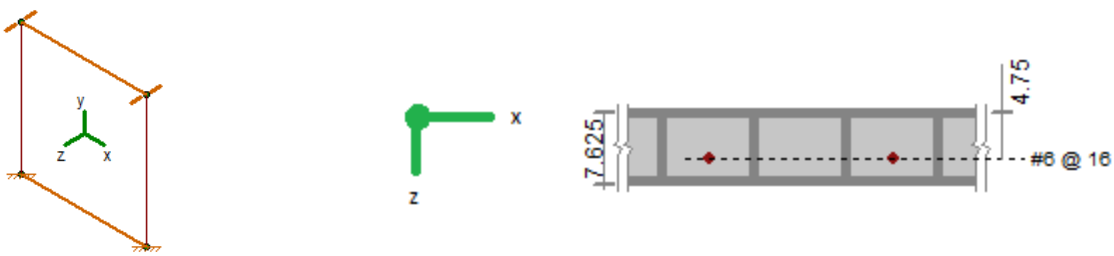
Allows you to specify the distance from the exterior of the masonry block to the extreme fiber of the reinforcement. The default cover value is defined as **Min**. The **Min** input will use whichever governs between these two sections of the specification:

- 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 = $t_{faceshell}$ + 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".

Thus, the **Min** term will create a cover equal to the maximum of $t_{faceshell}$ + 1/2" OR 1-1/2".

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***.

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 shows exactly where this bar is then located in the wall (d = 7.625" - 2.5" - 3/8" = 4.75").



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.

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.

Note:

- For the **Center** bar placement this field is ignored.

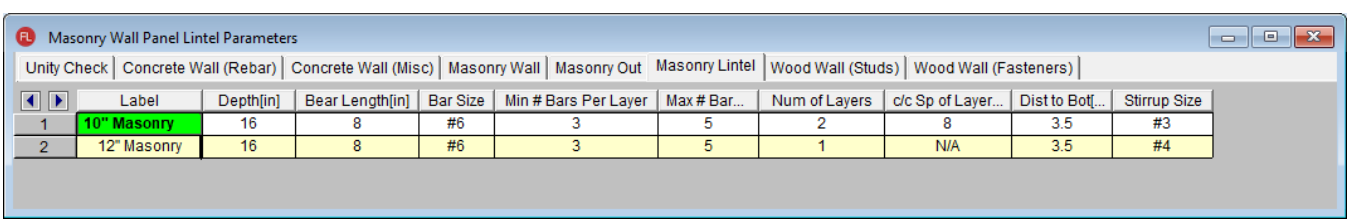
Mortar/Cement Types

Allows you to 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

This option will 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



	Label	Depth[in]	Bear Length[in]	Bar Size	Min # Bars Per Layer	Max # Bar...	Num of Layers	c/c Sp of Layer...	Dist to Bot[...	Stirrup Size
1	10" Masonry	16	8	#6	3	5	2	8	3.5	#3
2	12" Masonry	16	8	#6	3	5	1	N/A	3.5	#4

Depth

This is the total depth of your lintel.

Bear Length

This is the bearing length at either end of the lintel. This is used to calculate the effective length of the lintel.

Bar Size

This is the reinforcement size for your main reinforcing in the lintel.

Min/Max # Bars Per Layer

This is the number of bars you wish to have in a given layer of reinforcement. If you give a range between the max and min, then the program will optimize the reinforcement spacing based on geometry of the section and also the number of layers that you have defined.

Num of Layers

This is an option if you require multiple layers of reinforcement in the lintel.

c/c Sp of Layers

This is the distance between layers (if there is more than one).

Dist to Bot

This defines the distance from the centerline of the lowest-most bar to the bottom fiber of the lintel.

Stirrup Size

This is the size of stirrup that will be added to the lintel if required.

Masonry Wall Results

Masonry wall results are presented in the wall results spreadsheets and the detail reports. Results are reported on a region by region basis. In addition, the code checks for lintels spanning openings are reported separately.

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.

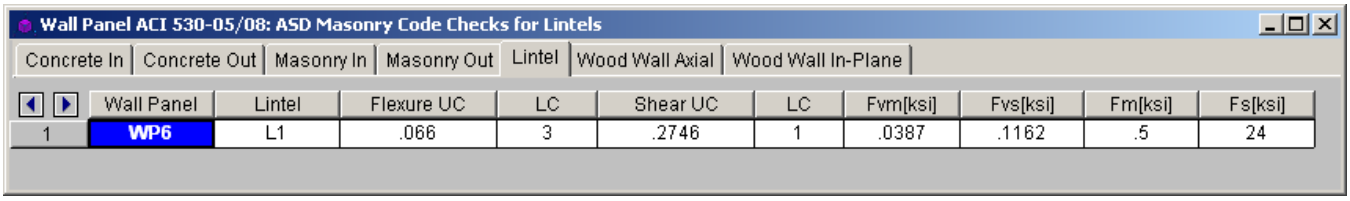
Note:

- All code references in this topic refer to the **TMS 402-16** specification unless noted otherwise.

Masonry Wall Spreadsheet Results

See Wall Panels - Results topic for spreadsheets not found here.

Lintels



	Wall Panel	Lintel	Flexure UC	LC	Shear UC	LC	Fvm[ksi]	Fvs[ksi]	Fm[ksi]	Fs[ksi]
1	WP6	L1	.066	3	.2746	1	.0387	.1162	.5	24

The Lintel results give the results for the masonry lintels that span over user defined openings in the wall. They can also be viewed by looking at the detail report associated with each opening.

The **Bending UC** entry gives the code check due pure flexure of the Lintel. Axial force is not considered in this code check at all. The **Shear UC** gives the code check for shear. A value greater 1.0 for either of these values would indicate failure.

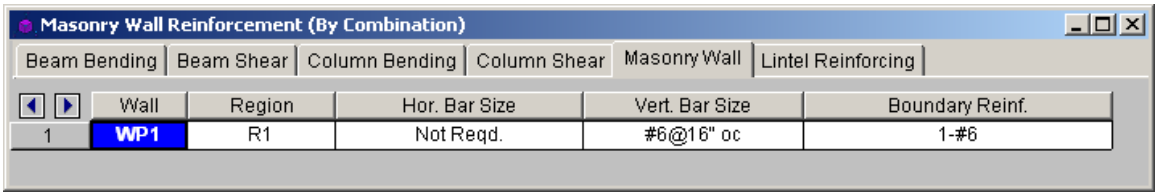
The **Fm** and **Fs** or **Mn*Phi** reports the calculated allowable bending stress or moment capacity for the region.

The **Fvm** and **Fvs** or **Vn*Phi** reports the allowable shear stress or shear capacity.

Concrete Reinforcing Spreadsheet Results

The last two tabs of this spreadsheet contain results for Masonry Wall reinforcement.

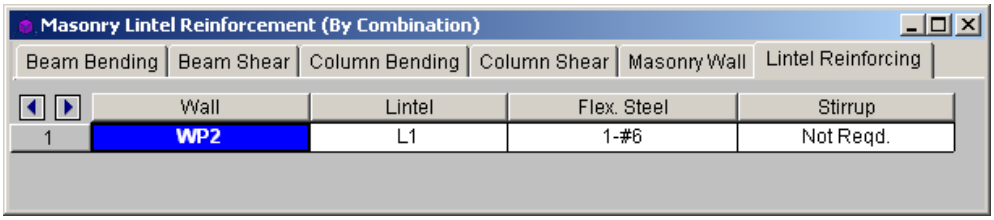
Masonry Wall



	Wall	Region	Hor. Bar Size	Vert. Bar Size	Boundary Reinf.
1	WP1	R1	Not Reqd.	#6@16" oc	1-#6

The Masonry Wall tab displays analysis results for the reinforcement of each region defined in your masonry wall.

Lintel Reinforcing



The Lintel Reinforcing tab displays analysis results for the reinforcement of each lintel defined in your masonry wall.

Masonry Wall Detail Reports

The detail reports show the overall geometry, analysis and design for the individual regions of the wall panel. The report also shows envelope diagrams for the forces and moments in the region.

Three basic types of detail reports are provided: Wall, Region and Opening.

Accessing the Detail Reports and the Specific Windows

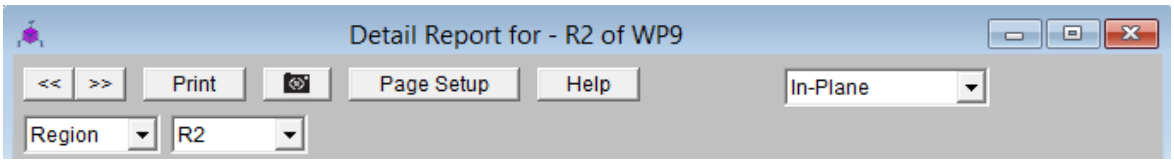
Once you have a solved model, the detail reports become available. They are accessible in two ways:

1. If you have the **Wall Results** spreadsheet open, there will be a button at the top of the screen: Detail Report for Current Item you can right click and select **Detail Report**. This will open up the detail report window.
2. If you are in a graphic view of your model, there is a Detail 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:

- If a wall panel is set to Transfer forces, then regions above and below opening will not have results. See the Concrete Wall - Design Rules topic for more information.

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



These options control the display of the Detail Report:

- << >> - The arrow buttons allow you to scroll quickly between the different wall panels in your model.
- - This 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.

-

- The first drop down list allows you to select between individual Region and Opening (Lintel) results and a summary of the entire Wall. Below we will explain the importance of each of these sections.

- - The second drop down list allows you to select between different Regions or Openings within the individual wall panel.

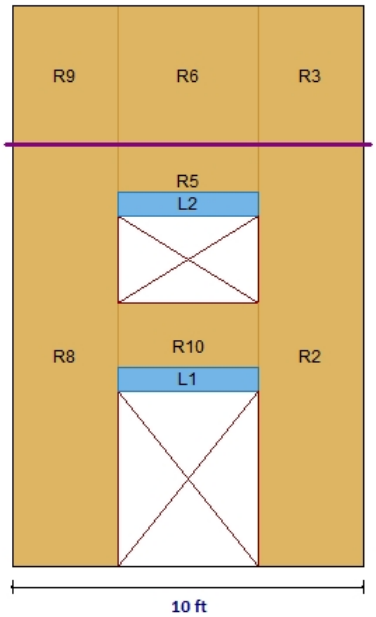
Note:

- In RISAFloor these checks will be much less extensive, as RISAFloor does not consider lateral forces.

Wall Report

This report gives an overview of the wall, a summary of the controlling code checks and deflection information. This report also displays information about the wall, similar to the Region Input Echo and also gives an image of the wall. The image shows region locations, wall length and story dimensions, and the nodes that define the corners of the wall panel.

CRITERIA		MATERIALS			GEOMETRY		
Code	: ACI 530-13: ASD	Masonry f'm	: 1.5	ksi	Total Height	: 16	ft
Special Insp	: Yes	Masonry Em	: 1350	ksi	Total Length	: 10	ft
Design Rule	: Typical						
Wall Area	: NCMA	Steel fy	: 60	ksi	Blk Nom Width	: 10"	
Transfer In?	: No	Steel E	: 29000	ksi	Blk Grouting	: Partially Grouted	
Transfer Out?	: No				1.5 Shear Factor	: Yes	
		Blk Material	: Conc 115 pcf		Parapet Height	: 4	ft
K	: 1	Grt Weight	: 140 pcf				
Use Cracked?	: Yes	Mortar Type	: Type M or S				
In Icr Factor	: .5	Cement Type	: Portland, Lime/Mortar				
Out Icr Factor	: .5						
Custom Regions	: No						



REGION RESULTS

Region	UC Max In Plane	LC
R1	.053	1
R4	.052	1

REINFORCEMENT RESULTS

Region	Vertical Reinforcement
R1	#5@72" oc ctr
R4	#5@72" oc ctr

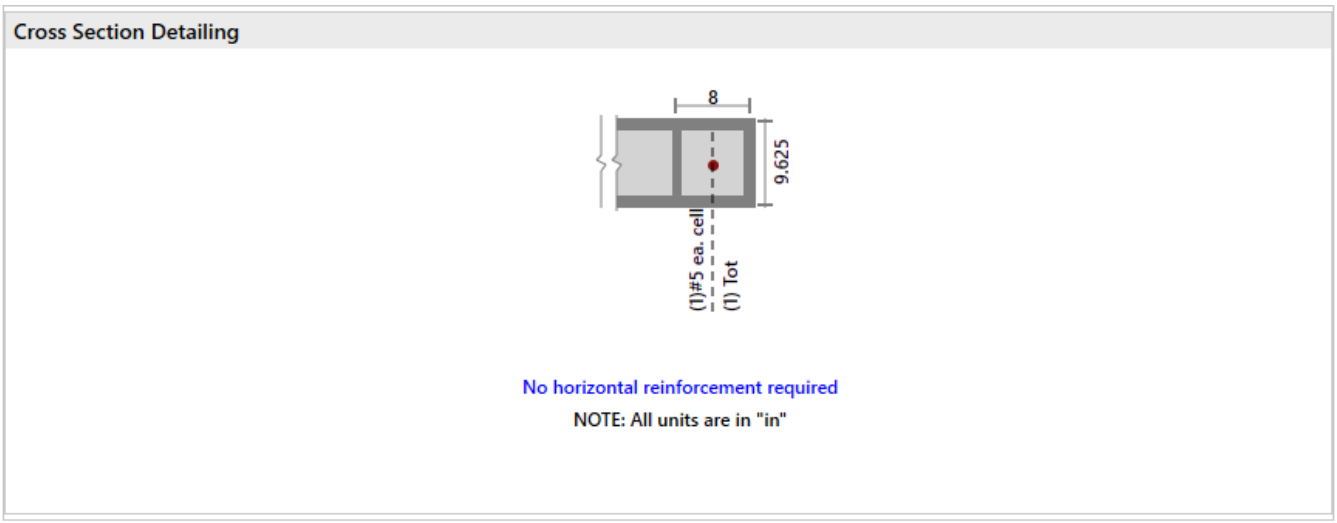
LINTEL REINFORCEMENT RESULTS

Lintel	Flexural Reinforcement	Shear Reinforcement
L1	1-#5	Not Reqd.

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.



Out of Plane Masonry Detail Reports

You can click any of the **Out-of-Plane** region sections in order to expand the section and view additional details pertinent for that specific region.

Out of Plane Walls - Input Echo

The top section of the detail report echoes back the input information used in the design of the wall region or lintel. This information is summarized below:

CRITERIA		MATERIALS			GEOMETRY		
Code	: ACI 530-13: ASD	Masonry fm	: 1.4	ksi	Total Height	: 10.5	ft
Special Insp	: Yes	Masonry Em	: 1050	ksi	Eq Sld Thickness	7.625"	
Wall Area	: RMEH						
		Steel fy	: 60	ksi	Blk Grouting	: Fully Grouted	
Type of Design	: ASD	Steel E	: 29000	ksi	Grt/Bar Spacing	: 16"	
Reinforced	: Yes						
		Blk Material	: Conc 135 pcf		Loc of r/f	: Non-Center	
Vertical Bar Size	: #6	Grt Weight	: 140 pcf				
		Wall Dead Wt	: .084	ksf			
		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.

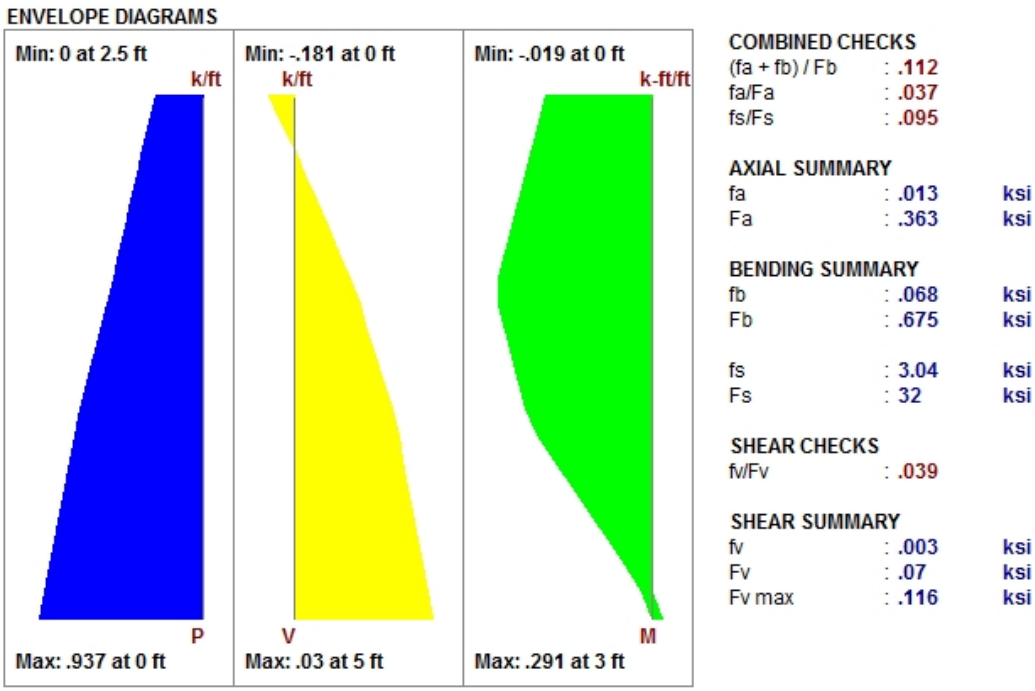
The **Eq Solid Thickness** comes from the "Wall Area Method" in the Wall Design Rules - Masonry Wall tab.

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.



The **Combined Check Summary** gives maximum overall code checks considering the effects of both bending and axial stresses. This section is not reported for MSJC strength design.

The **Axial Summary** and **Bending Summary** give the values used in the axial and flexural code checks at the location which controls the combined check (fa/Fa)+(fb/Fb).

For ASD design, the maximum bending stress in the flexural reinforcement is reported as **fs**, and the allowable steel bending stress as **Fs**. These values are given for the Load Combination and section that produce the maximum code check (fs/Fs).

The **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 **Shear Summary** reports the maximum shear demand on the beam as **fv** or **Vu**. The allowable shear stresses are then reported as **Fv**

DESIGN DETAILS

AXIAL DETAILS			BENDING DETAILS			SHEAR DETAILS		
Max Axial	: .8	k/ft	Max Moment	: 2.007	k-ft/ft	Max Shear	: .401	k/ft
Location	: 0	ft	Location	: 0	ft	Location	: 0	ft
Load Comb	: 1		Load Comb	: 1		Load Comb	: 1	
Rad gyration r	: 3.04	in	k	: .34		Width for Shear	: 11	in
h'/r	: 39.474		d	: 4.813	in	Corresponding M	2.007	k-ft/ft
			j	: .892		Corresponding P	.8	k/ft
						M / (V*d)	: 1	

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

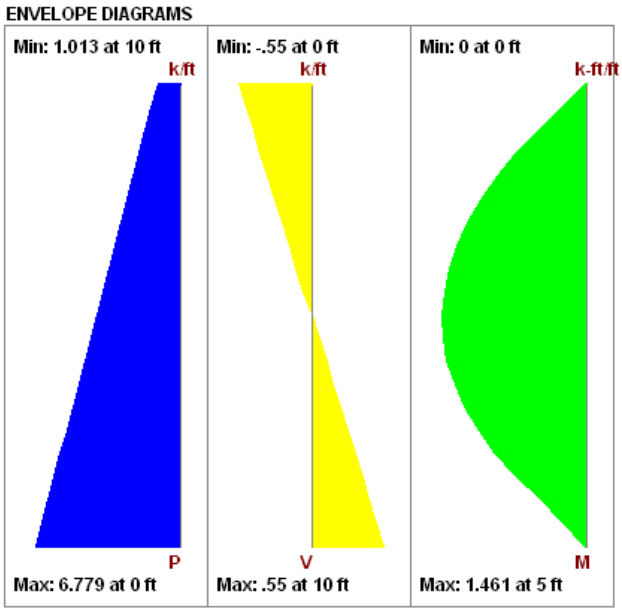
See the Masonry Wall - Design topic for information on a T-section analysis.

T-SECTION ANALYSIS

kd	: 1.609	in
Flange thickness	1.375	in
Web width	: 8.5	in

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 the summary of the code checks performed on the out-of-plane loaded slender wall.



AXIAL DETAILS

Pu / Pn	: 0.53	
Pu at max Mu	: 6.78	k/ft
Pn at max Mom	: 12.78	k/ft
H'/t	: 12.00	
Location	: 5.00	ft
Load Comb	: 2	

BENDING DETAILS

Mu / phi*Mn	: 0.54	
Mu, max	: 1.46	k-ft/ft
phi Mn	: 2.72	k-ft/ft
phi	: .9	
Location	: 5.00	ft
Load Comb	: 1	

DEFLECTION DETAILS

Dmax / DLimit	: 0.34	
Deflection Ratio	: H/416	
Delta max	: 0.29	in
Deflection Limit	: 0.84	in
Location	: 66.00	in
Load Comb	: 2	

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/ phi*Mn** represents the ratio of applied moment to moment capacity of the wall
- **Mu, max** is the out-of-plane bending moment at the controlling location of the wall.
- **phi Mn** is the out-of-plane moment capacity of the wall at the controlling location. It is calculated per Section 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 (Mu / phi*Mn) ratio. It is the location at which Mu and Pu are used to calculate phi Mn.
- **Load Combination** is the load combination which resulted in an Mu and Pu which yielded the highest (Mu / phi*Mn) ratio.

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*Mn	: Over-reinforced	
Mu, max	: -3.47	k-ft/ft
phi Mn	: 9.66	k-ft/ft
phi	: .9	
Location	: 0.00	ft
Load Comb	: 1	

Deflection Details

Dmax/Dlimit is a code check result. Thus, if the value is larger than one, your wall fails deflection criteria. **Delta max** is the calculated deflecting using the iterative slender wall design procedure detailed in the Masonry Wall - Design topic. The **Delta max** is the allowable deflection from 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 C/C OF REINFORCEMENT

Total Width	: 40	in	Agross	: 167.7	in2	As	: .307	in2
Eff Width	: 40	in	Igross	: 2283	in4	Ase	: .307	in2
Flange thick tf	: 1.375	in	fr	: 0.083	ksi	Rho Provided (%)	: .002	
Effective Thick te	: 4.192	in	Crack Mom, Mcr	: 3.041	k-ft	Rho Maximum (%)	: .005	
			c	: 0.543	in			
			n	: 21.48				
			Icracked	: 136.6	in4			

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 tf** refers to the thickness of the face shell whereas the **Effective Thick te** 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 **Ase** are reported, with the effective area based upon As + Pu/fy.

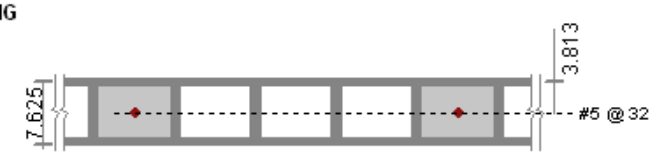
The gross steel ratio (rho gross) is calculated as the area of steel divided by the total area of the section.

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.

Masonry Detail Reports - Lintels

Lintels - Criteria / Materials / Geometry

The first section of the detail report echoes back the basic input parameters (Criteria, Materials, Geometry) entered by the user. An example is shown below:

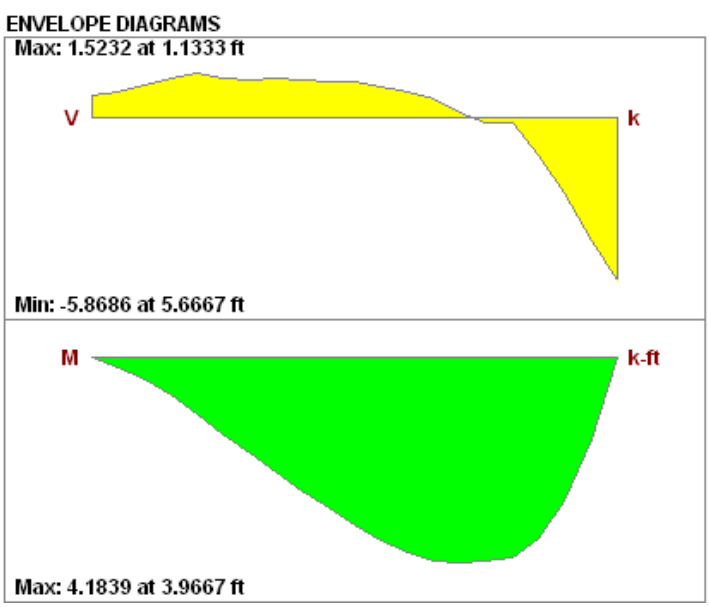
CRITERIA		MATERIALS			GEOMETRY		
Code	: ACI 530-11: ASD	Masonry fm	: 1.5	ksi	Dist to Top of Wall	: 3.167	ft
Special Insp	: Yes	Masonry Em	: 1350	ksi			
Type of Design	: ASD	Steel fy	: 60	ksi	Actual Length	: 5	ft
		Steel E	: 29000	ksi	Bearing Length	: 8	in
					Eff Length	: 5.667	ft
Stirrup Size	: #4	Beam Dead Wt	: .131	k/ft	Eff Width	: 9.625	in
Flex Steel	: 1-#5	Wall Dead Wt	: .098	ksf			
					Eff depth	: 12.5	in
					Total Depth	: 16	in

The geometry portion and the dead weight of the wall warrant further explanation

Attribute	Description
Beam Dead Weight	This is based on the user defined density for the Lintel / Opening.
Wall Dead Weight	This is based on the wall self weight or the self weight of the region immediately above the opening.
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.
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.

Lintel Detail Reports - Diagrams and Code Check Summary

The next section of the detail report provides the envelope shear and moment diagrams as well as a summary of the code checks for the Lintel.



SHEAR SUMMARY

Vu/(Vn*phi)	: .46	
Vu	: 5.869	k
Vn,total	: 15.956	k
Vn,masonry	: 15.956	k
Vn,steel	: 0	k
phi	: .8	

BENDING SUMMARY

Mu/(Mn*phi)	: .185	
Mu	: 4.184	k-ft
Mn	: 25.077	k-ft
Mn,masonry	: 3.802	k-ft
Mn,steel	: 21.275	k-ft
phi	: .9	

Lintel Detail Reports - Design Details

The next section of the detail report gives further details for the design checks performed on the lintel.

DESIGN DETAILS

BENDING DETAILS						SHEAR DETAILS		
Max Moment	: 3.945	k-ft	Steel Area As	: .307	in2	Max Shear	: 5.583	k
Location	: 2.3	ft	Per of steel p	: .003		Location	: 3.833	ft
Load Comb	: 1					Load Comb	: 1	

COMPRESSIVE STRESS BLOCK DETAILS								
Depth of Block	: 1.594	in	beta 1	: .8		Corresponding M	0	k-ft
N.A. Location	: 1.992	in	alpha	: .8		M / (V*d)	: .53	
						Tie Spacing	: Ties Not Reqd	

Mm	: 9.1057	k-ft	k	: .3261	
Ms	: 9.8442	k-ft	j	: .8913	

Code Check:

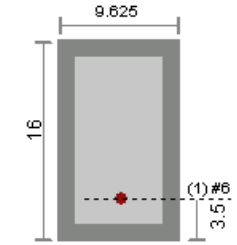
Limit State	Gov. LC	Required	Available	Unity Check	Result
Bending Design	1	17.165 k-ft	215.428 k-ft	0.089	PASS
M = 1.43 k-ft			Max moment		
Location = 3.333 ft			Governing location		
Mu = 1.43 k-ft					
Mn = 17.952 k-ft					
Mn masonry = 1.834 k-ft					
Mn steel = 16.119 k-ft					
phi = 0.9					
Shear Design	1	0.973 k	18.639 k	0.065	PASS
V = 0.973 k			Max shear force		
Location = 6.667 ft			Governing location		
Vu = 0.973 k					
Vn, total = 18.639 k					
Vn, masonry = 18.639 k					
Vn, steel = 0 k					
phi = 0.8					
Compressive Stress Block details					
Depth of block = 1.594 in					
N.A. Location = 1.992 in					
beta 1 = 0.8					
alpha = 0.8					

This **Bending Details** show the magnitude, location and load combination that correspond to the maximum moment used in the design as well as the area of flexural steel required (As) and the steel ratio (rho).

The **Compressive Stress Block Details** show the parameters used in calculating the strength of the lintel.

The **Shear Details** show the magnitude, location and load combination that correspond to the maximum shear used in the design, the M/(V*d) ratio for shear design, and the required spacing of ties (if applicable).

Lintel Detail Reports - Cross Section Detailing



This section of the detail report is meant to provide a visual confirmation to the user of the basic geometry and reinforcement provided in the lintel.

Wood Wall - Design

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

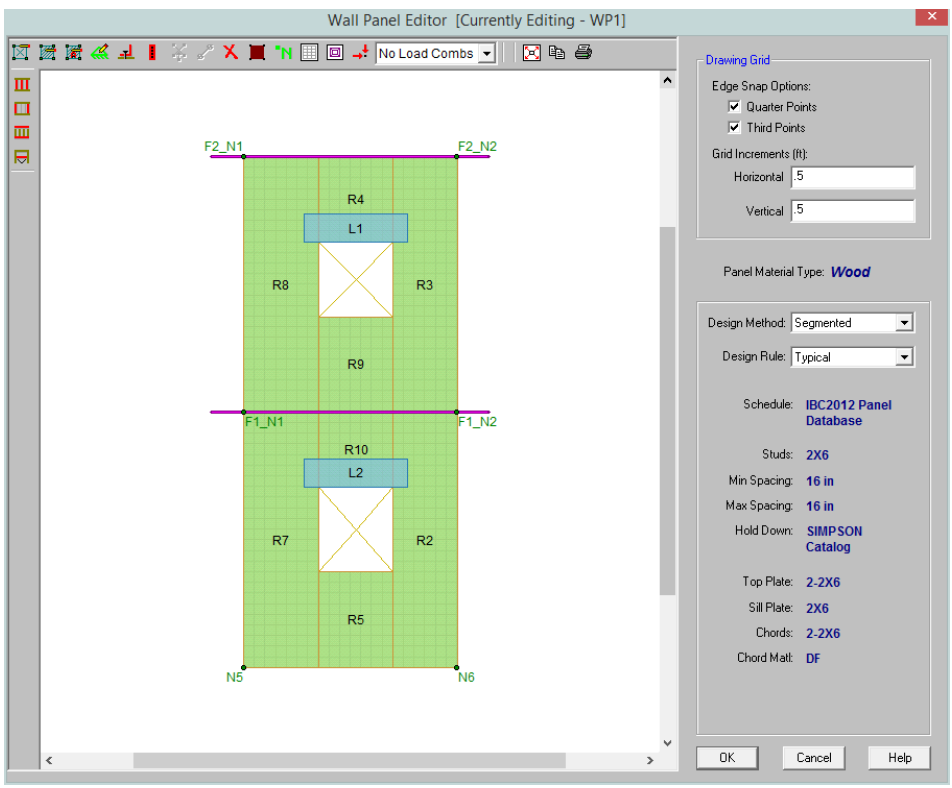
For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Wood Walls**.

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 **Wall Panel Editor** gives some specific information and options for modeling/analysis of wood walls.



Wood View Controls

Toggle Wall Studs Display allows you to turn the display of the studs on and off.

Toggle Wall Chords Display allows you to turn the display of wall panel region chords on and off.

Toggle Top/Sill Plate Display allows you to turn the display of the top/sill plates on and off.

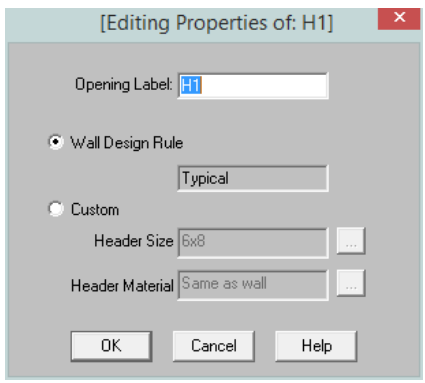
Toggle Opening Headers Display allows you to turn the display of headers on and off.

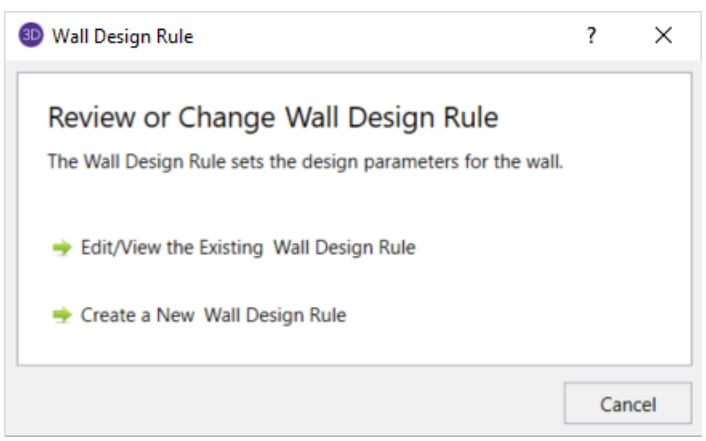
Design Rules

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

Creating Openings in Wood Walls with Headers

Within the **Wall Panel Editor** tab, you have the option of adding rectangular openings to wood wall panels. To draw an opening, select the **Create New Openings** button and then select two nodes or grid intersections which make up the two diagonal corners of your opening. When an opening is drawn, a header beam is automatically created above the opening. To view or edit the properties of a header beam go to the Wall Design Rules spreadsheet. In the case where you may have different opening headers in the same wall, you can define those different headers by double-clicking inside the boundary of the drawn opening. This will bring up the **Editing Properties** window for that particular header beam.





Label - This defines the name of this header and shows up in the results output for this header.

Wall Design Rule - This is the default setting here and is the default header information for all headers in the wall.

Custom - This allows you to define a header size and material that is different from the Wall Design Rule. You would only set a custom header if you have multiple headers in the same wall that are different sizes.

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 within the **Wall Panel Editor**.

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 **Add New Wood Hold Downs** button .

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

Straps represent the connection of the current wall panel to a wall panel below. To add straps to the base of your wall, first select the Add New Wood Straps button .

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

Note:

- All boundary conditions for wall panels should be defined in the wall panel editor. Adding external boundary conditions can create problems.
- Hold-downs and straps can only be added in RISA-3D.
- 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:

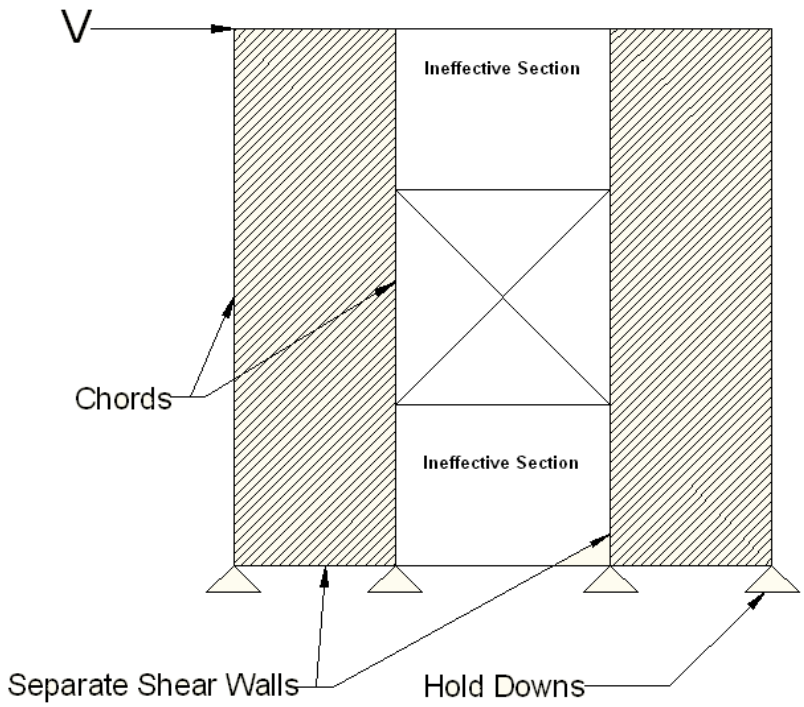
- The Canadian CSA O86 design code only supports the Segmented method, as described below.
- Because RISAFloor is only a gravity design program, it considers all design methods the same and is more of a Segmented design approach. If using RISAFloor together with RISA-3D, simply taking your model into RISA-3D will open up the Perforated and Force Transfer Around Openings design methods.

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.

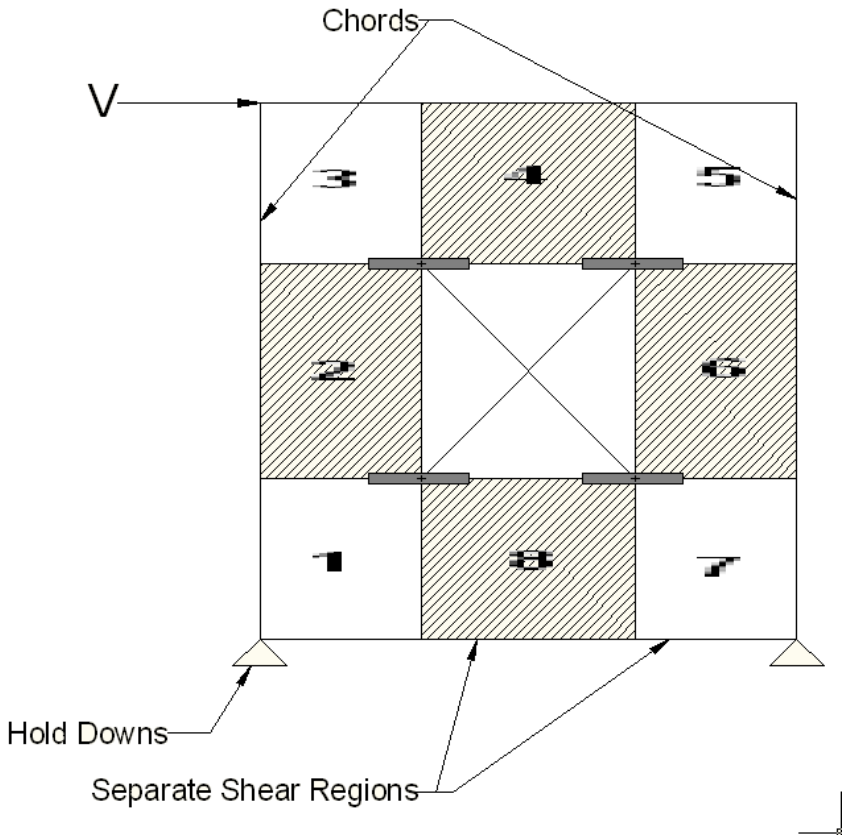
Note:

- 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 joints. This method essentially allows you to use the entire area of the wall (minus the opening) to resist the shear in the wall.



The basic assumptions made in the shear wall analysis are the following:

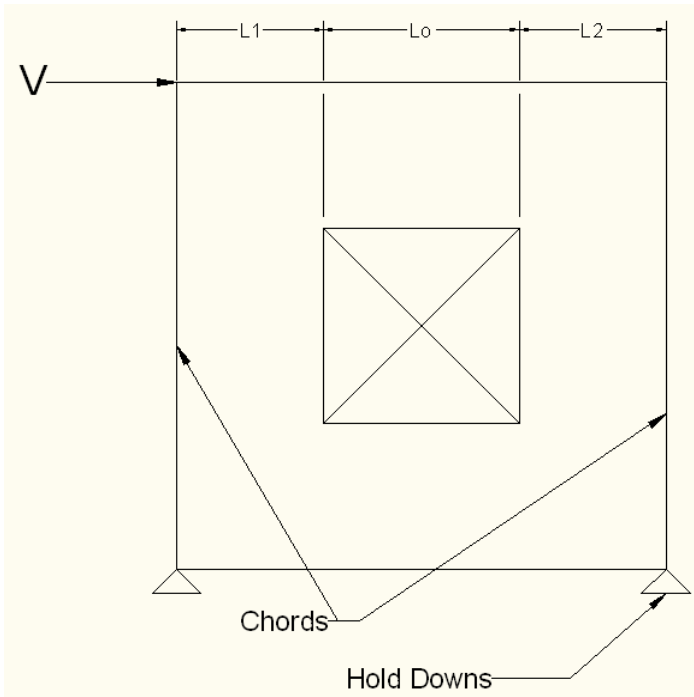
- The sheathing resists the shear forces. The average shear force in each block of the wall (numbered 1~8 as shown in the image above) is used as the controlling shear force in that location. The maximum shear in each of these locations will control the design of the wall. The program uses an area weighted average of the Fxy plate forces to determine the average shear for each block.
- The moment at the edge of each block that is above or below an opening is assumed to be transmitted across the opening interface by horizontal tension straps or compression blocks as shown in the image above. The required force is reported to the user, but the design and length of these elements is left to the engineer.
- The moment at the edge of each block that is to the right or left of an opening is assumed to be transmitted across the opening by tension straps or compression blocks. Since it is likely that the sheathing and king studs will be capable of transmitting these forces, these elements are not shown in the image above. However, these forces are reported so that the design of the studs and sheathing in these regions can be checked by the engineer to consider these effects.

Note:

- The program is limited in the automatic generation of regions for walls with multiple openings that are not aligned. Therefore, it is recommended that complex walls will multiple openings be simplified based on engineering 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:

$$T := \frac{V \cdot h}{C_o \cdot \Sigma L_i} \qquad v_{max} := \frac{V}{C_o \cdot \Sigma L_i}$$

Where:

$$\Sigma L_i := L1 + L2$$

Note:

- The perforated method of design also has many caveats that are given in Section 4.3.5.3 of the 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:

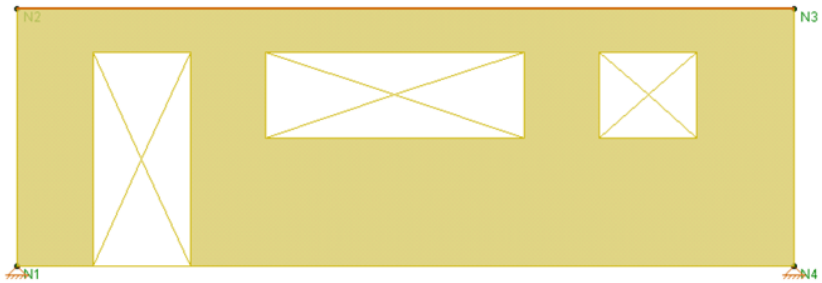
The NDS *Special Design Provisions for Wind & Seismic* lists Effective Shear Capacity Ratio (Co) values that are used in calculating the nominal shear capacity of perforated shear walls. Because the tabular values are limited to wall heights of 8' & 10', RISA instead uses equation (4.3-5) from the *2015 NDS SDPWS* (as shown below) or equation (4.3-6) from the *2021 NDS SDPWS* to calculate the Co factor for any height wall.

$$Co = \left(\frac{r}{3-(2*r)}\right) * \frac{Ltot}{\Sigma Li}$$

Where,

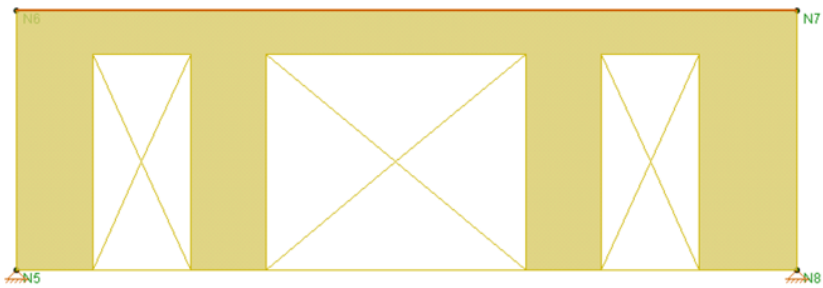
$$r = \frac{1}{1+\left(\frac{Ao}{h*\Sigma Li}\right)}$$

When using these equations, RISA takes Ao as the true area of the openings. However, Table 4.3.3.5 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.



Co = 0.84

VS.



Co = 0.53

(Full height openings assumption- matches NDS Table 4.3.3.4)

Note:

- When the assumption is made that all opening heights are equal to the maximum opening height then the equation produces values of Co that are within 1% for all the values shown in the NDS table.

C_o Limitation for Stacked Wall Panels

For wall panels that are stacked on one another, each wall panel has its own C_o value. These C_o values will be different if the openings in each wall are different. This is the source of a limitation in the program.

The program determines the design forces in each wall panel separately using the finite element solution. We then take those design forces and factor them for the C_o values. The problem is that the design forces for the bottom wall are affected by the C_o value for the upper wall. Thus, the forces that come down on a lower wall from an upper wall have been factored for the $C_o$from the upper wall. RISA-3D does not consider this. The program takes the forces from the finite element solution and uses the C_o value only for the wall in question.

Thus, the highest level in a stacked wall configuration will always use C_o in a correct manner. The lower wall(s), however, will be conservative if they have more openings than the upper wall(s) and can be unconservative if the upper wall(s) have more openings than the lower wall(s).

This limitation must be considered when designing Perforated walls that are stacked.

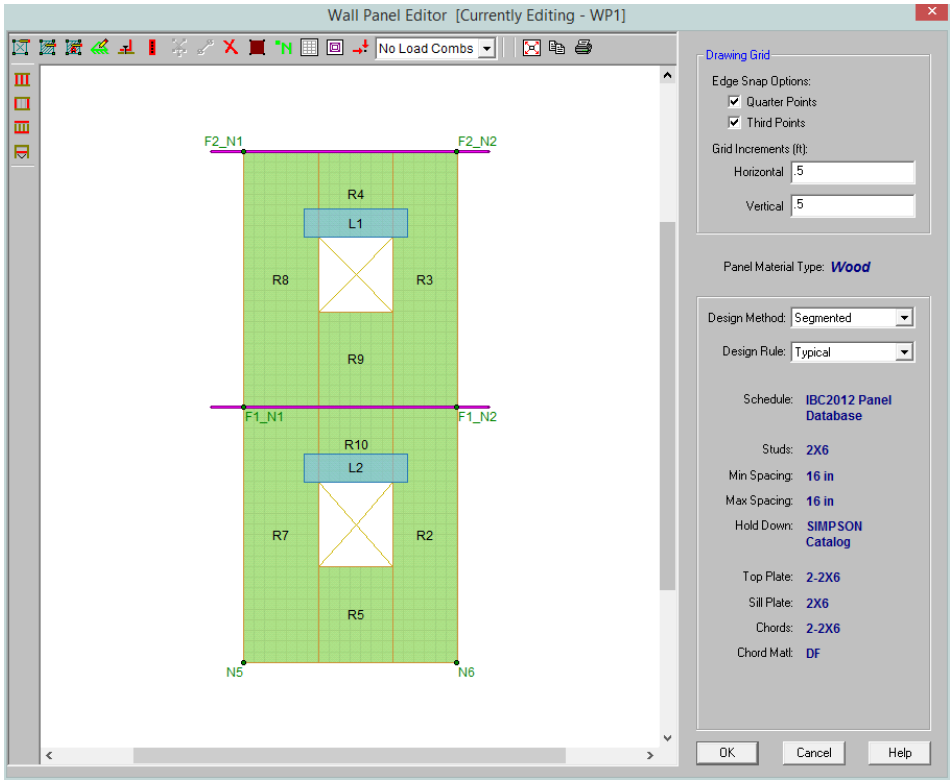
General Program Functionality and Limitations

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

Note:

- Hold downs and straps can not be added to wall panels in RISAFloor.
- Hold downs are only allowed to be added to the lower corners of the wall panel for the Perforated or Force Transfer Around Openings design methods.
- Hold downs are required at the corners of all full height regions in the wall panel for the Segmented design method.
- The Design Rule, Design Method and SSAF can be changed in either program at any time.

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.

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:

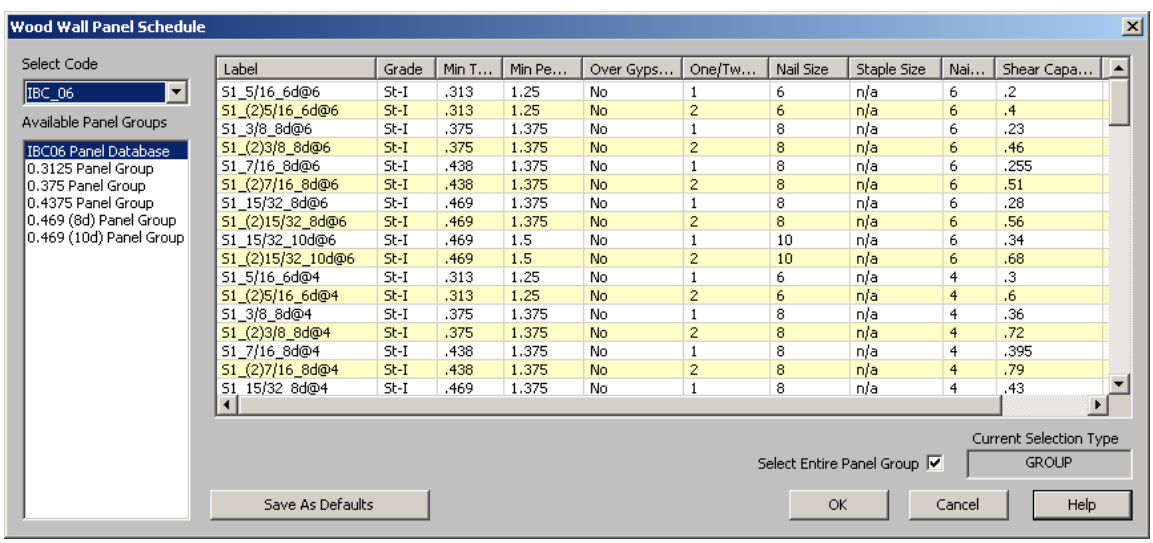
- For this calculation, stud height equals wall height minus the thickness of the sill plate and the top plate.
- The number of studs is calculated using the stud spacing specified in Design Rules.

Wood Wall Optimization

The program will optimize wood walls based on the required demand forces. The program can optimize:

- Stud spacing for axial design.
- Hold-downs for in plane design.

Panel Optimization



Label	Grade	Min T...	Min Pe...	Over Gyps...	One/Tw...	Nail Size	Staple Size	Nai...	Shear Capa...
S1_5/16_6d@6	St-I	.313	1.25	No	1	6	n/a	6	.2
S1_(2)5/16_6d@6	St-I	.313	1.25	No	2	6	n/a	6	.4
S1_3/8_8d@6	St-I	.375	1.375	No	1	8	n/a	6	.23
S1_(2)3/8_8d@6	St-I	.375	1.375	No	2	8	n/a	6	.46
S1_7/16_8d@6	St-I	.438	1.375	No	1	8	n/a	6	.255
S1_(2)7/16_8d@6	St-I	.438	1.375	No	2	8	n/a	6	.51
S1_15/32_8d@6	St-I	.469	1.375	No	1	8	n/a	6	.28
S1_(2)15/32_8d@6	St-I	.469	1.375	No	2	8	n/a	6	.56
S1_15/32_10d@6	St-I	.469	1.5	No	1	10	n/a	6	.34
S1_(2)15/32_10d@6	St-I	.469	1.5	No	2	10	n/a	6	.68
S1_5/16_6d@4	St-I	.313	1.25	No	1	6	n/a	4	.3
S1_(2)5/16_6d@4	St-I	.313	1.25	No	2	6	n/a	4	.6
S1_3/8_8d@4	St-I	.375	1.375	No	1	8	n/a	4	.36
S1_(2)3/8_8d@4	St-I	.375	1.375	No	2	8	n/a	4	.72
S1_7/16_8d@4	St-I	.438	1.375	No	1	8	n/a	4	.395
S1_(2)7/16_8d@4	St-I	.438	1.375	No	2	8	n/a	4	.79
S1_15/32_8d@4	St-I	.469	1.375	No	1	8	n/a	4	.43

The procedure that RISA uses for design optimization is fundamentally based upon the assumption that there is a 'cost' to shear capacity, and therefore the ideal panel design would have as little shear capacity as possible to meet code requirements. Once the program has determined the shear demand on the wall it will choose the most

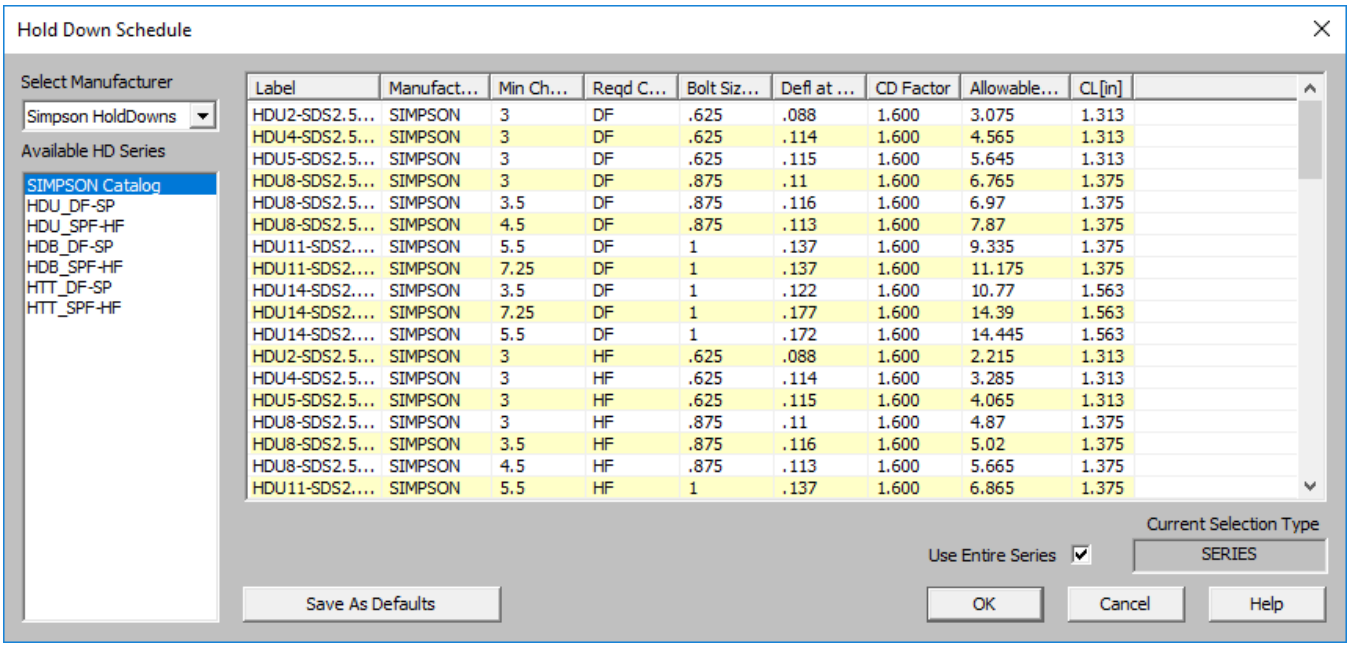
economical panel configuration based on that which has a **Shear Capacity** closest to, but not exceeding the shear demand.

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.

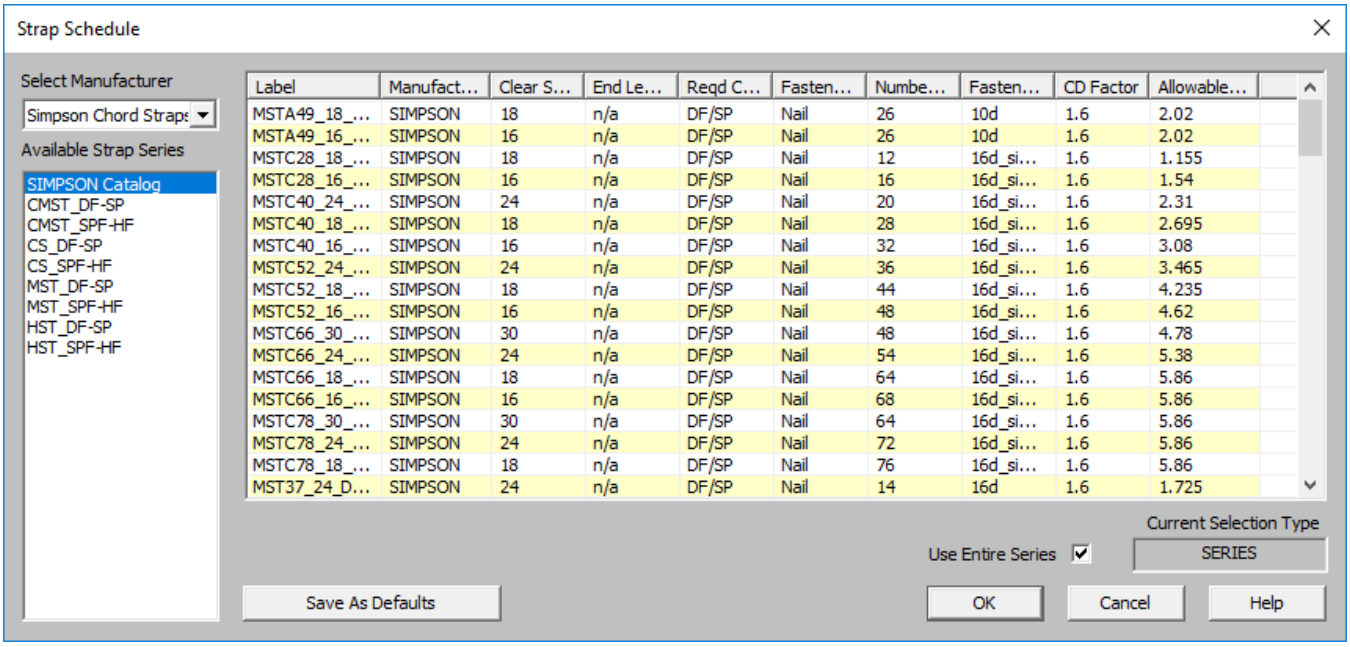
Hold Down Optimization



Label	Manufact...	Min Ch...	Reqd C...	Bolt Siz...	Defl at ...	CD Factor	Allowable...	CL[in]
HDU2-SDS2.5...	SIMPSON	3	DF	.625	.088	1.600	3.075	1.313
HDU4-SDS2.5...	SIMPSON	3	DF	.625	.114	1.600	4.565	1.313
HDU5-SDS2.5...	SIMPSON	3	DF	.625	.115	1.600	5.645	1.313
HDU8-SDS2.5...	SIMPSON	3	DF	.875	.11	1.600	6.765	1.375
HDU8-SDS2.5...	SIMPSON	3.5	DF	.875	.116	1.600	6.97	1.375
HDU8-SDS2.5...	SIMPSON	4.5	DF	.875	.113	1.600	7.87	1.375
HDU11-SDS2....	SIMPSON	5.5	DF	1	.137	1.600	9.335	1.375
HDU11-SDS2....	SIMPSON	7.25	DF	1	.137	1.600	11.175	1.375
HDU14-SDS2....	SIMPSON	3.5	DF	1	.122	1.600	10.77	1.563
HDU14-SDS2....	SIMPSON	7.25	DF	1	.177	1.600	14.39	1.563
HDU14-SDS2....	SIMPSON	5.5	DF	1	.172	1.600	14.445	1.563
HDU2-SDS2.5...	SIMPSON	3	HF	.625	.088	1.600	2.215	1.313
HDU4-SDS2.5...	SIMPSON	3	HF	.625	.114	1.600	3.285	1.313
HDU5-SDS2.5...	SIMPSON	3	HF	.625	.115	1.600	4.065	1.313
HDU8-SDS2.5...	SIMPSON	3	HF	.875	.11	1.600	4.87	1.375
HDU8-SDS2.5...	SIMPSON	3.5	HF	.875	.116	1.600	5.02	1.375
HDU8-SDS2.5...	SIMPSON	4.5	HF	.875	.113	1.600	5.665	1.375
HDU11-SDS2....	SIMPSON	5.5	HF	1	.137	1.600	6.865	1.375

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



Label	Manufact...	Clear S...	End Le...	Reqd C...	Fasten...	Numbe...	Fasten...	CD Factor	Allowable...
MSTA49_18_...	SIMPSON	18	n/a	DF/SP	Nail	26	10d	1.6	2.02
MSTA49_16_...	SIMPSON	16	n/a	DF/SP	Nail	26	10d	1.6	2.02
MSTC28_18_...	SIMPSON	18	n/a	DF/SP	Nail	12	16d_si...	1.6	1.155
MSTC28_16_...	SIMPSON	16	n/a	DF/SP	Nail	16	16d_si...	1.6	1.54
MSTC40_24_...	SIMPSON	24	n/a	DF/SP	Nail	20	16d_si...	1.6	2.31
MSTC40_18_...	SIMPSON	18	n/a	DF/SP	Nail	28	16d_si...	1.6	2.695
MSTC40_16_...	SIMPSON	16	n/a	DF/SP	Nail	32	16d_si...	1.6	3.08
MSTC52_24_...	SIMPSON	24	n/a	DF/SP	Nail	36	16d_si...	1.6	3.465
MSTC52_18_...	SIMPSON	18	n/a	DF/SP	Nail	44	16d_si...	1.6	4.235
MSTC52_16_...	SIMPSON	16	n/a	DF/SP	Nail	48	16d_si...	1.6	4.62
MSTC66_30_...	SIMPSON	30	n/a	DF/SP	Nail	48	16d_si...	1.6	4.78
MSTC66_24_...	SIMPSON	24	n/a	DF/SP	Nail	54	16d_si...	1.6	5.38
MSTC66_18_...	SIMPSON	18	n/a	DF/SP	Nail	64	16d_si...	1.6	5.86
MSTC66_16_...	SIMPSON	16	n/a	DF/SP	Nail	68	16d_si...	1.6	5.86
MSTC78_30_...	SIMPSON	30	n/a	DF/SP	Nail	64	16d_si...	1.6	5.86
MSTC78_24_...	SIMPSON	24	n/a	DF/SP	Nail	72	16d_si...	1.6	5.86
MSTC78_18_...	SIMPSON	18	n/a	DF/SP	Nail	76	16d_si...	1.6	5.86
MST37_24_D...	SIMPSON	24	n/a	DF/SP	Nail	14	16d	1.6	1.725

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.

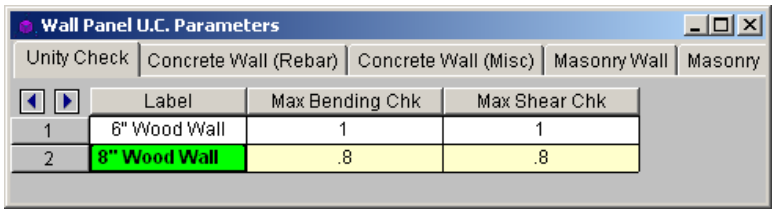
For experienced users who have more specific limitations in terms of the designs they would like to see, user-defined **Groups** (or families) are the solution. For example, an engineer who prefers to use only one sheathing thickness, or one nail type can create a custom Group that contains only the 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



	Label	Max Bending Chk	Max Shear Chk
1	6" Wood Wall	1	1
2	8" Wood Wall	.8	.8

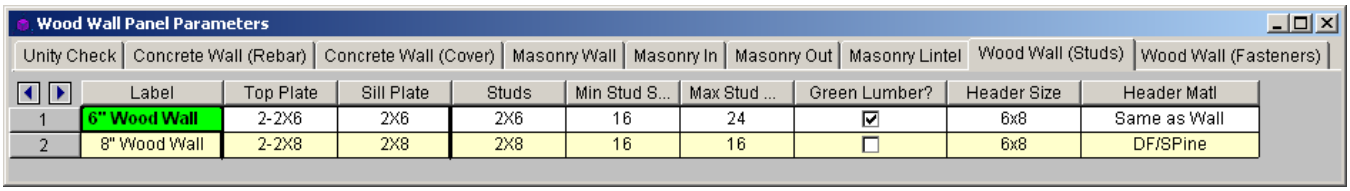
Max Bending Chk

This will define the hold-down maximum code check value. Typically this will always be 1.0, but this 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

This will define the shear panel selection maximum code check value. Typically this will always be 1.0, but this 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)



	Label	Top Plate	Sill Plate	Studs	Min Stud S...	Max Stud ...	Green Lumber?	Header Size	Header Matl
1	6" Wood Wall	2-2X6	2X6	2X6	16	24	☑	6x8	Same as Wall
2	8" Wood Wall	2-2X8	2X8	2X8	16	16	☐	6x8	DF/SPine

Top Plate

Use the Top Plate column to specify the member to be used as a top plate for your wall. A top plate is a member that runs continuously along the top of the wall studs. Note that you can use multiple plies of nominal lumber, or custom shapes.

Sill Plate

Use the **Sill Plate** column to specify the member to be used as a sill plate for your wall. A sill plate is a member that runs continuously along the bottom of the wall studs. Note that you can use multiple plies of nominal lumber, or custom shapes.

Studs

Use the Studs column to specify the member to be used for studs in your wall. Studs are vertical members in the wall, attached to the sill plate at the bottom and the top plate at the top. Note that you can use multiple plies of nominal lumber, or custom shapes.

Min/Max Stud Space

You may specify a minimum and maximum spacing of wall studs. The program can then optimize the stud spacing based on axial design only. For information on how the optimization works, see the Wood Wall - Design topic.

- If you specify the maximum and minimum stud spacing as the same value, then we will use that value exclusively.
- Out of plane design is not performed for wood walls, so any optimization is based only on axial forces in the studs.

Green Lumber

NDS Design: Check this box if your moisture content is greater than 19%. The program will then multiply the Ga value of the shear panel by 0.5 per Note 5 of Tables 4.3A and 4.3B of the NDS SDPWS.

CSA O86 Design: Check this box if you have wet service conditions. This will effect the Service Condition Factors, Ks per CSA O86-14 Table 6.4.2.

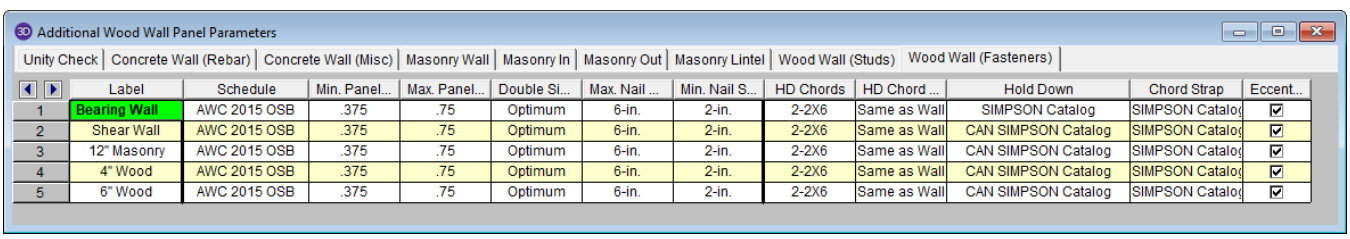
Header Size

This defines the default header size for all openings. Note that this can be modified in the Wall Panel Editor by double-clicking the opening and choosing **Custom**.

Header Matl

By default we will use the same material for the header as we are for the studs, chords, etc. However, you can change the material here.

Design Rules - Wood Wall (Fasteners)


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)

	Label	Schedule	Min. Panel...	Max. Panel...	Double Si...	Max. Nail ...	Min. Nail S...	HD Chords	HD Chord ...	Hold Down	Chord Strap	Eccent...
1	Bearing Wall	AWC 2015 OSB	.375	.75	Optimum	6-in.	2-in.	2-2X6	Same as Wall	SIMPSON Catalog	SIMPSON Catalog	☑
2	Shear Wall	AWC 2015 OSB	.375	.75	Optimum	6-in.	2-in.	2-2X6	Same as Wall	CAN SIMPSON Catalog	SIMPSON Catalog	☑
3	12" Masonry	AWC 2015 OSB	.375	.75	Optimum	6-in.	2-in.	2-2X6	Same as Wall	CAN SIMPSON Catalog	SIMPSON Catalog	☑
4	4" Wood	AWC 2015 OSB	.375	.75	Optimum	6-in.	2-in.	2-2X6	Same as Wall	CAN SIMPSON Catalog	SIMPSON Catalog	☑
5	6" Wood	AWC 2015 OSB	.375	.75	Optimum	6-in.	2-in.	2-2X6	Same as Wall	CAN SIMPSON Catalog	SIMPSON Catalog	☑

Schedule

You can select the Code, and Panel Group you would like to use for design optimization. By unchecking the **Select Entire Panel Group** box an individual panel type may be assigned. For information on how the optimization works, see the Wood Wall - Design topic. For more information on this schedule, as well as information on how to edit or create your own custom schedule, see Appendix F-Wood Design Databases

Min/Max Panel Thick

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

Double Sided

You can choose whether you want the program to force sheathing on only one side of the panel, both sides, or to choose the optimum based on weight.

Max/Min Nail Spacing

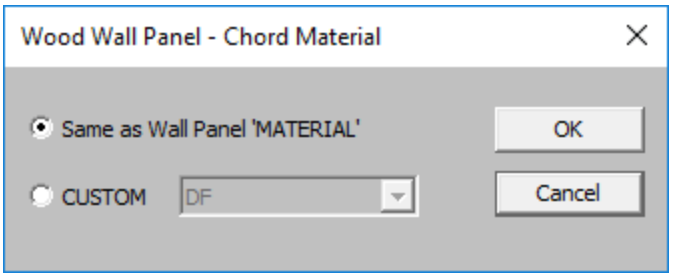
These values set minimums and maximums for the spacing of the nails that fasten the sheathing to the boundary members (top plate, sill plate, hold down chords). Note that a 12" spacing is assumed for all field nailing (nails fastening the sheathing to the internal studs).

HD Chords

You can choose what member size you would like to use for the Hold Down Chords (Posts) at both ends of the wall panel.

HD Chord Material

You can specify whether the hold down chords are of the same material as the wall, or another material.



Hold Down

You can select the Code, and Hold Down Series you would like to use for design optimization. By unchecking the **Use Entire Series** box you my select an individual hold down product to be assigned. For information on how the optimization works, see the Wood Wall - Design topic. For more information on this schedule, as well as information on how to edit or create your own custom schedule, see Appendix F-Wood Shear Wall Files

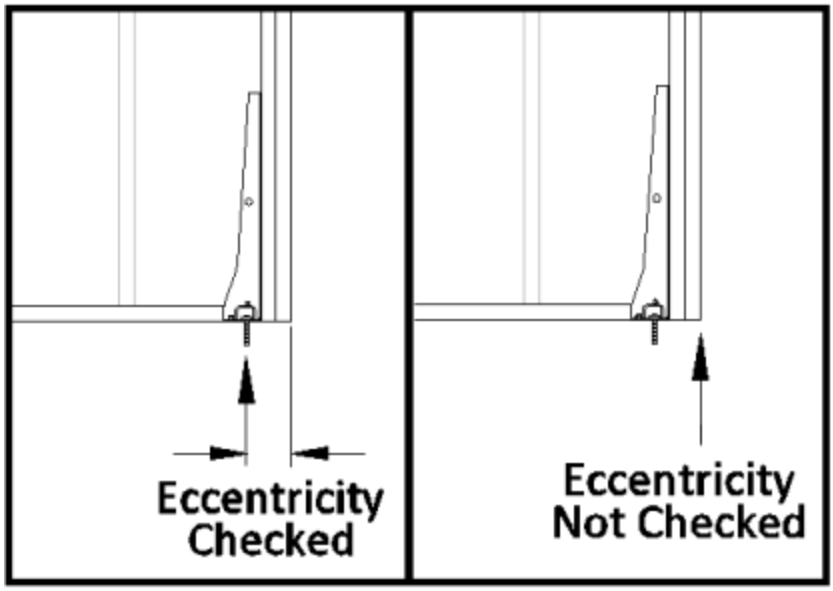
Chord Strap

You can select the Manufacturer and Strap Series you would like to use for design optimization. By unchecking the **Use Entire Series** box you my select an individual strap 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).



Note:

- **Eccentricity** option is only applicable to hold-downs, not straps.
- **Eccentricity** option does not affect the capacity of the wall sheathing.

