RISAFoundation

Rapid Interactive Structural Analysis – Foundations

Version 17- General Reference



RISA Tech, Inc.

27442 Portola Pkwy, Suite 200

Foothill Ranch, California 92610

(949) 951-5815

(949) 951-5848 (FAX)

risa.com

Copyright

Copyright © 2026 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
Software 1
Hardware 2
Program Limits 2
Software Limitations 2
Demonstration Version 2
License Agreement 2
Internet (Subscription License Only) 3
Maintenance (Perpetual License Only) 3
Main User Interface **4**
Start a Model 4
Title Bar 5
Ribbon Toolbar 6
Quick Access Toolbar 6
Customize the Quick Access Toolbar 8
Add an Icon You Want Displayed . 8
Remove an Icon You Don't Want Displayed 9
Rearrange Icons 9
Properties Panel 10
Explorer Panel 16
3D View 17
Window Toolbar 17
Custom Bins 18
Selection Toolbar 21
Results Toolbar 21
Shortcut Menu 22
Dynamic View Controls 22
Shortcut Keys and Hot Keys 23
General Hot Keys 24
Shortcut Keys available for Specific Windows 24
Spreadsheet Hot Keys that open spreadsheets 25
Status Bar 25
Windows Behavior 26
Model Views 26
Spreadsheets 27
Dialogs 28
Window Tiling 28
Modes 29
File Menu **31**
Home Ribbon **34**
Modify Ribbon **40**
View Ribbon **43**
Drawing Tools Ribbon **46**
Spreadsheet Ribbon **48**
Advanced Ribbon **52**
Results Ribbon **53**
Beams **55**

Table of Contents

Draw Beams 55
Modify Beam Properties 56
To Modify Beams 57
Beam Properties 57
Beams Spreadsheet - General Data 58
Beam Labels 58
Beam Start and End Points 59
Shape 59
Material 59
Design Rules 59
Start and End Releases 59
Analysis Offset 59
Beams Spreadsheet - Concrete Data 59
Icr Factor (Cracked Moment of Inertia Factor) 60
Service Level Stiffness 60
Flexural Layout 60
Shear Layout 60
Physical Members 60
Beams - Database 61
Rectangular Sections 62
Custom Rebar Layout Database 62
Bending Rebar Layouts 64
Shear Rebar Layouts 65
Rectangular Pedestal Rebar Layouts 66
Circular Pedestal Rebar Layouts 67
Beams - Design 68
Apply a Concrete Design Code 69
Parabolic vs. Rectangular Stress Blocks 69
British Eurocode Design Parameters (BS EN 1992-1-1: 2004) 70
General 70
Tension Development Length 70
Shear Capacity of Concrete 70
Limitations - General 71
Limitations - ACI 71
Limitations - Canadian Code 71
Limitations - Saudi Code 72
Special Messages 72
No Load Combinations for Concrete Design have been run. 72
Warning: No design for spans with less than 5 sections. 72
Warning: No design for spans less than 1 ft. 73
Warning: The shear tie spacing does not meet the code Minimum Requirement 73
Rebar spacing exceeds maximum allowable (ACI 318-14 Section 13.3.4.4, ACI 318-11 Section 15.10.4) 73
Beams - Design Results 73
Beam Results 73

Table of Contents

Beam Design Results Spreadsheet 73
Beam Bending Reinforcement Spreadsheet 74
Beam Shear Reinforcement Spreadsheet 75
Beam Detail Reports 76
Beams - Forces 83
Number of Reported Sections 83
Adjust Number of Sections 84
Number of Internal Sections 84
Adjust Number of Internal Sections 84
Beam Force Results 84
Boundary Conditions 87
Creating and Modifying Boundary Conditions 87
Modify Boundary Conditions for Nodes 87
Auto-Generated Soil Springs 88
Boundary Conditions Spreadsheet 89
Boundary Condition Options 90
Reaction Boundary Condition 90
Spring Boundary Condition 91
Compression-Only Springs 91
Tension-Only Springs 91
Customizing RISAFoundation 92
Save as Defaults 93
Application Settings (Preferences) 95
General Application Settings 95
Advanced Application Settings 98
Input Application Settings 100
Output Application Settings 102
File Location Application Settings 104
Design Optimization 106
Design Rules - Beam Rebar 106
Design Rules - Slab Rebar 107
Design Rules - Footing/Pile Cap Rebar 109
Design Rules - Pile/Pedestal Rebar 109
DXF Files 111
Import a DXF Drawing Grid 111
Export a Foundation Plan Drawing 113
Exporting Foundation Plot Plans 115
Exporting Slab Reinforcing Details 116
Exporting Footing Details 116
Exporting Pedestal Details 117
Exporting Pile Cap Details 118
Exporting Retaining Wall Details 118
File Operations 120
Start Off 120
Starting a Model Window 120
Create a New Model 121
Open an Existing Model 122
Open a Sample Model 123

Table of Contents

Save a Model 124
New Untitled Model 124
Existing Model 124
Exporting Files 124
Exporting to DXF Files 124
Exporting to Excel Spreadsheet 125
Automatic Backup 125
Footings - Design 127
Spread Footing Definitions Spreadsheet 127
Footing General Properties 127
Reinforcement Ratios 127
Material 128
Design Rules 128
Equal Bar Spacing 128
Enveloping Multiple Footing Designs 128
Concrete Bearing Check 128
Force Top Bars 128
Footing Geometry 128
Force Footing to be Square 129
Footing Pedestal 129
Boundaries and Eccentricities 130
Soil Properties 131
Overburden 131
Passive Resistance of Soil 131
Coefficient of Friction 131
Gross/Net 131
Local Axes 132
Limitations 133
Non-Linear Effects 133
Footing - Results 133
Solution Methodology 133
Sketch and Details 133
Footing Details 134
Soil Bearing Results 135
Footing Flexure Design 136
Footing Shear Check 137
Pedestal Design 138
Pedestal Flexure 139
Pedestal Shear 140
Pedestal Punching Shear 140
Stability Results 140
Overturning Check 141
Sliding Check 142
Concrete Bearing Check 142
Footings - Combined 142
Strap Footings 142
Using Slab Foundations to Model Combined Foundations 143
Footing Stability and Overturning Calculations 144
Calculation of Overturning Moment (OTM) Stability Ratio 144

Table of Contents

Traditional / Simplistic OTM Calculation 145
True Safety Factor Method (used by RISAFoundation) 145
Comparing the Two Methods 146
Calculation of Moment and Shear Demand for Unstable Footings 146
Generation 148
Circular Slab Generator 148
Graphic Display 150
Window Behavior 150
Multiple Windows 152
Open a New 3D View 152
Switch Between Open 3D Views 154
Adjust the Viewpoint (3D View) 155
Zoom Features 155
Use Your Mouse to Zoom 155
Use the Zoom Tools 155
Pan the View 157
Pan Using Your Mouse Wheel 157
Pan Using the Right-Click Menu 157
Snap to a View 158
Rotate a View 160
Click to Rotate 160
Drag to Rotate 160
Reset Rotation to Default 161
3D View Display Options 161
Display Loads 161
Display Results 162
Display Deflection 162
Animate Results 163
Create a Custom Snapshot of a Model 164
View Only Part of a Model 166
Save / Retrieve a Model 166
Save a Model View 166
Retrieve a Saved View 168
Delete a Model View 169
Graphic Editing 171
Draw / Modify Features 171
Where to Start 172
Apply Options 172
Undo / Redo Operations 172
Undo Operations 172
Redo Operations 173
Project Grid 173
Show / Hide Project Grid Lines 173
Show / Hide Project Grid Arcs 174
Project Grid Spreadsheet 175
Project Grid Line Generation 176
Project Grid Arc Generation 180
Edit the Project Grid Lines (Adding or Moving Grid Lines) 184
Drawing Grid 186

Table of Contents

Drawing Grid Features 186
Show / Hide the Drawing Grid 188
Drawing Grid Type 188
Rectangular Drawing Grid 188
Radial Drawing Grid 190
DXF Underlay Grid 190
Relocating the Drawing Grid 191
Skewed Drawing Grid 191
Save a Drawing Grid Default 192
Save a Drawing Grid 192
Recall a Drawing Grid 192
Snap Points 193
Import DXF Grid 194
Locating the DXF Drawing Grid 197
Delete Elements 197
Delete a Single Model Element 197
Delete Multiple Model Elements 198
Copy Model Elements 201
Copy Selected Items 201
Global Copy 202
Point to Point Copy 202
Rotate Copy 204
Mirror Copy 204
Move / Rotate Model Elements 205
Move or Rotate Selected Items 206
Global Move 206
Point to Point Move 207
Rotate Move 207
Drag and Move in 3D View 208
Re-Label Selection Items 209
Merge Model Elements 210
Perform a Model Merge 210
Match 211
Graphic Selection 213
Selection Shortcuts 214
Cycle Selection 215
Selection Modes 215
Select All and Unselect All 217
Box Select and Unselect Modes 217
Polygon Select and Unselect 218
Line Select and Unselect 218
Invert Selections 218
Criteria Selections 218
Nodes 219
Beams 220
Slabs/Pedestals 221
Spread Footings 222
Walls 223
Piles 224
Design Strips 225
Loads 226
Coordinates 227

Table of Contents

Lock & Dim Lock 228
Viewer Mode 230
Input Filter 233
Graphic Selection from Spreadsheets 233
Save Selections 234
Save a Selection 234
Retrieve a Saved Selection 235
Rename a Saved Selection 236
Delete a Saved Selection 237
Quick Find 238
Graphic View Settings 239
Graphics - Model View Settings 239
Set as Default 241
General 242
Beams 243
Footings 245
Walls/Stem Walls 247
Piles/Pile Caps 248
Slabs 250
Strips/Cuts 251
Loads 253
Graphics - Results View Settings 254
Nodes 256
Beams 257
Footings 259
Walls 260
Piles 262
Slabs 263
Design Strips/Cuts 265
Deflection 267
Help Options 269
Electronic Help File 269
Context Sensitive Help 269
RISA Tech, Inc. Online 269
Visit RISA Tech, Inc. on the web to: 269
Tool-tips 270
Tutorials 270
Hot Rolled Steel - Databases 271
Select a Database Shape 271
Add a Database Shape 273
Edit a Database Shape 275
Available Database Shapes 279
Wide Flange Database Shapes 279
Tube Database Shapes(Hollow Rectangular shapes) 279
Pipe Database Shapes 280
Channel and Double Channel Database Shapes 280
Tee Database Shapes 280
Double Angle Database Shapes 280
Single Angle Database Shapes 280
Solid Rectangle Shapes 281

Table of Contents

Solid Round Bar Shapes 281
Hot Rolled Steel - Databases 282
Select a Database Shape 282
Add a Database Shape 284
Edit a Database Shape 286
Available Database Shapes 290
Wide Flange Database Shapes 290
Tube Database Shapes(Hollow Rectangular shapes) 290
Pipe Database Shapes 291
Channel and Double Channel Database Shapes 291
Tee Database Shapes 291
Double Angle Database Shapes 291
Single Angle Database Shapes 291
Solid Rectangle Shapes 292
Solid Round Bar Shapes 292
Loads 293
Self Weight (Gravity Load) 293
Draw Loads 293
Modify Loads 294
Delete Loads 294
Loads - Load Categories 294
Loads - Load Combinations 297
Add Load Combinations Manually 298
Solve Load Combinations 298
Load Combinations Spreadsheet 298
Nesting Load Combinations 301
RISAFoundation Design Combinations 301
Service Combinations 302
Design Combinations 302
Generate Building Code Combinations 302
Wind Load Options 303
Seismic Load Options 303
Overstrength LC Options 304
Load Combination Generator - General Notes 304
Customize the Load Combination Generator 305
RSA Scaling Factor 306
Base Shears 307
Scaling Factor 307
Program Calculated 307
User Input 308
Apply Check box 308
Loads - Nodal Loads 308
Drawing Nodal Loads 308
Nodal Load Spreadsheet 309
Loads - Distributed Loads 311
Drawing Distributed Loads 311
Distributed Loads Spreadsheet 312
Loads - Area Loads 312
Drawing Area Loads 313

Table of Contents

Area Loads Spreadsheet 313
Material Properties **315**
Material Properties Spreadsheet 315
Hot Rolled Specific Material Data 316
Wood Specific Material Data 316
Commercial Species Groups 317
Concrete Specific Material Data 318
Masonry Specific Material Data 318
Material Takeoff 319
Model Settings **321**
Project Information 322
Settings 322
Solution 323
Processor Core Utilization 325
Codes 326
Concrete 328
Rebar 329
Seismic 330
Modeling Tips **332**
Soil Springs for Eccentric Footings 332
Modeling Complex Slabs 332
Nodes **333**
Define Nodes 333
Relabel Nodes 333
Round-Off Node Coordinates 334
Node Coordinates Spreadsheet 334
Node Information Dialog 335
Nodes - Results 335
Node Deflections Results 336
Node Reaction Results 336
Pedestals/Posts & Stem Walls **338**
Draw Pedestals/Posts 338
Modify Pedestals/Posts Properties 339
Draw Stem Walls 340
Modify Stem Walls 341
Pedestals/Posts Spreadsheet 342
Label 342
Type 342
Shape 342
Material 343
Design Rules 343
Angle 343
Height 343
Pedestal Rebar Design 344
Pedestal Layout 344
Shear Rebar Design 344
Shear Layout 344
Stem Walls Spreadsheet 344
Label 344
Start/End Node 345
Width/Height 345
Material 345

Table of Contents

Pedestal/Post - Design and Punching Shear Results 346
Pedestals/Posts Code Checks 346
UC (Unity Check) and Governing LC 347
Shear UC (Unity Check), Governing LC and Direction 347
Phi Used 347
Pedestal Reinforcement 348
Punching Shear Values 348
UC (Unity Check) and Governing LC 348
Location 348
Vuy 348
Muz and Mux 349
Total Stress 349
Phi*Vny 349
Minimum Slab Reinforcement at Punching Shear Critical Section 349
Punching Shear Geometries 350
Shape 350
Location 350
Effective Depth 350
L1 and L2 350
Polar Moment - xx and Polar Moment - zz 351
Gamma x and Gamma z 351
Pedestal/Post Detail Report 351
Input Data and Graphics 352
Design Checks 353
Punching Shear Check Results 355
Design Considerations 355
Moment and Axial Force Thresholds 355
Piles and Pile Caps 356
Draw Piles 356
Modify Pile Properties 358
Pile Definitions Spreadsheet 359
General Tab 359
Label 359
Shape 359
Material 359
Length 360
Pile Embedment 360
Pile Compressive and Tensile Capacities 360
Pile Shear Capacity 360
Pile Compressive and Tensile Stiffness 360
Pile Shear Stiffness 361
Concrete Tab 361
Pile Caps 362
Pile Tab 364
Label 364
Number of Piles 365

Table of Contents

Pile Definition 365
Max Depth/Diameter 366
Pile Spacing Factor 366
Min Edge Distance 366
Pile Location 366
Axial and Shear Capacity Reduction Factors 366
Pile Cap Tab 366
Reinforcement Ratios 366
Material 367
Design Rules 367
Cap Thickness 367
Overburden 367
Pedestal Tab 367
Pile and Pile Cap Modeling Considerations 368
Modeling with Piles, Slabs and Pedestals vs Pile Caps 368
Rigidity Assumptions 369
Pile Groups and Thickened Portions of a Slab within a Mat 370
Pile and Pile Cap Limitations 371
Pile - Results 371
Pile Design Checks 371
Pile UC Checks 371
Forces 372
Gov LCs 372
Soil UC Checks 372
Punching Shear Values 374
UC Check 374
Gov LC 374
Location 374
Vuy 374
Phi*Vny 374
Punching Shear Geometries 375
Effective Depth 375
L1 and L2 375
Pile Reinforcement (Concrete) 375
Phi Used 376
Pile Reinforcement 376
Pile Detail Report 376
Input Data and Graphics 377
Design Results 378
Punching Shear Check Results 379
Soil and Force Profile 380
Pile Caps - Design 380
Pile Cap Design Philosophy 381
Pile Capacity Checks 381
Allowable Pile Loads 381
Loads Developed in Piles in Pile Cap Applications 381
Pile Cap Flexural Design 382
Flexural Reinforcement Design 384
d Calculation 385

Table of Contents

Pile Location Tolerance Consideration 385

One Way (Deep Beam) Shear Check 385

Considerations 387

Pile Location Tolerance Considerations 388

Code Considerations 388

Pile Punching Shear Checks 388

bo Calculations 388

When there is no reinforcement at the top face of the pile cap (in either direction) 389

When the Pile's Local Axes do not correspond to the slab geometry 389

Using Equivalent Square Piles .. 389

d Calculation 390

Group Punching for Piles 390

Beams/Pedestal Intersecting the Shear Perimeter of a Pile 390

Pedestal Punching Shear Checks 390

d Calculation: 391

When Piles intersect the Shear Perimeter of the Pedestal 391

Pedestal Design 392

Pile Design 392

Pile Cap - Results 392

Pile Cap Design Spreadsheet 393

Pile Cap Code Checks 393

Pile Cap Geometry 393

Pile Cap Steel Reinforcement 394

Pile Cap Design Forces 394

Pedestal Design Checks 395

Pile Cap Detail Report 395

Header 395

Input Data, Material Properties and Design Properties 397

Sketch 398

Reinforcement Details 399

Pile Capacity Checks 400

Soil Results 400

Pile Cap Flexural Design 401

Pedestal Punching Shear Check 402

Pile Punching Shear Check 402

Punching Shear Geometry 403

One Way Shear Check 404

Pedestal Design Checks 405

Printing 406

Print a Report or Spreadsheet 406

Print Graphics 407

Printing Reports 408

Building a Report 408

Snapshot (Capture) Images 410

Adding Images 411

Report Print Settings 412

Table of Contents

Printing to a Flat File 412
Flat File Printing Options 413
Graphics Printing 414
Punching Shear Design 417
Punching Shear Basics 417
Punching Shear Calculation Values 417
Effective Depth, d 417
L1 and L2 418
Polar Moment Ixx and Polar Moment Izz 418
Centroid Location 418
Gamma xx and Gamma zz 419
Total Stress 419
Phi*Vny 419
Minimum Slab Reinforcement at Punching Shear Critical Section 420
Punching Shear Loading 420
Direct Punching Shear 420
Moment Consideration 421
Finding the Controlling Case (Interior, Edge or Corner) 422
Corner vs Edge Controlling Cases 424
Other Punching Shear Considerations/Limitations 425
Punching Shear Perimeters That Cross Different Slab Thicknesses 425
Pedestal/Post Groups 425
Beams, Columns and Walls 426
intersecting the influence area of a Pedestal/Post .....
When the Pedestal/Post Local Axes Do Not Correspond to the Slab Geometry 427
Edge Conditions 427
Corner Conditions 428
Using Equivalent Square Pedestal/Post 430
Punching Shear Reinforcement 430
Punching Shear Checks for Walls, Point Loads, and non-Concrete Columns 431
Results 432
Results Spreadsheets 432
Detail Reports 432
Saving Results 433
Results Spreadsheets 433
Finding Results 433
Sort Results 433
Exclude Results 433
Excluding Results After the Solution 434
Excluding Results While in Spreadsheets 435
Graphic Results 435
Plot Results Graphically 435
Clear Results 436
RISAFoundation Integration 438

Table of Contents

RISAFoundation Integration with RISA-3D 438

Foundations at Different Elevations 441

RISAFoundation Integration with RISAFloor 442

Gravity Elements in RISAFloor 442

Lateral Elements in RISAFloor 442

Limitations 442

RISAFoundation can not recognize moving loads 442

RISAFoundation requires your Y-axis to be vertical 442

RISAFoundation will not consider non-linear load effects 442

RISAFoundation will not consider second order effects 443

Shape Databases 444

Database Shape Types 444

Hot Rolled Shapes 444

Concrete Shapes 445

Wood Shapes 446

General Shapes 447

Pipe Database Shapes 448

Solid Rectangular Shapes 448

Solid Circular Shapes 448

Database Files 448

Slabs 450

Draw Slabs 450

Circular Slab Generator 450

Modify Slab Properties 452

Draw Thickened Regions 453

Modify a Thickened Region 454

Modify Edges 455

Create Slab Openings 457

Modify Slab Opening Edges 459

Slabs Spreadsheet 459

Label 460

Thickness 460

Material 460

Local Axis Angle 460

Analysis Offset 461

Passive Pressure 464

Soil Overburden 464

Icr Factor (Cracked Moment of Inertia Factor) 465

Service Level Stiffness 465

Automatic Slab Meshing 465

Control Points 466

Meshing Adjacent Slabs 466

Modeling Complex Slabs 466

Limitations 467

Slabs - Design Cuts 467

Create Design Cuts 467

Design Cuts Spreadsheet 467

Label 468

Table of Contents

Design Rule 468
Design Cut Results Spreadsheet 468
Design Cut Detail Report 469
Header 469
Input Data 469
Code Check Data 470
Area of Steel Required 470
Reinforcement Design Routine 471
Single Layer Rebar Design 472
Cross Section Detailing 472
Slabs - Design Strips 472
Create Design Strips 473
Setting the Width of Design Strips 474
ACI Definition of Strips (ACI 318-14 Section 8.4.1/ACI 318-11 Section 13.2) 474
Zero Shear Transfer Method 474
Zero Moment Method 475
Shear Perimeter Method 475
Hybrid Method / Engineering Judgment 475
Modify Design Strips 475
Design Strips Spreadsheet 476
Label 476
Rebar Angle from Plan Horizontal (deg) 476
No. of Design Cuts 476
Design Rule 477
Design Strip Results 477
Design Strip Detail Report 477
Header 477
Input Data 478
Envelope Shear and Moment Diagrams 478
Code Check Data 479
Slabs - Results 479
Plate Force Results 479
Plate Corner Force Results 481
Slab Contour Display Details 482
Slabs - Sliding & Overturning 484
Demand vs Resisting Forces 484
Overturning 486
Angle 486
Overturning Moments 486
Resisting Moments 488
Ratio 488
Limitations 488
Sliding 488
Angle 489
Applied Forces 489
Resisting Forces 489
Sliding Ratio (SR) 490
Limitations 490
Overturning by Category 490
Sliding by Category 491

Table of Contents

Soils **493**
Assigning Soil Properties 493
Soil Definitions Spreadsheet 493
Label 493
Layers 493
Subgrade Modulus 493
Allowable Bearing Pressure 494
Draw Soil Regions 494
Soil Layer Profile Spreadsheet 496
Label 496
Layer Start Depth 496
Unit Weight 496
Unit Skin Friction 497
Unit Tip-Bearing 498
Soil Springs Calculation (Automatic) 498
Adjusting the Subgrade Modulus 498
Single and Combined Footings 499
Grade Beams 500
Modeling Mat Foundations 500
Soil Pressure Results 500
Envelope Soil Pressure Results Spreadsheet 501
Plotting the Soil Pressure Results 501
Solution **504**
Solution Method 504
Static Solutions / Stiffness Matrix 504
Skyline Solver 505
Sparse Solver 505
Multi-Threaded Solutions 505
Spreadsheet Operations **507**
Moving and Scrolling 508
Spreadsheet Keyboard Commands 509
Selecting Spreadsheet Cells 509
Undoing Operations 510
Redoing Operations 510
Editing Spreadsheets 511
Inserting, Deleting and Clearing Cells 513
Moving and Copying Cell Contents 514
Sorting and Finding in Spreadsheets 516
Default Spreadsheet Data 517
Export to Excel Spreadsheet 517
Special Spreadsheet Functions 518
Load Combinations Spreadsheet 519
Selecting Items in the Model View 519
Units **521**
Standard Imperial Units 522
Standard Metric Units 522
Units Specifications 523
Wall Footings - Modeling **524**
Wall Footing Types 524
Retaining Wall 524
Strip Footing (Gravity) 524
Strip Footing (Lateral/Custom) 525

Table of Contents

Draw Wall Footings 525
Modify Wall Footings 528
Wall Footings Spreadsheet 530
Label 530
Start Point/End Node 530
Definition 530
Wall Footing Modeling Tips 530
Wall Footing Orientation 530
Load Categories 532
Loading of Wall Footings with Externally Applied Loads 533
Connectivity with Other Elements in the Model 536
Wall Footing Definitions 539
General Tab 539
Label 540
Wall Type 540
Wall Const 540
Wall Material 540
Foot Material 540
Wall Height 540
Soil Height 540
Water Height* 540
Propped?* 541
Foot Restrained?* 541
Wall/Foot Continuity?* 541
Foot Dowels* 541
Geometry Tab 541
Wall Thickness 541
Foot Thickness 542
Toe/Heel Length 542
Key Width/Depth* 542
Key Offset* 542
Wall Batter?* 542
Toe/Heel Batter* 542
Soil Tab 542
Method (Rankine or Coulomb)* 543
Gamma Heel/Gamma Toe 543
Phi Heel/Phi Toe* 543
K Lat Heel/K Lat Toe* 543
Delta Wall* 543
Backfill Angle* 543
Heel Surcharge* 544
Toe Depth* 544
EQ Max* 544
Kh* 545
Gamma Sat/Phi Sat Heel/K Lat Sat Heel* 545
Wall Tab (Conc) 545
Location 545
Int Bar and Ext Bar 545
Min/Max Int and Ext Bar Space 545
Bar Inc 546
Int and Ext Cover 546

Table of Contents

Hor Bar Size and Hor Bar Spacing 546
Outer Bars 546
Wall Tab (Masonry) 547
Block Grouting 547
Reinforced 547
Wall Area Method 547
Bar Size 548
Bar Space Min/Max 548
Bar Placement 548
Cover 548
Mortar/Cement Types 549
Footing Tab 549
Location 550
Bot Bar and Top Bar 550
Min/Max Bot and Top Bar Space 550
Bar Inc 550
Top Cover and Bot Cover 550
Long Bar Size and Long Bar Spacing 550
Wall Footing Definition Editor 551
Accessing the Wall Footing Editor 552
Wall Footings - Design 553
Soil Considerations 553
Lateral Force Coefficient (K) Calculations 554
Lateral Soil Summation 556
Loading Considerations 556
Seismic Loading Considerations .. 556
User Define the Magnitude of the Maximum Value of the Diagram 557
Allowing the Program to Calculate this Magnitude Using the Mononobe-Okabe/Seed-Whitman Formulation 557
Serviceability Considerations 558
Overturning 558
Sliding 560
Stem Wall Design Considerations 561
General Considerations 561
Concrete Wall Considerations 562
Axial/Bending Design 563
Moment and Axial Force Thresholds 563
Shear Check (Concrete) 563
Masonry Wall Considerations 564
ASD Design Considerations 565
Strength Design Considerations 568
ASD Masonry Design For Unreinforced Walls 570
Footing Design Considerations 572
Shear Friction Check (for dowels between the wall and footing) 572
Soil Pressure Considerations 575
Flexural Design 577

Table of Contents

Shear Checks 578
Wall Footing - Results 578
Wall Footing Results Browser 578
Footing Code Checks 579
Wall Code Check (Concrete) 579
Masonry Wall Results 580
Geometry 580
Reinforcement 581
Stability 581
Graphic Results 582
Wall Footing Detail Report Information 582
Header 582
Rebar Diagram 585
Loading Diagram 586
Input Data 588
Wall Design 590
Footing Design 595
Overturning Check (Service) 596
Warning Log 599
Wood - Database 600
Wood Material/Shape Selection 600
Material Selection 600
Create/Edit Wood Material dialog 601
Size Selection 602
View Design Properties 603
Wood Database Limitations 603
Round Shapes 603
Glulam Databases 603
Mechanically Graded Lumber 604
Structural Composite Lumber (SCL) 604
Design Values 605
NDS Emin Calculation 607
Custom Wood Sizes 607
Wood - Design 607
Glu-Lams 608
Glulam Material Properties 608
Glulam Dimensions 609
Glulam Limitations 609
Custom Wood Materials 609
Create and Apply a Custom Wood Material 611
Appendix B – Error Messages 611
Appendix D – File Format 612
Technical Support 613

Before You Begin

Welcome to the RISAFoundation 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 RISAFoundation 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

RISAFoundation is a foundation analysis and design program. It has been developed to make the definition, solution and modification of foundation models as fast and easy as possible. Analysis, up to and including calculation of maximum deflections and stresses, may be done on mat slabs, spread footings, and grade beams. Complete reinforcing detailing is also included in the program.

RISAFoundation has full graphical modeling capability. You can draw your model on the screen and also perform extensive graphical editing simultaneously in multiple views. To modify your model data directly, RISAFoundation employs a powerful, proprietary spreadsheet. All this combined with flexible data generation algorithms makes modeling very easy. Graphic display of the model along with applied loads, boundary conditions and much more, is always available. The model can be rapidly edited, solved, viewed, modified, re-solved, etc. The truly interactive nature and design output of RISAFoundation is its primary strength.

You may access the features in RISAFoundation by using the menu system, or the toolbars. The best way to learn RISAFoundation is to go through the Tutorials. The advantage to this is that you are exposed to the tools RISAFoundation provides and the ways that you can take advantage of them.

Minimum System Requirements

Software

One of the following operating systems is required:

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

Hardware

The following hardware is required:

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

Program Limits

Software Limitations

Points	500,000
Beams	10,000
Materials	500
Point Loads	250,000
Line Loads	10,000
Area Loads	10,000
Load Combinations	5,000
Slabs	250
Soil Regions	1000
Design Strips	2000
Point Supports (Footings, Piles)	7,500
Pile Definitions	800
Footing Definitions	800
Pile Cap Definitions	800
Pedestals	7,500

Demonstration Version

You may build and solve any model, however you may not save a model. There are also limits on some of the design values. For example, the allowable bearing stress and subgrade modulus for soil can not be adjusted.

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

Internet (Subscription License 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.

Maintenance (Perpetual License Only)

Program maintenance provides all upgrades to RISAFoundation.

When your maintenance expires, you will be given the opportunity to continue program maintenance on an annual basis. You are under no obligation to continue program maintenance, of course, but if you decide to discontinue maintenance you will no longer receive RISAFoundation program upgrades and technical support. In the event maintenance is discontinued, reinstatement of maintenance will require payment of an "upgrade" fee.

Complete program support is available to registered owners of RISAFoundation and is included in the purchase price. This support is provided for the life of the program. See Technical Support for a list of your support options.

The “life of the program” is defined as the time period for which that version of the program is the current version. In other words, whenever a new version of RISAFoundation is released, the life of the previous version is considered to be ended.

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

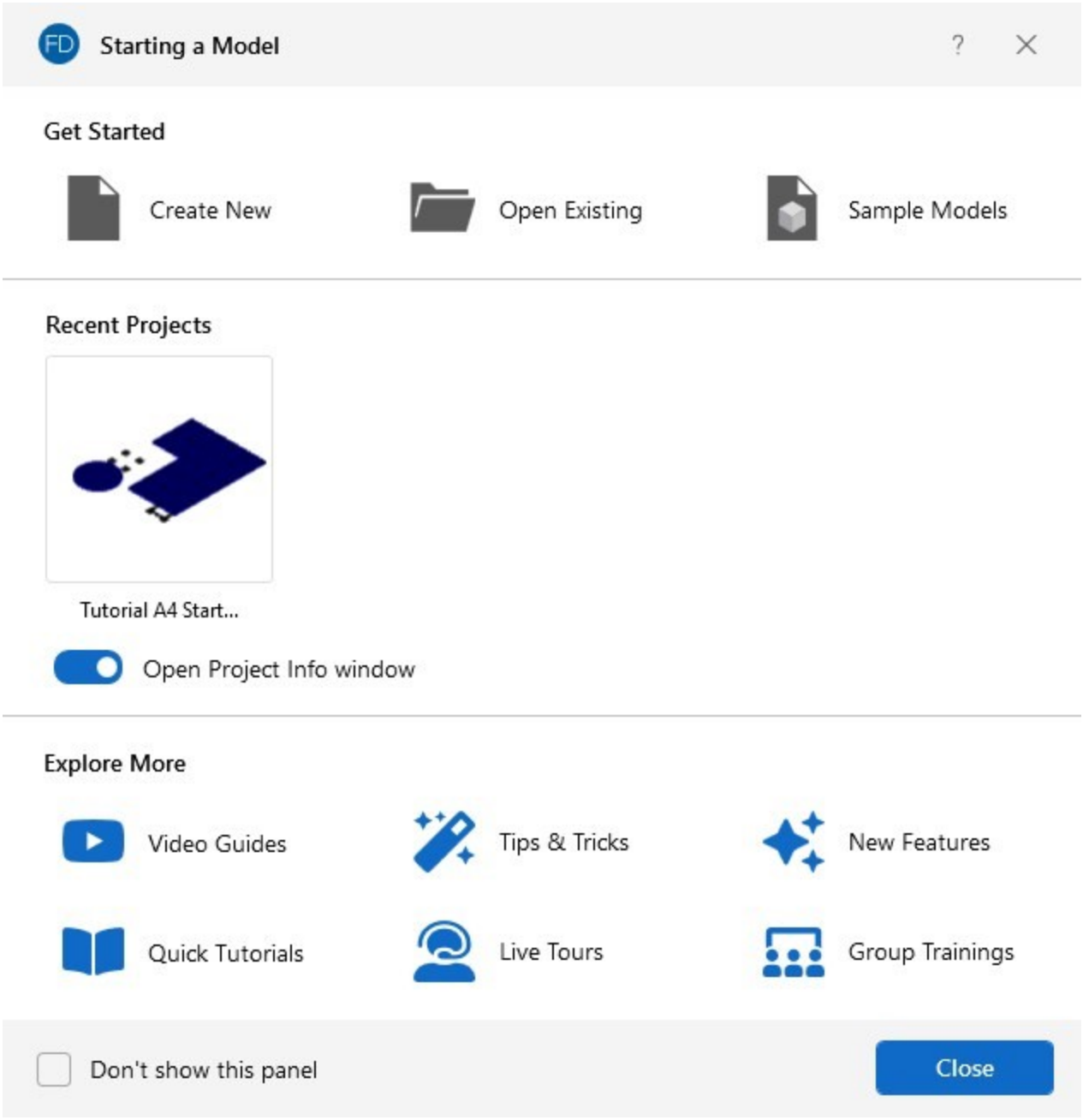
Main User Interface

The Tutorials (a separate document) contain a tutorial that leads you through the RISAFoundation interface with an actual model. If you have not done so already, consider going through the tutorial, 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 RISAFoundation may be accessed through the Ribbon toolbar, Quick Access toolbar, Properties panel, Explorer panel, 3D view and shortcut keystrokes. You may use any or all of these vehicles to interact with the software. The Ribbon toolbar has the advantage of containing all of the program options and features. The Quick Access toolbar contains more common options and invoke with one click. The Properties panel displays relevant properties for selected elements within your model and allows you to make quick changes to those elements. The Properties panel is also where you will modify and execute most of the program features. The Explorer panel provides a quick way to access all of the data entry and results spreadsheets. Shortcut menus present options relevant to the task at hand. The shortcut keys provide a fast way to access features, should you use the program often enough to make them familiar to you. All of these features are discussed in the sections below. There are many ways to access features and the method that you will use will simply be a matter of personal preference. The good news is that you have the options.