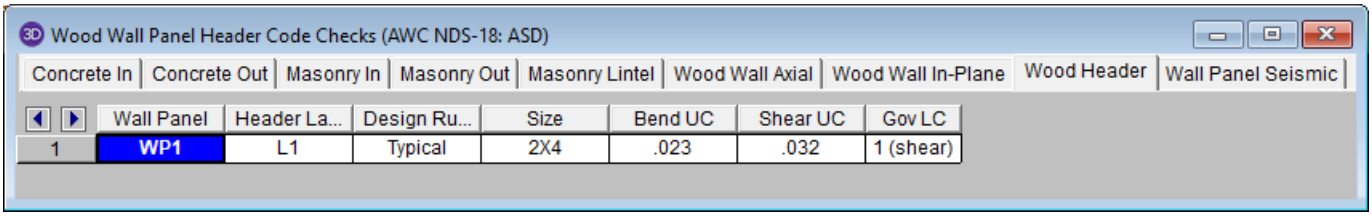
Wood Wall Results

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**.

Wood Wall Results Spreadsheets

See Wall Panels - Results for this topic.



Wood Wall Self Weight

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

Wood Wall Detail Reports

The detail report gives detailed information about the wall design. The detail reports are specifically molded to the type of design specified. Here we will walk through how to access the different information for each of the types of design: Segmented, Perforated and Force Transfer Around Openings.

Note:

- RISAFloor only considers the **Segmented** design method. If using RISAFloor together with RISA-3D, simply taking your model into RISA-3D will open up the **Perforated** and **Force Transfer Around Openings** design methods.
- Many of the values for design checks seen below are not performed in RISAFloor as it is strictly a gravity design program.

Accessing the Detail Reports and the Specific Windows

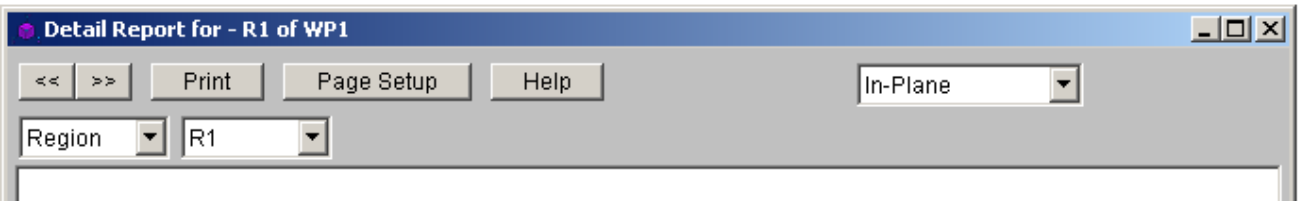
Once you have a solved model, the detail reports become available. They are accessible in two ways:

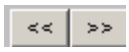
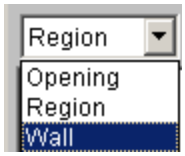
1. If you have the Wall Results spreadsheet open, you can right-click on a line in the spreadsheet and select **Detailed Report**. there will be a button at the top of the screen: Detail Report for Current Item. This will open up the detail report window.
2. If you are in a graphic view of your model, there is a Detail button that will open up the detail report window.

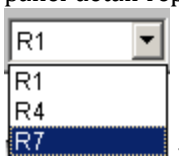
Note:

- Detail report information is not available for an envelope solution.

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



-  - Allows you to click between the different wall panels in your model.
- - This 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.
- 

- This dropdown list allows you to select between the three different parts of the wood wall panel detail report. Below we will explain the importance of each of these sections.
- 

- This dropdown list allows you to select between different regions or headers defined within the individual wall panel.

The Wall detail report gives an overall summary of your wall, complete with governing code checks and opening information. The Opening detail report gives information to the header design for the opening as well as detailed information for the **Force Transfer Around Openings** method. The Region detail report only applies for **Segmented** walls. Below we have give detailed information on each type of design: Segmented, Perforated and Force Transfer Around Openings.

Segmented Method

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

Wall Window

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

Input Echo

GENERAL

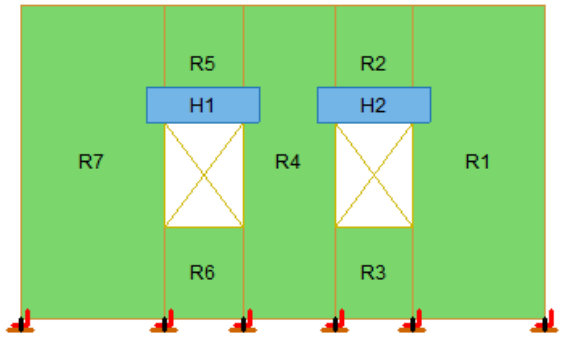
Code	:	AWC NDS-15:ASD
Design Method	:	Segmented
Wall Material	:	DF
Panel Schedule	:	IBC06-09 Panel Data...
Sel. Shear Panel	:	RS_7/16_8d@6

GEOMETRY

Total Height	:	12	ft
Total Length	:	20	ft
Max H/W Ratio	:	3.43	
K	:	1.00	

MATERIALS

Description	Material	Size
Top Pl	DF	2-2X6
Sill	DF	2X6
Wall Stud	DF	2X6
Chord	DF	2-2X6



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

Design Details

DESIGN DETAILS

REGION INFORMATION

Full-Height Region Label	Stud Required Cap	Stud Provided Cap	Stud UC	Stud LC
R1	.621	7.616	.082	2
R4	2.901	7.616	.381	2
R7	2.926	7.616	.384	2
R10	2.071	7.616	.272	2

OPENING INFORMATION

Header Label	Header Size	Shear UC	Bending UC	Governing LC
H1	6x8	.072	.047	2
H2	6x8	.142	.094	2
H3	6x8	.208	.081	2

The **Region Information** gives the tabulated results of all of the full-height regions in your wall.

The **Opening Information** gives the label, size, shear and bending code checks and the governing LC for each header in the wall.

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.

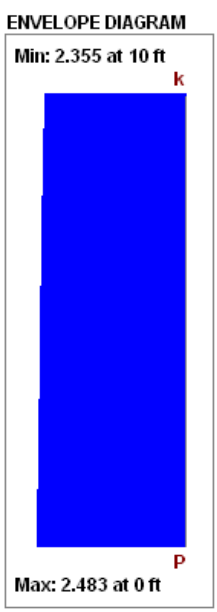
CRITERIA		MATERIALS		GEOMETRY		
Code	: AWC NDS-15:ASD	Wall Studs	: DF	Total Height	: 12	ft
		Stud Size	: 2X6	Total Length	: 5	ft
Wall Material	: DF	Chord Material	: DF	Region H/W	: 2.40	
		Chord Size	: 2-2X6	Cap. Adj. (2w/h)	: 1.00	
		Top Pl & Sill	: DF	Stud Spacing	: 16	in
		Top Pl Size	: 2-2X6	K	: 1.00	
		Sill Pl Size	: 2X6			

Criteria	Description
Code	Gives the code used to design your wall panel.
Wall Material	Specifies the wood type assigned to the entire wall

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

Geometry	Description
Total Height	This is the height of the wall panel region
Total Length	This is the length of the wall panel region
Region H/W Ratio	This is the ratio of wall height to length, using the minimum wall height
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 SDPWS section 4.3.3.4 Exception 1.
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



DESIGN SUMMARY

STUDS
Required Cap : .621 k
Provided Cap : 7.62 k
Ratio : .082
Governing LC : 2

Envelope Diagrams

These diagrams show the axial forces in the wall.

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.

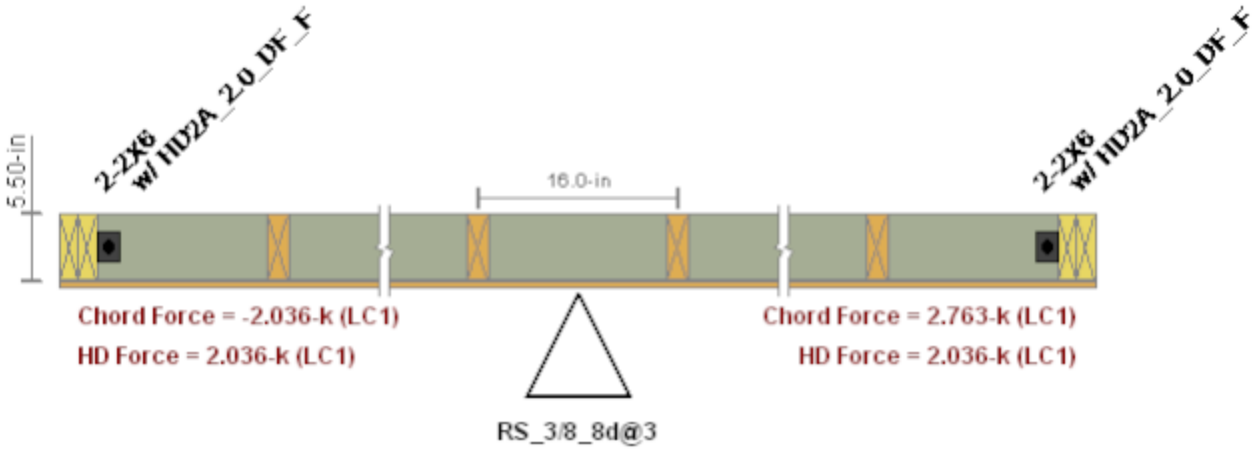
Design Summary

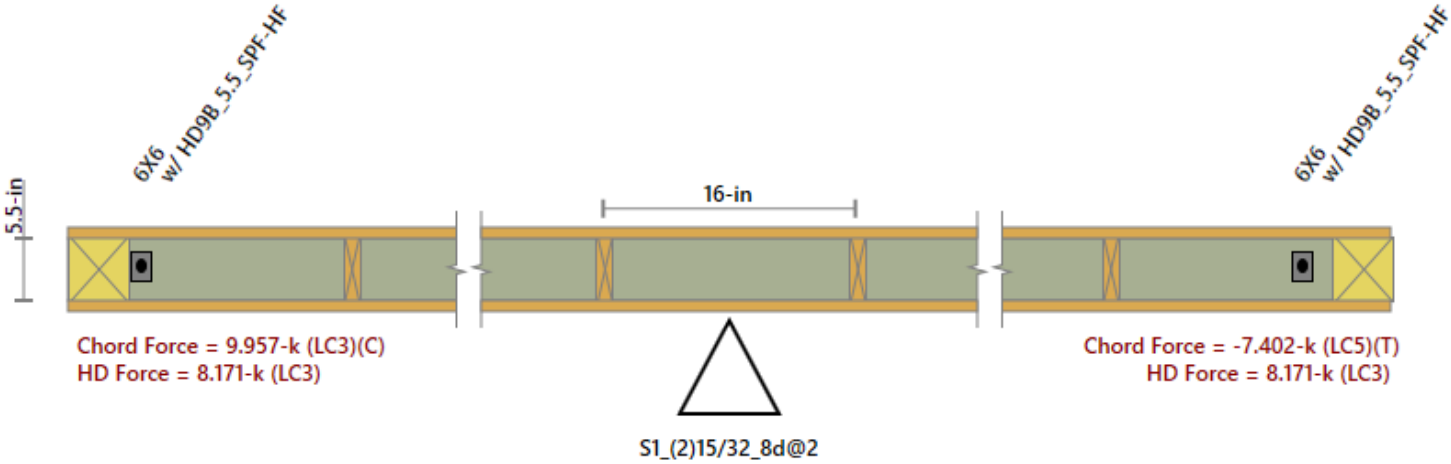
This portion gives you the capacity and strength values at the section in the wall where the combined check is maximum, as well as the governing load combination. Much of this information is also reported in the **Wood Wall Panel Design** spreadsheets.

Studs

The provided capacities of these members are calculated using the standard provisions for tension/compression members. These members are assumed to be fully braced in the weak axis, and unbraced in the strong axis. For more information on the chord force calculations, see the Wood Wall - Design topic.

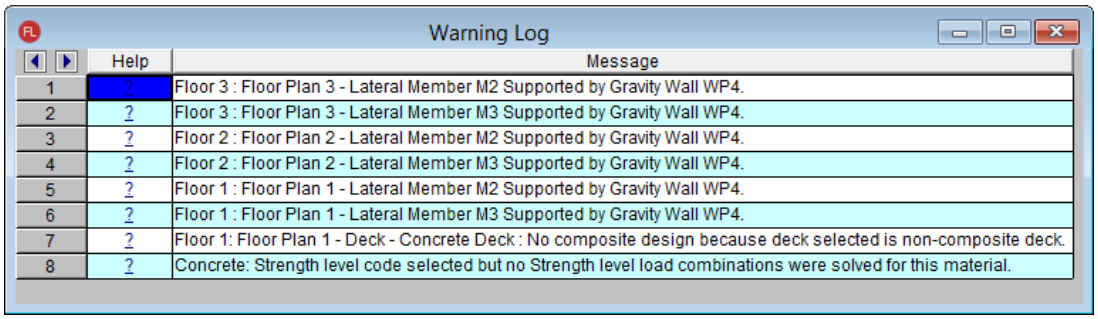
Cross Section Detailing





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 chord sizes/forces and hold down or strap designations/forces are shown at either end.

Warning Log



Warning Log

	Help	Message
1	?	Floor 3 : Floor Plan 3 - Lateral Member M2 Supported by Gravity Wall WP4.
2	?	Floor 3 : Floor Plan 3 - Lateral Member M3 Supported by Gravity Wall WP4.
3	?	Floor 2 : Floor Plan 2 - Lateral Member M2 Supported by Gravity Wall WP4.
4	?	Floor 2 : Floor Plan 2 - Lateral Member M3 Supported by Gravity Wall WP4.
5	?	Floor 1 : Floor Plan 1 - Lateral Member M2 Supported by Gravity Wall WP4.
6	?	Floor 1 : Floor Plan 1 - Lateral Member M3 Supported by Gravity Wall WP4.
7	?	Floor 1: Floor Plan 1 - Deck - Concrete Deck : No composite design because deck selected is non-composite deck.
8	?	Concrete: Strength level code selected but no Strength level load combinations were solved for this material.

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.

You can view the warning log by clicking on the Spreadsheets menu item and then clicking on the Warning Log selection, or by clicking on Warning Log on Results toolbar.

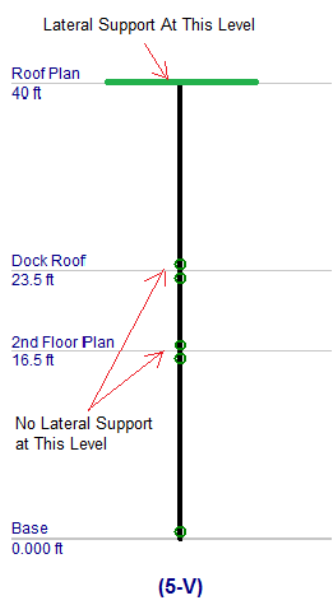
Pressing the **?** hyperlink will open the help file to give further explanation on this item.

Here is a common warning which could require some additional explanation:

Column Stack XX has more than one shear splice in between braces.

If you double-click on the column referenced you will open the **Column Stack Manager** and see that the column has too many shear splices. This creates an unstable situation. Note that a column is only braced laterally at a floor level if it either passes through a diaphragm at that floor, or has members framing into it at that floor.

Note the image below. This column has two shear splices in between lateral support locations (the Roof Plan and Base levels). This column is thus unstable. To correct this you would need to eliminate one or all shear splices in the column stacks, which can be done through the Column Stack Manager.



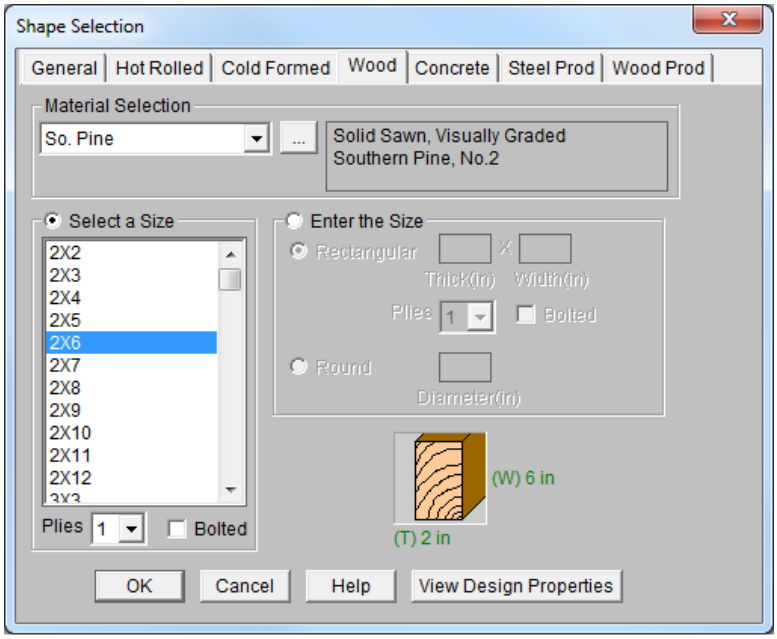
*Please refer to the **Help File** in RISAFloor for detailed information about each error message. The Help File may be accessed by selecting **Help Topics** from the **Help Menu**. Choose the **Warning Log** topic*

Wood - Database

The Wood Database may be accessed by clicking the **Shape Database** button on the **RISA** toolbar and then clicking the **Wood** tab of the **Shape Database** dialog.

Note:

- See Wood Products for information on the Wood Products Database.



Wood Material/Shape Selection

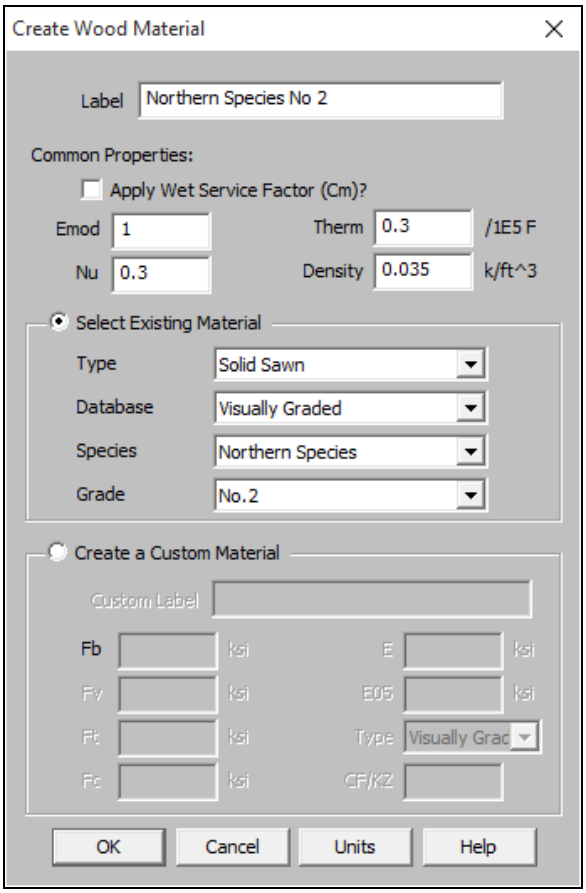
Unlike any other material, the design properties for wood members depends both on the material selection as well as the shape selection. Therefore, the **Wood** tab of the **Shape Selection** database contains shape and material selections.

Material Selection

The **Material Selection** menu reads in material options that are currently listed on the **Wood** tab of your Materials spreadsheet. You may select one of these existing materials, or click on the **Add Material** button to open the **Create Wood Material** dialog which will allow you to add in a new material without having to go to the **Materials** spreadsheet.

Create Wood Material dialog

The **Create Wood Material** dialog allows you to create or edit a material that will then be automatically added to your Materials spreadsheet. This allows you to enter/select this information directly from the **Draw Members** dialog without having to first manually enter the information in the **Materials** spreadsheet.



- Click on the **Select Existing Material** button to select a material using the drop down menus to select a solid sawn, glulam, SCL, or already created custom material from the material databases.
- Click on the **Create a Custom Material** 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,**Select a Size** menu or by using the **Enter the Size** option (typically referred to as Full Sawn).

- The **Select a Size** menuis 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 will automatically reduce the shape properties to their net sizes. If the CSA O86 design code is selected, a list of *actual* metric sizes will be shown.
- The **Enter the Size** optionwill allow you to 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 may 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 may 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 many 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 many 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 click on the **View Design** properties button to view the un-adjusted wood design values (Fb, Fv, Ft, etc.) for the selected combination. 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

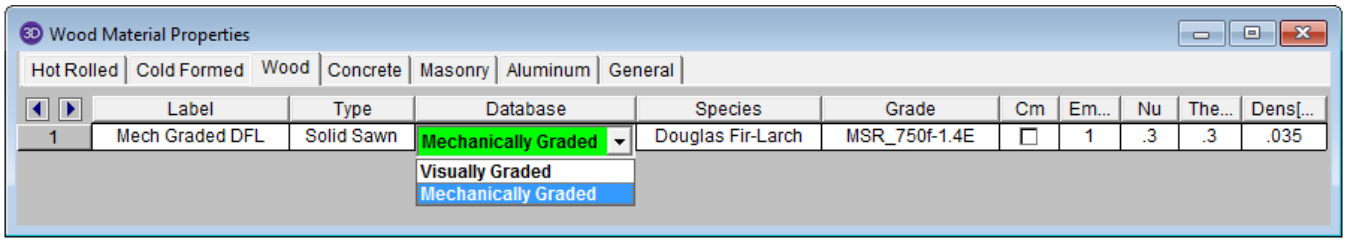
The program includes five tables of available glulam materials: NDS Tables 5A - 5D and CSA O86 Table 6.3. The values in NDS tables 5B & 5D depend on the number of laminations in the glulam. Therefore you will see three listings (2 Laminations, 3 Laminations, or 4 Laminations) for each material. If you 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 lamintations.
- 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.



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	-

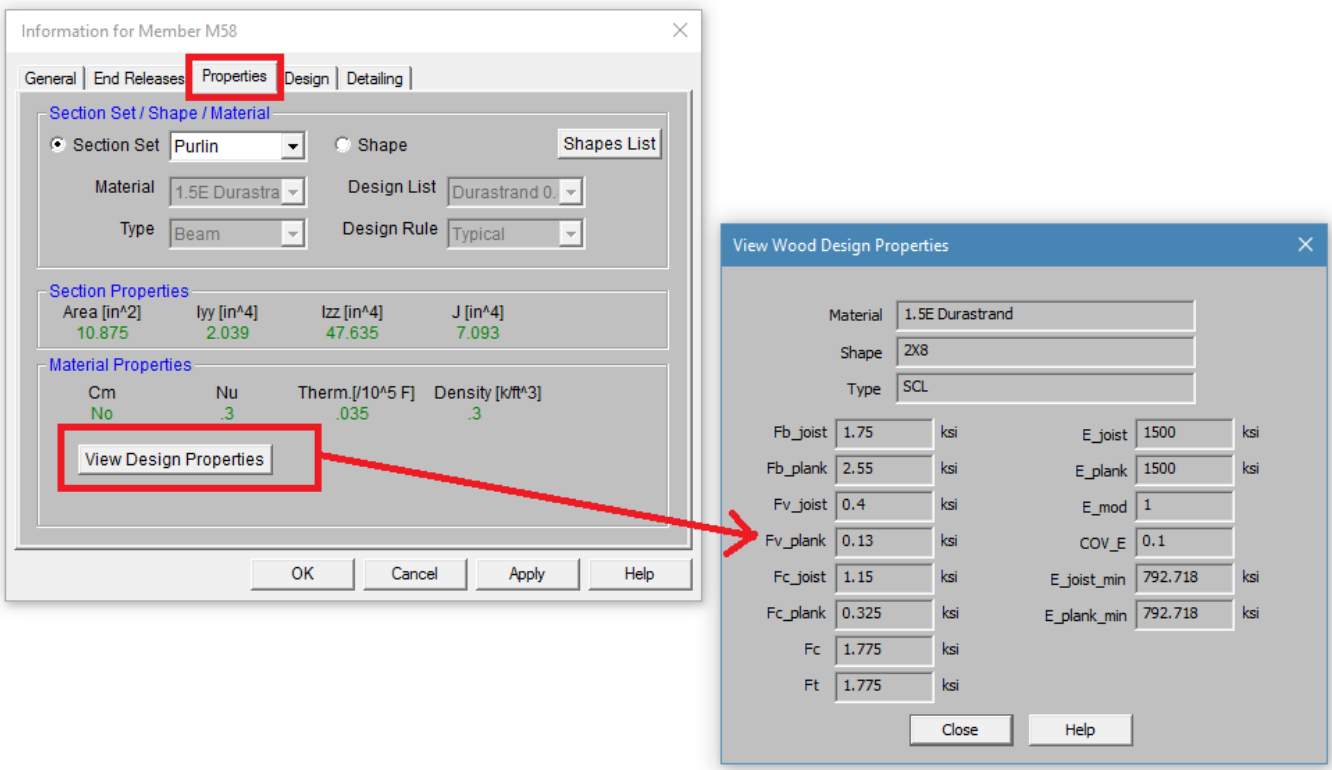
Footnote Limitations:
Each of the above reports includes tables with design values which were used to create the RISA-3D and RISAFloor SCL design property material databases. However, due to the difficulty of design variation, the following limitations are present:

- 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 View Design Properties button on the Properties tab of the Member Information dialogs.



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 checking the **Use Full Sawn Size** check box and entering the exact dimensions of the member in the boxes below. This applies to regular wood species, custom wood species, Structural Composite Lumber (SCL), and Glu-Lam members. The member thickness should be entered in the box on the left and the member width should be entered in the box on the right.

Wood - Design

Full code checking can be performed on Dimension Lumber and Post and Timber size wood shapes based on the following codes:

- The 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.

Information on the design of "Wood Products", such as wood I-joists, may be found in the Wood Products topic.

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 Table5A are always assumed to have the special tension laminations. Therefore, the Fbx value is not reduced.
- RISA is NOT applying any of the footnotes to Table 5A and 5C at this time except for the following:
    - Footnote #1 from Table 5A - For balanced materials Fbx- shall equal Fbx+ for the stress class.
    - Footnote #3 from Table 5A - Fvx and Fvy 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

To use a custom wood material that is not part of the standard NDS or CSA O86 databases, you will need to define the custom design properties. This can be done through the **Custom Wood Species** spreadsheet or directly through the **Materials** spreadsheet.

Custom Wood Properties Spreadsheet

To access the **Custom Wood Properties** spreadsheet, select **Custom Wood Species** from the **Spreadsheets** menu.

Custom Wood Properties

	Label	Fb [psi]	Ft [psi]	Fv [psi]	Fc [psi]	E [ksi]	E05 [ksi]	Type	CF
1	LVL_Microllam_1.9E_2600F	2600	1555	285	2510	1900	1900	SCL	
2	PSL_Parallam_2.0E_2900F	2900	2025	290	2900	2000	2000	SCL	
3	12F-V1 Hardwood	600	600	125	800	1000	1000	Glulam	
4	2005 NDS M-12 DFL	1600	850	180	1675	1600	1000	MEL	1.3

- **Fb** - Bending Stress Limit
- **Ft** - Tension Stress Limit
- **Fv** - Shear Stress Limit
- **Fc** - Compression (Parallel to Grain) Stress Limit
- **E** - Modulus of Elasticity
- **E05** - Fifth Percentile Modulus of Elasticity (for Canadian CSA O86 design only, ignored for NDS design)
- **Type** - Wood Type, used for determination of design variables (COV_E, E_{min}, c, etc.) and design calculations.
- **CF/KZ** - Wood Size Factor. The program will default this to 1.0 unless manually entered by the user.

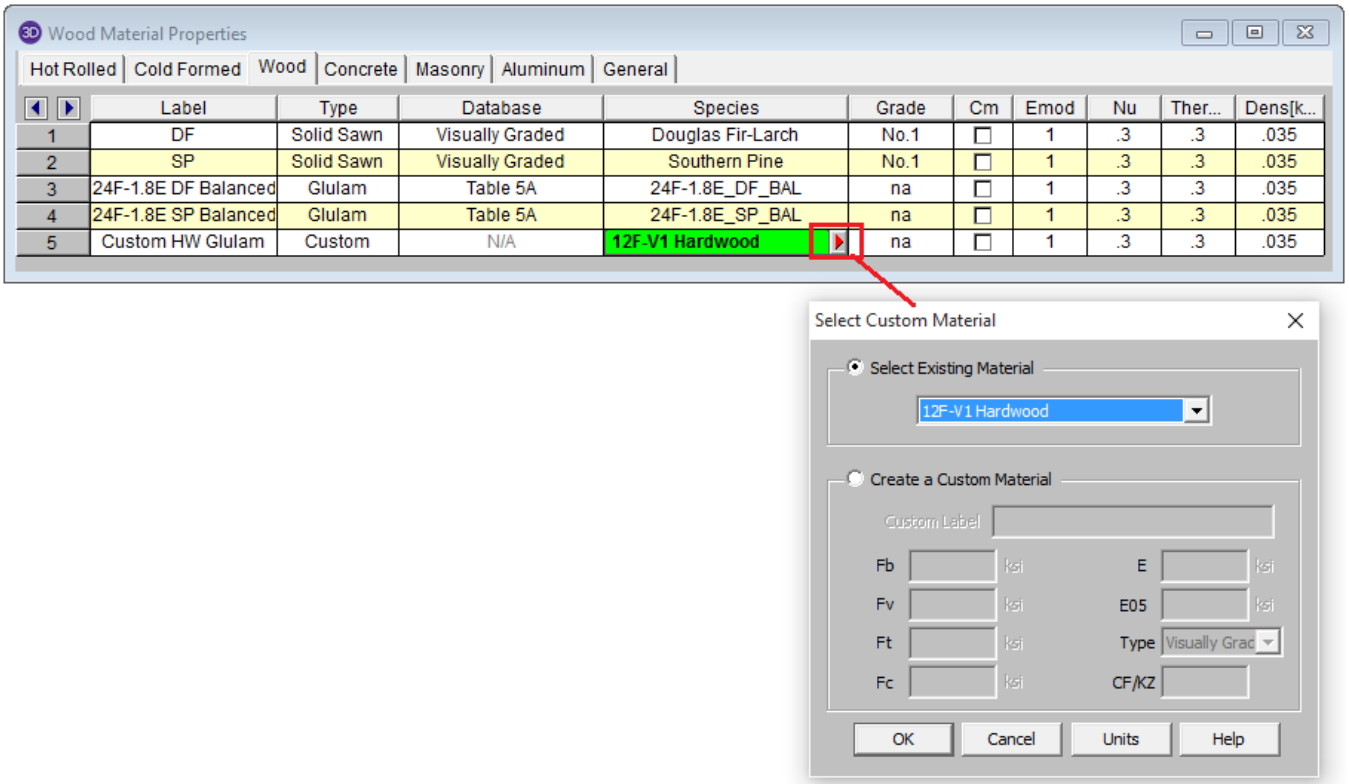
Notes:

- 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:
  - Fb is taken as the Fbx+, Fbx-, and Fby values for glulam design.
  - Fv is taken as the Fvx and Fvy values for glulam design.
  - E is taken as the Ex and Ey 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

Create and apply your custom material by doing the following:

1. First, open the **Materials** spreadsheet to the **Wood** tab.
2. Select "**Custom**" as the material **Type**.
3. Click within the **Species** cell and then click on the red arrow. This will open the **Select Custom Material** dialog box.
4. Either select an **Existing Material** (this is read in from the **Custom Wood Properties** spreadsheet, see above), or **Create a Custom Material** by typing the properties directly into this dialog.



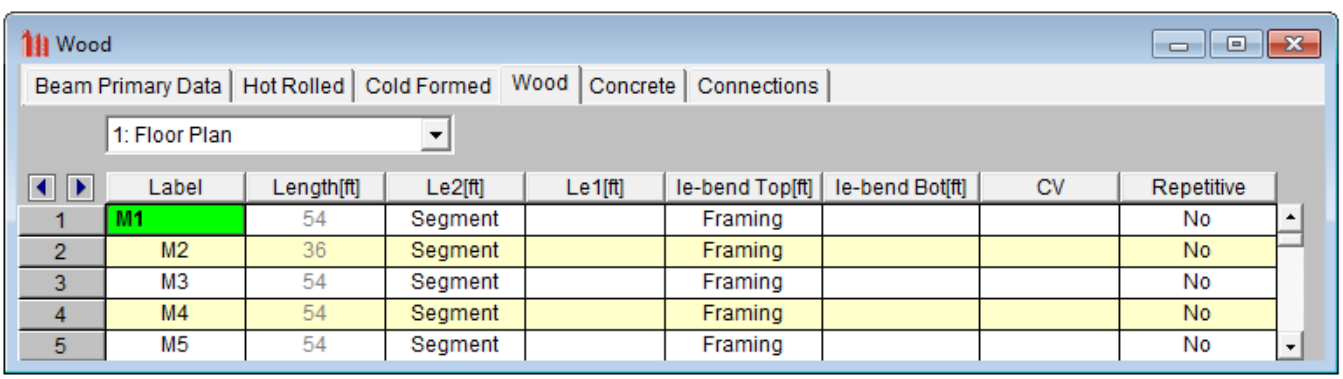
Note

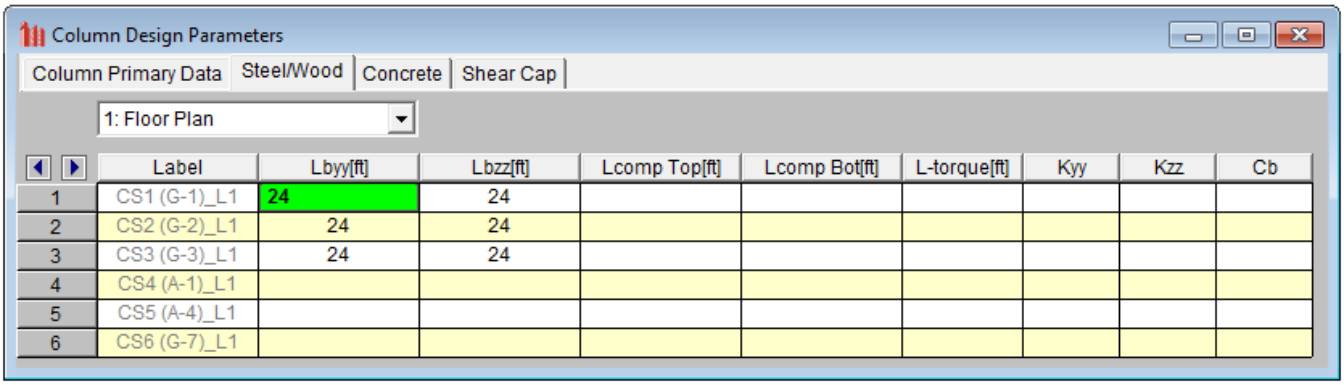
- The **Type** menu is used to designate whether the entry is a **Visually Graded** sawn lumber, a **MSR** mechanically graded lumber, a **MEL** mechanically graded lumber, a **Glulam** material, or a **SCL** (structural composite lumber). This selection effects the design factors and the applicable 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 Member Design Parameters

The **Beams** spreadsheet records the design parameters for the timber code checks and may be accessed by selecting **Beams** or **Columns** from the **Data Entry** toolbar and then clicking on the **Wood** tab. These parameters may also be assigned graphically. See Modifying Beam Design to learn how to do this.





	Label	Lbyy[ft]	Lbzz[ft]	Lcomp Top[ft]	Lcomp Bot[ft]	L-torque[ft]	Kyy	Kzz	Cb
1	CS1 (G-1)_L1	24	24						
2	CS2 (G-2)_L1	24	24						
3	CS3 (G-3)_L1	24	24						
4	CS4 (A-1)_L1								
5	CS5 (A-4)_L1								
6	CS6 (G-7)_L1								

These parameters are defined for each member. The pull down list at the top of the spreadsheet allows you to toggle between floor levels.

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. The **Label** field for column members is dictated by the **Label** entry on the **Column Stacks Spreadsheet** and may not be edited here. If you would like to edit this entry, you must do so on the **Column Stacks** spreadsheet.

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 beam **Length** is reported in the second column. This value may not be edited as it is dependent on the beam start and end points 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.

Adjustment Factors

Please see below for information about the various wood adjustment factors.

Sway Flags

See the Unbraced Lengths topic.

NDS Adjustment Factors

The NDS 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

- Cb (Bearing Area Factor), CT (Buckling Stiffness Factor), Cc (Curvature Factor), CI (Stress Interaction Factor), and Cvr (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 Cm (Wet Service Factor)

Cm is the Wet Service adjustment factor. It is applied when you check the **Cm** 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 **Bending Results 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 **Beams 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 **Beams Spreadsheet**. If you enter 'Yes' in the Repetitive 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 **Beams 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 **Beams 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 **Bending Results 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 Cc 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

- 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 Es 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, CT, is not currently accounted for.
- **Bearing Area Factor** - The NDS bearing area factor, Cb, is not currently accounted for.
- **Curvature Factor** - The NDS curvature factor for glulams, Cc, is not currently accounted for.
- **Stress Interaction Factor** - The NDS stress interaction factor for glulams, CI, is not currently accounted for.
- **Shear Reduction Factor** - The NDS shear reduction factor for glulams, Cvr, 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 **Beams** 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.

$$Emin = E * (1 - 1.645 * COV_E) * \left(\frac{1.03}{1.66}\right)$$

COV_E (the coefficient of variation in modulus of elasticity) comes from Table F1 in Appendix F.

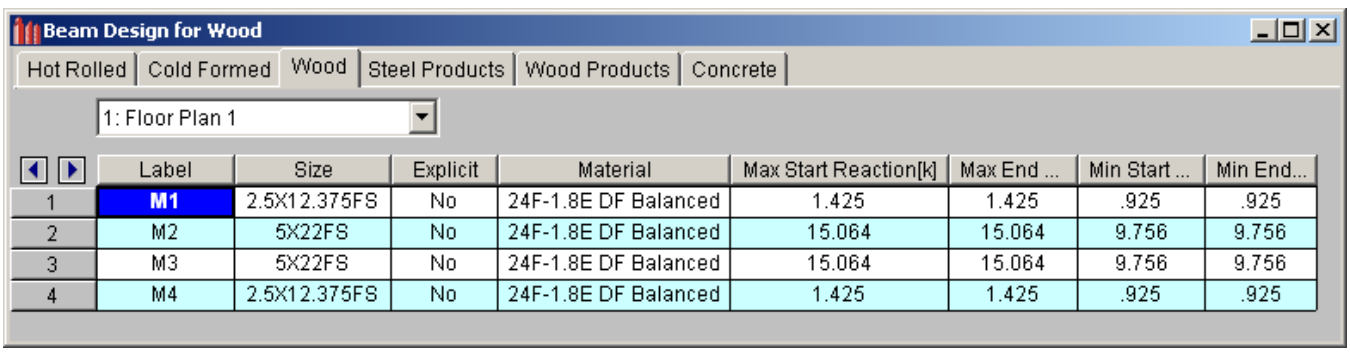
Note

- If your member is a glulam or SCL material, the Emin equation becomes:

$$Emin = E * (1 - 1.645 * COV_E) * \left(\frac{1.05}{1.66}\right)$$

Beam Design Results

The **Design Results Spreadsheet** displays the optimized design results for the beam elements and may be accessed by selecting **Designs** on the **Results** menu. The spreadsheet has six tabs: Hot Rolled, Cold Formed, Wood, Steel Products, Wood Products, and Concrete. The pull down list at the top of the spreadsheet allows you to toggle between floors.



	Label	Size	Explicit	Material	Max Start Reaction[k]	Max End ...	Min Start ...	Min End...
1	M1	2.5X12.375FS	No	24F-1.8E DF Balanced	1.425	1.425	.925	.925
2	M2	5X22FS	No	24F-1.8E DF Balanced	15.064	15.064	9.756	9.756
3	M3	5X22FS	No	24F-1.8E DF Balanced	15.064	15.064	9.756	9.756
4	M4	2.5X12.375FS	No	24F-1.8E DF Balanced	1.425	1.425	.925	.925

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

The **Size** column displays the optimized 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 or deflection requirements (see Design Optimization), or adding more shapes to the available Redesign List (see Appendix A – Redesign Lists).

The **Explicit** column displays "Yes" if the beam has been locked to an explicit beam size by the user. When you have chosen a specific shape to override the programs automatic redesign, that beam becomes "locked" and will not be automatically redesigned by the program.

Note

- To "unlock" a beam, you can use the beam – modify tool to assign a shape group. If the model has already been solved, you may optimize a beam by using the Member Redesign dialog. See Member Redesign for more details.
- Live load reduction effects are considered for the applicable values in this spreadsheet.

The **Material** column displays the material label assigned to the beam.

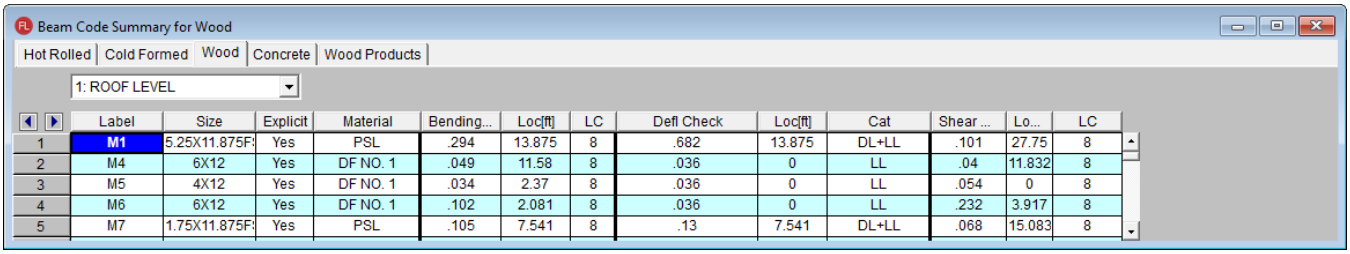
End Reactions

The **Max Start & End Reactions** column displays the maximum start and end reactions of the beam for ALL load combinations. If "Show Factored End reactions" in **Model Settings** is left unchecked, these displayed forces are not factored. If it is checked, then the displayed forces will have been multiplied by the factors in the load combinations. The sign convention assigns positive reactions to downward forces. Negative reactions, if they occur, would indicate uplift.

The **Min Start & End Reactions** column displays the minimum start and end reaction of the beam.

Beam Code Checks

The **Code Checks Spreadsheet** summarizes the code check results for the beams and may be accessed by selecting **Code Checks** on the **Results** menu. The spreadsheet has five tabs: Hot Rolled, Cold Formed, Wood, Concrete, and Wood Products. The pull down list at the top of the spreadsheet allows you to toggle between floors. For more information on Hot Rolled steel design, see Hot Rolled Steel Design or Steel Design – Composite.



	Label	Size	Explicit	Material	Bending...	Loc[ft]	LC	Defl Check	Loc[ft]	Cat	Shear ...	Lo...	LC
1	M1	5.25X11.875F:	Yes	PSL	.294	13.875	8	.682	13.875	DL+LL	.101	27.75	8
2	M4	6X12	Yes	DF NO. 1	.049	11.58	8	.036	0	LL	.04	11.832	8
3	M5	4X12	Yes	DF NO. 1	.034	2.37	8	.036	0	LL	.054	0	8
4	M6	6X12	Yes	DF NO. 1	.102	2.081	8	.036	0	LL	.232	3.917	8
5	M7	1.75X11.875F:	Yes	PSL	.105	7.541	8	.13	7.541	DL+LL	.068	15.083	8

The **Label** column displays the beam label.

The **Size** column displays the beam 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 **Explicit** column displays "Yes" if the beam has been locked to an explicit beam size by the user. When you have chosen a specific shape to override the programs automatic redesign, that beam becomes "locked" and will not be automatically redesigned by the program.

Note

- To "unlock" a beam, you can use the beam – modify tool to assign a shape group. If the model has already been solved, you may optimize a beam by using the Member Redesign dialog. See Member Redesign for more details.

The **Material** column displays the material label assigned to the beam.

Bending, Shear, and Deflection Checks

The **Bending Check** and **Shear Check** columns display the maximum bending check and shear check calculated by the program. This value is equal to the actual bending or shear demand (stress or force) divided by the actually beam resistance (allowable stress or ultimate capacity). You can see the details of these values in the **Bending Results** or **Shear Results** spreadsheet. This check is calculated at 100 sections along each beam for each load combination and the maximum check is reported. See Results Spreadsheets for more information.

The **Deflection Check** displays the maximum deflection check. This value is equal to the ratio of actual deflection to allowable deflection. You can see the details of these values in the **Deflection Results** spreadsheet. This check is calculated at 100 sections along each beam and the maximum check is reported. See Beam Results - Deflection for more information.

The **Location** columns display the location along the member where the maximum bending, shear, or deflection check occurs.

The **LC** column displays the controlling load combination which resulted in the maximum bending or shear check.

Deflection checks are based on Load Categories (Dead, Live or DL+LL), not Load Combinations. Therefore, the controlling load Category for deflections is reported in the **Cat** column.

Beam Shear Results

The **Shear Results Spreadsheet** records the shear results for the beam elements and may be accessed by selecting **Shear** on the **Results** menu. The spreadsheet has four tabs: Hot Rolled, Cold Formed, Wood, and Concrete. The pull

down list at the top of the spreadsheet allows you to toggle between floors.

Beam Shear Results for Wood

Hot Rolled | Cold Formed | Wood | Concrete

1: Floor Plan 1

	Label	Size	F'v[ksi]	fv[ksi]	Shear Check	Loc[ft]	LC
1	M1	2.5X12.375FS	.265	.069	.261	0	2
2	M2	5X22FS	.265	.205	.775	20	2
3	M3	5X22FS	.265	.205	.775	20	2
4	M4	2.5X12.375FS	.265	.069	.261	0	2

The **Label** column displays the beam label.

The **Size** column displays the beam size. When no adequate member could be found from the available shapes, this field will display the text "not designed". When this occurs, 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).

Shear Stress and Capacity

If you have selected an NDS wood design code, the **F'v** column will display the calculated allowable shear stress based on Chapter 3 of the applicable NDS code. The **fv** column displays the maximum actual shear stress that the member experiences.

If you have selected the CSA 086 wood design code, the **Vr** column will display the calculated allowable shear force based on clause 5.5.5.1 for sawn lumber, or clause 6.5.7.2.1a for glulam members (per the CSA 086-09 code). The **Vf** column displays the maximum actual shear force that the member experiences.

The **Shear Check** column displays the maximum shear check. This value is equal to the actual shear demand (stress) divided by the actual beam resistance (allowable stress). This shear check is calculated at 100 locations along each beam for each load combination. The maximum shear stress, its location, and the controlling load combination are reported. See Results Spreadsheets for more information.

The **Location** column displays the location along the member where the maximum shear check occurs.

The **LC** column displays the controlling load combination which resulted in the maximum shear check.

Note

- The CSA 086 **Vr** shear resistance value for glulams depends on the CV value. Unless manually entered by the user, this value is always assumed 1.0.

Beam Bending Results

The **Bending Results Spreadsheet** records the bending results for the beams and may be accessed by selecting **Bending** on the **Results** menu. The spreadsheet has four tabs: Hot Rolled, Cold Formed, Wood, and Concrete. The pull down list at the top of the spreadsheet allows you to toggle between floors.

Beam Bending Results for Wood

Hot Rolled | Cold Formed | Wood | Concrete

1: Floor Plan 1

	Label	Size	le-ben...	le-ben...	Rb	CL	CP	F'b[k...	fb[ksi]	Bend...	Loc[ft]	LC	Eq...
1	M1	2.5X12.375FS	3	20	8.443	.99	.047	2.376	1.34	.564	10	2	3.9-3
2	M2	5X22FS	2	20	4.596	.997	.97	2.276	2.243	.986	10	2	3.9-3
3	M3	5X22FS	2	20	4.596	.997	.97	2.276	2.243	.986	10	2	3.9-3
4	M4	2.5X12.375FS	3	20	8.443	.99	.047	2.376	1.34	.564	10	2	3.9-3

The **Label** column displays the beam label.

The **Size** column displays the beam size. When no adequate member could be found from the available shapes, this field will display the text "not designed". When this occurs, 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).

Unbraced Lengths

The **Lb Top** and **Lb Bottom** columns display the unbraced lengths associated with the controlling bending check. See Unbraced Lengths for more information on how these values are calculated.

Wood Slenderness and Stability Factors - NDS Design

The following adjustment factors are included in the spreadsheet:

- The **RB** column displays the slenderness ratio of the beam per equation 3.3-5 of the NDS code.
- The **CL** column displays the beam stability factor per equation 3.3-6 of the NDS specification.
- The **CP** column displays the column stability factor per equation 3.7-1 of the NDS specification.
    - NOTE: The column stability factor, CP, is not used for beam members in RISAFloor, only reported.

Wood Slenderness and Stability Factors - CSA 086 Design

The following adjustment factors are included in the spreadsheet:

- The **CB** column displays the slenderness ratio of the beam per clause 6.5.6.4.3.
- The **KL** column displays the lateral stability factor per clause 6.5.6.4.
- The **KC** column displays the slenderness factor per clause 5.5.6.2.4 (for sawn lumber) or clause 6.5.8.5 (for glulams).

Bending Stress and Capacity

If you have selected an NDS wood design code, th **F'b** column displays the calculated allowable bending stress based on Chapter 3 of the applicable NDS code. The **fb** column displays the maximum actual bending stress that the member experiences. The sign convention is defined so that positive bending will result in tension in the bottom fiber.

If you have selected the CSA 086 design code, the **Mr** column displays the calculated bending moment resistance based on clause 5.5.4.1 for sawn lumber or clause 6.5.6.5.1 for glulams. The **Mf** column displays the maximum actual bending force that the member experiences.

The **Bending Check** column displays the maximum bending check calculated by the program. This check is equal to the actual demand (bending stress/force) divided by the beam resistance (allowable stress/force). This bending check is calculated at 100 locations along each beam for each load combination and the maximum demand (stress), the location, and the controlling load combination are reported. See Results Spreadsheets for more information.

The **Location** column displays the location along the member where the maximum bending check occurs.

The **LC** column displays the controlling load combination which resulted in the maximum bending check.

The **Equation** column displays the code equation that controlled in the calculation of the bending check.

Column Results

The **Column Results Spreadsheet** summarizes the code check results and records the design results for columns and may be accessed by selecting **Column Results** on the **Results** menu. The spreadsheet has four tabs: Hot Rolled, Cold Formed, Wood, and Concrete.

Wood Column Code Checks

Hot Rolled | Cold Formed | Wood | Concrete

	Stack	Lift	Shape	Cod...	Ele...	LC	Shea...	Ele...	D.	LC	Fc' [k...	Ft' [ksi]	Fb1' [...	Fb2' [...	Fv' [k...	Eqn
1	CS1 (K-1)	1	8X8	.812	0	2	.000	0	z	2	.666	.825	1.2	1.2	.165	3.6.3
2		2	6X6	.891	12	2	.000	12	z	2	.607	.825	1.2	1.2	.165	3.6.3
3	CS2 (K-3)	1	8X8	.814	0	2	.000	0	z	2	.666	.825	1.2	1.2	.165	3.6.3
4	CS3 (I-3)	1	8X8	.812	0	2	.000	0	z	2	.666	.825	1.2	1.2	.165	3.6.3
5		2	6X6	.891	12	2	.000	12	z	2	.607	.825	1.2	1.2	.165	3.6.3
6	CS4 (I-1)	1	8X8	.814	0	2	.000	0	z	2	.666	.825	1.2	1.2	.165	3.6.3

The **Stack** column displays the column stack label.

The **Lift** column displays the lift number for the physical column. Lift No. 1 is the lowermost physical column in a stack and the lifts are numbered sequentially moving up the column stack.

The **Shape** column displays the physical column 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).

Code Checks

The **Code Check** column displays the maximum combined axial and bending check calculated by the program. This value is equal to the actual combined axial and bending demand (stress or force) divided by the actually column resistance (allowable stress or ultimate capacity). You can see the details of this value in the subsequent **Axial Resistance** and **Flexural Resistance** columns of this spreadsheet. This check is calculated at 100 sections along each physical column for each load combination and the maximum check is reported. See Results Spreadsheets for more information.

The **Shear Check** column displays the maximum shear check calculated by the program. This value is equal to the actual shear demand (stress or force) divided by the actual column resistance (allowable stress or ultimate capacity). You can see details of this value in the subsequent **Shear Resistance** column of this spreadsheet. This check is calculated at 100 sections along each column for each load combination and the maximum check is reported. See Results Spreadsheets for more information.

The **Elev** columns displays the absolute elevation along the column stack where the maximum code check occurs.

The **LC** column displays the controlling load combination which produced the maximum code check and/or shear check.

The **Dir** column displays the column local axis along which the maximum shear check occurs.

Column Stress and Capacity

If you have selected an *NDS design code*, the following capacities will be displayed:

- The **Fc'** column displays the calculated factored allowable axial compressive stress based on Chapter 3 of the applicable NDS specification.
- The **Ft'** column displays the calculated factored allowable axial tensile stress based on Chapter 3 of the applicable NDS specification.
- The **Fb1'** and **Fb2'** columns display the calculated factored allowable bending stress based on Chapter 3 of the applicable NDS specification.
- The **Fv'** column displays the calculated factored allowable shear stress based on Chapter 3 of the applicable NDS specification.

If you have selected the *CSA 086 design code*, the following capacities will be displayed:

- The **Pr** column displays the calculated allowable axial compressive force parallel to grain based on clause 5.5.6.2.3 for sawn lumber or clause 6.5.8.4.2 for glulam members.

    - The axial compressive resistance for sawn lumber members will have a 75% reduction taken into account (per clause 5.5.6.4.3) for bolted multi-ply members.
- The **Tr** column displays the calculated allowable axial tensile force based on clause 5.5.9 for sawn lumber or clause 6.5.11 for glulam members.
    - The **Tr**, axial tensile resistance for glulams only considers the gross area of the member when calculated per clause 6.5.11.

- The **Mrz** and **Mry** columns display the calculated allowable bending force in each direction based on clause 5.5.4.1 for sawn lumber or clause 6.5.6.5.1 for glulam members.
    - The glulam material Table 6.3 in the CSA 086-09 does not include weak axis bending or shear values. Per the commentary to clause 6.5, glulams will use the sawn lumber values for the glulams species' grade No. 2 listing in Table 5.3.1A.
- The **Vr** column displays the calculated allowable shear force based on clause 5.5.5.1 for sawn lumber or clause 6.5.7.2.1a for glulam members.

The **Eqn** column displays the code equation used in the calculation of the controlling code check.

Special Messages - Wood Design

In some instances code checks are not performed for a particular member. A message explaining why a code check is not possible will be listed instead of the code check value. You may click the cell that contains the message and look to the status bar to view the full message. Following are the messages that may be listed:

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 Le1/d or Le2/b 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

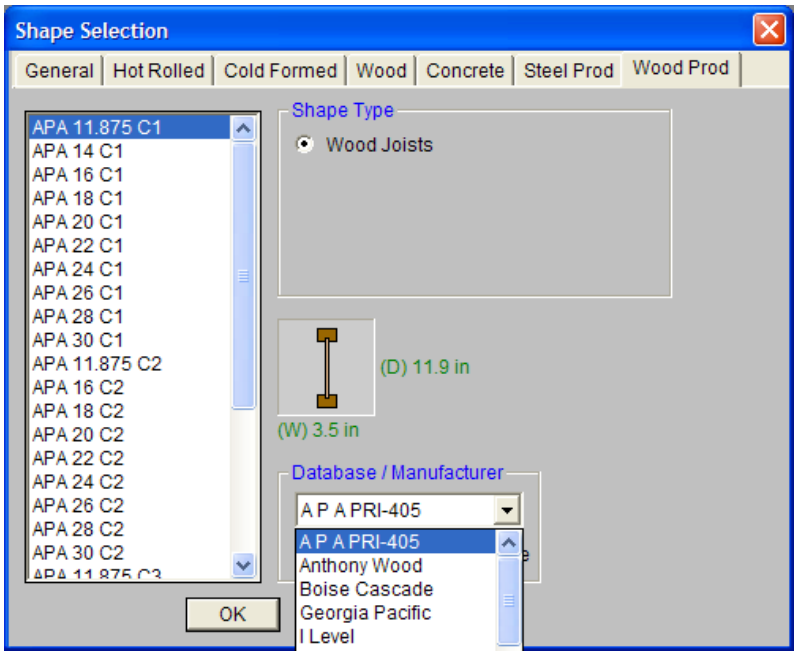
Section 3.9.2 of the NDS code limits the actual strong axis bending compressive stress to be less than the term FbE. This term is approximately the lateral buckling stress.

Wood Products

Full beam wood design can be performed on Wood Joist Products based on the NDS and CSA wood engineering codes. The allowable moment and shear values were obtained from manufacturer's catalogs. Each database of I-Joists are specified by an XML file in the **I-Joist** sub-directory of the **RISA Wood Schedules** directory. The location of this directory is based on the information in the **File Locations** tab of the **Tools - Application Settings** dialog. The default XML files are listed below:

Wood Products Database

The various Wood Product databases may be accessed from the **Beams** spreadsheet by clicking in the **Shape** field and then clicking .



The Wood Product database is based on data contained in the product's manufacturer's catalog. You may choose the Manufacturer and product from the drop down lists.

When you return to the **Beams** spreadsheet you will notice that a special label has been used to represent the choices that you made in the database. You may also type in this designation directly.

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

APA PRI-405

The APA Performance rated I-Joists bring standardization to the wood I-Joist industry. The standard provides design information for numerous types and sizes of I-Joists. This document is a draft version and is scheduled to be published soon. For all the design values that were used for this standard, refer the XML file located in the "I-Joist" sub-directory of the RISA Wood Schedules directory. For more information refer to APA website: www.apawood.org

XML Databases

Each database of I-Joists are specified by an XML file in the **I-Joist** sub-directory of the **RISA Wood Schedules** directory. The location of this directory is based on the information in the File Locations tab of the Tools - Application Settings dialog.

The program comes pre-loaded with XML files each of which contain a database of the manufacturers I-Joists:

Manufacturer

- APA PRI-405
- Anthony Wood
- Boise Cascade
- Georgia Pacific
- iLevel
- International Beams
- Louisiana Pacific
- Nordic
- Pacific Woodtech
- RedBuilt
- Roseburg Forest Products
- Standard Structures
- Trus Joist
- Canadian Anthony Wood
- Canadian Boise Cascade
- Canadian Georgia Pacific
- Canadian International Beams
- Canadian Louisiana Pacific
- Canadian Nordic
- Canadian Pacific Wood
- Canadian RedBuilt
- Canadian Roseburg Forest Products
- Canadian Trus Joist

The name of the XML file itself will be used in the list of databases in the Wood Products Dialog. The first sheet of the XML file contains all of the joist name, joist size for rendering purposes, and design information used for each I-Joist. All fields are required information and if they are not provided or are left blank, then that I-Joist will not be available for use in that database. The Label field is used to identify the I-Joist and this field must be a unique name. The Depth, Flange Width, Thickness, and Web Thick entries are used to render the I-Joist shapes. The Joist Wt is used to calculate the self weight of the I-Joist. The maximum moment and shear forces are used as the allowables and are described below as V' and M'. The EI and the K, shear deformation coefficient is used in the deflection equation:

$$\Delta = \frac{5wl^4}{384\,EI} + \frac{wl^2}{K}$$

Wood Products Adjustment Factors

The NDS code applies adjustment factors to the allowable moments and shears of a wood joist. Currently RISAFloor only accounts for the Load Duration Factor CD. The other adjustment factors (CL, Cm, Cr, Ct, and CH) are all applied as 1.0.

Wood Design Temperature Factor

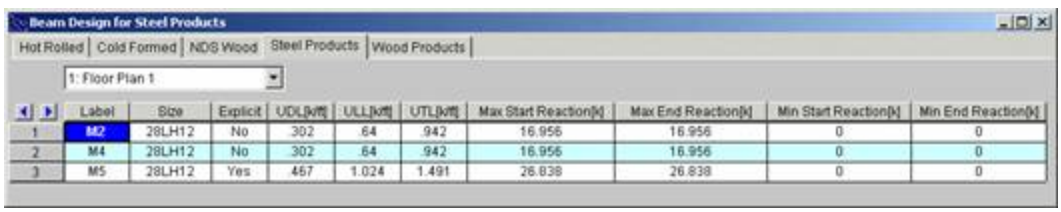
Load Duration Factor

The CD factor is entered on the **Load Combination** spreadsheet for each load combination that you want to use for wood code checks. If this field is left blank, it will be automatically calculated by the program. Different load combinations will have different CD factors. The CD factor will only be applied to wood code checks on wood

members. See the Table below for the CD factors that are assumed for the various load categories. Note that the CD factor used for a load combination will be for the load category with the shortest load duration in that load combination.

Load Category	CD
DLPre	0.9
LLConst	1.25
DLConst	1.25
DL	0.9
LL	1.0
LLS	1.0
RLL	1.0
SL	1.15
SLN	1.15
RL	1.15
OL1, OL2, OL3, OL4	1.0

Design Results - Wood Products



	Label	Size	Explicit	UDL[k/ft]	ULL[k/ft]	UTL[k/ft]	Max Start Reaction[k]	Max End Reaction[k]	Min Start Reaction[k]	Min End Reaction[k]
1	M2	28LH12	No	.302	.64	.942	16.956	16.956	0	0
2	M4	28LH12	No	.302	.64	.942	16.956	16.956	0	0
3	M5	28LH12	Yes	.467	1.024	1.491	26.838	26.838	0	0

The results for the **Steel and Wood Products** tab are explained below. The steel product results give the uniform loads applied to the joists. The results for the wood products give the maximum bending moments and shears as well as the allowable moments and shears for the joist. The pull down list at the top of the spreadsheet allows you to toggle between floors.

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

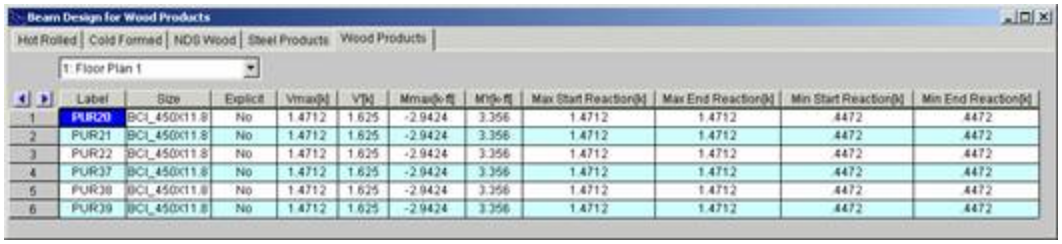
The **Size** column displays the designation of the steel or wood product. When no adequate member could be found from the redesign list, this field will display the text "not designed". Consider reframing, relaxing the design or deflection requirements (see Design Optimization), or adding more shapes to the available Redesign List (see Appendix A – Redesign Lists).

The **Explicit** column displays "Yes" if the beam has been locked to an explicit beam size by the user. When you have chosen a specific shape to override the programs automatic redesign, that beam becomes "locked" and will not be automatically redesigned by the program.

Note

- Live load reduction effects are considered for the applicable values in this spreadsheet.
- To "unlock" a beam, you can use the beam – modify tool to assign a shape group.

Shear and Moment Check for Wood Products


Beam Design for Wood Products

Hot Rolled | Cold Formed | NDS Wood | Steel Products | Wood Products

1: Floor Plan 1

	Label	Size	Explicit	Vmax[k]	V`[k]	Mmax[k-ft]	M`[k-ft]	Max Start Reaction[k]	Max End Reaction[k]	Min Start Reaction[k]	Min End Reaction[k]
1	PUR20	BCI_450X11.8	No	1.4712	1.625	-2.9424	3.356	1.4712	1.4712	.4472	.4472
2	PUR21	BCI_450X11.8	No	1.4712	1.625	-2.9424	3.356	1.4712	1.4712	.4472	.4472
3	PUR22	BCI_450X11.8	No	1.4712	1.625	-2.9424	3.356	1.4712	1.4712	.4472	.4472
4	PUR37	BCI_450X11.8	No	1.4712	1.625	-2.9424	3.356	1.4712	1.4712	.4472	.4472
5	PUR38	BCI_450X11.8	No	1.4712	1.625	-2.9424	3.356	1.4712	1.4712	.4472	.4472
6	PUR39	BCI_450X11.8	No	1.4712	1.625	-2.9424	3.356	1.4712	1.4712	.4472	.4472

The **V max** column for Wood Products displays the maximum applied shear that resulted from the load combinations that were selected for Wood Product design.

The **V`** column for Wood Products displays the allowable shear for the wood product under consideration. The program obtains this value from the manufacturers catalog.

The **M max** column for Wood Products displays the maximum applied moment that resulted from the load combinations that were selected for Wood Product design.

The **M`** column for Wood Products displays the allowable moment for the wood product under consideration. The program obtains this value from the manufacturers catalog.

Note:

- Wood I-Joists can be drawn continuous. It is assumed that the I-Joist member's compression flange is be sufficiently braced per manufacturer's/ICC recommendations.

End Reactions

The **Max Start & End Reactions** column displays the maximum start and end reactions of the beam for ALL load combinations. If "Show Factored End Reactions" in Model Settings is left unchecked, these displayed loads are not factored. If it is checked, then the displayed loads will have been multiplied by the factors in the load combinations. The sign convention assigns positive reactions to downward forces. Negative reactions, if they occur, would indicate uplift.

The **Min Start & End Reactions** column displays the minimum start and end reaction of the beam.

Deflections

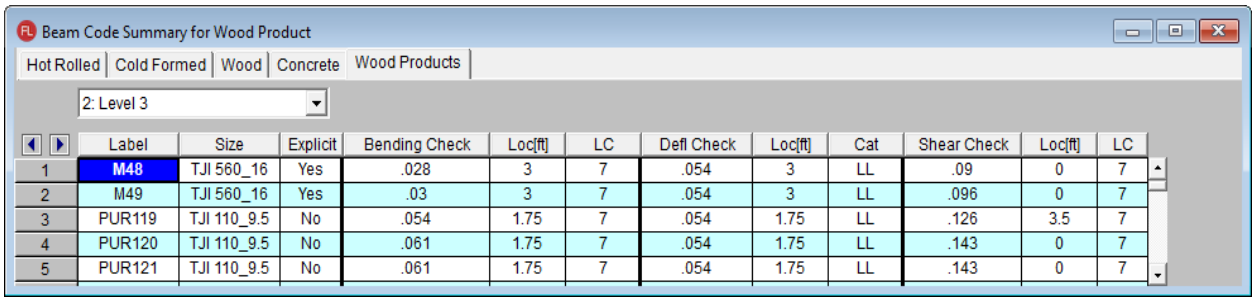
Shear deflection is only calculated for wood products that are pinned at both ends and have a uniform distributed load. Shear deflection may have significant effects for wood products, especially on beams with shorter spans.

End Reactions and Stiffeners

RISAFloor does not check the maximum end reactions per the manufacturer's documentation. The end reactions and intermediate reactions including the bearing lengths must be checked outside of the RISAFloor program.

Code Checks

The **Code Checks Spreadsheet** summarizes the code check results for the beams and may be accessed by selecting **Code Checks** on the **Results** menu. The spreadsheet has five tabs: Hot Rolled, Cold Formed, Wood, Concrete, and Wood Products. The pull down list at the top of the spreadsheet allows you to toggle between floors.



	Label	Size	Explicit	Bending Check	Loc[ft]	LC	Defl Check	Loc[ft]	Cat	Shear Check	Loc[ft]	LC
1	M48	TJI 560_16	Yes	.028	3	7	.054	3	LL	.09	0	7
2	M49	TJI 560_16	Yes	.03	3	7	.054	3	LL	.096	0	7
3	PUR119	TJI 110_9.5	No	.054	1.75	7	.054	1.75	LL	.126	3.5	7
4	PUR120	TJI 110_9.5	No	.061	1.75	7	.054	1.75	LL	.143	0	7
5	PUR121	TJI 110_9.5	No	.061	1.75	7	.054	1.75	LL	.143	0	7

The **Label** column displays the beam label.

The **Size** column displays the beam 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 **Explicit** column displays "Yes" if the beam has been locked to an explicit beam size by the user. When you have chosen a specific shape to override the programs automatic redesign, that beam becomes "locked" and will not be automatically redesigned by the program.

Note

- To "unlock" a beam, you can use the beam – modify tool to assign a shape group. If the model has already been solved, you may optimize a beam by using the Member Redesign dialog. See Member Redesign for more details.

Bending, Shear, and Deflection Checks

The **Bending Check** and **Shear Check** columns display the maximum bending check and shear check calculated by the program. This value is equal to the actual bending or shear demand (stress or force) divided by the actually beam resistance (allowable stress or ultimate capacity). You can see the details of these values in the **Bending Results** or **Shear Results** spreadsheet. This check is calculated at 100 stations along each beam for each load combination and the maximum check is reported.

The **Deflection Check** displays the maximum deflection check. This value is equal to the ratio of actual deflection to allowable deflection. You can see the details of these values in the **Deflection Results** spreadsheet. This check is calculated at 100 stations along each beam and the maximum check is reported. See Beam Results - Deflection for more information.

The **Location** columns display the location along the member where the maximum bending, shear, or deflection check occurs.

The **LC** column displays the controlling load combination which resulted in the maximum bending or shear check.

Deflection checks are based on Load Categories (Dead, Live or DL+LL), not Load Combinations. Therefore, the controlling load Category for deflections is reported in the **Cat** column.

Wood Product Limitations

Shear deflection is only calculated for wood products that are pinned at both ends and have a uniform distributed load. Shear deflection may have significant effects for wood products, especially on beams with shorter spans.

Elevated Slabs - Overview

Elevated slab design is a feature that is only available in RISAFloor ES (as opposed to RISAFloor). This feature allows you to model a concrete floor system that is self supporting, as opposed to a Beam Supported Floor, where the floor is a deck that is supported by beams. Elevated Floor Slab floors are unique in that they do not require any beams. Instead, the floor itself spans between columns/walls. Slabs may be analyzed and designed using traditional reinforcement bars or welded deformed wire reinforcement (WWR). WWR can be enabled by going to the Global Model Settings and checking the "Include Welded Wire Reinf." option in the Concrete tab. Two rebar design options are available, namely the Construction method and the Theoretical method. More info on rebar design options can be found in Global Model Settings - Concrete tab.

Note:

- This topic refers explicitly to elevated slab floors. For information on deck information for beam supported floors see the Decks and Diaphragms topic.
- Currently the **ACI 318-19**, **ACI 318-14**, **ACI 318-11**, **ACI 318-08**and **CSA A23.3-14** are the only codes supported for elevated slab design.
- See the Elevated Slabs - Analysis topic for more information on the analysis procedure.
- See the Elevated Slabs - Design topic for more information on the design procedure.

Suggested Workflow

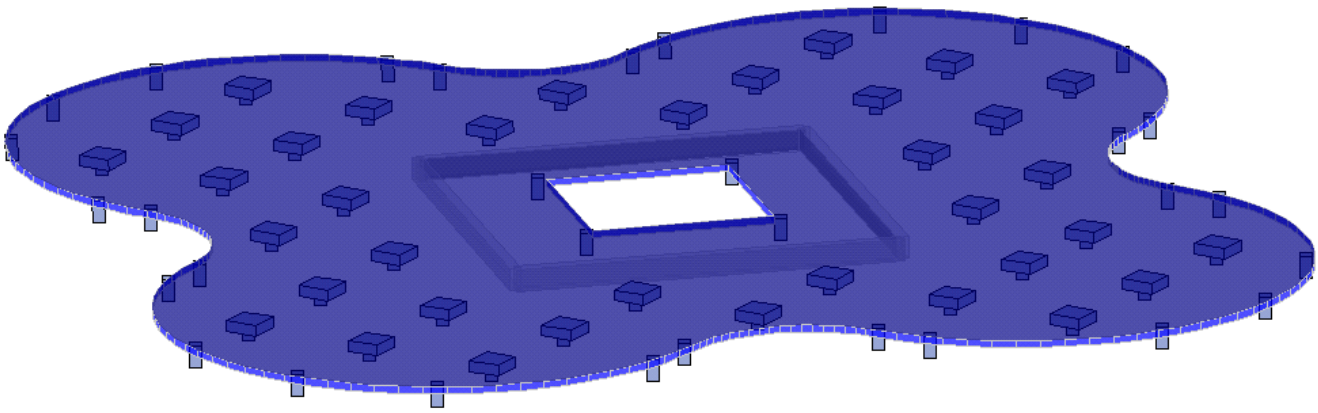
You'll want to follow these steps to create your model concrete floor plan model.

1. In the Create New Floor Plan dialog select Concrete Floor Slab. Choose the Design Method that you would like to use. See Design Methods for Design Strip Modeling or User Defined.
2. Import a DXF Drawing Grid or define a Drawing Grid manually to aid your drawing of the columns, slabs and beams. See DXF Drawing Grid for more information.
3. Define column and wall properties, and draw columns and walls that will support the floor level. See Drawing Columns and Drawing Walls for more information.
4. Define slab properties, draw the outline of the slab edge and add any openings. See Drawing Slabs for more information.
5. Draw any beams in the model. See Drawing Beams for more information.
6. Review the default area loads and apply any additional loads to the slab. See Area Loads, Point Loads, and Line Loads for more information.
7. **Design Strip Modeling**: Draw the Support Lines in both directions (N-S and E-W). Generate the Design Strips by clicking the Generate Strips button. Modify the Design Strip Areas. See Design Strip Modeling for more information. Or **User Defined Modeling**: Define the default reinforcement to be used for the entire slab. See Slab Design Rules.
8. Review the Load Combinations and create new combinations. Run the solution. See the Load Combinations for more information.
9. Review the Design Results including reinforcement and deflections. See the Design Results for more information.

Slab Modeling Overview

Slab Perimeter/Edges/Openings

Unlike Beam Supported Floors, Concrete Floor Slabs have no restrictions on the shape or size of the floor's perimeter (edge). This allows for complete flexibility in designing floor slabs with curved edges, or protruding balconies. Modeling a floor system such as the one below would be difficult or impossible as a Beam Supported Floor. Openings may also take any shape, and do not require beams around their perimeter.

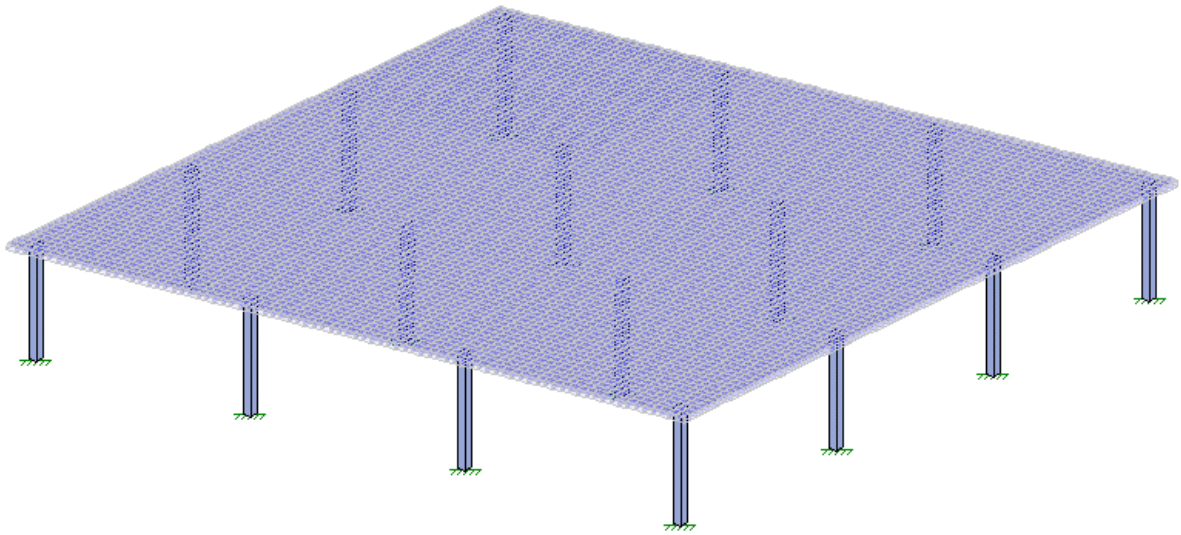


Supports

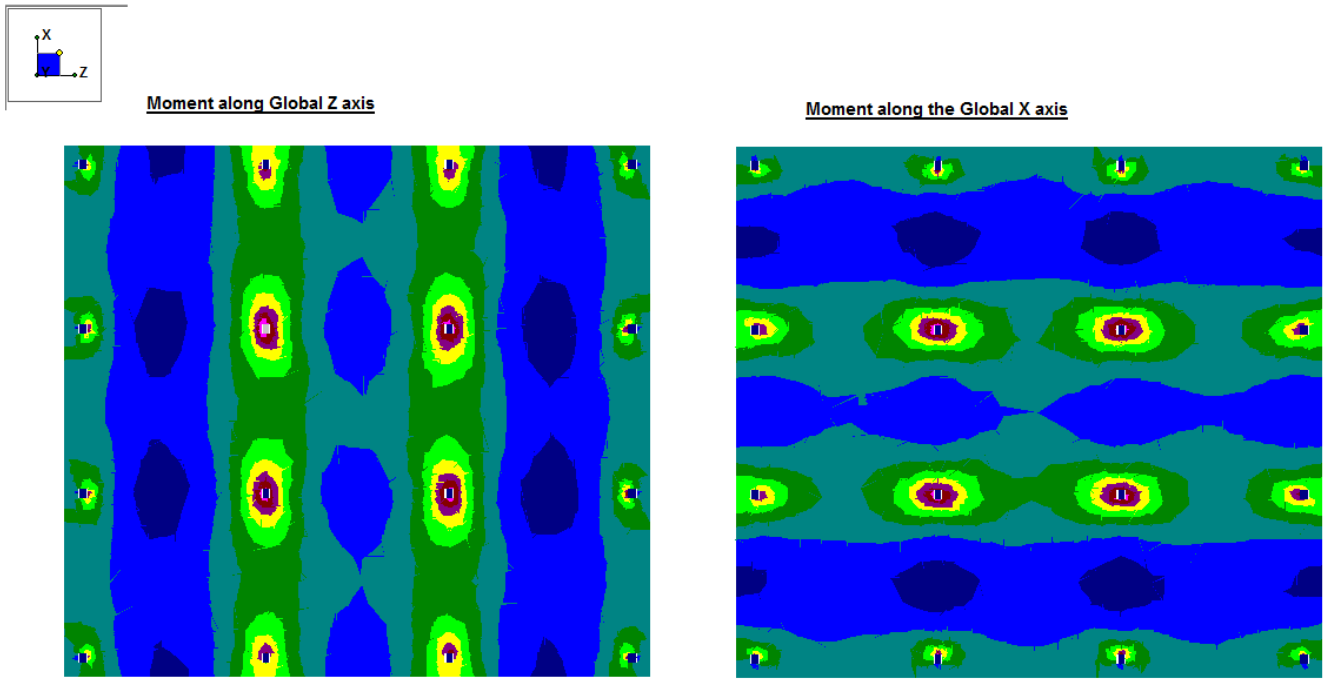
The concrete floor slab must be supported by vertical supports. These can be modeled as either columns or walls. In cases where punching shear is an issue, columns may include an integrated Shear Cap, which is essentially a thickening of the slab near the column. The interior columns in the image above show square Shear Caps, while the perimeter columns do not.

Load Path and Reinforcement Placement

The weight of the floor slab (which is usually significant) is combined with the superimposed vertical (gravity) loads, and is attributed to the supports based on the slab's stiffness. Unlike Beam Supported Floors, where the load on a support is a direct function of the support's tributary area, in Concrete Floor Slabs a finite element analysis (FEA) is undertaken in order to determine which portion of the load travels to which supports. In FEA the majority of the load will always travel along the route of greatest stiffness. Note the uniform slab below, with uniform loading and regularly spaced supports:



The bending stress contours of the slab show flow of forces in two directions with the slab experiencing the greatest tensile stresses (color coded in pink) at the supports. There are regions in the slab with virtually zero moment (color coded in greens). In order to optimize the design of the reinforcement over the areas of high moment, the design strip method is used in both directions.



Traditional Rebar Placement Methods

The quantity of rebar required in a given area of the slab is proportional to the bending stress in that area and prescribed minimums. Therefore, locations with higher bending stress such as the area directly between columns require more rebar than locations with lower bending stress such as the space in the "middle" of the four columns on the images above. As part of their Equivalent Frame Method, the ACI code recommends dividing reinforcement design into "strips":

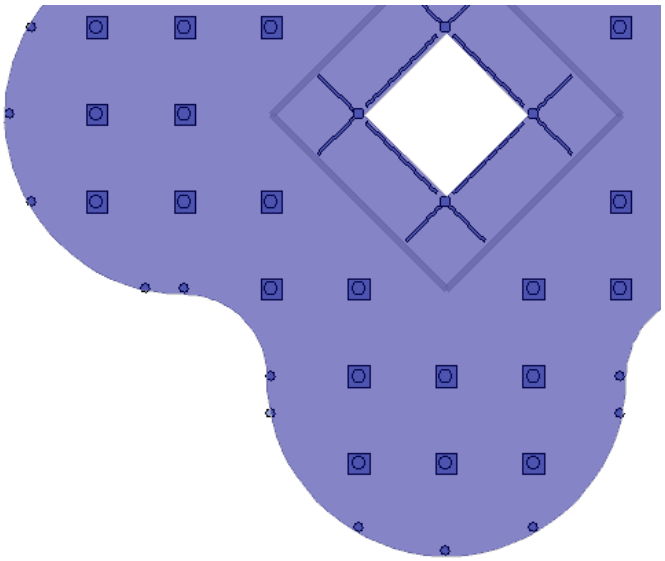
Column Strips run directly from column-to-column in each of two orthogonal directions. In slabs where the rebar is laid out orthogonally there are two nominal directions (North-South and East-West) that column strips run along. This means that the direction of the rebar in the North-South column strips is perpendicular to the direction of the rebar in the East-West column strips, regardless of the direction of True North. By using this compass terminology it is easy for the user to keep track of which set of rebar is which. Column Strips are assigned a width which is intended to capture the area of the slab with the highest bending stress, and thus the need for the most reinforcement.

Middle Strips run both parallel to, and adjacent to Column Strips. They also use the North-South, East-West terminology for organization. The middle strips fill in the gaps between column strips, to ensure that the areas of the slab with little or no bending stress still receive a minimum of reinforcement.

By dividing the slab into Column Strips and Middle Strips the designer can ensure that the entire slab is receiving adequate reinforcement, and that reinforcement is placed only where it is needed. Beware that any area of the slab which is not part of a column strip or middle strip receives no reinforcement design.

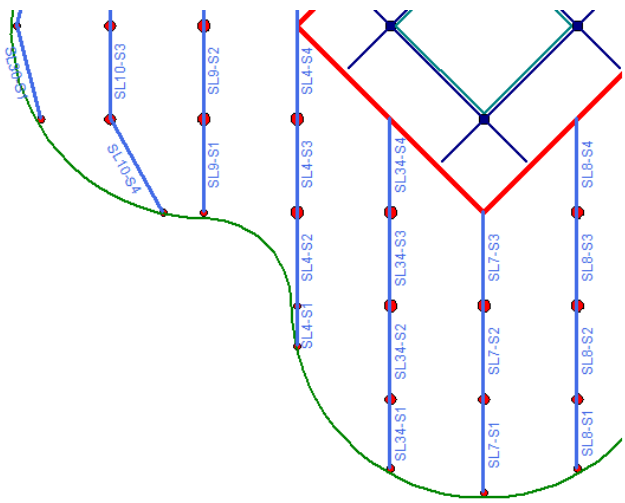
Design Strip Rebar Placement in RISAFloor ES

RISAFloor ES gives the user flexibility for rebar placement. This flexibility is necessary to achieve adequate designs for diverse building designs where supports may not be regularly spaced, or orthogonal. This overcomes the limitations of the ACI Direct Design method, which is predicated on simplified building layouts that make hand calculations possible. In RISAFloor ES, rebar design is performed based on the hierarchy explained below. For easier comprehension an example slab as shown below will be used to illustrate each element in the hierarchy.

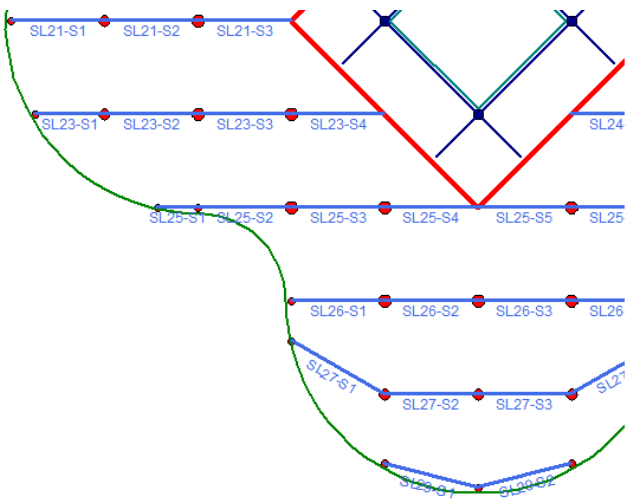


1. Support Lines

Support Lines are drawn by the user, and are used to define a line of supports for the slab that run along a nominal North-South or East-West direction. Support Lines should run along the shortest possible path between supports. Adjacent support lines should be approximately parallel, however complex support configurations can make this difficult to achieve. Note that support lines may cross or terminate at walls (in addition to columns).



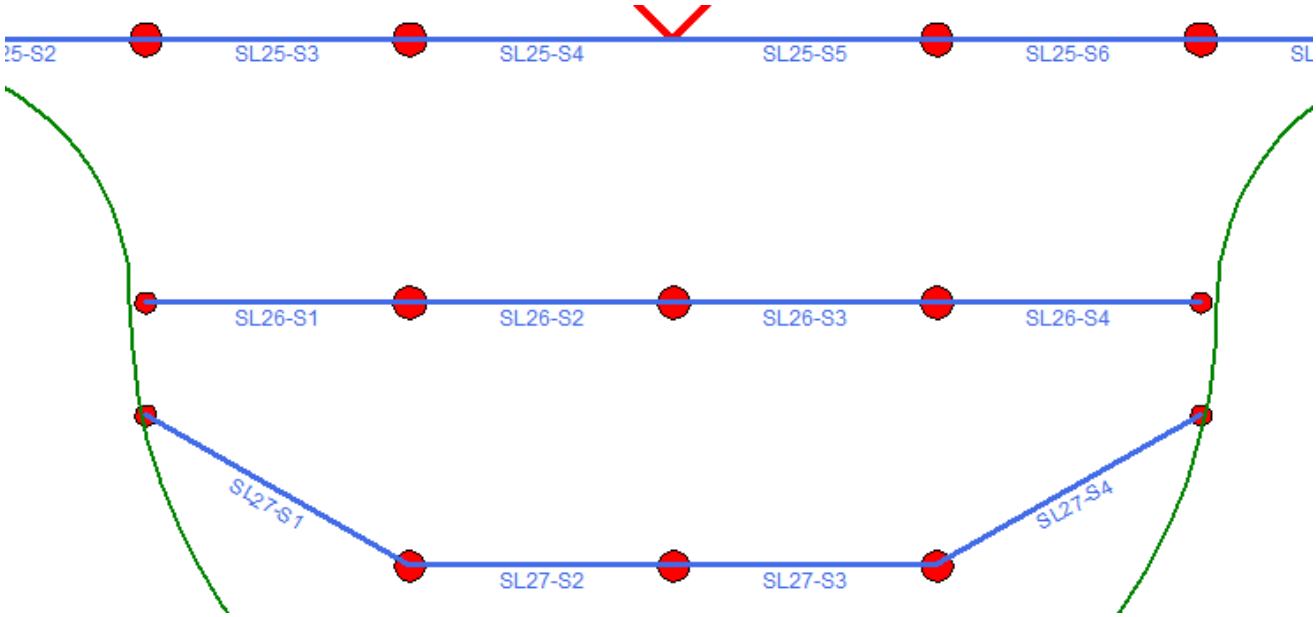
North-South Support Lines



East-West Support Lines

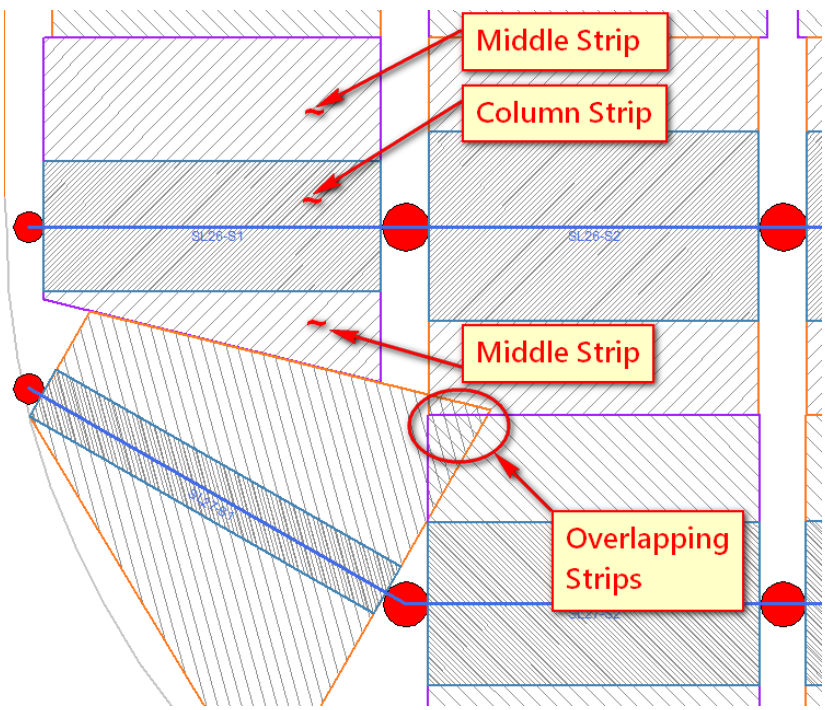
2. Spans

Each Support Line is divided into Spans. RISAFloor ES automatically creates the spans for each support line, based on the supports which the Support Line crosses. Each Span is intended to run from support-to-support, although spans may also dead-end into a location other than a support, such as a slab edge. In the image below, Support Line 26 (SL26) which is nominally an East-West Support Line is split into four spans (Span 1, Span 2, etc.) which are labeled SL26-S1, SL26-S2, etc.



3. Design Strips

Per the Equivalent Frame method, each Span is divided into three Design Strips: Middle Strip- Left, Column Strip, Middle Strip- Right. The Column Strip is approximately centered on the Support Line. The Middle Strips (Left and Right) fill in the gaps between the Column Strips of adjacent Spans. Each Span has its own Middle Strips, so the Middle Strips of adjacent spans butt up against each other.



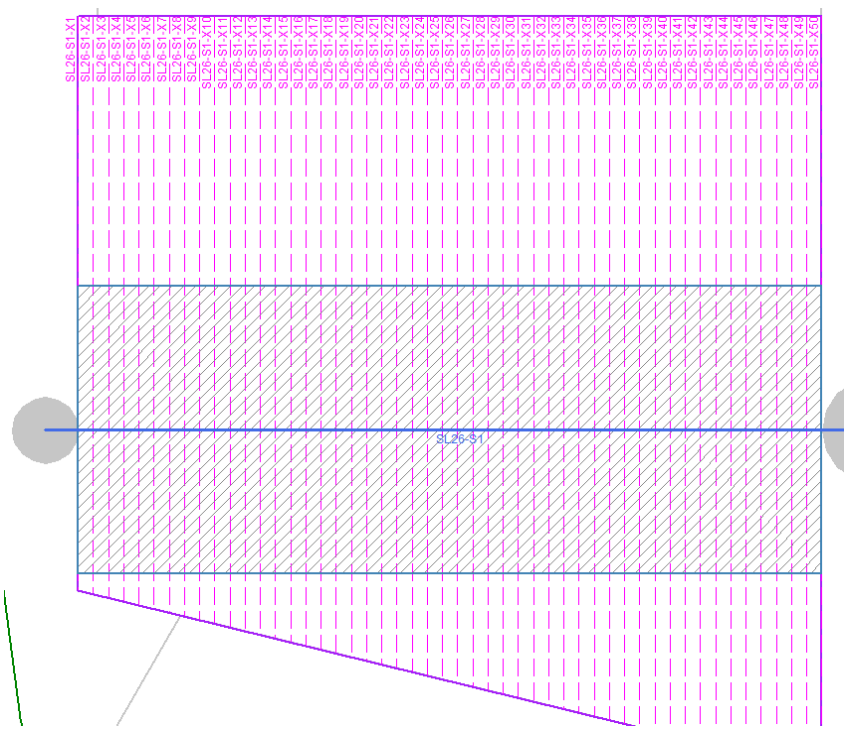
The Middle Strip- Left is the strip to the left of the Support Line. In the example above, Support Line 26 (SL26) was drawn left-to-right, so the "Left" strip is the upper strip (as that would be the Left direction when looking along the line the Support Line was drawn). Note that the Middle Strip- Right is tapered so that it does not overlap with the Middle Strip- Left of Support Line 27, Span 1 (SL27-S1). Also note that there is a location where multiple strips overlap. In locations where design strips overlap the required rebar for all of the overlapping strips is used. This results in a conservative design.

The Column Strip width is automatically calculated, and is based on the distance to adjacent Support Lines or Slab Edges. In the example above the column strip for Span 2 is wider than the column strip for Span 1. Click here for the exact description of the Design Strip Width equations. The Design Strip width is calculated with the intention of capturing the highest moment demands in the column strip. The most ideal width would be the bound by the zero shear however the widths are currently only based on the Equivalent Frame Method.

There will be top reinforcement designed over the supports due to the tension at the top of the slab (positive moment). The gap between design strips of adjacent spans (shown as empty white space in the image above) is addressed by running the Top Rebar continuously through the column (or through the gap in the case of Middle Strips). RISAFloor ES automatically does this as part of the rebar design. The bottom reinforcement is designed with the same spacing over the entire design strip to resist the tension at the bottom of the slab (negative moment).

4. Auto Design Cuts

Each Design Strip is automatically divided into Design Cuts. A Design Cut is a two dimensional cut through the thickness of the slab, perpendicular to the Support Line. The width of the cut is equal to the width of the Design Strip. In the image below, the Design Strips of Support Line 26, Span 1 (SL26-S1) contains 50 Design Cuts. These Cuts are shown as dashed pink lines running perpendicular to the Support Line. The Cuts are numbered starting at the left as Support Line 26, Span 1, Cut 1 (SL26-S1-X1) through Cut 50 (SL26-S1-X50). In this example, the Design Cuts for the Middle Strip-Left and the Column Strip have a uniform width, but the Design Cuts for the Middle Strip-Right have varying width since the Middle Strip-Right is tapered.



At each Design Cut, RISAFloor ES sums up the total forces across the Cut, as determined by FEA. The total shear for the Cut is used to perform a one way shear check. The total bending moment across the Cut (taken as the bending moment about the Cut axis) is used to determine the required amount of rebar at the Cut location. All this analytical and design information is presented in the Design Cut Detail Report.

Design Strip Summary

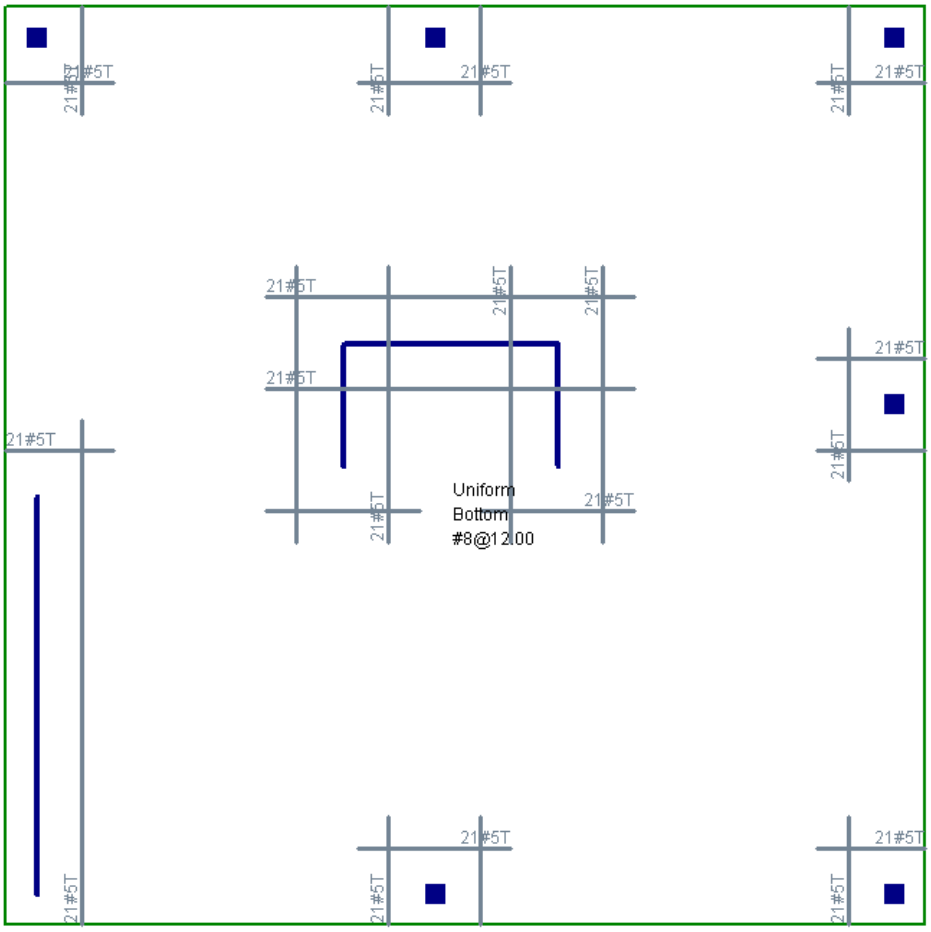
RISAFloor ES calculates the rebar requirements for each Design Cut and uses that to determine the required rebar for each region within the Design Strip, including where the bars must start and terminate along the length of the strip. With the design of rebar for all three strips, the required placement of rebar for each Span is determined. To

address the bending at supports, the rebar for each Span is matched to the rebar for adjacent Spans so that the top bars can be run continuously. Compatible designs for all adjacent Spans result in an overall rebar design for the entire Support Line.

If all of the Support Lines for a given direction (such as nominal East-West) are arranged such that their Design Strips butt up against each other then all of the required rebar for the slab is calculated for that direction by RISAFloor ES. If the same is true for the other nominal direction, and if there are no significant gap areas (where no Design Strip from either direction is present) then RISAFloor ES provides you with a complete rebar design for the entire slab, sufficient to meet all code requirements.

User Defined Rebar Placement in RISAFloor ES

Using the User Defined Rebar option in RISAFloor ES gives the user a simple option to check rebar pattern for the entire slab. This option allows the user to set a regularly spaced top and bottom rebar to check whether it meets code requirements. You can set this method from the start or flip back and forth to the Design Strip Method. For a full step by step, see User Defined.



Elevated Slabs - Modeling

Elevated slab analysis and design is done with RISAFloor ES. A slab must be defined that will then be loaded and analyzed. At solution the program will create a finite element mesh of plates to represent the slab. From there an analysis is considered and the program will then design the slab based on design strips.

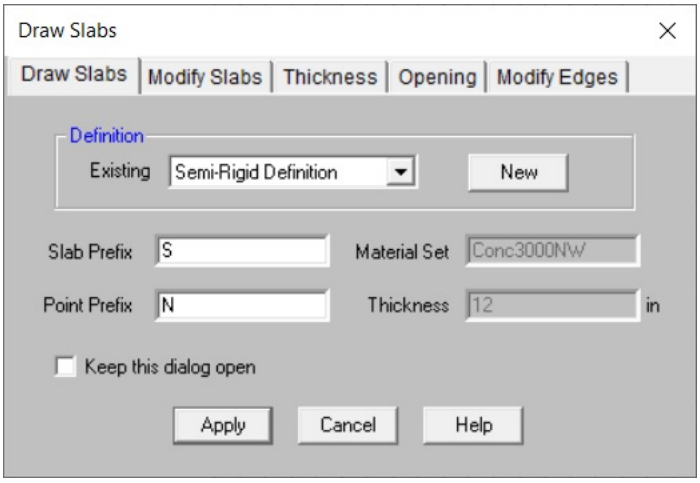
Note

- This topic refers explicitly to elevated slab floors. For information on deck information for beam supported floors see the Decks and Diaphragms topic.
- For information regarding analysis see the Elevated Slabs - Analysis topic.
- For information regarding design strips see the Design Strip Modeling, Design Strip Modeling Tips, Design Strip Design, Design Strip Detail Reports and Design Strip Spreadsheet Results topics.

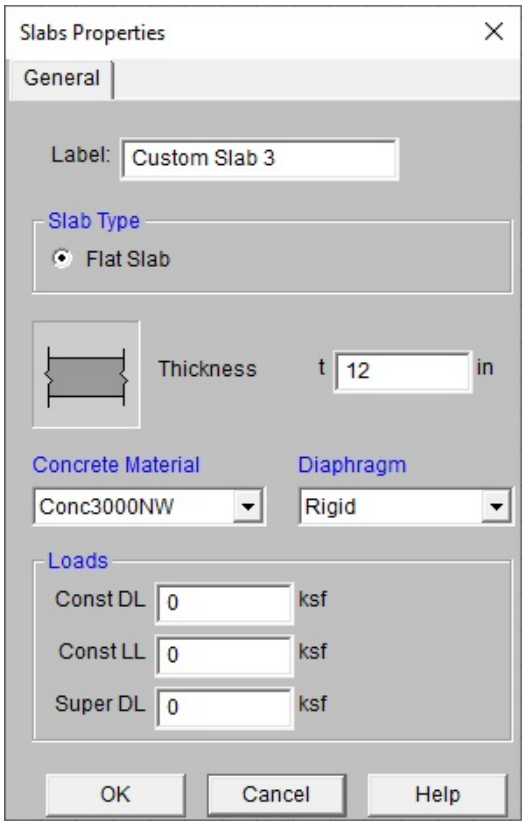
Slab edges and must be defined to establish the extents of the slab. Properties for the slab are defined in the Slab Definitions spreadsheet.

Drawing Slabs

To create slab edges or openings click on the **Drawing Toolbar** to open the **Draw Slabs** dialog.



Enter the Slab Definition you want to use or create a new one by pressing the **New** button.



Here is where to define the properties of the slab. See the Slab Definitions spreadsheet information below for more information.

To Draw a Slab

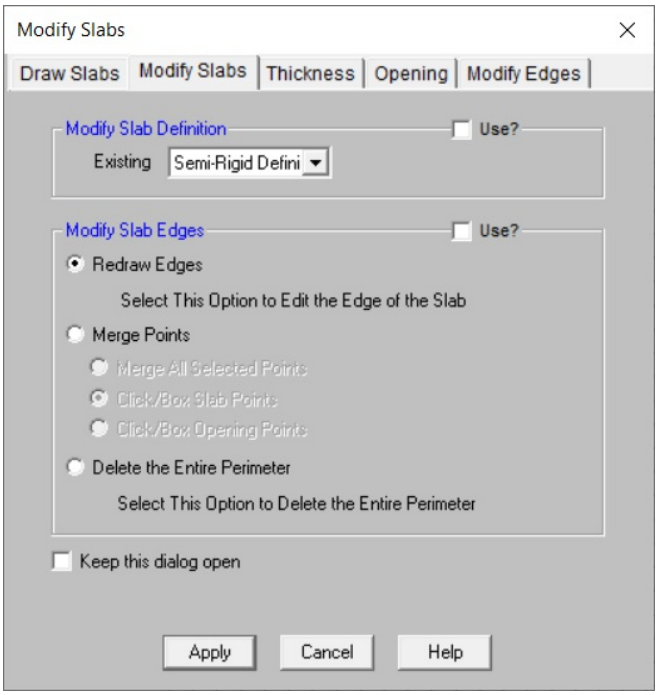
1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. If you do not already have beams or walls modeled to support the diaphragm edge you will need to create them first.
3. Click the button and set the edge distance.
4. Click **Apply.** Using the Project Grid, Drawing Grid Universal Snap points or member snap points draw the slab.

Note

- Press CTRL-D to quickly recall any of the dialogs and make changes.
- The coordinates of the closest grid or point to your cursor are displayed in the lower right hand corner.
- You may undo any mistakes by clicking the Undo button.

Modifying Slabs

To modify slabs edges click on the **Drawing Toolbar** and click on the **Modify Slabs** tab.



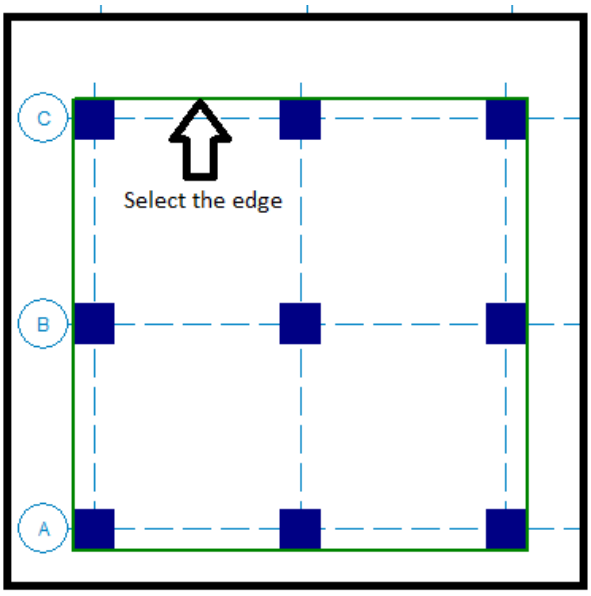
Modify Slab Definition

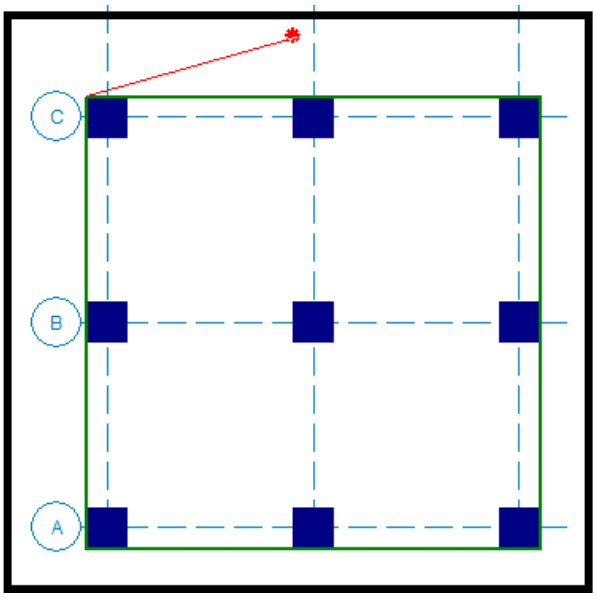
This will allow you to click on a slab graphically to change its**Slab Definition**. The Slab Definition defines the type of slab, thickness and loading. Refer to the Slab Definitions spreadsheet for more information.

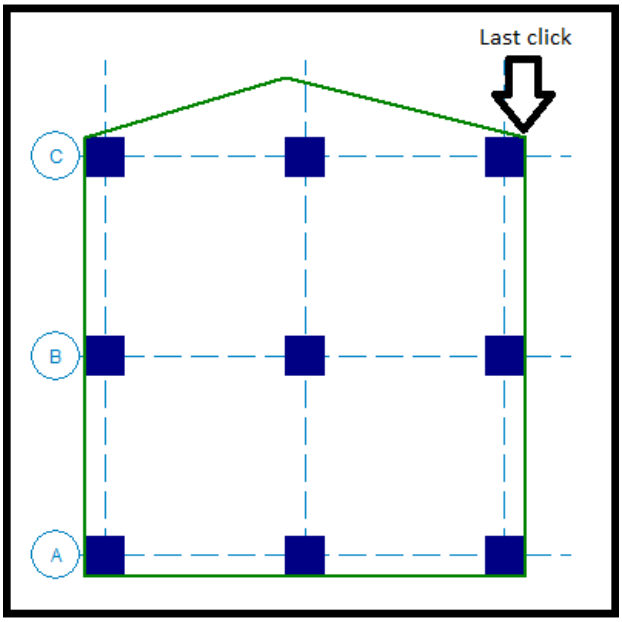
Modify Slab Edges

Redraw Edges

This option will let you take individual edges of the slab and re-draw them to a different geometry than they were individually drawn. This tool works on one entire straight line edge. The first step is to click on the edge you want to modify and then the cursor will be attached to the starting point of that edge with a red line. You can continue to click on any number of points to redefine this edge and the final click should end at the original end point of the edge.

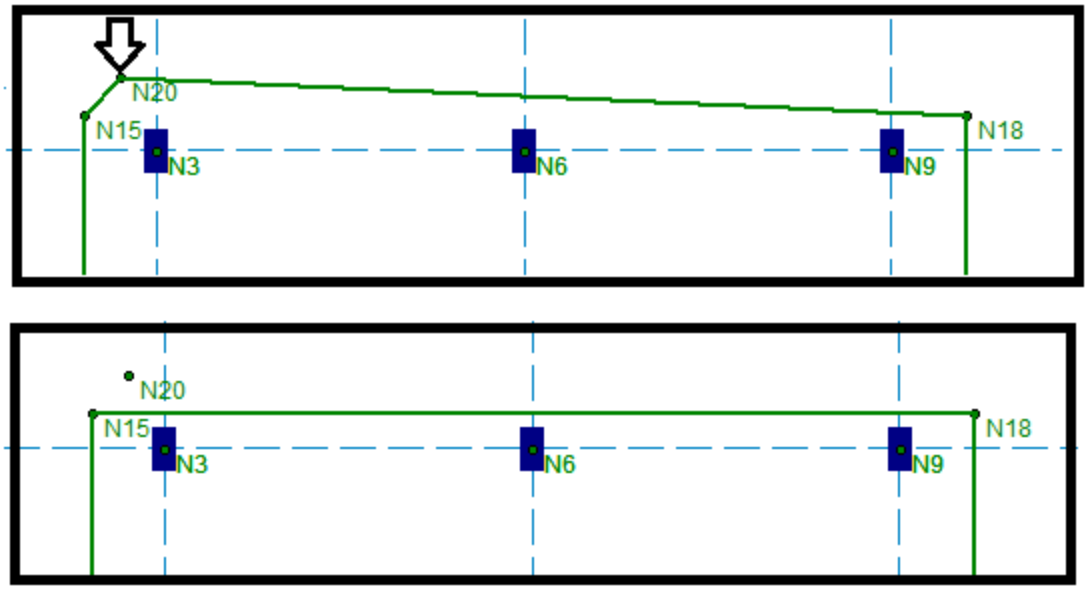






Merge Points

This option allows you to remove a point or node from the slab edge. Below is an example of a corner defined by two points. Selecting **Merge Points** and clicking on N20 removes that point from the slab edge. This tool can be used in conjunction with the **Redraw Edges** to redefine the slab edge if it's draw incorrectly. This tool can be used on the slab edges or you can use it for the openings.



Delete the Entire Perimeter

This option can be used to remove the entire perimeter so that it can be re-drawn. Select this option and you can then click on any edge in the model and it will be deleted.

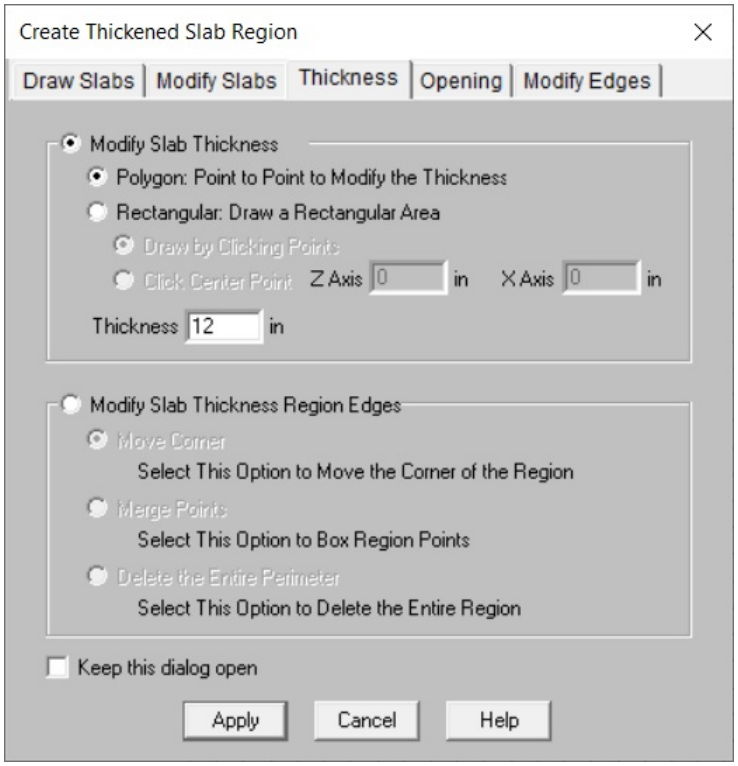
Note

- Make sure to check the "Use" checkbox to actually update your slab.
- It may be necessary to merge edges prior to redrawing them.
- You may undo any mistakes by clicking the Undo button.

Thickness

The slab thickness can be adjusted anywhere within a slab. The slab thickness will affect the design of the slab including design strip reinforcement design, punching shear and the FEA plate thickness.

To modify slabs thickness, click on the **Drawing Toolbar** and click on the **Thickness** tab.



Modify Slab Thickness

Polygon: Point to Point

Click Point to Point to define a polygon of any size or shape by selecting three or more points.

Rectangular: Draw by Clicking Points

To define a rectangular shape of any size, first click is the one corner, then stretch the mouse to create the desired size and the second click is the diagonal opposite corner.

Rectangular: Click Center Point

To define a rectangular shape of any size, define the size of the rectangular in the Z and X axis and then click the center point.

Modify Slab Thickness Region Edges

Move corner

To move the corners of thickened region(s).

Merge Points

To remove the corners of thickened region(s).

Delete the Entire Perimeter

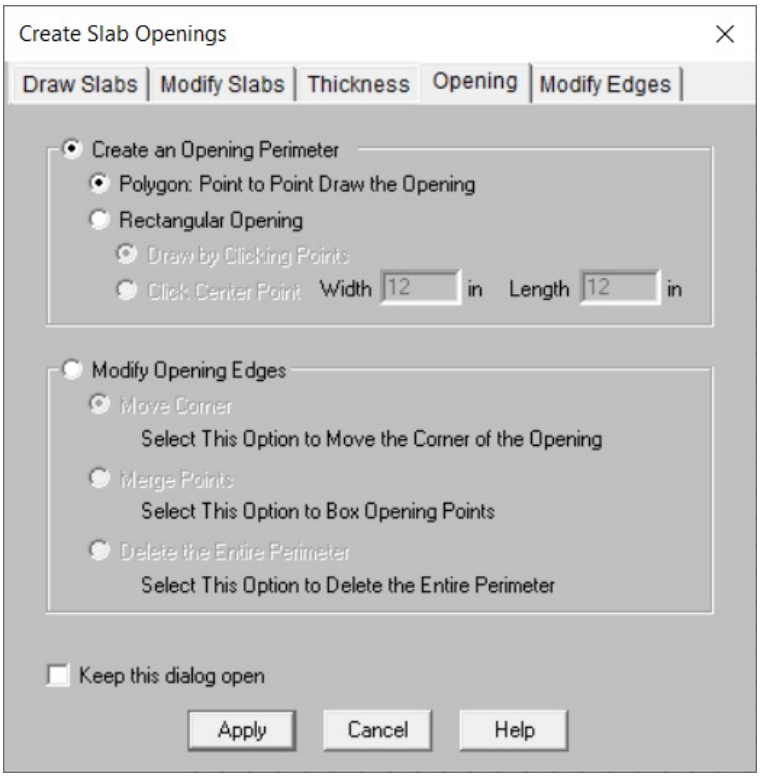
To delete thickened region(s).

Note:

- A slab thickness region cannot overlap an opening.
- The punching shear must be fully contained in a thickened region in order to use the thickness.
- The design strip cut must be fully contained in a thickened region in order to use the thickness.
- In the event of a conflict with a thickened region, drop cap or shear cap, the smallest thickness will be used for design.

Drawing Slab Openings

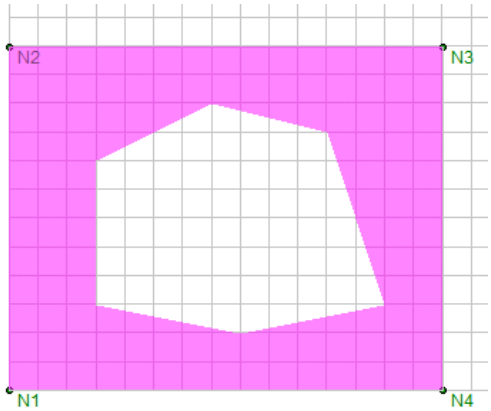
To add openings to slabs click on the **Drawing Toolbar** and click on the **Opening** tab.



Create an Opening Perimeter

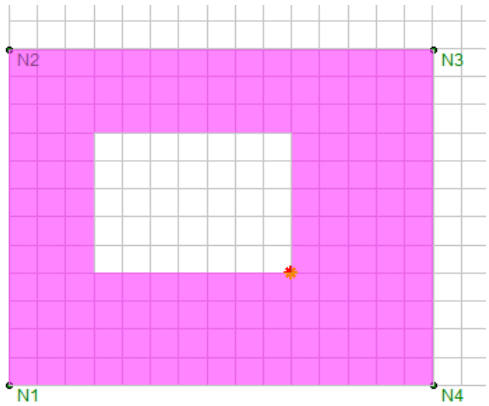
Polygon: Point to Point Draw

You can create an opening of any shape by selecting points to form a polygon shape. Click twice on the last point to indicate that the polygon is complete. Below shows an opening with a polygon shape.



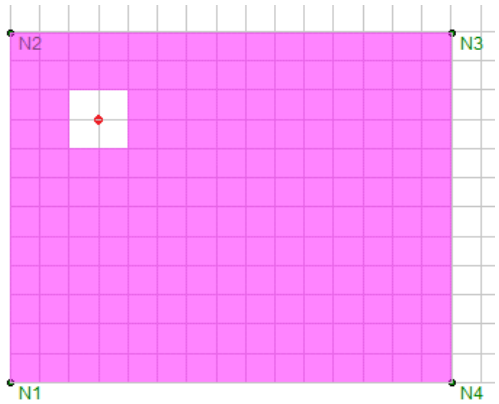
Rectangular: Draw by Clicking Points

You can create a rectangular opening of any size by selecting two points to form the rectangle. The first click is the one corner, then stretch the mouse to create the desired size and the second click is the diagonal opposite corner. Below shows a rectangular opening.



Rectangular: Click Center Point

You can create a rectangular opening of a desired size by clicking in the center of the opening. The dimensions entered are the overall dimensions of the opening. Below shows the center point (red dot) with Z axis dimension of 24 in and X axis dimension of 24 in.



Modify Opening Edges

Move corner

To move the corners of opening(s).

Merge Points

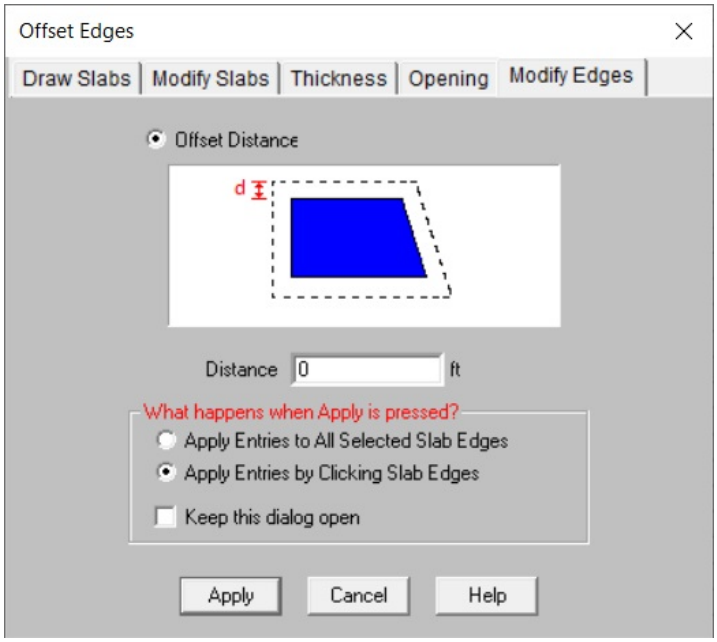
To remove the corners of opening(s).

Delete the Entire Perimeter

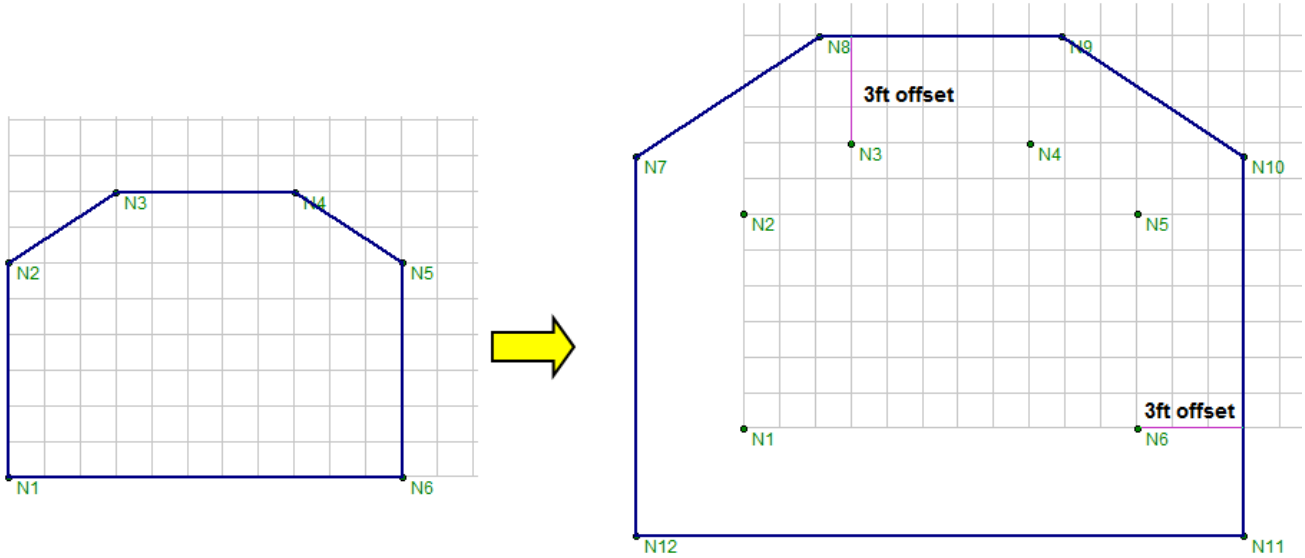
To delete opening(s).

Modify Edges

To offset a slab click on the **Drawing Toolbar** and click on the **Modify Edges** tab. This tool allows you to enlarge or minimize the size of a slab using an distance offset by entering a offset distance and clicking on the slab to be offset. You can input a positive number to enlarge the slab or a negative number to minimize the slab.

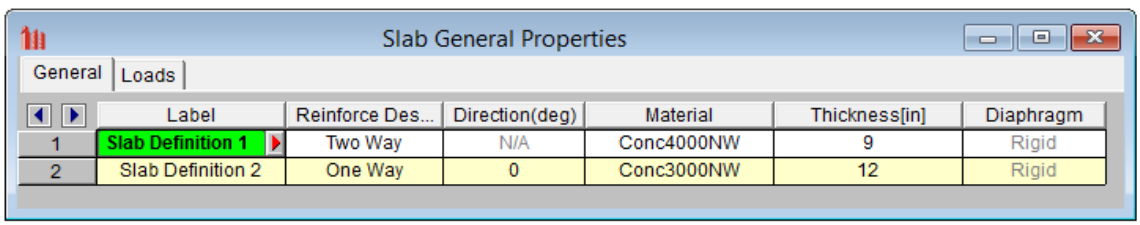


For example, if you indicate a 3 ft offset distance the slab edges will be offset linearly by 3 ft on all sides.



Slab Definitions - General

The **Slab Definitions** spreadsheet records slab properties and may be accessed by selecting **Slab Definitions** on the **Spreadsheets** menu. The spreadsheet has two tabs: General and Loads.



	Label	Reinforce Des...	Direction(deg)	Material	Thickness[in]	Diaphragm
1	Slab Definition 1	Two Way	N/A	Conc4000NW	9	Rigid
2	Slab Definition 2	One Way	0	Conc3000NW	12	Rigid

The **General** tab holds the general data for the slabs. The Loads tab contains self-weight and other load information for the slab.

Note:

- In the Label column is a red arrow that can be used to pull up the Slab Properties dialog to add the spreadsheet information in a dialog which will then populate the spreadsheet.

Label

A unique label may be assigned to all of the slab definitions manually or automatically. You can then refer to the slab definition by its label. You may relabel the slab definition at any time without affecting the application of the properties to the model.

Reinforce Design

This defines whether the design is a Two-Way slab design or One-Way slab design. The One-Way model behavior is very different than Two-Way slab, see One-Way Slab Design for more information.

Direction

When choosing a One-Way slab design, then a default direction of the span of the slab must also be defined. This direction can be overwritten with the**Set One-Way Slab Properties**button in the drawing toolbar. See One-Way Slab Design for more information. If you choose Two-Way slab design, this column doesn't apply.

Material

This defines which concrete material is being used for the slab definition.

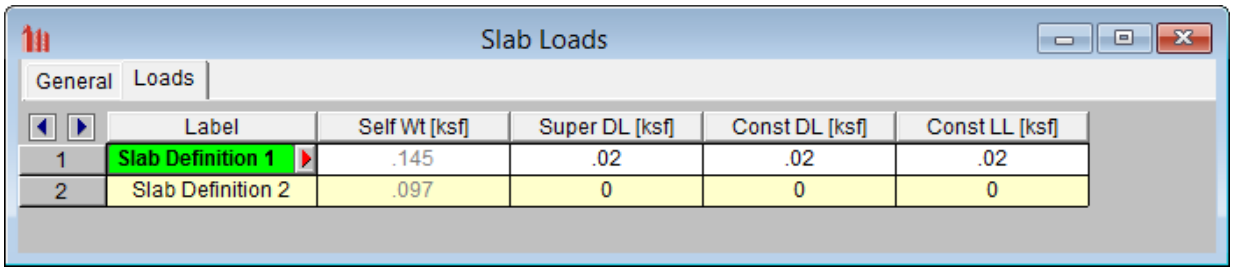
Thickness

This defines the thickness of the slab.

Diaphragm

This defines a Semi-Rigid or Rigid diaphragm in RISA-3D. Semi-Rigid can only be used with Two-Way Slabs. For more information on these options see the Diaphragms topic.

Slab Definitions - Loads



	Label	Self Wt [ksf]	Super DL [ksf]	Const DL [ksf]	Const LL [ksf]
1	Slab Definition 1	.145	.02	.02	.02
2	Slab Definition 2	.097	0	0	0

The Loads tab holds all the load data for the slab.

Self Weight

Self Weight is the weight of the slab system. This weight will be used as part of the pre-composite dead load (PreDL) and also as part of the post-composite dead load (DL) when creating load combinations for solution.

Superimposed Dead Load

The Superimposed Dead Load (Super DL) is additional load that will be treated as additional slab self weight. This might be useful to account for deck topping or flooring materials that will be part of the dead load of the floor system. This load will be included for any calculations that include deck self weight is included (seismic mass, beam camber and any places the refer to load categories DLPre and DL).

Construction Dead Load

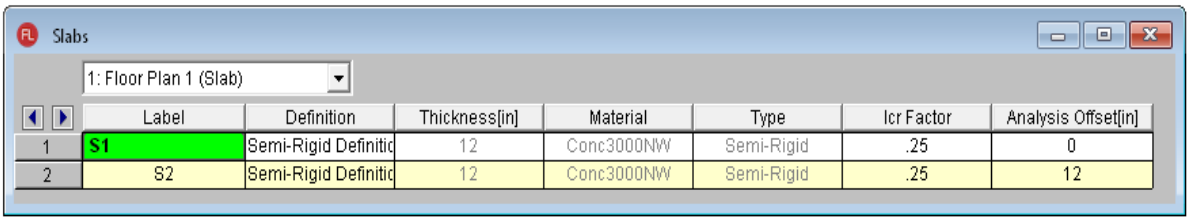
You may define construction dead load that occurs during construction but is not part of the final load on the system. This weight will be used as part of the pre-composite dead load (PreDL) but not included as part of the post-composite dead load (DL) in the load combinations for solution.

Construction Live Load

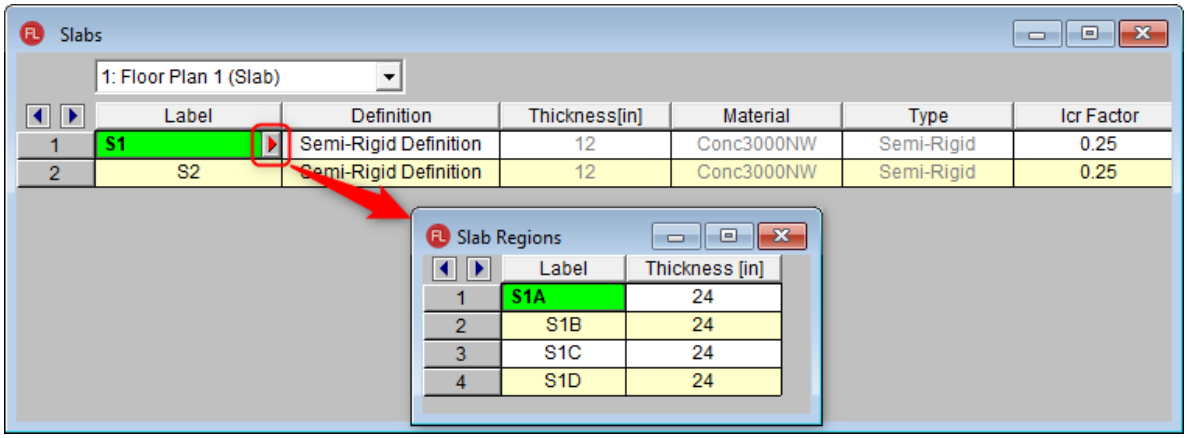
You may define construction live load that occurs during construction but is not part of the final load on the system. This weight will be used as part of the pre-composite live load (LLConst) but not included as part of the post-composite live load (LL) in the load combinations for solution.

Slabs Spreadsheet

All elevated slabs will show up in the **Slabs** spreadsheet on a per floor basis. Here you can update the **Slab Definition** and the **Icr Factor**.



If you click the triangle button next to a slab Label, there is a separate spreadsheet to show all the slab regions for the current slab. You can modify the Label, and Thickness for each slab region.



Icr Factor

The **Icr Factor** is used to reduce the bending stiffness of slabs per ACI 318-14 Section 6.6.3.1.1 (ACI 318-11 Section 10.10.4.1). This value defaults to 0.25.

Note

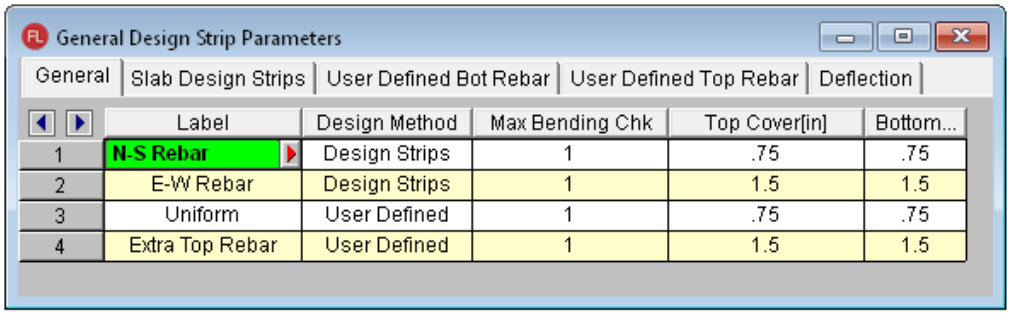
- The **Icr Factor** will be ignored if the **"Use Cracked Slabs"** is not selected in the **Concrete** tab of the **Model Settings** dialog.
- The alternative calculation in ACI 318-14 Table 6.6.3.1.1(b) (ACI 318-11 Equation 10-9) is not considered.
- The sustained load reduction of ACI 318-14 Section 6.6.3.1.1 (ACI 318-11 Section 10.10.4.2) is not considered.
- Long Term Deflection will also apply the Icr for the Service Load Combinations. These results will be displayed for the Service Load Combinations in the graphical display, spreadsheet results and Detail Report.
- See the Elevated Slabs - Analysis topic for more information.
-

Elevated Slabs - Design Strip Modeling

Once you have a slab you can apply Support Lines and Design Strips to your model. Design Strips are used to design reinforcement for slabs. If a design strip is drawn over various slab thicknesses, each design cut within the design strip will use the thinnest thickness that particular cut intersects. Here we will discuss the drawing of Design Strips and talk about design strip rules.

The first step in getting reinforcement design for an elevated slab floor is defining design strip rules. To define design strip rules, click the **Slab Design Rules** button on the **Data Entry** toolbar.

Slab Design Rules - General



	Label	Design Method	Max Bending Chk	Top Cover[in]	Bottom...
1	N-S Rebar	Design Strips	1	.75	.75
2	E-W Rebar	Design Strips	1	1.5	1.5
3	Uniform	User Defined	1	.75	.75
4	Extra Top Rebar	User Defined	1	1.5	1.5

Max Bending Check

Setting a **Max Bending Check** value controls the rebar which the program chooses for the slab design. A value of 1.0 denotes that the program may choose a rebar layout that is at 100% of capacity. To build in extra safety a value < 1.0 may be used.

Top & Bottom Cover

This is used to define the cover to the outer-most bars in the slab. The default N-S and E-W rules are slightly offset to account for the overlapping nature of the bars in a two-way slab. You should adjust the cover to reflect the specific bar size selected.

Slab Design Rules - Slab

Design Method

There are two methods available; Design Strips or User Defined. Design Strips use the traditional methods that use Column/Middle Strips and the User Defined allows the user to define a uniform rebar.

- When you select User Defined, the program will ignore Slab Design Strips.
- When you select Design Strips, the program will ignore the User Defined Bot Rebar and Top Rebar.

Rebar Sizes

The **Top Bar** and **Bottom Bar** columns define the reinforcement bar sizes that are to be used. You may specify your rebar set in the Model Settings - Design.

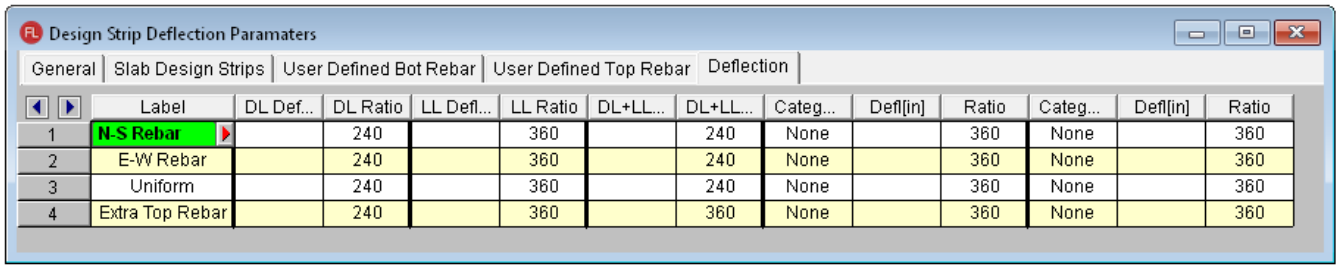
Deformed welded wire sizes can be selected from the **Top Bar** and **Bottom Bar** drop down list when enabled in the Model Settings - Design.

Rebar Spacing

The **Max Top Bar Spacing**, **Min Top Bar Spacing**, **Max Bot Bar Spacing**, and **Min Bot Bar Spacing** columns are used to set the maximum and minimum bar spacing for the slab reinforcement. The units are listed in the column headers.

The **Spacing Increment** column sets the rebar spacing increment that will be used by when analyzing various rebar layouts between the maximum and minimum bar spacing set in the previous columns. The units are listed in the column header.

Slab Design Rules - Deflection Parameters



	Label	DL Def...	DL Ratio	LL Defl...	LL Ratio	DL+LL...	DL+LL...	Categ...	Defl[in]	Ratio	Categ...	Defl[in]	Ratio
1	N-S Rebar		240		360		240	None		360	None		360
2	E-W Rebar		240		360		240	None		360	None		360
3	Uniform		240		360		240	None		360	None		360
4	Extra Top Rebar		240		360		360	None		360	None		360

The deflection is calculated based on the span length measured along the support line from face of support to face of support.

For further information see the Deflection Results Spreadsheet.

DL Deflection and Ratio

Enter the maximum allowable dead load deflection and/or the minimum dead load deflection to slab span length ratio.

LL Deflection and Ratio

Enter the maximum live load deflection and/or the minimum live load deflection to slab span length ratio.

DL+LL Deflection and Ratio

Enter the maximum dead + live load deflection and/or the minimum dead + live load deflection ratio.

Category Specific Deflection and Ratio

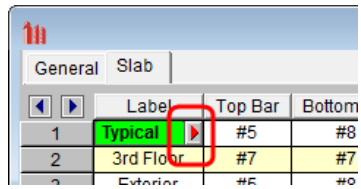
The remaining columns allow you to set deflection criteria for other load categories. Enter the category, the maximum deflection and/or the minimum deflection ratio.

Note:

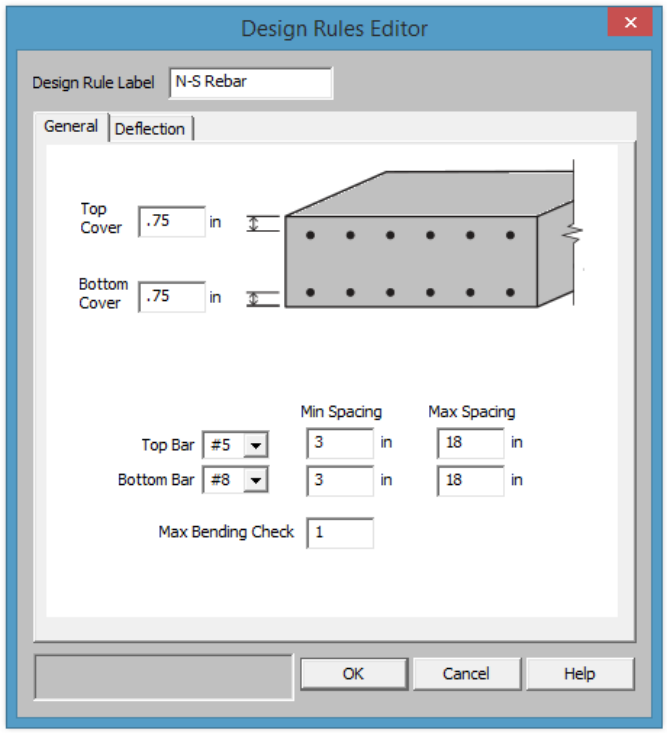
- There is no design for deflection done in the program. This information is used to perform deflection checks on the existing slab thickness. If there are deflection problems the slab thickness or geometry will need to be edited manually.

Design Strip Rules Editor

Pressing the red arrow in the **Label** column will open the **Design Rules Editor**, where the spreadsheet information can be edited with an intuitive dialog.

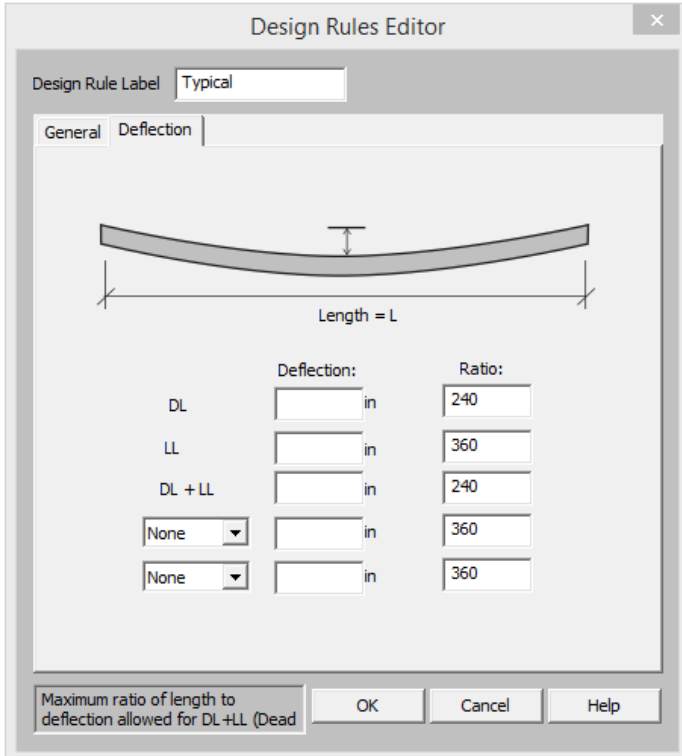


The **General** tab is used to enter all the cover spacing, bar size and spacing range, and maximum bending check.



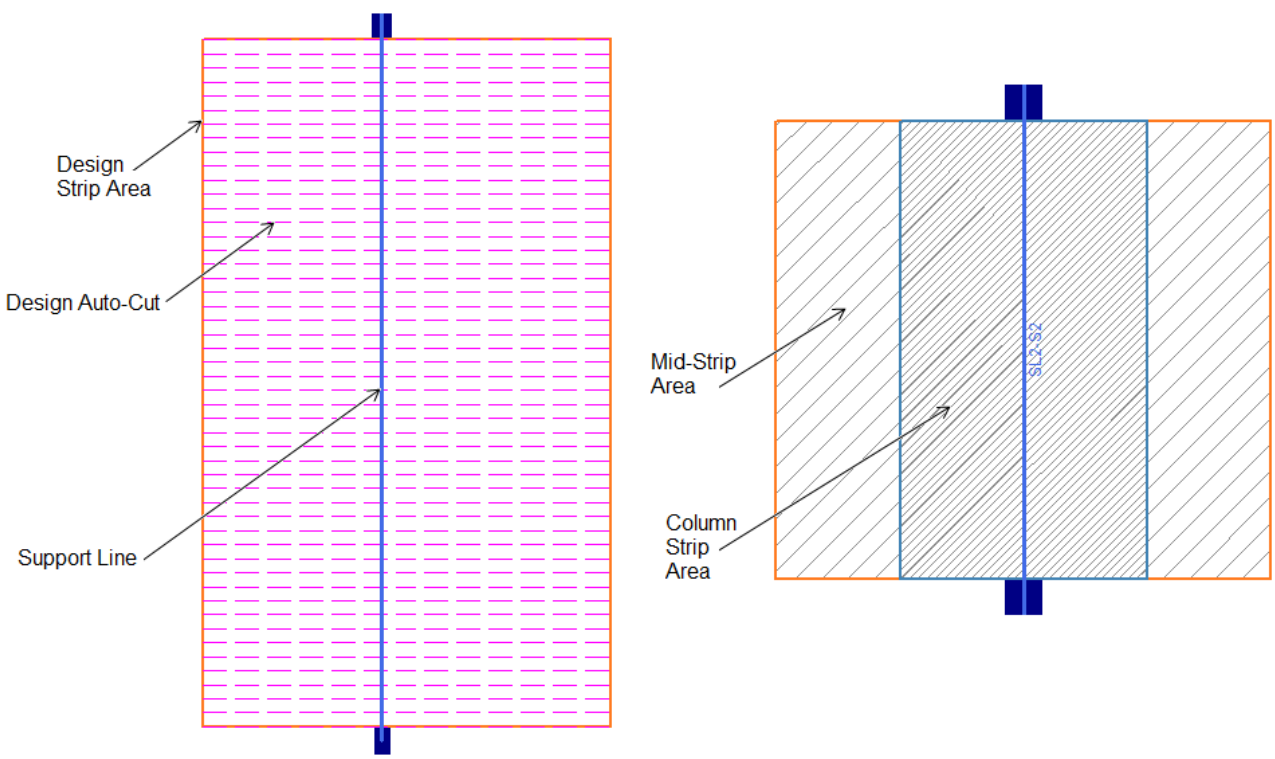
Deformed welded wire sizes can be selected from the **Top Bar** and **Bottom Bar** drop down list when enabled in the Model Settings - Design.