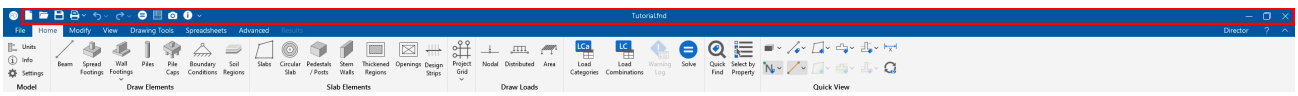
Start a Model

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



Title Bar

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



The three buttons on the upper 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. Below these three buttons is the Director button, Help File button and Minimize Ribbon Toolbar button.



The Director button will allow you to transfer back to your RISA-3D model. The question mark will pull up the RISAFoundation help file. The caret allows you to minimize the contents within the Ribbon toolbar.

To the left of the title bar next to the FD logo is the Quick Access toolbar.

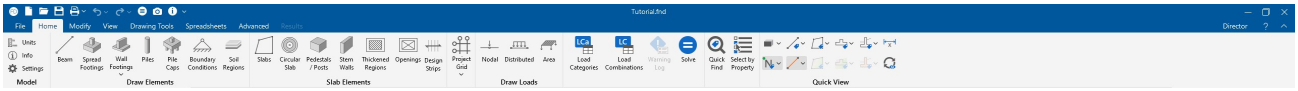


The Quick Access toolbar provides a quick way to access commonly used features and functions within the software. See the Quick Access toolbar section for more information.

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

Ribbon Toolbar

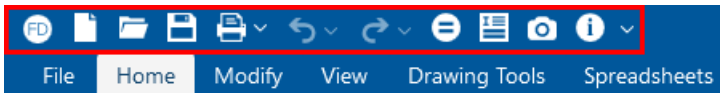
All of the program features may be accessed through the Ribbon toolbar system beginning with the **File** tab on the far left and ending with the **Results** tab on the far right.



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

Quick Access Toolbar

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



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

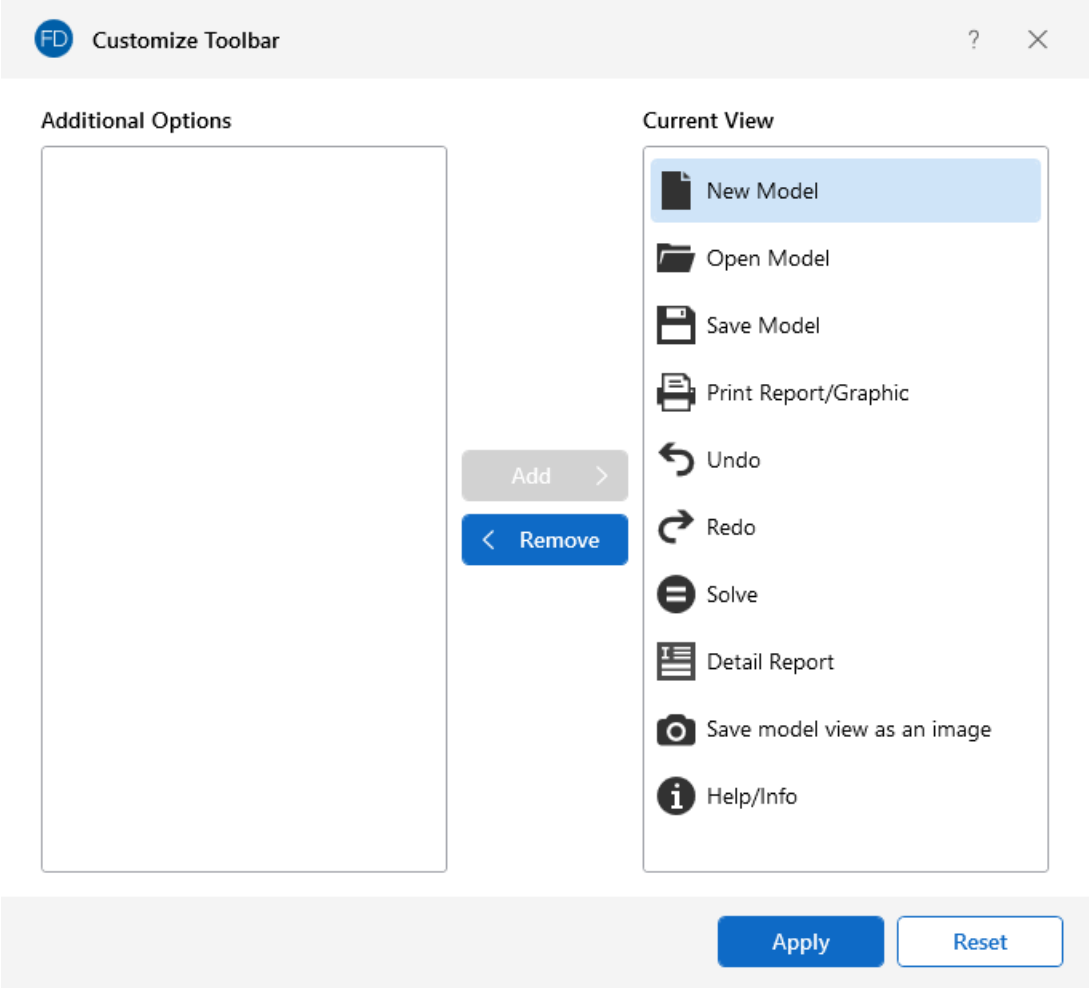
Quick Access Toolbar Icons

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

Icon	Name	Description
		• Move the Quick Access Toolbar above the Ribbon Toolbar. • Minimize the Ribbon.

Customize the Quick Access Toolbar

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



Add an Icon You Want Displayed

To add an icon to the Quick Access Toolbar:

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

Remove an Icon You Don't Want Displayed

To remove an icon from the Quick Access Toolbar:

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

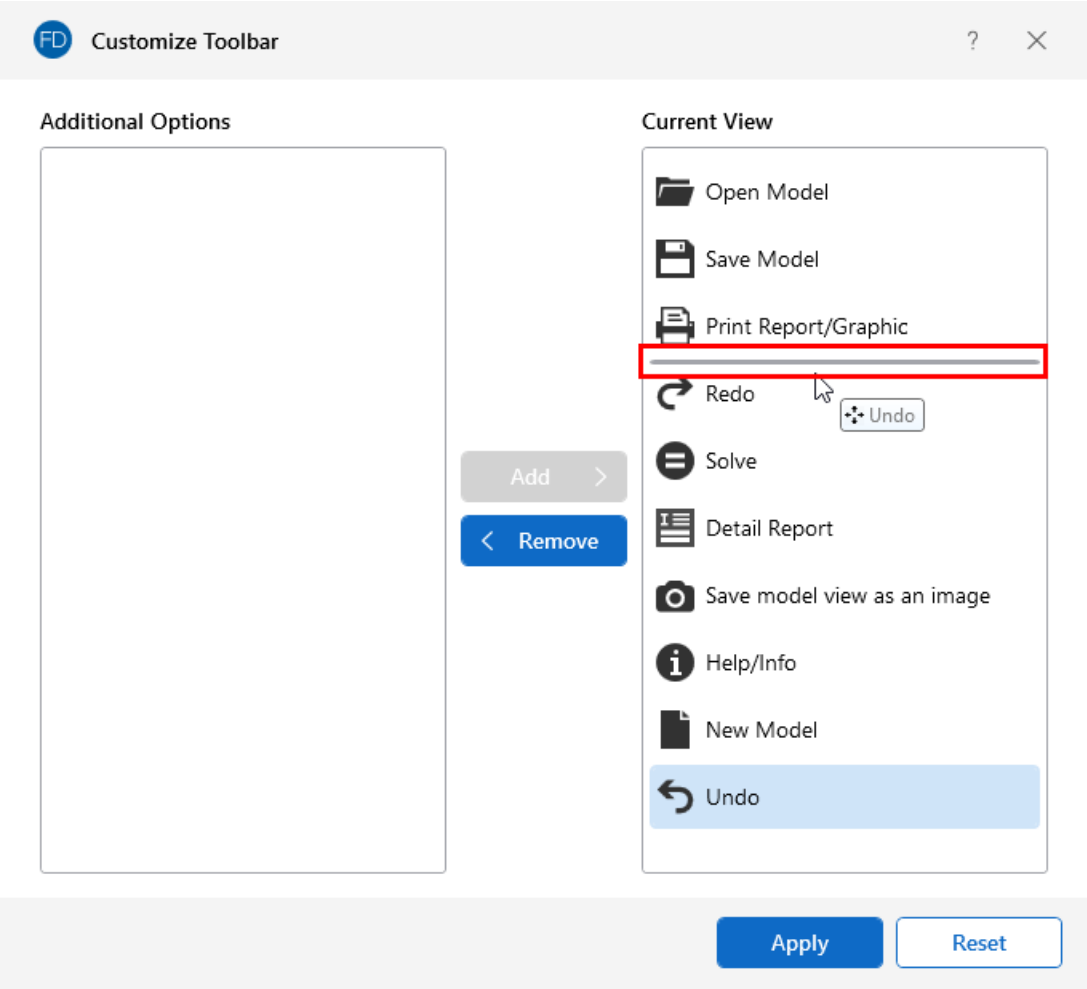
Rearrange Icons

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

To rearrange the position of an icon:

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

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

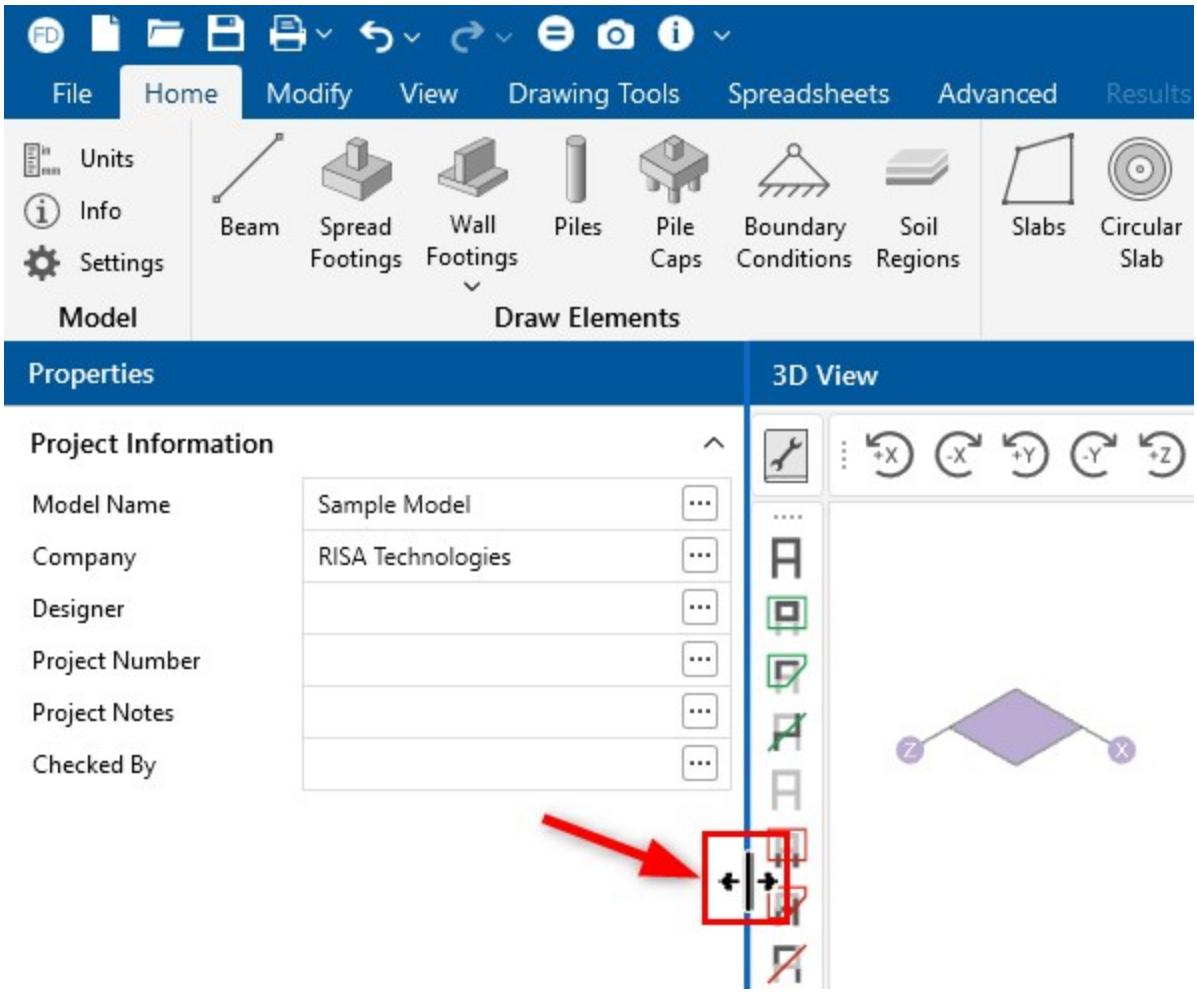


Dragging up moves the icon to the left on the Quick Access Toolbar, while dragging down moves it to the right.

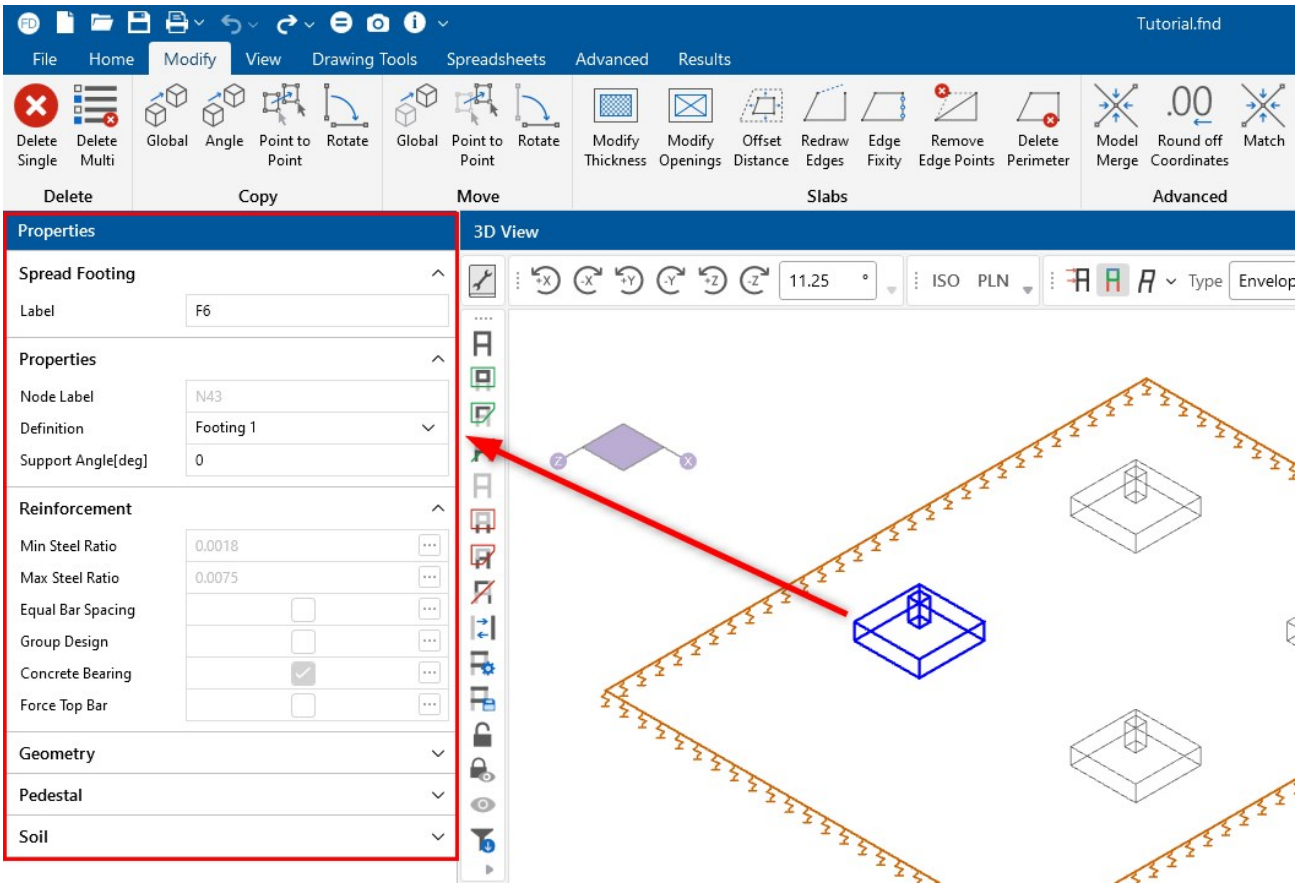
3. Repeat for each icon you want to move.
4. Click **Apply**, after you have moved the icon(s) into the position(s) you want them in.

Properties Panel

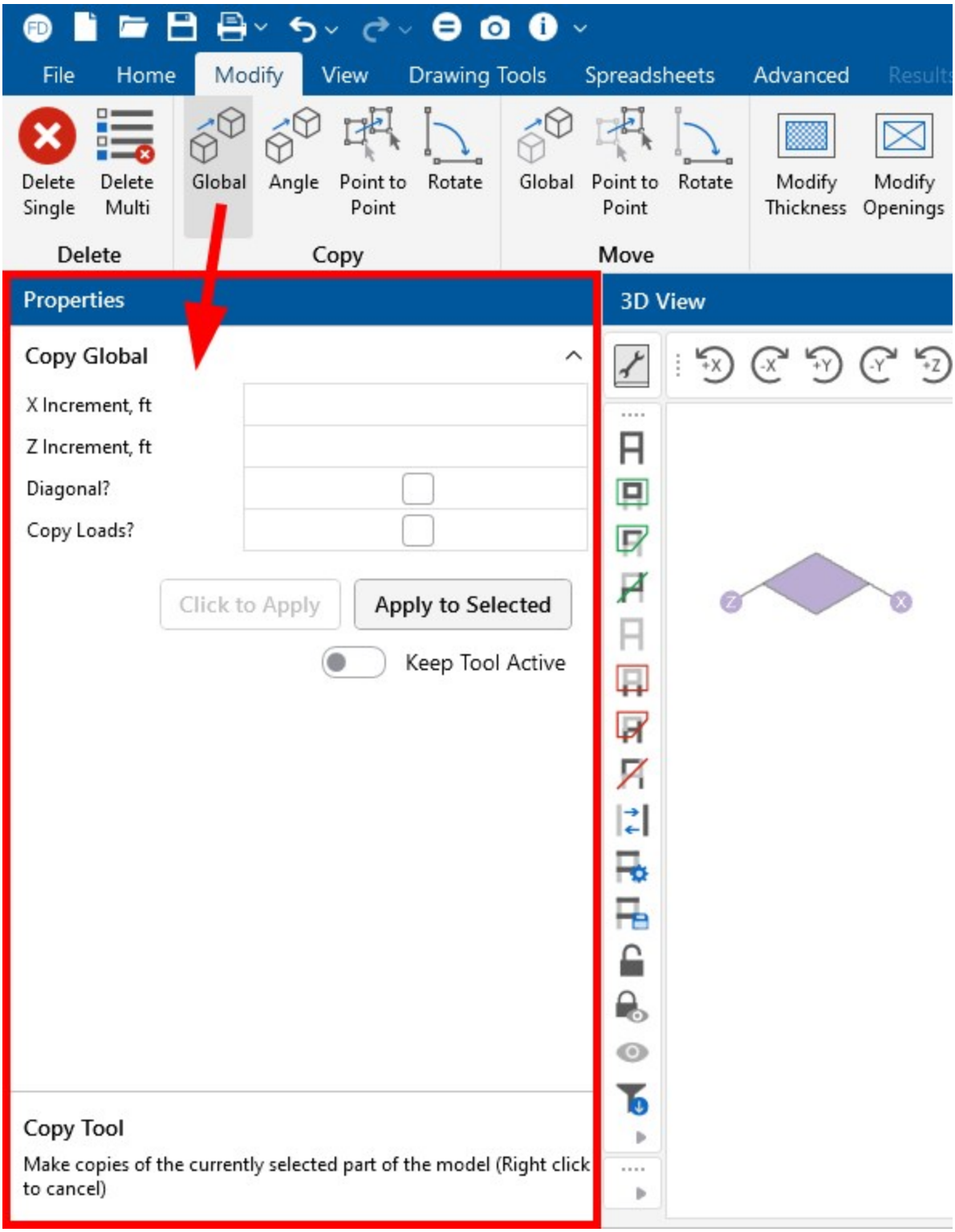
The **Properties** panel is an affixed panel on the left side of the program interface. This panel serves different functions depending on which elements are selected or what feature is activated. When there are no features activated or elements selected, the **Properties** panel displays the Project Information entered for the current model. Note that hovering your mouse over the right edge of the Properties panel (as shown in the following image) allows you to drag and expand or shrink the panel horizontally.



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

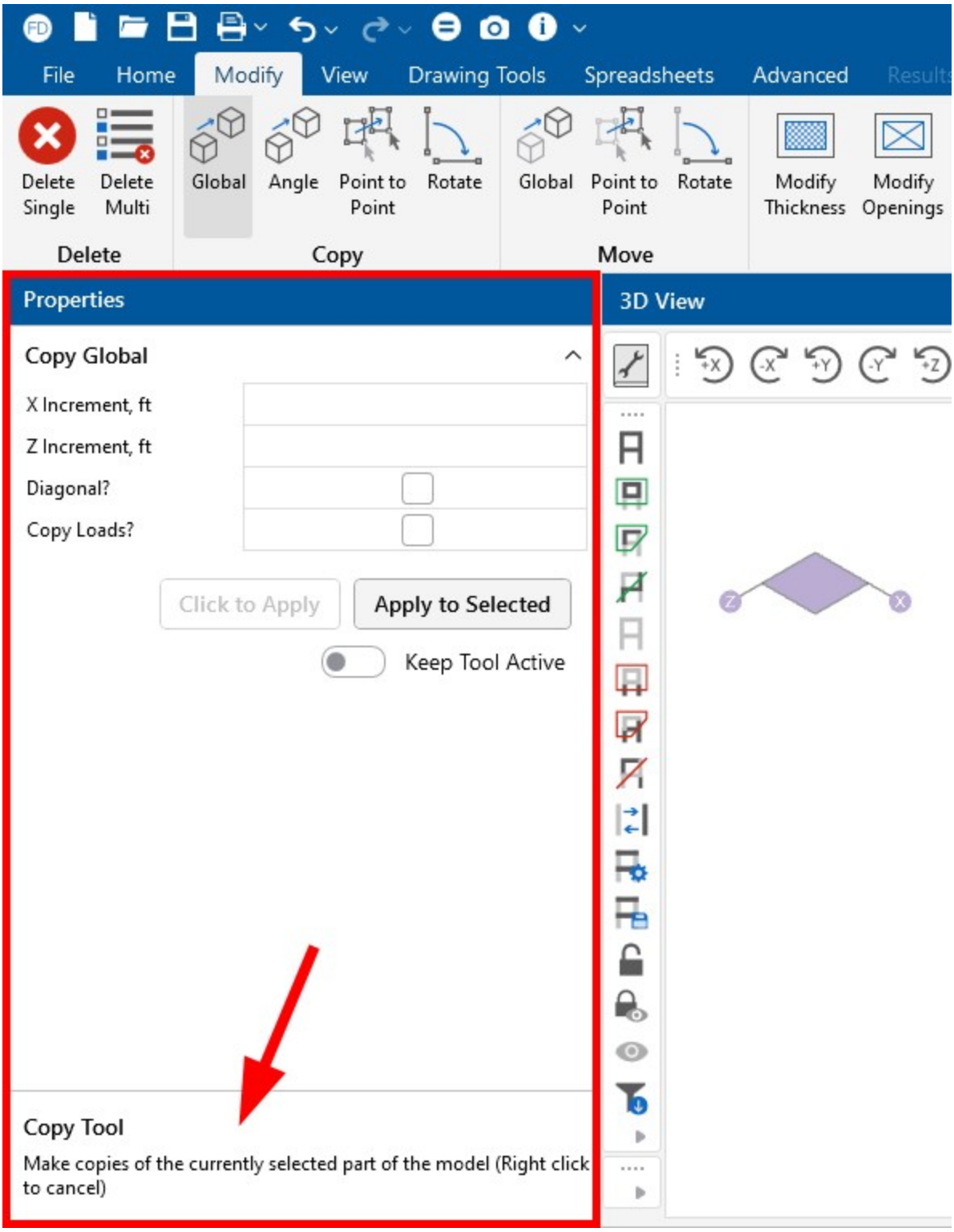


When activating a feature within the program, such as adding a load or a copy/move tool, the **Properties** panel will serve as the method to specify and apply that feature's parameters.


File
Home
Modify
View
Drawing Tools
Spreadsheets
Advanced
Delete Single
Delete Multi
Delete
Global
Angle
Point to Point
Rotate
Copy
Global
Point to Point
Rotate
Move
Modify Thickness
Modify Openings
Properties
Copy Global
X Increment, ft
Z Increment, ft
Diagonal?
Copy Loads?
Click to Apply
Apply to Selected
Keep Tool Active
Copy Tool
Make copies of the currently selected part of the model (Right click to cancel)
3D View

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

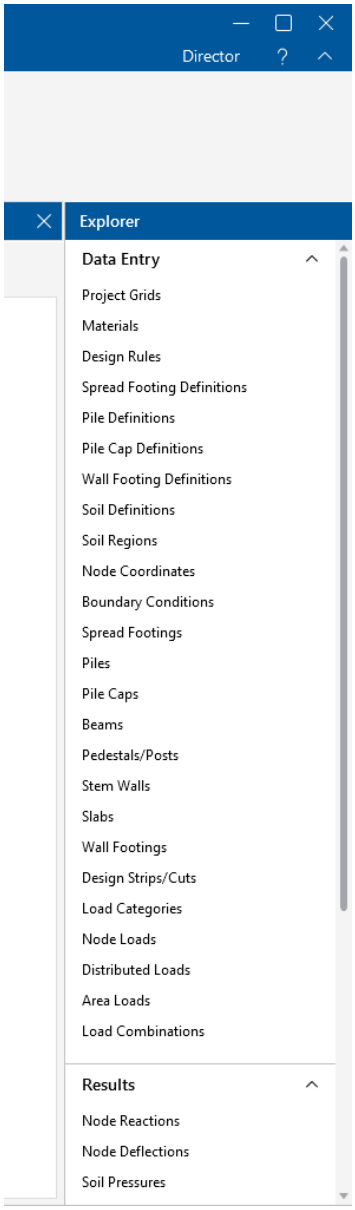
When navigating through a spreadsheet or inputs for a specific function, you will find a short description at the bottom of the **Properties** panel.



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

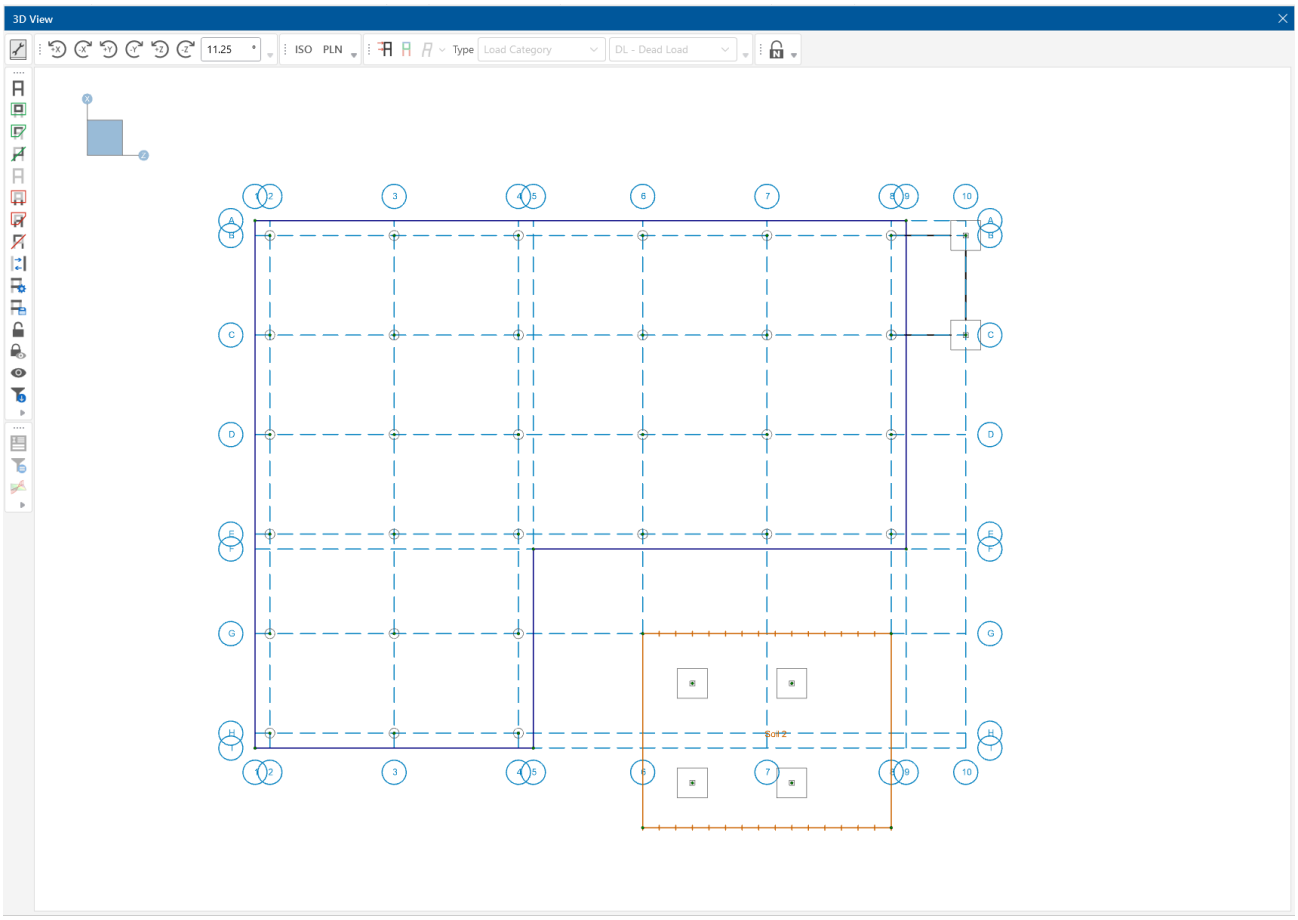
Explorer Panel

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



3D View

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



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

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

See the Graphic Display section for additional details on working with **3D View** windows.

Window Toolbar

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



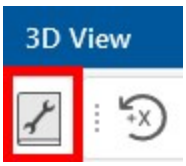
Other 3D view windows that are open are not affected, so that each may show different information. For more information, see the Graphic Display section.

Custom Bins

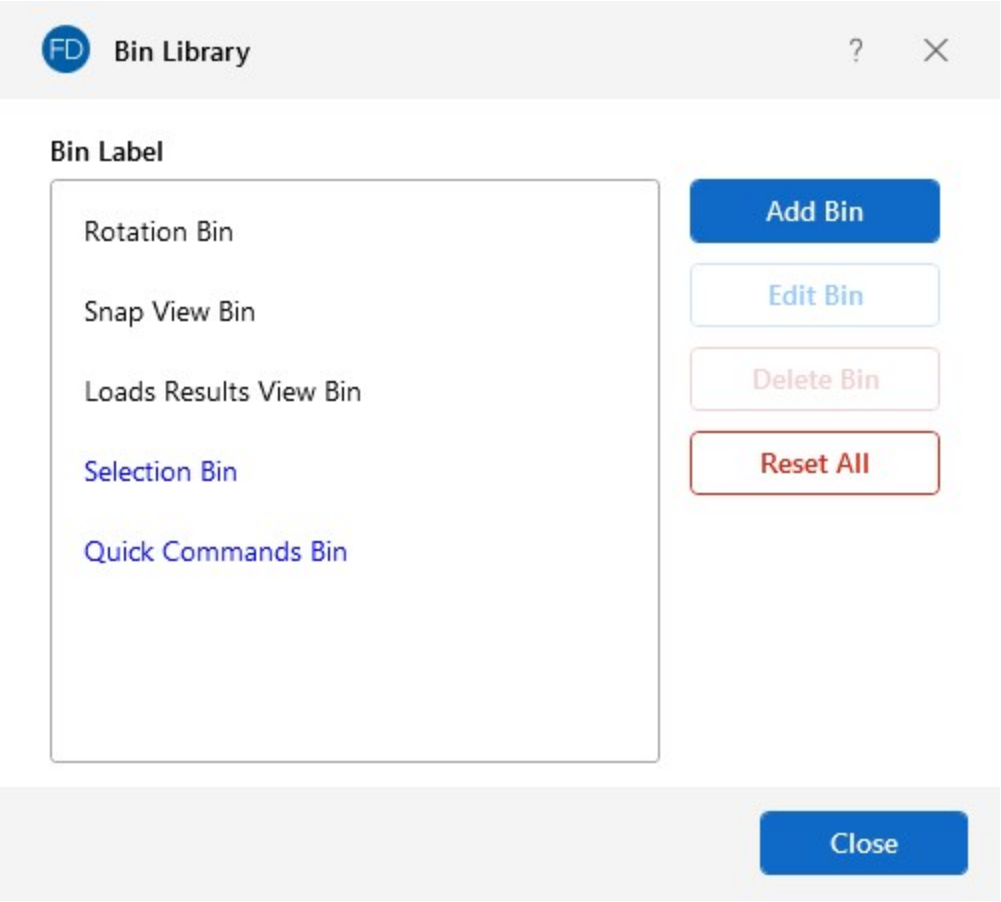
Each grouping of quick buttons in the **3D View** panel is called a **Bin**. You can customize your bins so that your most-used tools are easily accessible.