The **Deflection** tab is used to set the deflection limits and ratio.



Design Strip Terminology

Below is a road map to all the Design Strip terms used in the program. There are a variety of Plot Option settings as well as toolbar buttons to turn on and off the hatch and labeling. Refer to the [Model Display Options - Design Strips/Cuts] for more information.



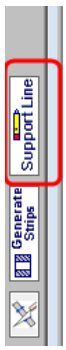
SL is the default prefix for Support Lines. You can adjust the prefix in the Slab Design Strips spreadsheet.

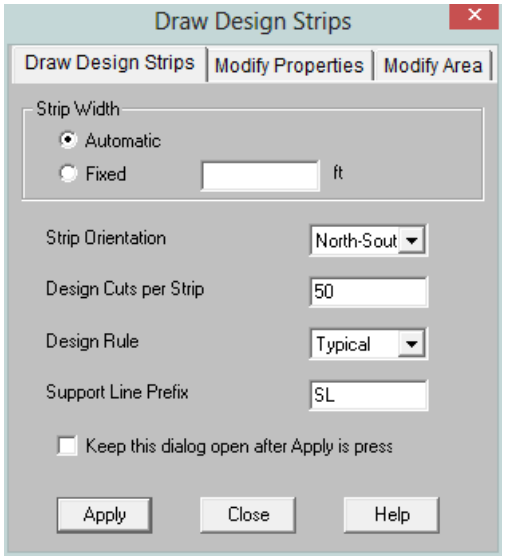
S indicates the Span and each span will be assigned a number starting at the first point the Support Line was created.

Drawing Design Strips - Support Lines

Once **Slab Design Rules** are defined then the Support Lines and Design Strips are drawn. The first step is defining Support Lines in the model.

To define Support Lines for your model press the **Support Line** button:





Strip Width

The **Automatic** option will automatically (at solution or after pressing the **Generate Strips** button) define a strip width that extends to a slab edge or a parallel design strip. The **Fixed** option allows you define the exact width of the Design Strip.

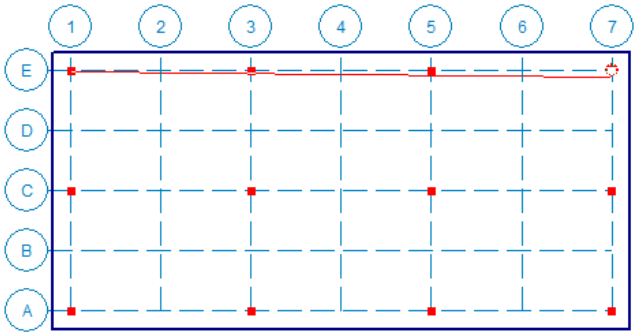
After making modifications to the slab edge or surrounding Design Strip areas, you can click the **Generate Strips** button to re-draw the extents of the Design Strip areas. If the Design Strip has been modified it's width will not be over-written and it will be drawn in purple to indicate it's modified.

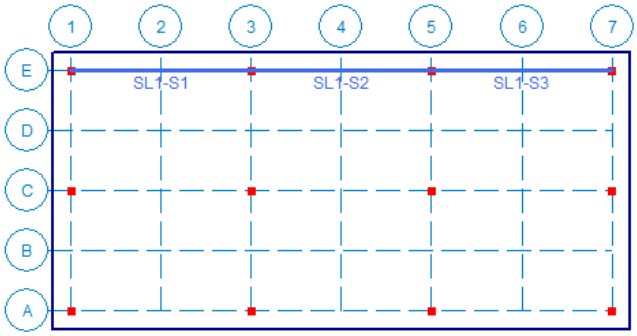
The **Strip Orientation**, **Design Cuts per Strip** and **Design Rule** entries are explained further in the Slab Design Strips spreadsheet.

Modeling the Support Lines

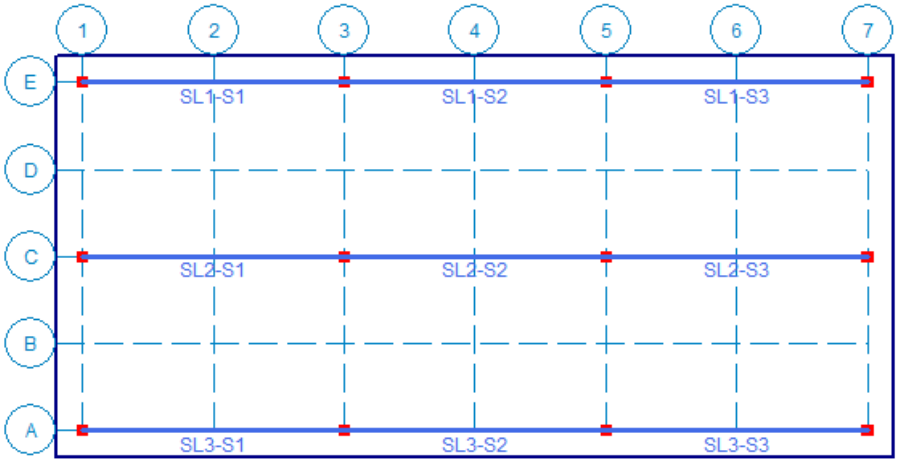
Once these items are selected, press **Apply** and we are ready to draw Support Lines.

In the example below we will start with East-West Support Lines and click on the columns as our supports. For the first Support Line we will click on the column at Grid E1 and then click the column at Grid E7.

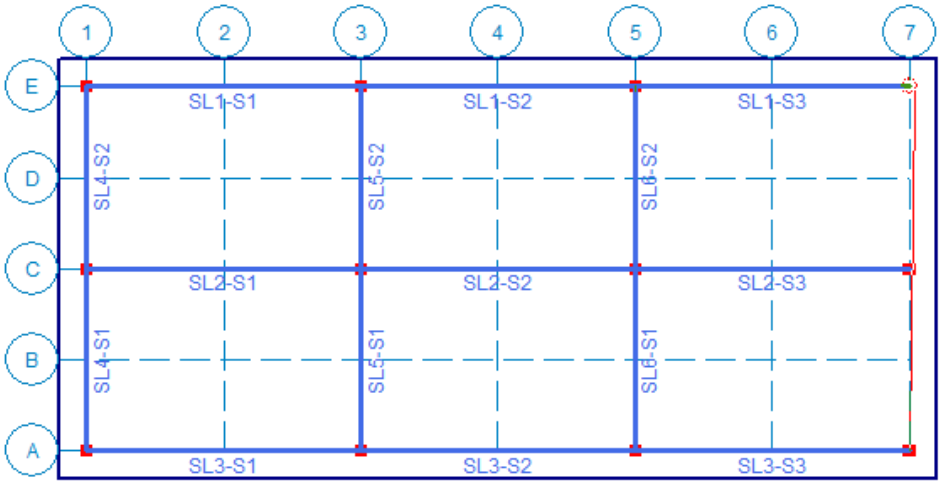




Here we see that we get a single Support Line (SL1) with multiple spans (S1-S3). We continue by drawing the rest of the Support Lines in the East-West direction.



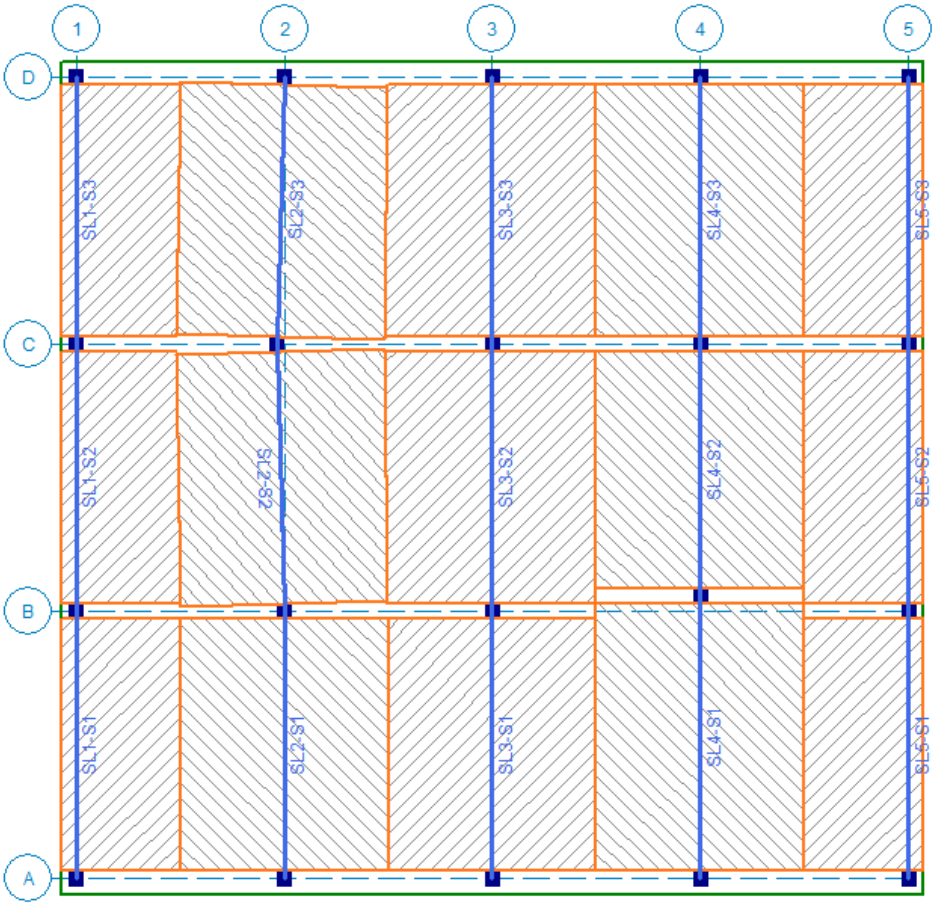
From here edit the **Draw Design Strip** dialog to draw the North-South strips.



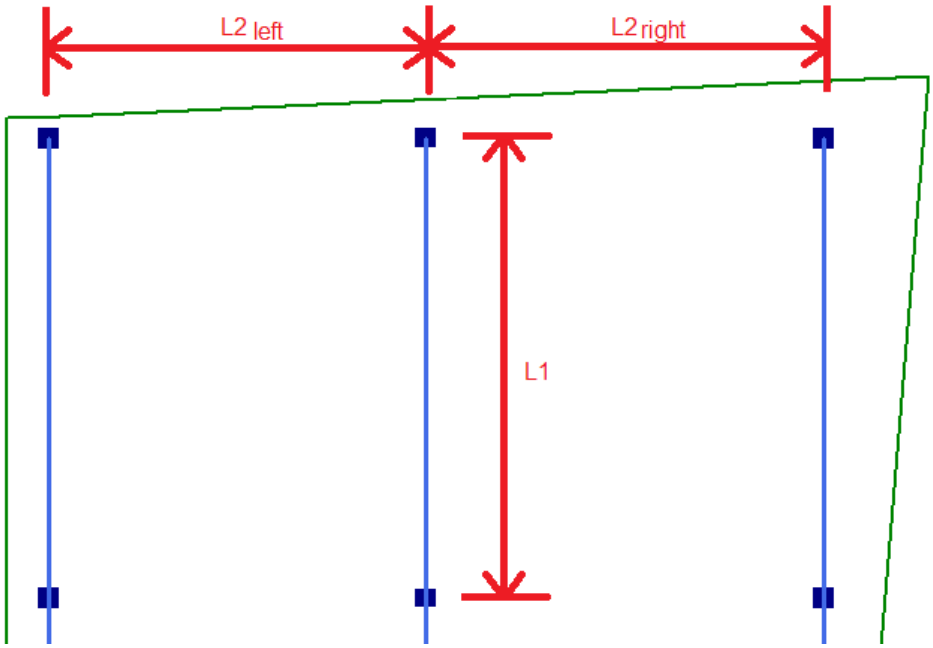
Note: You can draw a Support Line to a non-support point to create spans that don't end on support points. However, the results are based on the assumption that the span is defined from support point to support point.

Automatic Creation of Design Strip Widths

After all Support Lines are drawn then the program will automatically draw the Design Strip widths by either solving or pressing . Once this is done the strips are shown graphically. In the image below, the width of the strip is outlined in orange. If the Strips and Support Lines are drawn correctly, the design results can be reviewed after solution. See the Elevated Slab - Design Strip Results topic for more information.

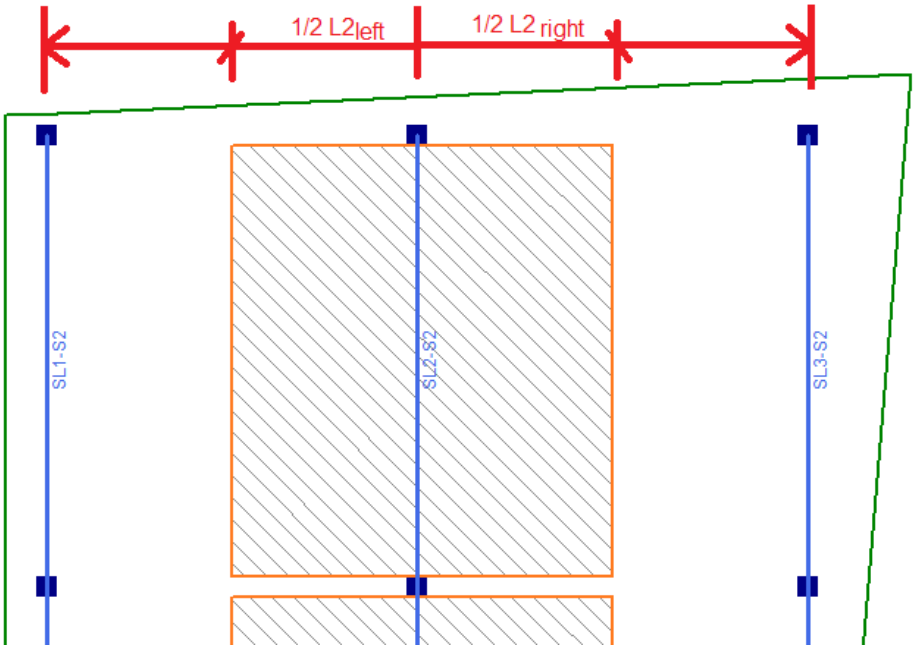


The automation of the Design Strip width is calculated based on half the distance from the Support Line to the neighboring Support Line. Neighboring Support Lines must be the same direction (N-S or E-W). Below shows a simple example of how this is calculated.

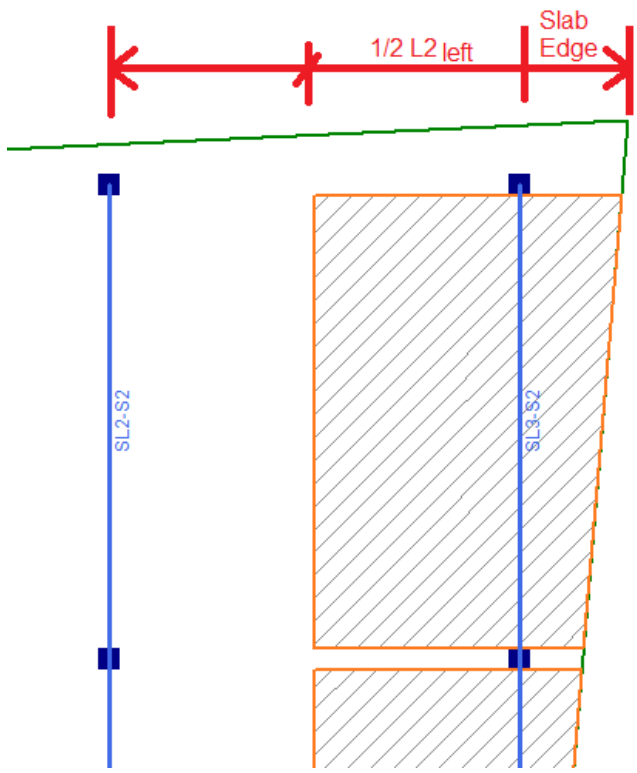


L1 = Span along the Support Line from support to support.

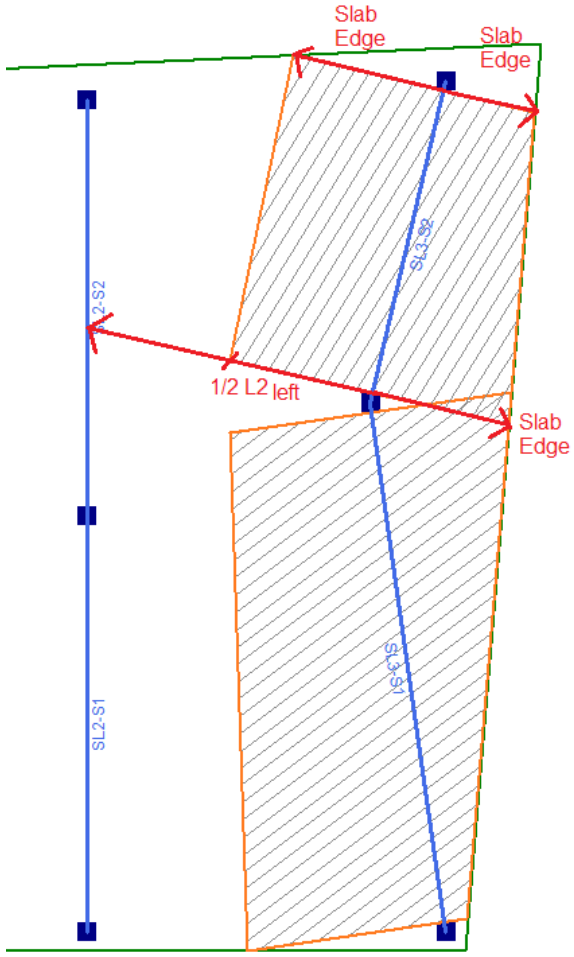
L2 = Span from the Support Line to neighboring Support Line (left and right).



If the Support Line is at an edge and there is no adjacent Support Line then the Design Strip will extend to the slab edge.



The slab edge and the Support Line distance can both vary along the distance of the Support Line. To solve this scenario, the program will find two widths for the start of the Support Line and the end. These two widths will be connected to form the Design Strip.

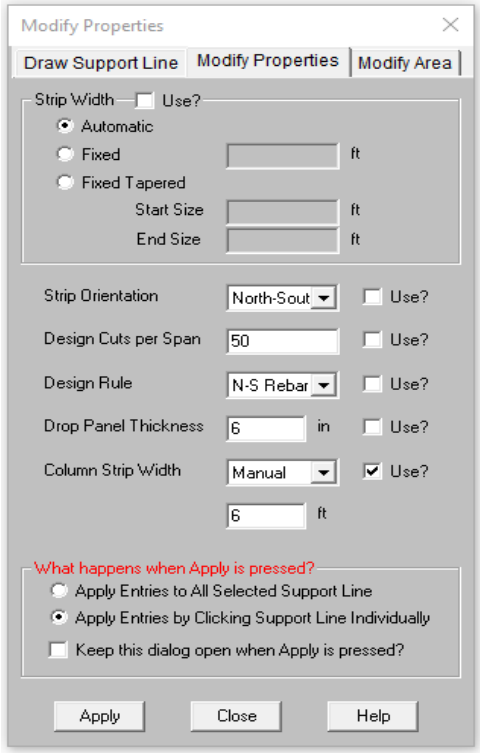


Modifying Slab Design Strips

From here we can modify these slab edges, edit/add Support Lines, etc., from the other tabs of the **Draw Design Strip** dialog.

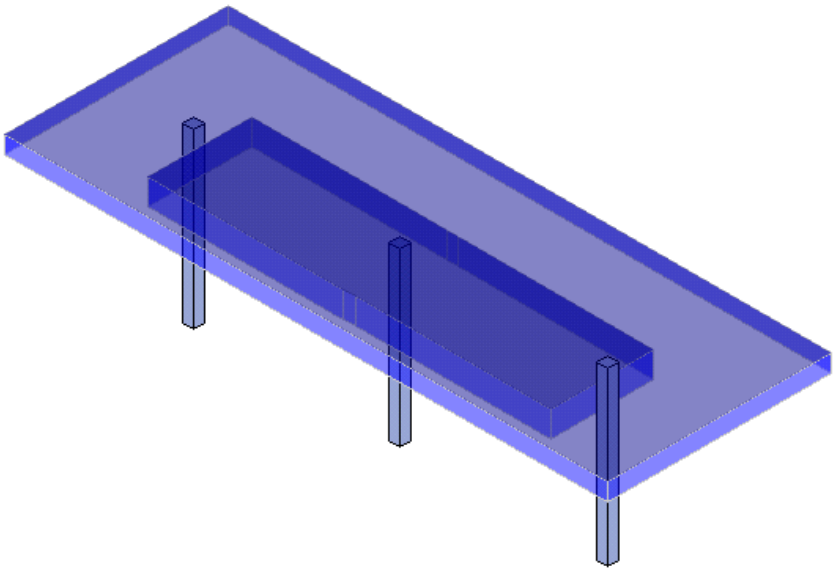
To modify Design Strips click

and click on the **Modify Properties** tab.



This allows you to edit the **Strip Width**, **Strip Orientation**, **Number of Design Cuts per Span**, **Design Rule** and **Drop Panel**. Check the appropriate **Use?** checkboxes, press **Apply** and click on individual Support Line.

Drop Panels



This allows you to edit the Column Strip thickness by adding a Drop Panel. The Drop Panel Thickness is the additional thickness below the surrounded slab. For example, the Slab Definitions spreadsheet defines the slab thickness of 12" and the Drop Panel Thickness is applied as 6" then the Column Strip thickness will be changed to 18". To thicken the slab beyond the column strip, see **Thickened Slabs.**

Once a Drop Panel is applied to the Design Strip- the Column Strip will graphically change colors to indicate the Drop Panel has been applied. Only the thickness of the Column Strip is affected by the Drop Panel, the middle strip remain the thickness of the surrounding slab. The Drop Panel will adjust the thickness for the entire support line.

Drop Panels are used to reduce the amount of negative moment reinforcement per ACI. The depth of the drop panel must project below the slab at least 1/4 of the surrounding slab thickness. The drop panel columns strip is thickened for design of reinforcement and the (FEA) plate mesh is also thickened. When the slab floor is designated as a Semi-Rigid diaphragm- the drop panel will thicken the plate mesh in RISA-3D also.

Drop Panels will affect the punching shear calculations if the shear perimeter is entirely within the drop panel. In cases where there is ever multiple thicknesses for a shear perimeter- the program will use the thinnest thickness.

Note:

The slab thickness can also be altered using Thickened Regions.

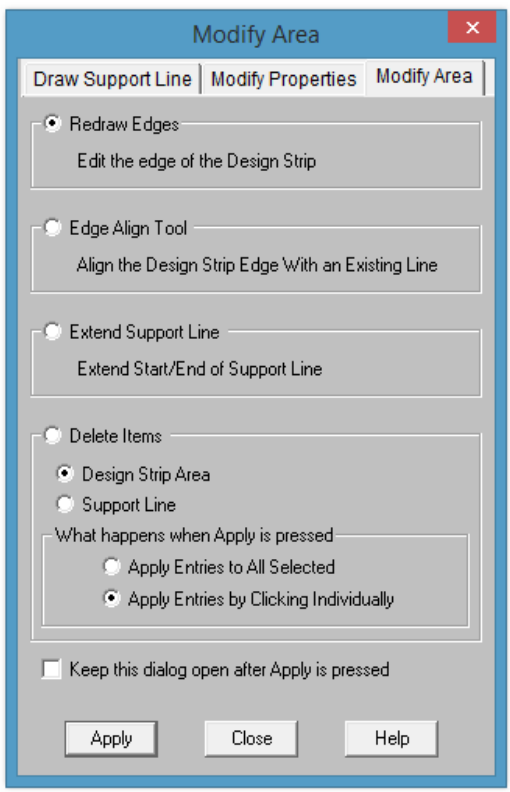
Column Strip Width

This allows you to edit the Column Strip width manually. The program will automatically calculate the Column Strip width based on the ACI code provisions, see **Column Strip Width Calculation** for full calculation details. However, this manual entry will allow you to override this ACI calculated value with your own width.

Note: When the Column Strip is defined manually the entire design strip will be modified. The ACI calculated column strip widths are automatically adjusted per span.

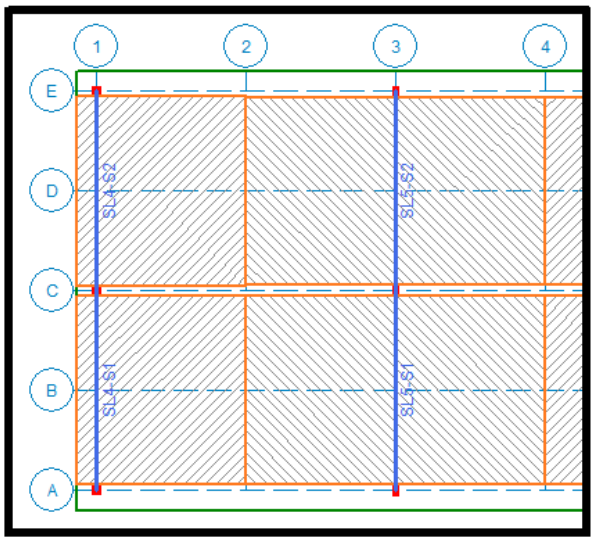
Additional Modify Tools for Design Strips

There are additional options to edit and refine Design Strips.

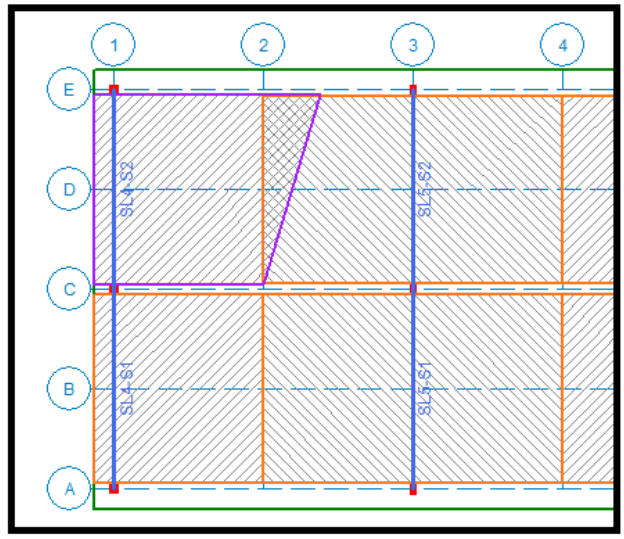


After making any modifications to the Design Strip area, the outside edges will turn purple to indicate they have been modified. If you can click the **Generate Strips** button, the modified Design Strips will not be affected and the

surrounding Design Strips will re-generate. If you intend to go back to the original Design Strip area, you can delete the Design Strip area and then click **Generate Strips** again.



Before Modifications

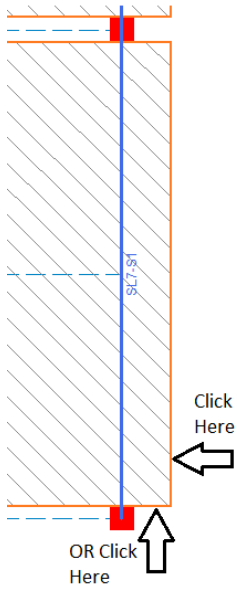


After Modifications

Redraw Edges

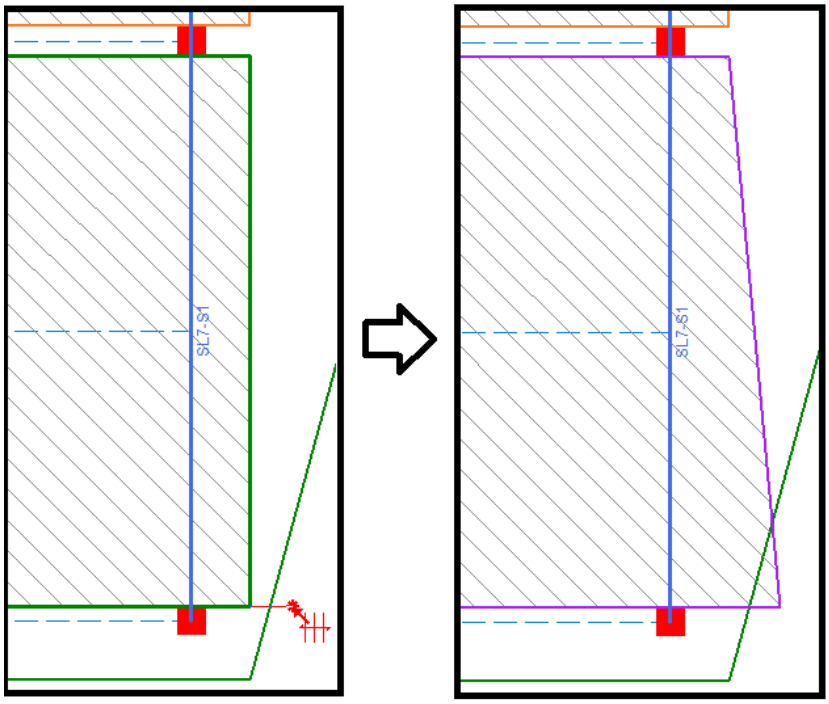
With this tool click on the edge of a Design Strip and you can then pull it out in a perpendicular direction to re-draw it.

Click on the either edge closest to the corner of the Design Strip area.



The Design Strip area edges will turn green and you can then drag the mouse closer or away from the Support Line perpendicularly. Left-click at the new location for the Design Strip corner. The Design Strip will turn purple indicating it's been modified.

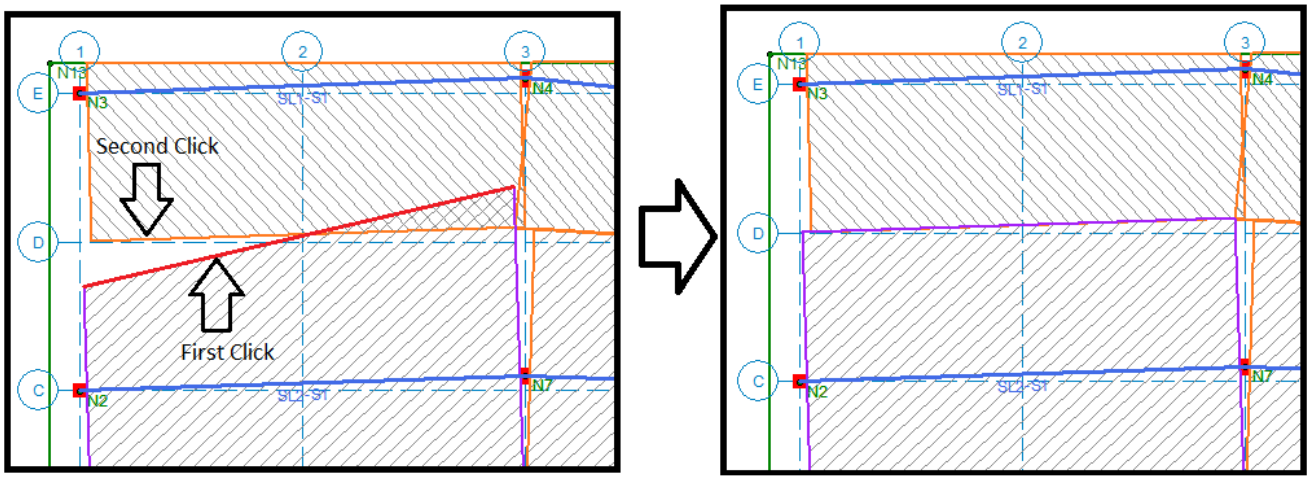
There is a minimum width of 6" from the Support Line in which the Redraw tool will not allow you to draw the Support Line smaller than that minimum.



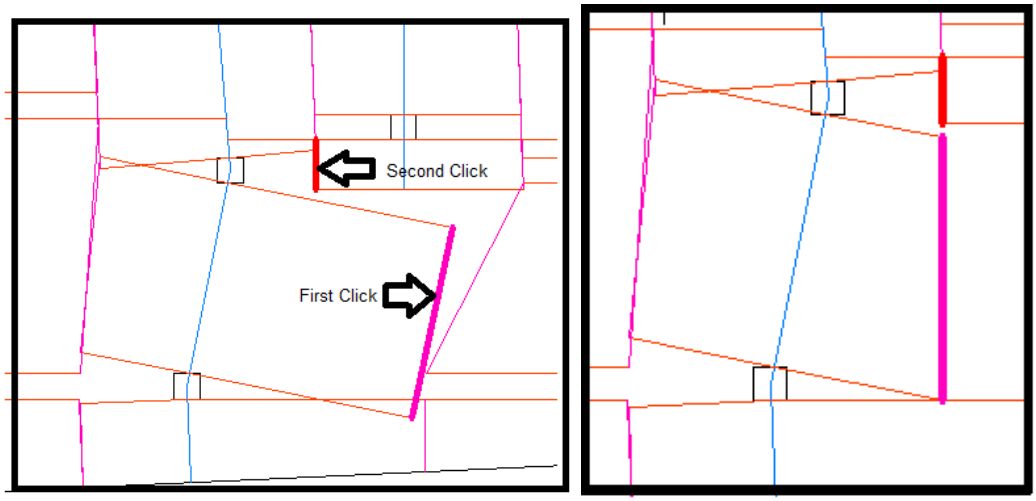
Edge Align Tool

With this tool you can click on the edge of a Design Strip and then click on a slab edge to align the Design Strip edge with the deck edge.

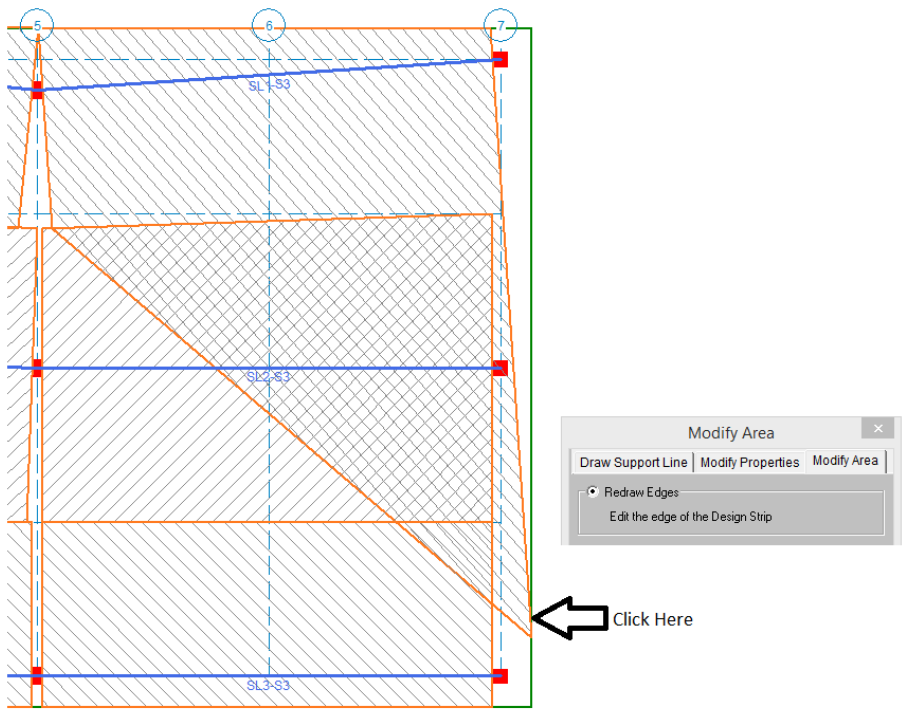
Example 1: You can align two edges using this tool by clicking first on the edge you move, and second click on the edge to align to.

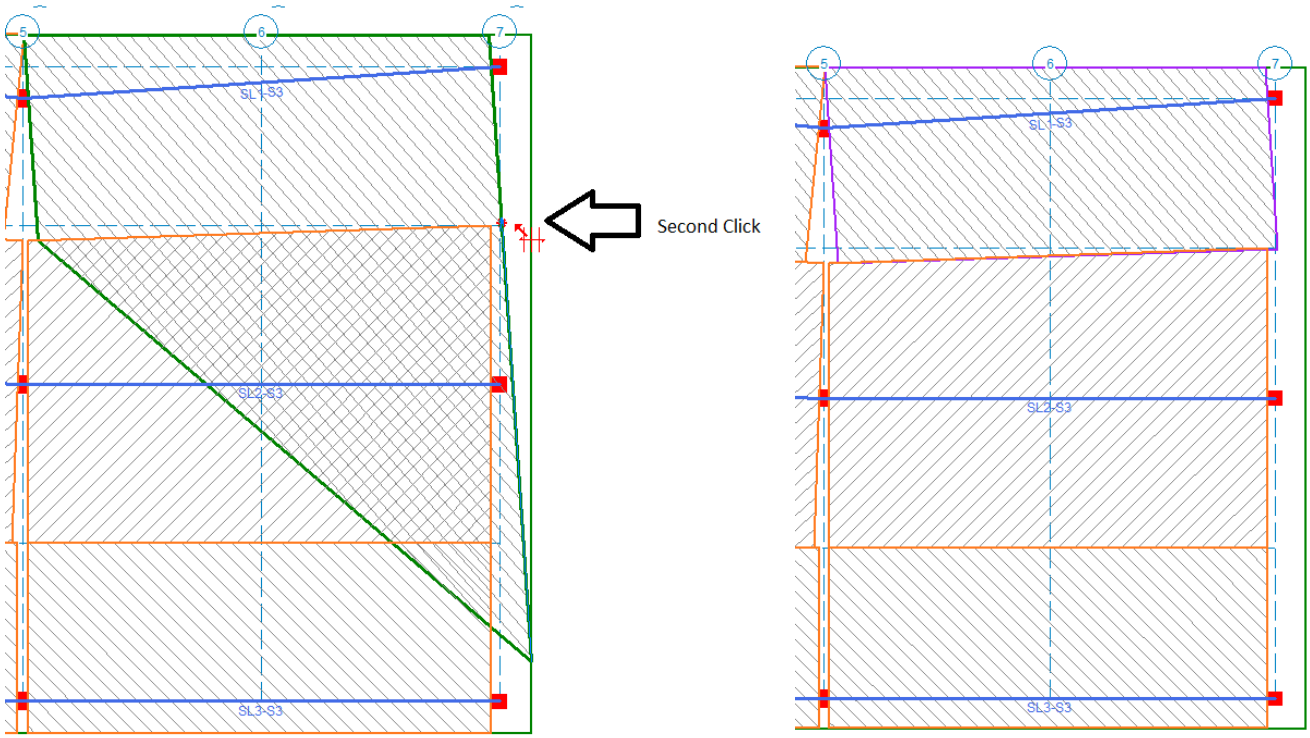


Example 2: You can align edges to any other edge in the model including non-adjacent edges.



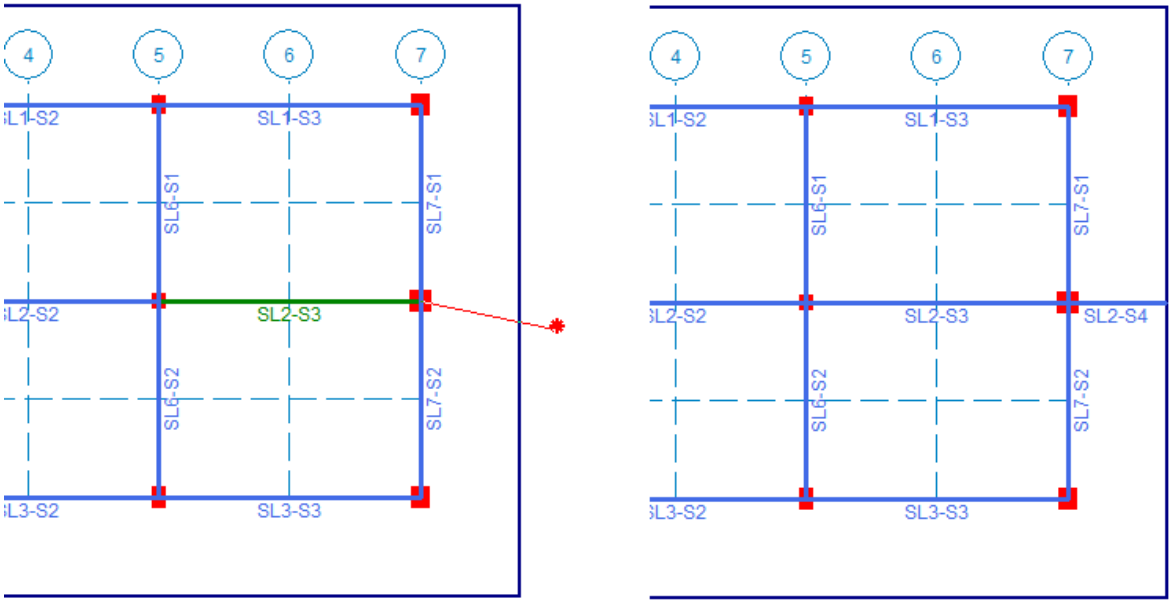
Example 3: Since the program is determining the Design Strip width based on the adjacent support line or slab edge, this example demonstrates how it may not find a perimeter nearby. This is easily corrected with the Redraw Edges tool shown below.





Extend Support Line

With this tool you can click on the end of an existing support line and drag it to lengthen it. This is a good tool to add cantilever spans to an existing support line.

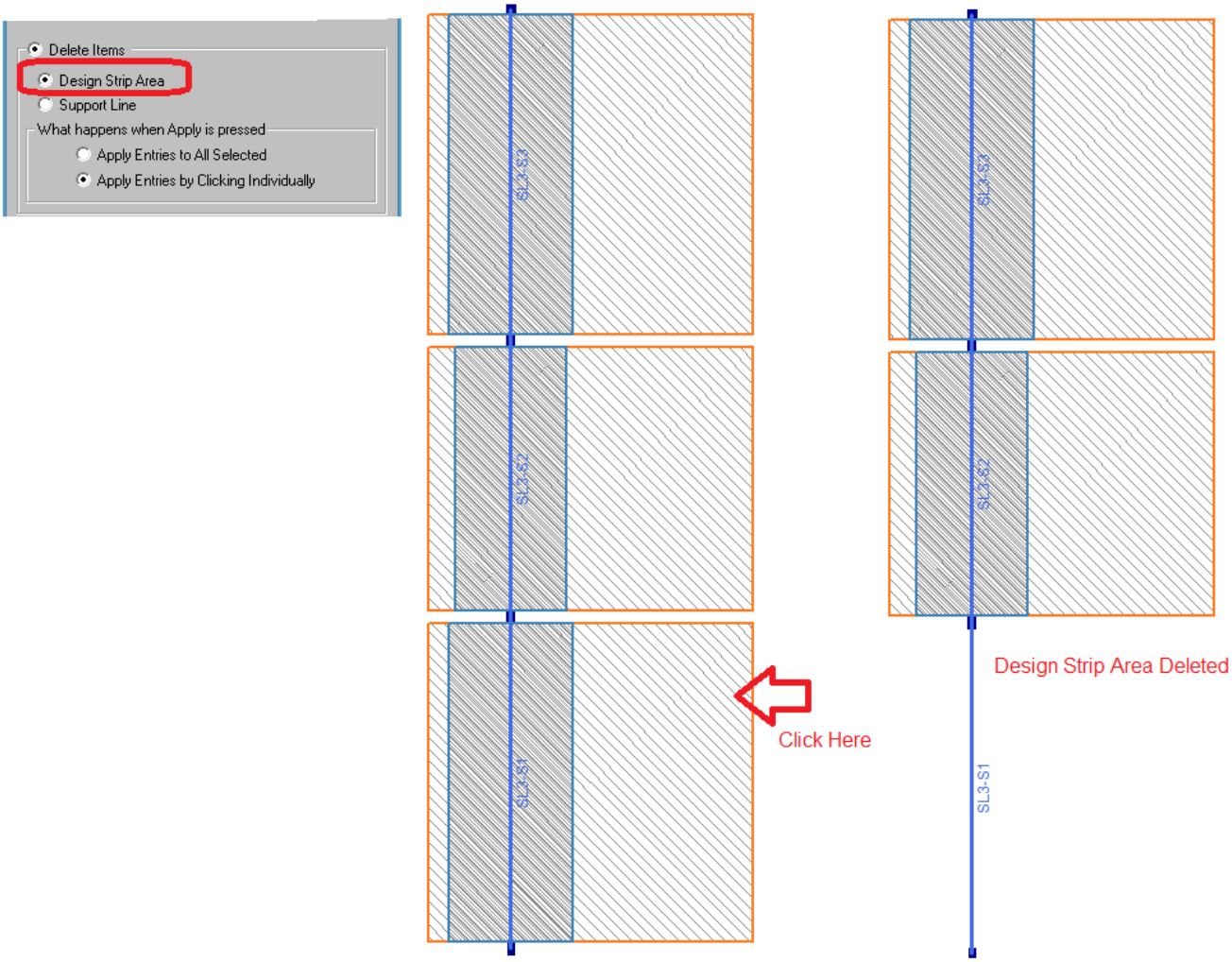


Delete Items

This tool can be used to delete specific Design Strips or Support Lines. You can delete the individual items or selected items.

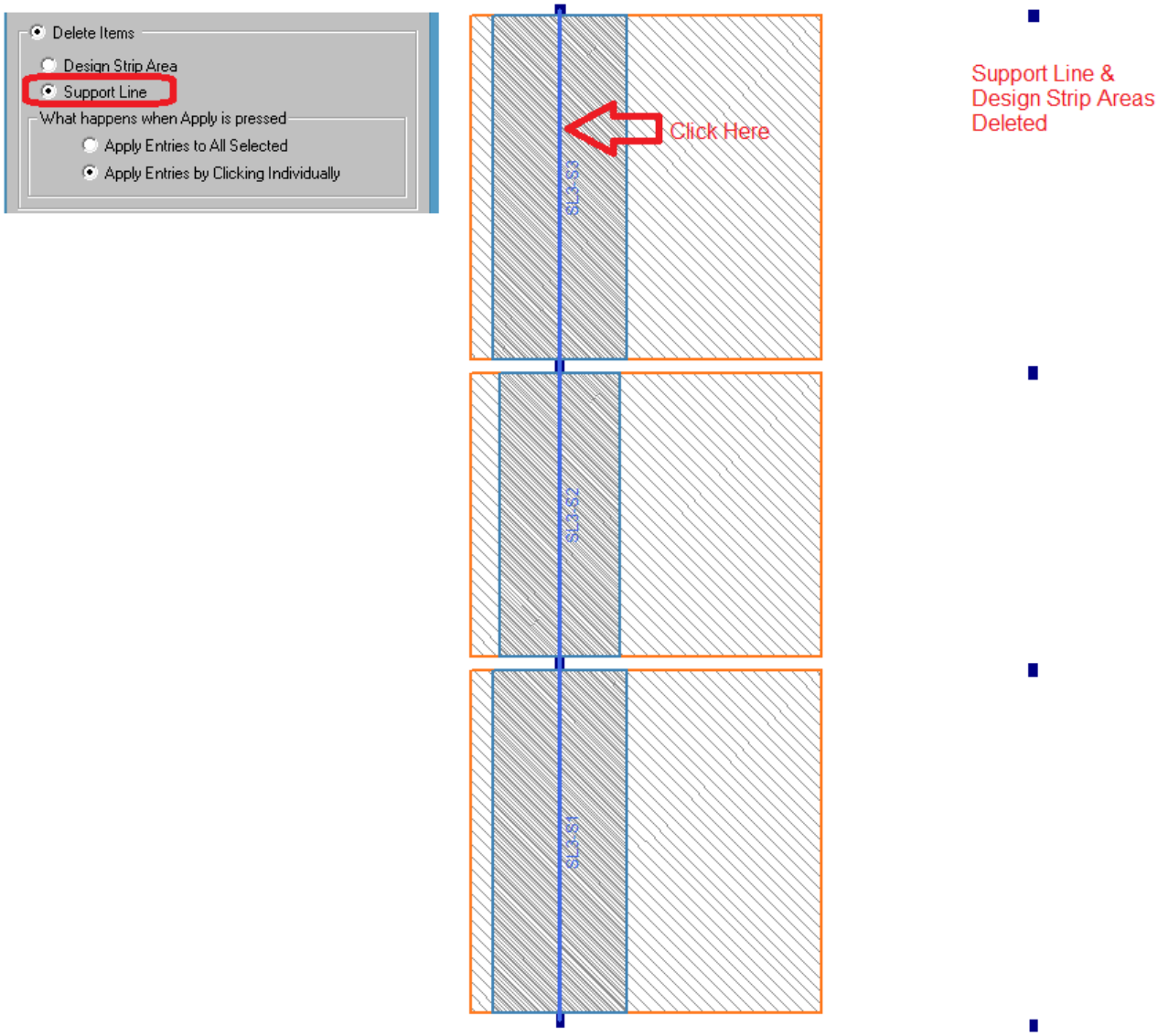
Example 1: Delete Strip Area

This allows you to delete an individual Design Strip Area when you select "**Apply Entries by Clicking Individually**". This can be helpful when you've altered the Design Strip area and you want to the program to go back to the original geometry.



Example 2: Delete Support Line

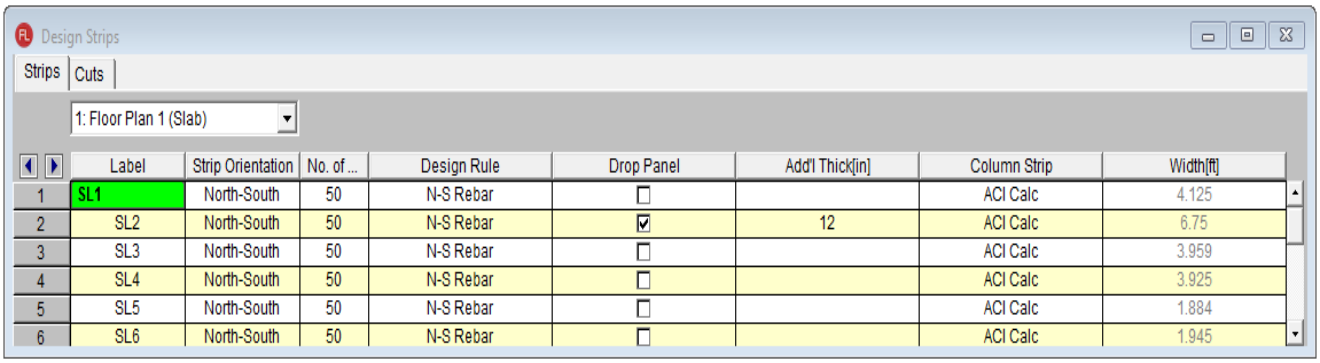
This will delete the entire Support Line AND the associated Design Strip areas. The Support Line cannot be deleted in segments so if you want to delete a section, you will need to delete the entire Support Line and re-draw it in separate segments.



You can also use the **Delete Items** tool from the Graphic Editing toolbar to delete the Support Lines and Design Strip areas.

Slab Design Strips Spreadsheet

The **Slab Design Strips** spreadsheet records all design strip support lines on each floor and may be accessed by selecting **Slab Design Strips** on the **Data Entry** toolbar.



	Label	Strip Orientation	No. of ...	Design Rule	Drop Panel	Add'l Thick[in]	Column Strip	Width[ft]
1	SL1	North-South	50	N-S Rebar	☐		ACI Calc	4.125
2	SL2	North-South	50	N-S Rebar	☑	12	ACI Calc	6.75
3	SL3	North-South	50	N-S Rebar	☐		ACI Calc	3.959
4	SL4	North-South	50	N-S Rebar	☐		ACI Calc	3.925
5	SL5	North-South	50	N-S Rebar	☐		ACI Calc	1.884
6	SL6	North-South	50	N-S Rebar	☐		ACI Calc	1.945

Strip Orientation

The **North-South** and **East-West** option allows you to define two separate sets of design strips for two orthogonal directions in the slab. The two main functions of this option:

- If using the **Automatic** strip width generation the program uses adjacent strips to determine the strip width of the current strip. If the current strip is a North-South strip the program will only look for adjacent North-South strips when defining the strip width. East-West strips would be ignored in this case.
- When viewing strip results the North-South and East-West designations can be turned on/off to make viewing of results easier. The reinforcement design can also be easily viewed in each direction separately. There are a variety of Plot Option settings as well as toolbar buttons to turn on and off the display- refer to the Model Display Options - Design Strips/Cuts for more information.

No. of Design Cuts

A Design Strip is a collection of individual Design Cuts through the width of the strip. Each strip has the design forces calculated to produce a shear and moment diagram for the strip. This setting defines how many cuts make up the Design Strip. The larger the number of strips the more accurate the results.

Note:

- The number of Design Cuts is per span. So a Support Line with 50 cuts and four spans will have 4*50 = 200 total Design Cuts.

Design Rule

The design rule defines reinforcing, deflection, cover and unity check design information for the strip. See the Design Strip Rule section for more information.

Drop Panel/Add'l Thick

The Drop Panel checkbox will add a thickened column strip region. The Add'l Thick column is the thickness below the surrounding slab. See"Drop Panels" section for more information.

Column Strip/Width

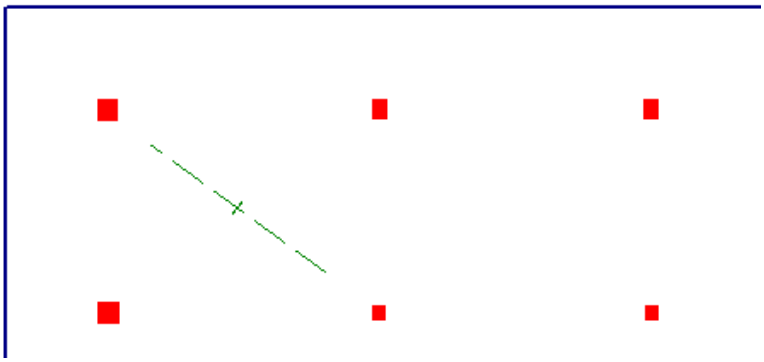
The Column Strip defines whether the program automatically calculates the Column Strip Width based on the ACI code parameters or if you Manually enter the width. See Column Strip Width for the full details of the ACI calculations.

Slab Manual Design Cuts

It is also possible to draw a single Design Cut in a slab to calculate forces at a specific location in a slab. This tool will not design the reinforcement; it will only provide analysis information. To do this press the manual **Design**

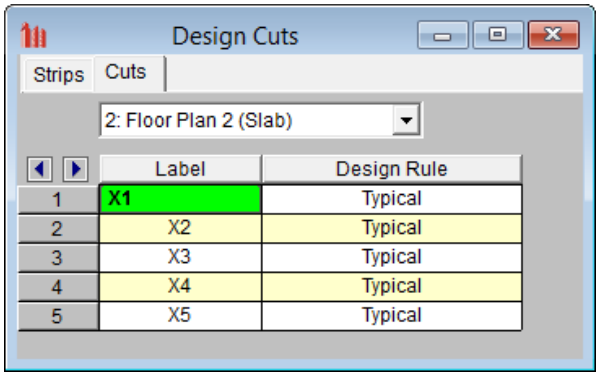
Cuts button.

From here you can free draw Design Cuts on the model.



Slab Design Cuts Spreadsheet

The **Slab Design Cuts** spreadsheet records all Design Cuts on each floor and may be accessed by selecting **Slab Design Strips** on the **Data Entry** toolbar and clicking on the **Cuts** tab.



Note:

- For information on results see the Slab Design Cuts Results section.
- Design Cuts are solely for analysis only.

Elevated Slabs - User Defined

The User Defined reinforcement selection allows the user to select uniform reinforcement and RISAFloor ES will check that reinforcement. The program is using the maximum moments found in each FEA slab plate to check the referenced code.

Step 1: Define the Slab as a User Defined Slab when you create the floor.

Step 2: Determine how you want to define the Rebar.

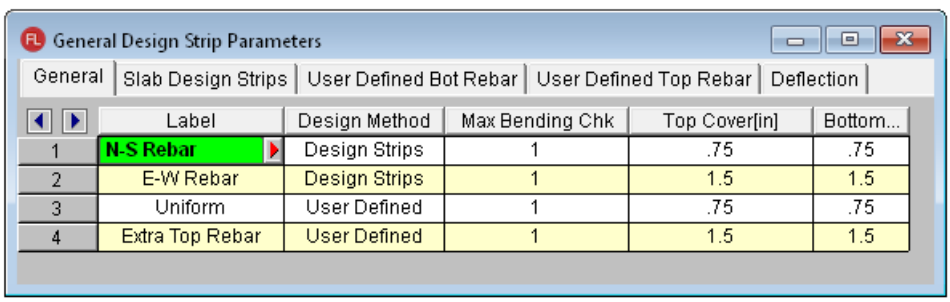
Option 1: Use continuous reinforcement for both the top and bottom of the slab.

The Slab Design Rules will define the Top and Bottom rebar. See the Floors spreadsheet to define the Top Continuous.

Option 2: Use continuous bottom rebar only and on the top place rebar in regions over the columns and walls.

The Slab Design Rules will define the Top and Bottom rebar. The Columns and Walls spreadsheets allow you to define the regions.

Slab Design Rules - General

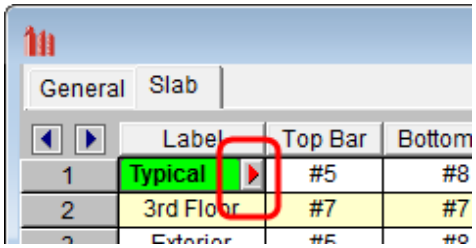


	Label	Design Method	Max Bending Chk	Top Cover[in]	Bottom...
1	N-S Rebar	Design Strips	1	.75	.75
2	E-W Rebar	Design Strips	1	1.5	1.5
3	Uniform	User Defined	1	.75	.75
4	Extra Top Rebar	User Defined	1	1.5	1.5

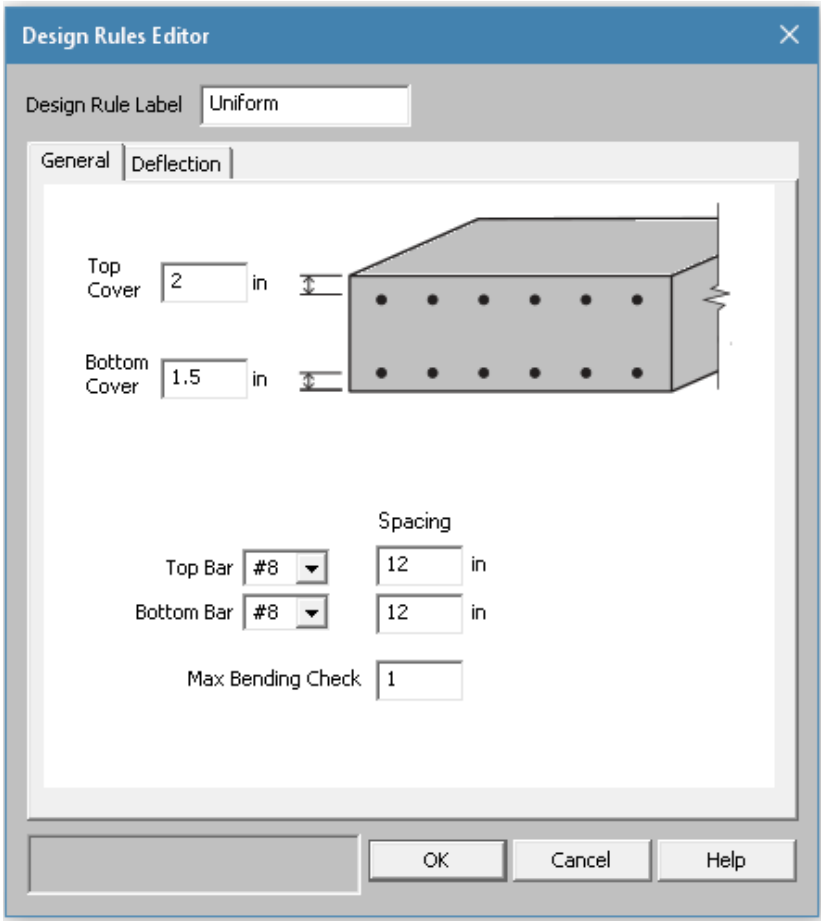
Design Method

There are two methods available; Design Strips or User Defined. Design Strips use the traditional methods that use Column/Middle Strips and the User Defined allows the user to define a uniform rebar. When you select User Defined option the program will ignore Slab Design Strips tab.

Pressing the red arrow in the **Label** column will open the **Design Rules Editor**, where the spreadsheet information can be edited with an intuitive dialog.

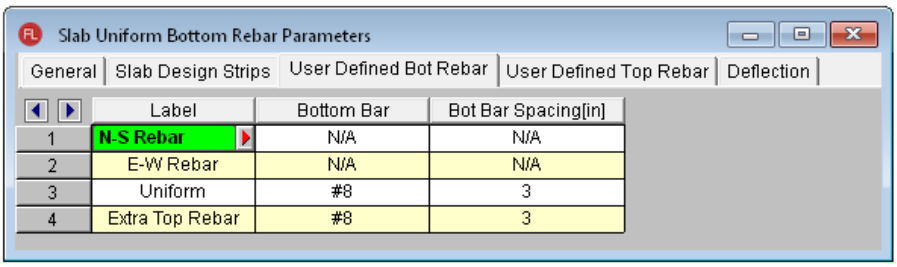


The **General** tab is used to enter all the cover spacing, bar size and spacing.



User Defined Bottom Rebar

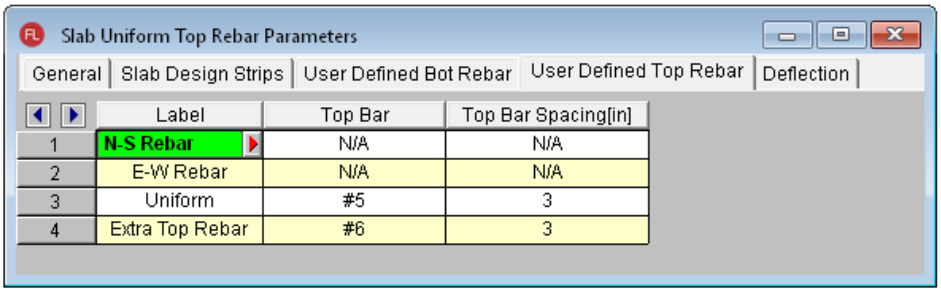
The **Bottom Bar** and **Bottom Bar Spacing** is continuous bottom rebar for both global directions.



	Label	Bottom Bar	Bot Bar Spacing[in]
1	N-S Rebar	N/A	N/A
2	E-W Rebar	N/A	N/A
3	Uniform	#8	3
4	Extra Top Rebar	#8	3

User Defined Top Rebar

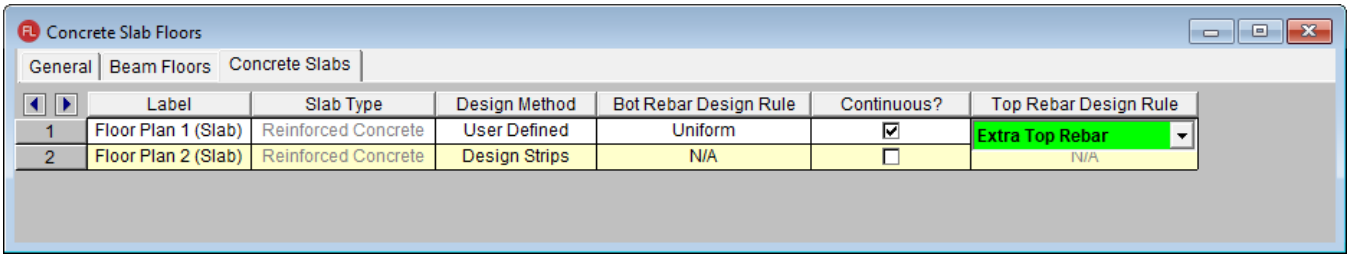
There are two options for the Top Rebar; Continuous over the entire slab or over the supports (columns and walls).



	Label	Top Bar	Top Bar Spacing[in]
1	N-S Rebar	N/A	N/A
2	E-W Rebar	N/A	N/A
3	Uniform	#5	3
4	Extra Top Rebar	#6	3

Continuous Top Rebar

In the Floors spreadsheet, you can select Continuous checkbox. This will apply continuous top rebar for both global directions based on the Top Rebar Design Rule.

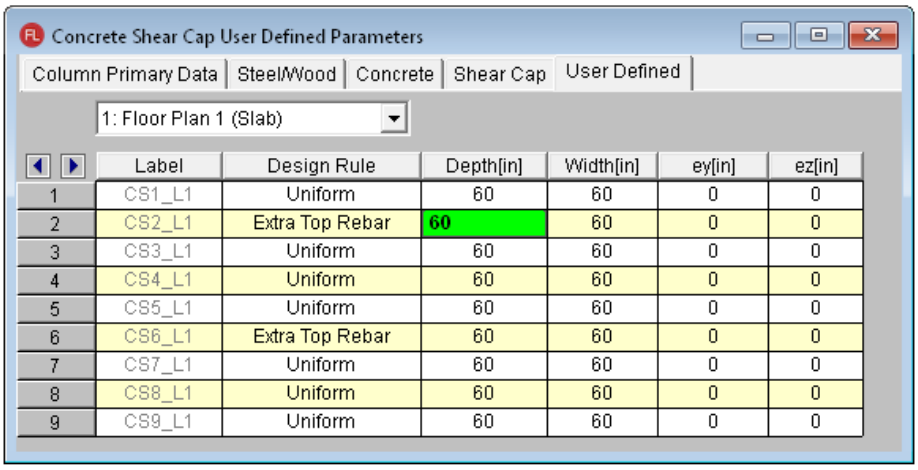


Top Rebar Over Supports

If the Continuous is not checked, the Top Rebar will only be placed over select regions over supports.

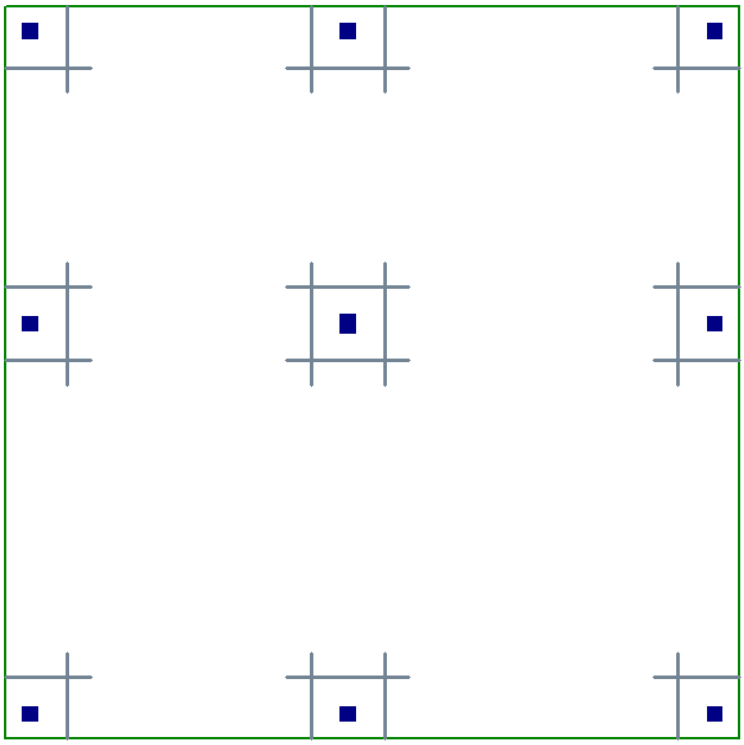
Columns

The Columns spreadsheet- User Defined tab allows you to customize the size of the rebar region over the column. The size will be determined by the Depth (X direction) and Width (Z direction). The top rebar region can also be moved eccentrically using ey and ez. The regions will automatically be trimmed at the edges of the slab.



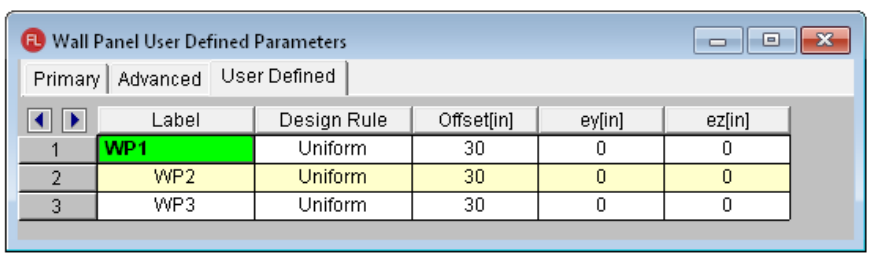
	Label	Design Rule	Depth[in]	Width[in]	ey[in]	ez[in]
1	CS1_L1	Uniform	60	60	0	0
2	CS2_L1	Extra Top Rebar	60	60	0	0
3	CS3_L1	Uniform	60	60	0	0
4	CS4_L1	Uniform	60	60	0	0
5	CS5_L1	Uniform	60	60	0	0
6	CS6_L1	Extra Top Rebar	60	60	0	0
7	CS7_L1	Uniform	60	60	0	0
8	CS8_L1	Uniform	60	60	0	0
9	CS9_L1	Uniform	60	60	0	0

For example: this slab shows the top rebar regions only over the columns.



Walls

The Top Rebar will only be placed over select regions over supports. The Walls spreadsheet- User Defined tab allows you to customize the size of the rebar region over the walls. The size will be determined by the Depth (X direction) and Width (Z direction). The top rebar region can also be moved eccentrically using ey and ez. The regions will automatically be trimmed at the edges of the slab.

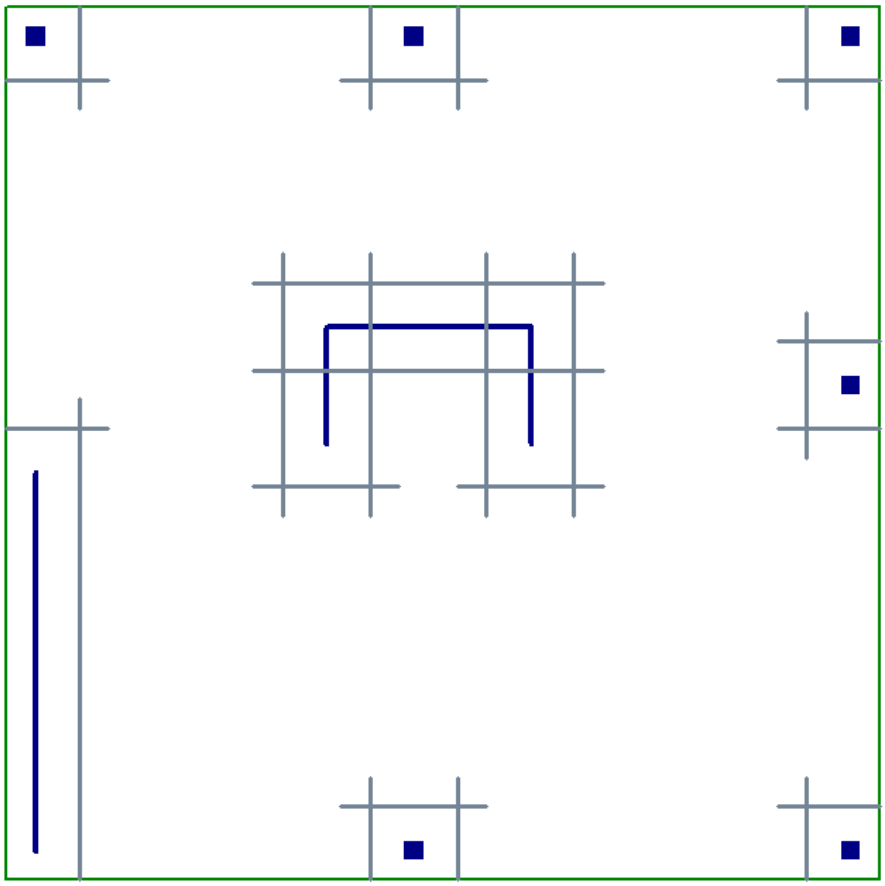


Wall Panel User Defined Parameters

Primary | Advanced | User Defined

	Label	Design Rule	Offset[in]	ey[in]	ez[in]
1	WP1	Uniform	30	0	0
2	WP2	Uniform	30	0	0
3	WP3	Uniform	30	0	0

For example: this slab shows the top rebar regions only over the columns and walls.

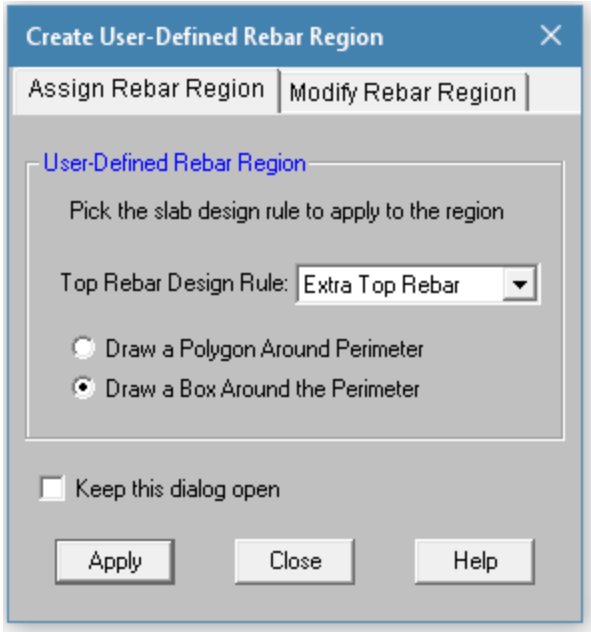


User-Defined Rebar Regions

If there is an area in the model that needs additional or different rebar than the default, you can define a special region that will replace the default top rebar. The User-Defined Rebar Region will replace all Top Rebar defined in that area with this new Top Rebar Design Rule. If it's on top of a column/wall, this Region will replace the Top Rebar defined in the column/wall regions.

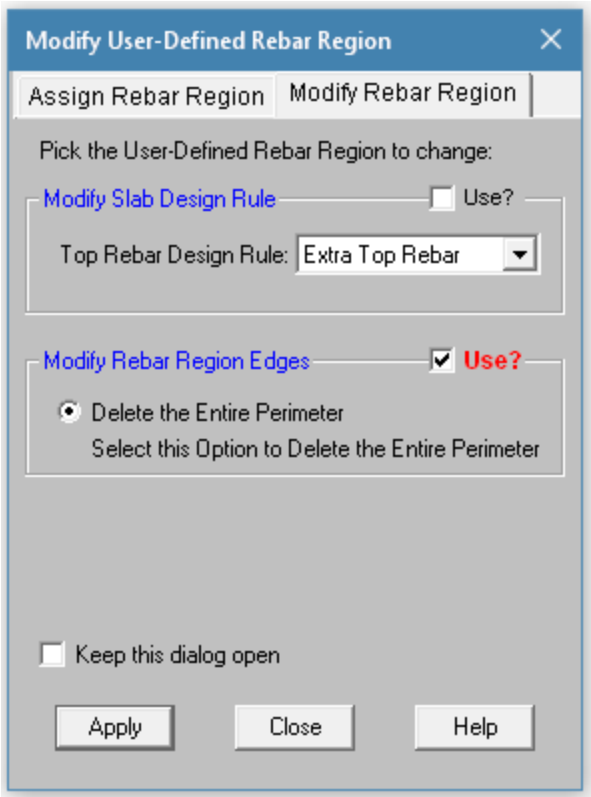
This can be accessed from the Graphical Editing Toolbar:

The Top Rebar Design Rule can be selected from a drop-down list and then defined by drawing a polygon or box.



Modify Rebar Region

This can be used to change the User-Defined Rebar Region or Delete it.



Note: The User-Defined rebar regions will be displayed graphically as Top Rebar regions. They will replace the existing Top Rebar region only if it's fully encompassed.

User Defined Deflection

Design Strip Deflection Paramaters

General | Slab Design Strips | User Defined Bot Rebar | User Defined Top Rebar | Deflection

	Label	DL Def...	DL Ratio	LL Defl...	LL Ratio	DL+LL...	DL+LL...	Categ...	Defl[in]	Ratio	Categ...	Defl[in]	Ratio
1	N-S Rebar		240		360		240	None		360	None		360
2	E-W Rebar		240		360		240	None		360	None		360
3	Uniform		240		360		240	None		360	None		360
4	Extra Top Rebar		240		360		360	None		360	None		360

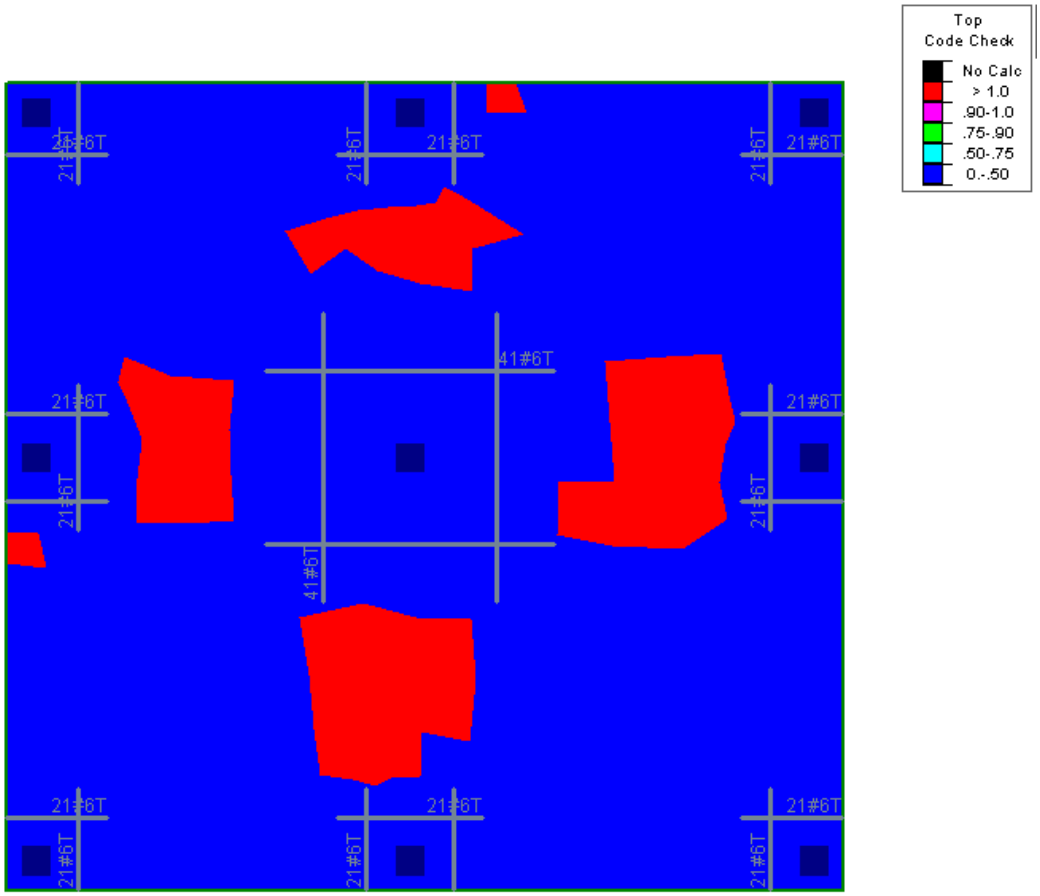
Although the Support Lines are not needed for design of the reinforcement using the User Defined methods, they are required for the Deflection to be calculated for the spans and cuts. For further information see the Deflection and Deflection Results Spreadsheet.

The program will use the Design Strip methodology to define the Design Strips. Therefore the Deflection criteria must be defined in the Design Rules defined as Design Strip. For example (above) N-S Rebar and E-W rebar will be used for Deflection criteria.

User Defined Results

The User Defined Results are graphical. This allows you to see the Unity Check directly on model to see where it passes and fails.

After solution, click the User Defined Contours to show the model with color coded Top and Bottom Unity Check.

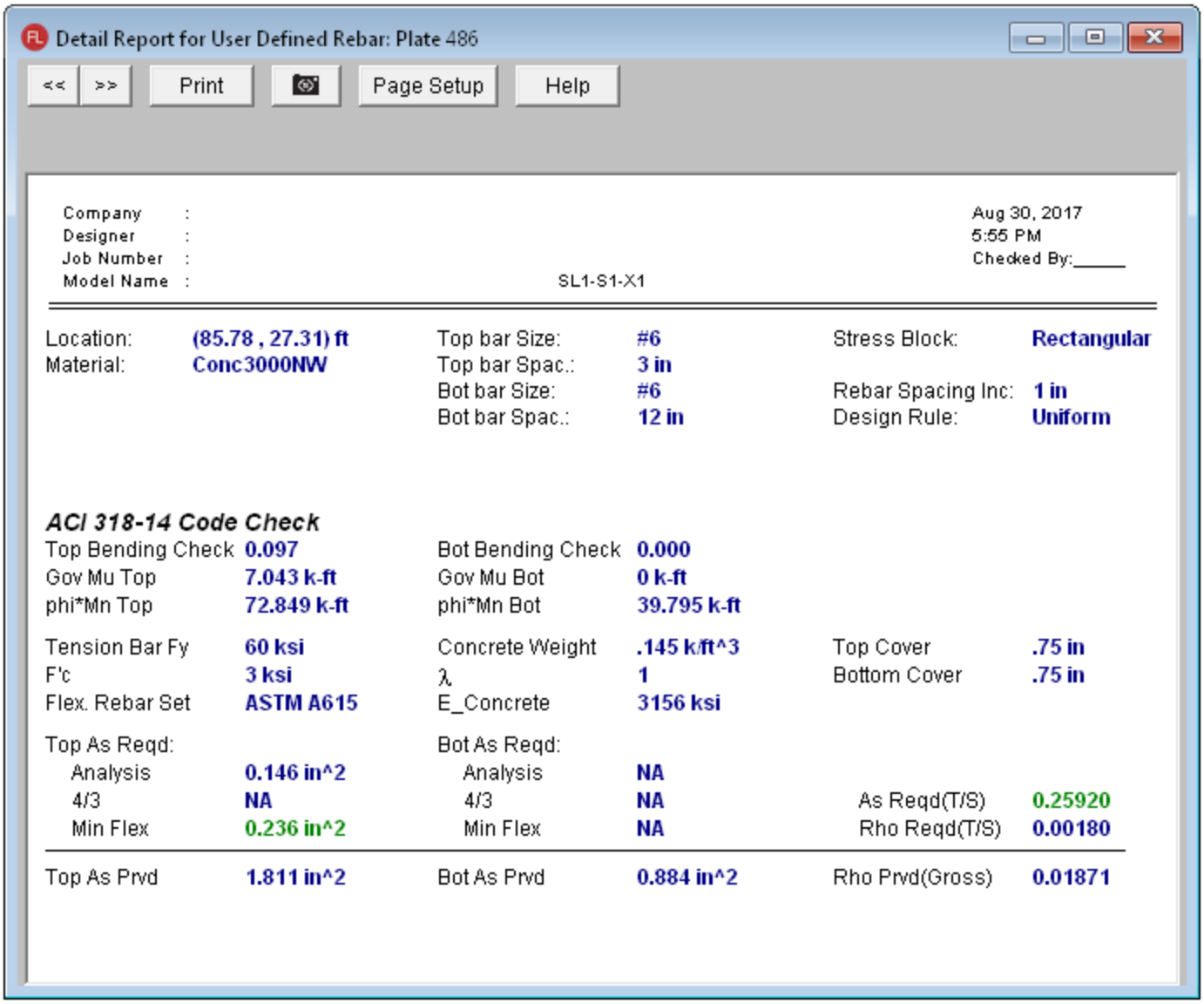


User Defined Detail Report

After solution, the Detail Report can be found by clicking on the Detail button

and then clicking anywhere on the slab.

The Detail Report for a User Defined floor is per plate. The Plate number is listed at the top including the Location of the plate. The forces are averaged per plate to provide a per foot check of reinforcement capacity vs demand. For further information, see the Rebar Design.



Note:

- Long Term Deflection is only available for User Defined floor levels.
- The Slab Strip Deflection results requires that there are Support lines drawn in the model.
- One-Way Shear checks require that the Support Lines are drawn in the model.

- A limited Detail Report for Design Strips and Cuts will provide deflection and one-way shear results. Access to the Detail Report can be found by opening the Slab Strip Deflection or Slab Results/Rebar spreadsheets.

Elevated Slabs - Modeling Tips

This section will provide helpful modeling tips and best practices for modeling slabs in RISAFloor ES.

Drawing Long Support Lines

If you have a support line that is very long, you can breakup the support line into two segments in order to get only half the support line in the Detail Report. The only limitation to breaking up a support line is that at the intersection of the two support lines, the top reinforcement will not be forced to be uniform spacing. Otherwise the design will be the same no matter how many spans the support line has.

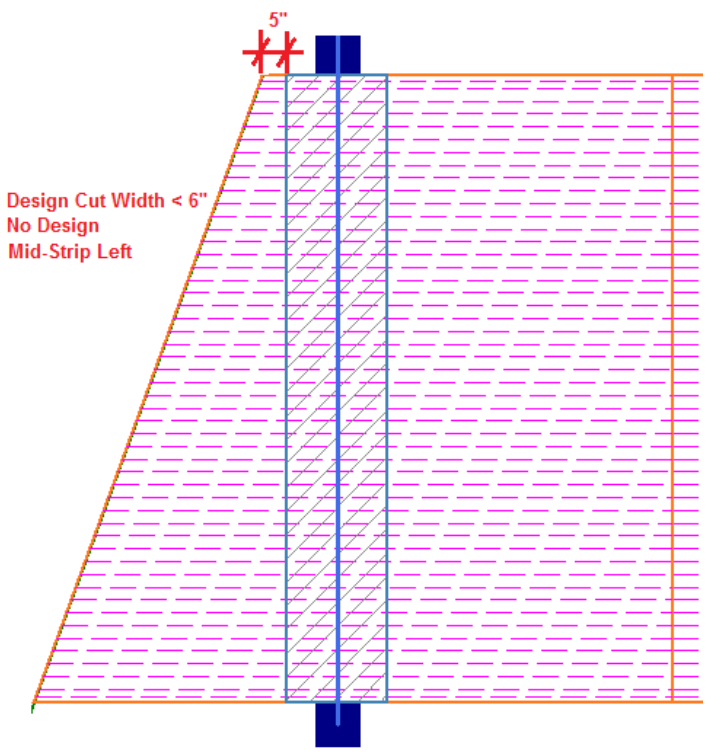
Modifying Mesh Size

The mesh size can contribute to slow run-time. If you increase the mesh size, usually the model will run faster. The mesh size defined in the Model Settings - Solution tab defines the maximum mesh size of finite elements created for the slab floor. There are minimum numbers of plates required between control points so even if you set the mesh size larger than the default the program will make sure there are enough plates. To can review the mesh size, go to the Model Display Options - Slabs tab and check the **Outline Plates** box.

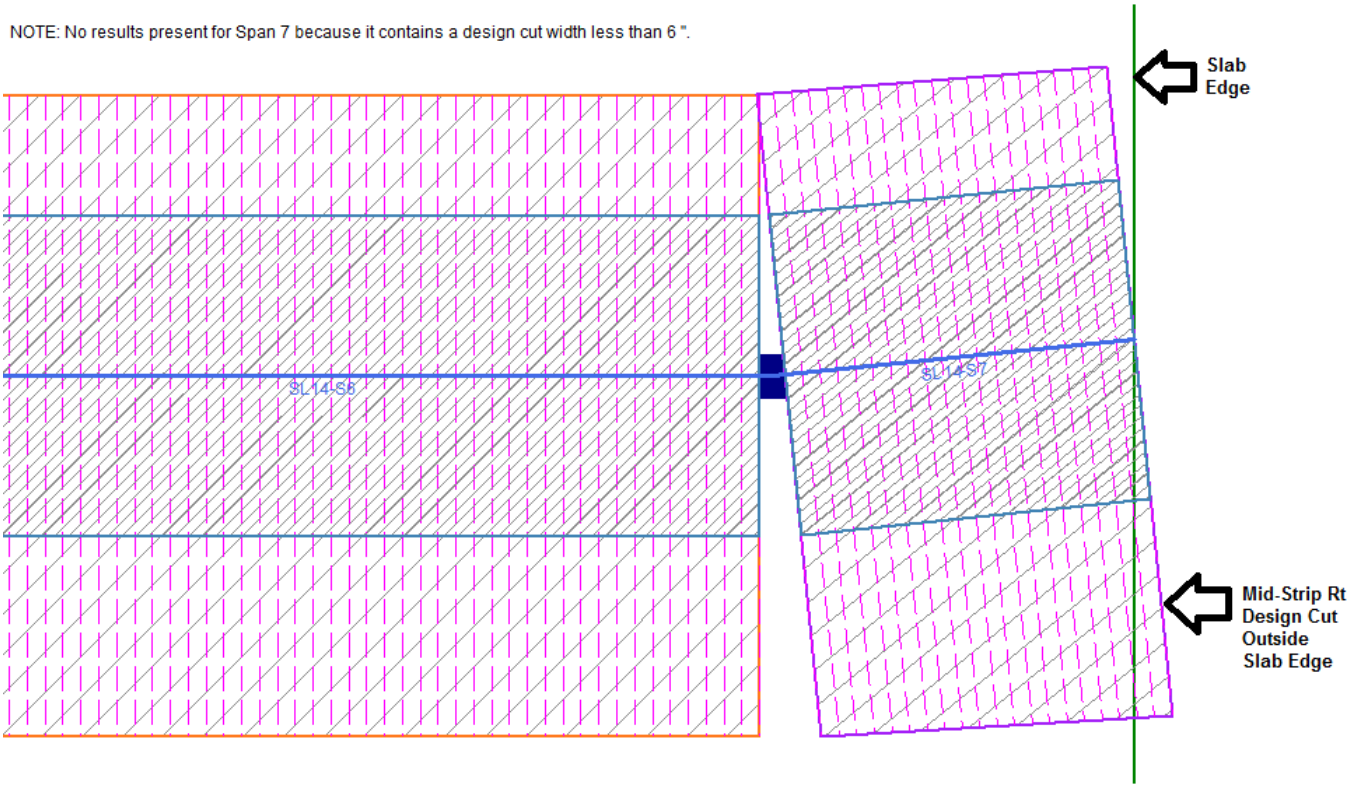
6" Minimum Design Cut Width

In order to get reinforcement design and analysis information, there must be a cut of at least 6" at the start and end of the Design Strip span. Below demonstrates the different ways that you might not get design for all or part of the strip.

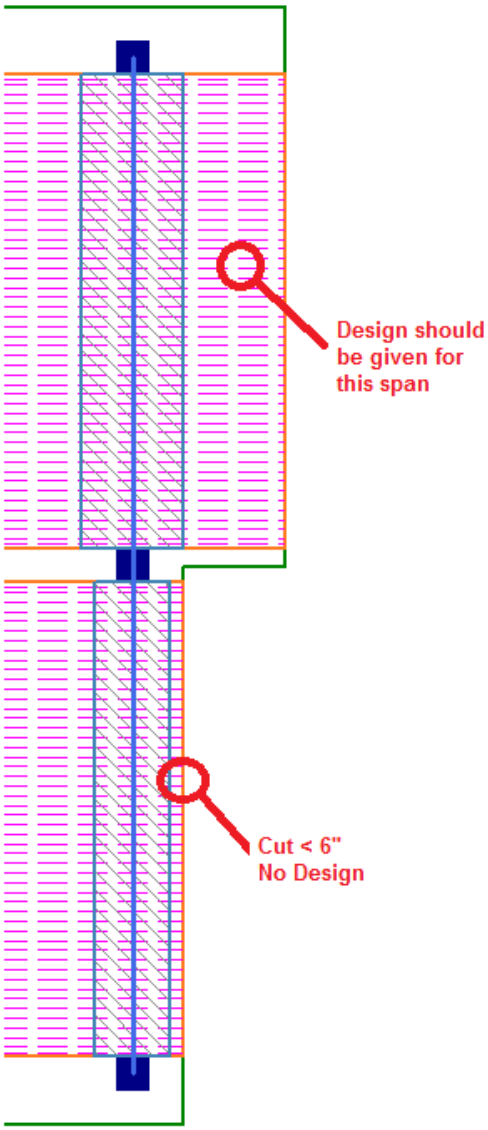
Example 1: The Mid-Strip Left has the first cut of 5" which is less than 5". The Column Strip and Mid-Strip Right will have design.



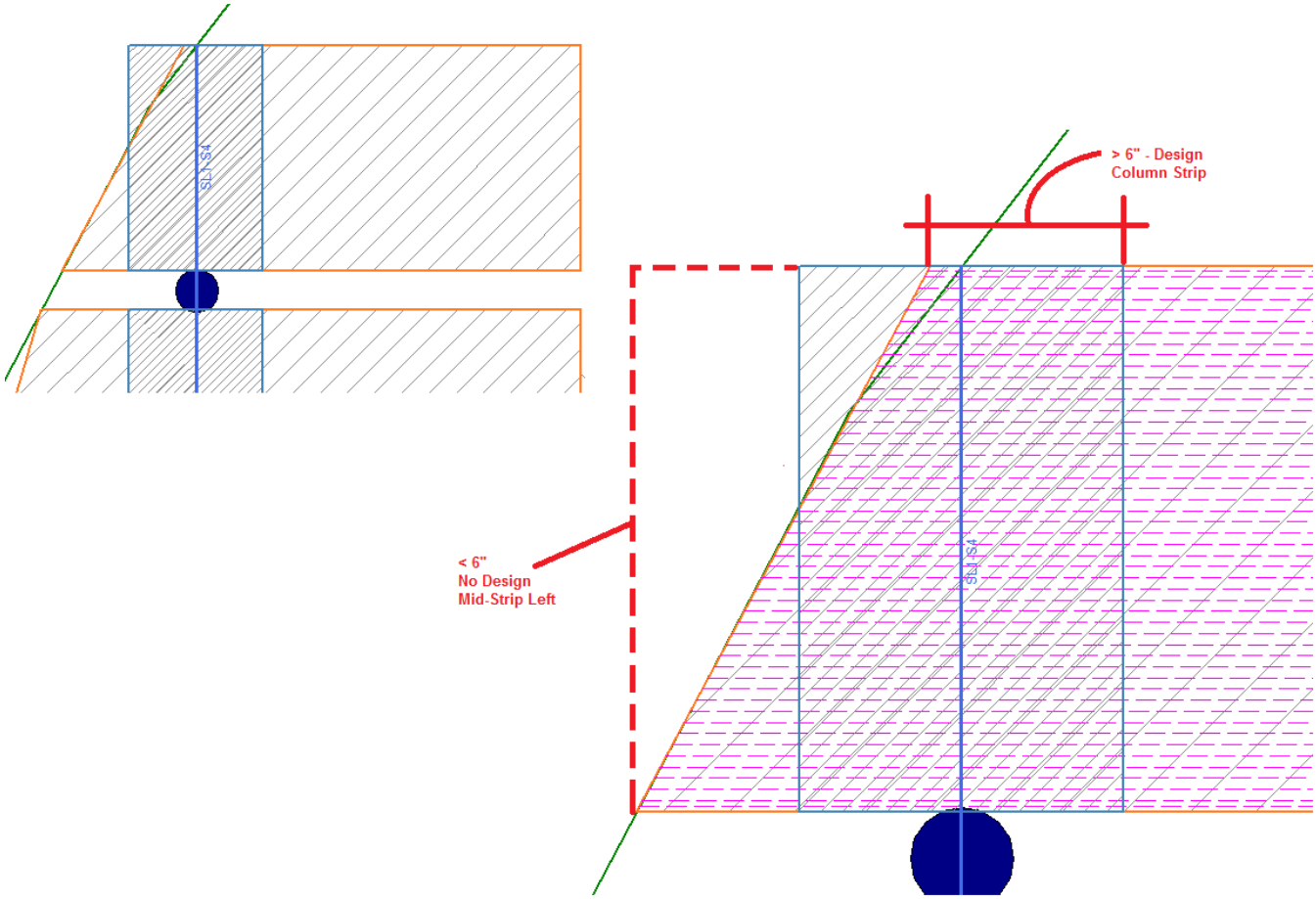
Example 2: The first Mid Strip Rt Design Cut is outside the Slab edge.



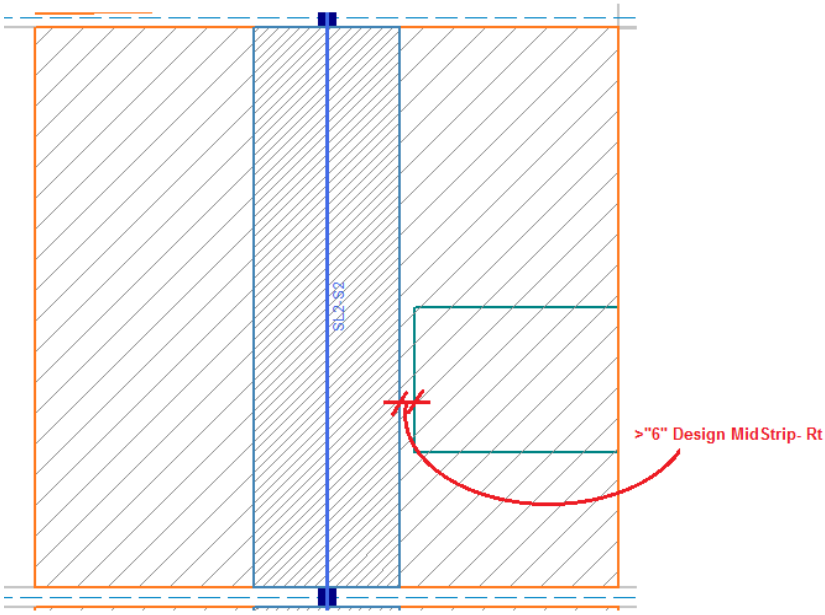
Example 3: Each span is determined individually within a support line.

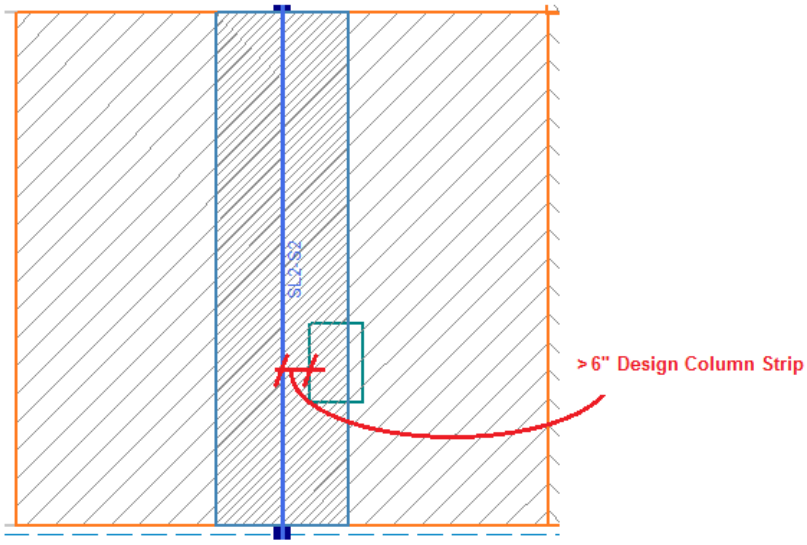


Example 4: The slab edge can interrupt the design strip and that can cause no design for a portion of the strip. In this example, the Column Strip still has at least 6" and the Support Line is not interrupted by the slab edge so it will get a design.

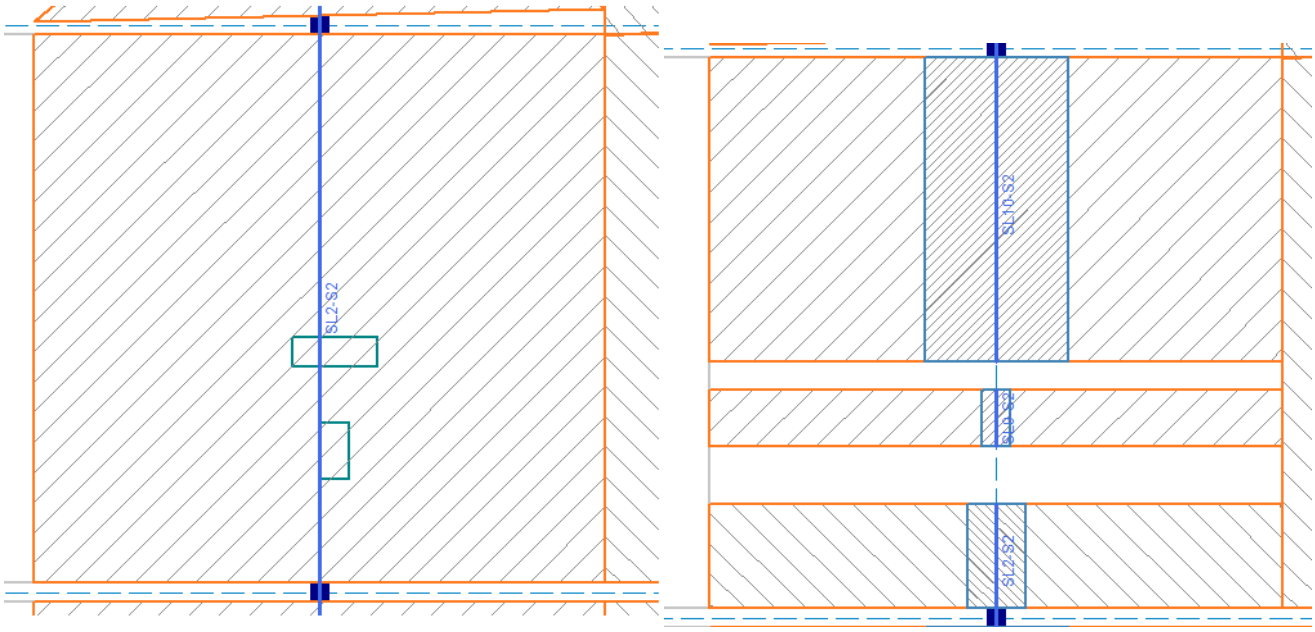


Example 5: Openings can cause the Design cuts to be less than 6".

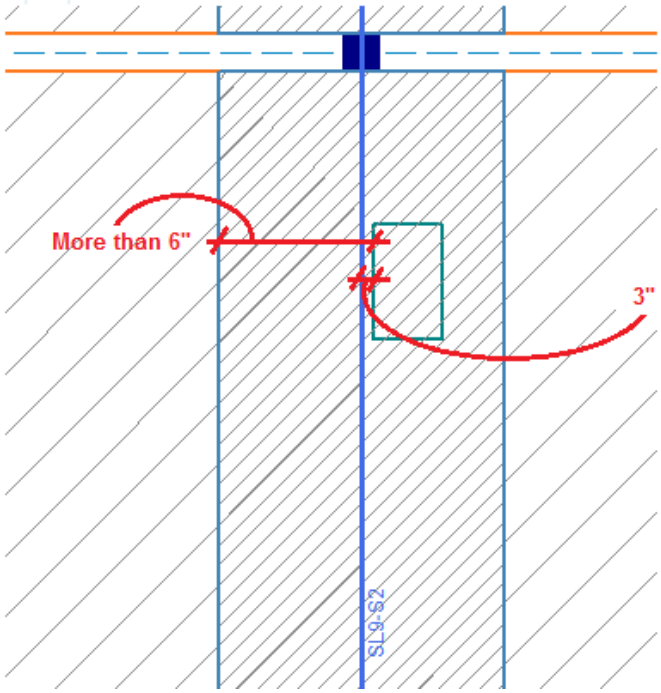




Example 6: Openings can cross over support lines. The openings will split up the support lines into separate segments around the openings.



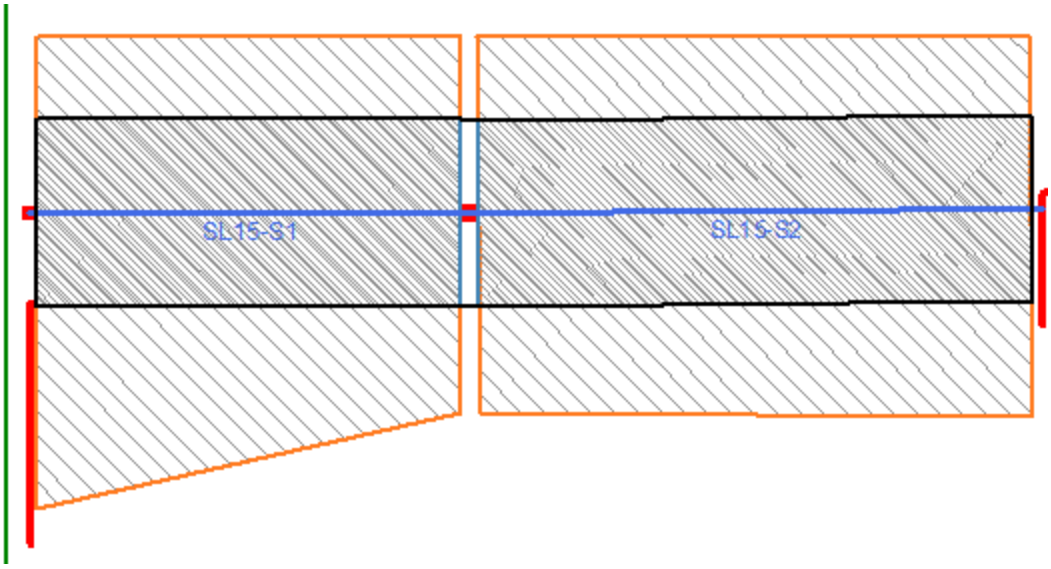
Example 7: This example shows an opening that doesn't cross the support line and it's fully contained within the column strip. It will not break up the support line as shown above unless it crosses the support line. There will be design for this column strip example as long as the cuts are at lease 6".



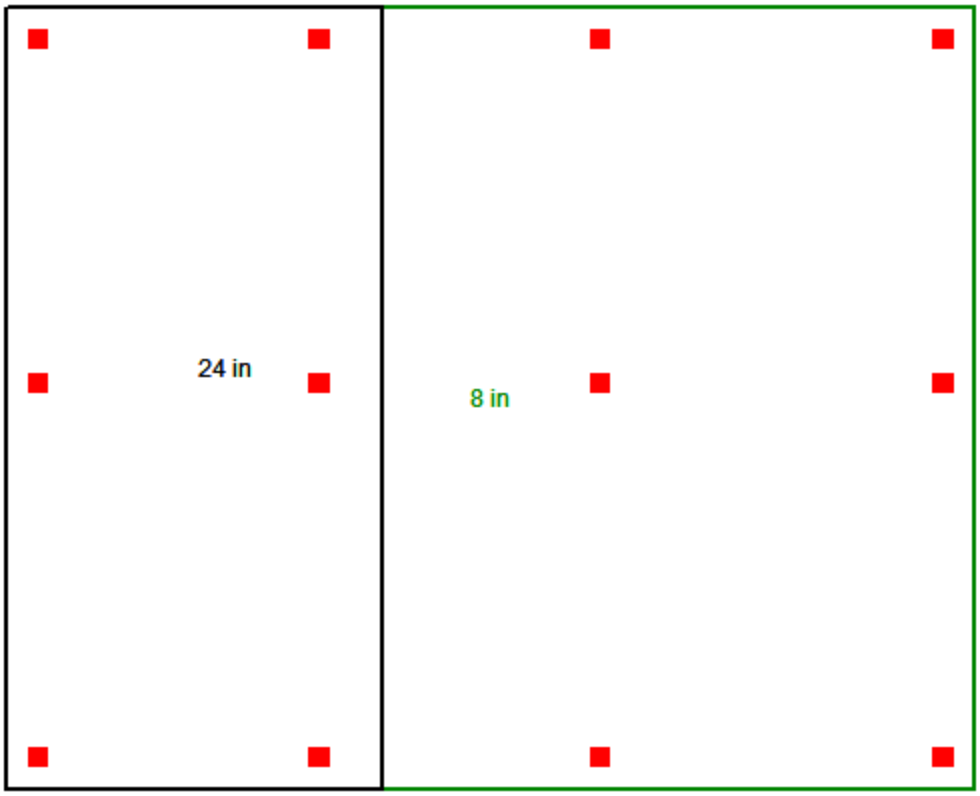
Varying the Slab Thickness

There are three methods to vary the slab thickness; Drop Panels, Shear Caps and Thickened Slab Region.

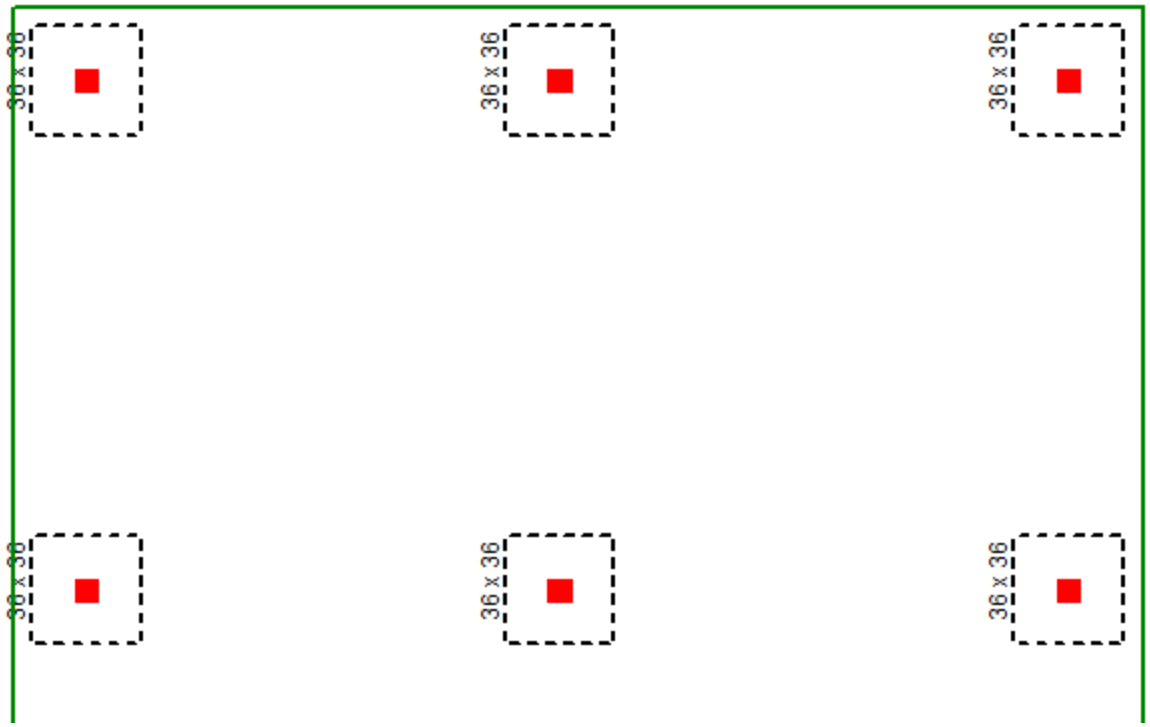
The **Drop Panel** will thicken the Column Strip only. This can be helpful when the forces are concentrated in the column strip and the added depth of the slab will reduce the reinforcement. The drop panel will be indicated in black outline around the Column Strip. For more information on adding a Drop Panel, see Drop Panels section.



Thickened Slab also allows you to change the thickness of the slab using a Polygon or Rectangular shape any location on the slab. This tool allows you to draw any free-formed shape so it can be used for entire section of the floorplan (example shown below) or just a smaller section. For more information on adding a Thickened Slab region, see Drop Panels section.



Shear Caps allow you to thickened the slab around a column. In practice this is refered to a shear cap, drop cap, or capital. Shear Caps are used when the punching shear calculations indicate that the slab thickness isn't adequate. The shear cap will thicken the slab in the area around the column top. For further information on Shear Caps, see the Punching Shear Considerations section.



Elevated Slabs - Analysis

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:

- 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.

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 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 the user. The mesh can be turned on using the setting on the Model Display Options - Slabs tab. Listed under the Wireframe selection, the **Outline Plates** check box will turn 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 will automatically generate point constraints at the following locations:

1. Support points which include: joints along a wall panel and column joints (see Column Meshing for full description).

2. Beams points which include: the start, end and intermediate joints along the beam.

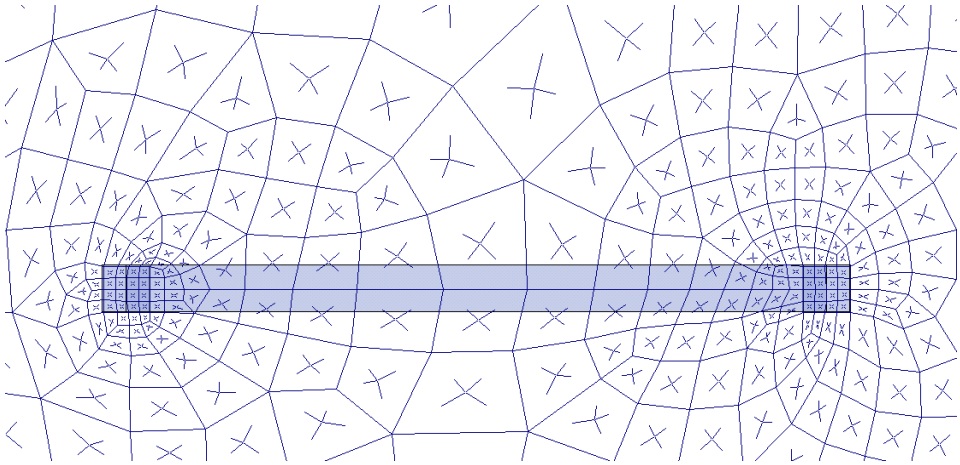
3. Column shear cap extents.

Note:

- Not all loads points generate point constraints. However, each point load is attributed to the nearest plate corners according to tributary distance.

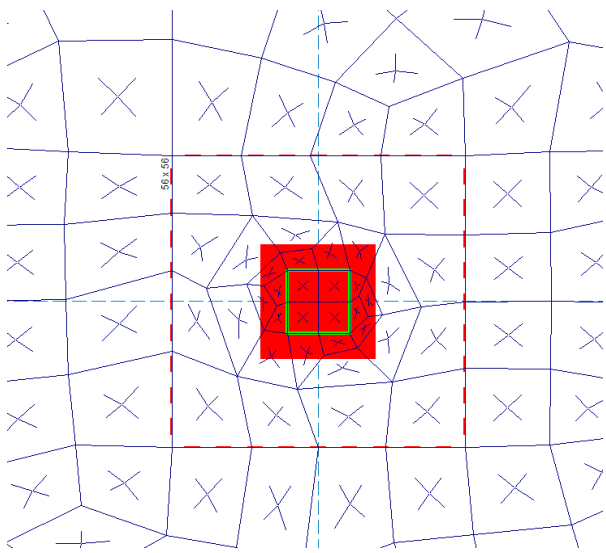
Beam Connectivity to Mesh

Beams drawn on a Two-Way slab are meshed directly to the surrounding plates. The picture below shows the center-line of the beam is meshed to the plates.

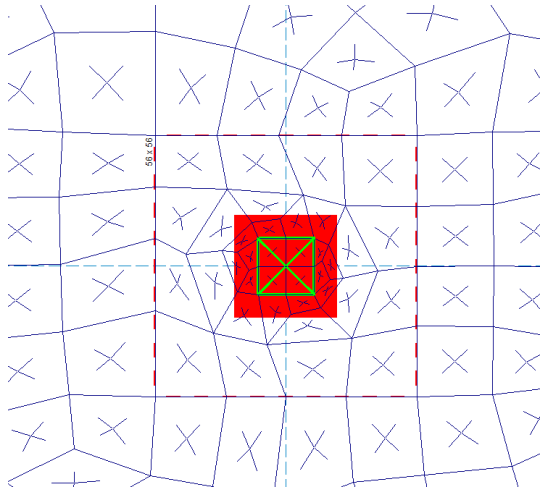


Column Meshing

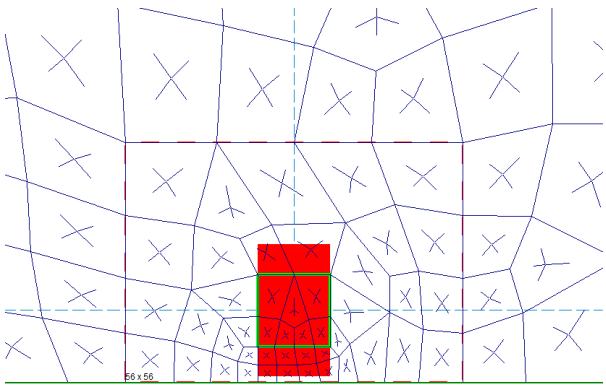
The top of the column will be submeshed and connected to the surrounding plates with rigid links in a 12" x 12" square. This approach is more accurate than modeling the column connection at only one node. When only one node is used the slab will have a higher peak moment and higher deflection at that 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.



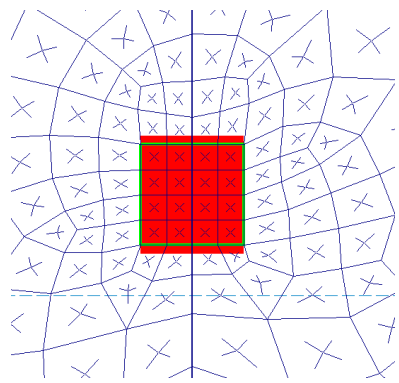
The column node will connect to the slab via the additional diagonal rigid links. After the slab is meshed, additional diagonal rigid links are drawn to connect the column node to the square rigid links (shown below). The column node has rotational and translational support so there will be vertical 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



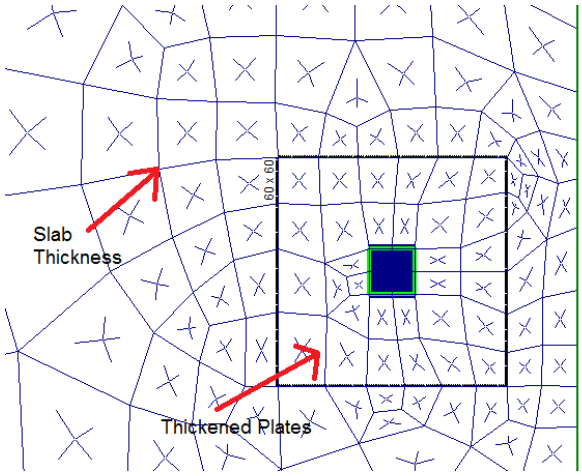
Beam Intersecting a Column

Note:

- The slab connection to the column rigid link top depends on the model merge tolerance being within the 12"x12" rigid link square. If the model's merge tolerance has been set too high in the Model Settings, the slab mesh may not connect to the columns as seen by the deflected shape and slab forces. If this occurs, reduce the merge tolerance back to the default 0.12" and the slab mesh will attach to the column.

Column Shear Caps

Shear caps can be added to columns. See the Shear Cap Considerations section to learn more about how to model Shear Caps. The Shear Cap will automatically thicken the plates in the area designated by the shear cap. For example, the Shear Cap shown below as 60"x60" will have thicker plates (the sum of the regular slab depth and the depth of the shear cap). The Shear Cap will alter the analytical force distribution by thickening the plates in this area.



General Slab Deflection Considerations

Aside from the Strength considerations mentioned above (Flexure and Shear), Elevated Slabs must be checked for Serviceability (i.e. Deflection). RISAFloor ES checks deflection on slabs in three ways:

Slab Minimum Thickness

ACI 318-14 Section 8.3.1.1 (ACI 318-11 Section 9.5.3.2) specifies a minimum permissible slab thickness as a function of clear span between supports. RISAFloor ES checks this value using Span Length and Support Dimensions and produces a warning if these requirements are not met.

Relative Deflection

ACI 318-14 Section 24.2.2 (ACI 318-11 Section 9.5.2.6) specifies a maximum permissible deflection as a function of span between supports. RISAFloor ES calculates a deflection for each Span based on FEA and checks this value using Span Length. A warning is produced if these deflection requirements are not met. The default Slab Design Rules conform to the requirements of ACI 318-14 Table 24.2.2 (ACI 318-11 Table 9.5b), however you may manually override these Design Rules to impose other deflection criteria. See the Elevated Slabs - Spreadsheet Results topic for more information.

Absolute Deflection

There is no ACI requirement regarding absolute deflection, however absolute deflections of the slab may be important for design considerations specific to your building. RISAFloor ES calculates the absolute deflection at many locations in the slab and reports those in the Slab Point Deflections spreadsheet. Because there is no way to specify deflection limits for absolute deflection, no checks or warnings are produced. See the Elevated Slabs - Spreadsheet Results topic for more information.

Slab Point Deflections

The **Slab Point Deflections** are calculated using finite element analysis. The program runs deflection analysis for all of the Service Load Combinations (those with the Service Checkbox turned on). Deflections are reported for all Points within a slab. Points are typically located at the corners of the plate elements which comprise the slab mesh.

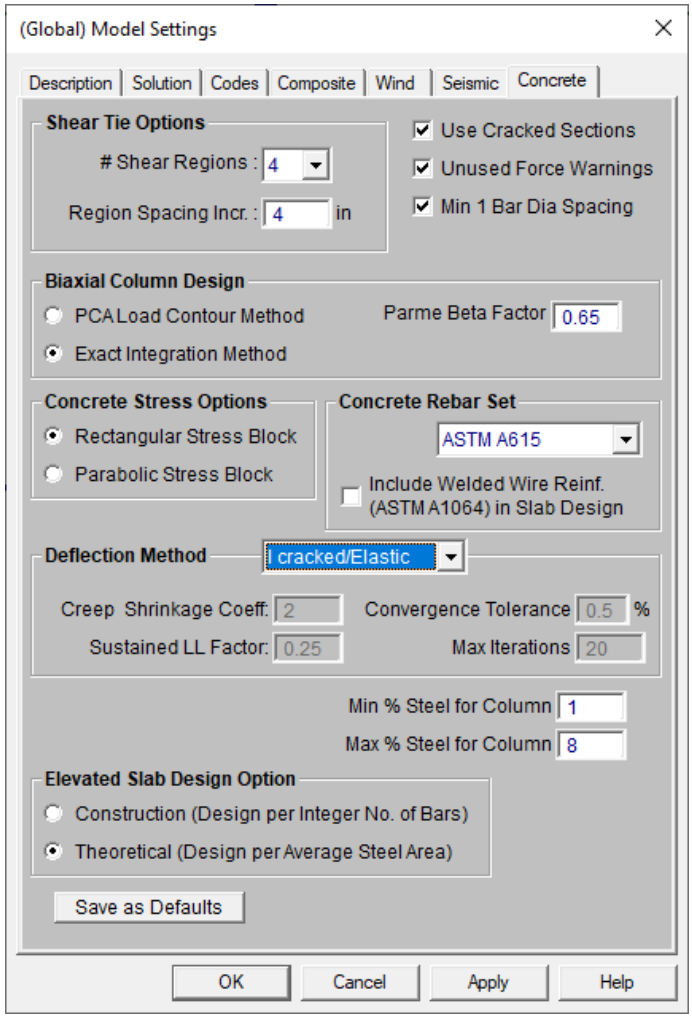
To view the point deflections results, see the Slab Point Deflections spreadsheet.

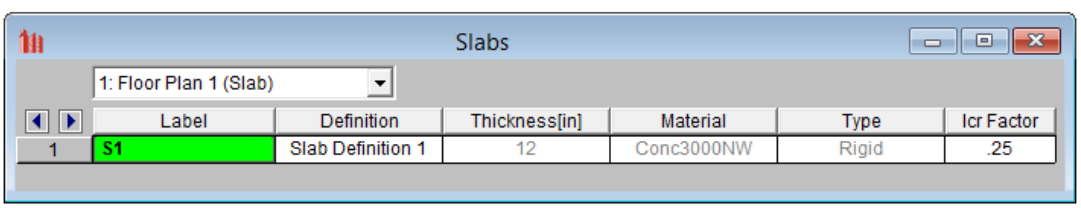
Note:

- The Slab Point Deflections are based on Load Combinations, not Load Categories.

Cracked Slabs

Selecting a Deflection Method of **I Cracked/Elastic** in the Model Settings controls the stiffness reduction based on the Icr factor in the **Slabs** spreadsheet .

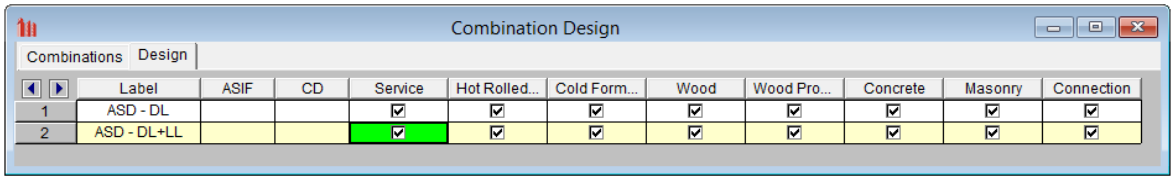




The Icr factor default is set to 0.25 per ACI 318-14 Section 6.6.3.1.1 (ACI 318-11 Section 10.10.4.1). You can modify this factor with any value between zero and 1.0 by typing in this field. The Icr factor adjusts the stiffness of the plate elements comprising the slab by altering the thickness during 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.



	Label	ASIF	CD	Service	Hot Rolled...	Cold Form...	Wood	Wood Pro...	Concrete	Masonry	Connection
1	ASD - DL			☑	☑	☑	☑	☑	☑	☑	☑
2	ASD - DL+LL			☑	☑	☑	☑	☑	☑	☑	☑

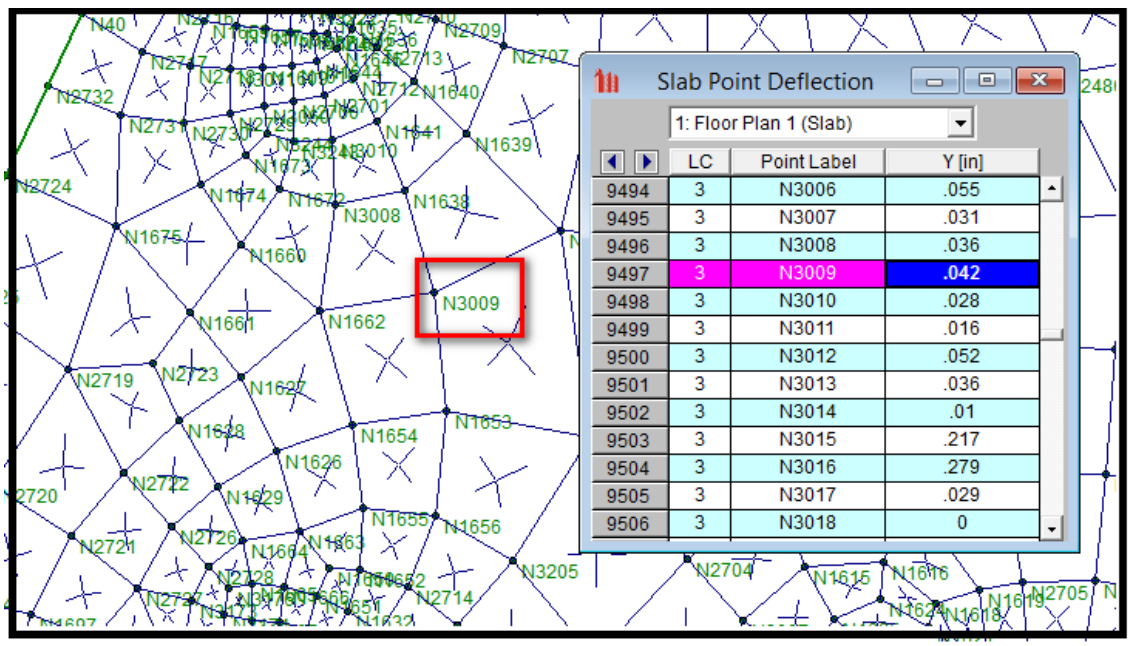
The commentary in ACI 318-14 Section 6.6.3.2.2 (ACI 318-11 Section 10.10.4.1) recommends a 1.43 factor to be applied to the cracked moment of inertia for all service loads. The program applies this factor to all Load Combinations that have the **Service** checkbox turned on, as long as the **Use Cracked Slabs** is selected in the **Model Settings**. This results in an effective default slab moment of inertia of $(0.25)*(1.43)*I_g = 0.3575*I_g$

Note: The Long Term Deflection will also give the cracking analysis for the Service Load Combinations in the graphical display, spreadsheet results and Detail Report.

Point Deflections vs Strip Deflections

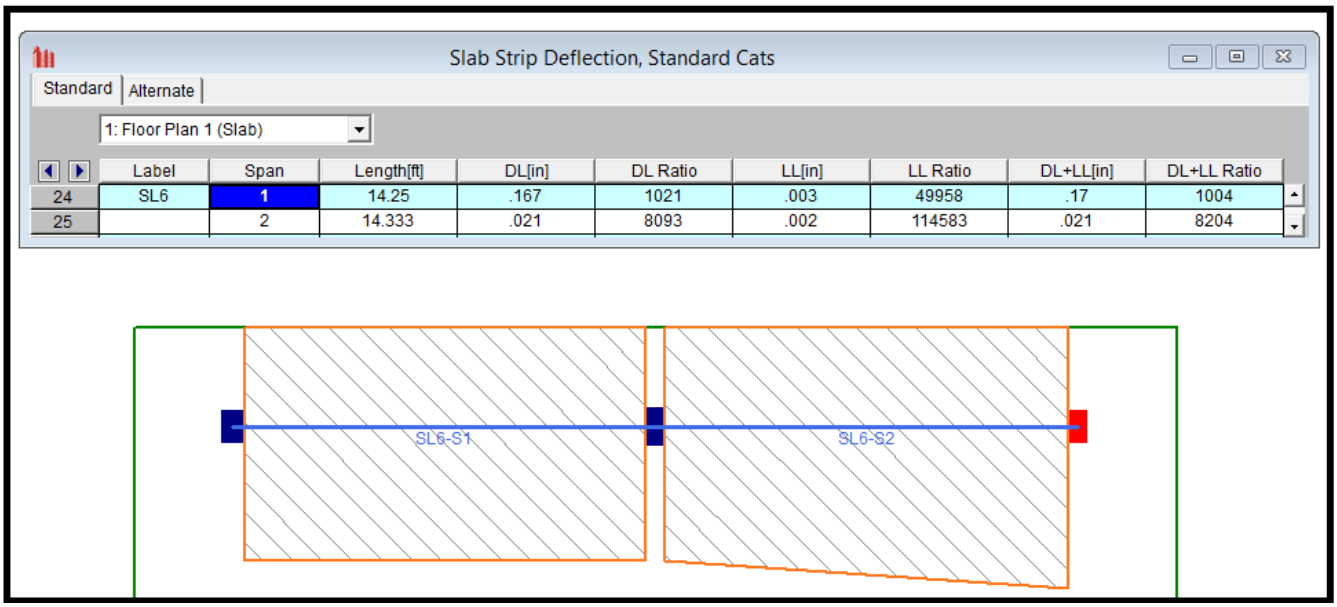
The **Slab Point Deflections** are the absolute deflections of every point inside the plate mesh based on the Load Combination. The **Design Strip Deflections** are the deflections within the Design Strip based on each Load Category. These are both very useful in different ways. The Point Deflections can help you identify exact low points in a slab. The **Design Strip Deflection** is closer to building code requirements because it helps you check the deflection ratio within a span.

Slab Point Deflection example:



	LC	Point Label	Y [in]
9494	3	N3006	.055
9495	3	N3007	.031
9496	3	N3008	.036
9497	3	N3009	.042
9498	3	N3010	.028
9499	3	N3011	.016
9500	3	N3012	.052
9501	3	N3013	.036
9502	3	N3014	.01
9503	3	N3015	.217
9504	3	N3016	.279
9505	3	N3017	.029
9506	3	N3018	0

Slab Strip Deflection Example:

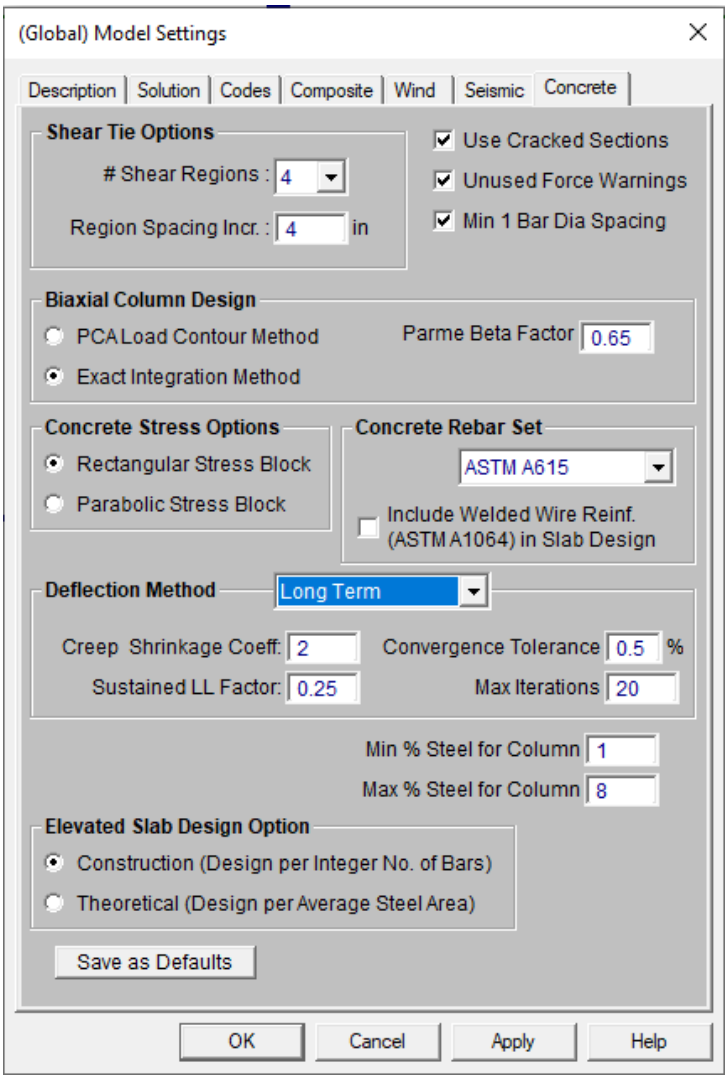


	Label	Span	Length[ft]	DL[in]	DL Ratio	LL[in]	LL Ratio	DL+LL[in]	DL+LL Ratio
24	SL6	1	14.25	.167	1021	.003	49958	.17	1004
25		2	14.333	.021	8093	.002	114583	.021	8204

Long Term Deflection

Long Term Deflection is available for Concrete Floor Slabs using Design Method: User Defined.

With Long Term Deflection turned on, the program will automatically adjust the plate stiffness by adjusting the Moment of Inertia to be I_e (ACI-14 24.2.3.5). This stiffness adjustment is an iterative analysis per plate based on the global axis.



Turn on the Long Term Deflection analysis in the Model Settings- Concrete tab.

Creep Shrinkage Coeff: Multiplier used for additional deflection due to long-term effects. This coefficient is described as λ_{Δ} for further information refer to ACI-14 24.2.4.

Sustained LL Factor: There are internal Load Combinations included when the Long Term Deflection analysis is turned on. The Sustained LL Factor gets used in the Load Combination listed below.

> **Sustained Load**: DL + (Sustained LL Factor)*LL
>
> **Long Term Sustained**: Sustained Load * (1 + Creep Shrinkage Coeff)
>
> **Total Load**: DL + LL
>
> **Long Term**: Long Term Sustained + (Total Load - Sustained Load)

These internal Load Combinations can be shown in the Slab-Contours- Displacements as well as the Slab Point Deflections.

Convergence Tolerance: The long term deflection is an iterative analysis to find the Effective Moment of Inertia, Ie. This tolerance is used to set how close a subsequent solution must be to the previous solution to be considered converged.

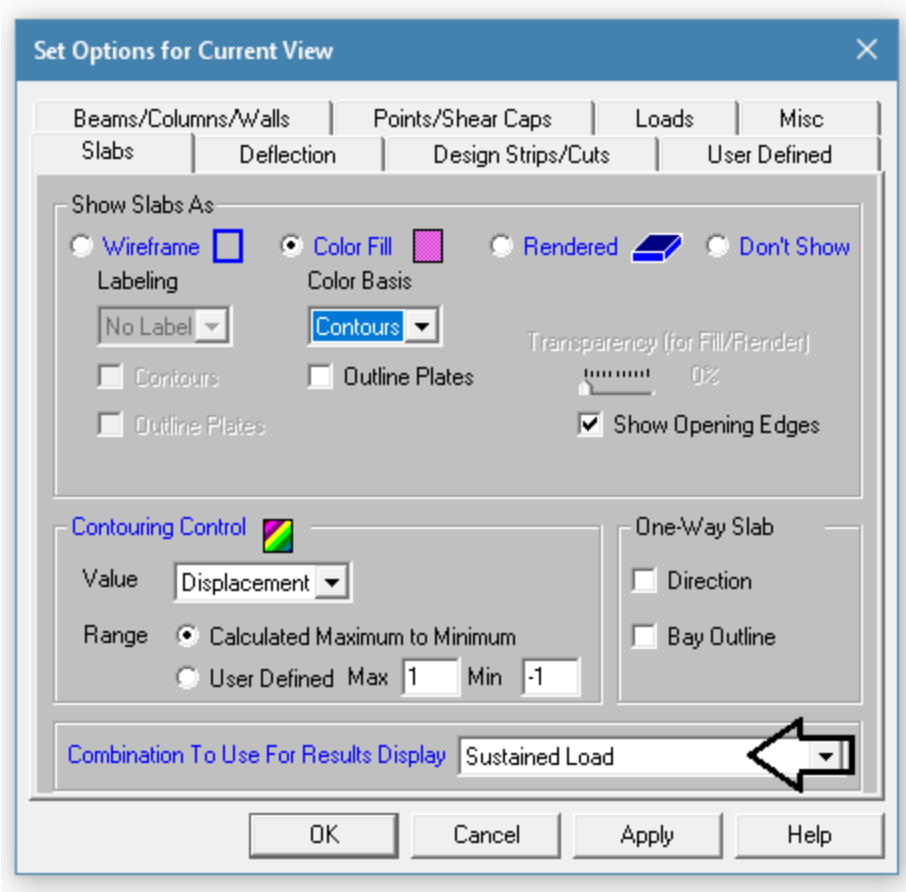
Max Iterations: This is the maximum number of analytical iterations the program will go before it stops.

Note: When the Long Term Deflection is turned on, the program will NOT use simplified methods for Cracked Slabs (Icr).

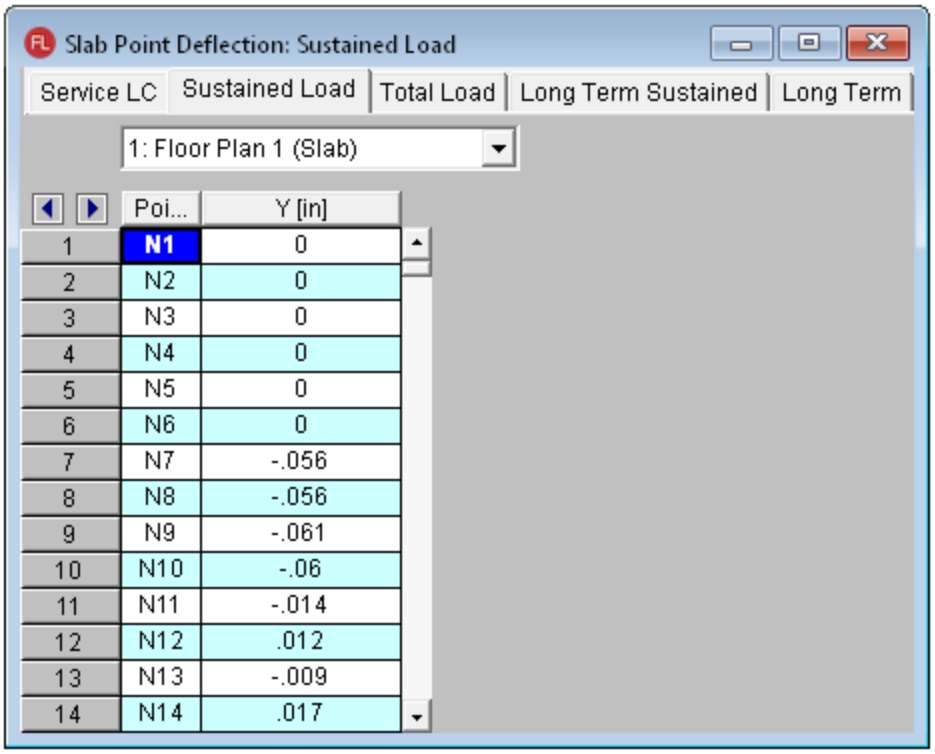
Long Term Deflection Results

The Long Term Deflection can be viewed graphically using the Slab Contours-Displacement. The internal Load Combinations are listed in the Plot Options.

There is also an icon that can be added to quickly display all the graphical views of displacement.



The Slab Point Deflection spreadsheet will show the node displacement per each internal load combination.



	Poi...	Y [in]
1	N1	0
2	N2	0
3	N3	0
4	N4	0
5	N5	0
6	N6	0
7	N7	-.056
8	N8	-.056
9	N9	-.061
10	N10	-.06
11	N11	-.014
12	N12	.012
13	N13	-.009
14	N14	.017

Notes and Limitations:

- Long Term Deflection does not apply for Design Method: Design Strips.
- The Global Axis is used to calculate stiffness adjustments. Therefore for the most accurate results the model to be situated mostly in the Horizontal Z and Vertical X directions respectively.
- The Support Line- Detail Reports and Strip Deflection spreadsheets will only show the Service Load Combinations results, not the Long Term Deflection.