To customize your bins:

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

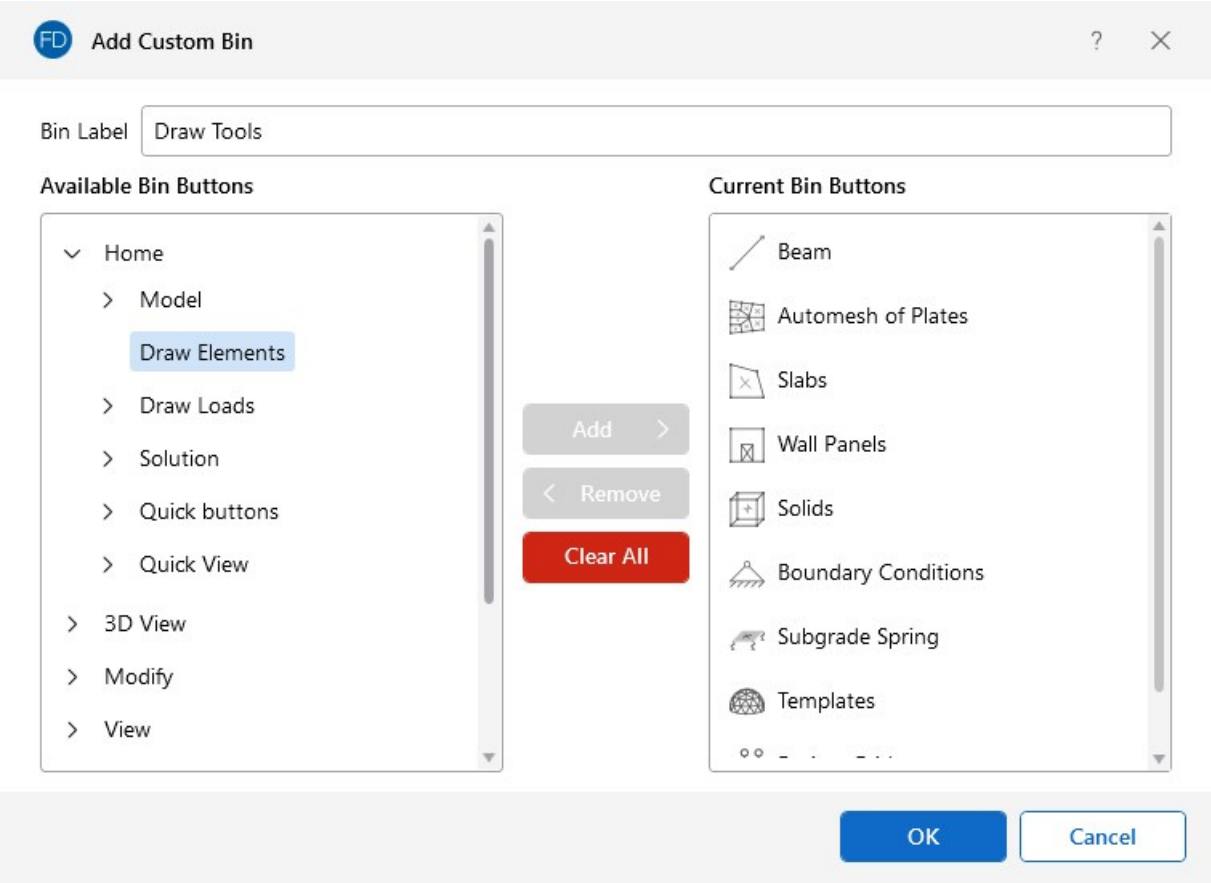
   

   The Bin Library window opens.



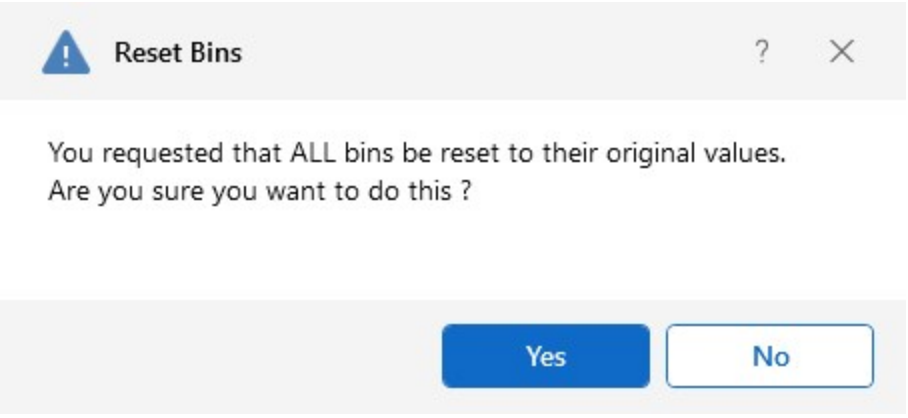
In this window you can add your own bins and customize the quick buttons within each bin. You can also edit and delete bins. Note that the default bins can not be deleted (Rotation Bin, Snap View Bin and Loads Results View Bin).

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



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

2. Click **OK** in the Add Custom Bin window to save and return to the Bin Library window.
3. (Optional) Click the **Reset All** button in the Bin Library window to reset all your bins to their default configuration.



4. Click **OK** in the Bin Library window if you are finished customizing bins.

Selection Toolbar

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



Results Toolbar

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

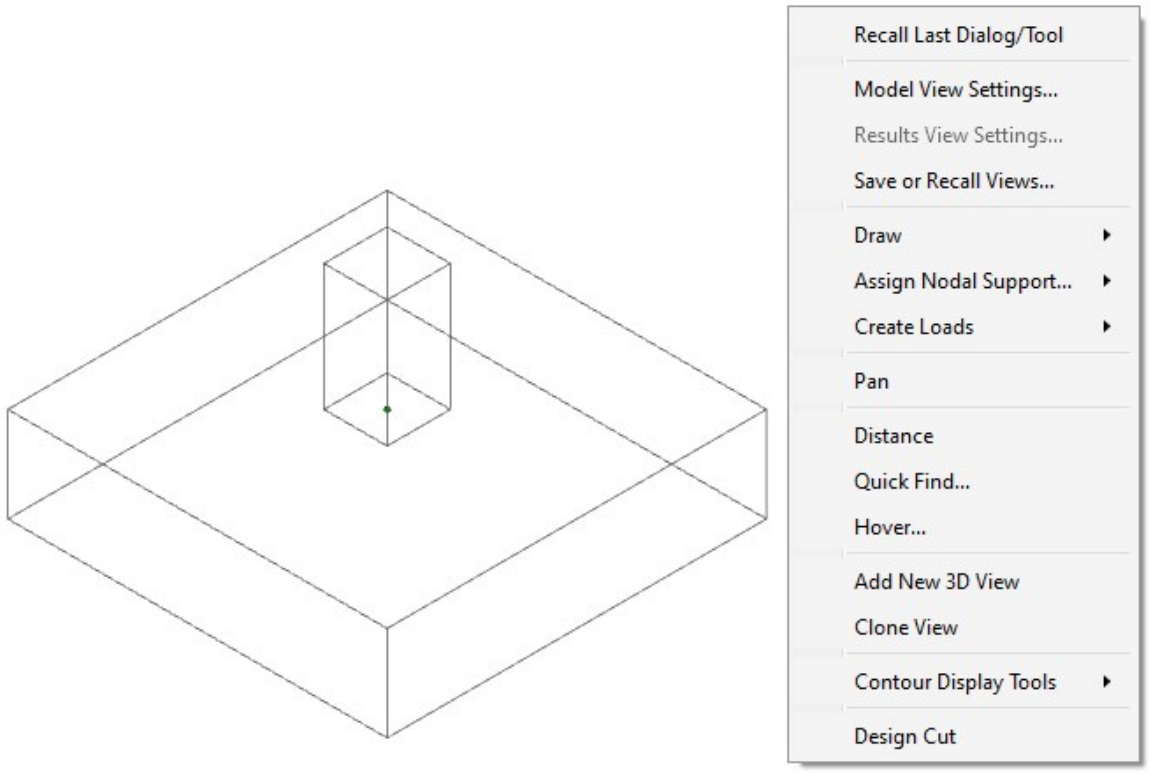


The tools in this toolbar become active when a solution is present. They provide quick access to **Result** tools that would otherwise be selected in the **Results** tab in the **Ribbon** toolbar. For more information, see Results.

Shortcut Menu

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

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



Dynamic View Controls

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

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

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

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

Shortcut Keys and Hot Keys

Shortcut Keys and **Hot Keys** allow you to use the keyboard to access features quickly. 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 almost 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 a new view
F7	Solve the Model
Ctrl-C	Copy to the clipboard
Ctrl-V	Paste from the 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

Shortcut Keys available for Specific Windows

Key Combination	Model View Window	Spreadsheet
Ctrl-D	Open last graphic editing dialog (opens in Properties panel)	Delete Marked Lines
Ctrl-A	Select All	Select all
Ctrl-U	Unselect all	Unselect all
Ctrl-F		Block Fill
Ctrl-M		Block Math
Ctrl-I	Invert Selection	
Ctrl-L	Toggle Lock unselected	Unmark lines
F2	Open Model Display Options	Start/Stop Cell Edit
F3		Insert line
F4		Delete line
F5	Initiates the "Distance" tool	Find
F8		Repeat current line

Key Combination	Model View Window	Spreadsheet
+	Zoom in	
-	Zoom out	
PgUp PgDn	Scrolling	Scrolling

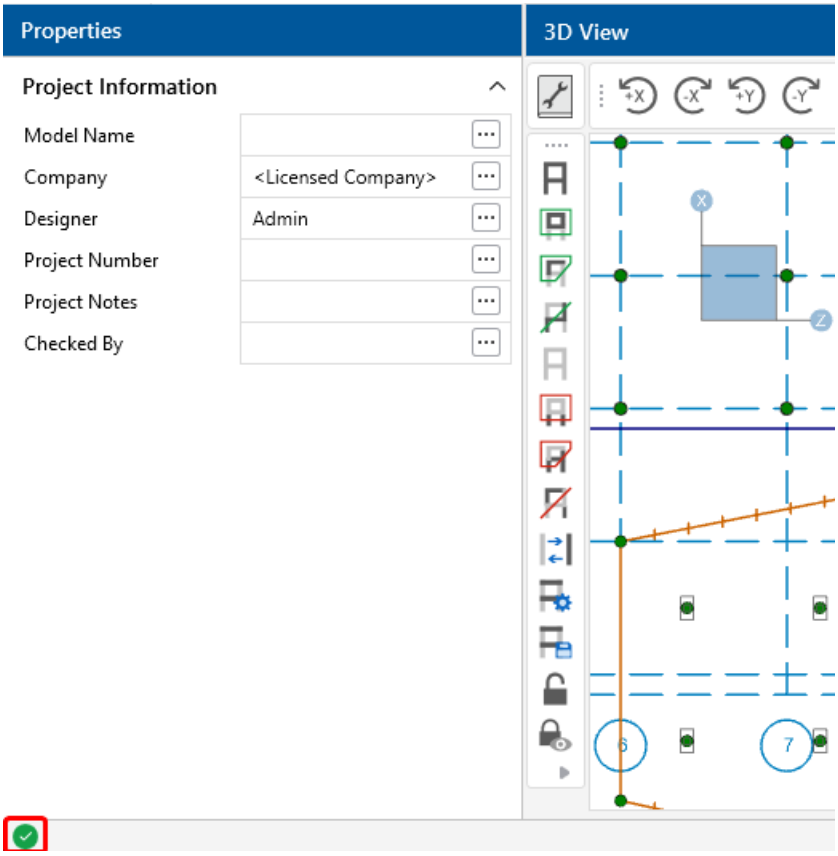
Spreadsheet Hot Keys that open spreadsheets

Key Combination	Unsolved Model	Solved Model
Ctrl-Alt-C	Point Coordinates	Corner Forces
Ctrl-Alt-D	Distributed Load	Joint Deflections
Ctrl-Alt-G	Model Settings	
Ctrl-Alt-P	Point Loads	
Ctrl-Alt-R	Design Rules	Reactions

Status Bar

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

The flag will indicate whether or not your model currently has a solution. When the flag is gray (as shown in the image above), a solution is not present. The flag turns green when a solution is present.



Windows Behavior

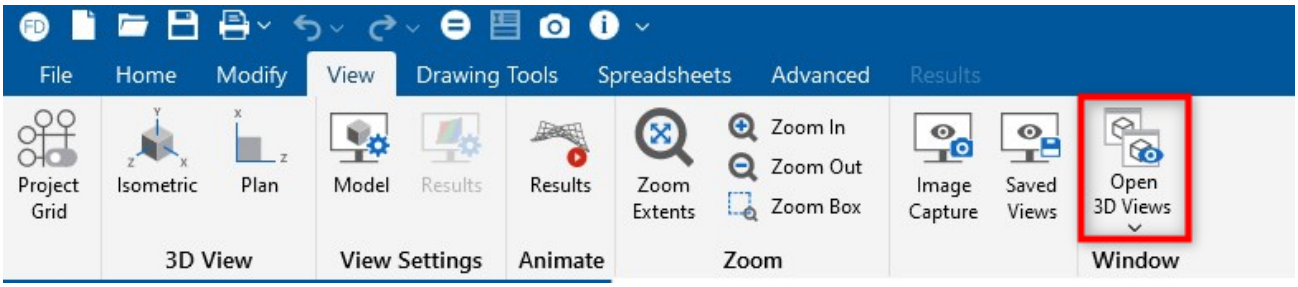
Modeling the structure takes place within model views and spreadsheets, each in its own window that may be moved around the workspace and sized as you wish. The ability to have multiple model views and multiple spreadsheets open at one time is a powerful feature. You can move the windows around by clicking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

These windows all contain an **x** 

in the upper right corner to help you close the window. There are also scroll boxes to help you view information outside of the window viewing area. Click the scroll bar buttons or drag the scroll box to advance the display in a particular direction.

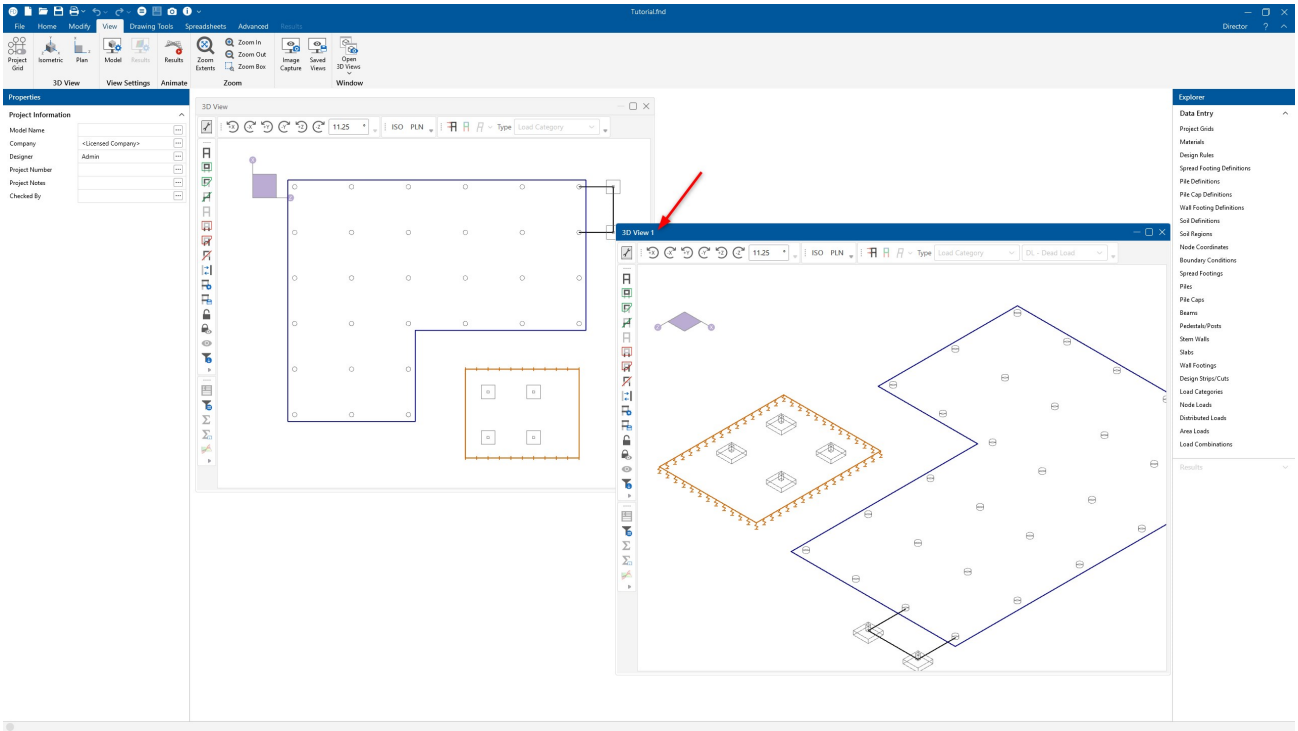
Model Views

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



You may open as many model view windows as you like, which can be 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 display are dependent on that active window. The active window is the one with the highlighted blue bar, whereas an inactive window will be grayed out. In the example below, the 3D View 1 is the active window.



For more information on this view, see the Window Behavior section in Graphics Display.

Spreadsheets

Spreadsheet windows are made up of rows and columns of data cells. To add or edit data in a spreadsheet cell, 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.

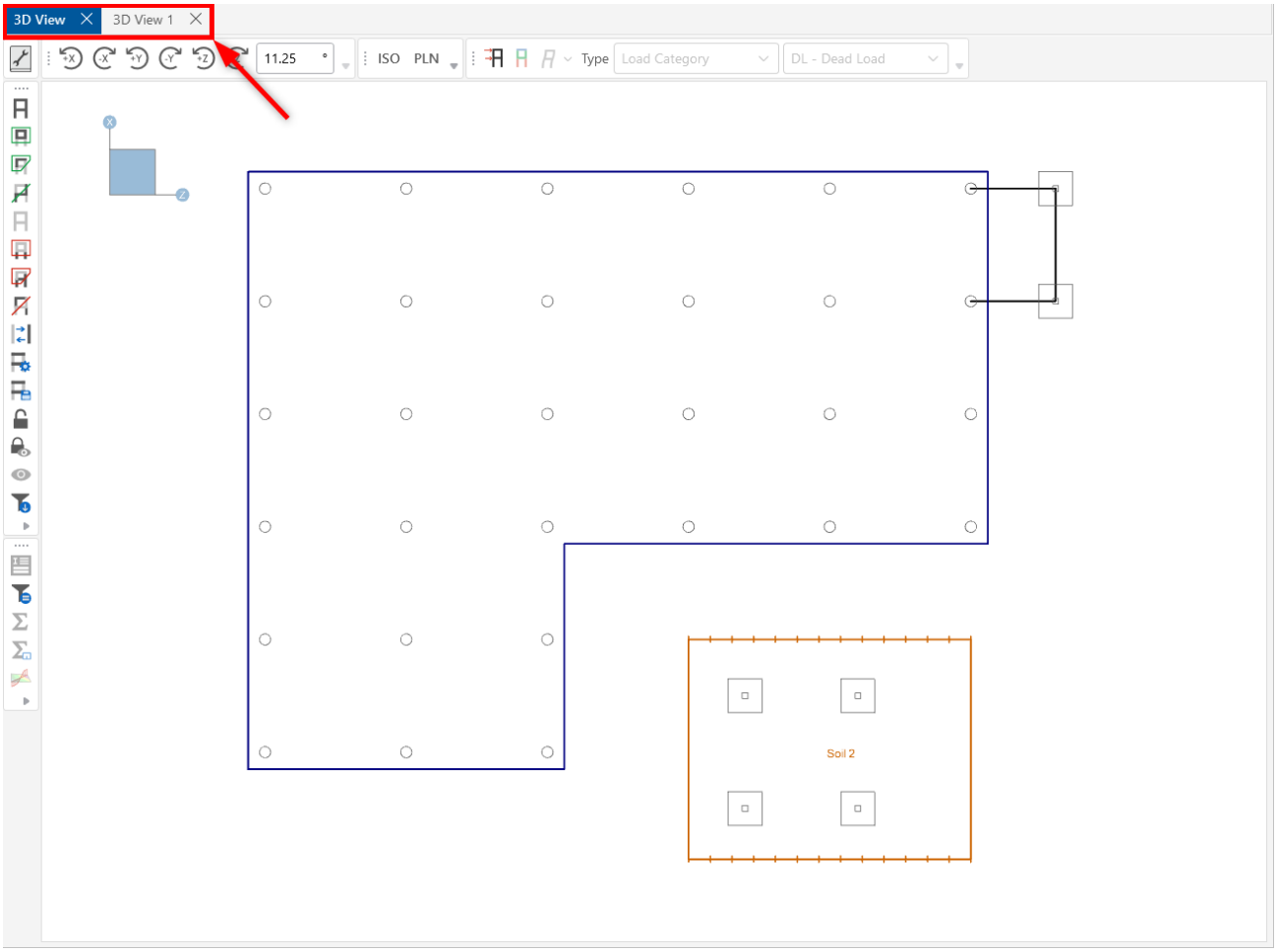
Window Tiling

Standard window tilings help you set up your workspace. With any 3D view or spreadsheet, you can click the blue bar at the top of the window and drag the window into a configuration that best suits your needs. When dragging a window, a circle with four arrows displays, as shown below.



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

By placing windows over the dot within other windows, the windows tab together. See the below example where 3D View 1 and 3D View have been tabbed together.



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

For additional information on working with viewpoint windows, see the Windows Behavior section in Graphic Display.

For information about working with multiple views, see the Multiple Windows section in Graphic Display, Windows Behavior.

Modes

There are three basic program modes (**View, Select** and **Edit**) and a mode hierarchy that allows 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 indicate 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 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 model elements. 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.

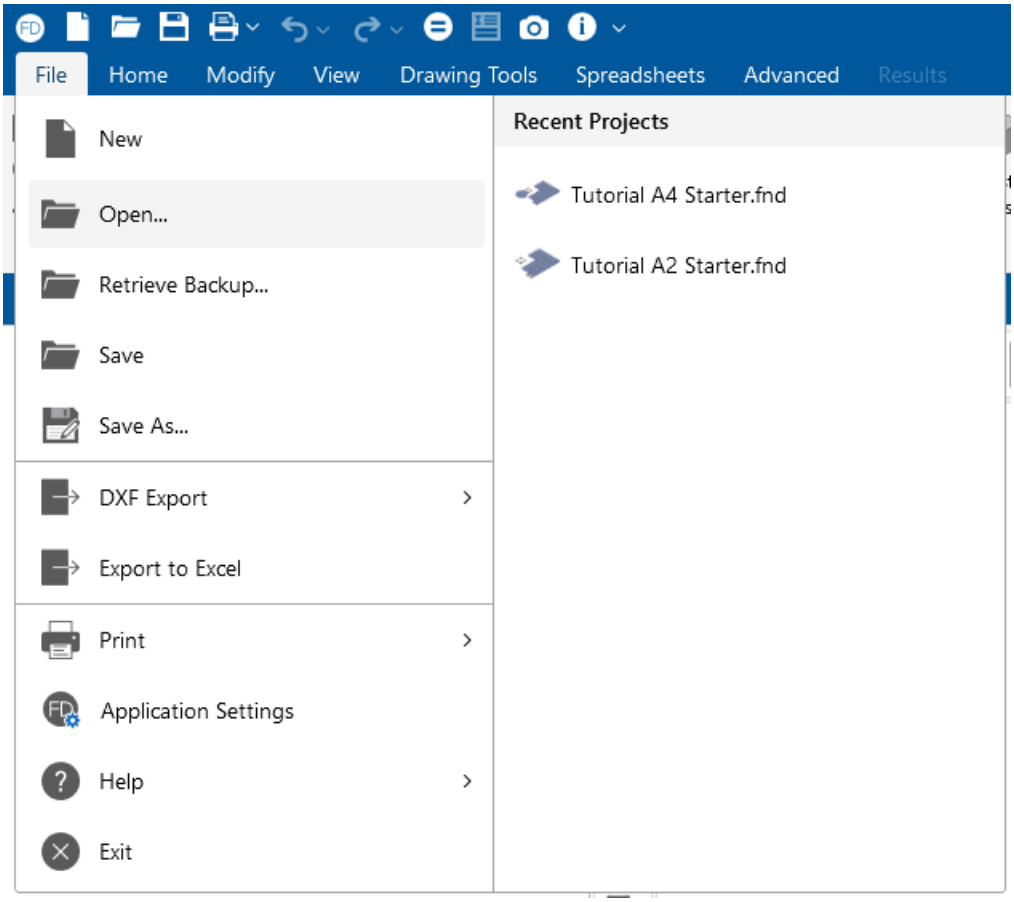
Edit Mode is the lower-level mode that allows you to graphically edit the model. You may make selections and adjust the view while in the edit mode such that when you are finished selecting you will be returned to the **Edit Mode**. Some **Edit Mode** features have options on how you apply the edit.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



Note: The default mode is a selection mode that is indicated by the standard  mouse cursor. While in this mode you can select/deselect individual items by clicking on them. You can view information about the selected item in the Properties panel. Use the ESC key or the right mouse button to cancel an active mode.

File Menu

Clicking the File tab will expand the **File** menu.



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

The following table provides descriptions of all the menu options.

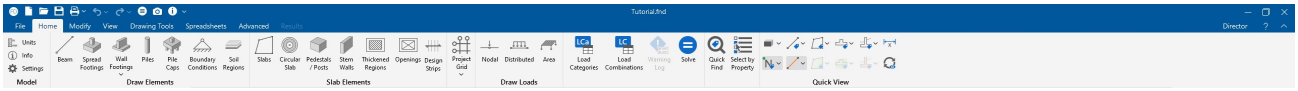
File Menu Icons / Options

Icon	Option	Description
	New	Allows you to close the current file, prompting you to save it, and opens a new file.
	Open	Allows you to close the current file, prompting you to save it, and opens an existing file.
	Retrieve Backup	Allows you to close the current file, prompting you to save it, and allows you to open an available backup file.

Icon	Option	Description
	**Save**	Allows you to save the current file, prompting you for a file name if necessary.
	**Save As**	Allows you to save a copy of the current file as a new file, prompting you for a new file name.
	**DXF Export**	Allows you to export the current file to another file format to communicate with external programs.
	**Print**	Allows you to access RISAFoundation printing options.
	**Application Settings**	Opens an Application Settings window to allow you to view or modify settings that are specific to the application. For more information, see Customizing RISAFoundation.
	**Help**	Allows you to access RISAFoundation supplemental information.
	**RISAFoundation Manual**	Available from the 'Help' sub-menu, this icon opens the RISAFoundation General Reference Manual. This manual has explanations of how the software should/can be used, as well as any limitations. Searchable content helps make this a valuable tool when using the software.
	**RISAFoundation Tutorial**	Available from the 'Help' sub-menu, this icon opens the RISAFoundation Tutorial document. The step-by-step instructions that guide you through the entire process of building a model will cover most of the features in the program, as well as demonstrate several tips and shortcuts.
	**Licensing**	Available from the 'Help' sub-menu, this icon provides license information for subscription users and provides the option to Borrow or Return a license. For additional subscription licensing information, see Subscription Licensing.
	**Check for Updates**	Available from the 'Help' sub-menu, this icon allows you to run an internal check for possible program updates. If your program is up to date, you receive a message saying you are up to date. If you are not using the current version of the software, the check will offer you options of how to update.
	**About RISAFoundation**	Available from the 'Help' sub-menu, this icon allows you to view your RISAFoundation version and hardware key information.

Icon	Option	Description
	**Log Out & Exit**	Allows you to close the current file and logs you out, prompting you to save if necessary. The next time you open the program, you will need to re-enter your subscription license username and password.
	**Exit**	Allows you to close RISAFoundation, prompting you to save if necessary.

Home Ribbon



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

Icon	Tool	Description
Model		
	Units	Allows you to define the units to be used for the input and output data of the model. For more information, see Units.
	Info	Allows you to enter descriptive information such as company and designer name. This information will display when printing reports and graphic output. Some information can also be saved using the 'Set As Default' icon. For more information, see Model Settings.
	Settings	Allows you to modify settings specific to the model which is currently open. Designate the code standard to use for the analysis, input criterion for concrete and seismic analysis and adjust parameters used in the solution, such as mesh size and processor utilization. Information can also be saved using the Set As Default button. For more information, see Model Settings.
Draw Elements		
Beam	**Beam**	Allows you to create new beams by clicking a start and end point. Designate general properties, such as material and shape. Then, draw the beam in your model at the desired location. For more information, see Beams.
Spread Footings	**Spread Footings**	Allows you create a spread footing by clicking a single point. Spread footing properties will be based on the spread footing definition that you assign to the footing. See the Spread Footing Definition spreadsheet to adjust material, geometry, pedestal and soil attributes of the footing.
Wall Footings	**Wall Footings**	Allows you create a retaining wall or strip footing by clicking a start and end point. Wall footing properties will be based on the wall footing definition that is assigned. See the Wall Footing Definitions spreadsheet to adjust the material, geometry and soil

Icon	Tool	Description
		properties of the wall.
 Piles	**Piles**	Allows you to create a pile by clicking a single point. Pile properties will be based on the pile definition that you assign. See the Pile Definitions spreadsheet to adjust material, shape, length and capacities of the pile.
 Pile Caps	**Pile Caps**	Allows you to create a pile cap foundation by clicking on a single point. Pile caps are based on the pile cap definition that you assign. See the Pile Cap Definitions spreadsheet to adjust material, geometry, number of piles and pedestal location.
 Boundary Conditions	**Boundary Conditions**	Allows you to define how the model is externally restrained. The support point(s) can be completely restrained, or partially restrained, by using a Spring with a specified stiffness. For more information, see Boundary Conditions.
 Soil Regions	**Soil Regions**	Allows you to create a soil region by clicking a series of points. Default soil properties will be applied to the entire model based on the default soil within the Soil Definitions spreadsheet. You can draw a soil region for an area of the model that you would like to have different soil properties other than the specified default soil definition.
Slab Elements		
 Slabs	**Slabs**	Allows you to create a mat slab by clicking a series of points. Select a material and thickness to assign to the mat slab. You can then pick from the slab elements in the Home tab to add pedestals, stem walls or openings to the slab.
 Circular Slab	**Circular Slab**	Allows you to quickly create a circular mat slab. Use the generator to specify a slab material, polar origin, radius and thickness to quickly generate a circular slab. There are also circular line and area load generation capabilities within the generator.
 Pedestals / Posts	**Pedestals/Posts**	Allows you to create a pedestal or post by clicking a single point on a slab. Pedestals or posts will transfer load directly to the slab. Pedestals will provide a code check and rebar after solution. Posts will represent an outline of the baseplate on the slab and the slab will be checked for punching shear based on the post outline.

Icon	Tool	Description
	**Stem Walls**	Allows you to create a stem wall by clicking a start and end point on a slab. Stem walls are solely used to transfer load to a mat slab. No design of the stem wall will be done, but the self-weight will be accounted for.
	**Thickened Regions**	Allows you to create a region with a different thickness on a slab by clicking a series of points within the slab. Use this tool to change the thickness of an interior region of a slab. The modified thickness can be lesser than or greater than the already modeled slab thickness.
	**Openings**	Allows you to create an opening within a slab by clicking a series of points within the slab perimeter. Slab openings can be drawn in contact with one perimeter of the slab. Line loads and area loads will not be considered in the analysis of the slab if they are within the perimeter of an opening.
	**Design Strips**	Allows you to create a design strip by clicking a series of points on a slab. Specify the rebar orientation and design rule to be used in the flexural reinforcement design. Design strips may be defined either before or after a solution is performed.
	**Project Grid**	Allows you to add project grids to your model file. Project grids are utilized as a reference to draw new elements in the model. Columns and walls are attached to project grids and will move with the project grid. For more information, see Project Grid.
Draw Loads		
	**Nodal**	Allows you to apply concentrated loads and moments to nodes. Nodal loads may only be applied at node locations. If a nodal load is being added to any beam span location, a new node must be added to that location prior to applying nodal load.
	**Distributed Loads**	Allows you to graphically draw distributed loads in your model. Click on the start and end nodes to define the distributed load. The load can be applied in varying magnitude from start to end. For more information, see Loads - Distributed Loads.
	**Area**	Allows you to apply an area load to slabs. Click on points to define the area to apply the area load. Nodes are not required at the application point.

Icon	Tool	Description
Miscellaneous These tools allow you to access load categories and combinations, and run a solution which will reveal errors if any, in a warning log.		
	**Load Categories**	Opens the Load Categories spreadsheet. The Load Categories spreadsheet groups and summarizes the loads applied to the model.
	**Load Combinations**	Opens the Load Combinations spreadsheet. The Load Combination spreadsheet allows you to define combined load categories with factors applied for solving the model. For more information, see Loads - Load Combinations.
	**Warning Log**	Opens the Warning Log spreadsheet if warnings are available after solution. The Warning Log Spreadsheet provides you with a record of any warnings or errors that occurred during the solution of your model. For more information, see Warning Log.
	**Solve**	Performs the analysis and design calculations to solve the current model file. Solution will be of all selected load combinations that have the "Solve" checkbox checked. For more information, see Solution.
	**Quick Find**	Quick Find performs a quick search to find specific elements in your model by label and add them to the selection state. For more information, see Quick Find.
	**Select by Property**	Allows you to select or unselect items within the model based on specific criteria. Selects or unselects elements based on various parameters for either nodes, beams, slabs, pedestals, spread footings, wall footings, piles, pile caps and design strips. For more information, see Graphic Selection.
Quick View These tools can be used to change the graphical display of the model. You can use these icons to turn on beam and node labels, and view color-coded properties and results. Additional view options can be specified in the View Settings on the **View** ribbon.		
	**Render**	Allows you to toggle through wireframe, color coded wireframe and a rendering of model elements. Rendering is dependent on material type. The color coding criteria can be set in the Model View Settings.