Elevated Slabs - Design

Concrete slab reinforcement design and code checks can be performed based on the following codes:

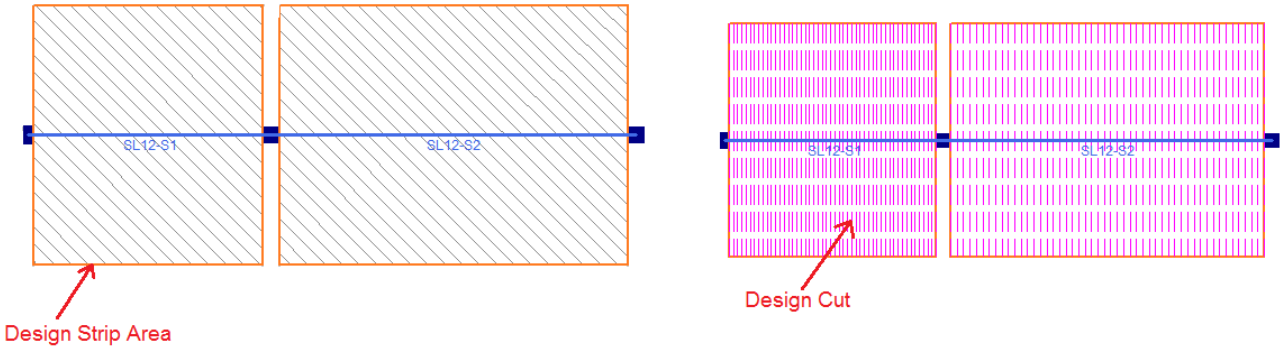
- ACI 318-19 (22)
- ACI 318-19
- ACI 318-14
- ACI 318-11
- ACI 318-08

Note:

- Unless otherwise specified, all code references for elevated slabs are to ACI 318-14/ACI 318-11. ACI 318-14 Section 8.2 (ACI 318-11 Section 13.5.1) provides the code basis.
- Refer to Design Strip Modeling for a full description of the Design Strip terms used in this section.
- See the Elevated Slabs - Analysis topic for more information on the analysis.
- Because of its limitations no part of the Direct Design Method (ACI 318-14 Section 8.10/ACI 318-11 Section 13.6) is used in the program.
- 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.

Concrete Reinforcement Design

The reinforcement design is calculated based on the Design Strips which are defined by the Support Lines. See Elevated Slabs - Modeling for a full description of how to model the Support Lines and Design strips. Within a Design Strip, there are Auto-generated Design Cuts that are "slices" thru the design strip perpendicular to the Support Line. The Design Cut finds all the forces across it's length and then finds adequate reinforcement for these forces. In the example below, there are 50 Design Cuts per span. The program will envelope all the reinforcement requirements per span and come up with a final design. The reinforcement spacing and size are defined in Slab Design Rules spreadsheet. The reinforcement design checks the analytical requirements, minimum ratios and detailing requirements.



The number of Auto Design Cuts are defined when you draw the Support Line and can be modified in the Slab Design Strip spreadsheet. The design cuts will start at the face of the columns with the exception set by ACI 318-14 Section 8.11.6.1 (ACI 318-11 13.7.7.1) specifies that the design should be not farther away from than 0.175*L1 (L1 is the span).

Two rebar design options are available, namely the Construction method and the Theoretical method, for the design of the slab rebar. The Construction method, which is the default method employed by the program for a significant period, determines the design of the slab based on an integer number of rebar. This method operates under the assumption that if the strip width divided by rebar spacing is not an integer, an additional rebar will be provided to guarantee the presence of a rebar at each end of the design strip. On the other hand, the Theoretical method is a new method added later that enables the program to calculate the rebar area based on the average value, where the

rebar area equals As (single rebar) multiplied by the Strip Width divided by Spacing. However, it is crucial to note that this method may not always result in an integer number of rebar so unlike the Construction method, the results for Theoretical method will have no info of rebar amount. You can select different rebar design options in Global Model Setting.

Slabs may be analyzed and designed using traditional reinforcement bars or welded deformed wire reinforcement (WWR). WWR can be enabled by going to the Global Model Settings and checking the "Include Welded Wire Reinf." option in the Concrete tab.

Note:

- Welded wire reinforcement is only available for Elevated Slabs.
- Only ASTM A1064 deformed bar sizes are available.
- Development length is conservatively calculated in the similar fashion as traditional reinforcement bars. Ψw factor is conservatively ignored which reduce the development length due to contribution of "welded intersections".

Reinforcement Design Routine

Because a design strip has multiple cuts we need to first consider each cut and then take the enveloped value for the cuts for the overall strip.

For each design cut:

1. Calculate As-Req'd to satisfy all provisions (Strength, T&S, Min_Flex, 4/3*Strength).
2. Calculate an integer number of bars (or equivalent rebar number if Theoretical method is used) that meets or exceeds As-Req'd from Step 1.
3. Calculate a number of spaces (equals Step 2 value minus 1).
4. Spacing = Divide cut width by Number of spaces (from Step 3). This will almost always be a decimal value.
5. Calculate Min_Spacing as the greater of ACI min spacing requirements (2*db, etc) and Min Spacing from Design Rules.
6. Calculate Max_Spacing as the lesser of ACI min spacing requirements (6*db, etc) and Max Spacing from Design Rules.
7. If Min_Spacing < Spacing < Max_Spacing.
    1. Round Spacing down to closest value divisible by Increment (Increment comes from Design Rules). Report this as your Spacing value. Do not adjust As
8. If Spacing > Max_Spacing.
    1. Calculate the lowest integer number of bars which satisfies Max spacing.
    2. Calculate a number of spaces (equals number from Step 8.1 minus 1).
    3. Spacing = Divide cut width by number of spaces from Step 8.2.
    4. Round Spacing down to closest value divisible by Increment (Increment comes from Design Rules). Report this as your Spacing value. Report As from Step 1.
9. If Spacing < Min_Spacing.
    1. Calculate the lowest integer number of bars which satisfies Min spacing.
    2. Calculate a number of spaces (equals number from Step 9.1 minus 1).
    3. Spacing = Divide cut width by number of spaces from Step 9.2.
    4. Report Decimal Spacing. The cut will fail anyway because Req'd spacing was below what's allowed, so who cares that we report a decimal value for a failing cut. Report As from Step 1.

Once this is complete we have a rebar spacing and As for each cut.

For each design strip then r eport the smallest value of Spacing from all of the Design Cuts.

Reinforcement Lengths - Top Rebar

The top reinforcement is shown with a heavy blue line and the length is listed below in parenthesis. The rebar is measured from the center-line of the column to the end of it's required length. The development length is shown

beyond the top rebar length and it's indicated by the red dashed line at the end and the length is shown in parenthesis below.

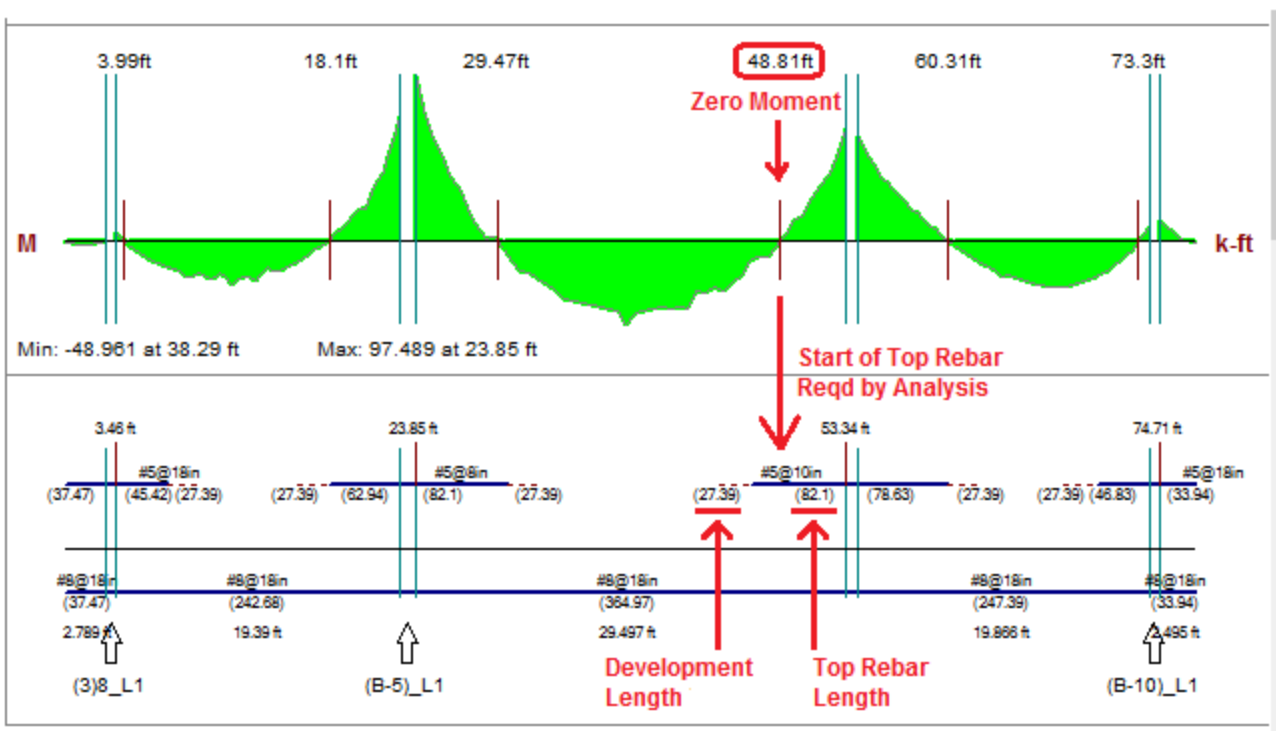
The Top Rebar length is determined by the following steps:

Step 1: Find the required length by analysis determined by the point of zero moment. This location indicates the no demand for tensile reinforcement to resist flexure. The zero moment location is indicated on the moment diagram as a vertical red line and the location of that line is listed above. This dimension is an overall dimension from the start of the support line to that point.
Step 2: The development length is calculated using ACI 318-14 Section 25.4.2 (ACI 318-11 Section 12.2.2, CSA A23.3-14 Section 12.2.2). Check the development length is longer than the minimum required by ACI 318-14 Section 7.7.3.3 (ACI 318-11 Section 12.10.3, CSA A23.3-14 Section 12.10.4), if not use this value as the development length.
Step 3: Add the required by analysis length (Step 1) and development length (Step 2) and check that it meets the minimum extensions listed in ACI 318-14 Fig 8.7.4.1.3(a) (ACI 318-11 Fig 13.3.8, CSA A23.3-14 Figure 13.1). If the total minimum length is not met then the additional length will get added to the length required by analysis.

Note:

- ACI 318-14 Fig 8.7.4.1.3(a) (ACI 318-11 Fig 13.3.8) uses the "Without Drop Panels" section. The more conservative value is selected from this table as the program currently has no way to indicate different lengths for some bars. Therefore 0.3ln is used for Column Strip and 0.22ln for Middle Strip Top bars.

The example shown below demonstrates the case where the minimum was not met and the top rebar length indicated with a heavy blue line is longer than required by analysis.

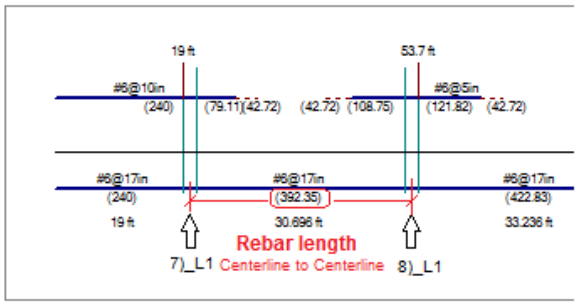


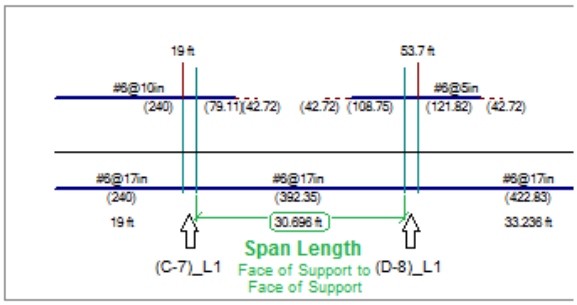
Reinforcement Lengths - Bottom Rebar

The bottom bars are considered continuous along the entire Design Strip throughout the entire support line. The ACI allows for a certain amount of reinforcement to be discontinuous however at this time RISA is not indicating any other length than fully continuous.

Note:

- There are two dimensions shown on the bottom of the diagram, one indicates the rebar length and the other is the design span length.





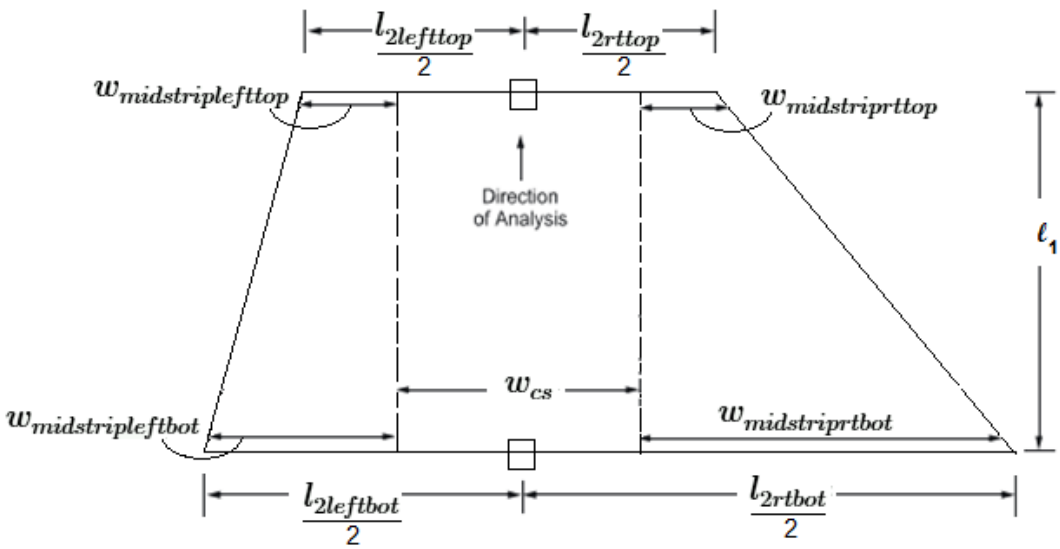
Column Strip Width Calculation

The image below describes how the program calculates the Column Strip width (w_{cs}). The variables l_1 and l_2 are also described above. This drawing shows how the Design Strip can be tapered and the equation accommodates the taper and non-tapered Design Strip. The width of the Column Strip is not allowed to vary along the span. The width of the Mid-Strip Right and Mid-Strip Left may vary.

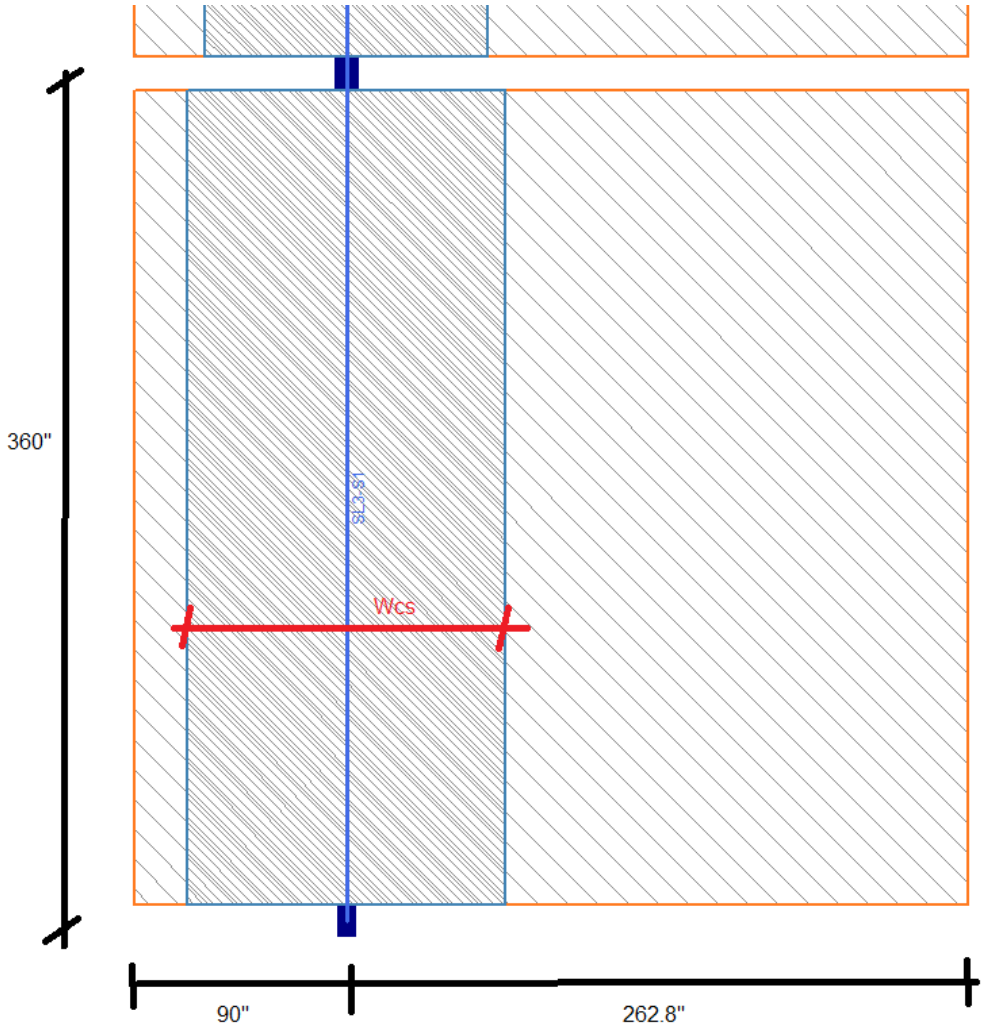
Note:

- The Column Strips in RISAFloor ES are centered on the Support Line except when the Design Strip is very near a slab edge or Design Strip edge. In those two cases, the column strip will be justified with the edge. The actual width of the Column Strip is listed in the Design Cut Detail Report.
- The Column Strip width is calculated for each span individually. A support line may have varying Column Strip widths for each span.

$$w_{cs} := \underbrace{min\left(\frac{l_{2lefttop}}{2} \times \frac{1}{2}, \frac{l_{2leftbot}}{2} \times \frac{1}{2}, \frac{l_1}{4}\right)}_{\text{Left Side}} + \underbrace{min\left(\frac{l_{2rttop}}{2} \times \frac{1}{2}, \frac{l_{2rtbot}}{2} \times \frac{1}{2}, \frac{l_1}{4}\right)}_{\text{Right Side}}$$



Example calculation: This shows how the Column Strip width is calculated for a non-tapered Design Strip.



Wcs= Min (90"/2, 360/4) + Min(262.8/2,360/4)

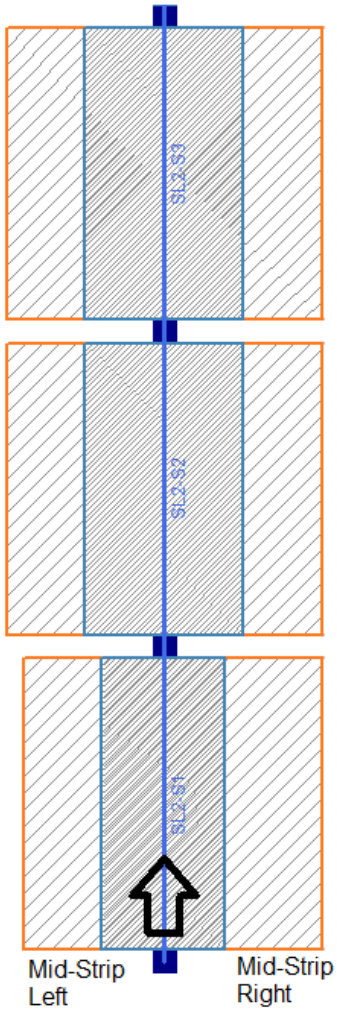
= Min (45", 90") + Min (131.4", 90")

=135"

Calculation of Mid-Strip Width

The Mid-Strip is calculated by centering the Column Strip over the middle of the Support Line and the remainder is the Mid-Strip on each side. The actual width of the Mid-Strip for each side is listed in the Design Cut Detail Report.

The Mid-Strip Left and Mid-Strip Right are defined by the way the Support Line is drawn. Below is an example that shows the Support Line is drawn from the bottom to the top and the Support Line is numbered accordingly (SL1 at the bottom, SL3 at the top). From the vantage point of the start of the Support Line, the Mid-Strip Right falls to the right of the Support Line and the Mid-Strip Left falls to the left of the Support Line.



Slab Shear Design

Concrete slabs are susceptible to two different types of shear failure:

One Way Shear

One Way Shear can cause a localized shear failure along a plane perpendicular to the direction of the shear. This type of shear is checked across the total design strip width (column strip plus middle strip) as part of the Design Cut. The total amount of shear for each Design Cut is determined through finite element analysis, and is compared against the shear capacity of that cut. Because it is not possible to provide stirrup shear reinforcing for concrete slabs, the plain concrete shear strength is used. RISAFloor ES automatically calculates the ratio of shear force to shear capacity for each Design Cut, and produces a warning if that value exceeds 1.0 anywhere in the model.

When ACI 318-19 is selected, the shear strength of concrete (Vc) uses equations in Table 22.5.5.1. Note that slabs do not have shear reinforcement (Av≤Av,min), Vc is calculated by the equations (c) in the table. ACI 318-19 code suggests ρw may be taken as the sum of the areas of longitudinal bars located more than two-thirds of the overall slab depth away from the extreme compression fiber. Therefore, RISA calculates ρw as the sum of the areas of longitudinal bars on the tension face of the slab.

When other ACI 318 editions are selected, the shear strength of the concrete alone is limited to the standard 2*λ*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).

Two Way Shear (Punching Shear)

Two Way Shear(Punching Shear) can cause a shear failure consisting of a plane through the slab which forms a closed circuit around a support or a location of high load concentration. This type of shear is checked for each column that supports the slab, and for any columns which terminate on the slab without a column below. With this type of failure the column "punches" through the slab, leaving a hole in the slab and allowing the slab to move vertically relative to the column. Because it is not possible to provide stirrup shear reinforcing for concrete slabs, the plain concrete shear strength is used. The most common method of addressing punching shear is to increase the perimeter of the failure plane, thereby increasing the amount of concrete to resist the shear. This can be addressed either by using a larger column, or by thickening the slab in an area around the column (known as a Shear Cap). RISAFloor ES automatically calculates the ratio of shear stress to shear capacity around each column and produces a warning if that value exceeds 1.0 anywhere in the model.

Refer to the Punching Shear - Design and the Punching Shear - Results topics for more information about punching shear checks around a column.

Note:

- Shear stud reinforcement is not currently supported for punching shear checks.

Limitations

Post Tensioned Reinforcement design is not currently supported in RISAFloor ES.

Skip Loading is not currently considered on a two-way slab. The skip loading can be applied to a Beam Supported floor, see Skip Loading for more information.

Creep/Long Term Deflectionwill only be analyzed for concrete floor slabs when the User Defined method is selected for the reinforcement design. Creep/Long Term Deflection is not currently analyzed in RISAFloor ES for Concrete Floor slabs when using the design strip method. In this case, the program will check point deflection based on the cracked or uncracked moment of inertia. For further information on the deflection calculations, see theDeflection topic.

Elevated Slabs - Spreadsheet Results

After solving the model, spreadsheet results for the analysis and design become available. These can be accessed through the Results menu or the Results toolbar. The information presented below is only relevant for Concrete Floor Slabs. For detail report information see the Elevated Slab - Design Strip Detail Reports topic.

Slab Point Deflection

The Slab Point Deflection spreadsheet displays the absolute deflection of points in the slab for each load combination. Deflection is reported in the vertical global Y-axis. You can select the results per floor plan by selecting from the drop-down box at the top.

Slab Point Deflection

1: Floor Plan 1 (Slab)

	LC	Point Label	Y [in]
16	1	N16	-.036
17	1	N17	0
18	1	N18	.019
19	1	N19	-.018
20	1	N20	.002
21	1	N21	-.004
22	1	N22	.022
23	1	N23	-.004
24	1	N24	.022
25	1	N25	-.018

Note:

- For more information see Elevated Slabs - Analysis topic.
- If no results appear in this spreadsheet it is probable that you have not solved any Load Combinations that have the Service checkbox turned on.

Slab Strip Deflection

The maximum vertical deflection (in the global Y-axis) is reported for each Span of each Support Line. Under the **Standard** tab the deflection is reported separately for the Dead Load, the Live Load, and the combined Dead and Live Load. For each Load Category an absolute maximum deflection is reported over the span, and then a ratio of clear span to deflection is also reported. The clear span (face of support to face of support) is shown in the Length column.

Slab Strip Deflection, Standard Cats

Standard | Alternate

1: Second Floor (Slab)

	Label	Span	Length[ft]	DL[in]	DL Ratio	LL[in]	LL Ratio	DL+LL[in]	DL+LL Ratio
1	SL1	1	11.667	.464	302	.101	1381	.563	249
2		2	28.25	.292	1160	.064	5278	.357	950
3		3	11.583	.504	276	.11	1262	.612	227
4	SL2	1	18.5	.375	592	.081	2725	.454	490
5		2	28.333	.293	1161	.065	5255	.36	945
6		3	18.5	.492	451	.107	2074	.596	373
7	SL3	1	18.583	.184	1210	.041	5480	.226	986
8		2	28.333	.293	1161	.065	5255	.36	945

Values which exceed the allowable deflections defined in the Slab Design Rules are shown in red. Alternate deflection criteria defined in the **Slab Design Rules** is shown on the **Alternate** tab.

Note:

- See also the Design Strip Deflection for further information.

Slab Design Strip Results

The **Slab Results/Rebar** spreadsheet reports the design results for the Design Strips.

Strip Reinforcing

Design Strip Results | Design Cut Results

1: Second Floor (Slab)

	Label	Span	Region	Location	UC Top	Top Bars	Length[ft]	Gov Design Cut	UC Bot	Bot Bars	Length[ft]	Gov Design Cut	UC Shear	Gov Design Cut
1	SL1	1-1	Column	Start	.804	#5@5in	12.5	SL1-S1-X21	.078	#8@18in	12.5	SL1-S1-X1	.562	SL1-S1-X21
2				End	--	--	--	--	--	--	--	--	--	--
3			Mid-Rt	Start	.679	#5@15in	12.5	SL1-S1-X21	.029	#8@18in	12.5	SL1-S1-X1	.09	SL1-S1-X9
4				End	--	--	--	--	--	--	--	--	--	--
5			Mid-Left	Start	.5	#5@18in	12.5	SL1-S1-X16	0	#8@18in	12.5	SL1-S1-X21	.06	SL1-S1-X1
6				End	--	--	--	--	--	--	--	--	--	--
7		1-2	Column	Start	.744	#5@5in	9.477	SL1-S2-X1	.319	#8@18in	30	SL1-S2-X26	.488	SL1-S2-X47
8				End	.764	#5@5in	9.727	SL1-S2-X50	--	--	--	--	--	--
9			Mid-Rt	Start	.621	#5@15in	10.343	SL1-S2-X1	.226	#8@18in	30	SL1-S2-X26	.112	SL1-S2-X42

Below are descriptions of each column in the Design Strip Results tab:

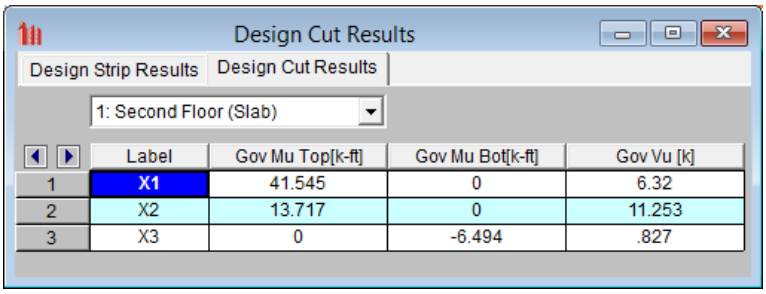
Spreadsheet Column	Description
Label	Support Line Label
Span	The Support Line Span defined by Support Point to Support Point. Note: When the Span is the same number (i.e. 1-1) this indicates a cantilever support line span.
Region	Label for the Design Strip within the Span
Location	The end of the Design Strip (Start or End) which the design results are presented for, based on the order in which the Support Line was drawn (Start is first click, End is second click)
UC Top	Controlling Unity Check for bending when the top face of the slab is in tension. This is the ratio of moment demand to moment capacity, based on the specified rebar.
Top Bars	Required rebar at the top face of the slab, as determined by analysis. This quantity of rebar is sufficient to meet all code requirements unless UC Top is greater than 1.0. If welded wire bars are designed, the output result will be preceded by a “D” instead of “#” to indicate wire bar size.
Length	Distance the top rebar must project from the centerline of support into this Span (including development length). The total piece length of the top bar would be equal to the sum of the Lengths from the two adjacent Spans.
Gov Design Cut	The Design Cut within the Design Strip which produced the controlling UC Top value.
UC Bot	Controlling Unity Check for bending when the bottom face of the slab is in tension. This is the ratio of moment demand to moment capacity, based on the specified rebar.
Bot Bars	Required rebar at the bottom face of the slab, as determined by analysis. This quantity of rebar is sufficient to meet all code requirements unless UC Bot is greater than 1.0. If welded wire bars are designed, the output result will be preceded by a “D” instead of “#” to indicate wire bar size.
Length	Length of the bottom bars in this Span. Since bottom bars are always designed as continuous over the entire span, this number is always the length of this Span (centerline to centerline).
Gov Design Cut	The Design Cut within the Design Strip which produced the controlling UC Bot value.
UC Shear	Controlling Unity Check for one-way out-of-plane shear in the slab. This is the ratio of shear demand to concrete shear capacity, ignoring any steel shear capacity.
Gov Design Cut	The Design Cut within the Design Strip which produced the controlling UC Shear value.

Note:

- For more information see the Design Strip Results topic.

Slab Design Cut Results

The Slab Results/Rebar spreadsheet reports the design results for the Manual Design Cuts.



	Label	Gov Mu Top[k-ft]	Gov Mu Bot[k-ft]	Gov Vu [k]
1	X1	41.545	0	6.32
2	X2	13.717	0	11.253
3	X3	0	-6.494	.827

This spreadsheet only reports Manual Design Cuts, as opposed to Automatic Design Cuts.

Below are descriptions of each column in the Design Cut Results tab:

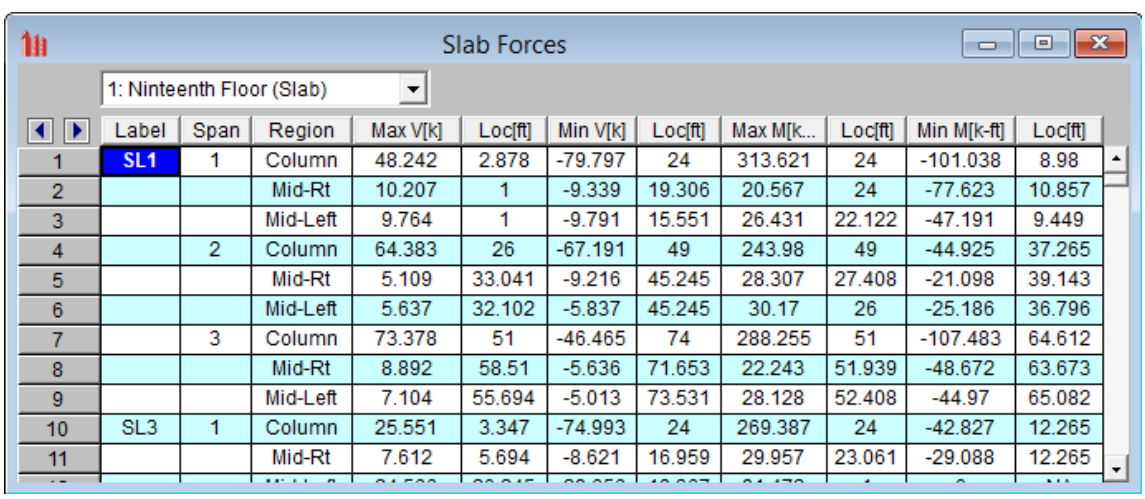
Spreadsheet Column	Description
Label	Manual Design Cut Label
Gov Mu Top	Total bending moment across the Cut which results in the top face of the slab being in tension. This is the maximum bending moment which occurred considering all solved Load Combinations.
Gov Mu Bot	Total bending moment across the Cut which results in the bottom face of the slab being in tension. This is the maximum bending moment which occurred considering all solved Load Combinations.
Gov Vu	Total vertical shear through the plane of the Cut. This is the maximum bending moment which occurred considering all solved Load Combinations.

Note:

- For more information see the Design Strip Results topic.

Slab Forces

The **Slab Forces** spreadsheet is accessible through the Results Toolbar and the Results Menu. This spreadsheet lists all of the slab forces for a given floor. The spreadsheet is organized by Support Line. The forces for each support line are broken out by Span, and the forces for each Span are broken out by Region.



	Label	Span	Region	Max V[k]	Loc[ft]	Min V[k]	Loc[ft]	Max M[k...	Loc[ft]	Min M[k-ft]	Loc[ft]
1	SL1	1	Column	48.242	2.878	-79.797	24	313.621	24	-101.038	8.98
2			Mid-Rt	10.207	1	-9.339	19.306	20.567	24	-77.623	10.857
3			Mid-Left	9.764	1	-9.791	15.551	26.431	22.122	-47.191	9.449
4		2	Column	64.383	26	-67.191	49	243.98	49	-44.925	37.265
5			Mid-Rt	5.109	33.041	-9.216	45.245	28.307	27.408	-21.098	39.143
6			Mid-Left	5.637	32.102	-5.837	45.245	30.17	26	-25.186	36.796
7		3	Column	73.378	51	-46.465	74	288.255	51	-107.483	64.612
8			Mid-Rt	8.892	58.51	-5.636	71.653	22.243	51.939	-48.672	63.673
9			Mid-Left	7.104	55.694	-5.013	73.531	28.128	52.408	-44.97	65.082
10	SL3	1	Column	25.551	3.347	-74.993	24	269.387	24	-42.827	12.265
11			Mid-Rt	7.612	5.694	-8.621	16.959	29.957	23.061	-29.088	12.265

Below are descriptions of each column in the **Slab Forces** spreadsheet:

Spreadsheet Column	Description
Label	Support Line Label
Span	Span Number of Support Line
Region	Region within Span.
Max V & Loc	Maximum shear in that region with in that span. The Loc Column states where in that span the maximum is located.
Min V & Loc	Minimum shear in that region with in that span. The Loc Column states where in that span the maximum is located.
Max M & Loc	Maximum moment in that region with in that span. The Loc Column states where in that span the maximum is located.
Min M & Loc	Minimum moment in that region with in that span. The Loc Column states where in that span the maximum is located.

Elevated Slabs - Design Strip Detail Reports

This topic is specific to design strip detail report results.

Note

- See the Elevated Slabs - Design Strip Modeling topic for information on modeling design strips.
- See the Elevated Slabs - Spreadsheet Results topic for information regarding spreadsheet results.

Design Strip Detail Reports

The detail report gives detailed information about the design strip. Here we will walk through how to access the analytical and design information for the Design Strips.

Note:

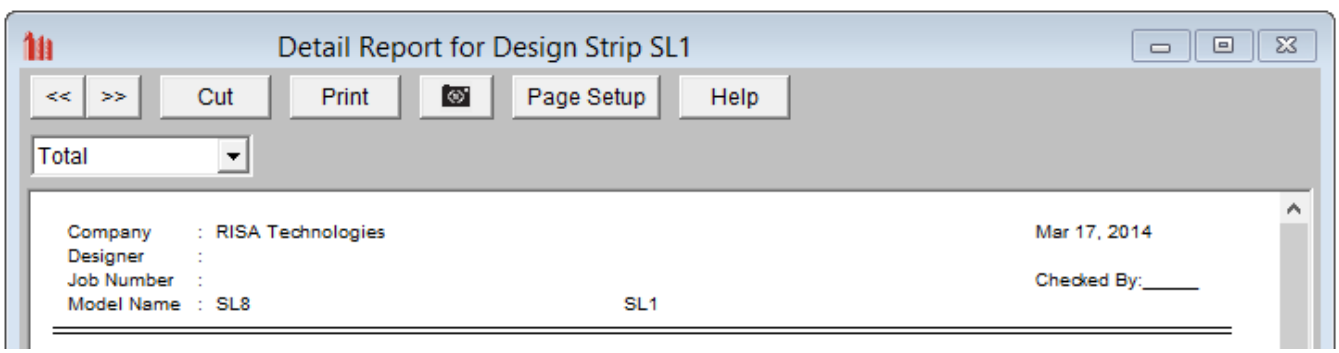
- In order to get results for a Design Strip, you must first draw Support Lines.

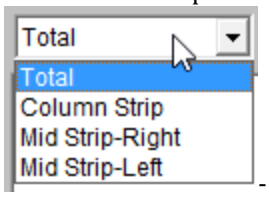
Accessing the Detail Reports and the Specific Windows

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

1. If you have the **Slab Results/Rebar** spreadsheet open in the Design Strip Results tab, there will be a button at the top of the screen: Detail Report for Current Item. This will open up the detail report window.
2. If you are in a graphic view of your model, there is a Detail button that will open up the detail report window when you click on a particular Design Strip.

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

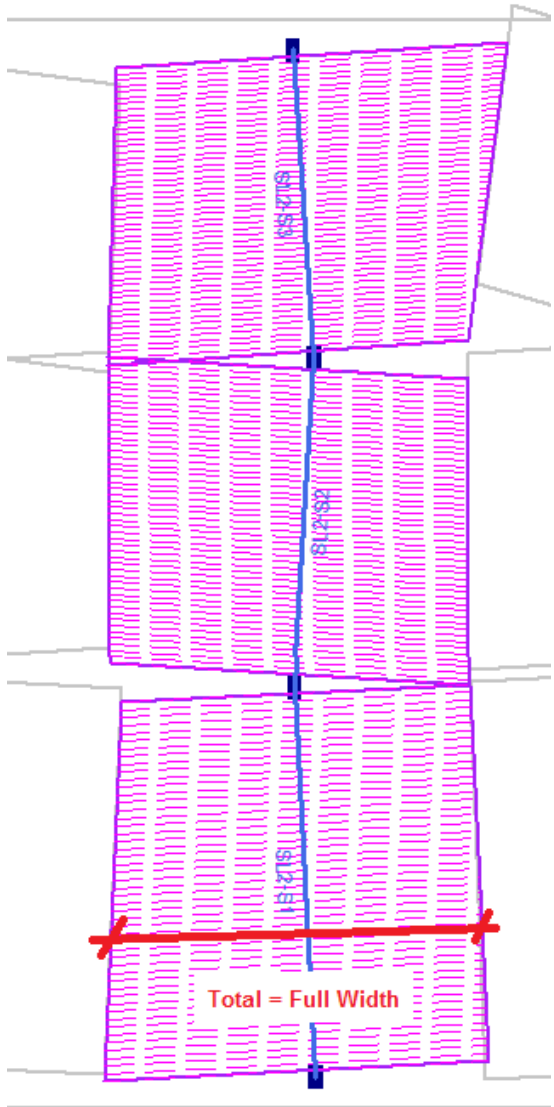


- << >> - Allows you to click between the different design strips in your model.
- - This button will allow you to take a snapshot of the current detail report you are viewing so that it can be added to a report. View the Printing topic for more information.
- 

- This dropdown list allows you to select between the total design strip results or the three different design region results. Below we will explain the importance of each of these sections.

Total Design Strip Detail Report

The Total Design Strip report gives an overall summary of the full width of the design strip. The Total Design Strip will only give analysis information (not design/reinforcement information). The picture below illustrates the Design Cut across the Total Design Strip (at only one span for simplicity).



Input Echo

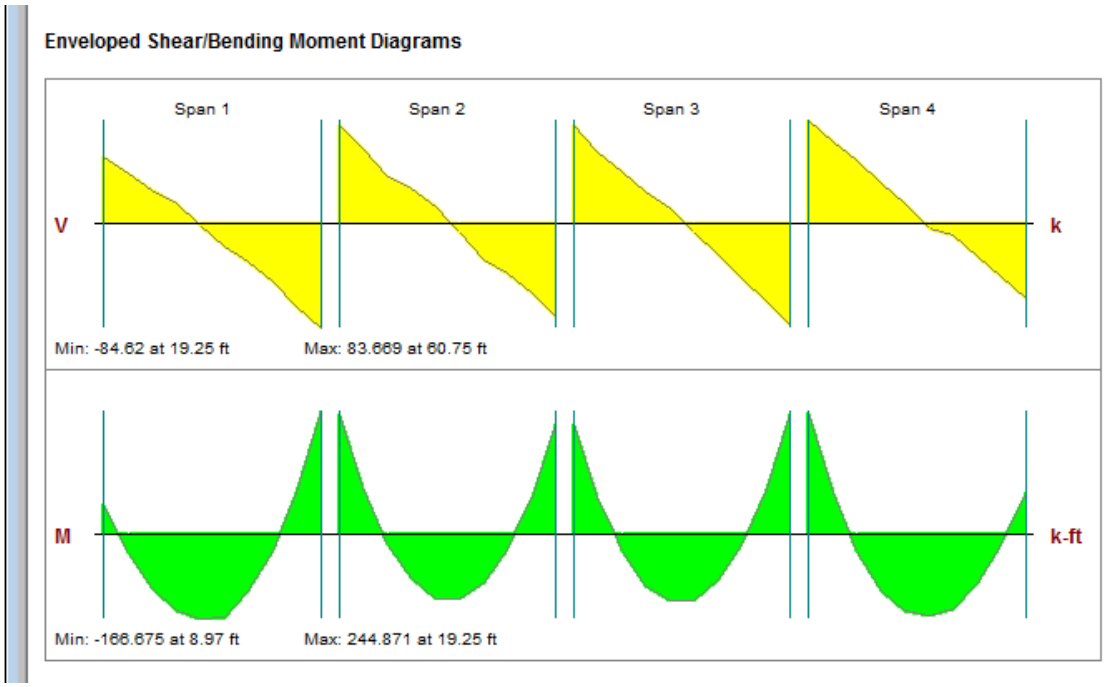
This lists information about the total design strip details. Below is the input echo portion of the detail report.

Strip:	SL1	Concrete Weight:	.145 k/ft^3	Stress Block:	Rectangular
Material:	Conc3000NW	λ	1	Top Cover:	.75 in
Total Cuts/Span:	50 / 5	E_Concrete:	3156 ksi	Bottom Cover:	.75 in
Design Rule:	N-S Rebar	Tension Bar Fy:	60 ksi		
Strip Orientation:	North-South	F'c:	3 ksi	Rho Min (T&S):	.0018
		Flex. Rebar Set:	ASTM A615	Slab Thickness:	14 in

Criteria	Description
Strip	Slab Design Strip Label defined in the Slab Design Strips spreadsheet
Material	Slab Material defined in the Slab Definitions spreadsheet
Total Cuts/Span	Number of Design Cuts defined in the Slab Design Strips spreadsheet/Number of spans per support line
Design Rule	Design strip Design Rule defined in the Slab Design Strips spreadsheet
Strip Orientation	N-S or E-W defined in the Slab Design Strip Spreadsheet
Concrete Weight	Slab weight defined in the Slab Definitions spreadsheet
Lamda	Lightweight concrete modification factor defined in the Concrete Materials spreadsheet
E_Concrete	Elastic modulus defined in the Concrete Materials spreadsheet
Tension Bar Fy	Strength of Bar in tension defined in the Materials spreadsheet- Concrete tab- Flex Steel
F'c	Concrete Yield Stress from Material spreadsheet
Flex. Rebar Set	The Concrete Rebar Set is selected in the Global Paremeters- Concrete tab. The Standard ASTM A615 (imperial), ASTM A1064 (imperial) and ASTM A615M (metric) are supported.
Stress Block	Stress Block from the Model Settings- Concrete tab. Only Rectangular is supported.
Top Cover	Clear cover from the top bar to the top of slab
Bottom Cover	Clear cover from the bottom bar to the bottom of the slab
Rho Min (T&S)	The ratio of reinforcement corresponding the area of steel to bd
Slab Thickness	Overall thickness of the slab defined in the Slab Definitions spreadsheet.

Diagrams

The Shear and Moment diagrams display the Envelope result based on the Load Combinations checked. The program assumes that your spans start and end at a support. The supports are represented in the diagrams as gaps. The column strip and Mid-strip detail reports will show the support labels in the Rebar Detailed Drawing for reference. The minimum and maximum values are listed at the bottom for the entire support line.



Analysis Results

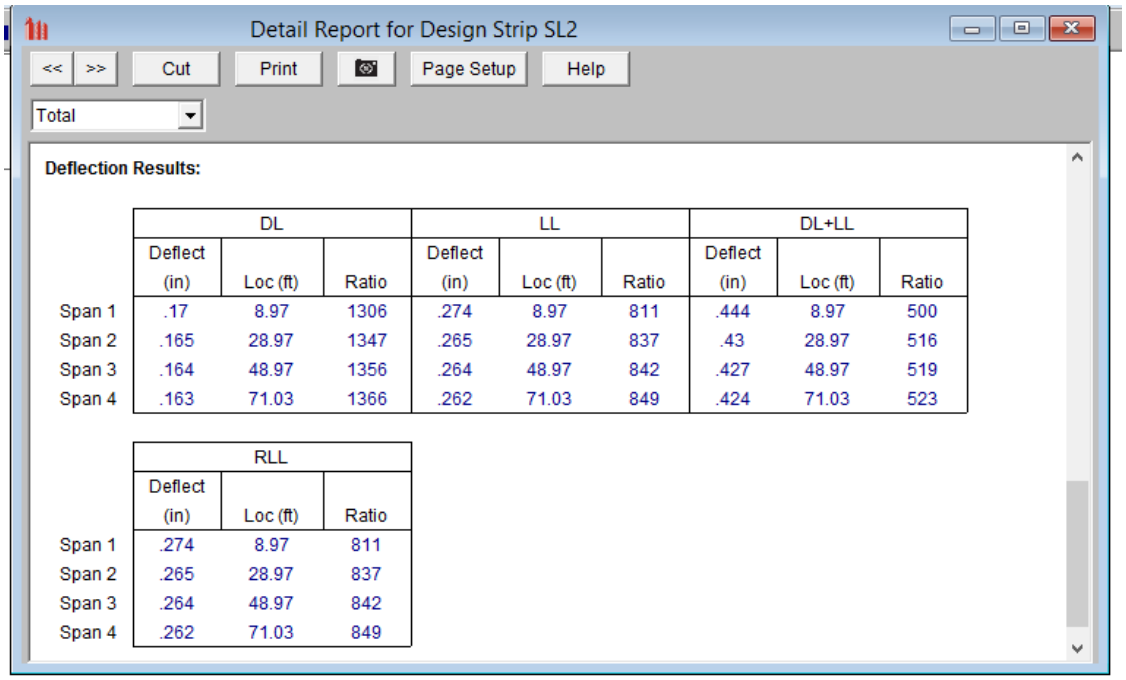
This portion gives the detailed information of the diagrams above displaying the minimum and maximum shear and moment in each span, along with the controlling location.

Analysis Results:

	Shear V (k, ft)				Moment M (k-ft, ft)			
	Max	Loc	**Min**	Loc	**Max**	Loc	**Min**	Loc
Span 1	55.302	.75	-87.92	19.25	243.852	19.25	-165.941	8.97
Span 2	79.677	20.75	-78.185	39.25	239.955	20.75	-129.773	31.03
Span 3	77.289	40.75	-51.905	59.25	214.581	40.75	-108.333	48.97
Span 4	85.352	60.75	-54.277	79.25	239.502	60.75	-160.103	71.03

Slab Strip Deflections

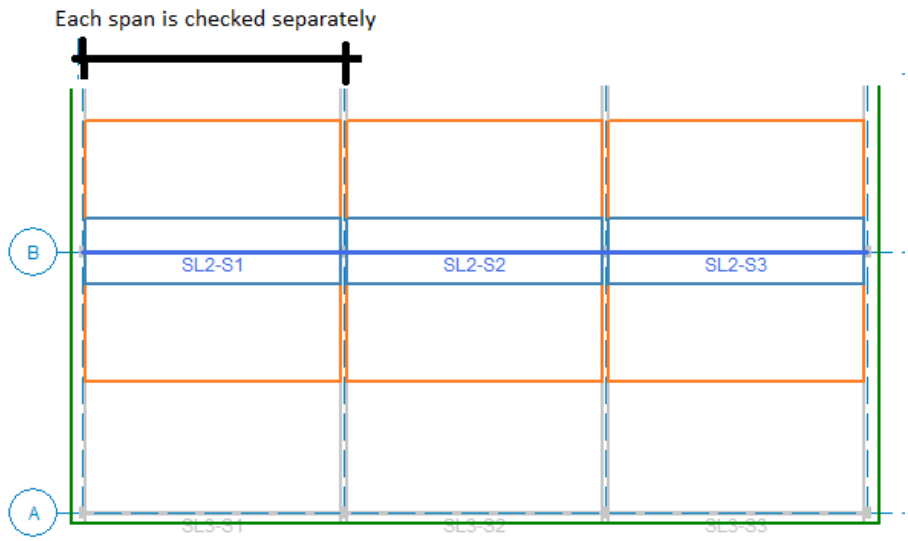
These deflections are based on the Total Design Strip width along the Support Line Spans. The program also reports this value in the Slab Strip Deflections spreadsheet. The Slab Strip Deflection is based on the Load Categories (not Load Combinations). Each Support Line has a **Slab Design Rule** which defines the Deflection criteria. The program finds the actual deflection and compares it to the limits defined in the **Slab Design Rule** to calculate the ratio.

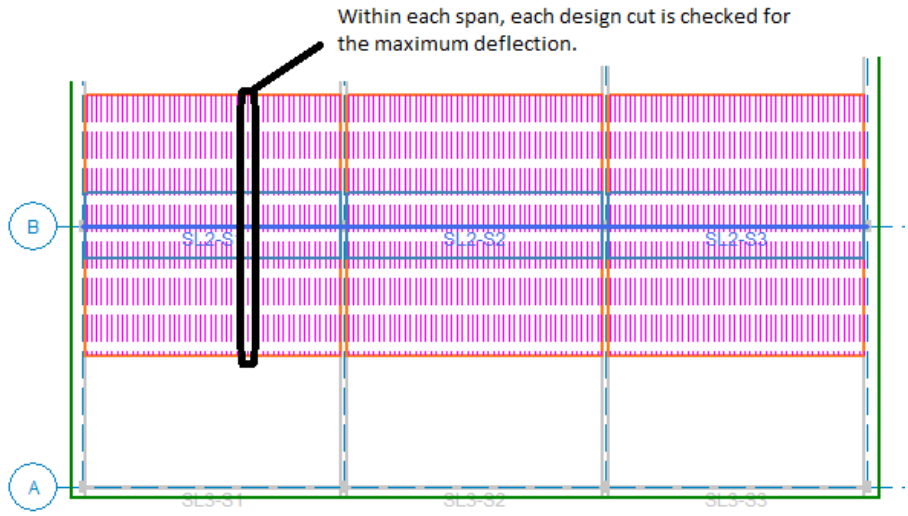


	DL			LL			DL+LL		
	Deflect (in)	Loc (ft)	Ratio	Deflect (in)	Loc (ft)	Ratio	Deflect (in)	Loc (ft)	Ratio
Span 1	.17	8.97	1306	.274	8.97	811	.444	8.97	500
Span 2	.165	28.97	1347	.265	28.97	837	.43	28.97	516
Span 3	.164	48.97	1356	.264	48.97	842	.427	48.97	519
Span 4	.163	71.03	1366	.262	71.03	849	.424	71.03	523

	RLL		
	Deflect (in)	Loc (ft)	Ratio
Span 1	.274	8.97	811
Span 2	.265	28.97	837
Span 3	.264	48.97	842
Span 4	.262	71.03	849

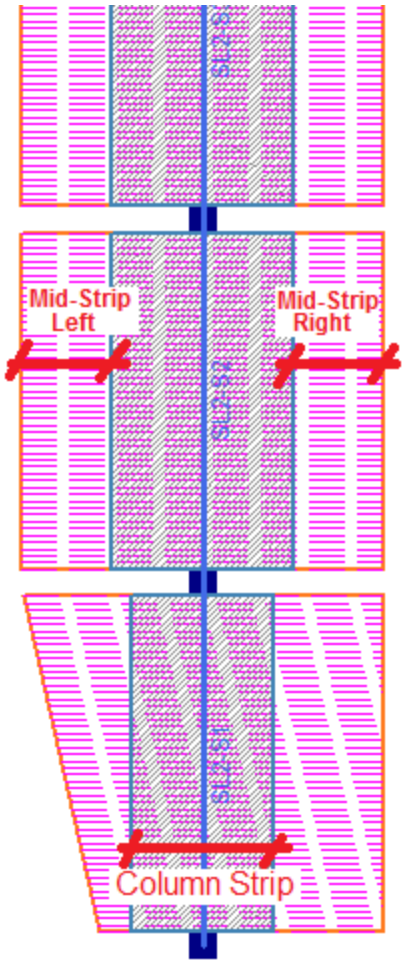
The program checks each span separately, and within those spans each Design Cut is checked to find the maximum deflection.





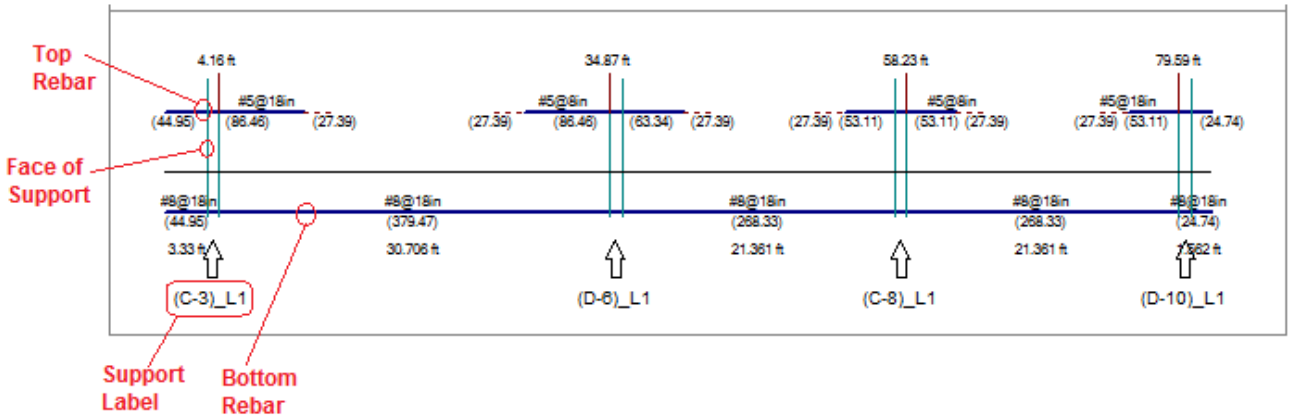
Column Strip/Middle Strip Detail Report

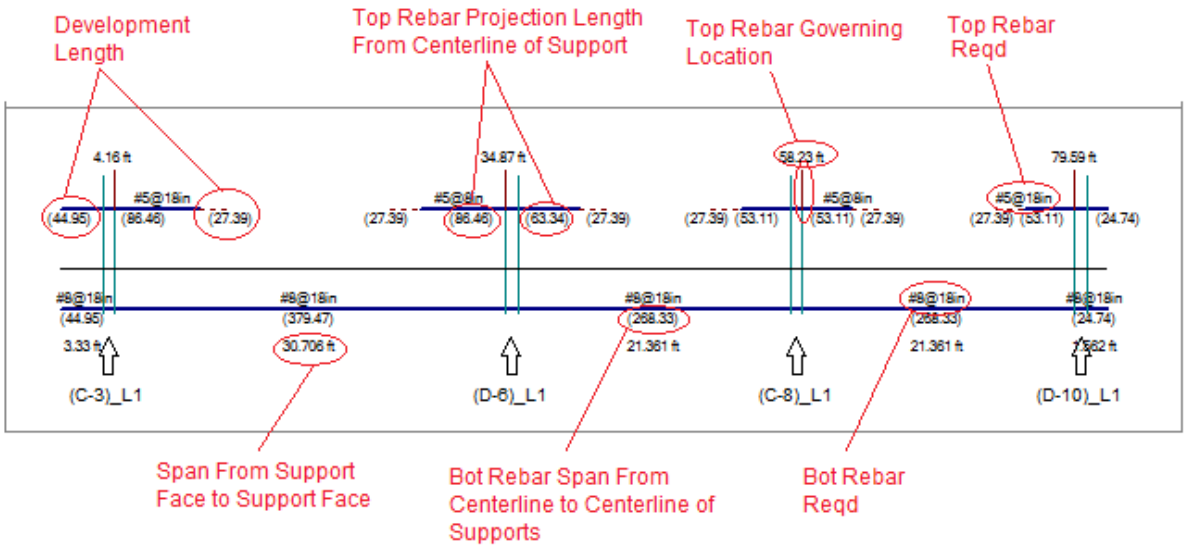
The Column Design Strip, Mid-Strip Right and Mid-Strip Left Detail Report gives the summary of the all the cuts within that strip. The picture below illustrates the Design Cuts (pink dashed lines) across the strip (at only one span for simplicity).



Rebar Detailed Drawing

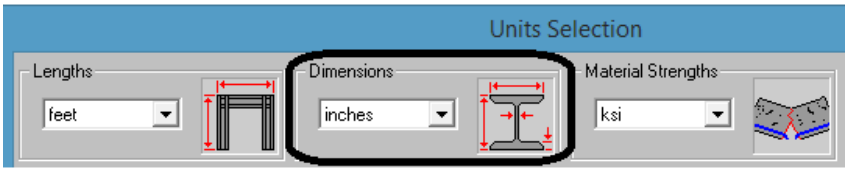
The Column Strip and Mid-Strip Detail Reports have reinforcement design. The Rebar Detailed Drawing is directly in-line with the Moment diagrams so that you can see the analytical demand. Below shows a picture describing all of the labels and symbols on the drawing.



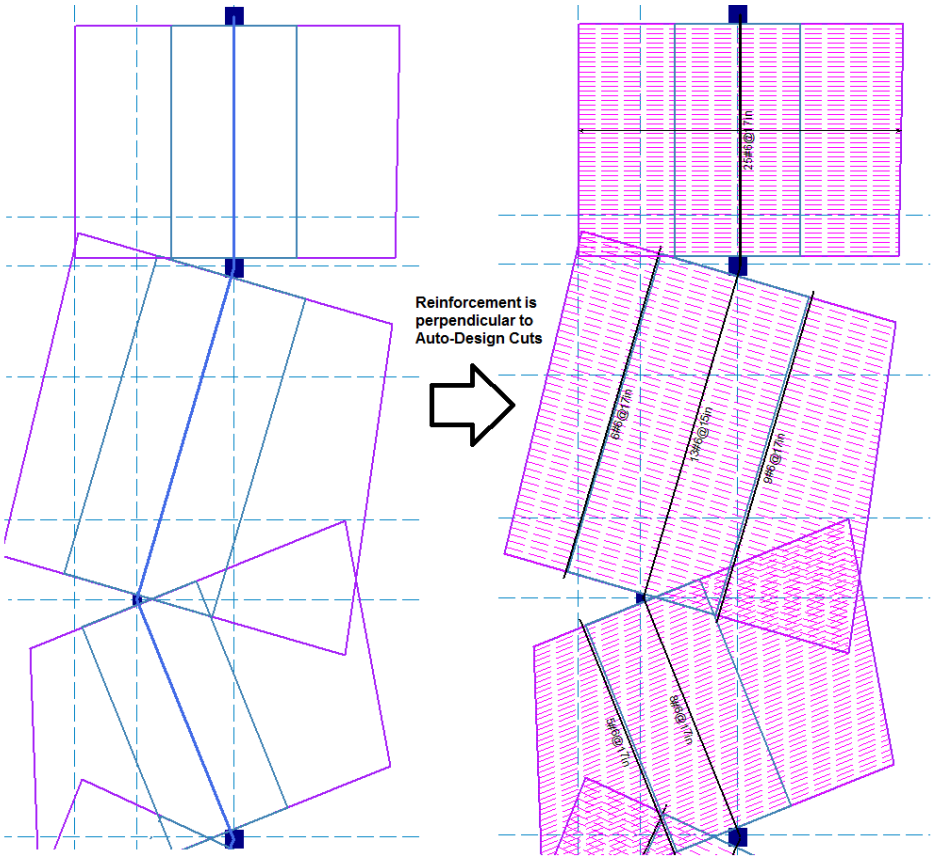


Note:

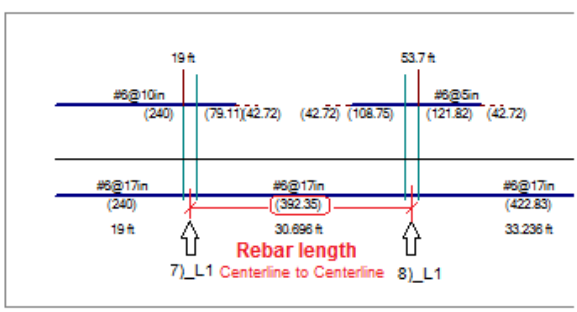
- For full description of how the reinforcement is designed, see the Reinforcement Design topic.
- The units shown in the Rebar Detailed Drawing are based on the **Dimensions** units in the Unit Selection dialog unless noted otherwise.

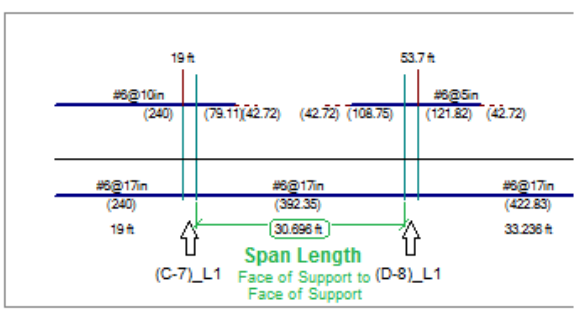


- If there are any overlapping items in the detail report diagrams when using metric units, change the dimensions to centimeters instead of millimeters and most of the overlapping goes away.
- The reinforcement is reported per Design Region and it's orientation is perpendicular to the Auto-Design Cut. Below is an example that shows a Design Strip that has spans that are orthogonal and skewed.



- There are two numbers listed at the bottom of the Reinforcement drawing. One indicates the Rebar length, the other indicates the Span length.





Analysis Results

This portion gives the detailed information of the diagrams above displaying the minimum and maximum in each span and the controlling location. The location is the overall location in relation to the Support Line.

Analysis Results:

	Shear V (k, ft)				Moment M (k-ft, ft)			
	Max	Loc	Min	Loc	Max	Loc	Min	Loc
Span 1	55.302	.75	-87.92	19.25	243.852	19.25	-165.941	8.97
Span 2	79.677	20.75	-78.185	39.25	239.955	20.75	-129.773	31.03
Span 3	77.289	40.75	-51.905	59.25	214.581	40.75	-108.333	48.97
Span 4	85.352	60.75	-54.277	79.25	239.502	60.75	-160.103	71.03

Code Check and Bending Steel Results

The first table describes the positive moment demand and the top reinforcement.

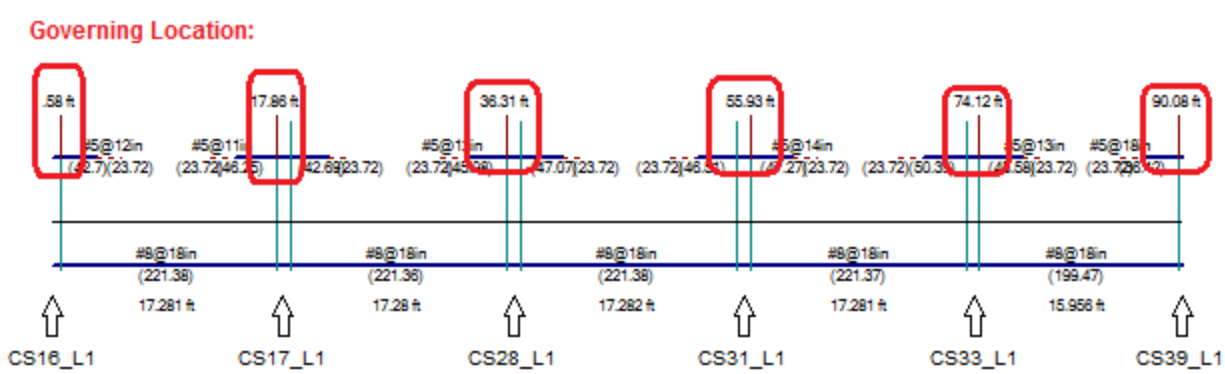
ACI 318-11 Code Check and Bending Steel Req and Provided, Units: in^2

Gov Loc (ft)	Top Bending Check	Gov Mu Top (k-ft)	Gov LC	phi*Mn top (k-ft)	Gov Cut	Top Rebar	Top As Prvd	Top As Reqd
19	.856	399.546	2	466.528	SL1-S1-X31	**#6@10**	8.394	8.087
53.7	.856	804.888	2	939.83	SL1-S3-X1	**#6@5**	17.671	14.927
88.93	.742	327.895	2	441.825	SL1-S4-X1	**#6@10**	7.952	7.601
118.45	.848	291.531	2	343.668	SL1-S5-X1	**#6@10**	6.185	5.922
139.45	.451	90.101	2	199.976	SL1-S5-X50	**#6@18**	3.534	2.095

Criteria	Description
Gov Loc	This gives the overall location along the span that yielded the worst case design cut for top reinforcement design. See below for more information.
Top Bending Check	This is a calculated code check based on the top reinforcement design capacity vs. moment demand.
Gov Mu Top	This is the largest moment demand in this segment. The top reinforcement spacing must be equal on both sides of a support so this governing positive moment could be from the left or right of the support. See Gov Loc for the actual location.
Gov LC	This is the governing Load Combination for the top reinforcement.
phi*Mn top	This is the capacity based on the amount of top reinforcement.
Gov Cut	This is the governing Design Cut that controls the design for the top reinforcement.
Top Rebar	Size and spacing of flexural reinforcement at the top of the slab. If welded wire bars are designed, the output result will be preceded by a “D” instead of “#” to indicate wire bar size.
Top As Prvd	Area of Steel provided in the top of the slab
Top As Reqd	Area of Steel required in the top of the slab

Governing Location- This gives the overall location along the span that yielded the worst case design cut. This table is organized by the governing location because at each support point there will be one governing location that controls the design. The top reinforcement is required to be the same spacing at each side of support point.

This governing location is also displayed on the Rebar Detailed Drawing with a red "tick" mark and the dimension listed.



The second table describes the negative (mid-span) moment demand and the bottom reinforcement. This table is span based since the bottom reinforcement is designed span by span.

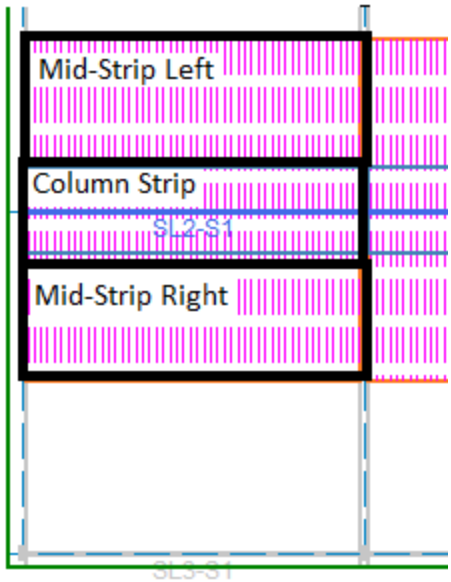
	Bot Bending Check	Gov Mu Bot (k-ft)	Gov LC	phi*Mn bot (k-ft)	Gov Cut	Bot Rebar	Bot As Prvd	Bot As Reqd	Rho Prvd (Gross)
Span 1	.187	-56.01	2	299.256	SL1-S1-X8	**#6@17**	5.301	4.749	0.00519
Span 2	.501	-149.855	2	299.256	SL1-S2-X22	**#6@17**	5.301	4.749	0.00201
Span 3	.637	-190.856	2	299.591	SL1-S3-X29	**#6@17**	5.301	4.959	0.00192
Span 4	.707	-211.343	2	298.751	SL1-S4-X26	**#6@16**	5.301	4.959	0.00214
Span 5	.509	-114.093	2	224.299	SL1-S5-X28	**#6@17**	3.976	3.478	0.00206

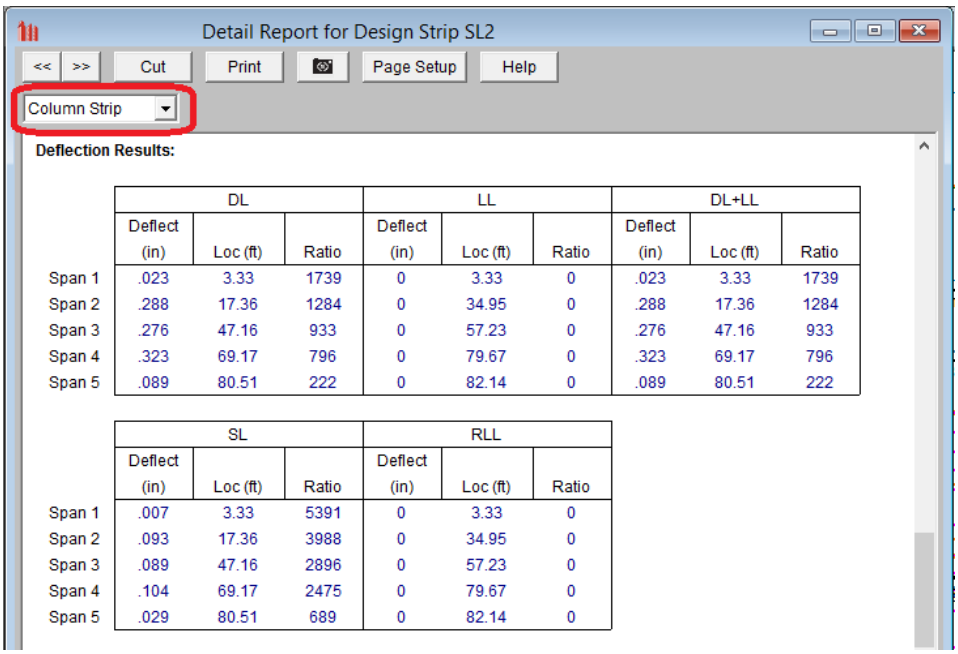
Criteria	Description
Bot Bending Check	This is a calculated code check based on the bottom reinforcement design capacity vs. moment demand.
Gov Mu Bot	This is the largest negative moment demand in the span.
Gov LC	This is the governing Load Combination for the bottom reinforcement.
phi*Mn bot	This is the capacity based on the amount of bottom reinforcement.
Gov Cut	This is the governing Design Cut that controls the design of the bottom reinforcement.
Bot Rebar	Size and spacing of flexural reinforcement at the bottom of the slab. If welded wire bars are designed, the output result will be preceded by a “D” instead of “#” to indicate wire bar size.
Bot As Prvd	Area of Steel provided in the bottom of the slab
Bot As Reqd	Area of Steel required in the bottom of the slab
Rho Prvd(Gross)	The ratio of reinforcement corresponding the area of steel (entire slab) to bd

Column Strip/Middle Strip Deflection

The Total Design Strip is broken into 3 regions: Column Strip, Mid-Strip Left and Mid-Strip Right. The Detail Report gives more information on the deflection by reporting the maximums in each of these regions.

The Slab Strip Deflection is based on the Load Categories (not Load Combinations). Each Support Line has a **Slab Design Rule** which defines the Deflection criteria. The program finds the actual deflection and compares it to the limits defined in the **Slab Design Rule**.