<table>
<tr><th>Icon</th><th>Tool</th><th>Description</th></tr>
<tr><td></td><td></td><td>
<table>
<tr><th>Icon</th><th>Option</th><th>Description</th></tr>
<tr><td></td><td>Wireframe</td><td>Allows you to display elements as wireframe.</td></tr>
<tr><td></td><td>Color Coded</td><td>Allows you to display elements as wireframe and color coded based on the Model Settings.</td></tr>
<tr><td></td><td>Rendered</td><td>Allows you to render elements based on material type.</td></tr>
</table>
For more information, see Model View Settings.
</td></tr>
<tr><td></td><td>Show Node Labels</td><td>Allows you to toggle through the display of nodes and node labels.

Node labels and corresponding node locations can be seen in the Node Coordinates spreadsheet. You can also relabel all nodes from the Node Coordinates spreadsheet.</td></tr>
<tr><td></td><td>Beam Labels</td><td>Allows you to display beam labels when viewing beams in wireframe.

Label options include beam properties, such as shape and material type.</td></tr>
<tr><td></td><td>Beam Color Codes</td><td>Allows you to toggle the display of the various beam parameters using color coding options.

Display options include material, shape group and beam start and end designations.</td></tr>
<tr><td></td><td>Slab Labels</td><td>Allows you to toggle the display of slab labels.

View the slab labels as text. Other label display options include slab properties, such as material, thickness and local axes.</td></tr>
<tr><td></td><td>Slab Contours</td><td>Allows you to toggle the display of slab contours.

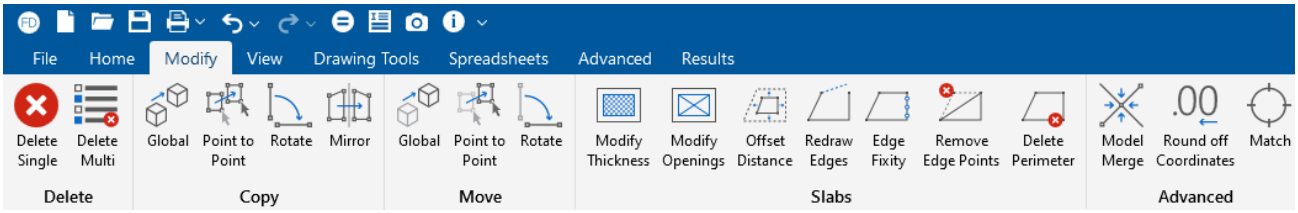
Use the contour legend in the top right of the 3D View to view slab forces and stresses.</td></tr>
<tr><td></td><td>Spread Footing Labels</td><td>Allows you to toggle the display of spread footing labels.

View the footing labels as text. Other label display options include footing properties, such as local axes.</td></tr>
<tr><td></td><td>Spread Forces</td><td>Allows you to toggle the color coded display of spread footing results.

Display options in color coded view, such as bearing ratio, bearing pressure, fixture and stability checks.</td></tr>
</table>

Icon	Tool	Description
	**Wall Footing Labels**	Allows you to toggle the display of wall footing labels. View the footing labels as text. Other label display options include footing properties such as local axes.
	**Wall Footing Color Code**	Allows you to toggle the color coded display of wall footing results. Display options include color coded view, such as bearing ratio, bearing pressure, fixture and stability checks.
	**Design Strip Labels**	Allows you to toggle the display of design strip labels when viewing design strips in wireframe. Label options include design strip label and design rules.
	**Design Strip Forces**	Allows you to toggle the display of design strip force diagrams. Click to turn on the graphical display of the design strip out of plane moment or out of plane shear force diagrams.
	**Rebar Labels**	Allows you to toggle the display of rebar labels within a design strip. Click to turn on or off the rebar labels within a design strip. There are also options to set the labels at different span locations.
	**Soil Region Labels**	Allows you to toggle the display of soil region labels. Display options include soil region label and subgrade modulus.
	**Distance Measure Tool**	Allows you to measure the distance between two points. Click on two points in the model to measure the distance. The measurement result is displayed in the bottom-left of the Properties Panel.
	**Reset View**	Clears the currently displayed options and sets the model view back to the default wireframe view.

Modify Ribbon



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

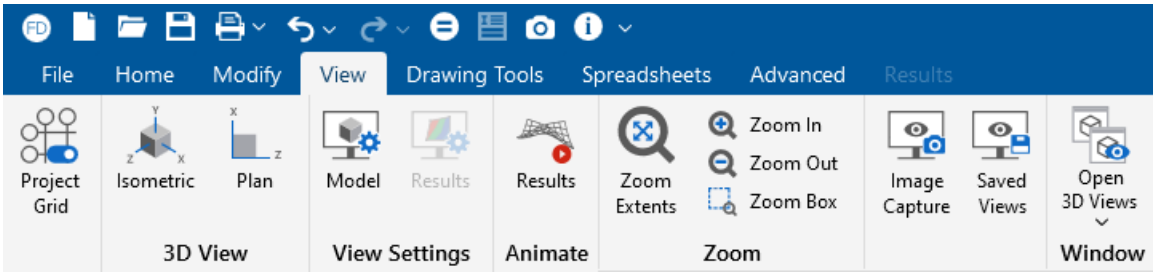
Icon	Tool	Description
Delete These tools provide a way to delete specific elements in your model. For more information, see Delete Elements.		
	**Delete Single**	Allows you to delete a single item in the model. Delete a single item by clicking on it individually in the model view. Alternatively, you can select an element and press the DELETE key on your keyboard to perform the same action.
	**Delete Multi**	Allows you to delete elements within the model based on criteria. Delete selected items based on elements types such as, nodes, unattached nodes, beams, spread footings, wall footings, piles, pile caps, and displayed loads.
Copy These tools provide various ways to copy elements within the model. For more information, see Copy Model Elements.		
	**Global**	Allows you to copy elements in the global X and Z direction. Specify a distance to copy an element in the X and/or Z direction. Click diagonal to apply the distances simultaneously and/or click Copy Loads to also copy associated member loads in the duplicate.
	**Point to Point**	Allows you to copy selected elements by specifying two points for distance and direction. Specify a start point and end point for copying selected elements in the direction and the distance between the two specified points.

Icon	Tool	Description
 Rotate	**Rotate**	Allows you to copy elements by specifying a rotation. Specify the axis of rotation, angle and how many increments to create rotational copies of the element.
Move These tools provide various way to move or rotate elements within the model. For more information, see Move / Rotate Model Elements.		
 Global	**Global**	Allows you to move selected elements in the global X and/or Z direction. Specify a distance to move an element in the global X and Z direction.
 Point to Point	**Point to Point**	Allows you to move selected elements by specifying two points for distance and direction. Specify a start point and an end point for moving selected elements in the direction and the distance between the two specified points.
 Rotate	**Rotate**	Allows you to rotate elements about a specific origin. Specify a rotation axis and angle of rotation. Then, specify an origin by clicking a node in the model or entering specific coordinates.
Slabs These are graphical tools that provide ways to modify slabs within the model. For more information, see Slabs.		
 Modify Thickness	**Modify Thickness**	Allows you to modify and/or delete the edges of slab regions. You can specify whether to move a corner, remove a corner or delete the entire perimeter of the slab region.
 Modify Openings	**Modify Openings**	Allows you to modify and/or delete the edges of slab openings. You can specify whether to move a corner, remove a corner or delete the entire perimeter of the slab opening.
 Offset Distance	**Offset Distance**	Allows you to offset the entire perimeter of a slab. Specify an offset distance to adjust the slab perimeter. You can input a positive number to enlarge the slab or a negative number to minimize the slab.

Icon	Tool	Description
 Redraw Edges	**Redraw Edges**	Allows you redraw an edge of the slab. Select this option to edit an edge of the slab or opening edge in the slab. Click on the edge you would like to edit and you will be able to redraw the edge.
 Edge Fixity	**Edge Fixity**	Allows you to adjust the slab edge fixity to an adjacent slab. Specify the slab edge fixity to either be Fixed or Pinned and then click on the slab edge that you would like to apply the fixity to.
 Remove Edge Points	**Remove Edge Points**	Allows you to delete the corner of the slab. After removing the corner of the slab, the slab perimeter will connect the two adjacent points.
 Delete Perimeter	**Delete Perimeter**	Allows you to delete the entire slab perimeter.
Advanced		
 Model Merge	**Model Merge**	Allows you to merge elements in your model based on your merge tolerance. Model merge can help detect and fix modeling errors. For more information, see Merge Model Elements.
 Round off Coordinates	**Round off Coordinates**	Allows you to specify the number of decimal places for node coordinates. Use this option to indicate the decimal precision used for node coordinates. This is especially useful when importing a model from another program to reduce the possibility of element misalignment due to excessive decimal precision.
 Match	**Match**	Allows you to match the properties from one element to another. Select a base element whose properties will be copied to additional elements. This tool is useful when making mass modifications to the model. For more information, see Match.

View Ribbon

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



The following table provides a short description of each tool.

View - Icons

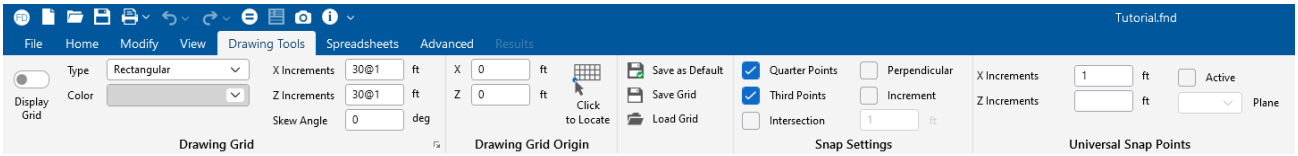
Icon	Tool	Description
	**Project Grid**	Allows you to toggle the display of the Project Grid. Create new project gridlines from the Home tab. Modify existing project gridlines using the Project Grid spreadsheet. For more information, see Project Grid.
3D View Tools for snapping your model to a specific view.		
	**Isometric**	Allows you to snap your model to an isometric view.
	**Plan**	Allows you to snap your model to an XZ planar view.
View Settings These tools let you specify how to plot nodes, beams, footings, piles, slabs, loads and the information that is to accompany them.		
	**Model**	Allows you to select global model display settings. Select display settings for the model display using dialog selection options. Customize the display settings for beams, spread footings, wall footings, piles, slabs and design strips. For more information, see Graphics - Model View Settings.

Icon	Tool	Description
 Results	**Results**	Allows you to select result display settings. Select how to display the results for beams, spread footings, wall footings, piles, slabs and design strips. In addition, the display of reactions or deflections can be adjusted. For more information, see Graphics - Results View Settings.
Animate		
 Results	**Animation Results**	Allows you to view an animation of the model's deflected shape. A new model view window will open, displaying a video animation of the model's behavior. For more information, see Animate Results.
Zoom		
 Zoom Extents	**Zoom Extents**	Allows you to reset the view to display the full model. For more information, see Zoom Extent Tool .
	**Zoom In**	Allows you to zoom the view in for a closer view of the model. Each time the Zoom In icon is clicked, the model view will be magnified. For more information, see Zoom In Tool.
	**Zoom Out**	Allows you to zoom the view out for a wider view of the model. Each time the Zoom Out icon is clicked, the model view expand. For more information, see Zoom Out Tool.
	**Zoom Box**	Allows you to zoom in on a specific area of the view using a box selection. Select an area to zoom into using a box selection tool. Click on one point and hold your mouse button down as you drag down to the opposite corner of the box. Once you lift up the mouse button, your area will be selected and the view zoomed in. For more information, see Zoom Box Tool.
Miscellaneous		
 Image Capture	**Image Capture**	Allows you to capture and save an image of the current graphical view. Create custom snapshot images that can be added to a printed report. For more information, see Capture a Custom Snapshot. For more information on printing your image captures, see

Icon	Tool	Description
		Printing.
	**Saved Views**	Allows you to save or recall a model view. Save or recall specific selection states for a model view. For more information, see Save / Retrieve a Model.
Window		
	**Open 3D View**	Allows you to open or add a new 3D View. Add a new 3D View window or choose from a list of any other 3D View window that is open. For more information, see Open a New 3D View.

For more details about the various options on the View ribbon, see the Graphic Display section.

Drawing Tools Ribbon



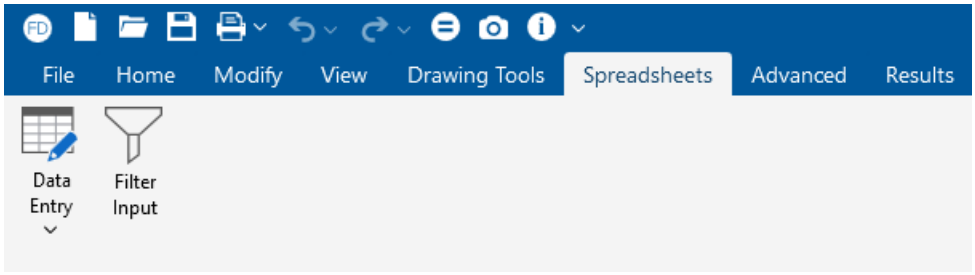
The **Drawing Tools** ribbon provides options to aid drawing elements in the model.

Icon	Tool	Description
Drawing Grid		
Display Grid	**Display Grid**	Allows you to toggle the display of the drawing grid.
Drawing Grid Origin		
Click to Locate	**Click to Locate**	Allows you to position the origin of the drawing grid by selecting a point in the 3D View.
Drawing Grid Options		
	Save as Default	Allows you to save the existing drawing grid as the default grid.
	Save Grid	Allows you to save the existing drawing grid.
	Load Grid	Allows you to load a previously saved drawing grid.
Snap Settings		
	Quarter Points	Allows you to create snap points at quarter intervals along a beam. A check indicates that this feature is turned on. To check or uncheck, click inside the Quarter Points box.
	Third Points	Allows you to create snap points at third intervals along a beam. A check indicates that this feature is turned on. To check or uncheck the box, click inside the Third Points box.
	Intersection	Allows you to create a snap point where elements intersect. A check indicates that this feature is turned on. To check or uncheck the box, click inside the Intersection box.
	Perpendicular	Allows you to create a snap point to draw perpendicular beams.

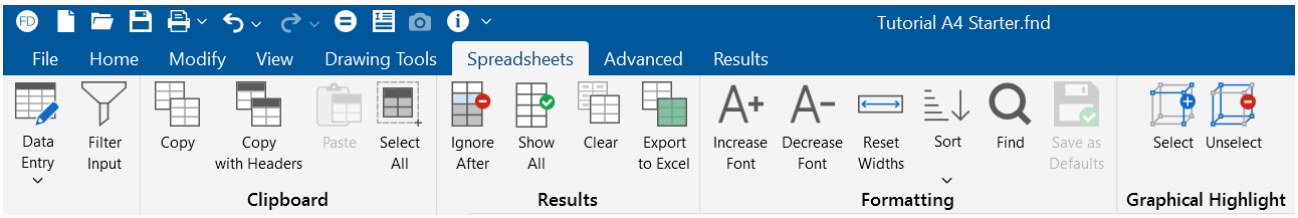
Icon	Tool	Description
		A check indicates that this feature is turned on. To check or uncheck the box, click inside the Perpendicular box.
☐	**Increment**	Allows you to create snap points based on a specified distance. A check indicates that this feature is turned on. To check or uncheck the box, click inside the Increment box.
Universal Snap Points		
☐	**Active**	Allows you to activate snap points in the model based on universal snap point options. A check indicates that this feature is turned on. To check or uncheck the box, click inside the Active box.

For more details about the various options on the Drawing Tools ribbon, see Graphic Editing.

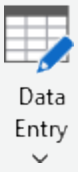
Spreadsheet Ribbon



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



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

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

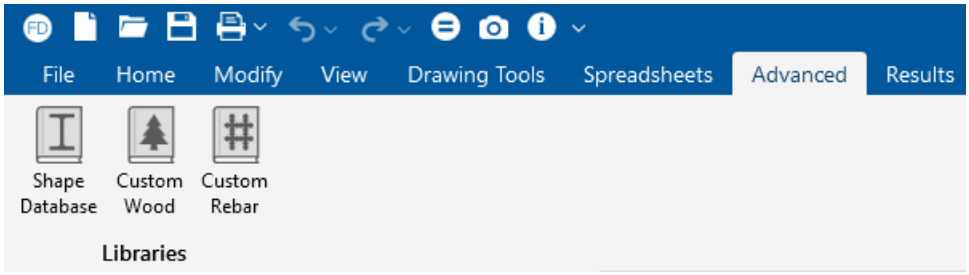
Icon	Tool	Description
 Add Row	**Add Row**	Allows you to add a single row. The tool adds a new row to the bottom of the spreadsheet.
 Insert Row	**Insert Row**	Allows you to insert a single row. The tool inserts a row above the currently selected row in the spreadsheet.
 Delete Row	**Delete Row**	Allows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

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

Icon	Tool	Description
 Reset Widths	**Reset Widths**	Allows you to reset the width of the columns in the spreadsheet. When selected, this tool resets the width of the columns in the spreadsheet to the program defaults.
 Sort	**Sort**	Allows you to sort information/data by a selected column in the spreadsheet. Select a column header in a spreadsheet and click this tool. Then, choose high to low or low to high to reorganize the spreadsheet based on the sort selection in that column.
 Find & Replace	**Find & Replace**	Allows you to search for and replace selected information. This tool searches for the data in the current spreadsheet and replaces it with the selected data. This tool is only available when working with a Data Entry spreadsheet.
 Save as Defaults	**Save as Defaults**	Allows you to save common entries as the default values in a spreadsheet so that they are readily accessible for new, future models.
Graphical Highlight These tools are only available when you select one or more rows in a spreadsheet.		
 Select	**Select**	Allows you to select marked lines in the current model view. You can quickly select elements in the model view that correspond to the marked lines in the spreadsheet.
 Unselect	**Unselect**	Allows you to unselect the marked lines in the current model view. You can quickly unselect elements in the model view that correspond to the marked lines in the spreadsheet.

For more details about the various options on the Spreadsheets ribbon, see Spreadsheet Operations.

Advanced Ribbon

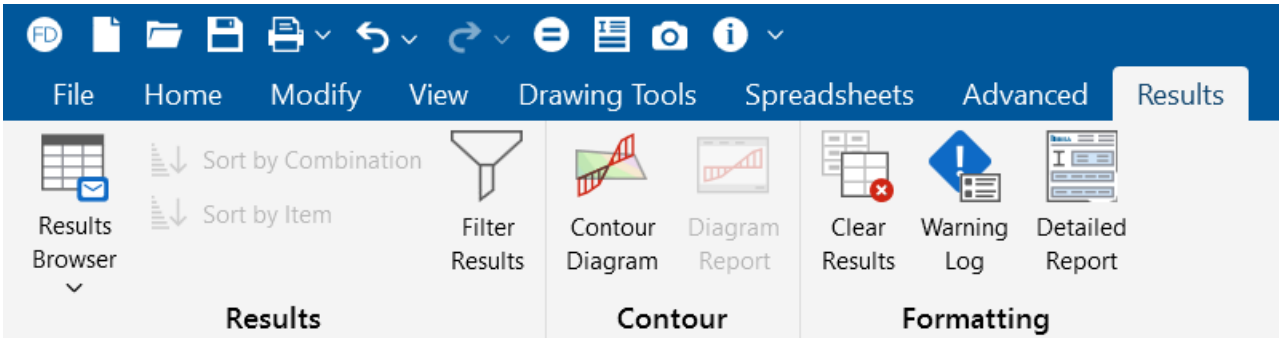


The following table provides a list of the various tools on the Advanced ribbon and their descriptions.

Icon	Tool	Description
 Shape Database	**Shape Database**	Allows you to view the Shape Database. The Shape Database contains hot rolled steel, wood, concrete, and general shapes and their properties. View, edit, or create a shape to use in the model. For more information, see Shape Database.
 Custom Wood	**Custom Wood**	Allows you to view the properties for Custom Wood species. Modify customized wood properties in this database, which can then be used as the Custom wood material found in the Materials > Wood spreadsheet. For more information, see Wood Design.
 Custom Rebar	**Custom Rebar**	Allows you to view Custom Rebar layouts. Create custom flexural or shear rebar layouts to be used in the analysis of concrete columns or beams. For more information, see Concrete Design.

Results Ribbon

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



The following table provides a short description of each tool.

Icon	Tool	Description
Results		
Results Browser	**Results Browser**	Allows you to select to open one or more spreadsheets from a list of all the Results spreadsheets. The tool displays a list of all the Results spreadsheets and whether or not they are currently open. A check in the checkbox indicates that the spreadsheet is open.
	Sort by Combination	Allows you to sort the spreadsheet results of a batch solution by load combination. Results are listed in chronological order, based on load combination. For example, all the member forces under Load Combination 1 are listed first. The forces under Load Combination 2 are listed next.
	Sort by Item	Allows you to sort the spreadsheet results of a batch solution by item. For example, you can group results from each load combination under a particular member.
Filter Results	**Filter Results**	Allows you to filter selected elements from the spreadsheet results. Choose to filter out the results for selected, or unselected, elements from the results spreadsheets. This is useful when you would like to focus on the results for a specific portion of the model.

Icon	Tool	Description
Results		
 Contour Diagram	**Contour Diagram**	Displays a diagram for stresses and/or forces along a defined cut in Plates or Walls. Select two points within a slab to display a diagram representation of color contours for stresses and/or forces along a defined cut in the slab.
 Diagram Report	**Diagram Report**	Displays an enlarged image of the Contour Diagram. Use this tool to view a graphical report that shows the contours as diagrams along with numerical data for every section along the length of the diagram.
Formatting		
 Clear Results	**Clear Results**	Allows you to delete all solution results. This tool clears all solution results to make changes in the model or solve another solution.
 Warning Log	**Warning Log**	Opens the Warning Log spreadsheet, which provides you with a record of any warnings or errors that occurred during the solution of your model. For more information, see Warning Log.
 Detailed Report	**Detailed Report**	Allows you to access detailed analysis and design information for elements. Click on an element to view all analysis and design results pertaining to that specific element. View shear, moment and stress diagrams, as well as design information and code checks.

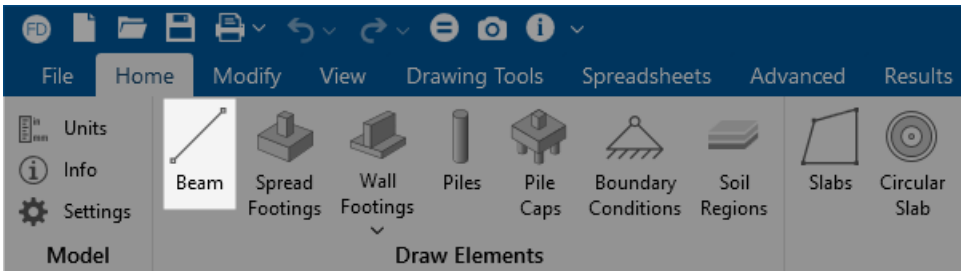
For more details about the various options on the Results ribbon, see Results.

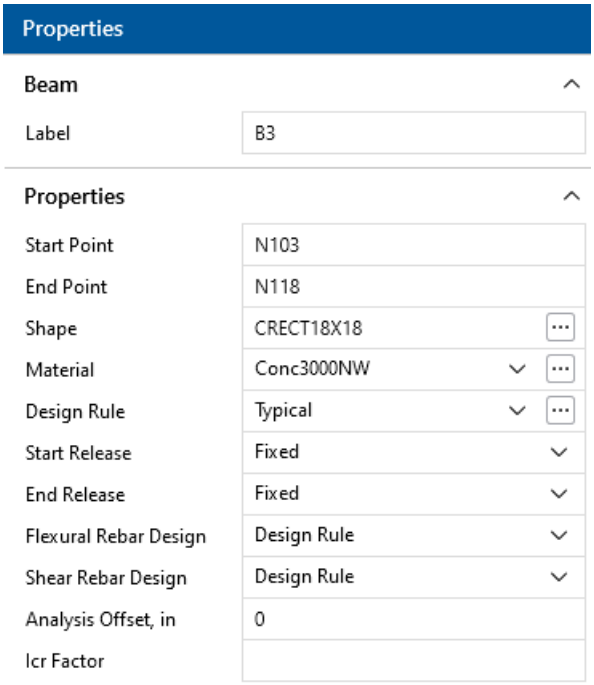
Beams

RISAFoundation uses a Physical Member that is automatically sub-meshed into general-purpose beam elements. Beam data may be viewed and edited in two ways: graphically in a model view or in the **Beams Spreadsheet**. For descriptions of the beam data, see the Beams Spreadsheet section. Concrete design parameters are recorded on the Concrete tab of the **Beams Spreadsheet**, discussed in Beams - Design.

Draw Beams

To create new beams, you can draw them using a Drawing Grid, Project Grid or drawing point-to-point clicking existing nodes. You can set all of the beam properties up front, or you can modify these properties after you draw the beam. Graphically modifying properties is discussed in the next section.





To draw beams:

1. If there is not a model view already open, go to the **View** ribbon.
2. Click the **Open 3D Views** icon in the **Window** section to open a new view.
3. If you are not drawing between existing nodes, you need to create a drawing grid or define nodes in the **Node Coordinates Spreadsheet**.
4. Go to the **Home** ribbon.
5. Click the **Beam** icon in the **Draw Elements** section.
6. In the **Properties** panel, set the beam properties.
7. To start drawing beams, click on **nodes** or **grid intersections** with the left mouse button. The coordinates of the closest node or grid intersection to your cursor are displayed on the screen near your cursor.

   The first click defines the Start Point of the first beam. The second click, and each click thereafter, defines the End Point of the first beam and also the Start Point of the next beam so that you can continue to draw. To stop drawing, click the **right mouse button**. You can then start drawing somewhere else with the left button.

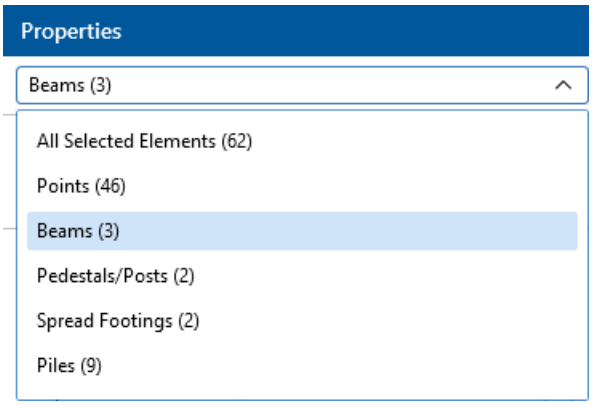
   The new beams are shown on screen and recorded in the **Beams Spreadsheet**.
8. To stop drawing altogether, click the **right mouse button** again or press the **Esc** key.

Note:
- You may also specify or edit beams in the **Beams Spreadsheet**.
- You may undo any mistakes by clicking the **Undo** icon in the **Quick Access toolbar**.

Modify Beam Properties

There are a number of ways to modify beams. You may view and edit the beam data in the Beams Spreadsheet. You may also click a single member or select multiple members to view and edit the properties in the **Properties** panel. You can use the tools in the **Modify** tab to graphically modify the beams in the 3D View.

In this section, we discuss the **Properties** panel for beams, which lets you modify the properties of beams that already exist in your model. For more information about selecting elements, see Graphic Selection.



To Modify Beams

To modify beams properties:

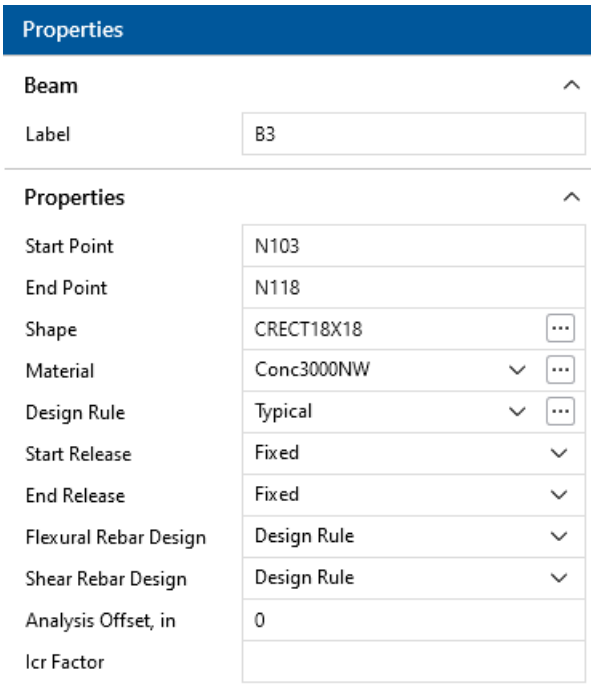
1. If there is not a model view already open, go to the **View** ribbon.
2. Click the **Open 3D Views** icon to open a new view.
3. Click a single beam, or select multiple beams that you wish to modify.
4. If you have multiple elements selected, specify which element type to view properties for using the **Selection Properties** drop-down in the **Properties** panel.
5. Set the parameters for the beams within the **Properties** panel.

   If certain properties vary within your selection of beams, the input field will show 'Various'.

Note: You may undo any mistakes by clicking the **Undo** icon.

Beam Properties

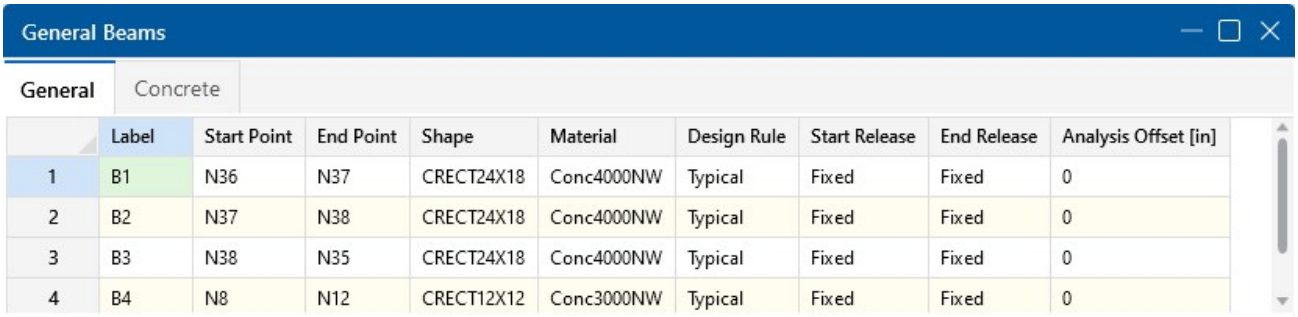
The beam label displays at the top of the **Properties Panel**. Below this are the general properties where you can specify beam parameters, such as end releases, material and shape size.



Note: You may also modify beams in the **Beams Spreadsheet**.

Beams Spreadsheet - General Data

The **Beams Spreadsheet** records the properties for the beam elements and may be accessed by selecting **Beams** from the **Data Entry** drop-down on the **Spreadsheets** tab of the ribbon, or by clicking **Beams** from the **Data Entry** section of the **Explorer** panel.



	Label	Start Point	End Point	Shape	Material	Design Rule	Start Release	End Release	Analysis Offset [in]
1	B1	N36	N37	CRECT24X18	Conc4000NW	Typical	Fixed	Fixed	0
2	B2	N37	N38	CRECT24X18	Conc4000NW	Typical	Fixed	Fixed	0
3	B3	N38	N35	CRECT24X18	Conc4000NW	Typical	Fixed	Fixed	0
4	B4	N8	N12	CRECT12X12	Conc3000NW	Typical	Fixed	Fixed	0

The following data columns hold the **General** data for the beams:

Beam Labels

You may assign a unique **Label** to any or all of the beams. You can then refer to the beam 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.

Beam Start and End Points

The **Start Point** and **End Point** entries define the start and end locations of the beams using node labels.

Shape

The **Shape** entry is used to set the beam size. To enter a shape, type the shape name directly or click the ··· (ellipsis) button in the cell to access the **Shape Selection** dialog. For more information, see Beams - Database.

Material

The **Material** column displays the concrete material used for each beam. Available materials can be selected from the drop-down list, which contains all entries in the **Materials** spreadsheet. For more information, see Material Properties.

Design Rules

The **Design Rules** entry displays the selected design rule for each beam. Available design rules may be selected from the drop-down list, which contains all entries in the **Design Rules Spreadsheet**. For more information, see Design Optimization.

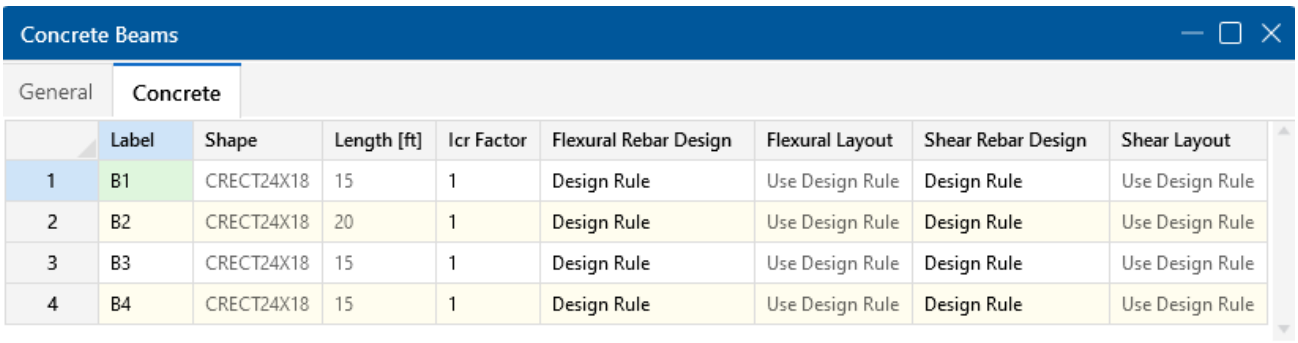
Start and End Releases

The **Start** and **End** release columns allow you to set the beam end releases to **Fixed** or **Pinned**. This determines how the beam connects to other elements at the end nodes.

Analysis Offset

This is a positive value that moves the beam downward in relation to any other element in the model. This not just a graphics change; the program is internally adjusting the location of the beam and connecting it to surrounding elements with rigid links. You can expect to see different distribution of forces due to the offset.