	DL			LL			DL+LL		
	Deflect (in)	Loc (ft)	Ratio	Deflect (in)	Loc (ft)	Ratio	Deflect (in)	Loc (ft)	Ratio
Span 1	.023	3.33	1739	0	3.33	0	.023	3.33	1739
Span 2	.288	17.36	1284	0	34.95	0	.288	17.36	1284
Span 3	.276	47.16	933	0	57.23	0	.276	47.16	933
Span 4	.323	69.17	796	0	79.67	0	.323	69.17	796
Span 5	.089	80.51	222	0	82.14	0	.089	80.51	222

	SL			RLL		
	Deflect (in)	Loc (ft)	Ratio	Deflect (in)	Loc (ft)	Ratio
Span 1	.007	3.33	5391	0	3.33	0
Span 2	.093	17.36	3988	0	34.95	0
Span 3	.089	47.16	2896	0	57.23	0
Span 4	.104	69.17	2475	0	79.67	0
Span 5	.029	80.51	689	0	82.14	0

Design Cut Detail Reports

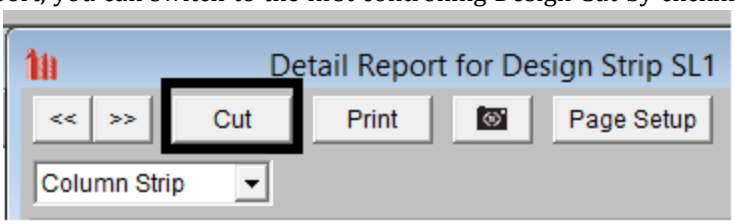
Similar to the detail report for Design Strips, you have the ability to view the detail report for a Design Cut. Each Design Strip is made up of multiple Design Cuts. The Design Cuts reports are given for each cut in the Design Strip and allow you to get detailed calculation information.

Note:

- It is also possible to manually create single cuts and get results for these as well. See Elevated Slab - Design Strip Modeling topic for more information.

Accessing the Design Cut Detail Reports

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

1. From within the Design Strip detail report, you can switch to the first controlling Design Cut by clicking on the cut button at the top of the screen. 

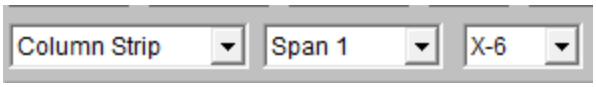
2. If you are in a graphic view of your model, there is a Detail button that will open up the detail report window when you click on a Design Cut.

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

<< >> - Allows you to toggle between the different Design Cuts in your model.

- This 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.

There are drop-down menus to help navigate between design regions, spans and cuts.



Input Echo

Company : RISA Technologies
Designer :
Job Number :
Model Name : Tutorial B6 Starter File SL1-S1-X1

Apr 25, 2023
12:04 PM
Checked By:_____

Cut:	SL1-S1-X1	Max Top bar Spac.:	18 in	Stress Block:	Rectangular
Material:	Conc3000NW	Min Top bar Spac.:	3 in	Slab Design Option:	Construction
Start:	(5 , 9.33) ft	Max Bot bar Spac.:	18 in	Rebar Spacing Inc:	1 in
End:	(16 , 9.33) ft	Min Bot bar Spac.:	3 in	Design Rule:	N-S Rebar

This lists information about the Design Cut details. Below is the input echo portion of the detail report.

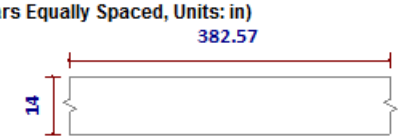
Criteria	Description
Cut	Slab Design Cut Label automatically generated within each Design Strip
Material	Slab Material defined in the Slab Definitions spreadsheet
Start	Point Location defining the start of the Design Cut
End	Point Location defining the end of the Design Cut
Max Top bar Spac	Maximum Top Rebar Spacing defined in the Slab Design Rules spreadsheet
Min Top bar Spac	Minimum Top Rebar Spacing defined in the Slab Design Rules spreadsheet
Max Bot bar Spac	Maximum Bottom Rebar Spacing defined in the Slab Design Rules spreadsheet
Min Bot bar Spac	Minimum Bottom Rebar Spacing defined in the Slab Design Rules spreadsheet
Stress Block	Stress block used in the analysis defined in Model Settings
Slab Design Option	Slab Design Option used in the rebar design defined in Model Setting
Rebar Spacing Inc	The increment of bar spacing the program will try during the iteration of design
Design Rule	Design Rule defined in Slab Design Rules spreadsheet

Design Cut - Total

ACI 318-11 Code Check

Top Bending Check	NA	Bot Bending Check	NA	1 Way Shear Check	NA
Gov Mu Top	0 k-ft	Gov Mu Bot	-61.036 k-ft	Gov Vu	8.962 k
phi*Mn Top	NA	phi*Mn Bot	NA	phi*Vn	NA
Tension Bar Fy	60 ksi	Concrete Weight	.145 k/ft^3	Top Cover	.75 in
F'c	3 ksi	λ	1	Bottom Cover	.75 in
Flex. Rebar Set	ASTM A615	E_Concrete	3156 ksi		
Top As Reqd:		Bot As Reqd:			
Analysis	NA	Analysis	NA		
4/3	NA	4/3	NA	As Reqd(T/S)	NA
Min Flex	NA	Min Flex	NA	Rho Reqd(T/S)	NA
Top As Prvd	NA	Bot As Prvd	NA	Rho Prvd(Gross)	NA
Max Deflection	1.26 in				
Gov Category	DL+LL				

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



The Total Design Cut contains only analysis results in the Design Cut detail report. Since the Design Strip reinforcement design is based on the Design region (Column Strip, Middle Strip) this Detail Report does not show any reinforcement. It is a useful to present the summation of forces in the entire Design Cut. Listed below are the available controlling values along the total length of the Design Cut.

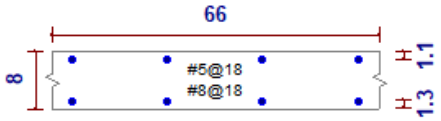
Criteria	Description
Gov Mu Top	Governing maximum positive moment value
Gov Mu Bottom	Governing maximum negative moment value
Gov Vu	Governing maximum shear value
Max Deflection	Maximum deflection anywhere along Total Design Cut
Gov Category	The governing categories defined in the Slab Design Rules- Deflection tab. This in not LC based rather Load Category based. See the Deflection Calculations section for further information.

Design Cut - Column/Middle Strip

ACI 318-19 Code Check

Top Bending Check	0.658	Bot Bending Check	0.000		
Gov Mu Top	24.426 k-ft	Gov Mu Bot	0 k-ft		
phi*Mn Top	37.103 k-ft	phi*Mn Bot	87.509 k-ft		
Tension Bar Fy	60 ksi	Concrete Weight	0.145 k/ft^3	Top Cover	0.75 in
F'c	3 ksi	λ	1	Bottom Cover	0.75 in
Flex. Rebar Set	ASTM A615	E_Concrete	3156 ksi		
Top As Reqd:		Bot As Reqd:			
Analysis	0.799 in^2	Analysis	NA		
4/3	NA	4/3	NA	As Reqd(T/S)	0.95040
Min Flex	0.950 in^2	Min Flex	NA	Rho Reqd(T/S)	0.00180
Top As Prvd	1.227 in^2 (4 #5)	Bot As Prvd	3.142 in^2 (4 #8)	Rho Prvd(Gross)	0.00827
Max Deflection	0.069 in				
Gov Category	DL+LL				

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



Criteria	Description
Top Bending Check	This is a calculated code check based on the top reinforcement design capacity vs. moment demand.
Gov Mu Top	Total positive moment along the cut. (CSA A23.3-14 uses Mf)
phi*Mn Top	This is the capacity based on the amount of top reinforcement. (CSA A23.3-14 uses Mr)
Bot Bending Check	This is a calculated code check based on the bottom reinforcement design capacity vs. moment demand.
Gov Mu Bottom	Total negative moment along the cut. (CSA A23.3-14 uses Mf)
phi*Mn Bot	This is the capacity based on the amount of bottom reinforcement. (CSA A23.3-14 uses Mr)
1 Way Shear Check	This is a calculated code check based on the shear capacity of the concrete slab v shear demand.
Gov Vu	Total shear along the cut. (CSA A23.3-14 uses Vf)
phi*Vn	Shear capacity of the concrete slab. (CSA A23.3-14 uses Vr)
Strain(ey)	CSA A23.3-14 longitudinal strain calculated per the General Method in section 11.3.6.4
Tension Bar Fy	Minimum yield stress of tension reinforcement.
F'c	Concrete compressive strength defined in the Slab Definitions spreadsheet.
Flex. Rebar Set	Flexural Rebar Set defined in the Model Settings- Concrete tab.
Concrete Weight	Slab weight defined in the Slab Definitions spreadsheet.
Lamda	Lightweight concrete modification factor defined in the Concrete Materials spread-sheet.
E_Concrete	Elastic modulus defined in the Concrete Materials spreadsheet.
Top Cover	Distance from top slab edge to top reinforcement defined in the Slab Design Rules Spreadsheet.
Bottom Cover	Distance from bottom slab edge to bottom reinforcement defined in the Slab Design Rules Spreadsheet.

Area of Steel Required

The program calculates the area steel required based on the steps listed below. The green text indicates the number that governed for that Design Cut.

Step 1 (**Analysis**): Calculate the area steel required by analysis. Select a spacing and number of bars that fit the design region.
Step 2 (**Min Flex**): Check that the minimum reinforcement for flexure is being met per ACI 318-14 Section 9.6.1 (ACI 318-11 Section 10.5, CSA A23.3-14 Section 10.5.1.2).
Step 3 **(4/3)**: Check the 4/3*As required by analysis per ACI 318-11 Section 10.5.3, CSA A23.3-14 Section 10.5.1.3.

Note: ACI 2014 does not use the 4/3 As for slabs, the results will show NA.

In the example shown below, the green text indicates that the Top Area of Steel Provided met the Analysis requirements and was increased to meet the Minimum Flexural requirements. Since the 4/3 was greater than Min Flex, it was not used.

ACI 318-11 Code Check

Top Bending Check	0.770	Bot Bending Check	0.000	1 Way Shear Check	0.454
Gov Mu Top	359.001 k-ft	Gov Mu Bot	0 k-ft	Gov Vu	89.838 k
phi*Mn Top	466.405 k-ft	phi*Mn Bot	323.423 k-ft	phi*Vn	198.087 k
Tension Bar Fy	60 ksi	Concrete Weight	.145 k/ft^3	Top Cover	.75 in
F'c	3 ksi	λ	1	Bottom Cover	.75 in
Flex. Rebar Set	ASTM A615	E_Concrete	3156 ksi		
Top As Reqd:		Bot As Reqd:			
Analysis	6.396 in^2	Analysis	NA		
4/3	8.528 in^2	4/3	NA	As Reqd(T/S)	4.71908
Min Flex	8.037 in^2	Min Flex	NA	Rho Reqd(T/S)	0.00180
Top As Prvd	8.394 in^2	Bot As Prvd	5.743 in^2	Rho Prvd(Gross)	0.00539

The bottom reinforcement design is the same as listed above except the temperature and shrinkage requirements are also checked. The bottom reinforcement must be uniform across the entire span but the top reinforcement will stop and start over supports. Therefore the mid-span cuts where there is only bottom reinforcement will always govern for temperature and shrinkage. Below shows an example where the As required is being governed by Temperature and Shrinkage.

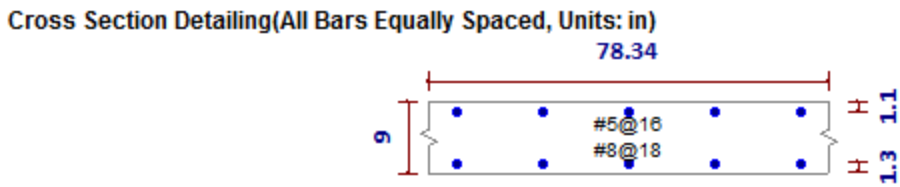
ACI 318-11 Code Check

Top Bending Check	0.000	Bot Bending Check	0.333	1 Way Shear Check	0.203
Gov Mu Top	0 k-ft	Gov Mu Bot	-107.782 k-ft	Gov Vu	40.117 k
phi*Mn Top	0 k-ft	phi*Mn Bot	323.423 k-ft	phi*Vn	198.087 k
Tension Bar Fy	60 ksi	Concrete Weight	.145 k/ft^3	Top Cover	.75 in
F'c	3 ksi	λ	1	Bottom Cover	.75 in
Flex. Rebar Set	ASTM A615	E_Concrete	3156 ksi		
Top As Reqd:		Bot As Reqd:			
Analysis	NA	Analysis	1.878 in^2		
4/3	NA	4/3	2.503 in^2	As Reqd(T/S)	4.71908
Min Flex	NA	Min Flex	8.037 in^2	Rho Reqd(T/S)	0.00180
Top As Prvd	NA	Bot As Prvd	5.743 in^2	Rho Prvd(Gross)	0.00219

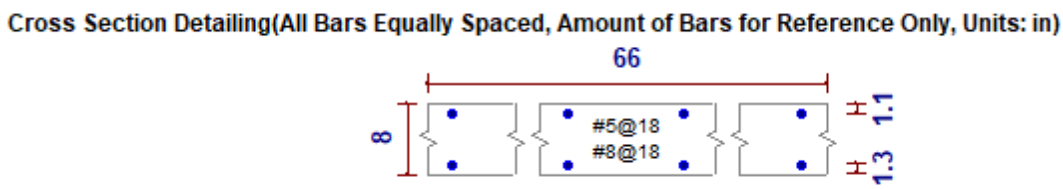
Cross Section Detailing

Depending on the Slab Design Option users selected in Global Model Setting - Concrete tab, the Cross Section Detailing can be different.

If the Construction Method is selected, the cross section will show the actual rebar amount:



If the Theoretical Method is selected, the cross section will show a generic view and the rebar displayed are for reference only. We do not show the rebar amount because the rebar calculation is based on average rebar area and the rebar amount may not be an integer:



Manual Design Cut Detail Reports

Similar to the detail report for Design Strips, you have the ability to view the detail report for a manual Design Cut. You will not receive design information for this option but will be able to view analysis results including governing moment and shear magnitudes.

Note:

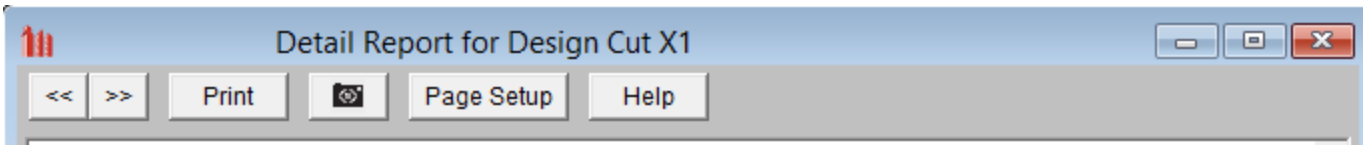
- In order to get results for this feature, you must first draw the Design Cut. See Manual Slab Design Cuts for more information.

Accessing the Design Cut Detail Reports

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

1. If you have the Slab Results/Rebar spreadsheet open under the Design Cut Results, there will be a button at the top of the screen: Detail Report for Current Item. This will open up the detail report window.
2. If you are in a graphic view of your model, there is a Detail button that will open up the detail report window when you click on a Design Cut.

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



- << >> - Allows you to toggle between the different Design Cuts in your model.
- - This 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 thePrintingtopic for more information.

Input Echo

Cut:	X7			Stress Block:	Rectangular
Material:	Conc3500NW			Design Rule:	Typical
Start:	(18.25 , -35) ft				
End:	(18.25 , -18.5) ft				

ACI 318-11 Code Check

Gov Mu Top	406.843 k-ft	Gov Mu Bot	0 k-ft	Gov Vu	108.326 k
Top d	18.5 in	Bot d	18.5 in		
Tension Bar Fy	60 ksi	Concrete Weight	.145 k/ft^3	Top Cover	1.5 in
Shear Bar Fy	60 ksi	λ	1	Bottom Cover	1.5 in
F'c	3.5 ksi	E_Concrete	3409 ksi		

This lists information about the Design Cut details. Below is the input echo portion of the detail report.

Criteria	Description
Cut	Slab Design Cut Label defined in the Slab Design Strips spreadsheet under the Cuts tab
Material	Slab Material defined in the Slab Definitions spreadsheet
Start	Point Location defining the start of the Design Cut
End	Point Location defining the end of the Design Cut
Stress Block	Stress block used in the analysis defined in Model Settings
Design Rule	Design Rule defined in Slab Design Rules spreadsheet
Top d	Slab thickness minus top cover
Bot d	Slab thickness minus bottom cover
Tension Bar Fy	Minimum yield stress of tension reinforcement
F'c	Concrete compressive strength defined in the Slab Definitions spreadsheet
Concrete Weight	Slab weight defined in the Slab Definitions spreadsheet
Lamda	Lightweight concrete modification factor defined in the Concrete Materials spreadsheet
E_Concrete	Elastic modulus defined in the Concrete Materials spreadsheet
Top Cover	Distance from top slab edge to top reinforcement defined in the Slab Design Rules Spreadsheet
Bottom Cover	Distance from bottom slab edge to bottom reinforcement defined in the Slab Design Rules Spreadsheet

Analysis Results

The analysis results are included the Design Cut detail report. Listed below are the available controlling values along the total length of the Design Cut.

Criteria	Description
Gov Mu Top	Total positive moment along the cut.
Gov Mu Bottom	Total negative moment along the cut.
Gov Vu	Total shear along the cut.

These values are also available in the **Slab Results/Rebar** spreadsheet after an analysis is ran. See Elevated Slabs - Spreadsheet Results for more information.

Elevated Slabs - One Way Slab Considerations

One Way slabs are available on Concrete Slab Floors (RISAFloor ES). The One Way Slab floor will act like a beam supported floor distributing the loads onto the beams in one-direction. Loads modeled are attributed to the beams and the beams transfer the loads through the floor framing to walls and columns and ultimately down to the foundation.

The primary function of One Way slab floor is to design the concrete beams. Beams on One Way slabs can be specified as Concrete rectangular or Concrete composite T and L beams. See Concrete Beams for more information.

One Way Slab Requirements

1. One Way slabs must be supported by beams and require beam framing in a "closed circuit" just like Beam Supported Floors.

2. Slab edges must be parallel to edge beams.

Note:

- One Way slabs are used for load attribution only. No rebar design will be done.
- One Way slabs will not create a plate submeshed floor "slab" like the Two Way slab.
- Composite beams can only be designed on One Way Slab floor.

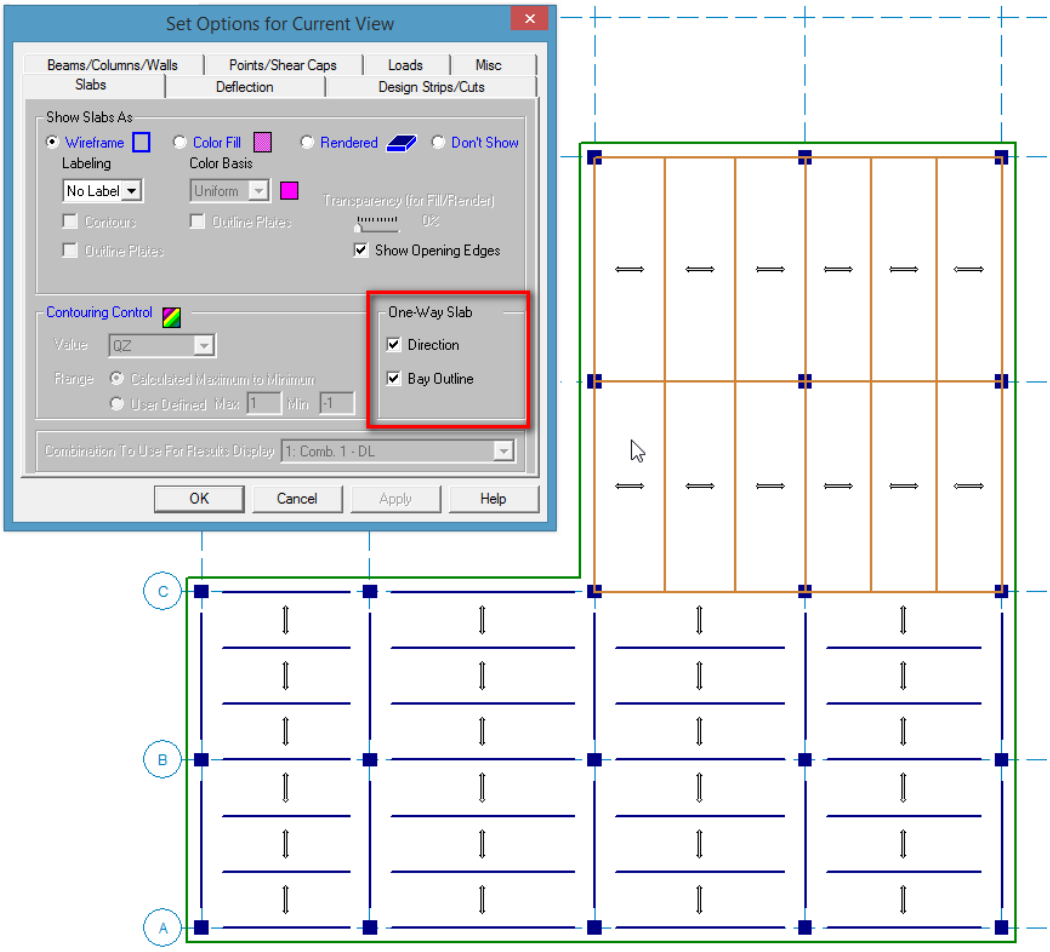
One Way Slab Direction

When choosing a One-Way slab design, there is a default load attribution direction that must be defined in the Slab Definitions. However, this direction can be overwritten by clicking on the**Set One-Way Slab Properties**button located in the drawing toolbar. This will bring up the **One-Way Slab Direction** dialog box.

The **Slab Direction** section allows the user to define the load attribution direction as parallel to the global Z Axis, parallel to the global X Axis, or a user-defined angle with the angle measured from the positive Z Axis in a counter-clockwise direction.

The one-way slab direction can be applied to all bays that are currently selected in the graphics view or the user can choose to click or box the applicable bays individually. The selected bays must be a closed perimeter area.

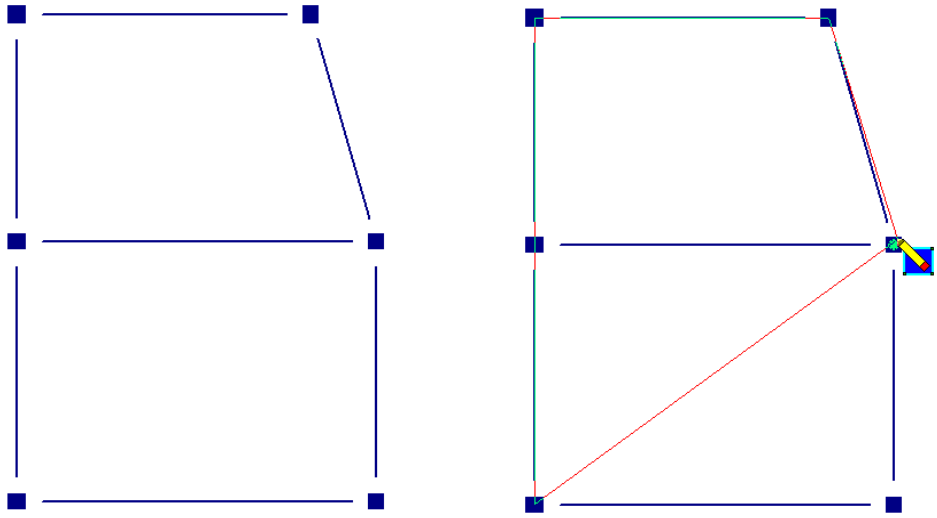
The direction is visible by going to Model Display Options (or Press F2)- Slabs tab- One-Way Slab section. This allows you to turn on the Direction which shows the default direction and any overridden direction. This allows you to visually verify the direction of the slab. The Bay Outline will show up in Orange outline for any bay that has been overridden.



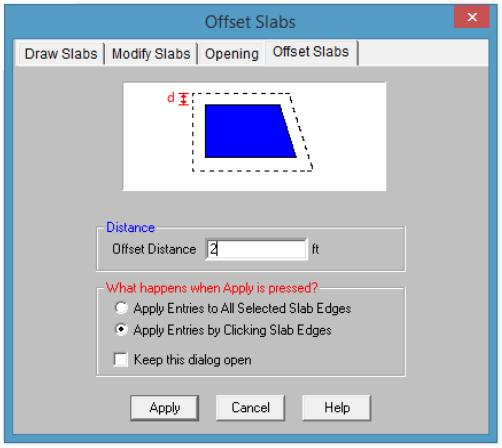
Modeling One-Way Slab Edges

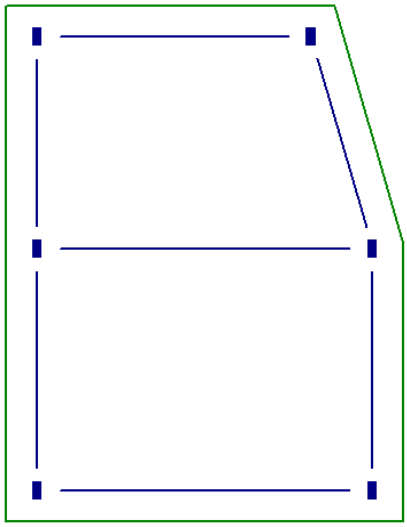
One-Way slabs are required to have slab edges that are parallel to the beam edges (just like a beam supported floor). In Concrete Slab floors, the Draw slab edge tool allows you to draw any shaped edge which is very powerful for a 2-way slab but will break the slab edge rules for one-way slabs. Below is the suggested method for modeling the slab edge for one-way slabs.

Step 1: Draw the slab edge, clicking on every support point. The slab edge will be directly on top of the beams.

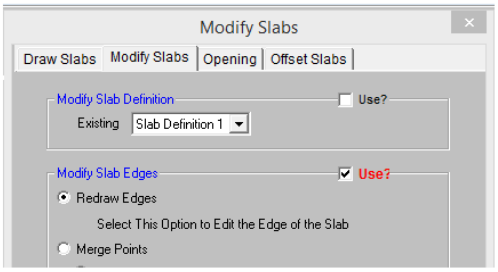


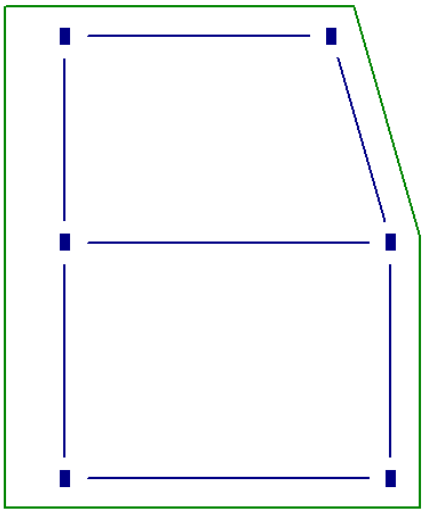
Step 2: Use Offset Slab tool to create an slab overhang.





Step 3: To create an larger overhang on one side, use the Redraw Edges tool.



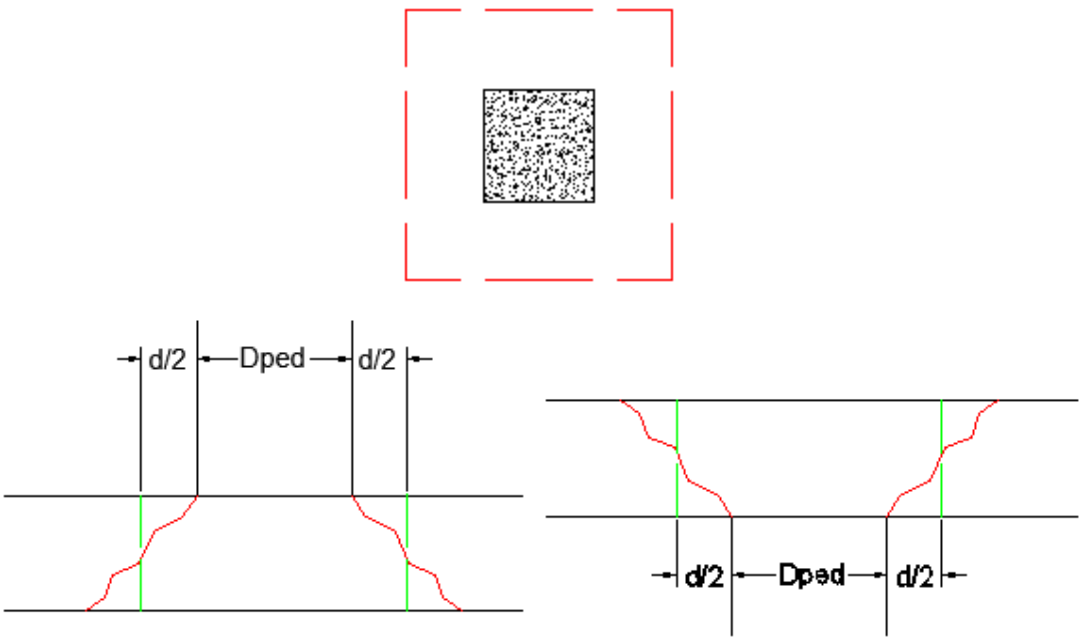


Elevated Slabs - Punching Shear Design

Punching shear calculations are provided for columns/pedestals that land on concrete slabs. Here is some information on those calculations.

Punching Shear Basics

Punching shear is a phenomenon where a concentrated force on a slab causes a shear failure cone that "punches" through.



Punching shear calculations require a demand value and a capacity value. Many hand calculations will consider a punching shear force against a punching shear capacity force. This works if only shear force is present, but is insufficient if a moment is occurring on the column/pedestal. Thus, a stress demand/stress approach is taken. The applied shears and moments, along with the shape of the punching shear failure cone are used to calculate a maximum punching shear stress.

Punching Shear Calculation Values

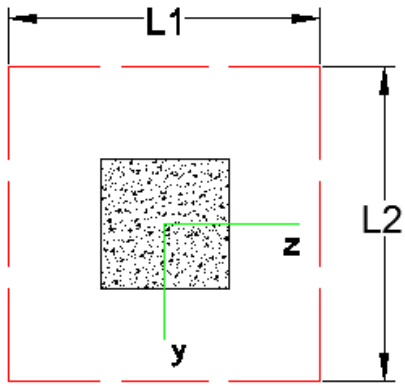
Effective Depth, d

The effective depth of the slab is used to define the punching shear perimeter at a distance of d/2 beyond the edge of the column/pedestal.

The effective depth is based on the smallest depth to centroid of reinforcing for the Design Strips that encompass the pedestal. In cases where no design strips are defined at the pedestal location, the program will assume the bar size and cover specified in the first rule defined in the Design Rules spreadsheet.

L1 and L2

Length of the critical section along the local axes. This value is used to define the punching shear perimeter as well as to define how much of the moment is transferred by eccentricity of shear (as opposed to the portion transferred by flexure).



Note:

- For further discussion refer to ACI section 11.12.1.2 and 13.5.3

Polar Moment Iyy and Polar Moment Izz

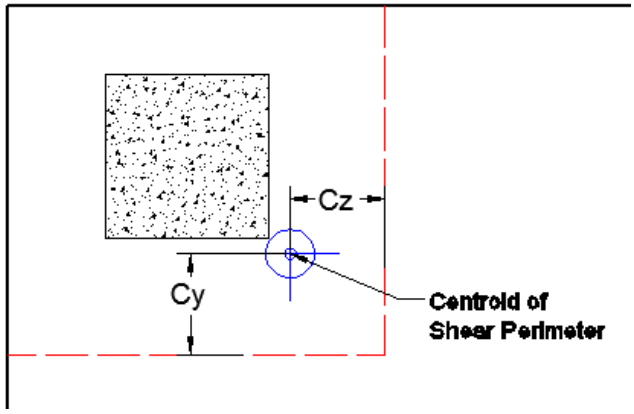
This is a property (Jc) of the critical section which is analogous to polar moment of inertia about the local xx axes. This value is used to determine the shear stress in the critical location due to unbalanced moment.

Note:

- This calculation is very different depending on interior, edge or corner punching shear cases.
- For further discussion and equations refer ACI 318-14 Section 48.4.4.2.3 (ACI 318-11 Section R11.11.7.2) and Chapter 13-8 of the Reinforced Concrete Mechanics and Design textbook by MacGregor & Wight.

Centroid Location, cz and cy

These give the distance from the shear perimeter edge to the centroid of the shear perimeter. For an interior punching shear perimeter this will always be L1/2 or L2/2. For edge and corner cases this value is calculated as the moment of area of the shear perimeter/area of the sides.



Note:

- For further discussion and equations refer to Chapter 13-8 of the Reinforced Concrete Mechanics and Design textbook by MacGregor & Wight.

Gamma xx and Gamma zz

This is calculated from the dimensions of the assumed punching shear perimeter. It represents the fraction of the unbalanced moment for each axis that is transferred by eccentricity of shear.

Note:

- For further discussion refer to ACI 318-14 Sections 8.4.4.2.2 and 8.4.2.3 (ACI 318-11 sections 11.11.7.1 and 13.5.3).

Total Stress

The Total Stress is the direct shear stress plus the shear stress due to moment. This calculation uses the geometry of the punching shear failure cone along with the induced shear and moment forces to come up with a maximum stress. This will always occur at one of the failure cone corner locations. For an example calculation see the ACI 318-14 Section R8.4.4.2.3 (ACI 318-11 Section R11.11.7.2).

Note:

- See the Punching Shear Loading section for more information.

*Phi*Vny*

This represents the shear ***stress*** capacity of the slab calculated based on the three ACI equations for two way or punching shear capacity of the slab. Refer to ACI 318-14 Section 22.6 (ACI 318-11 Section 11.11.7.2) for more information. When ACI 318-19 is selected, the shear strength of concrete (Vc) uses equations in Table 22.6.5.2. Note that an additional size effect factor λs is added in the equation, λs is a function of effective depth of the slab d, and is calculated per Equation (22.5.5.1.3).

Note:

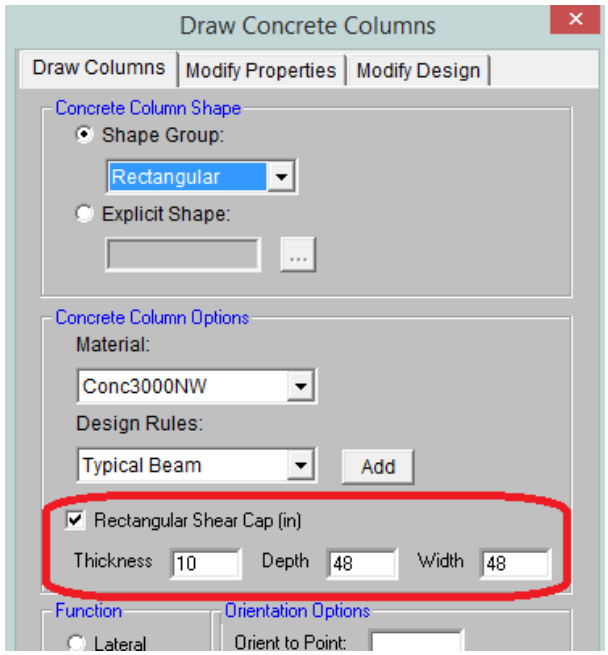
- This value does NOT include any additional capacity due to shear reinforcing in the slab.

Minimum Slab Reinforcement at Punching Shear Critical Section

ACI 318-19 requires a minimum slab reinforcement at punching shear critical section if vuv>φ2λφ√f'c per Section 8.6.1.2. The As,min is calculated and presented in the punching shear result of columns under this condition. Please note that the reinforcement design in elevated slabs does NOT consider this requirement, users need to consider separately the required additional slab reinforcement at punching shear critical section to account for this code requirement. RISA provides the calculations of As,min per Equation (8.6.1.2) in the Detail Report. Please refer to Elevated Slabs - Punching Shear Results for additional information.

Shear Cap Considerations

In RISAFloor ES there is the ability to add a shear cap, a thickened slab portion around the column to help with punching shear. A shear cap can be drawn at the same time as drawing a column:

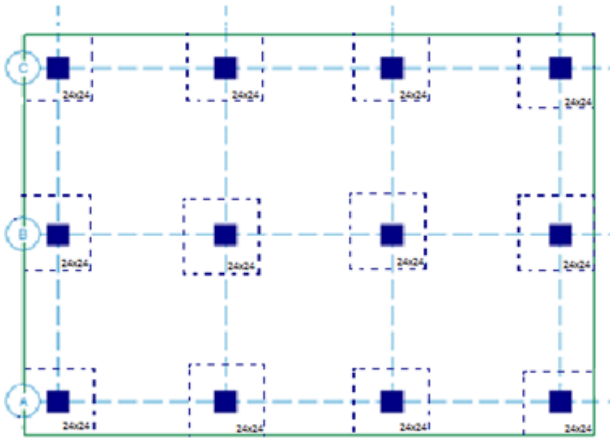


It can also be added graphically using the **Modify Properties** tab of the **Draw Columns** dialog, or it can be added via the **Shear Caps** tab of the **Columns** spreadsheet.

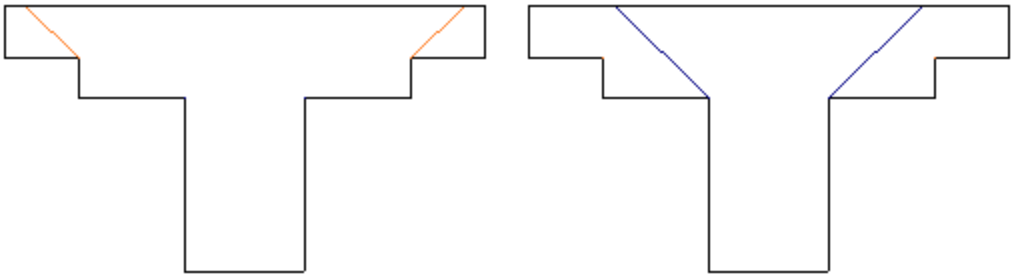
Note:

- See the Shear Caps Spreadsheet information for more information.
- Shear caps will alter the analysis by using a thicker plate in the plate mesh for the cap. See the Elevated Slabs - Analysis topic for more information on this.
- If the shear cap falls off the edge of the slab it will be clipped at the slab edge.

When a shear cap is applied to a column dashed lines will show up in the model view to indicate the shear caps.

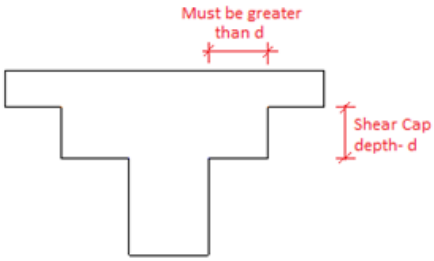


When a shear cap is present at a column location the calculations are very similar. However, two checks are done, one for the slab depth punching shear and one for the shear cap depth punching shear.



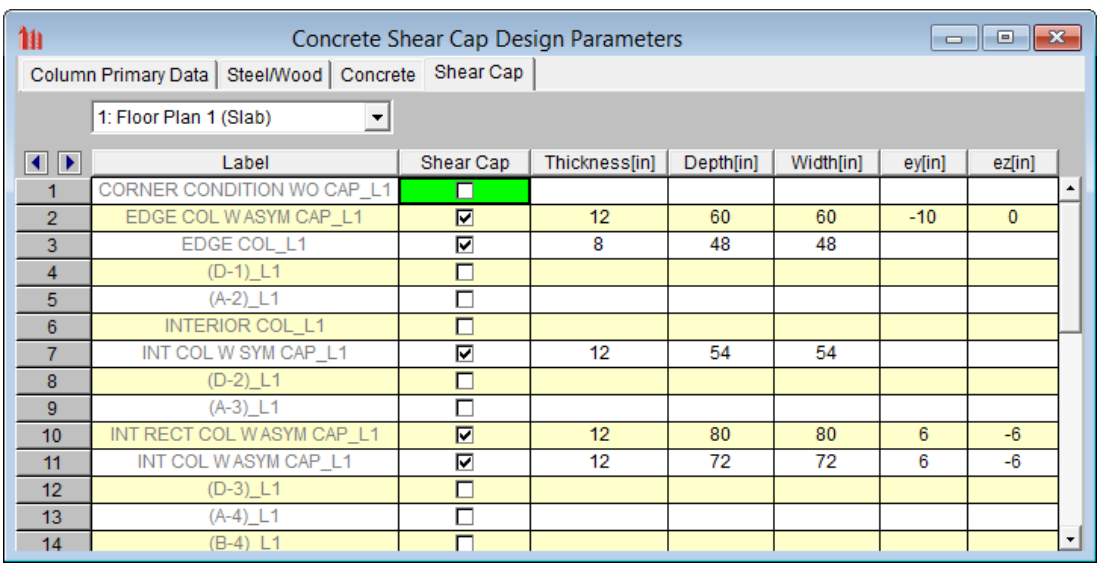
Note:

- If the shear cap does not meet ACI 318-14 Section 8.2.5 (ACI 318-11 Section 13.2.6) criteria (distance of shear cap from face of column) then a warning log message will be given and the shear cap will be ignored completely. A punching shear cap will be done for the column based only on the slab.



Shear Caps Spreadsheet

The **Shear Caps** spreadsheet gives a tabular view of all shear caps defined on each slab floor in the model.



	Label	Shear Cap	Thickness[in]	Depth[in]	Width[in]	ey[in]	ez[in]
1	CORNER CONDITION WO CAP_L1	☐					
2	EDGE COL W ASYM CAP_L1	☑	12	60	60	-10	0
3	EDGE COL_L1	☑	8	48	48		
4	(D-1)_L1	☐					
5	(A-2)_L1	☐					
6	INTERIOR COL_L1	☐					
7	INT COL W SYM CAP_L1	☑	12	54	54		
8	(D-2)_L1	☐					
9	(A-3)_L1	☐					
10	INT RECT COL W ASYM CAP_L1	☑	12	80	80	6	-6
11	INT COL W ASYM CAP_L1	☑	12	72	72	6	-6
12	(D-3)_L1	☐					
13	(A-4)_L1	☐					
14	(B-4)_L1	☐					

Label

This is the column label at the specific lift.

Shear Cap

This allows you to toggle on or off a shear cap at a specific column at a specific floor.

Thickness

This is the thickness of the shear cap, which is the additional thickness beyond the regular slab thickness.

Depth/Width

These are the plan dimensions of the shear cap and are based on the local axes orientation of the column. The depth is the dimension in the local y axis of the column and the width is the dimension in the local z axis.

ey/ez

These are the eccentricities of the shear cap from the column. A positive value indicates a positive eccentricity along the local axes of the column and vice versa for a negative eccentricity. See the eccentricity section for more information.

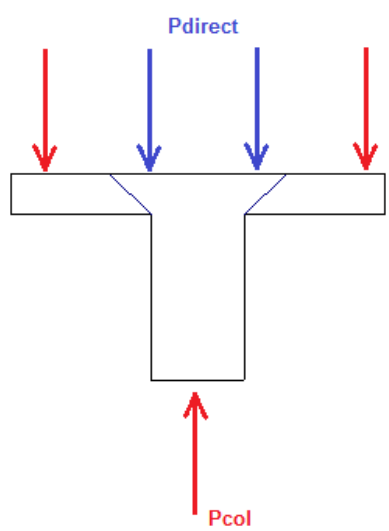
Punching Shear Loading

Punching shear loading includes a direct shear component, as well as biaxial moments. It is these combined forces that are used to determine the punching shear stress and code check.

Direct Punching Shear

The direct punching shear demand is not exactly just the axial force in the Column. The area within the punching shear perimeter loading is assumed to pass directly into the support, so that force must be discounted.

This includes applied point loads, line loads, area loads, slab self-weight, and column loads from above that occur within the punching shear perimeter.

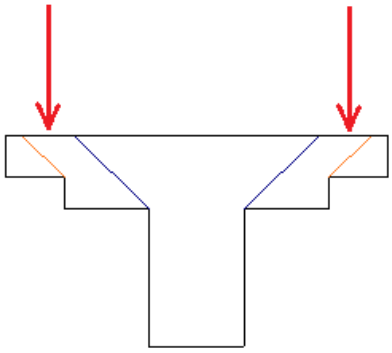


For the image above the Vu demand equals P_{col} - P_{direct}

Note:

- The program is removing the equivalent nodal loads from nodes inside the punching shear perimeter. Because the slab mesh does not create a line constraint along the punching shear perimeter these equivalent nodal loads may not be exactly what you would find in a hand calculation. In general these differences should be very minimal.

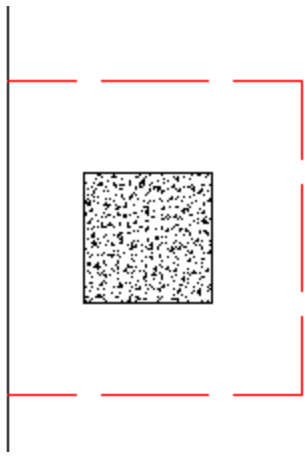
When there is a shear cap the program uses the exact same Vu for both the slab and shear cap punching shear checks. This is a conservative assumption. It is possible that a load will occur between the slab punching shear perimeter and the cap punching shear perimeter, but the program conservatively assumes this load goes into both punching shear calculations.



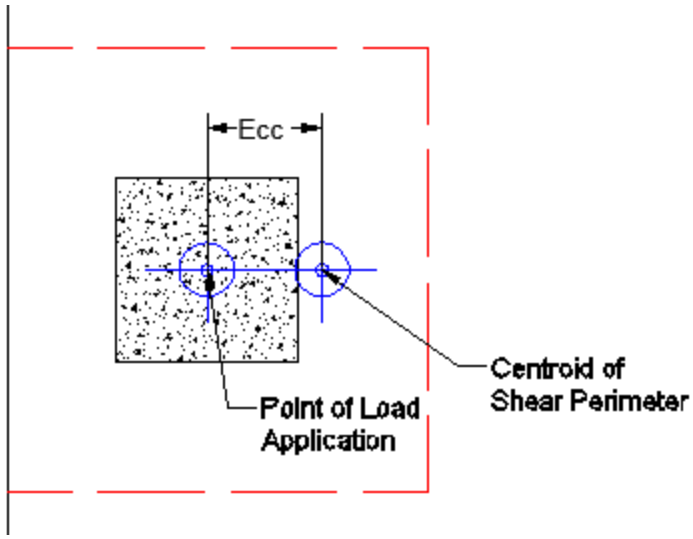
Moment Consideration

The moments used are also not necessarily just the moments directly from the Column. If the centroid of punching shear area is not the same as the centerline of the Column, then there will be an eccentricity that also adds/subtracts moment.

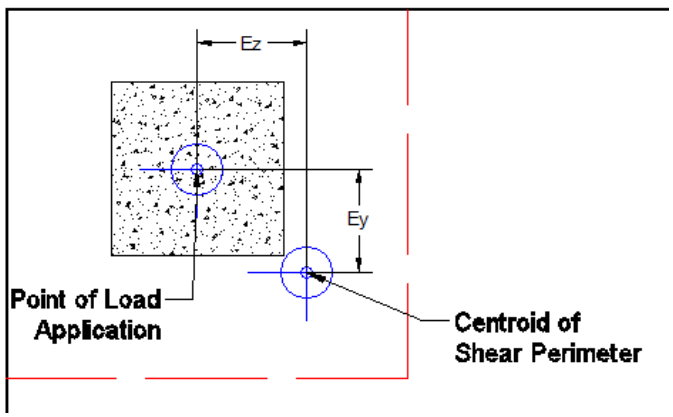
Take the edge punching shear situation shown below:



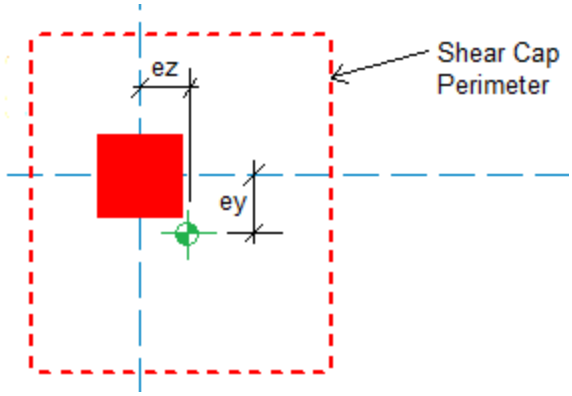
The punching shear perimeter is to the right of the center of Column location. This causes a moment in the punching shear calculation even if a moment is not explicitly applied on the Column. This eccentricity is automatically accounted for in the punching shear stress calculation.



This same inherent eccentricity occurs in corner punching shear conditions. In this case, however, you get an induced moment in both local axes directions because of the centroid of shear perimeter location. This, similar to the edge condition, is automatically considered in the calculation of the maximum shear stress.



When using shear caps they can be drawn asymmetrically from the center of the column. If this is done then there will automatically be an eccentric moment created. This moment will automatically be accounted for when determining the maximum punching shear stress.



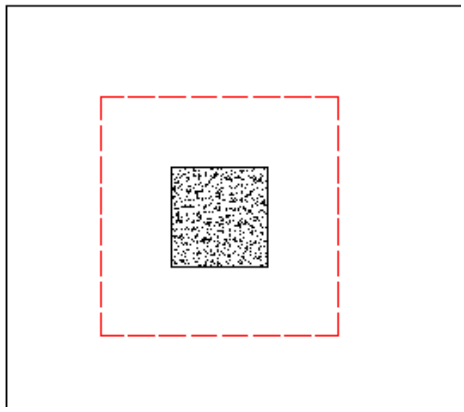
Note:

- The moments due to eccentricity are not accounted for in the detail report moment output. The moments shown there are taken directly from the Column.

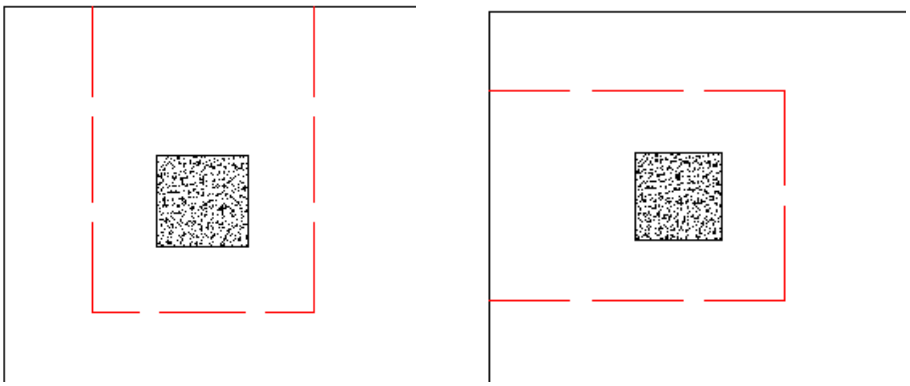
Finding the Controlling Case (Interior, Edge or Corner)

The program considers as many as nine different punching shear perimeters for each column in the model. different scenarios. We look at the interior, four edges, and four corner cases and determine which configuration produces the largest code check and present that in the results.

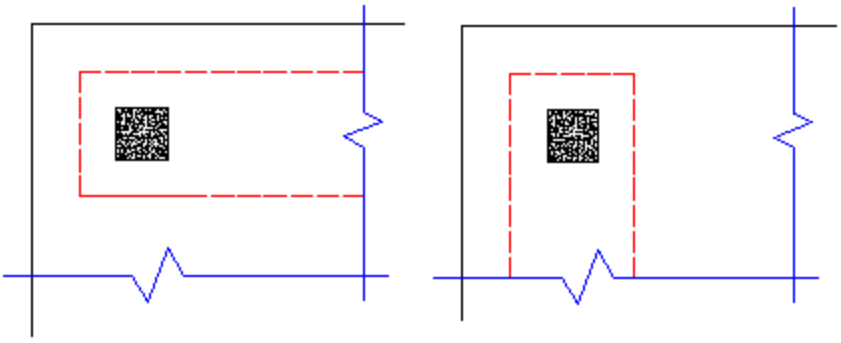
For example, let's consider a Column that has an interior punching shear perimeter close to a corner.



Here the program will also check out any EDGE punching shear perimeter considerations. The two below would be the most likely ones for this given geometry.

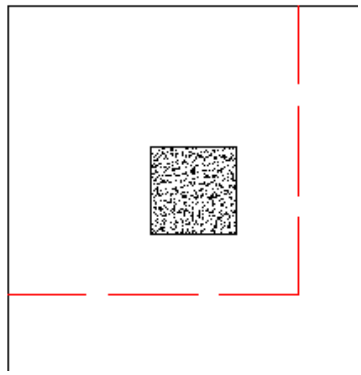


However, these other two edge conditions will also be checked:

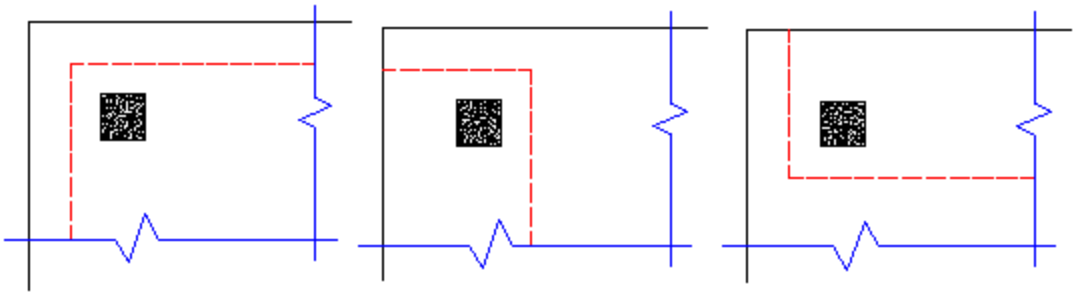


In most cases these odd situations will not govern. However, large moments in a given direction can cause punching shear perimeters with larger areas to govern in specific cases.

The program will also check out any CORNER punching shear considerations. The one below would be the most likely for this given geometry.



However, these other three corner conditions will also be checked:



In most cases these odd situations will not govern. However, large moments in a given direction can cause punching shear perimeters with larger areas to govern in specific cases.

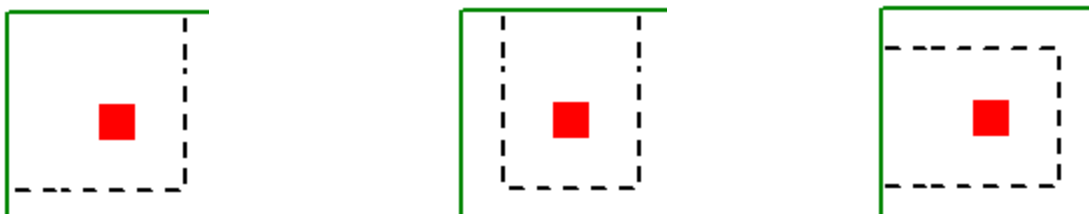
Whichever configuration produces a maximum code check will be reported in the punching shear results.

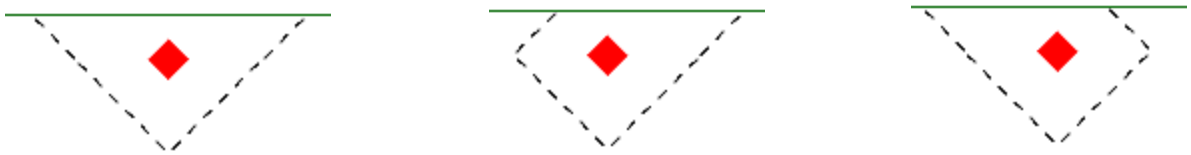
Note:

- The punching shear perimeter lengths shown pictorially above will be shown as the L1 and L2 value for punching shear calculations.
- To limit the total punching shear checks for a given column we have added two caveats to the above behavior. For any of the four Edge or four Corner cases, the program will NOT perform a punching shear check if:
    1. The punching shear perimeter of any punching shear case is more than 50% larger than the minimum Edge/Corner punching shear perimeter.
    2. The punching shear perimeter of a punching shear case encompasses a column within it.

Corner vs Edge Controlling Cases

When the program tests all different cases to determine whether a corner or edge situation governs, it bases the controlling case on the maximum code check produced. There are cases where an edge case is governing that by inspection you might think it would be a corner case. See the images below:





Corner Case Edge Case 1 Edge Case 2

In these two punching shear scenarios shown above, by inspection you would likely expect the corner condition to control over the edge ones. Punching shear capacity is based on a minimum of three equations in the ACI code. One of these equations has an alpha term in it, where alpha is smallest for a corner condition. If this equation governs, this would guarantee the corner condition would have the smallest capacity. However, if this equation does not govern then these capacities may not be as different as you might expect.

Also, the demand portion of the equation will be different based on the geometry of the shear perimeter. It is possible that the edge cases produce a larger maximum stress than the corner cases.

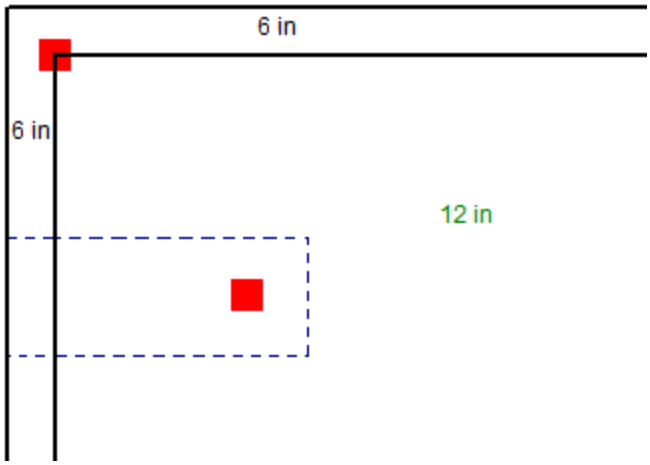
In summary, our testing has shown that these non-intuitive scenarios where an edge controls over a corner have occurred in some instances.

Other Punching Shear Considerations/Limitations

Punching Shear Perimeters That Cross Different Slab Thicknesses

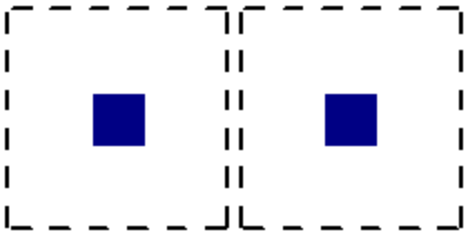
It is possible to have different slab thicknesses within a single column/pedestal's punching shear perimeter. If there is a different thickness at any point along the punching shear perimeter, the program will use the thinnest thickness for the entire calculation. When there is a different thickness, there will be a warning message in the Detail Report. The L1 and L2 values, for use in calculating the b_o, will use the depth at the location of the pedestal/column.

A slab thickness can be changed using the thickened slab tool or drop panels. The slab can be thickened anywhere and in this case the edge of the slab is thinner than the center. Since the punching shear perimeter will be checked for all cases including edge cases, in this example the slab will be assumed to be 6" because it crosses both a 12" section and a 6" slab section. The thinnest thickness will be used and it may govern.

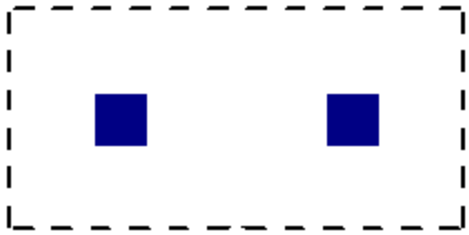


Column Groups

It may be possible for multiple, closely spaced Column to create a punching shear perimeter around the GROUP of piers at a lower force level than would be observed by investigating any one of the individual Column. This is NOT currently considered in the punching shear code checks.



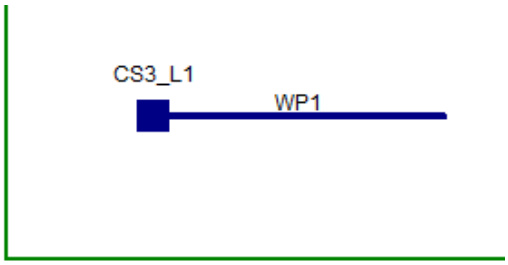
Individual Punching Shear Perimeter



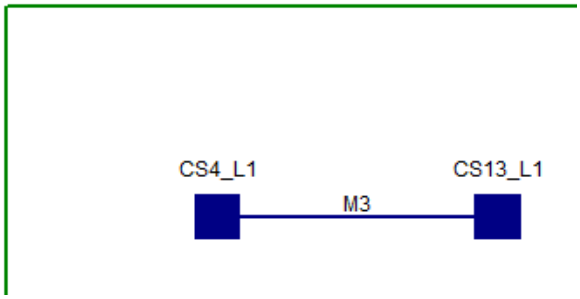
Group Punching Shear Perimeter

Beams, Columns and Walls intersecting the influence area of a Column

When a beam or wall panel is co-linear with the Column joint, the program will not provide a punching shear check on the Column. In these cases a code check of NC or "no calc" will be listed.



Wall Co-Linear with a Column



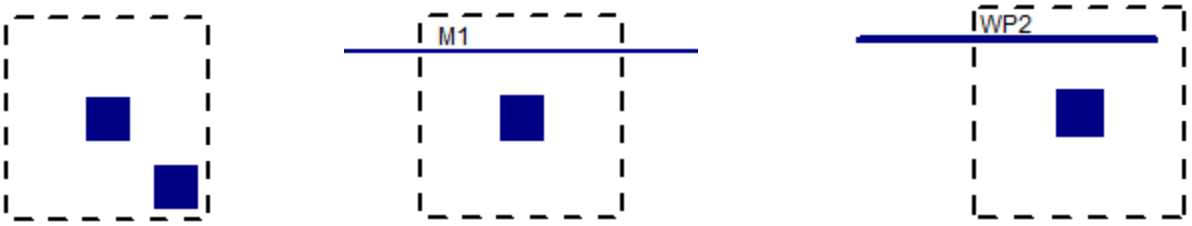
Beam Co-Linear with a Column

Note:

- This would also refer to situations where a column from a floor above lands on either a beam or a wall. In that case punching shear checks would be omitted as well.

When a beam, wall or column isn't co-linear with the column but is located within the punching shear perimeter then punching shear checks will still be performed. However, the calculations will not be influenced by the presence of that wall, column or beam.

If a beam, column or wall lands inside of or crosses a shear cap we will give code checks, but again the calculations will not be influenced by these elements.

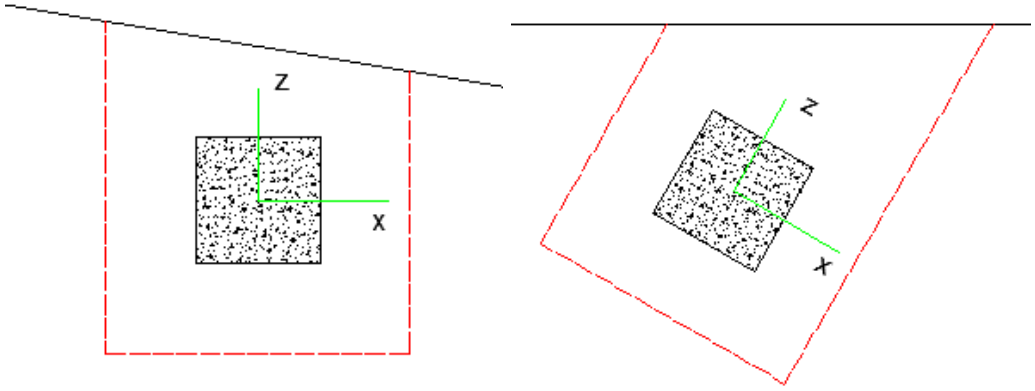


When the Column Local Axes Do Not Correspond to the Slab Geometry

The punching shear perimeters are always investigated with respect to the Column local axes. As shown below, there may be cases where these local axes do not correspond to the slab edge or corner geometry adjacent to that Column. In cases like these it is possible for this to result in an artificially large assumed punching shear perimeter.

Edge Conditions

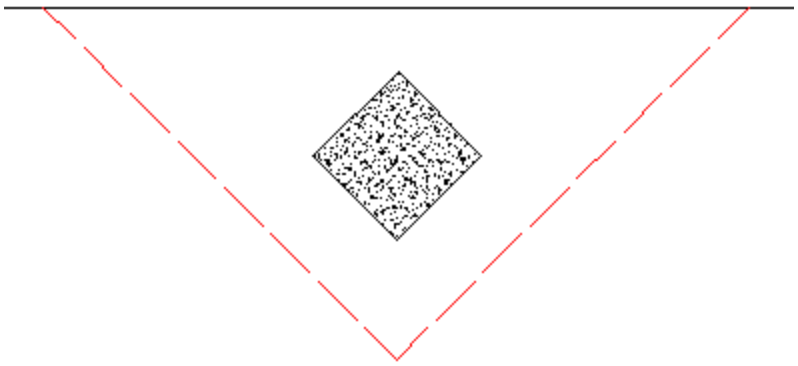
If either the slab edge is sloping or a column local axis is rotated it is possible to get edge shear perimeters as shown below:



In these cases the program will calculate the Polar Moment of Inertia by summing up the three individual sides to get a total value.

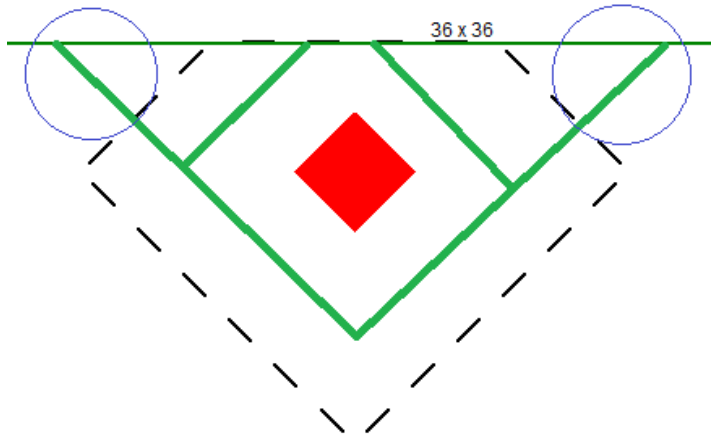
Corner Conditions

If either the slab edge is sloping or a column local axis is rotated it is possible to get a corner shear perimeter. The program will start by trying to draw the full shear perimeter. When this happens in this case we find that the shear perimeter has two parts of it that run into a slab edge. When this occurs the program considers this a corner. In this case the program will use the side dimensions shown below in red to determine the L1 and L2 values.



Special Case With a Skewed Shear Cap

Note in the image below a case where when checking the punching shear on a shear cap the program may still have edge and corner conditions to consider. When the rectangular perimeter of the shear cap hits the slab edge it will be clipped back to the slab edge as shown.

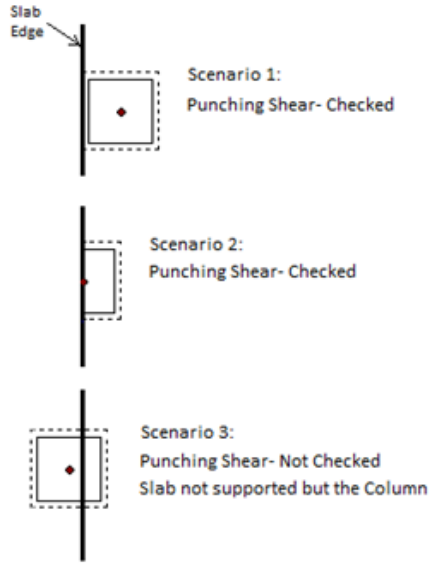


In green you can see what the punching shear perimeter looks like for interior, edge and corner cases. Because the interior case falls off of the edge this case automatically gets discarded and a corner or edge failure must occur. However, in the case above both the corner and edge cases must punch through the shear cap AND a portion of the regular thickness slab.

The program calculates these shear perimeters appropriately but always uses a consistent depth that is the depth of the cap. Thus, the program simplifies this calculation.

Note:

- Columns are defined as a single node in the model. If this node lands inside of the slab then a punching shear check will be performed. If the single node does NOT land on the slab then the Column will not be considered attached to the slab and no punching shear check will be done.



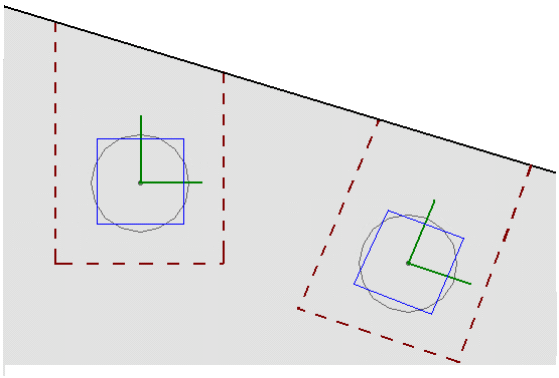
Using Equivalent Square Column

When calculating the punching shear perimeter for a round Column, the program always converts the dimension into an equivalent square of equal perimeter. This greatly simplifies the calculation of punching perimeters when close to an edge or corner.

For an example of this let's use a 12" diameter. The perimeter = Pi*Diameter = 37.7 in.

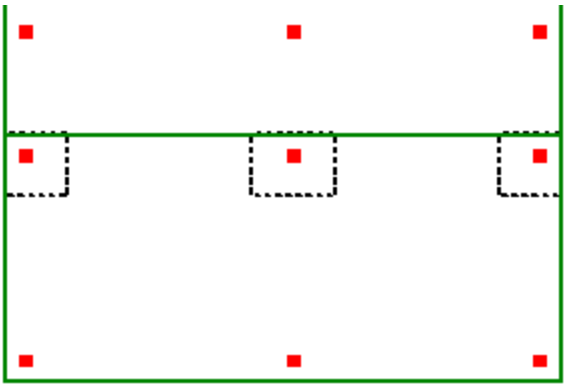
An equivalent square would have a side dimension = 37.7 in/4 = 9.424"

Thus, a 12" diameter round Column would be considered a 9.424" x 9.424" square for use in punching shear checks.



Punching Shear at Slab Intersections

The program will not consider an adjacent slab when performing a punching shear check. Only the slab that the column is drawn on will be consider and the slab edge will be considered free for punching shear checks.



Punching Shear Reinforcement

The program currently does not allow for shear reinforcement. The only capacity considered for punching shear is the concrete itself. ACI 318-14 sections 22.6.6 (shear reinforcement), 22.6.9 (shearheads) and 22.6.8 (headed shear stud reinforcement) are not supported. ACI 318-11 sections 11.11.3 (shear reinforcement), 11.11.4 (shearheads) and 11.11.5 (headed shear stud reinforcement) are not supported.

Punching Shear Checks for Walls, Point Loads, and non-Concrete Columns

It is possible that punching shear failures can occur around walls (especially walls with a short length). It is also possible that punching shear failures can occur around point loads or non-concrete columns bearing on the slab. RISAFloor ES does not currently check for punching shear for these situations.

Elevated Slabs - Punching Shear Results

The Punching Shear spreadsheets reports the two way/punching shear code checks for the defined Column as described in the following sections.

Note:

- For detailed information on punching shear calculations see the Punching Shear - Design topic.

Punching Shear Values

Punching Shear Values

Punching Shear Values | Punching Shear Geometries

1: Floor Plan 1 (Slab)

	Label	Failure Plane	UC	Gov LC	Location	Vux[k]	Muz[k-ft]	Muy[k-ft]	Total Stress[ksi]	Phi*Vnx[ksi]
1	CORNER CONDITION W	Slab	.619	4	Corner +y -z	45.257	-96.679	-71.894	.102	.164
2	EDGE COL W ASYM CAP	Slab	.919	4	Edge -z	91.97	27.239	-195.101	.151	.164
3		Shear Cap	NC	NC	NC	NC	NC	NC	NC	NC
4	EDGE COL_L1	Slab	.792	4	Corner -y -z	77.248	-218.708	-244.521	.091	.115
5		Shear Cap	.393	4	Corner -y -z	77.248	-206.474	-203.426	.06	.152
6	(D-1)_L1	Slab	.629	4	Corner -y -z	46.484	97.163	-71.754	.103	.164
7	(A-2)_L1	Slab	.859	4	Corner +y -z	74.812	-161.071	171.395	.107	.125
8	INTERIOR COL_L1	Slab	.584	4	Interior	116.651	26.42	14.923	.096	.164
9	INT COL W SYM CAP_L	Slab	.41	4	Interior	137.721	-51.01	40.013	.06	.147
10		Shear Cap	.244	4	Interior	137.721	-51.01	40.013	.04	.164
11	(D-2)_L1	Slab	.566	4	Corner -y +z	71.315	96.789	.875	.076	.135
12	(A-3)_L1	Slab	.618	4	Edge +y	70.216	-95.746	-8.353	.102	.164

This section reports the basic / summary data for the punching shear code check.

Failure Plane

This gives the designation between the slab punching shear check and the shear cap. If there is a shear cap present then the punching shear check will always show both failure planes. If there is no shear cap then you will only have a single slab shear check.

UC (Unity Check) and Governing LC

The **UC** value is the governing code check (Total Stress /phi*Vnx) for the pedestal punching shear. The **Gov LC** value is the Load combination that resulted in this maximum code check.

Location

The pedestal and slab geometry is investigated and checked for all three types of punching failure (Interior, Edge and Corner). This field represents the type of failure method that controlled the code check calculation for the given pedestal. For a Corner or Edge case the program will also give the direction of the punching shear failure based on the local axes of the column. For more information on how this works, see the Punching Shear - Design topic.

Vux

This value represents the factored load transferred from the pedestal into the slab for the governing load combination. This corresponds to the axial force in the Column.

Muz and Muy

These values represent the factored moments about the local axes of the Column.

Total Stress

The Total Stress is the direct shear stress plus the shear stress due to moment. For more information, see the Punching Shear - Design topic.

*Phi*Vny*

This represents the shear ***stress*** capacity of the slab calculated based on the three ACI equations for two way or punching shear capacity of the slab. Refer to ACI 318-14 Section 22.6.5 (ACI 318-11 Section 11.11.7.2) for more information. When ACI 318-19 is selected, the shear strength of concrete (Vc) uses equations in Table 22.6.5.2. Note that an additional size effect factor λs is added in the equation, λs is a function of effective depth of the slab d, and is calculated per Equation (22.5.5.1.3).

Note:

- This value does NOT include any additional capacity due to shear reinforcing in the slab.
- For the CSA A23.3-04 code provision 13.3.4.3 is used.
- For the Eurocode2 : BS EN 1992-1-1:2004+A1:2014 code section 6.4.4 is used.

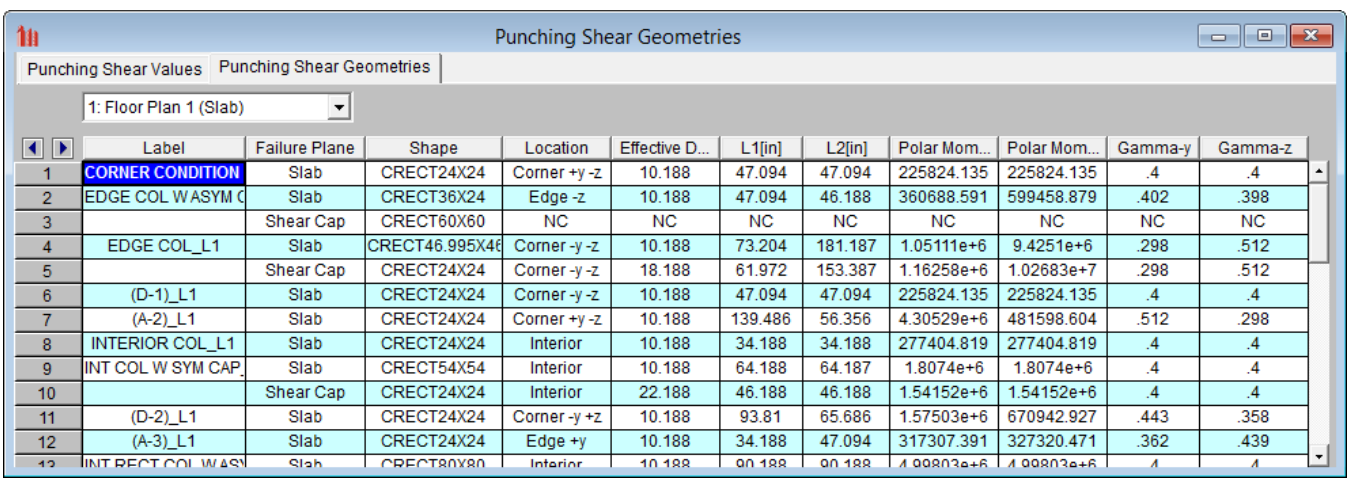
Minimum Slab Reinforcement at Punching Shear Critical Section

ACI 318-19 requires a minimum slab reinforcement at punching shear critical section if vuv>φ2λφ√f'c per Section 8.6.1.2. The As,min is calculated and presented in the punching shear result of columns under this condition. Please note that the reinforcement design in elevated slabs does not consider this requirement, users need to adjust the slab reinforcement at punching shear critical section to account for this code requirement. RISA provides the calculations of As,min per Equation (8.6.1.2) in the Detail Report for columns. Please note that the As,min calculated is based on area of steel per foot, i.e., bslab was taken as 12". The calculated As,min shall be multiplied by bslab (specified in 8.4.2.2.3) to obtain the total area of steel required.

Minimum Slab Reinforcement Requirements (per foot) provided over the effective slab width (bslab) per ACI318-19 Sec8.6.1.2

v_{uv}	0.204 ksi	alpha	40
lambda_s	0.977	bo	67.75 in
As_min	0.461 in^2/ft		

Punching Shear Geometries


Punching Shear Geometries

Punching Shear Values | Punching Shear Geometries

1: Floor Plan 1 (Slab)

	Label	Failure Plane	Shape	Location	Effective D...	L1[in]	L2[in]	Polar Mom...	Polar Mom...	Gamma-y	Gamma-z
1	CORNER CONDITION	Slab	CRECT24X24	Corner +y -z	10.188	47.094	47.094	225824.135	225824.135	.4	.4
2	EDGE COL W ASYM C	Slab	CRECT36X24	Edge -z	10.188	47.094	46.188	360688.591	599458.879	.402	.398
3		Shear Cap	CRECT60X60	NC	NC	NC	NC	NC	NC	NC	NC
4	EDGE COL_L1	Slab	CRECT46.995X46	Corner -y -z	10.188	73.204	181.187	1.05111e+6	9.4251e+6	.298	.512
5		Shear Cap	CRECT24X24	Corner -y -z	18.188	61.972	153.387	1.16258e+6	1.02683e+7	.298	.512
6	(D-1)_L1	Slab	CRECT24X24	Corner -y -z	10.188	47.094	47.094	225824.135	225824.135	.4	.4
7	(A-2)_L1	Slab	CRECT24X24	Corner +y -z	10.188	139.486	56.356	4.30529e+6	481598.604	.512	.298
8	INTERIOR COL_L1	Slab	CRECT24X24	Interior	10.188	34.188	34.188	277404.819	277404.819	.4	.4
9	INT COL W SYM CAP_	Slab	CRECT54X54	Interior	10.188	64.188	64.187	1.8074e+6	1.8074e+6	.4	.4
10		Shear Cap	CRECT24X24	Interior	22.188	46.188	46.188	1.54152e+6	1.54152e+6	.4	.4
11	(D-2)_L1	Slab	CRECT24X24	Corner -y +z	10.188	93.81	65.686	1.57503e+6	670942.927	.443	.358
12	(A-3)_L1	Slab	CRECT24X24	Edge +y	10.188	34.188	47.094	317307.391	327320.471	.362	.439
13	INT RECT COL W ASY	Slab	CRECT80X80	Interior	10.188	90.188	90.188	4.99803e+6	4.99803e+6	.4	.4

Note:

- For detailed information on many of these values see the Punching Shear - Design topic.

Location

The pedestal and slab geometry is investigated and checked for all three types of punching failure (Interior, Edge and Corner). This field represents the type of failure method that controlled the code check calculation for the given pedestal. For a Corner or Edge case the program will also give the direction of the punching shear failure based on the local axes of the column. For more information on how this works, see the Punching Shear - Design topic.

Effective Depth

The effective depth of the slab is used to define the punching shear perimeter at a distance of d/2 beyond the edge of the pedestal.

L1 and L2

Length of the critical section along the local axes.

Note:

- For a skewed Edge condition L1 or L2 may have two values. See the Punching Shear - Design topic for more information.

Polar Moment - xx and Polar Moment - zz

This is a property (Jc) of the critical section which is analogous to polar moment of inertia about the longitudinal column/pedestal axes.

Gamma x and Gamma z

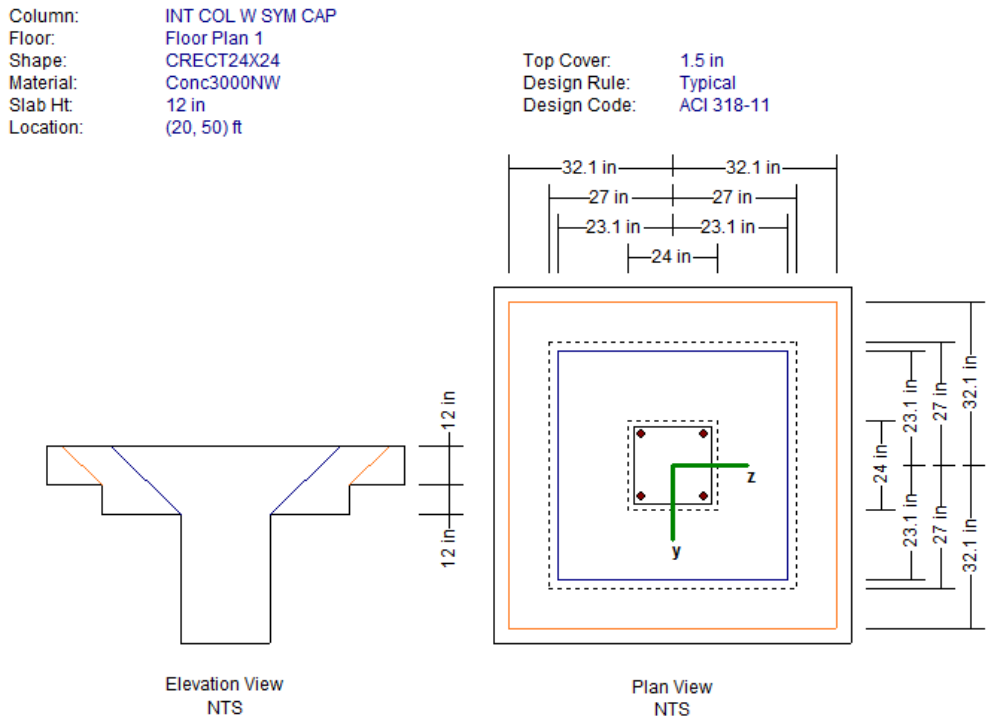
This is calculated from the dimensions of the assumed punching shear perimeter. It represents the fraction of the unbalanced moment for each axis that is transferred by eccentricity of shear.

Column Detail Report

Note:

- - This 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.

Input Echo/Graphic View



The input echo lists off the column label, floor, column shape, slab material, slab thickness, column location, cover, design rule and design code. The graphic view shows the dimensions used in the calculations. The graphics shown above shows the punching shear information when a shear cap is present. If there is no shear cap (or the dimensions of the cap violate the ACI code requirements and the cap is ignored) then the diagram will be much simpler.

Note:

- The plan view diagram is not to scale, however the dimensions have been updated properly.

Punching Shear Check Results

Punching Shear Check Results

Punching Shear Values							
Unity Check	.352						
Gov LC	4						
Governed by	SLAB						
Punching Shear Geometries - Slab							
Location	Interior						
Effective depth	10.19 in						
L1 along zz	82.2 in	Polar moment Iyy	3.78494e+6 in^4	cz	41.1 in	Gamma yy	.4
L2 along yy	82.2 in	Polar moment Izz	3.78494e+6 in^4	cy	41.1 in	Gamma zz	.4
Total Stress	.047 ksi	Vuy	145.5 k				
ΦVnx	.133 ksi	Muy	42.7 k-ft	Muy Column	-30.1 k-ft		
		Muz	22.1 k-ft	Muz Column	-50.6 k-ft		
Punching Shear Geometries - Shear Cap							
Location	Interior						
Effective depth	22.19 in						
L1 along zz	46.2 in	Polar moment Iyy	1.54152e+6 in^4	cz	23.1 in	Gamma yy	.4
L2 along yy	46.2 in	Polar moment Izz	1.54152e+6 in^4	cy	23.1 in	Gamma zz	.4
Total Stress	.041 ksi	Vuy	145.5 k				
ΦVnx	.164 ksi	Muy	-30.1 k-ft	Muy Column	-30.1 k-ft		
		Muz	-50.6 k-ft	Muz Column	-50.6 k-ft		

The **Unity Check** gives a ratio of demand to capacity, while the **Gov LC** gives the load combination that produced this maximum unity check. If there is both a shear cap and slab check the worst case of the two will be reported. The results shown above occur if a shear cap is present. If no shear cap is present then anything relating to the shear cap calculations will be removed.

For an explanation of the rest of these values see both the spreadsheet result explanations above, as well as the Punching Shear - Design topic.

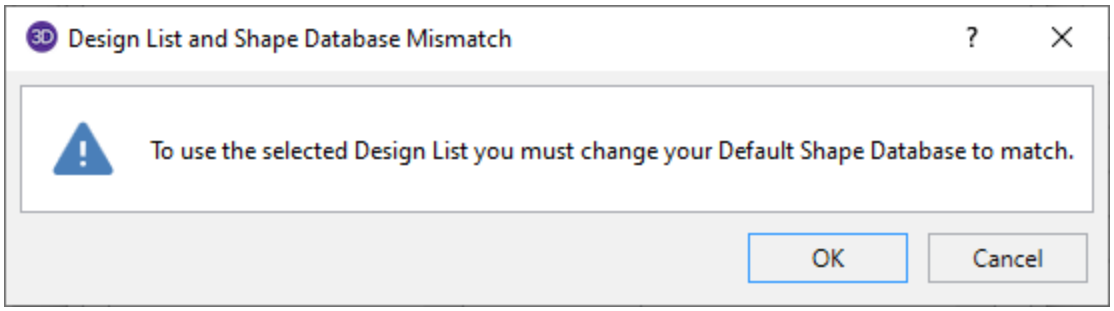
Appendix A – Redesign Lists

RISAFloor 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**.

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 Documents\RISA folder. However, this folder can be changed by selecting Tools - Preference - File Locations from the main menu toolbar.

File Naming Convention for the Design Lists

The design lists must be ASCII text files with a file extension of ASC. In addition, the first 3 characters of the Design indicate which Region the lists will be available for. The region can be changed by selecting Tools - Prefereces - General from the Main Menu Toolbar. Each region 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 may view these files using any text editor such as Window's Notepad. However, this file must be saved in the risa__redesign_lists folder as an ASCII text file with a file extension ASC. If the file is saved with a txt extension, it will not be read intoRISAFloor. 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]

Creating or Modifying Design List

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 Beams or Columns spreadsheet Shape pull down box (See Beam Primary Data o rColumn Primary Data 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 B – Error Messages

Extensive error checking is performed during data entry and model solution in order to prevent you from having to scan input files for offending data. If an error or questionable entry is detected, an error or warning message will be displayed. Errors are divided into two groups such that data entry error numbers are between 1000-1999 and solution errors are between 2000-2999.

*Please refer to the **Help File** in RISAFloor for detailed information about each error message. The Help File may be accessed by selecting **Help Topics** from the **Help Menu**. Choose **Appendices : Appendix B - Error Messages**.*

Appendix D – File Format

RISAFloor uses a plain text, keyword driven input file. The input data is delimited by sets of keyword labels that divide the input into a number of segments containing semantically related data.

Each input segment may include one or more input records. The input records are text strings that define sets of properties for single input items, e.g., label, coordinates, etc. for nodes. Each record is terminated with a semicolon (;), except for a few project description records (between the [.PROJECT_DESCRIPTION] and [.END_PROJECT_ DESCRIPTION] labels). Those records are delimited by their corresponding keyword labels.

Multiple records are used to describe multiple instances of input items. The number of records immediately follows the keyword label and is given in the <> brackets.

Lines starting with a // are ignored by the program and may be used for user comments.

The reading of Label fields (such as member names, or shape names) uses a combination of fixed length fields and field delimiter. The field lengths are set in the Label Length Data. Care should be taken to maintain the proper length for all label fields. If the length of the field is not entered correct, the program will attempt to use the field delimiters (") to read the data. A combination of incorrect field length and the use of (") marks within a shape label itself (e.g. a shape label of: 3/4" rod bracing) will result in the program producing an error during the reading of the file.

*Please refer to the **Help File** in RISAFloor for detailed information about the file format. The Help File may be accessed by selecting **Help Topics** from the **Help Menu**.*

Appendix E - Interfacing with Other Programs

RISAFloor 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 RISAFloor and RISASection see Shape Databases.

For information on the integration between RISAFloor and Foundation see RISAFoundation Integration.

For information on the integration between RISAFloor and RISAConnection see RISAConnection Integration

Linking your Autodesk Revit Structure model with RISAFloor

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 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

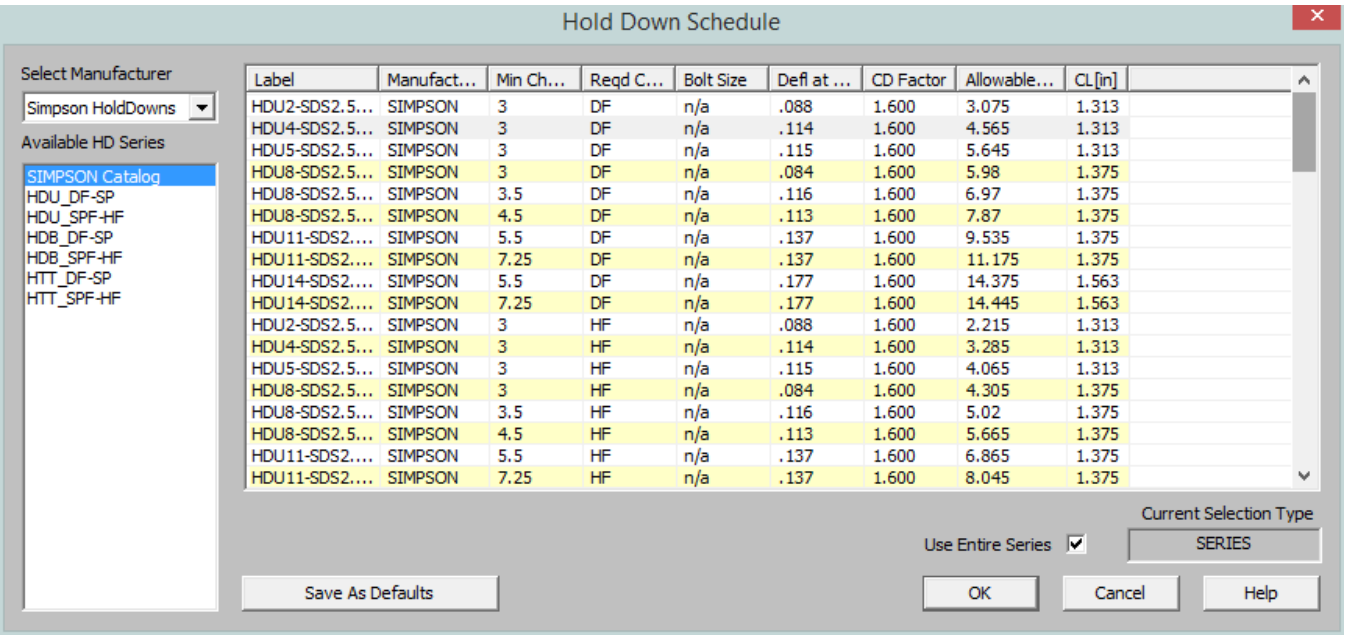
Importing or Exporting DXF Files

For DXF importing / exporting, refer to the DXF Files section of the main manual.

Appendix F – Wood Shear Wall Files

RISAFloor 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



Label	Manufact...	Min Ch...	Reqd C...	Bolt Size	Defl at ...	CD Factor	Allowable...	CL[in]
HDU2-SDS2.5...	SIMPSON	3	DF	n/a	.088	1.600	3.075	1.313
HDU4-SDS2.5...	SIMPSON	3	DF	n/a	.114	1.600	4.565	1.313
HDU5-SDS2.5...	SIMPSON	3	DF	n/a	.115	1.600	5.645	1.313
HDU8-SDS2.5...	SIMPSON	3	DF	n/a	.084	1.600	5.98	1.375
HDU8-SDS2.5...	SIMPSON	3.5	DF	n/a	.116	1.600	6.97	1.375
HDU8-SDS2.5...	SIMPSON	4.5	DF	n/a	.113	1.600	7.87	1.375
HDU11-SDS2....	SIMPSON	5.5	DF	n/a	.137	1.600	9.535	1.375
HDU11-SDS2....	SIMPSON	7.25	DF	n/a	.137	1.600	11.175	1.375
HDU14-SDS2....	SIMPSON	5.5	DF	n/a	.177	1.600	14.375	1.563
HDU14-SDS2....	SIMPSON	7.25	DF	n/a	.177	1.600	14.445	1.563
HDU2-SDS2.5...	SIMPSON	3	HF	n/a	.088	1.600	2.215	1.313
HDU4-SDS2.5...	SIMPSON	3	HF	n/a	.114	1.600	3.285	1.313
HDU5-SDS2.5...	SIMPSON	3	HF	n/a	.115	1.600	4.065	1.313
HDU8-SDS2.5...	SIMPSON	3	HF	n/a	.084	1.600	4.305	1.375
HDU8-SDS2.5...	SIMPSON	3.5	HF	n/a	.116	1.600	5.02	1.375
HDU8-SDS2.5...	SIMPSON	4.5	HF	n/a	.113	1.600	5.665	1.375
HDU11-SDS2....	SIMPSON	5.5	HF	n/a	.137	1.600	6.865	1.375
HDU11-SDS2....	SIMPSON	7.25	HF	n/a	.137	1.600	8.045	1.375

Each database of hold downs is specified by an XML file in the "Hold Downs" sub-directory of the Wood Wall panels directory. The location of this directory is based on the information in the File Locations tab of the Tools - Application Settings dialog.

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.

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.

The **Deflection at Peak Load** entry is used to calculate the deflection of the shear wall per APA / NDS formulas. This deflection is then reported on the shear wall detail report for each wall panel.

The **CD Factor** is the assumed load duration factor that was used as the basis for specifying the listed **Allowable Tension** value for that hold down.

A load combination may be solved with a load duration factor different from the CD Factor described above. When this is the case, the **Allowable Tension** for that hold down will be adjusted based on the difference between the assumed and actual load duration factors.

Full Database - Optional Fields

The following fields are optional. They are not currently used in the design or capacity calculations, but are reported on the detail reports for reference purposes only.

The **Manufacturer** field is an identifier for the hold down. It is provided so that the engineer can more easily identify the callouts for their final design drawings.

The allowable capacity of the hold down will vary based on the Chord Thickness. Therefore, the **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.

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.

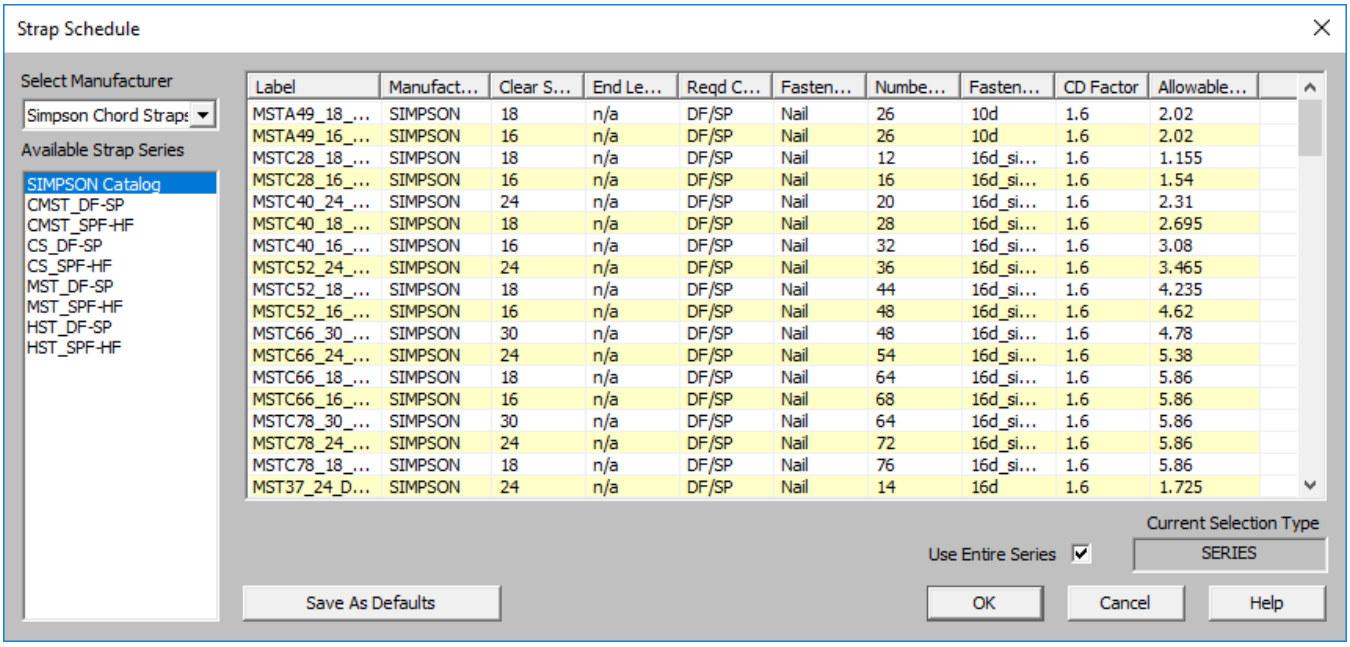
The **Bolt Size** when specified is used to reduce the axial capacity of the hold down chord itself. The only change to the calculation is that the program will perform the allowable tension check on the net area of the chord member rather than the gross area.

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



Label	Manufact...	Clear S...	End Le...	Reqd C...	Fasten...	Numbe...	Fasten...	CD Factor	Allowable...
MSTA49_18_...	SIMPSON	18	n/a	DF/SP	Nail	26	10d	1.6	2.02
MSTA49_16_...	SIMPSON	16	n/a	DF/SP	Nail	26	10d	1.6	2.02
MSTC28_18_...	SIMPSON	18	n/a	DF/SP	Nail	12	16d_si...	1.6	1.155
MSTC28_16_...	SIMPSON	16	n/a	DF/SP	Nail	16	16d_si...	1.6	1.54
MSTC40_24_...	SIMPSON	24	n/a	DF/SP	Nail	20	16d_si...	1.6	2.31
MSTC40_18_...	SIMPSON	18	n/a	DF/SP	Nail	28	16d_si...	1.6	2.695
MSTC40_16_...	SIMPSON	16	n/a	DF/SP	Nail	32	16d_si...	1.6	3.08
MSTC52_24_...	SIMPSON	24	n/a	DF/SP	Nail	36	16d_si...	1.6	3.465
MSTC52_18_...	SIMPSON	18	n/a	DF/SP	Nail	44	16d_si...	1.6	4.235
MSTC52_16_...	SIMPSON	16	n/a	DF/SP	Nail	48	16d_si...	1.6	4.62
MSTC66_30_...	SIMPSON	30	n/a	DF/SP	Nail	48	16d_si...	1.6	4.78
MSTC66_24_...	SIMPSON	24	n/a	DF/SP	Nail	54	16d_si...	1.6	5.38
MSTC66_18_...	SIMPSON	18	n/a	DF/SP	Nail	64	16d_si...	1.6	5.86
MSTC66_16_...	SIMPSON	16	n/a	DF/SP	Nail	68	16d_si...	1.6	5.86
MSTC78_30_...	SIMPSON	30	n/a	DF/SP	Nail	64	16d_si...	1.6	5.86
MSTC78_24_...	SIMPSON	24	n/a	DF/SP	Nail	72	16d_si...	1.6	5.86
MSTC78_18_...	SIMPSON	18	n/a	DF/SP	Nail	76	16d_si...	1.6	5.86
MST37_24_D...	SIMPSON	24	n/a	DF/SP	Nail	14	16d	1.6	1.725

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 Tools - Application Settings dialog.

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.

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.

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 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.

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.

The **Clear Span** is the distance the strap is required to span between elements it is resisting the tension of.

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.

The **Fastener Type** shows whether Nails or Bolts is used to fasten the strap to the tension chords.

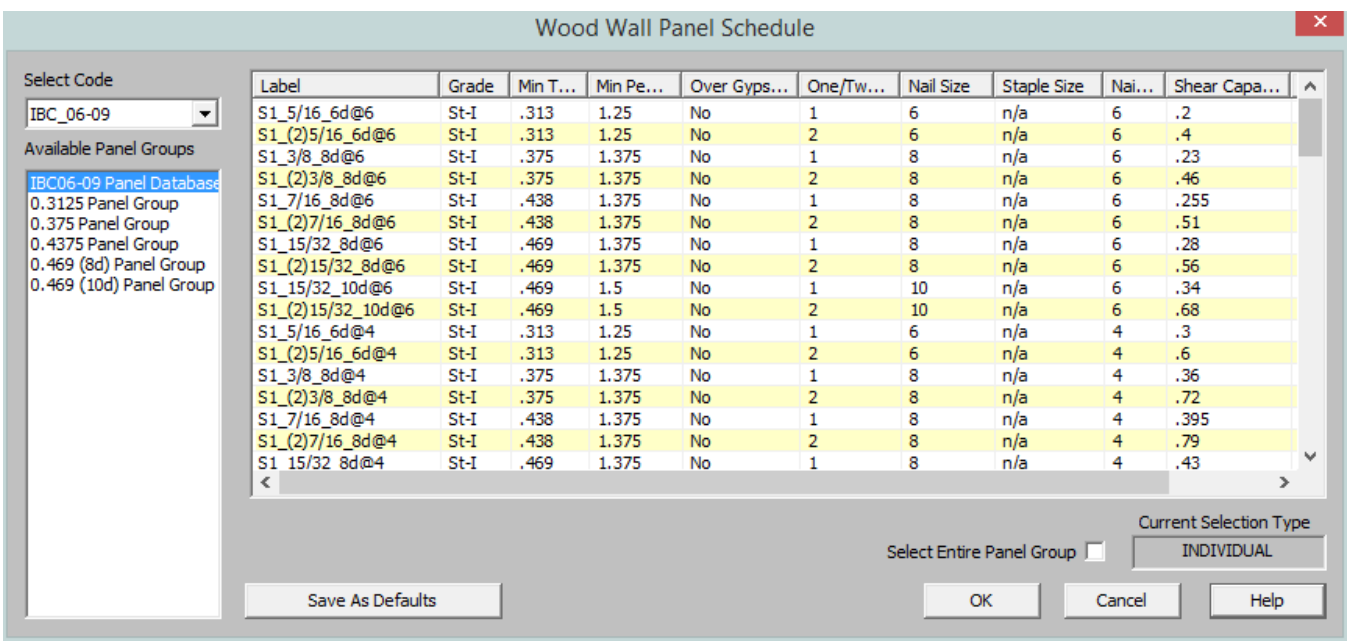
The **Number of Fasteners** field show how many fasteners are required to be used to meet the requirements of the tension capacity given by the strap.

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 thestrap 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 T...	Min Pe...	Over Gyps...	One/Tw...	Nail Size	Staple Size	Nai...	Shear Capa...
S1_5/16_6d@6	St-I	.313	1.25	No	1	6	n/a	6	.2
S1_(2)5/16_6d@6	St-I	.313	1.25	No	2	6	n/a	6	.4
S1_3/8_8d@6	St-I	.375	1.375	No	1	8	n/a	6	.23
S1_(2)3/8_8d@6	St-I	.375	1.375	No	2	8	n/a	6	.46
S1_7/16_8d@6	St-I	.438	1.375	No	1	8	n/a	6	.255
S1_(2)7/16_8d@6	St-I	.438	1.375	No	2	8	n/a	6	.51
S1_15/32_8d@6	St-I	.469	1.375	No	1	8	n/a	6	.28
S1_(2)15/32_8d@6	St-I	.469	1.375	No	2	8	n/a	6	.56
S1_15/32_10d@6	St-I	.469	1.5	No	1	10	n/a	6	.34
S1_(2)15/32_10d@6	St-I	.469	1.5	No	2	10	n/a	6	.68
S1_5/16_6d@4	St-I	.313	1.25	No	1	6	n/a	4	.3
S1_(2)5/16_6d@4	St-I	.313	1.25	No	2	6	n/a	4	.6
S1_3/8_8d@4	St-I	.375	1.375	No	1	8	n/a	4	.36
S1_(2)3/8_8d@4	St-I	.375	1.375	No	2	8	n/a	4	.72
S1_7/16_8d@4	St-I	.438	1.375	No	1	8	n/a	4	.395
S1_(2)7/16_8d@4	St-I	.438	1.375	No	2	8	n/a	4	.79
S1_15/32_8d@4	St-I	.469	1.375	No	1	8	n/a	4	.43

Each database of wall panels is specified by an XML file in the "Shear Panels" sub-directory of the Wood Wall panels directory. This directory is located based on the information File Locations tab of the Tools - Application Settings dialog.

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.

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.

Note:

- Panels with a label containing the characters "_W" together will be ignored during design optimization.

The **Min Panel Thickness** is used during the design optimization to limit the selected panels based on the Design Rules chosen by the user. It is also used to help set the elastic stiffness of the wall panel used during the FEM solution.

The **Ga** value is the **Apparent Shear Stiffness** from nail slip and panel deformation as defined in equation 4.3-1 of the NDS' Special Design Provisions for Wind and Seismic. This value (in combination with the Min Panel Thickness defined above) is used to set the elastic stiffness of the wall panel that will be used during the FEM solution.

Note:

- When a family or group of panels / nailing schedules are assigned to a shear wall, the lowest value of Ga and Min Panel thickness will be used to determine the elastic stiffness of the plate elements in the FEM solution.

The **One/Two Sided** field is used during the design optimization to limit the available panels based on the Design Rules specified by the user.

The **Boundary Nail Spacing** field is used during the design optimization to limit the available panels based on the Design Rules specified by the user.

Note:

- The maximum field spacing is never entered in the program but is generally equal to 12 inches for the nailing schedules defined in the 1997 UBC and 2006 IBC databases. If a different nail spacing is present, then the user should add in a new nailing schedule to the existing database with a user defined shear capacity.

The **Shear Capacity** listed in the spreadsheet is the primary value that controls the code checking of the shear wall. This is the seismic capacity, which the program can automatically increase for wind loads if the Wind ASIF function is enabled.

Full Database - Optional Fields

The following fields are optional. They are not currently used in the design or capacity calculations, but are reported on the detail reports for informational purposes only.

The **Panel Grade** and **Min Penetration** fields are identifiers for the engineer, but are not used in the design calculations. They are provided so that the engineer can most easily identify the panels in their design results and drawings.

The **Panel Applied Over Gypsum** field is also a identifier for the engineer that will not be used in the design calculations.

The **Nail Size** listed in the spreadsheet is intended to refer to the Common nail size, but is reported only for reference purposes and are NOT used in the capacity calculations. If the nail size is changed by the user, then the user should also change the Shear Capacity entry accordingly. Below is a reference table for common, box, and sinker nails.

	Diameter				Length	
Penny Weight	**Common (in)**	**Common (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

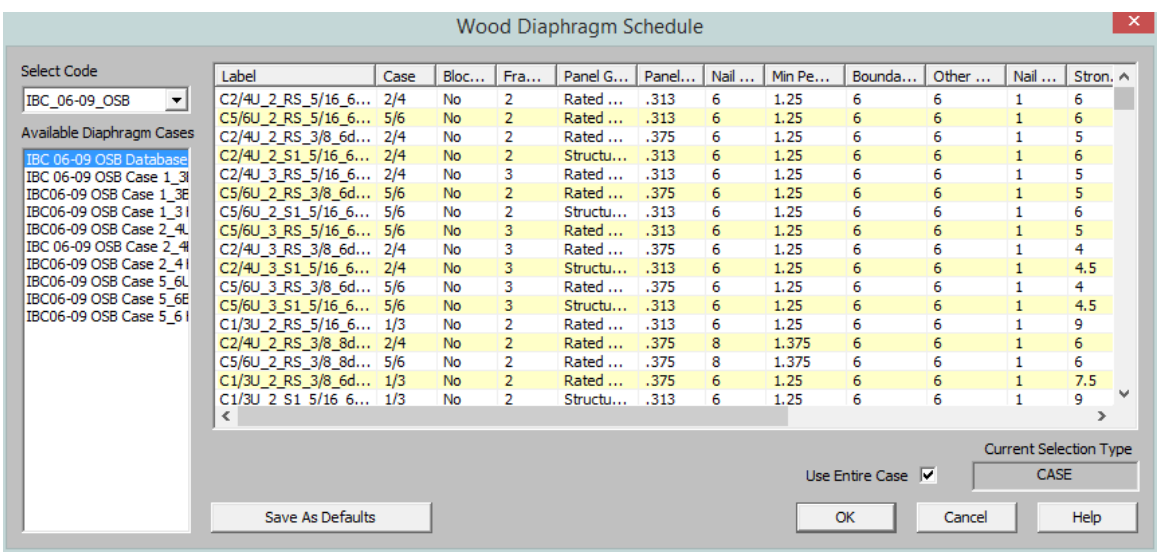
The Staple size listed in the database is reported for reference purposes only. If the staple size is entered or changed by the user, then the user should also change the shear capacity entry to the appropriate value.

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



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 ...	.313	6	1.25	6	6	1	6
C5/6U_2_RS_5/16_6...	5/6	No	2	Rated ...	.313	6	1.25	6	6	1	6
C2/4U_2_RS_3/8_6d...	2/4	No	2	Rated ...	.375	6	1.25	6	6	1	5
C2/4U_2_S1_5/16_6...	2/4	No	2	Structu...	.313	6	1.25	6	6	1	6
C2/4U_3_RS_5/16_6...	2/4	No	3	Rated ...	.313	6	1.25	6	6	1	5
C5/6U_2_RS_3/8_6d...	5/6	No	2	Rated ...	.375	6	1.25	6	6	1	5
C5/6U_2_S1_5/16_6...	5/6	No	2	Structu...	.313	6	1.25	6	6	1	6
C5/6U_3_RS_5/16_6...	5/6	No	3	Rated ...	.313	6	1.25	6	6	1	5
C2/4U_3_RS_3/8_6d...	2/4	No	3	Rated ...	.375	6	1.25	6	6	1	4
C2/4U_3_S1_5/16_6...	2/4	No	3	Structu...	.313	6	1.25	6	6	1	4.5
C5/6U_3_RS_3/8_6d...	5/6	No	3	Rated ...	.375	6	1.25	6	6	1	4
C5/6U_3_S1_5/16_6...	5/6	No	3	Structu...	.313	6	1.25	6	6	1	4.5
C1/3U_2_RS_5/16_6...	1/3	No	2	Rated ...	.313	6	1.25	6	6	1	9
C2/4U_2_RS_3/8_8d...	2/4	No	2	Rated ...	.375	8	1.375	6	6	1	6
C5/6U_2_RS_3/8_8d...	5/6	No	2	Rated ...	.375	8	1.375	6	6	1	6
C1/3U_2_RS_3/8_6d...	1/3	No	2	Rated ...	.375	6	1.25	6	6	1	7.5
C1/3U_2_S1_5/16_6...	1/3	No	2	Structu...	.313	6	1.25	6	6	1	9

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 Tools - Application Settings dialog.

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 RISAFloor and brought into RISA-3D.

The first sheet of the XML file should always be descriptive of the contents of the database (such as **IBC_06-09_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:

Full Database - Required Fields

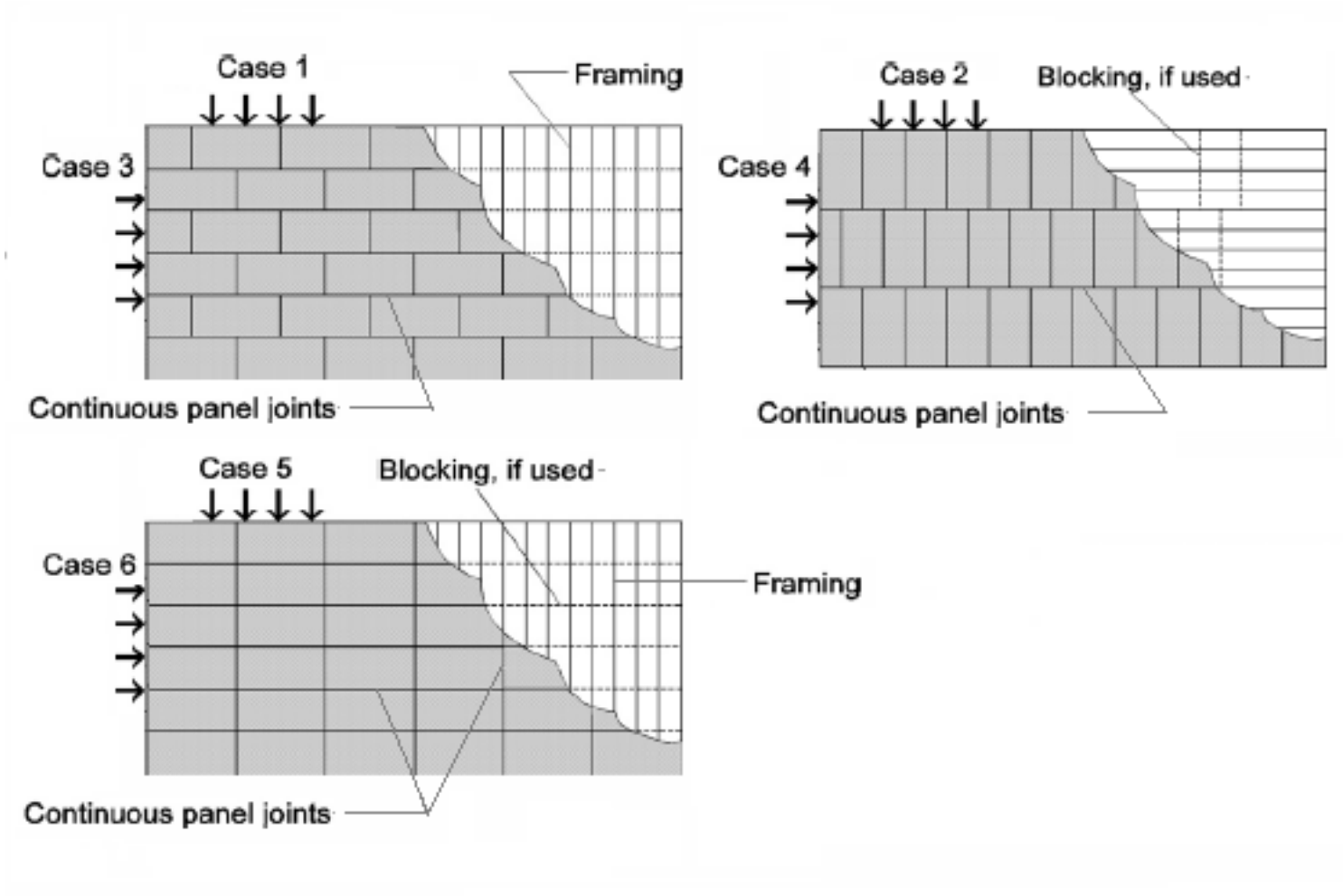
The following fields are required information. If they are not provided or are left blank, then that nailing schedule will not be available for use in the database.

The **Label** field is a used to identify the diaphragm nailing. This field must be referenced on the sheets that identify families or groups of panels.

Note:

- Panels with a label containing the characters "_W" together will be ignored during design optimization.

The **Case** field is used to specify the layout of the shear panels as shown below. Any diaphragm that has a Case 1 layout also has a Case 3 layout, and the same goes for 2/4 and 5/6.



The program considers the deck span defined in RISAFloor to coincide with the long direction of the plywood. Therefore, Cases 2, 3 and 6 are considered parallel to the to the RISAFloor deck span. Whereas, Cases 1, 2 and 5 would be considered perpendicular to the same deck span.

The **Blocked** field specifies whether blocking is used to achieve the associated design strength. It also determines the method by which the diaphragm deflection will be calculated. For more information see Diaphragm Deflection.

The **Panel Grade** field specifies what grade of structural panel is used in the diaphragm. This may be set to either "Structural-I", "Rated Sheathing" or "Other" and is used as a criteria in the Design Rules.

The **Panel Thickness** field specifies the thickness of the structural panel used for the diaphragm. This is a decimal value that is rounded to four places for reporting convenience. For example, a 15/32" panel is listed as 0.4688

The **Boundary/Cont Edge Spacing** field specifies the nail spacing at the boundary and along any continuous edges. These must be specified as the same value.

The **Other Edge Spacing** field specifies the nail spacing at non-continuous edges.

The **Nail Lines** field specifies the number of lines of nails along each panel edge. This value is greater than (1) for High Load diaphragms. This value is not currently used in design optimization, but is reported on the output for reference purposes only.

The **Strong Shear Capacity** field specifies the shear strength of the diaphragm (lbs/ft) based on its stronger case. For example, while Case 1/3 represents the same panel layout, Case 1 has greater strength than Case 3 (based on load direction). This is the seismic capacity, which the program can automatically increase for wind loads if the Wind ASIF function is enabled.

The **Weak Shear Capacity** field specifies the shear strength of the diaphragm (lbs/ft) based on its weaker case. There are many situations where strong and weak capacities are identical. In these cases the same value must be specified for both fields. This is the seismic capacity, which the program can automatically increase for wind loads if the Wind ASIF function is enabled.

The **Strong Ga** field specifies the apparent shear stiffness (kips/in) of the diaphragm as specified in the NDS document *Special Design Provisions for Wind and Seismic* Since this is the strong direction it will be based on the stronger direction / case for loading. For example, while Case 1/3 represents the same panel layout, Case 1 has greater stiffness than Case 3 (based on load direction). For more information see Diaphragm Deflection.

The **Weak Ga** field specifies the apparent shear stiffness (kips/in) of the diaphragm as specified in the NDS document *Special Design Provisions for Wind and Seismic*.

The **Gt** field specifies the shear stiffness of panel depth. It is always the same for both strong weak directions, hence it does not need to be specified twice. For more information see Diaphragm Deflection.

The **Strong Nail Slip (en)** field specifies the nail slip used for deflection calculations based on the stronger case. For example, while Case 1/3 represents the same panel layout, Case 1 may have less nail slip than Case 3 (based on load direction). For more information see Diaphragm Deflection.

The **Weak Nail Slip (en)** field specifies the nail slip used for deflection calculations based on the weaker case. There are many situations where strong and weak nail slips are identical. In these cases the same value must be specified for both fields. For more information see Diaphragm Deflection.

Note:

- The Gt and Nail Slip fields are ignored for unblocked diaphragms.

Full Database - Optional Fields

The following fields are optional. They are not currently used in the design, capacity or deflection calculations, but are reported on the detail reports for informational purposes only.

The **Framing Width** field identifies the minimum required framing width for the nailing layout. A higher shear capacity can typically be achieved for a diaphragm by using wider supporting framing, thereby reducing the tension perpendicular to the grain of supporting members.

The **Minimum Penetration** field identifies the minimum required nail penetration specified in the IBC/NDS tables.

The **Nail Size** listed in the spreadsheet is intended to refer to the Common nail size, but is reported only for reference purposes and are NOT used in the capacity calculations. If the nail size is changed by the user, then the user should also change the Shear Capacity entry accordingly. The section on shear walls contains a good reference table for common, box and sinker nails.

Grouping Panel / Nailing Schedules for Design Optimization

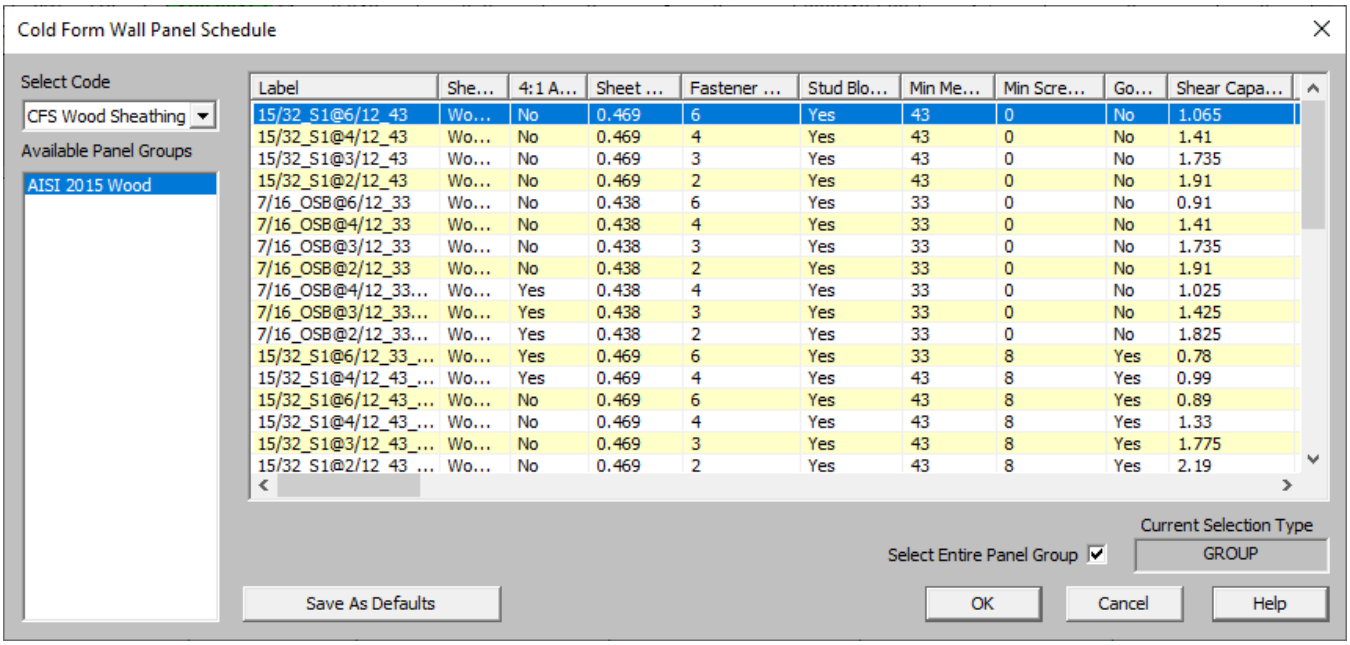
The other sheets in the database allow the user to organize multiple nailing schedules into groups or families for design optimization purposes. These additional sheets CANNOT be the first sheet in the XML file as that first sheet must always be the one where the full database information resides.

The labels specified on these sheets refer only to nailing schedules that have already been defined on the first (full) database sheet.

Appendix G – CFS Shear Wall Files

RISAFloor 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



Label	She...	4:1 A...	Sheet ...	Fastener ...	Stud Blo...	Min Me...	Min Scre...	Go...	Shear Capa...
15/32_S1@6/12_43	Wo...	No	0.469	6	Yes	43	0	No	1.065
15/32_S1@4/12_43	Wo...	No	0.469	4	Yes	43	0	No	1.41
15/32_S1@3/12_43	Wo...	No	0.469	3	Yes	43	0	No	1.735
15/32_S1@2/12_43	Wo...	No	0.469	2	Yes	43	0	No	1.91
7/16_OSB@6/12_33	Wo...	No	0.438	6	Yes	33	0	No	0.91
7/16_OSB@4/12_33	Wo...	No	0.438	4	Yes	33	0	No	1.41
7/16_OSB@3/12_33	Wo...	No	0.438	3	Yes	33	0	No	1.735
7/16_OSB@2/12_33	Wo...	No	0.438	2	Yes	33	0	No	1.91
7/16_OSB@4/12_33...	Wo...	Yes	0.438	4	Yes	33	0	No	1.025
7/16_OSB@3/12_33...	Wo...	Yes	0.438	3	Yes	33	0	No	1.425
7/16_OSB@2/12_33...	Wo...	Yes	0.438	2	Yes	33	0	No	1.825
15/32_S1@6/12_33_...	Wo...	Yes	0.469	6	Yes	33	8	Yes	0.78
15/32_S1@4/12_43_...	Wo...	Yes	0.469	4	Yes	43	8	Yes	0.99
15/32_S1@6/12_43_...	Wo...	No	0.469	6	Yes	43	8	Yes	0.89
15/32_S1@4/12_43_...	Wo...	No	0.469	4	Yes	43	8	Yes	1.33
15/32_S1@3/12_43_...	Wo...	No	0.469	3	Yes	43	8	Yes	1.775
15/32_S1@2/12_43_...	Wo...	No	0.469	2	Yes	43	8	Yes	2.19

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 Tools - Application Settings dialog.

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.

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.

Note:

- 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.

The **Type of Wall** dictates the design method that this row should be associated with.

The **Sheathing Material** indicates which material of the sheathing. This should be Wood Structural, Steel Sheet, Fiberboard or Gypsum Board Panels.

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.

The **Thickness** column is used during the design to help set the elastic stiffness of the wall panel used during the FEM solution.

The **Fastener Spacing** is used during the design optimization to limit the available panels based on the Design Rules specified by the user.

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.

Stud thickness	Sheathing entries with Min Member Thickness values equal to:
XXXSXXX - **33**	33
XXXSXXX - **43**	33, 43
XXXSXXX - **54**	33, 43, 54

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).

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 S240 and S400. The program will then apply the appropriate factors depending on if ASD or LRFD design is selected.

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 RISAFloor package. There is no charge for technical support for all users with a **current** subscription of RISAFloor. Technical support is very important to the staff at RISA Tech, Inc.

Hours: 6AM to 5PM Pacific Time, Monday through Friday (excluding holidays)

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 RISAFloor. The Table of Contents or Search can 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 will help you get a feel for the program and likely increase your modeling confidence.
2. Ensure that you are running the latest version of the software. Under the *Help* menu click **Check for Updates**
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 solve it.

Email: support@risa.com. If your question is related to a specific model, please send us that model. That makes it much easier for us to debug a problem or answer questions about specific issues. The model file has a .RFL 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: (949) 951-5815. Feel free to call, especially if you need a quick answer and your question is not model specific and therefore doesn't require us to look at your file.

Index

1

10 Feet 412

A

Adding Shapes 45, 247, 384
AISC Design Code Limitations 261
AISC Design Code Special Messages 262
AISC Design Codes 259
Ambient Temperature 364
Animation 210
  Animating Deflections 210
APA PRI-405 558
Application Interface 4
Application Settings 126
Applying Changes 216
Arbitrary Shapes 385
Area Calculations 311
Area Load Default 185
Area Loads 298, 301
Area Loads Spreadsheet 300
Assigning Deck Properties 133
Australian & New Zealand Design Codes 270
Australian & New Zealand Design Codes - Limitations 271
Auto Design Cuts 568
Automatic Backup 126
  Customizing 126
  File Operations 181
Axes
  Plot Options 212

B

Background 212
Backup 126, 181
  Customizing 126
  File Operations 181
Bar Shapes 250
  Database 250
Base Elevation 336
Base Shear 332
Baseplate Connections 122
Beam
  Modifying Design Parameters 28
  Modifying Properties 26
Beam End Reactions 43
Beam Load Takedown 317
Beam Stability Factor (wood) 546
Beams
  Deflection 41
  Drawing 20
  Overview 19
  Spreadsheet 33
Black Background 212
Block operations (spreadsheet) 5
Block Operations (spreadsheet) 397
Box Selections 241
Brace to Base Plate Connections 122
British Design Code - Limitations 265
British Design Code - Member Design Parameters 265
Building a Report 367
Building Codes 323

C

Camber Input 255

Canadian Design Codes - Limitations 263

Canadian Design Codes - Member Design Parameters 263

Canadian Design Codes - Special Messages 264

Cantilevers 22

Cardinal Points 289

Cb Factor (AISC) 260

CFS View Controls 455

CFS Wall - Design 454

CFS Wall Detail Reports 464

CH factor (Wood) 547

Channel Shapes 249

Clearing Cells 400

Cloning Views 198

Cm Coefficient 355

- Cold Formed 49
- Concrete 92
- Hot Rolled 259
- Wood (moisture) 355

Coefficient of thermal Expansion 353

Cold Formed Steel Design 48

- AISI 48
- Bending Coefficients 49-50
- Limitations 55
- R-Value 50

Color 203, 208, 212

Column Eccentricity 77

Column Load Takedown 317

Column Stability Factor (Wood) 546

Column Stack Manager 73

Column Stacks 65

Column Stacks Spreadsheet 71

Column Strips 565

Columns

- Analysis 75
- Drawing 59
- Modeling 58
- Modifying Design Parameters 64
- Modifying Properties 62
- Physical 65
- Spreadsheet 67

Commands 4

- Main Menu 4
- Shortcuts and Hot Keys 12

Company Name (printed) 370

Composite 272

- Beams 272
- Canadian 278
- Design 272

Composite Deck Database Files 137

Compressive Strength 355

Concrete Cover 145

Concrete Design 90

- Beam Results 100
- Code Options 90
- Controlling Bar Sizes 145
- Cover 145
- Cracking 92-93
- Design Rules 145
- Detail Reports 107
- K Factors 91
- Limitations 96
- Messages 98
- Rebar Layout 85
- Rebar Layout (Beams) 93
- Spans 90
- Stress Block 94
- Sway 92

T-beams & L-beams 93
Tie Spacing 196
Unbraced Lengths 91
Concrete Design Considerations 470
Concrete Design Results 100
Concrete Lintel Considerations 472
Concrete Reinforcing Spreadsheet Results 480, 502
Concrete Stiffness 93
Icracked (Beams) 93
Icracked (Columns) 92
Concrete Wall Design 469
Concrete Wall Design Rules 476
Concrete Wall Detail Reports 481
Concrete Wall Modeling Considerations 473
Concrete Wall Regions 470
Concrete Wall Results 480
Concrete Wall Spreadsheet Results 480
Concrete Wall View Controls 469
Connection Results 118
Connection Rules 114
Construction Lines 224
Context Help 246
Coordinates 16, 364
Copy 229-230
Linear Copy 229-230
Offset 233
Spreadsheet Cells 400
Copy by Angle 231-232
Cover 145
Cr factor (Wood) 547
Creating a New Floor 182
Creating Diaphragm Edges and Openings 131
Creating Openings 435
Creating Regions 437
Criteria Selections 241
Cs 332
CS shapes 46
CSA 086 Adjustment Factors 548
CU shapes 46
Custom Shapes 385
Arbitrary 385
Cold Formed 45
Hot Rolled 247
Customizable Model View Toolbar 8
Customization 126
CV 547

D

Data Entry Preferences 126
Data Entry Toolbar 11
Database 384
Arbitraty Shapes 385
Cold Formed 46, 384
Concrete 85, 384
Hot Rolled Steel 247, 384
NDS Wood 384, 536
On-Line Shapes 387
RISA Section 385
Structural Desktop (SDT) 387
Date 369
Deck and Angle Default 186
Deck Loads 301
Deck Properties - General 135
Deck Properties - Loads 140
Deck Properties - Material Type 136
Deck Unbraced Length 136
Decks and Diaphragms 131
Defaults 126, 401
Load Combinations 321

Materials 353
Mode 17
Plot Options 202
Deflection 41
Deleting 237
Deleting a Floor 184
Density 353
Material Properties 353
Design Category 332
Design for Joists with Non-Uniform Loads 406
Design Lists 682
Design Parameters
Cold Formed 48
Concrete Beams 92
Concrete Columns 91
Hot Rolled 254
Wood 544
Design Results - Steel Products 410
Design Results - Wood Products 560
Design Rules 142
Concrete 145
Deflection 143
Size/UC 142
Design Strip Rules Editor 582
Design Strips 567
Detail Reports 374
Concrete Detail Reports 107
Detailing Input 290
Detailing Layer 289
Dialog Boxes 17
Diaphragm Deflection Calculations 169
Diaphragm Design Limitations 170
Diaphragm Key Plan 170
Diaphragm Nail Spacing Schedule 168
Diaphragm Results - Detail Reports 164
Diaphragm Stiffness 153
Diaphragms 152
Diaphragms - Analysis and Results 160
Direct Analysis Method - Ratio 363
Directionality Factor (Kd) 336
Director Menu 7
Distance Below (Walls) 432
Distance Tool 12
Double Angles 249
Database 249
Drawing 216
Grid 221
Import DXF 227
Snap Points 226
Wall Panels 428
Drawing Area Loads 298
Drawing Grid 437
Drawing Slab Openings 575
Drawing Slabs 570, 610
Drawing Toolbar 10
DXF Files 172
Dynamic Pan / Zoom / Rotate 11

E

E (Young's Modulus) 353
Edit menu 5
Edit mode 17
Elevated Slabs 570, 581, 601, 610, 617, 627, 635
Elevated Slabs - Design Strip Reports 640
Email (support) 700
Emin Calculation 550
End Reactions 43
Equivalent Square Pedestals/Posts 675
Error Messages 685

EuroCode Design Codes - Limitations 268
EuroCode Design Codes - Member Design Parameters 266
EuroCode Design Codes - Special Messages 269
Excluding Results 373
Exporting Files 180
Exposure Category 336
Exposure Coefficient 337, 339
Exposure Constant Alpha 337
Exposure Constant zg 337

F

F'm compressive strength 356
Fc' (compressive strength) 355
File Locations 130
File Menu 4
File Operations 180
Filling Spreadsheets 399
Find 401
Flat File 369
Flat Use Factor -CSA 550
Flat Use Factor (Wood) 547
Flexible Diaphragms 156
Floor Level 339-340
Floor Type 185
Floors 182
Floors Spreadsheet 184
Floors Spreadsheet - Beam Floors 186
Floors Spreadsheet - Concrete Slabs 187
Font Control 129
Form Factor (wood) 548
Framing 417
Fu (Ulitmate Stress) 354
Fy (Yield Stress) 353

G

G (shear Modulus) 353
General Member Design Limitations 271
General Preferences 126
geometric axes 418, 421
Global Axes 212
Governing Equation 332
Graphic 198
  Display 198
  Editing 216
  Printing 370
  Results 373
  Selection 240
Grid 221
  Drawing 221
  Project 217
Grid Increments 438
Ground Elevation Factor 336-337
Gust Effect Factor 338

H

Hardware Requirements 1
Height 339
Help Menu 7
Help Options 246
Hold Downs 455, 515
Hot Keys 12
Hot Rolled Steel Design 252
  AISC
    ASD Limitations 261
  Allowable Stress Increase Factor 260
  Code Limitations 252
  Composite 272
  K Factors 421

Lateral-Torsional Buckling Modification Factor 260
Parameters 254
Results 281
HSS Shapes 248
HU Shapes 47

I

Icr (Concrete Walls) 433
Icr Factor - Slabs 580
Image Capture 368
Importance Factor 332, 337
Importing Files 180
Indian Design Codes - Limitations 270
Indian Design Codes - Member Design Parameters 269
Infill Beams 24
Information Dialog 365
Joints 365
Insert Cells 400
Insert Menu 6
Integration Error Messages 122
Interface 4
Inverting Selections 241
Isometric View 5, 199

J

Joints 364
Joist Girders 407
Joists and Joist Girders 404

K

K (Concrete Walls) 433
K Factors 421
KCS Joists 406
Keyboard Commands 12
Kf Factor 548

L

L Shapes 249
Labels 143, 200
Lateral Stability Factor - CSA 549
Layers (CAD) 172
Lbyy 417
Lbyy, Lbzz or Lb-in Lb-out 91, 413
Leeward Pressure 340
License Agreement 3
Limitations 475
Limits 2
Line Loads 302
Line Selections 241
Lintel Force Calculation 441
Live Load Reduction 307
LL Shapes 249
Load Attribution 314
Load Cases 294
Load Categories 295
Load Combinations 321
Design Options 322
Load Direction 296
Load Duration Factor - CSA 548
Load Generation 335
Load Generation - Seismic Loads 329
Load Generation - Wind Loads 327
Load Takedown 317
Loads 294
Cases 322
Categories 322
Combinations 321
Display 209
Duration factors 323, 546

Notional Load Generation 327
Seismic Load Generation 329
Time Effect Factor 548
Loads - Load Types 298
Local Axes
Plates 208
Locking Selections 245

M

Main Menu 4
Manuals 1
Masonry Lintels 489
Masonry Regions 490
Masonry View Controls 488
Masonry Wall Design 493
Masonry Wall Design Rules 498
Masonry Wall Detail Reports 503
Masonry Wall Modeling 488
Masonry Wall Optimization 492
Masonry Wall Results 502
Material Properties 353
Material Set 431
Material Take-Off 357
Material Type 431
Max Span 136
Mean Roof Height (h) 337
Member Design Lists 142
Member Design Rule Editor 146
Member Design Rules – Camber 144
Members
Detail Reports 374
Menus 4
Messages 535, 685
Error Message 685
Warning Log 535
Mezzanine Levels 349
Middle Strips 565
Model Display Options 202
Model Settings 188
Codes 190
Composite 192
Concrete 196
Description 188
Seismic 194
Solution 189
Wind 194
Model View 17, 199
Modeling Tips 358
Dummy Members 358
Stranded Columns 358
Stranded Walls 359
Modes (program) 17
Modify Menu 6
Modify Slab Edges 577
Modifying
Wall Panels 429
Modifying Deck Properties 134
Modifying Diaphragm Edges and Openings 132
Modifying Slab Design Strips 590
Modifying Slabs 571, 610
Moment Connection 120
Moving Elements 233
Multi-Story Column Analysis 75

N

NDS 542
New Models 126
Not Designed 551
Nu (Poisson's Ratio) 353

O

Offset Slabs Tool 577
On-Line Help 246
On-Line Shapes 387
On-Line website 246
One Way 314
One Way Shear 633
Opening Template 439
Openings 435, 439
Optimization 142
Options 126
Outriggers 23
Overview 1

P

P-Delta 361
  Turning off the P-Delta Requirement 128
P-Little Delta - AISC DAM 363
P - Little Delta Analysis 362
Page Number (for printing) 367
Panning (graphics) 199
Parabolic Attribution 314
Parapet Height (Walls) 432
Parapet Wind Loading 341
Parent 432
Parent / Child 187
Partial Views 201
Paths 130
Phone (support) 700
Physical Columns 65
Pipe Shapes 249, 387
Point Loads 304
Poisson's Ratio 353
Polygon Selections 241
Pressure Coefficient 337, 339
principal axes 418, 421
Printing 366
Program Defaults 126
Program Interaction 380
Program Limits 2
Program Modes 17
Project Grid 217
Punching Shear 634
Punching Shear Check Results 681
Punching Shear Considerations/Limitations 672
Punching Shear Detail Report 679
Punching Shear Geometries 678
Punching Shear Values 677

R

Ranking of Area Loads 314
Re-Labeling Selected Elements 239
Rectangular Drawing Grid 223
Rectangular Shapes 85, 250, 387
Redo 217, 398
Reducible Load Types 311
Region Force Calculation 441
Regions 437
Rendering (Graphics)
  Plates 208
Reports 367, 373
Resistance Factor 548
Results 372
  Menu 6
  Preferences 128
  Toolbar 11
Right-Click Menu 7
Rigid Diaphragms 152
RISA-2D Files 180

RISA Toolbar 7
RISAConnection Integration 113
RISAConnection Spreadsheet 116
RISAFloor
Buckling-Restrained Braced Frame (BRBF) System | Primary.Not_2D,Primary.Not_Foundation | [87] 379
Detach 379
Diaphragm Mass 378
Seismic Loads 379
Wind Loads 379
RISAFoundation Interaction 380
RISASection 385
Rotating 11, 199
Views 199, 212

S

Save As Defaults 126
Saving 4
Files 4
Results 372
Selections 245
Views 201
Scrolling 397
Search Bar 14
Secondary Effects 361
Section Properties 384
Seismic Design Parameters 330
Seismic Load Results 332
Seismic Loads 329
Seismic Weight 329
Select Mode 17, 240
Selection 9
Toolbar 9
Using Graphics 240
Using Spreadsheets 245
Self-Weight
Material Density 353
Self Weight 294
Semi-Rigid Seismic Loads 333
Semi-Rigid Wind Loads 350
Service Condition Factor 549
Shape Database 384
Cold Formed Steel 45
Concrete 85
General 384
Hot Rolled Steel 247
Wood 536
Shear
Modulus 353
Shear Connection Considerations 120
Shear Load Coefficient-CSA 549
Shored 138
Shortcut Keys 12
Shortcut Menu 7
Single Angle 41
Single Angles 262, 271
Size Factor- CSA 549
Size Factor (Wood) 547
Skewed Geometry for Punching Shear 673
Skip Loading 36, 79
Skyline Solver 395
Slab Definitions 578
Slab Design Cut Detail Report 656
Slab Design Rules 581
Slab Design Strips Spreadsheet 598
Slab Forces 638
Slab Manual Design Cuts 599
Slab Support Lines 585
Slabs Spreadsheet 579
Slenderness Factor 550

Sloped Members 390
Sloped Roof Wind Loads 349
Snap Points 226
Solution 394
  Preferences 128, 189
Solve Menu 6
Sorting 5, 401
SP Joists 406
Span 90, 196
Spans 566
Sparse Solver 396
Special Live Loads 311
Splice Connection 69
Splice Type 69
Spreadsheets 397
  Copying 400
  Defaults 401
  Editing 399
  Filling 399
  Finding 401
  Inserting 400
  Math 399
  Moving 397
  Sorting 401
  Wall Panels 431
Spreadsheets Menu 6
Stability Factors (Wood) 546
Stack Manager 65
Standard Combinations 323
Standard Skyline Solver 395
Static Solution 395
Status Bar 16
Steel Products Database 404
Straps 455, 515
Stud Height 138
Support Lines 566
Sway
  Concrete 92
Sway Flags 422
System Factor 549
System Requirements 1

T

T Method A 332
T Upper Limit 332
T Used 332
Tapered Area Loads 301
Technical Support 700
Tee Shapes 249
Temperature 364
Temperature Factor (Wood) 546
Thermal Coefficient 353
Toggle Buttons 200
Toolbars 7
Tools Menu 6
Tooltips 246
Top Chord Extensions 411
Topographic Factor (Kzt) 337
Topographic Factors 336
Transient Loads 323
Translucent Display 208
Trim/Extend Members 235
Troubleshooting 535
  Error Messages 685
  Warning Log 535
Troubleshooting RISAConnection Integration 120
Truss HSS T-Connections 122
Tube Shape 47
Tube shapes 248

Tutorial 1, 246
Two Way 315
Two Way Shear 634

U

UBC 323
Ultimate Stress 354
Unbraced Lengths 413
  Concrete 91
Undo 5, 217, 398
Unequal Flanges 262
Units 423
Uplift Forces and Bridging 412

V

Velocity Pressure 338, 340
Vertical Axis 174
Vertical Brace Connections 120
Vibrations 425
View menu 5
View mode 17
Volume Factor 547

W

Wall Load Takedown 317
Wall Opening Headers 455, 514
Wall Panel Editor 434
Wall Panel Function 432
Wall Panel Labels 431
Wall Panel Load Attribution 441
Wall Panel Load Transfer 445
Wall Panel Results 449
Wall Panel Spreadsheets 431
Wall Panel Thickness 432
Wall Panel View Controls 438
Wall Panels 428
Wall Panels - Results 449
Walls 428
Warning Log 535
Warnings 535
Web Site 246
Wet Service Factor (Wood) 546
White Background 212
Wind Code 336
Wind Load Results 337
Wind Loads 327, 335
Wind Speed 336
Window Selections 241
Window Toolbar 8
Windows 16
Windward Pressure 340
Wood Database Limitations 538
Wood Design 542
  Adjustment Factors 545
  Design Code 190
  GluLams 542
  Limitations 550
  Messages 556
  Parameters 544
  Shapes 536
Wood I-Joist Databases 558
Wood Product Limitations 562
Wood Products 558
Wood View Controls 513
Wood Wall - Design 461, 513, 525
Wood Wall Detail Reports 529
Wood Wall Floor/3D Interaction 458, 521
Wood Wall Results 464, 529
WT Shapes 249

Y

Yield Stress 145, 353

Young's Modulus 353

Z

Zooming 11, 199

ZS Shapes 47

ZU Shapes 47