Beams Spreadsheet - Concrete Data



	Label	Shape	Length [ft]	Icr Factor	Flexural Rebar Design	Flexural Layout	Shear Rebar Design	Shear Layout
1	B1	CRECT24X18	15	1	Design Rule	Use Design Rule	Design Rule	Use Design Rule
2	B2	CRECT24X18	20	1	Design Rule	Use Design Rule	Design Rule	Use Design Rule
3	B3	CRECT24X18	15	1	Design Rule	Use Design Rule	Design Rule	Use Design Rule
4	B4	CRECT24X18	15	1	Design Rule	Use Design Rule	Design Rule	Use Design Rule

The following data columns hold the **Concrete** data for the beams:

Icr Factor (Cracked Moment of Inertia Factor)

The **Icr Factor** is used to reduce the bending stiffness of beams. If this entry is left blank, the Icr factor defaults to a value of 0.35.

Note: The **Icr Factor** is ignored if the **"Use Cracked Sections"** box is not checked on the **Concrete** tab in the **(Global) Model Settings** window.

Service Level Stiffness

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

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

Flexural Layout

The **Flexural Layout** entry displays the selected custom rebar layout for beam flexural reinforcement. When the **Flexural Rebar Design** column is set to **Custom**, available layouts can be selected from the drop-down list which contains all **Bending Rebar Layouts** from the **Rebar Layout Database**. For more information, see Custom Rebar Layout Database. When **Design Rule** is selected in the **Flexural Rebar Design** column, RISAFoundation will select the reinforcement for the beam based on the specified design rule.

Shear Layout

The **Shear Layout** entry displays the selected custom rebar layout for beam shear reinforcement. When the **Shear Rebar Design** column is set to **Custom**, available layouts may be selected from the drop-down list, which contains all **Shear Rebar Layouts** from the **Rebar Layout Database**. For more information, see Custom Rebar Layout Database. When **Design Rule** is selected in the **Shear Rebar Design** column, RISAFoundation will select the reinforcement for the beam based on the specified design rule.

Physical Members

Physical Members provide fixity to all joints that occur along the length of the member, without breaking that member into multiple smaller members. You may use the physical member feature to

avoid defining one 'field' member with multiple members in your model. This saves time in building and editing your model and in understanding your results.

Continuous beams are examples of continuous 'field' members that can be modeled with one Physical Member. You may define them from start to finish without having to explicitly define their connection to other elements or boundaries through intermediate points. Subsequently, when it comes time to make changes to a beam you can edit the properties of one Physical Member rather than the multiple beams that might otherwise represent it.

Physical Members are also effective in managing results because the results for one Physical Member are reported together in the results spreadsheets and the beam detail report.

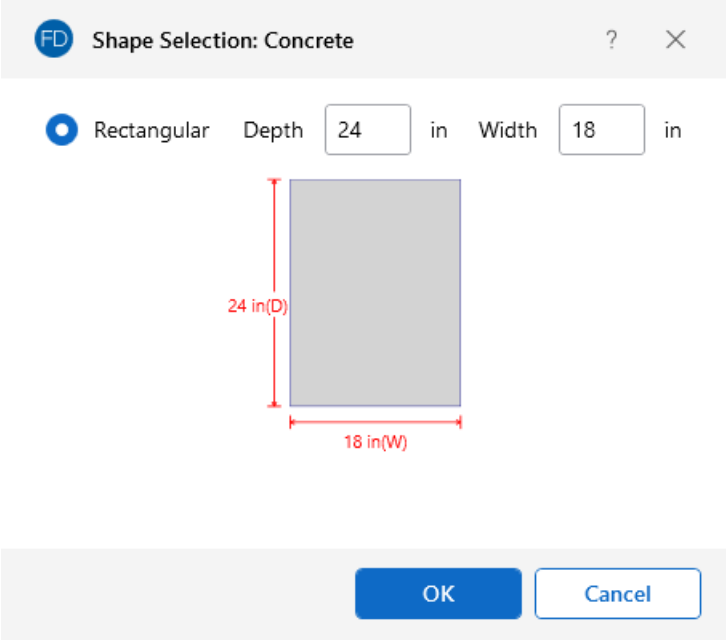
Note: You cannot convert multiple beams into one Physical Member. To achieve this you must delete the multiple beams and define a new Physical Member.

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

Beams - Database

Currently, only rectangular concrete shapes are supported in RISAFoundation. If you're familiar with the shape definitions, you can type the name directly into the appropriate field.

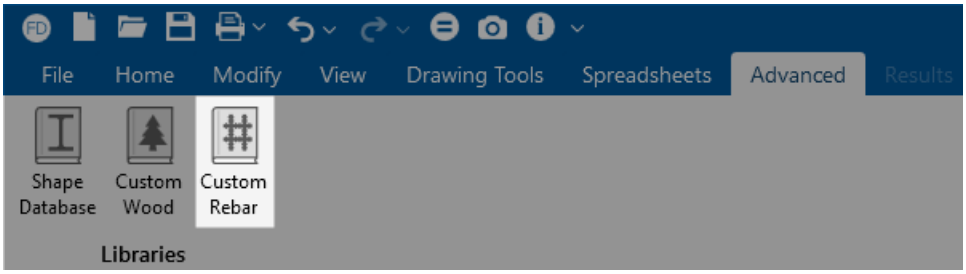
Alternatively, you can click the ··· (ellipsis) button to have the program generate the desired shape definition for you.



Rectangular Sections

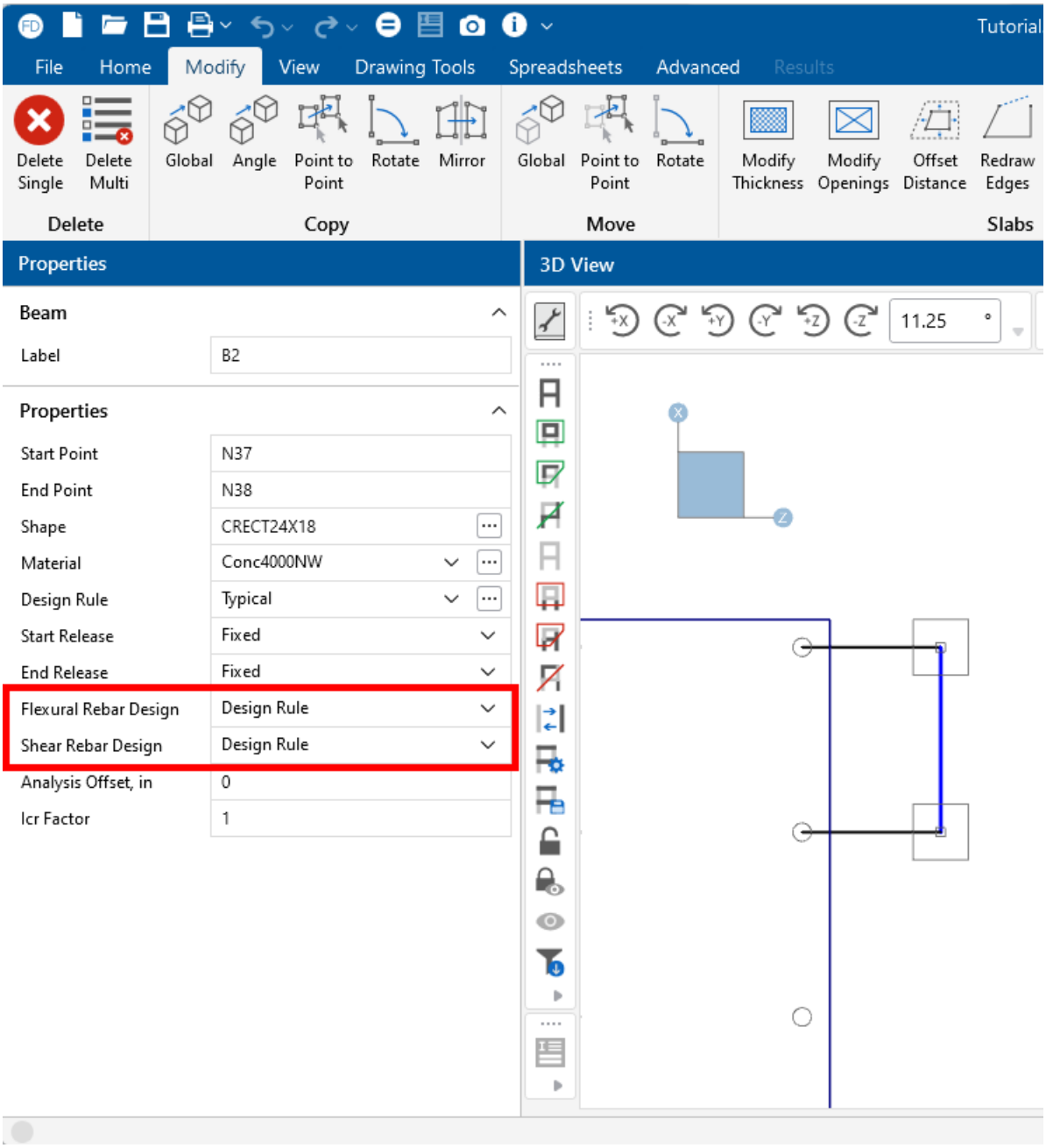
Rectangular sections are defined using a parametric shape code, since a rectangular shape may be any depth or width. The code is CRECT'depth'X'width', where 'depth' and 'width' are the values in the current dimension units. For example, if you wanted a beam that was 18" deep and 12" wide, you would enter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

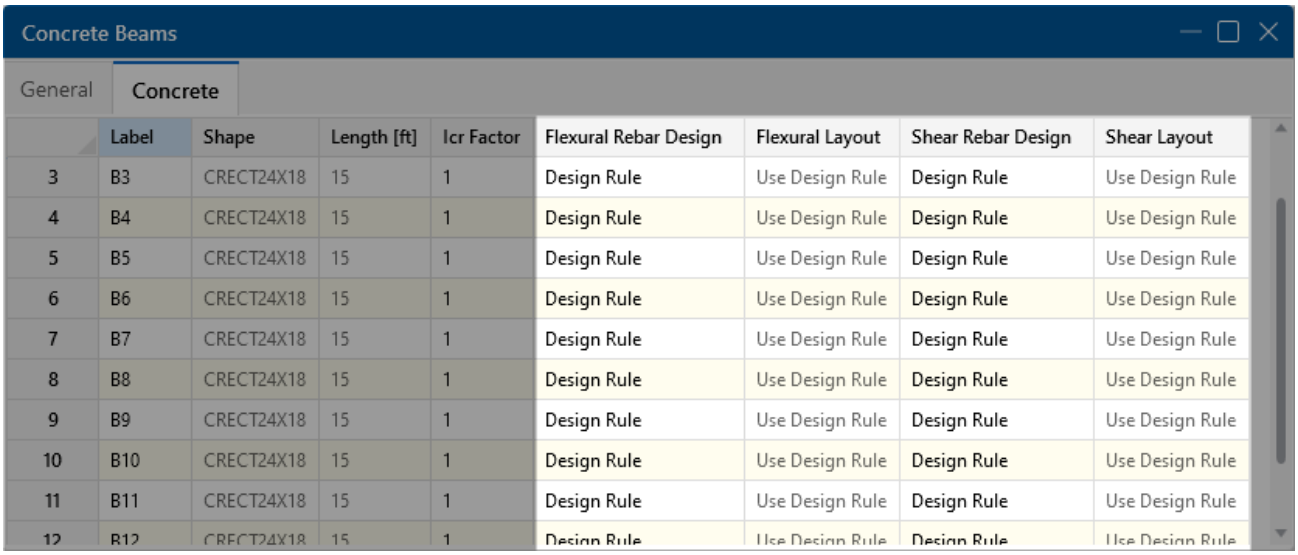
Custom Rebar Layout Database



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

These reinforcement layouts can be assigned to beam members in the **Beams Spreadsheet** and **Properties Panel** in the same way as the other concrete design parameters are assigned.


Tutorial
File
Home
Modify
View
Drawing Tools
Spreadsheets
Advanced
Results
Delete Single
Delete Multi
Delete
Global
Angle
Point to Point
Rotate
Mirror
Copy
Global
Point to Point
Rotate
Move
Modify Thickness
Modify Openings
Offset Distance
Redraw Edges
Slabs
Properties
Beam
Label
B2
Properties
Start Point
N37
End Point
N38
Shape
CRECT24X18
Material
Conc4000NW
Design Rule
Typical
Start Release
Fixed
End Release
Fixed
Flexural Rebar Design
Design Rule
Shear Rebar Design
Design Rule
Analysis Offset, in
0
Icr Factor
1
3D View
11.25



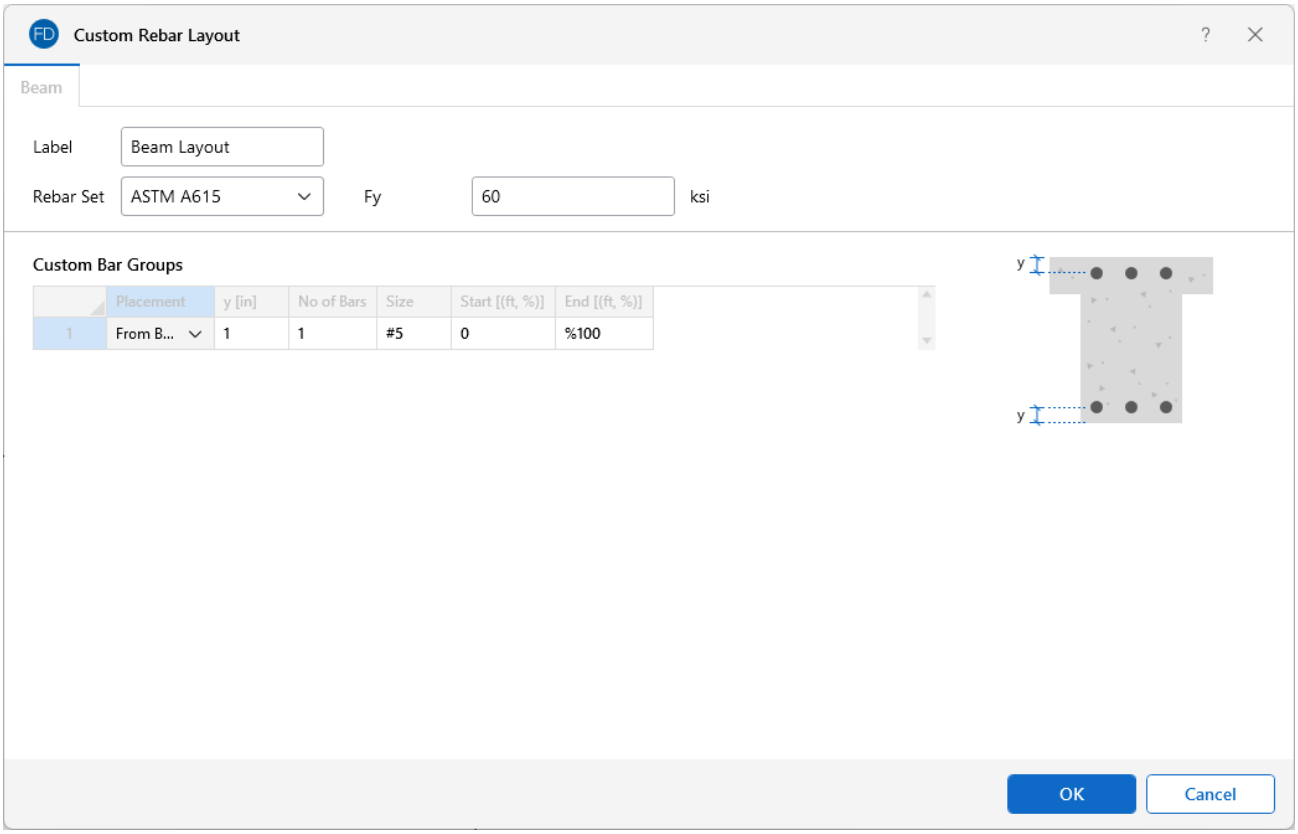
	Label	Shape	Length [ft]	Icr Factor	Flexural Rebar Design	Flexural Layout	Shear Rebar Design	Shear Layout
3	B3	CRECT24X18	15	1	Design Rule	Use Design Rule	Design Rule	Use Design Rule
4	B4	CRECT24X18	15	1	Design Rule	Use Design Rule	Design Rule	Use Design Rule
5	B5	CRECT24X18	15	1	Design Rule	Use Design Rule	Design Rule	Use Design Rule
6	B6	CRECT24X18	15	1	Design Rule	Use Design Rule	Design Rule	Use Design Rule
7	B7	CRECT24X18	15	1	Design Rule	Use Design Rule	Design Rule	Use Design Rule
8	B8	CRECT24X18	15	1	Design Rule	Use Design Rule	Design Rule	Use Design Rule
9	B9	CRECT24X18	15	1	Design Rule	Use Design Rule	Design Rule	Use Design Rule
10	B10	CRECT24X18	15	1	Design Rule	Use Design Rule	Design Rule	Use Design Rule
11	B11	CRECT24X18	15	1	Design Rule	Use Design Rule	Design Rule	Use Design Rule
12	B12	CRECT24X18	15	1	Design Rule	Use Design Rule	Design Rule	Use Design Rule

These reinforcement layouts may also be assigned to pedestals in the **Pedestal / Pile Spreadsheets** or in the **Properties Panel**.

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**.

Bending Rebar Layouts

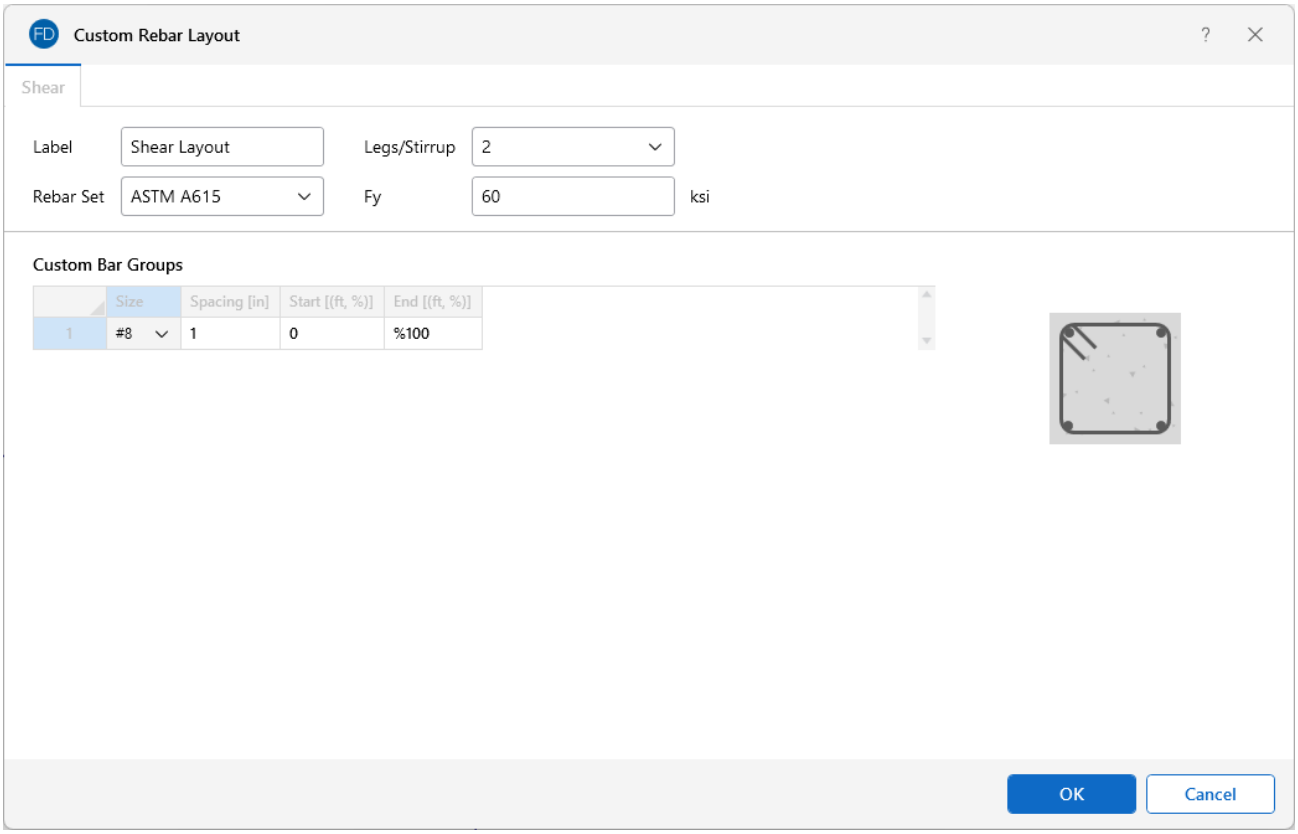
Since beams are only designed for uni-axial bending, the only requirements for the beam layouts are that you specify the depth at which the bars are located, as well as the size and number of the bars that are present at that depth. You can specify the depth with respect to the top or bottom surface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



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.

Shear Rebar Layouts

Shear rebar layouts are specified by giving the size and spacing of the bars, as well as the location on the beam where that reinforcement will be present.



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 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 beam, the start location should be '0' and the end location should be '%100', as shown in the dialog above.

Rectangular Pedestal Rebar Layouts

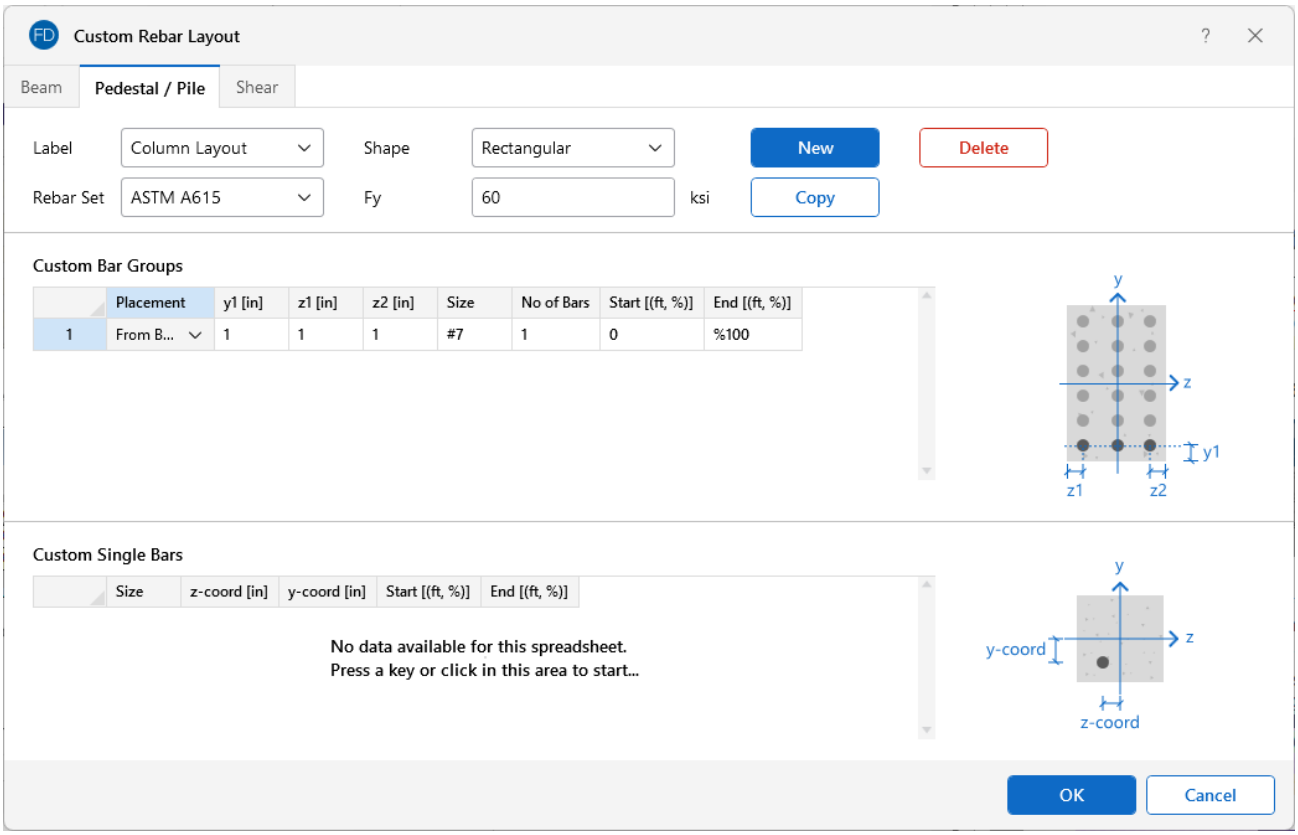
Since pedestals are designed for bi-axial bending, they require more information about the location and arrangement of the bars.

Normally, column bars are arranged in **layers**. One 'top' and one 'bottom' horizontal layer must **always** be defined, each containing at least two bars. These layers, as well as any additional horizontal layers, will be specified by entering a **y1** value to specify the depth from the top or bottom fiber to the center-line 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 center-line of the reinforcing steel. The number and size of the bars must then be entered. The **z1** and **z2** values are ignored for vertical layers because the bars are assumed to be evenly spaced between the required top and bottom layers referred to previously. If this is not desired, then the side bars should be entered individually as **custom single bars**.

Custom single bars are specified by their y- and z-coordinates, measured from the local y- and z-axis respectively. A positive y-coordinate would place the bar closer to the top fiber and a negative y-coordinate would place the bar closer to the bottom fiber (relative to the local axes of the pedestal). 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 (relative to the local axes of the pedestal).

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 below.

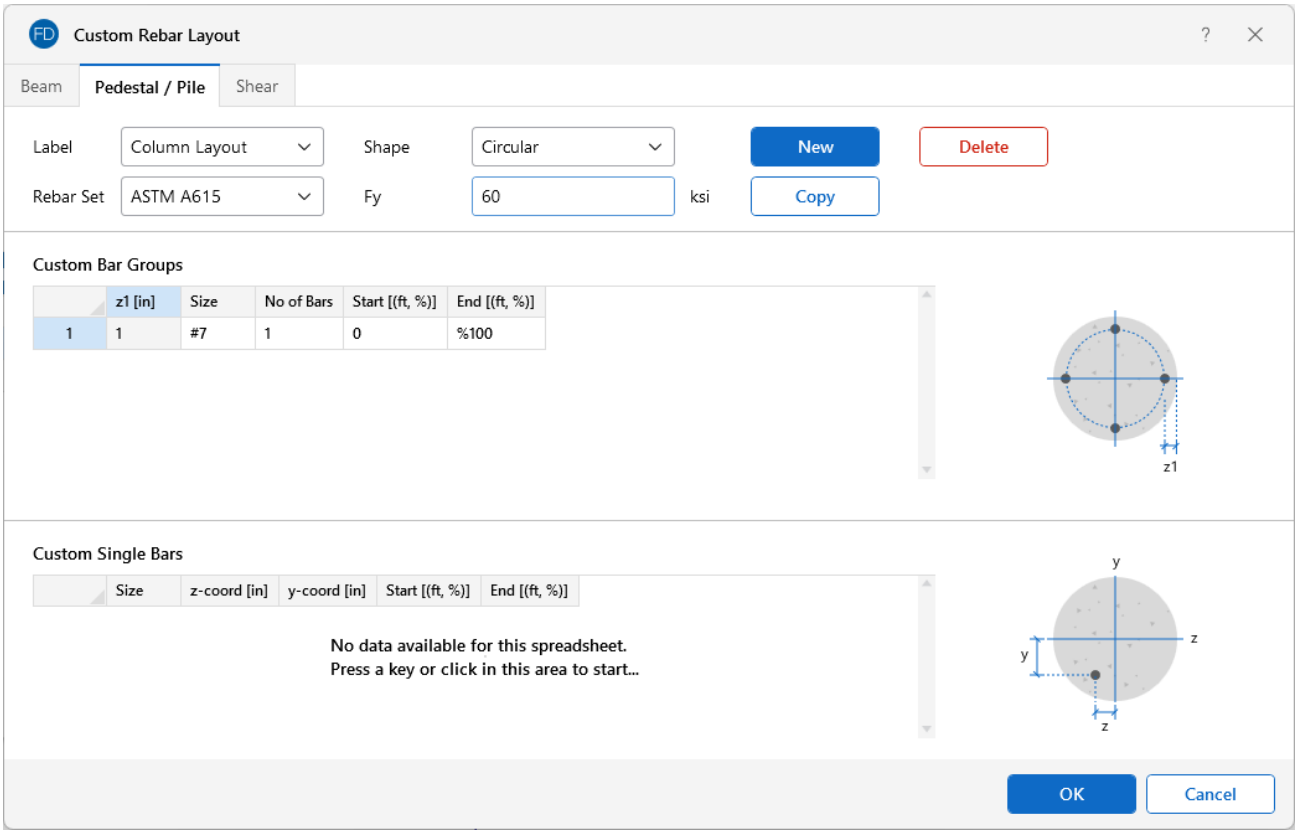


Circular Pedestal Rebar Layouts

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

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

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 below.



Beams - Design

Beam design and optimization can be performed for rectangular concrete shapes based on the following codes:

- The 2022, 2019, 2014, 2011, 2008, 2005, 2002 and 1999 Editions of ACI 318
- The 1992 EuroCode (EC2) and the British publication of the 2014 and 2004 Eurocode (BSEN)
- The 2004 and 2014 Editions of the Canadian code (CSA A23.3)
- The 2004 Edition of the Mexican code (NTC-DF)

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.

The program designs the longitudinal and shear reinforcement for rectangular beams. 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 in the beam detail reports.

Apply a Concrete Design Code

To apply a Concrete Design Code:

1. On the **Design** tab of the 'Model Settings' window, choose the **Concrete** code from the drop-down list.
2. Click **Apply**.

Beam Spans

RISAFoundation automatically breaks 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 is supported. Note that for a physical member to see a support, there must be a point at that support point.

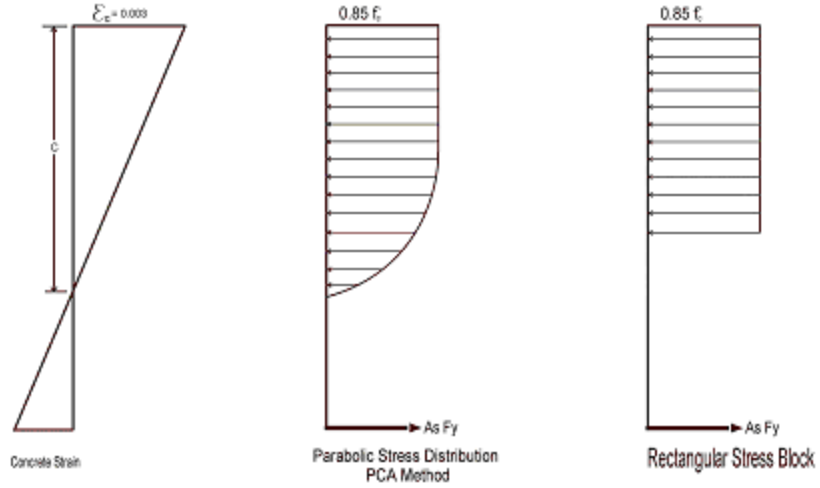
Beam members are supported by the following: Supports (Reaction, Spring, etc.) and other Beam Members that are supporting that member.

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

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.



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

Tension Development Length

- α_{ct}= 1 (assumed in Eq 3.16)
- $\eta_1 = \eta_2$=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 $k^{3/2}$ $f_{ck}^{1/2}$
- v =.6*[1- f_{ck} /250]
- V_{max} is calculated from Eq. 6.5.
- θ is assumed 45 degrees in Eq 6.8.

Limitations - General

Torsion – Beams ignore torsion with respect to the design of shear reinforcement. A message is shown in the detail report to remind you of this.

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

Cracking – The effects of concrete cracking are not considered in the Foundation analysis.

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. 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.

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 can be taken as the sum of the areas of longitudinal bars located more than two-thirds of the overall member depth away from the extreme compression fiber. Therefore, RISA calculates 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 ε_x with the moment and shear at the section taken from the envelope diagrams. The maximum ε_x for each span is conservatively assumed for the entire span. Currently, the program has no option for pre-stressing, so V_p and A_p are both taken as zero.

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

Limitations - 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 beam. 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.

All of the load combinations that were run had the **Service** box checked on the **Load Combinations Spreadsheet**. Since there are no concrete design specific load combinations, there are no results or force diagrams to show.

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

Certain, very short spans in physical members can end up with less than 5 design sections. No design is attempted without at least 5 sections because maximum values can be missed and an unconservative 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: The shear tie spacing does not meet the code Minimum Requirement

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

Rebar spacing exceeds maximum allowable (ACI 318-14 Section 13.3.4.4, ACI 318-11 Section 15.10.4)

The 2008 and newer editions of the ACI 318 forbid the placement of rebar in foundation slabs exceeding 18 inches. The program will still optimize rebar up to the largest spacing specified in the **Design Rules Spreadsheet**, and may violate this ACI provision. Reduce maximum spacing in the Design Rules - Beam Rebar.

Beams - Design Results

Access the **Beam Design Results Spreadsheet** by selecting the **Results** tab ▸ **Results Browser** button ▸ **Beam Code Check**. Alternatively, you can select **Beam Code Check** from the Explorer Panel. Note that concrete results are always based on envelope results.

For beam flexural design, the required bars are based on the envelope moment diagrams. Similarly, the required bars for beam shear steel design are based on the enveloped shear force diagrams.

Beam Results

Beam results are shown in the following spreadsheets: **Beam Design Results** and **Beam Reinforcement**.

Beam Design Results Spreadsheet

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

Beam Design Results (By Item)

	Beam	Shape	UC Max Top	Loc[ft]	LC	UC Max Bot	Loc[ft]	LC	Shear UC	Loc[ft]	LC	Phi*Mnz Top[k-ft]	Phi*Mnz Bot[k-ft]	Phi*Vny[k]
1	B1	CRECT...	0.025	15	2	0.124	3.125	2	0.06	12.031	2	125.099	153.149	36.927
2	B2	CRECT...	0.021	0	2	0.049	10	2	0.048	0	2	125.099	153.149	36.927
3	B3	CRECT...	0.025	0	2	0.13	11.875	2	0.062	0	2	125.099	153.149	36.927

These top and bottom code checks, **UC Max Top** and **UC Max Bot**, are based on the top/bottom moment capacities and maximum top/bottom moment. Currently, the moment capacity is only based

on the tension steel (NO compression steel is considered in the capacity calculation). The governing maximum shear check for all spans, **Shear UC**, is also shown. The capacities shown are only for the governing section. Capacities for each span, as well as beam reinforcement detailing diagrams, may be viewed in the Detail Report.

Beam Bending Reinforcement Spreadsheet

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



	Beam	Shape	Span	Left Top	Left Bot	Mid Top	Mid Bot	Right Top	Right Bot
1	B1	CRECT...	1		4 #6	3 #6	4 #6	3 #6	
2	B2	CRECT...	1		4 #6	3 #6			4 #6
3	B3	CRECT...	1	3 #6	4 #6	3 #6	4 #6		4 #6

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

The **Shape** column displays the beam size.

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 only have 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

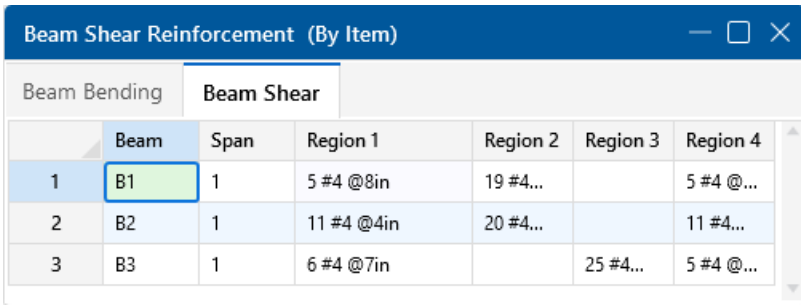
first number indicates the number of parallel reinforcement bars in that section. The second number, preceded by the '#' sign, indicates the size of reinforcement bars used.

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 **Beam Reinforcement** from the **Results Menu**. Results are listed on the **Beam Shear** tab.



The **Beam** 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 **Rebar** 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 the force at which code required reinforcement must be provided.

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

Note:

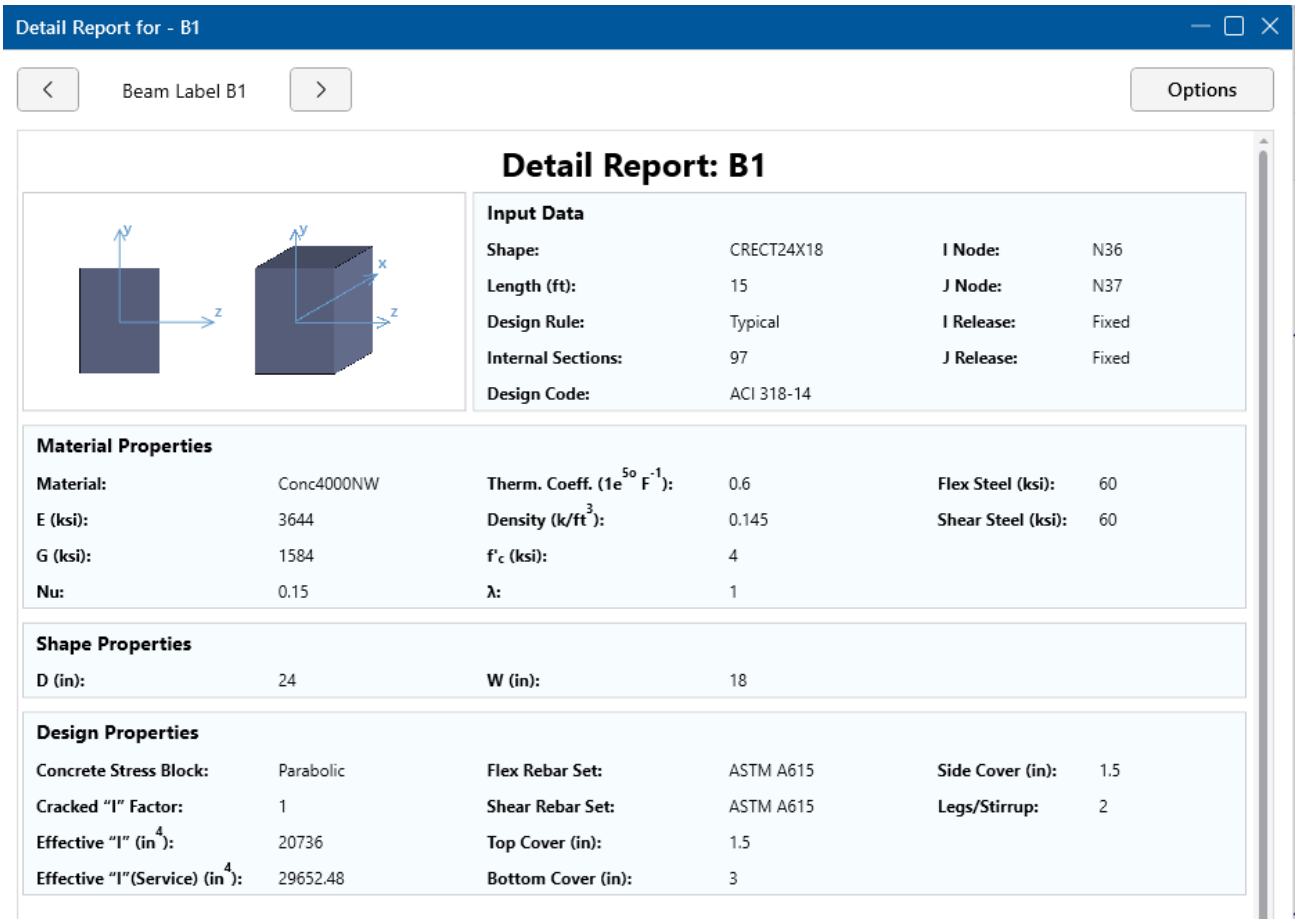
- If '2' shear regions are selected on the **Design** tab of the **Model Settings Dialog**, the Region 2 and Region 3 entries will be left blank in this spreadsheet.
- The concrete code checks are only performed at the sections where the internal forces are calculated. The number of internal force calculations 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.

Beam Detail Reports

The **Beam Detail Reports** allow you to see the overall force state for any particular beam. Detail reports for concrete beams are not based on individual load combinations; instead, they are based on an envelope of the solved load combinations.

Detail reports for concrete beams can, and often do, exceed one page in length due to the large amount of information that must be displayed for concrete design. One reason for this is that RISAFoundation determines the number of spans for concrete beams based on the number of internal supports. Thus, one physical member may have several spans that all must be reported.

The image below is the first portion of a detail report for a concrete beam showing the member information.

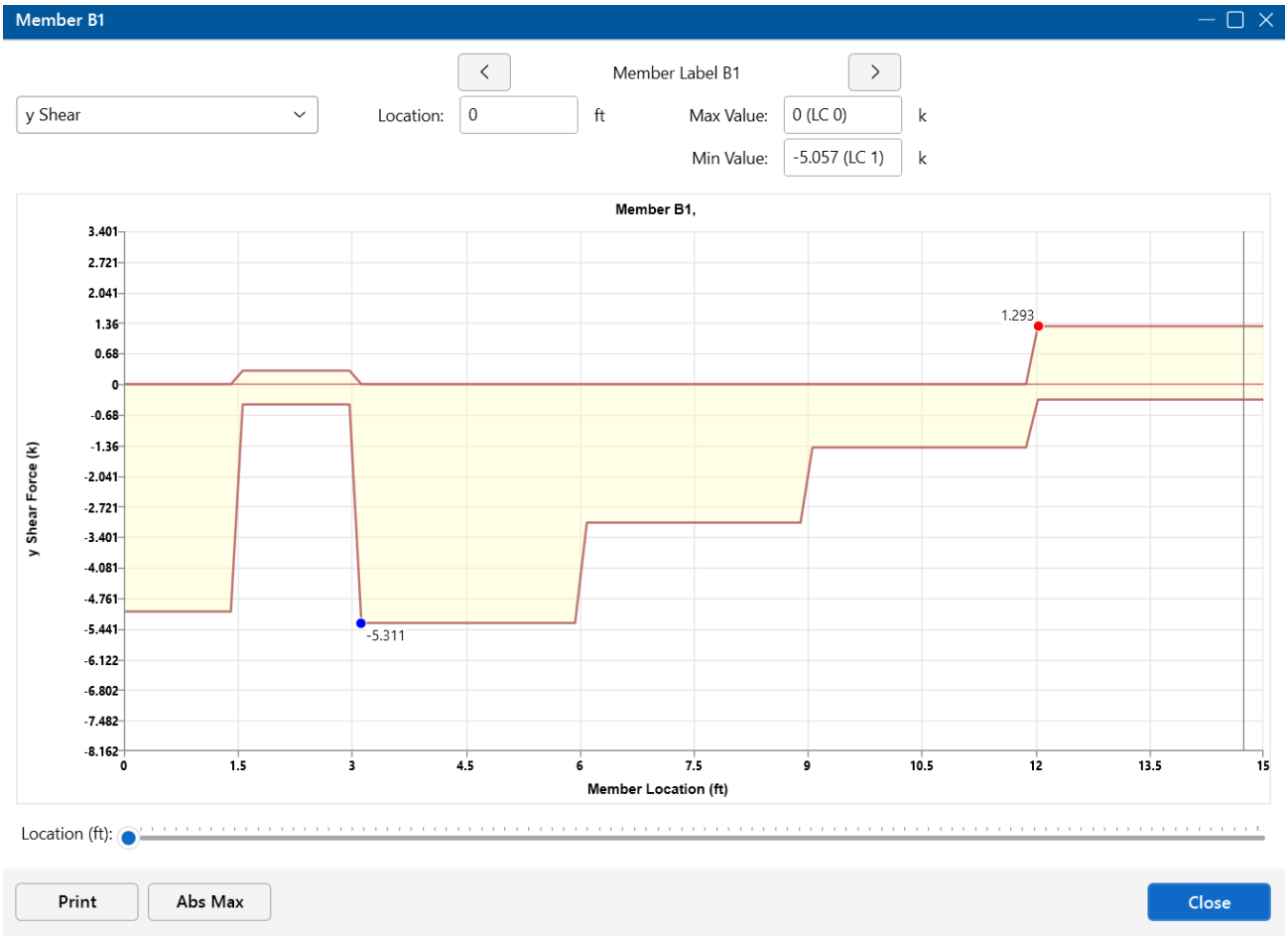


You can identify the **Member Type** by referring to the text between the navigation arrows in the upper-left corner of the **Detail Report** dialog window. This title will always show the member type (Beam).

The **Member Information** in the text above the force diagrams displays basic member information, as well as the **Concrete Stress Block** type used in the solution.

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.


Detail Report for - B1
Beam Label B1
Options
Effective "I"(Service) (in4): 29652.48
Bottom Cover (in): 3
Diagrams
Axial Force (k)
1.293 at 12.03 ft
-5.311 at 3.12 ft
y Shear Force (k)
z Shear Force (k)
2.053 at 3.12 ft
Torsion (k-ft)
4.782 at 12.03 ft
-22.911 at 3.12 ft
z-z Moment (k-ft)
y-y Moment (k-ft)



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 drop-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 provided. **Gov Mu Top** and **Gov Mu Bot** represent the governing ultimate moment in the top and bottom of the beam, respectively. **Gov Vu** represents the governing ultimate shear along the local y-axis of the beam.

The values **phi*Mn Top** and **phi*Mn Bot** represent the nominal moment strength at 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*Vn** represents the nominal shear strength in the beam, reduced by the appropriate Phi Factor.

General concrete, reinforcement and bar cover information about the section is also 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

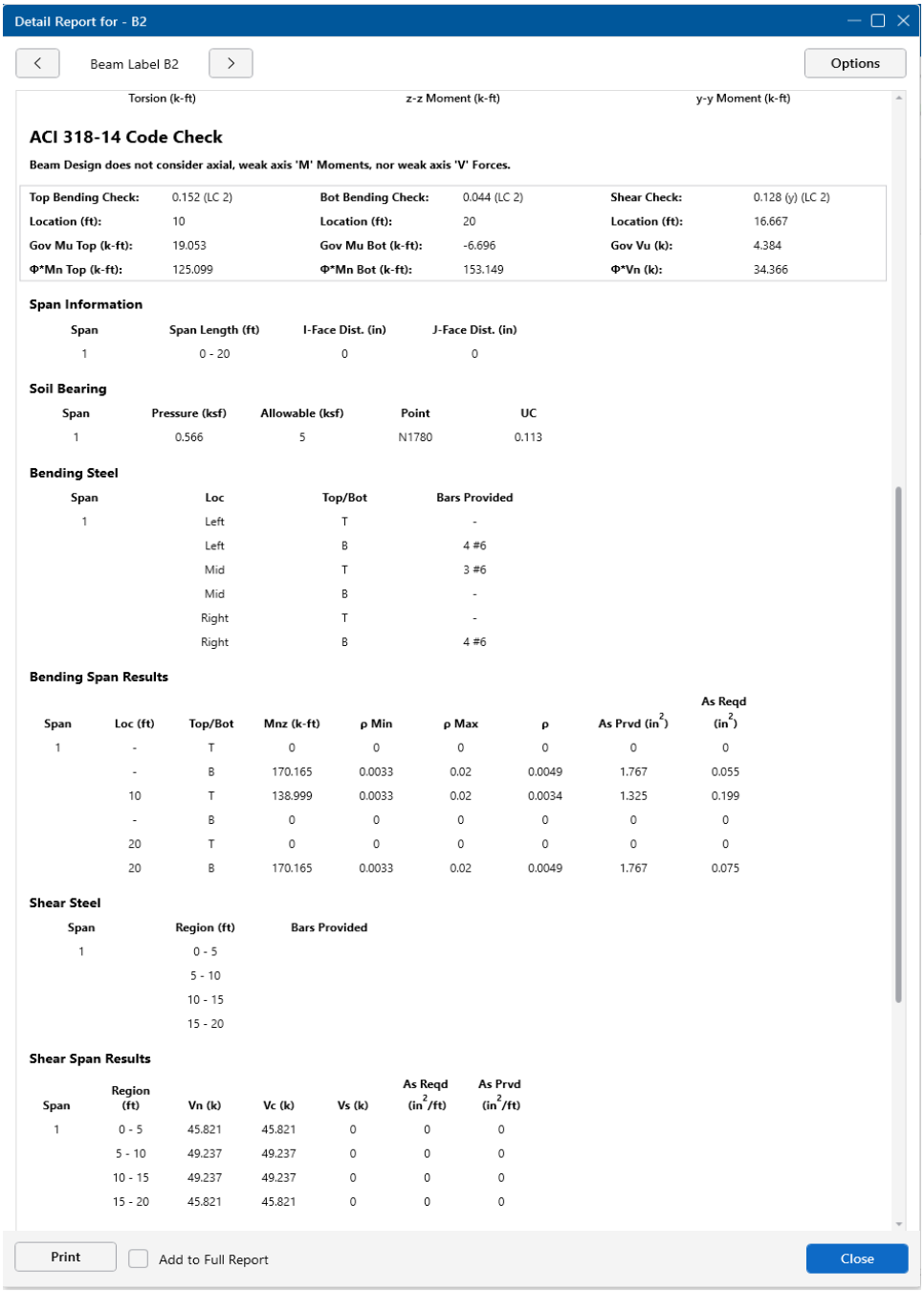
Materials Spreadsheet or the calculated value based on the given f'c and weight density (if the 'E' value was left blank on the Materials Spreadsheet).

The **Span Information** gives the start and end of each span centerline within the member, as well as the distance from the support centerline to the face of the support 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 **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 Reqd** 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 (also 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 taken from the **Shear Regions** setting on the **Design** tab of the **Model Settings Dialog**. The number, size and spacing of reinforcing bars is provided 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.

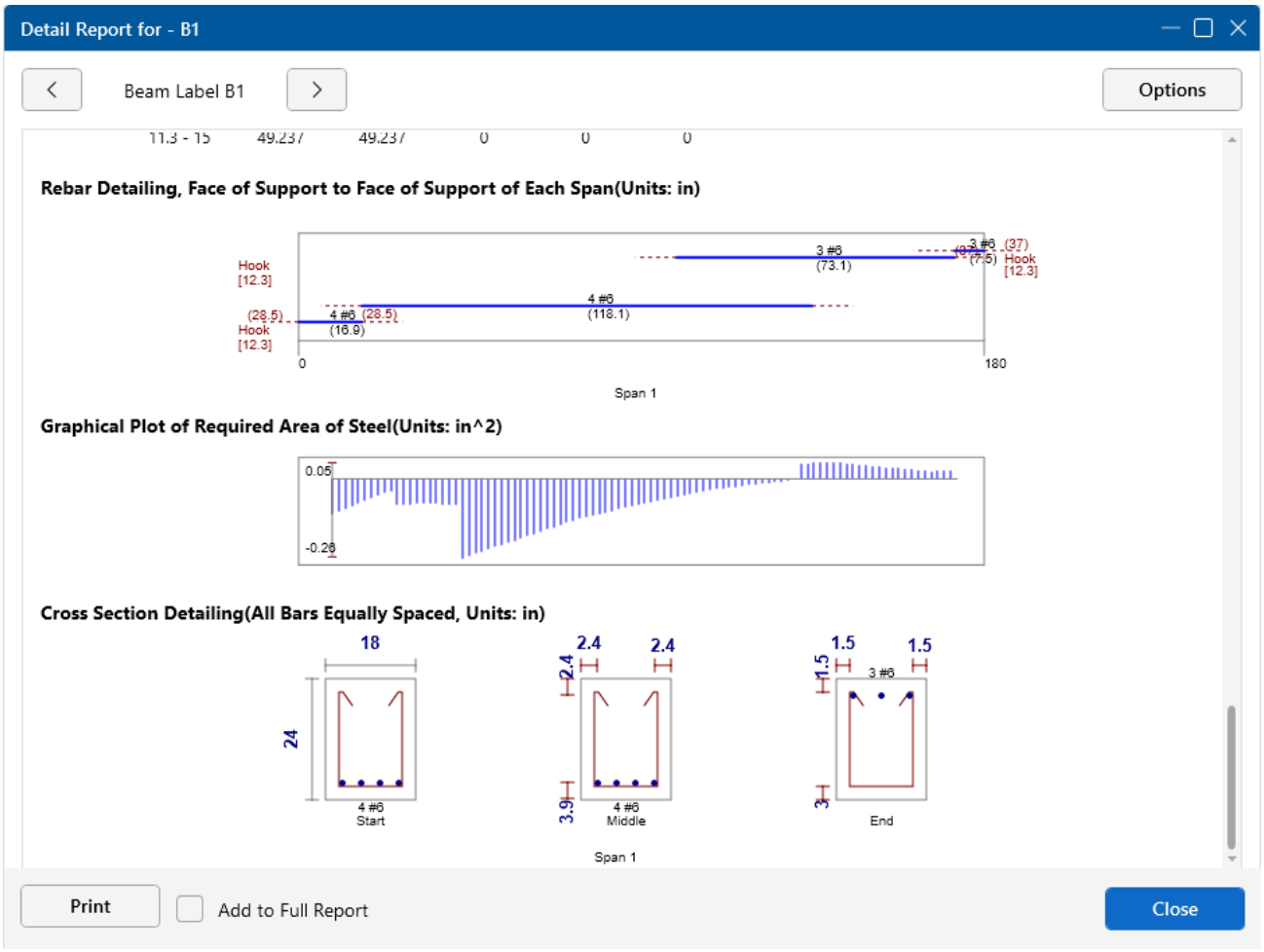
It should be noted that the values for **Mn** and **Vn** in this section of the detail report are the UNREDUCED nominal capacities of the member at each span/region. The actual design capacities would be obtained by multiplying these values by their respective **Phi Factors** indicated in the code.

The last section of the detail report, shown below, displays the **Beam Reinforcement Detailing Diagrams**. The **Rebar Detailing** portion of the report displays 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 represented by a dashed line. In addition to the development lengths, hook lengths are provided for bars at the ends of the beam, 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.

Beams - Forces

When the model is solved, the results are separated into specific design results and generic results. This section discuss the generic beam forces results. The design results are discussed in Beams - Design Results.

Number of Reported Sections

Note that the beam forces are only reported at the section locations. For example, if you set the **Number Of Reported Sections** in the **Model Settings** dialog to '2', you will not get any results for the middle of your beam, you will only get results for the end points. If you have a point load applied to your member at a location that is not a section location, you will probably not report the maximum moment in the section if it does not occur at an endpoint.

Adjust Number of Sections

To adjust the number of Sections:

1. Go to the **Model Settings** window.
2. If not already open, click on the **Solution** tab.
3. Adjust the **Number of Reported Sections** as needed.



Note: Adjusting the number of sections affects the amount of output.

Number of Internal Sections

Internally, the program subdivides the beam into equally spaced sections to calculate forces, code checks, etc. The **Number of Internal Sections** can be adjusted in the **Model Settings** dialog. If this value were set to '100', this would mean that for a beam that is 100ft long, RISAFoundation will calculate these values at approximately every foot. These values are then used in the beam code checks, diagrams in the model view and detail reports. The locations of the maximum code checks are reported at a distance from the Start Point.

Adjust Number of Internal Sections

To adjust the number of internal Sections:

1. Go to the **Model Settings** window.
2. If not already open, click on the **Solution** tab.
3. Adjust the **Number of Internal Sections** as needed.

All other results are reported at the **Number of Reported Sections** that you specify in the **Model Settings** dialog. This controls how many places you receive **reported** or **printed** beam force results. These locations are also equally spaced, so setting the value to '5' will give you 5 equally spaced sections; at the ends, the middle and the quarter points.



Note:

- Adjusting the **Number of Internal Sections** will not affect the amount of output in the results spreadsheets.
- You may want to stick with odd numbers for the **Number of Reported Sections**. Setting the **Number of Reported Sections** to an even number will not report forces at the midpoint of the beam, which is often the location of maximum moment.

Beam Force Results

Access the **Beam Section Forces** spreadsheet by going to the **Results** menu and selecting **Beam Forces**.

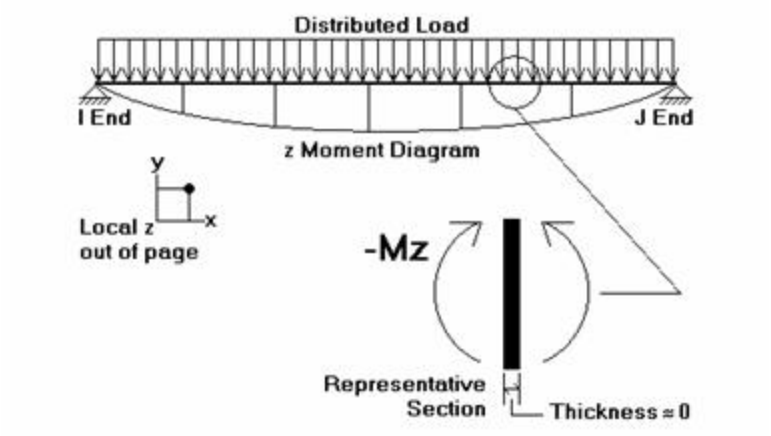
Beam Section Forces (By Combination)

	LC	Beam Label	Sec	y Shear[k]	Torque[k-ft]	z-z Moment[k-ft]
1	1	B1	1	-2.409	0.18	-6.985
2			2	0.302	0.028	-7.222
3			3	-4.094	0.401	-15.234
4			4	-2.667	0.401	-7.138
5			5	-1.406	0.401	-1.731
6			6	-0.342	0.401	1.252
7			7	-0.342	0.401	2.107
8	1	B2	1	-1.656	0.666	-1.591
9			2	-1.656	0.666	3.928
10			3	-0.871	0.666	6.832
11			4	0.36	0.666	7.591

These are the beam forces calculated along each beam. The units for the forces are shown at the top of each column. As for the sign convention, the signs of these results correspond to the beam's local axes, using the right hand rule. The left side forces at each section location are displayed. There are three force values for each section location.

These include shear parallel to the local y-axis (y Shear), torque moment and moment about the member's local z-axis (z-z Moment).

These section forces may also be displayed graphically. Remember that the section forces used for the plot are the left side forces. For an example of what you would see for the graphic plot of the moment diagram for a member, please see below:



RISAFoundation uses the right hand rule **joint convention** and is always consistent with this convention. Since the left side moment is being used, a beam under negative Mz moment would have the "holds water" deflected shape, which is contrary to some **beam conventions**. The opposite is true for My moments which will tend to "hold water" under a positive moment and "shed water" under a negative moment.

Note:

- To learn how to use **Find**, **Sort** and other options, see Spreadsheet Operations.
- To learn how to plot beam results, see Beams.

Boundary Conditions

Boundary Conditions define how the model is externally constrained. The soil regions you define create a grid of compression-only springs to support your slabs/footings. You may define additional points of support as completely restrained or as partially restrained with a **Spring**. You can also define a spring support that has stiffness in only one direction with tension-only or compression-only springs.

Points of support may also be defined as **Footings, Piles or Pile Caps**. This section will deal primarily with points of support not at a footing, pile or pile cap. For more information on support at foundation elements, see Footings - Design and Piles and Pile Caps.

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

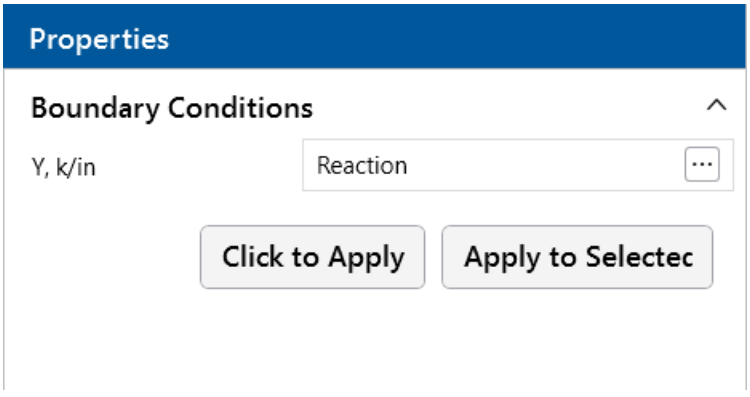
Creating and Modifying Boundary Conditions

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

Modify Boundary Conditions for Nodes

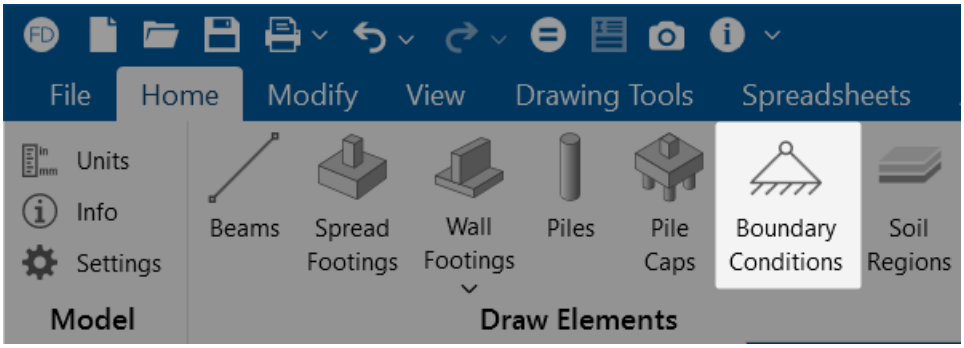
The graphical **Boundary Conditions** tool discussed here lets you specify and modify boundary conditions graphically.

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



To apply Boundary Conditions:

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



3. You can apply the boundary condition by choosing **Click to Apply** or **Apply to Selected.**

   To choose nodes individually, choose **Click to Apply**. Click the nodes with the left mouse button.

   To apply the boundary condition to a selection of nodes, choose **Apply to Selected.**

Note:

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

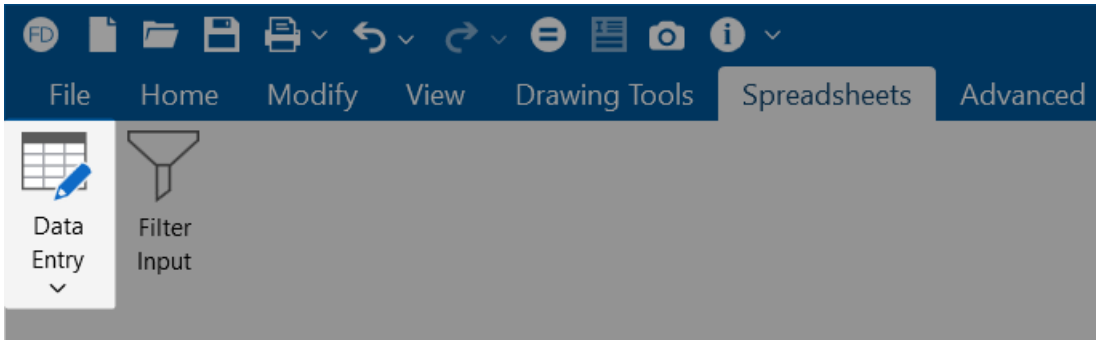
Auto-Generated Soil Springs

When a solution is performed, RISAFoundation auto-meshes each slab into a mesh of plate elements. In the process, points are automatically created to define the corners of each plate. A **Soil Spring**, or compression-only spring, is auto-generated at every plate corner point, in each slab. User-defined supports previously applied to any point within a slab will override any soil springs generated by the program. In other words, a soil spring will NOT be applied at those particular points in the slab.

RISAFoundation calculates the tributary area for each plate corner point individually and multiplies that area by the **Subgrade Modulus** defined in each particular **Soil Region** to determine the spring stiffness for the soil spring at that point. For more information, see Soil Springs Calculation.

Boundary Conditions Spreadsheet

The **Boundary Conditions Spreadsheet** records the boundaries for the nodes and may be accessed by selecting **Boundary Conditions** from the **Data Entry** drop-down in the **Spreadsheets** ribbon, or by selecting **Boundary Conditions** from the **Data Entry** section in the **Explorer** panel.



Boundary Conditions

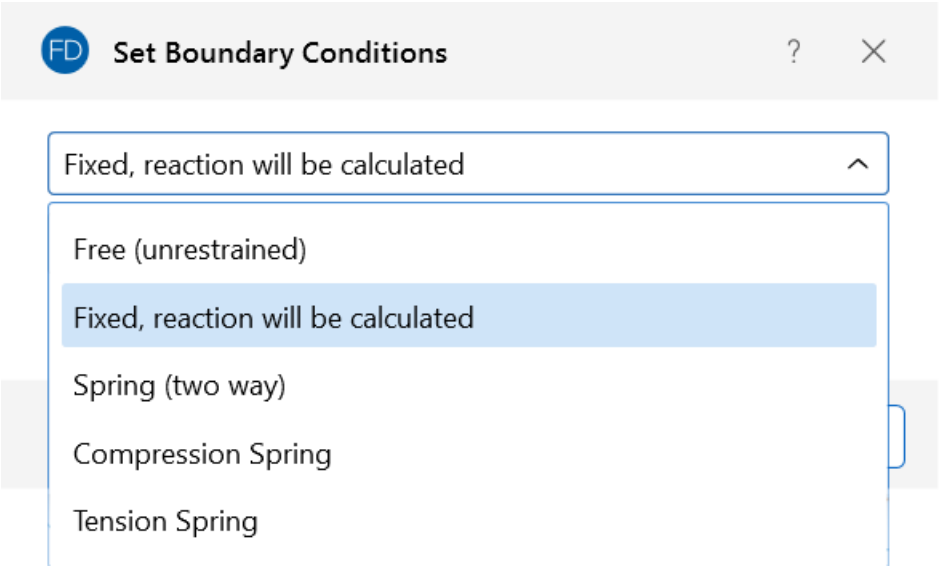
	Node Label	Y [k/in]
1	N9	Reaction
2	N61	Reaction
3	N72	Reaction
4	N5	Reaction
5	N36	Reaction
6	N2	Reaction
7	N122	Reaction
8	N50	Reaction
9	N1	Reaction
10	N3	Reaction
11	N209	Reaction
12	N197	Reaction

The **Node Label** column contains the label of the node that is restrained.

The boundary conditions are entered in the remaining column by selecting the cell, clicking ... (ellipsis) button and choosing from the boundary options. You may also type them in directly.

Boundary Condition Options

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



Note:

- These options are only available for nodes not defined by a footing, pile or pile cap.
- Models that contain compression-only or tension-only springs must be iterated until the solution converges. Convergence is achieved when no more load reversals are detected in the springs. During the iteration process, each spring is checked, and if any springs are turned off (or back on), the stiffness matrix is rebuilt and model is resolved.
- With this iteration procedure it is possible to run into a phenomenon known as "clapping". This occurs when a compression-only spring keeps going between a tension force present (where the spring is removed) to a compression force present (where the spring is re-added). This can be somewhat common in non-linear solutions. If you run into this phenomenon, you may see the program give Error 1162. The fix for this is to very slightly adjust either the loading or the spring/element stiffness to get the offending spring out of the range where it can bounce back and forth. Another option would be to move the location of the soil spring slightly so it's not right at an inflection boundary.

Reaction Boundary Condition

Reaction specifies full restraint for the vertical direction. No movement will be allowed in the vertical direction, perpendicular to the XZ plane, for the node and the reaction force will be calculated.

Spring Boundary Condition

The "Snnn" code for **Springs** models a spring attached to the node in the vertical direction. The "nnn" portion of the code is the numerical magnitude of the springs' stiffness. The appropriate units are shown at the top of the column.

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

Compression-Only Springs

The "CSnnn" code for **Compression-Only Springs** models a one way "compression-only" spring attached to the node in the vertical direction. This spring has stiffness for negative displacements (downward) and NO stiffness for positive displacements (uplift). The "nnn" portion of the code is the numerical magnitude of the springs' stiffness. The spring stiffness units are the same as those for a normal spring. Compression-only springs are used by RISAFoundation when auto-generating soil springs for slabs.

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

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

Tension-Only Springs

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

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

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

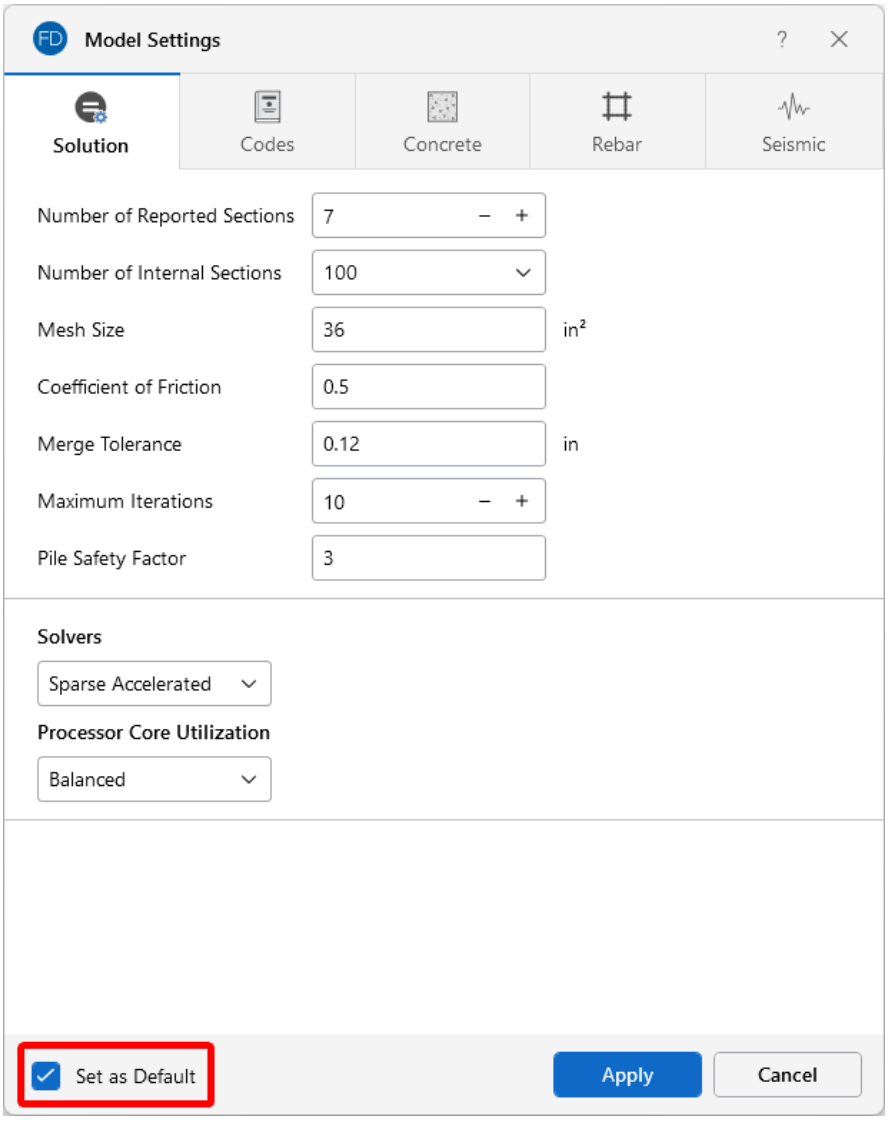
Customizing RISAFoundation

You may customize many of the default parameters, design and analysis options in RISAFoundation. 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** offer you control over the behavior of the software, accessible by clicking the **File** tab and selecting Application Settings. The **Save as Defaults** feature allows you to specify the default settings for new model files. Custom reports may also be defined and saved for future use. To learn how to build a custom report, see Printing.

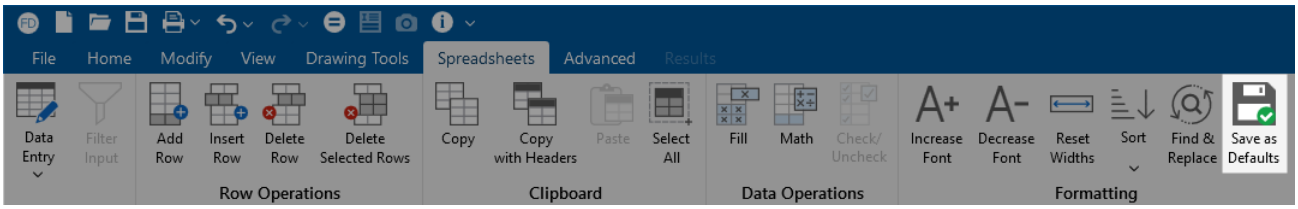
The following sections provide more details on the available settings:

- Save as Defaults
- Application Settings (Preferences)

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 checking the **Save as Defaults**checkbox: **Model Settings, Units** and **Drawing Grids**. This will cause the program to use these settings with any new files that are created.



Many of the spreadsheets also provide the option to save the current data as the default so that every subsequent new file will already have that data. Simply enter the data, then save it as the default by clicking on the **Save as Defaults** button on the **Spreadsheets** ribbon. This way, the office standards that you regularly use in your models are already entered and available in new models.

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 can be utilized to give you a good starting point for new files, but won't hold you to those settings.

Application Settings (Preferences)

The **Application Settings** window allows you to choose your program preferences. You can set general preferences, such as showing the “Starting a Model” window when you open RISAFoundation, opening the “Project Information” window when loading a model, showing a warning before filtering input and many other options for setting up your application to best suit your liking. Options may be accessed by clicking **File** in the ribbon toolbar and selecting **Application Settings**.

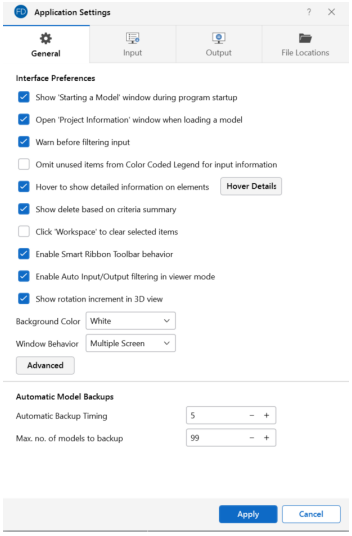
To open the Application Settings window:

1. Click on **File** in the ribbon toolbar, as shown in the following image.



2. Select **Application Settings**.

   The Application Settings window opens to the General tab.



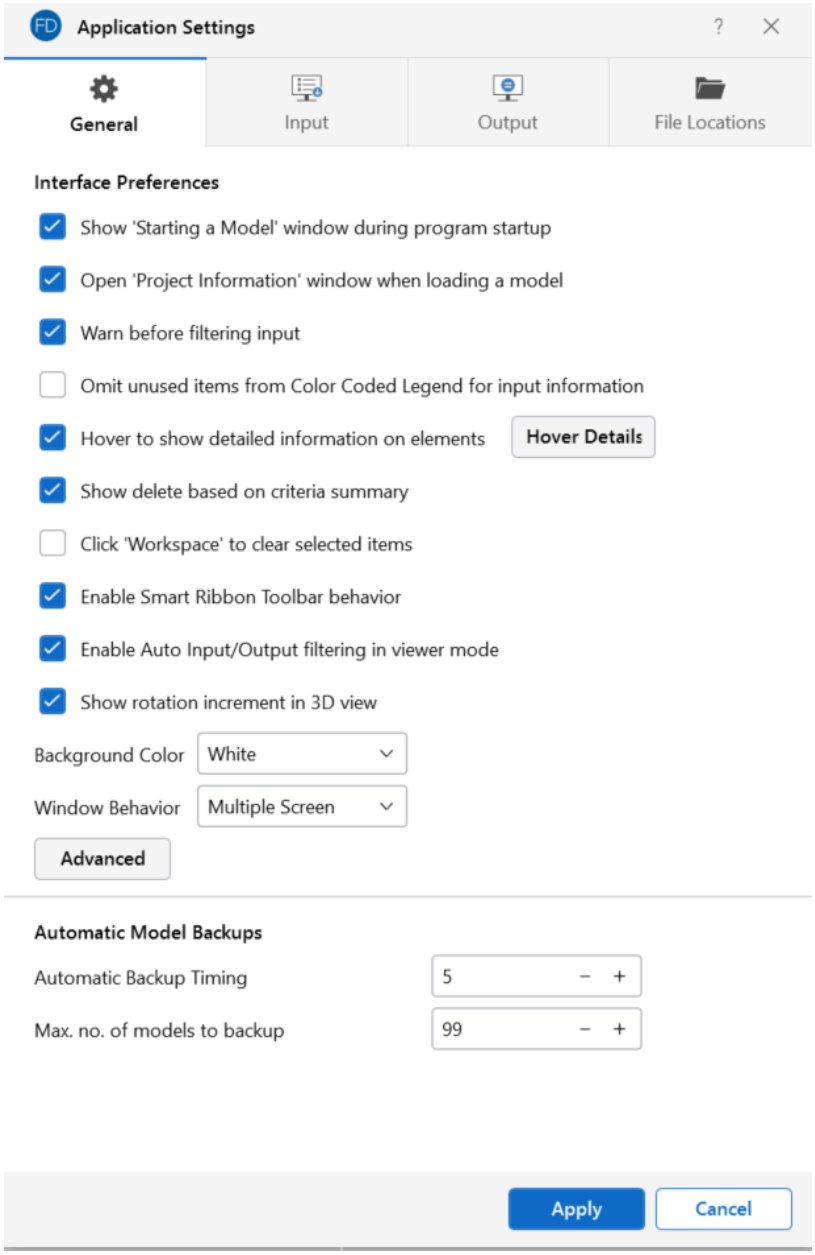
3. (Optional) Click on the tab associated with the preferences you want to set.

Many of the Application Settings themselves are self-explanatory. However, the following sections provide more details on the available preference settings.

General Application Settings

The general Application Settings are fairly straightforward. For more information on specific item, click the ? button in the top corner of the dialog to be taken to more detailed information on the available options. One of the options you can set on this tab is the frequency for which the program

will create a backup of your model. For more information about the backup capabilities of RISAFoundation, see Automatic Backup. The **Reset all Program Defaults** button in the Advanced General Application Settings will clear all of the Application Settings that you have set on any of the tabs.



The following table provides descriptions for the available settings.

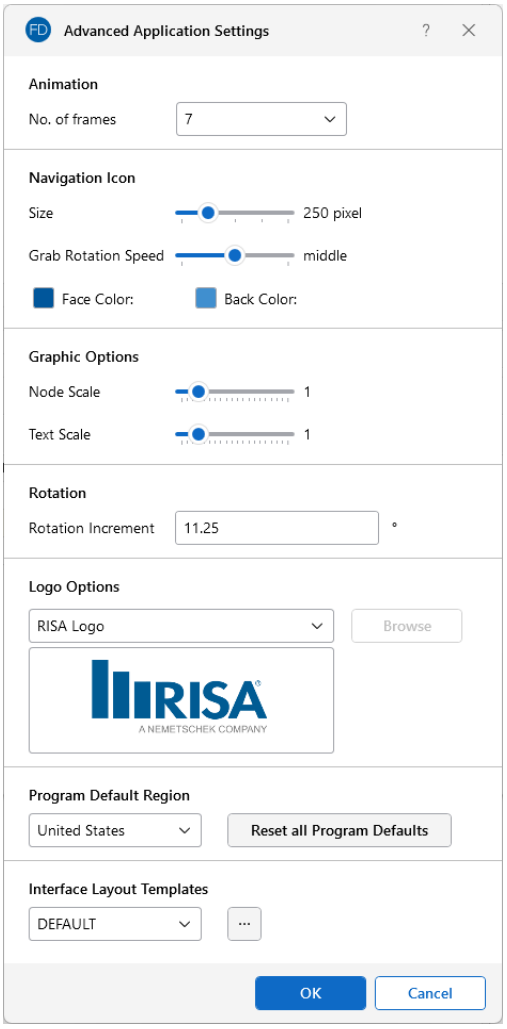
General Application Settings

Setting	Description
Show 'Starting a Model' Panel when starting a new model	The **New Model Dialog** will be displayed when opening the program or selecting 'New File' from the File menu.
Open 'Project Information' window when loading a model	Displays the **Project Information** settings automatically after loading a file.
Warn before filtering input	With this option checked, the program will always warn you before you filter input.
Hover to show detailed information on elements	Click the Hover Details button to customize the information shown when hovering over an element.
Show delete based on criteria summary	This provides an option to display a summary of all items that were deleted after deleting based on a specific criteria.
Click 'Workspace' to clear selected items	This option allows you to click the white space in the 3D View to clear your current selection.
Enable Smart Ribbon Toolbar behavior	The **Smart Ribbon Toolbar** behavior will predict which tab of the Ribbon toolbar to open based on the actions performed in the program.
Enable Auto Input/Output filtering in viewer mode	This option will automatically filter both input and output spreadsheets based on visible items in Viewer Mode.
Show rotation increment in 3D View	This option will show the degree increment field in the Incremental Rotate bin.
Background Color	Choose which color to make the background color in the user interface.
Window Behavior	Choose your window behavior option. For more information, see Window Behavior.
Automatic Backup Timing	Automatic backup of the currently open model occurs at the specified interval. Each backup overwrites the previous one, such that only one backup of a given model exists at any time.

Setting	Description
Max. no. of models to backup	This controls how many models are kept in the backup folder. Once the folder becomes "full" according to this setting, the oldest backups will be deleted automatically. Setting this to zero turns off automatic backups.

Advanced Application Settings

Additional program options can be found by clicking the **Advanced** button in the Application Settings window, under the General tab.



The following table provides descriptions for the settings.

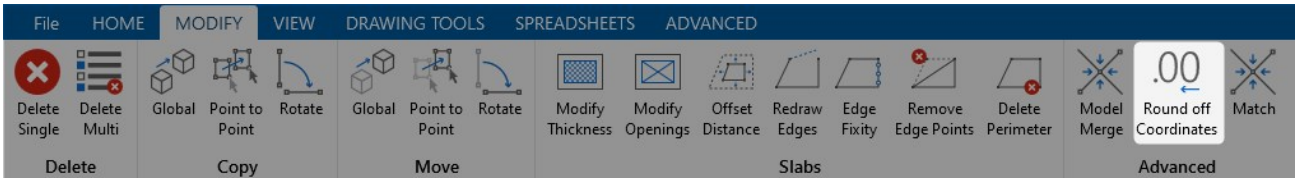
Advanced Application Settings

Setting	Description
No. of frames	This option allows you to specify how many frames are used in the animation to move from the undeflected state of the model to the fully deflected state.
Size	This option controls the size of the Global Axes icon.
Grab Rotation Speed	This option controls the rate of rotation when rotating the model with your mouse cursor. A small grab rotation coefficient will cause the model to rotate less as you move your mouse. A large grab rotation coefficient will cause the model to rotate at a higher rate as you move your mouse.
Face Color	This option controls the color of the front of the Global Axes icon.
Back Color	This option controls the color of the back face of the Global Axes icon.
Node Scale	This setting controls the size of the nodes displayed in the 3D View.
Text Scale	This setting controls the size of the text displayed in the 3D View.
Rotation	This setting controls the rotational increment for the **Click to Rotate** function.
No Logo	This option will not include a logo on your printed output.
RISA Logo	This option will place the RISA logo on all printed output.
Custom Logo	This option allows you to add a custom company logo to your printed output.
Program Default Region	Choose the default region for your projects.
Reset all Program Defaults	Choose this to reset all of the Application Settings that you have a set on any of the tabs.

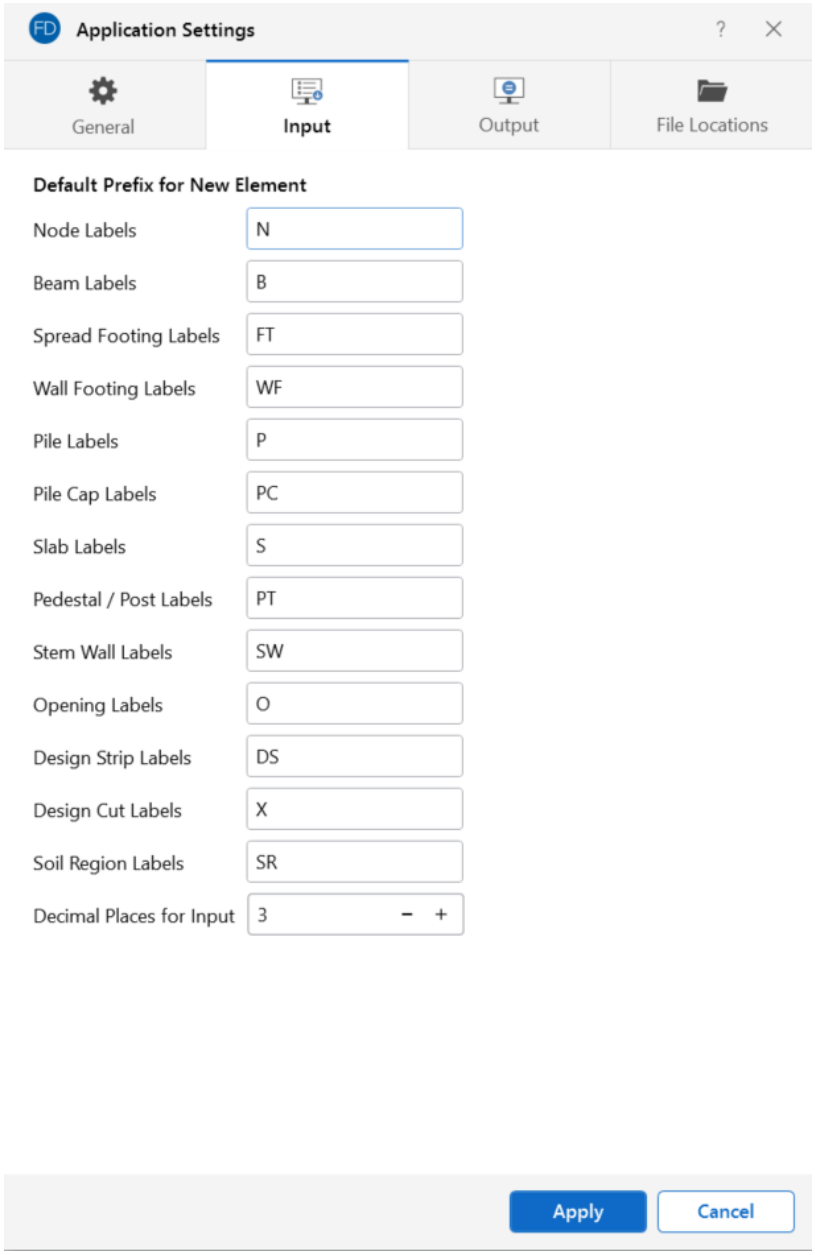
Setting	Description
Interface Layout Templates	This setting controls the layout of the bins in the 3D View.

Input Application Settings

Here you can specify the number of decimal places that are displayed. The one exception is the **Node Coordinates**, as RISAFoundation maintains the coordinates to 15 significant figures and the exact value is always displayed. You may also use the **Round off Coordinates** button in the **Modify** tab to round off node coordinates.



If you wish to use a prefix with your model elements, such as "N" with nodes, you can specify the default prefix on this tab. The prefixes may be changed as you build your model.



The following table provides descriptions for the settings.

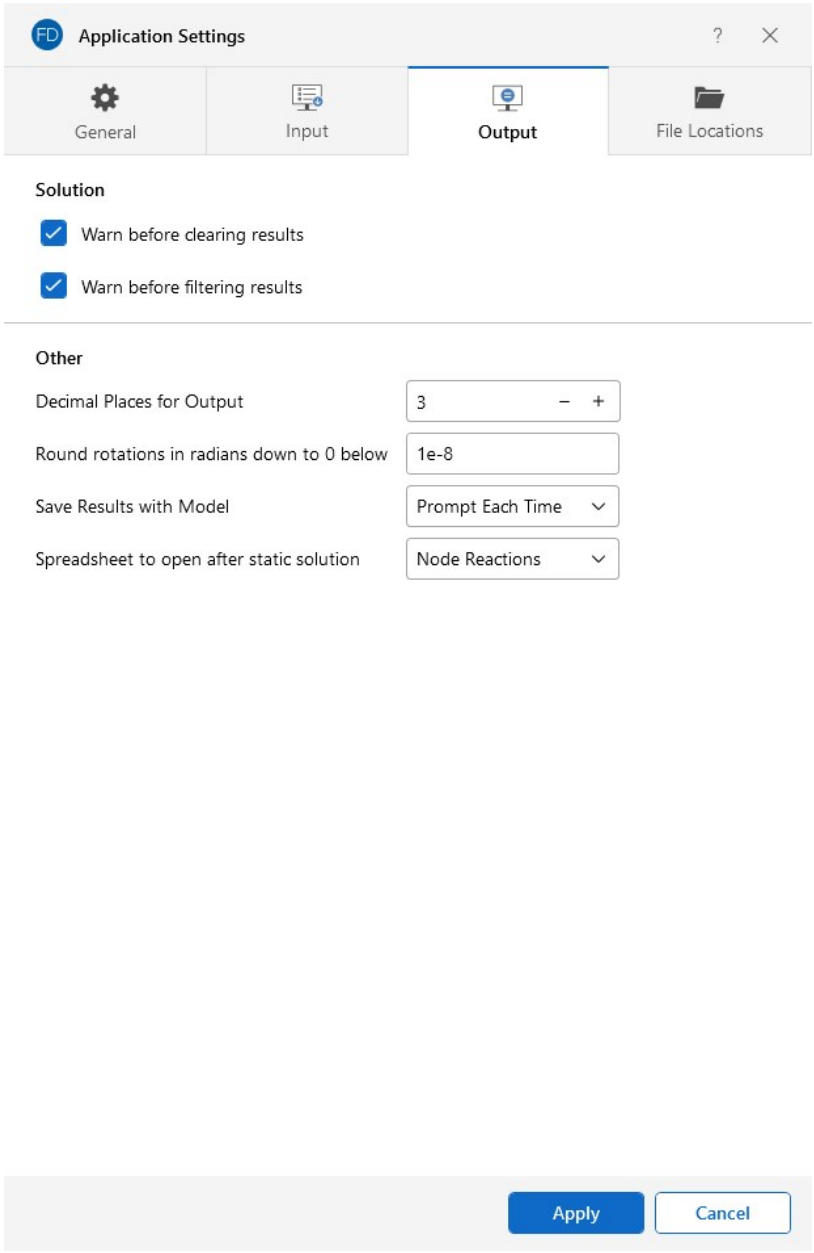
Input Application Settings

Setting	Description
Node Labels	Sets the default prefix to be used in node labels.
Beam Labels	Sets the default prefix to be used in beam labels.
Spread Footing Labels	Sets the default prefix to be used in spread footing labels.

Setting	Description
Wall Footing Labels	Sets the default prefix to be used in wall footing labels.
Pile Labels	Sets the default prefix to be used in pile labels.
Pile Cap Labels	Sets the default prefix to be used in pile cap labels.
Slab Labels	Sets the default prefix to be used in slab labels.
Pedestal / Post Labels	Sets the default prefix to be used in pedestal/post labels.
Stem Wall Labels	Sets the default prefix to be used in stem wall labels.
Opening Labels	Sets the default prefix to be used in opening labels for openings in slabs.
Design Strip Labels	Sets the default prefix to be used in design strip labels.
Design Cut Labels	Sets the default prefix to be used in design cut labels.
Soil Region Labels	Sets the default prefix to be used in soil region labels.
Decimal Places for Input	Sets the number of decimal places to display in the data spreadsheets, with a maximum of four places.

Output Application Settings

RISAFoundation can provide a warning when clearing results. You may also specify the number of decimal places that are displayed for results. The number of figures displayed may not be the true value in many cases because RISAFoundation maintains numbers to several decimal places behind the scenes.



The following table provides descriptions for the settings.

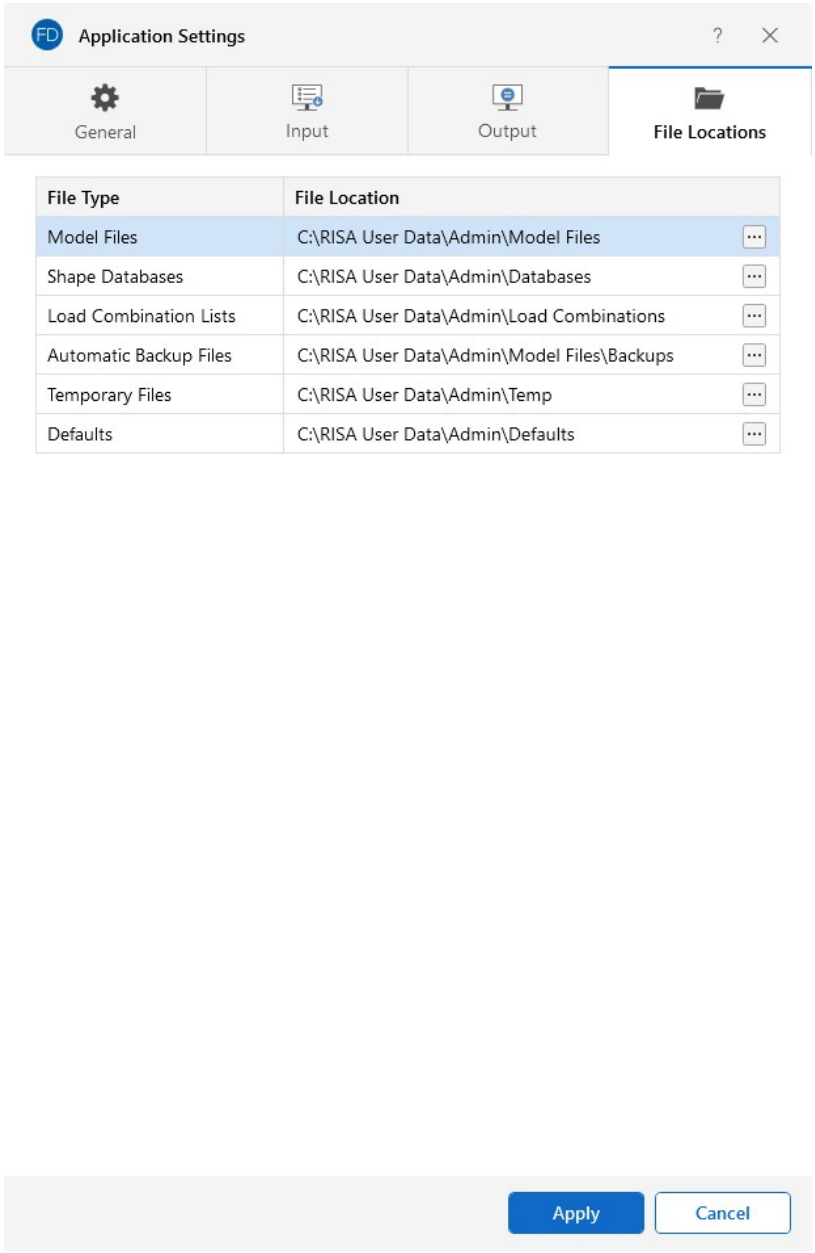
Output Application Settings

Setting	Description
Warn before clearing results	Verifies that results are to be cleared to edit the model.
Warn before filtering results	With this option checked, the program will always warn you before you filter output results.
Decimal Places for Output	Sets the number of decimal places to display in the

Setting	Description
	results spreadsheets, with a maximum of four places.
Round rotations in radians down to 0 below	Shows 0 for the rotation when smaller than this value.
Save Results with Model	These options let you control what is done with the results when saving a file.
Spreadsheet to open after static solution	This option allows you to select the default results spreadsheet to open after solution. You can also choose 'None'.

File Location Application Settings

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



For each file type in the list the current setting is displayed.

Click the ··· browse button to choose a different location.

Design Optimization

RISAFoundation will optimize Concrete beams and slab reinforcement. 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 **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. On each tab of the **Design Rules Spreadsheet**, you have the option to enter the maximum bending and shear unity checks in the **Max Bending Chk** and **Max Shear Chk** columns of the spreadsheet. This should usually be specified as '1', however, if you desire a larger factor of safety, provide a lower factor (i.e. 0.95).

You can assign the design rules graphically as you draw elements, or later as a modification. Design rules for slab design cuts may only be applied via the **Design Cuts Spreadsheet**. For more information, see Modifying Beam Properties, Slabs - Design Strips and Slabs - Design Cuts.

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

Design Rules - Beam Rebar



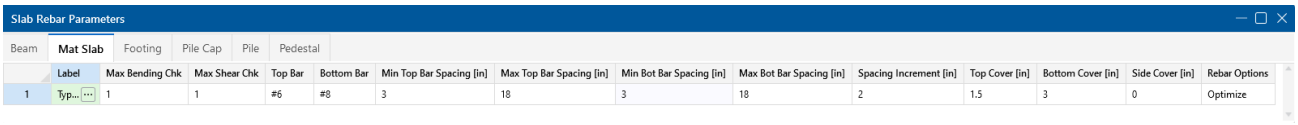
Note: If you would like to define specific flexural and shear rebar layouts for beams and columns, either use the **Explicit** option discussed below or see Custom Rebar Layout Database for Custom Rebar layouts.

The entries for the **Beam Rebar** tab are explained in the following table.

Column Header	Description
Optimize Rebar	The options in this column are **Optimize** and **Explicit**. **Optimize** tells the program to find the most optimal size and spacing of the reinforcement. The **Explicit** option indicates the exact number and size of reinforcement

Column Header	Description
	bars. When Explicit is selected, additional columns are editable and allow you to enter the quantity for **Top/Bottom Bars** and **Shear Bar Spacing**. These columns are grayed out when **Optimize** is selected.
Flexural Bars	Use the **Min Flex Bar** and **Max Flex Bar** columns to restrict bar sizes for your beam flexural reinforcing. Currently, we support the ASTM A615 (imperial) and ASTM A615M ("hard" metric, i.e. #8M is an 8mm bar) reinforcement standards. You may specify your rebar set on the Rebar tab of the Model Settings. You can force the program to analyze one bar size by setting the Min and Max values to be the same bar size.
Shear Bars	Use the **Shear Bar** column to enter the size of your shear ties. Currently, we support the ASTM A615 (imperial) and ASTM A615M ("hard" metric, i.e. #8M is an 8mm bar) reinforcement standards. You may specify your rebar set on the Rebar tab of the Model Settings.
Legs per Stirrup	Use the **Legs per Stirrup** column to enter the specific information about how many legs (2 to 6) each of your shear ties is expected to have.
Concrete Cover	The **Top Cover** and **Bottom Cover** columns are used to specify the clear cover measured to the beam flexural reinforcing steel. The **Side Cover** column is used to specify the clear cover measured to the shear reinforcing. The units are listed in the column header.

Design Rules - Slab Rebar



The entries for the **Mat Slab Rebar** tab are explained in the following table.

Column Header	Description
Rebar Sizes	The **Top Bar** and **Bottom Bar** columns register the flexural reinforcement bar sizes that are to be used in the footing design. Currently, we support the ASTM A615 (imperial), ASTM A615M ("hard" metric, i.e. #8M is an 8mm bar) and CSA G30.18 (Canadian) reinforcement standards. You may specify your rebar set in the Model Settings.

Column Header	Description
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 top and bottom slab flexural reinforcement. The units are listed in the column headers. The **Spacing Increment** column sets the rebar spacing increment that will be used by RISAFoundation 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.
Side Cover	The **Side Cover** column is used to specify the clear cover from the edge of a design strip or cut measured to the nearest slab flexural bar. The units are listed in the column header. Note that this entry should probably be '0' for design strips or cuts that do not extend to the edges of a slab, provided that they are not directly adjacent to another design strip or cut with a side cover greater than '0'.
Rebar Options	The last column allows the user to place their slab reinforcement at the mid-depth of the slab, force it to be placed at the top and bottom, one single layer at any depth or to be optimized by the program solution.

Note:

- For **mid-depth** rebar, the program will use the *bottom* bar size and spacing options. The cover to reinforcement specified in the design rule will have no effect on the placement of the reinforcement. The mid-depth option will always position the reinforcement such that it is centered within the slab.
- For **single layer** rebar, the design uses the *top* bar size and spacing options. The top cover under the General tab is used to define the rebar placement from the top of the slab. The top cover is the clear cover from the top face of the slab to the outer edge of the top layer reinforcing steel.

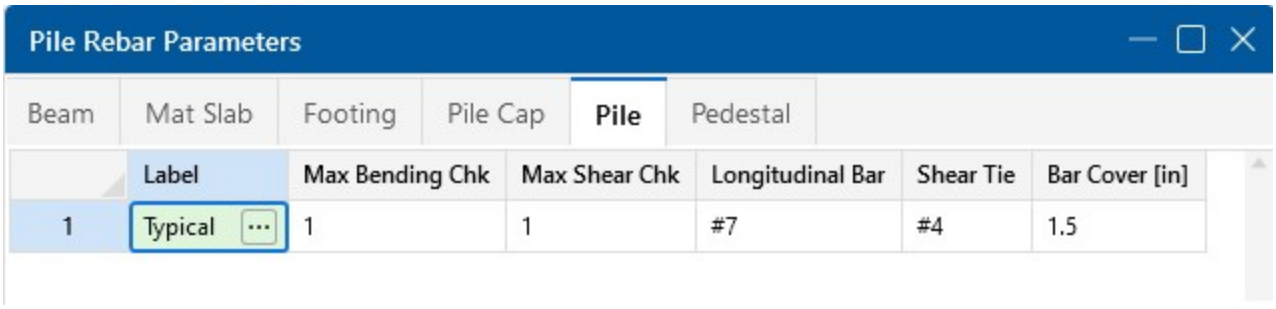
Design Rules - Footing/Pile Cap Rebar



The entries for the **Footing/Pile Cap** tabs are explained in the following table.

Column Header	Description
Rebar Sizes	The **Top Bar** and **Bottom Bar** columns register the flexural reinforcement bar sizes that are to be used in the design. Currently, we support the ASTM A615 (imperial), ASTM A615M ("hard" metric, i.e. #8M is an 8mm bar) and CSA G30.18 (canadian) reinforcement standards. You may specify your rebar set in the Model Settings.
Concrete Cover	The **Top Cover** and **Bottom Cover** columns are used to specify the clear cover measured to the beam flexural reinforcing steel. The units are listed in the column header.

Design Rules - Pile/Pedestal Rebar



The entries for the **Pile/Pedestal** tabs are explained in the following table.

Column Header	Description
Rebar Sizes	The **Longitudinal Bar** and **Shear Tie** columns register the flexural and shear reinforcement bar sizes to be used in the pile/pedestal design. Currently, we support the

Column Header	Description
	ASTM A615 (imperial), ASTM A615M ("hard" metric, i.e. #8M is an 8mm bar) and CSA G30.18 (canadian) reinforcement standards. You may specify your rebar set in the Model Settings .
Pedestal Bar Cover	The last column, **Pedestal Bar Cover**, is used to specify the clear cover measured to the shear reinforcing. The units are listed in the column header.

DXF Files

RISAFoundation provides the ability to export a drawing of your foundation plan (with footing and beam schedules), Footing Details or Slab Reinforcing to a DXF file.

Note: You can also import a DXF as a Drawing Grid. For more information, see Import DXF Grid.

The following sections provide more details:

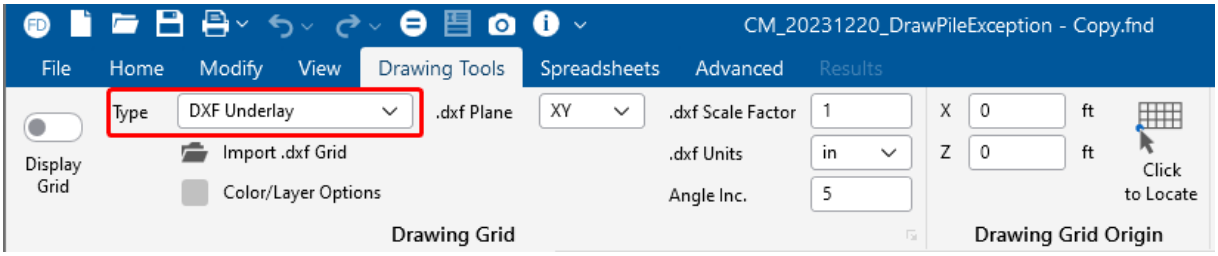
- Import a DXF Drawing Grid
- Export a Foundation Plan Drawing

Import a DXF Drawing Grid

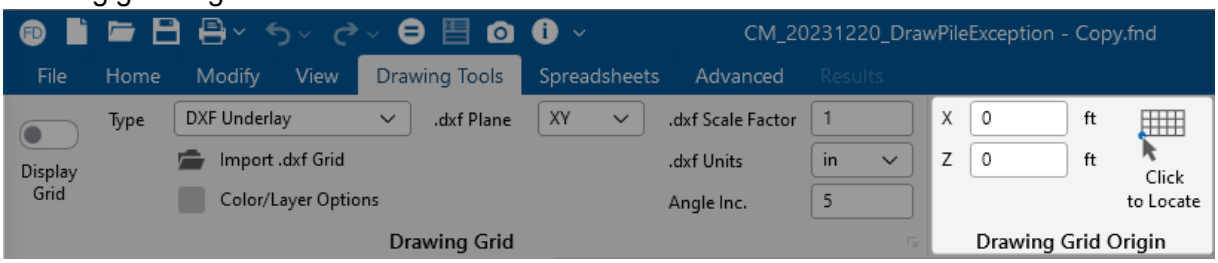
Import .dxf Grid lets you import a .dxf file into the model view as a drawing grid. The DXF image is imported so that you can snap to any point or intersection to aid in the drawing of your model. The DXF Import Drawing Grid supports Lines, Polylines, Circles, Arcs, Polylines with Arcs and Points.

To import a DXF drawing grid:

1. If there is not a model view already open, go to the **View** ribbon.
2. Click the **Open 3D Views** icon in the **Window** section to open a new view.
3. Select **DXF Underlay** from the Type drop-down menu in the **Drawing Tools** ribbon.

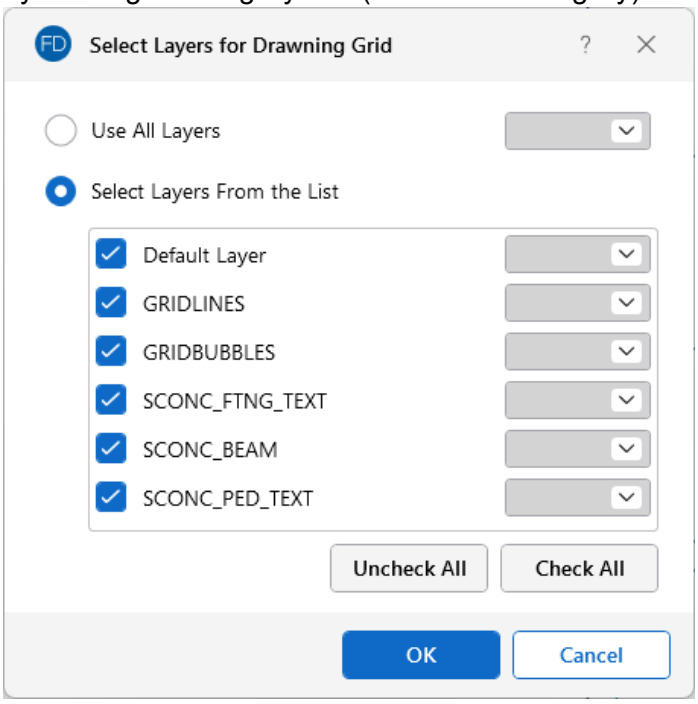


4. Set the **Drawing Grid Origin** directly or use the **Click to Locate** button to locate your drawing grid origin.



5. Choose the **.dxf Plane** from the drop-down menu.

6. Click **Import .dxf Grid** and select the DXF file you would like to us.
7. 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 for the layer display by clicking on the gray box (default color is gray).



The following table describes the options available when importing a DXF file.

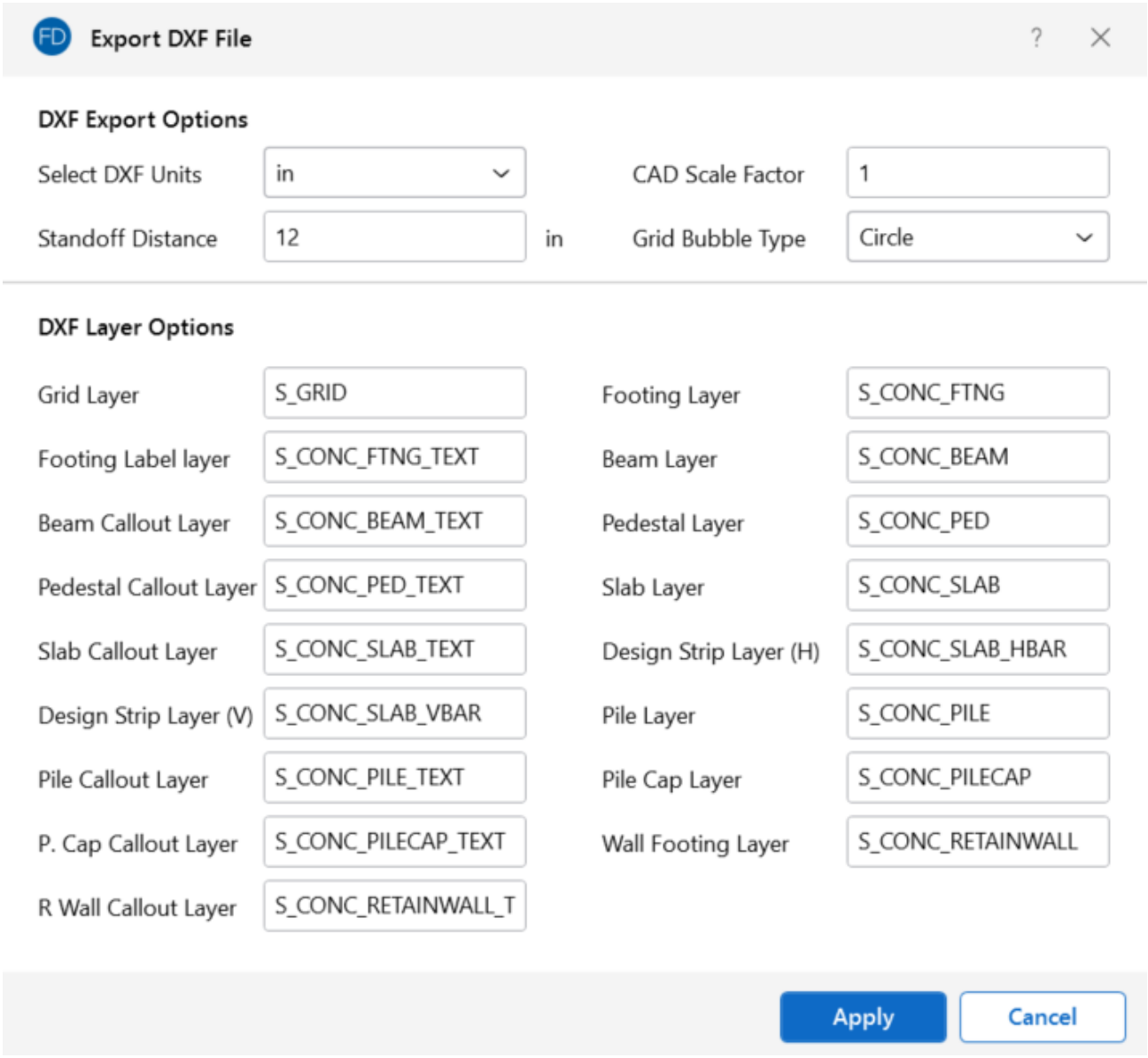
Import DXF File Options

Option	Usage / Description
.dxf Units	Allows you to choose the units. Supported DXF units are: none, inches, feet, mm, cm or meters. You need to choose the same units as used in the CAD model from which you produced the DXF file.
.dxf Scale Factor	Allows you to choose the scale factor that causes the DXF file to scale up or down in relation to the full scale. The default is 1.0. For example, if you created a scaled model in AutoCAD at a scale of 1/4”=12”, the appropriate scale factor to produce a full size RISA model would be 48.
Angle Inc.	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.
.dxf plane	Select the DXF drawing plane to import from the DXF file. The

Option	Usage / Description
	DXF file will be flattened to this plane and all lines/points visible from this plane will be imported.
Drawing Grid Origin	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 of the toolbar. The second option is to specify the origin by clicking on an existing point in the model. This option is available by selecting the **Click to Locate** button, and then clicking on the specific location in the model view where you want the drawing grid origin to be located.

Export a Foundation Plan Drawing

Only the node, member, wall and element geometry are translated and used to create an ASCII DXF file. Any other information such as the boundary conditions, loads, etc. are not translated at this time. You have several options available for controlling how DXF files are exported.



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

The following table describes the available options.

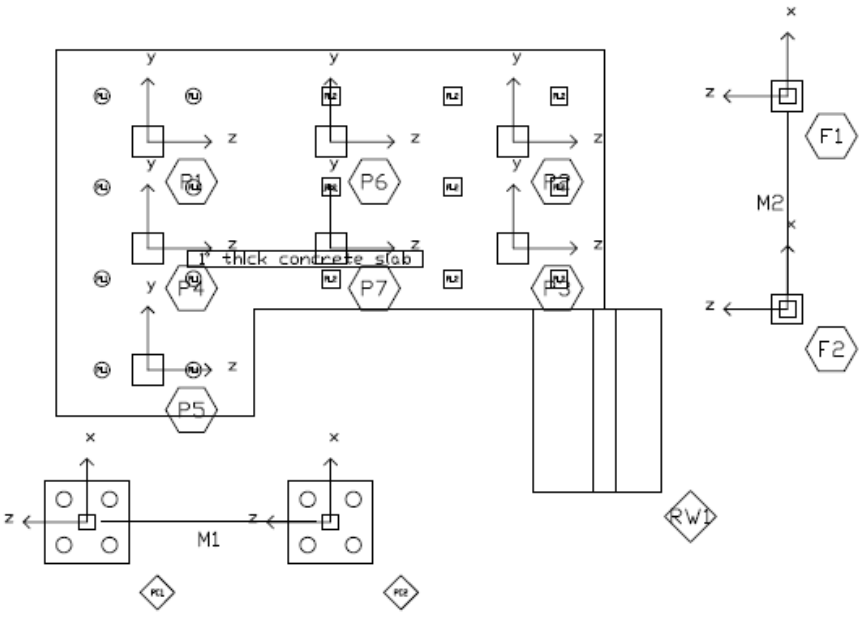
Export DXF File Options

Option	Usage / Description
Select DXF Units	Select the units you want the CAD model to be created in. The options for the DXF units are none, inches, feet, mm, cm and meters.

Option	Usage / Description
CAD Scale Factor	Enter the scale factor that causes the full scale RISA model to be scaled up or down to the desired drawing scale. For example, if you created a full scale model that you wanted scaled down to 1/4"=12", the factor would be 0.020833, which is (.25/12).
Standoff Distance	Enter the distance you wish to have the line entities “stand off” from the nodes to which they are attached. The standoff distance is measured along the axis of the line. The distance is used as entered and is not scaled by the CAD Scale factor.
Grid Bubble Type	You can export the project grid bubbles as either Circles or Hexagons.
DXF Layer Options	These controls allow you to customize the layer names in the DXF output of your Foundation Plan Drawing.

Exporting Foundation Plot Plans

Once you have all these parameters set, press OK and the program exports a plan view of the foundation model, complete with slabs, footings, grade beams, pedestals, piles, pile caps and retaining walls. Slab reinforcement can be found in the slab reinforcing details below.



Footing Pad Schedule							Footing Pedestal Schedule			
		Footing Size (ft x ft x in)	x-bars		z-bars					
Mark	No.	(X Width x Z Width x Thk)	Top	Bottom	Top	Bottom	Height (in)	Size (in x in)	Flex. Bars	Ties
F1	1	2 x 2 x 12	-	#6 @ 10.5 in max	-	#6 @ 10.5 in max	24	12 x 12	4 - #6	#4 @ 12 in max
F2	1	2 x 2 x 12	-	#6 @ 10.5 in max	-	#6 @ 10.5 in max	24	12 x 12	4 - #6	#4 @ 12 in max

Pedestal Schedule					
Mark	No.	Height (in)	Size (in x in)	Flex. Bars	Shear Ties
P1	1	12	24 x 24	8 - #6	#4 @ 12 in max
P2	2	12	24 x 24	8 - #6	#4 @ 12 in max
P3	3	16	24 x 24	8 - #6	#4 @ 16 in max
P4	4	12	24 x 24	8 - #6	#4 @ 12 in max
P5	5	12	24 x 24	8 - #6	#4 @ 12 in max
P6	6	12	24 x 24	8 - #6	#4 @ 12 in max

Pile Cap Schedule								Pile Cap Pedestal Schedule			
		Pile Cap Size (ft x ft x in)	# Pile	x-bars		z-bars					
Mark	No.	(X Width x Z Width x Thk)	Type	Top	Bottom	Top	Bottom	Height (in)	Size (in x in)	Flex. Bars	Ties
PC1	1	5.5 x 5.5 x 24	Pile 1	-	#6@10 in	-	#6@10 in	24	12 x 12	4 - #6	#4@12 in
PC2	1	5.5 x 5.5 x 24	Pile 1	-	#6@10 in	-	#6@10 in	24	12 x 12	4 - #6	#4@12 in

Pile Schedule				
Mark	No.	Shape	Diameter/Side (in)	Pile Enbed (in)
PL1	16	Round	12	4
PL2	9	Square	14	6

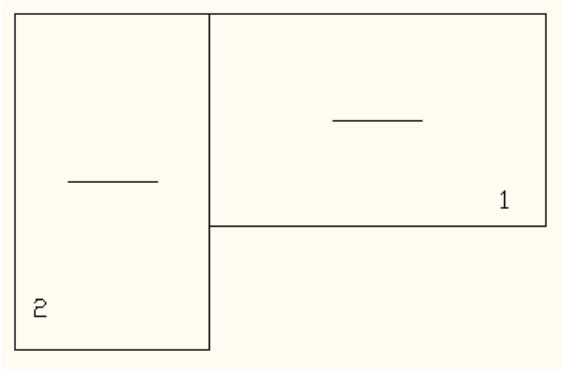
Retaining Wall Geometry Schedule							
Mark	Wall Height (ft)	Wall Thick (in)	Toe Length (ft)	Heel Length (ft)	Foot Thick (in)	Key Depth(in)	Key Width(in)
RW1	14	18	3	4	18	0	0

Retaining Wall Reinforcement Schedule							
Mark	Int Wall Verts	Ext Wall Verts	Wall Horz Bars	Foot Top Bars	Foot Bot Bars	Foot Long Bars	Dowel Bars
RW1	#7 @ 10 in	#8 @ 18 in	#4 @ 18 in	#6 @ 10 in	#5 @ 14 in	#4 @ 18 in	#7@10in,#8@18in

Grade Beam Schedule								
	Size(in x in)			Flex Reinforcing		Shear Ties		
Mark	(Width x Depth)	Span	Location	Top	Bottom	Size	Region	Max. Spacing
M1	12 x 12	1	Start	2 - #5	-	-	-	-
			Middle	-	2 - #5	-	-	-
			End	2 - #5	-	-	-	-
						-	-	-
M2	12 x 12	1	Start	2 - #5	-	-	-	-
			Middle	-	2 - #5	-	-	-
			End	2 - #5	-	-	-	-
						-	-	-

Exporting Slab Reinforcing Details

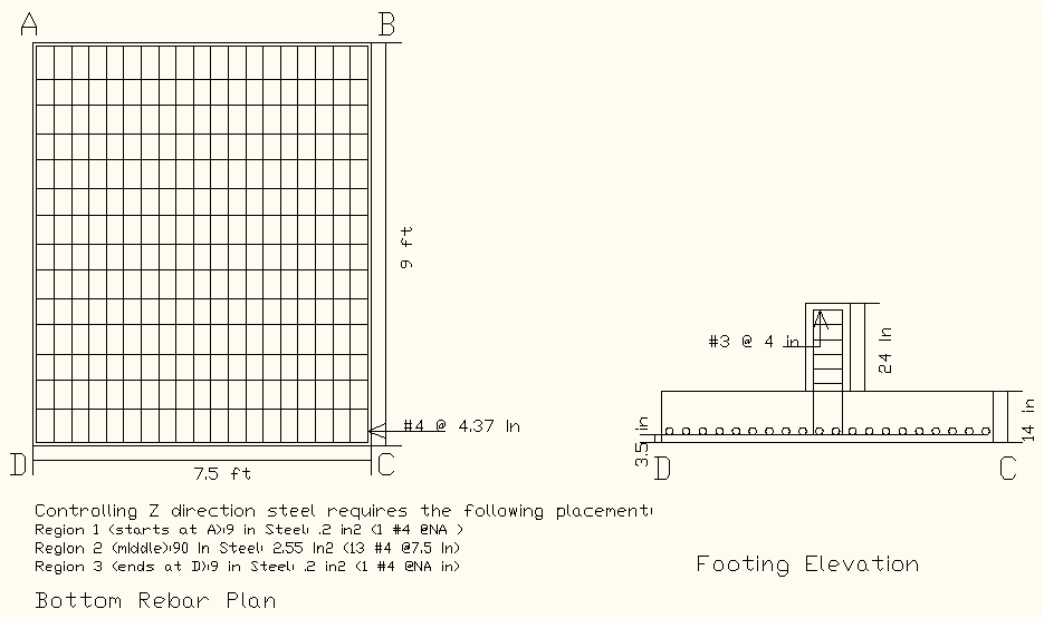
The program can export a schedule of the horizontal and vertical slab reinforcing details. Below is an example of the horizontal slab reinforcing details.

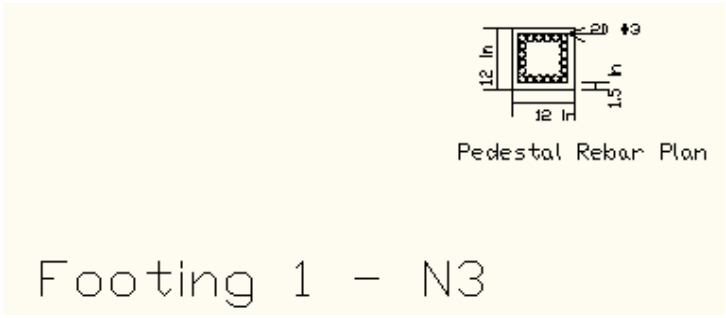


Plan Horizontal Design Strip Schedule		
	Flex Reinforcing	
Mark	Top	Bottom
1	#5 @ 18 in max	#8 @ 18 in max
2	#5 @ 18 in max	#8 @ 18 in max

Exporting Footing Details

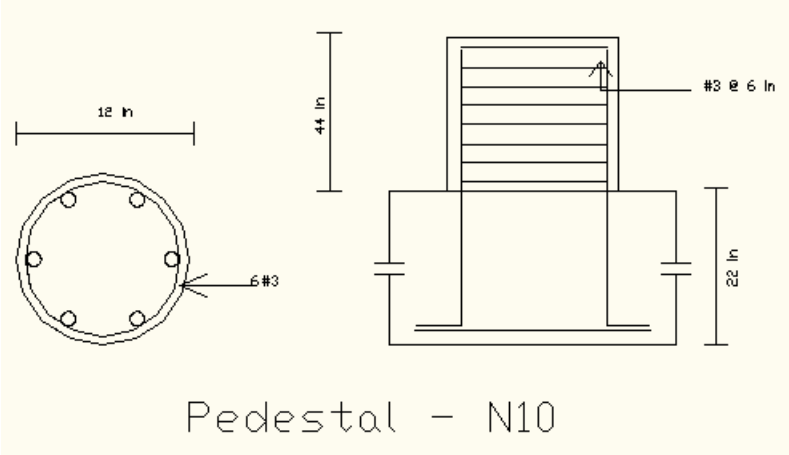
The program can export a sketch of each designed footing. The sketch shows a reinforcement plan, an elevation view and a pedestal reinforcement plan.





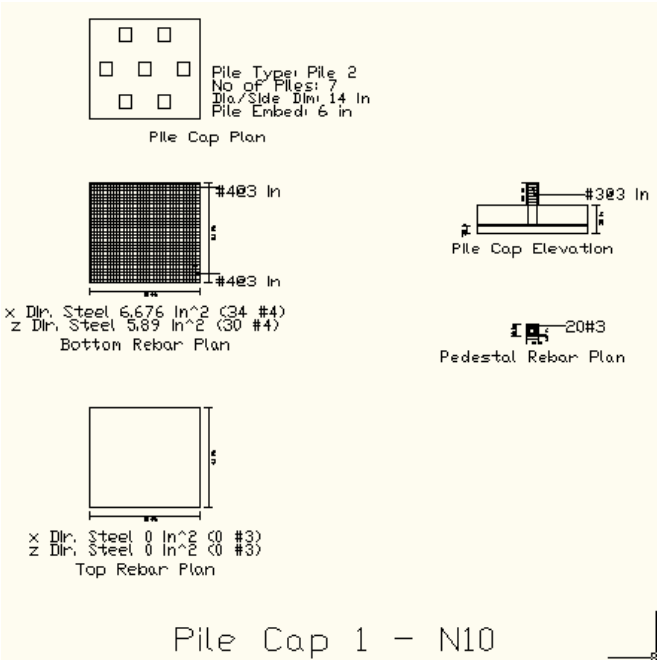
Exporting Pedestal Details

The program can export a sketch of each designed pedestal. The sketch shows the reinforcement, a plan view and an elevation view.



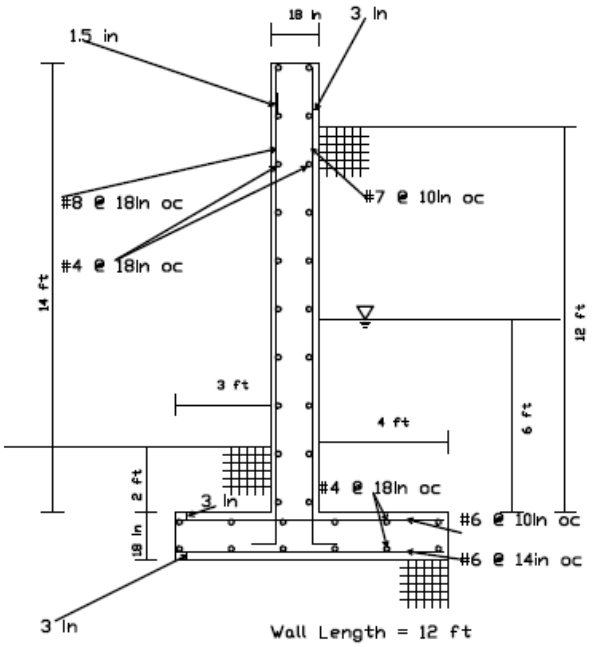
Exporting Pile Cap Details

The program can export detailed plans of each designed pile cap. The sketch shows a pile cap plan, reinforcement plans, an elevation view and a pedestal reinforcement plan.



Exporting Retaining Wall Details

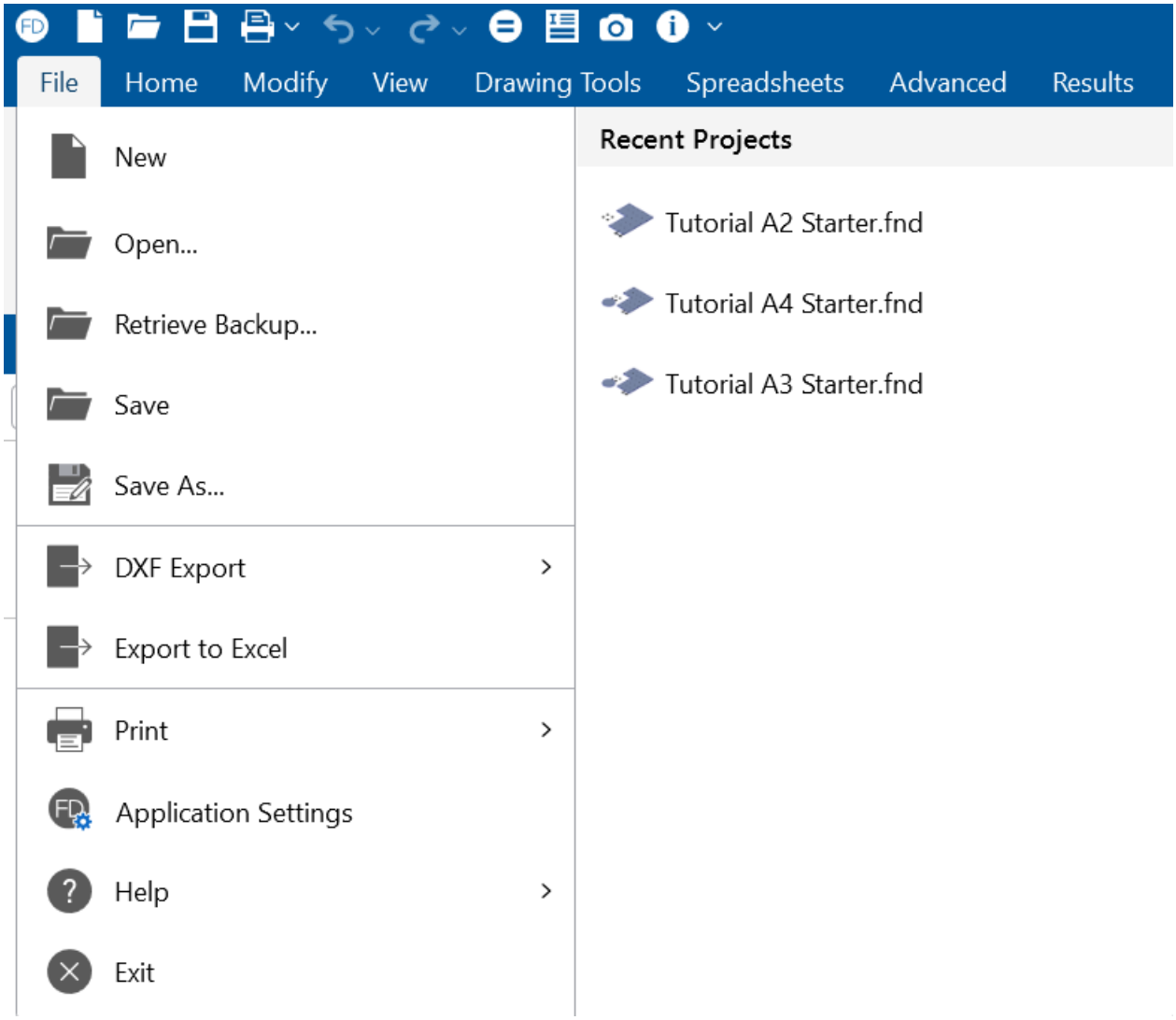
The program can export a detailed plan of each designed retaining wall. The sketch shows a section view complete with dimensions and reinforcement sizes and spacings.


18 in
3 in
1.5 in
#8 @ 18in oc
#7 @ 10in oc
#4 @ 18in oc
14 ft
12 ft
3 ft
4 ft
6 ft
2 ft
3 in
#4 @ 18in oc
#6 @ 10in oc
18 in
#6 @ 14in oc
3 in
Wall Length = 12 ft

RW1

File Operations

You may create, save and recall files from the File Menu ribbon or the Quick Access Toolbar. Exporting other file types may also be accomplished from the **File Menu**.

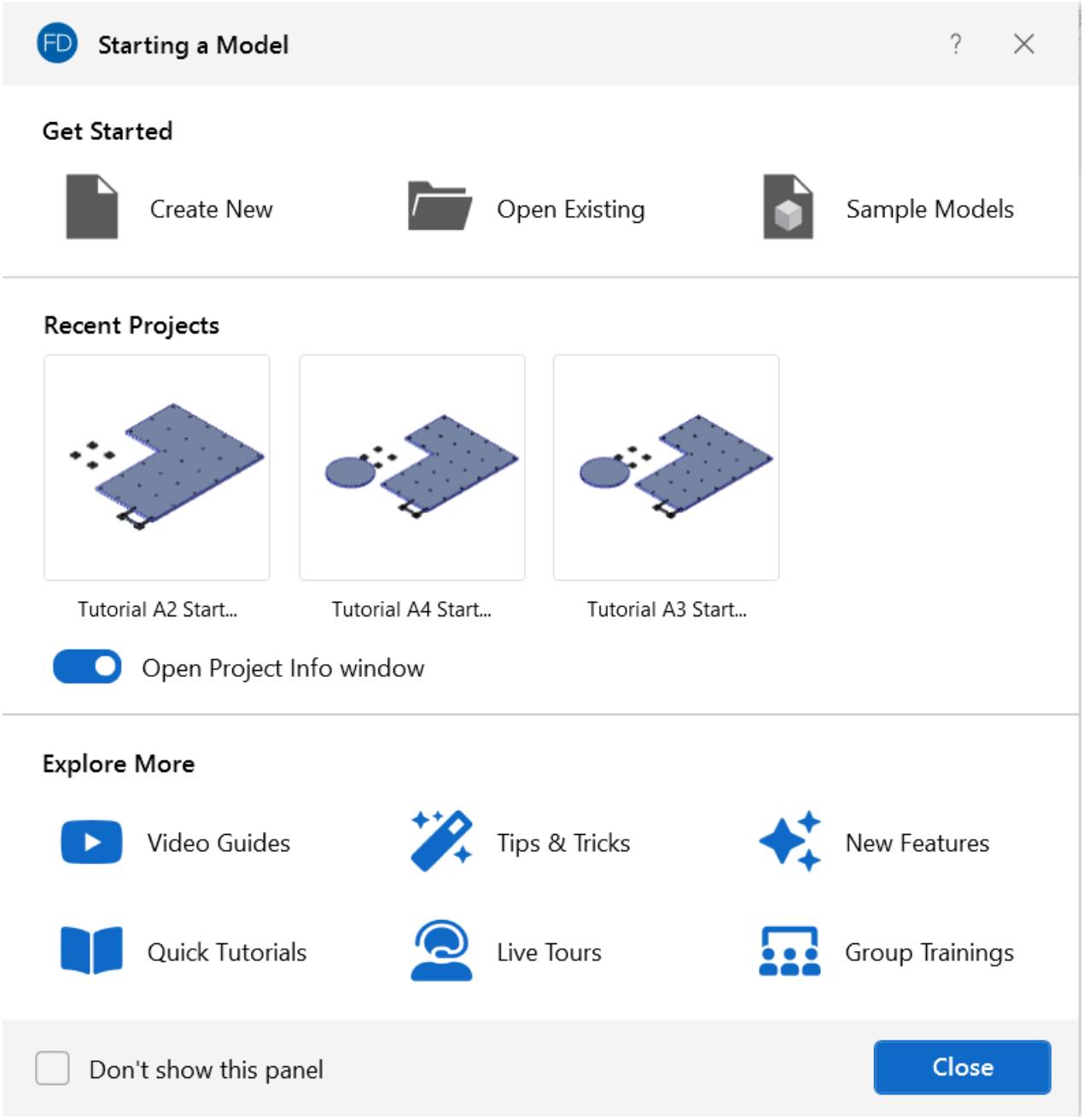


Start Off

Starting a Model Window

When you open the program, you are presented with a **Starting a Model** window that includes the following options: Create New, Open Existing, Sample Models and Recent Projects. The bottom area

of the window contains an **Explore More** section that provides links to other helpful options for learning about and working with RISAFoundation.



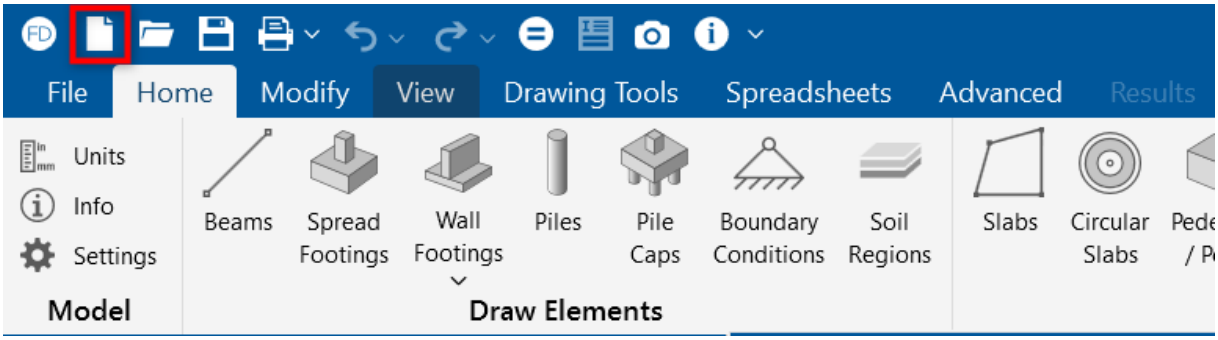
Create a New Model

To create a new model:

- Click the **Create New** icon in the Starting a Model window.

  OR

- Click the **New Model** icon on the Quick Access Toolbar if you've already closed the 'Starting a Model' window, turned off the display of it (checked the **Don't show this again** check box) or currently have a model open in the program.



If you made any changes to the current model that has not yet been saved, you will be prompted to **Save**.

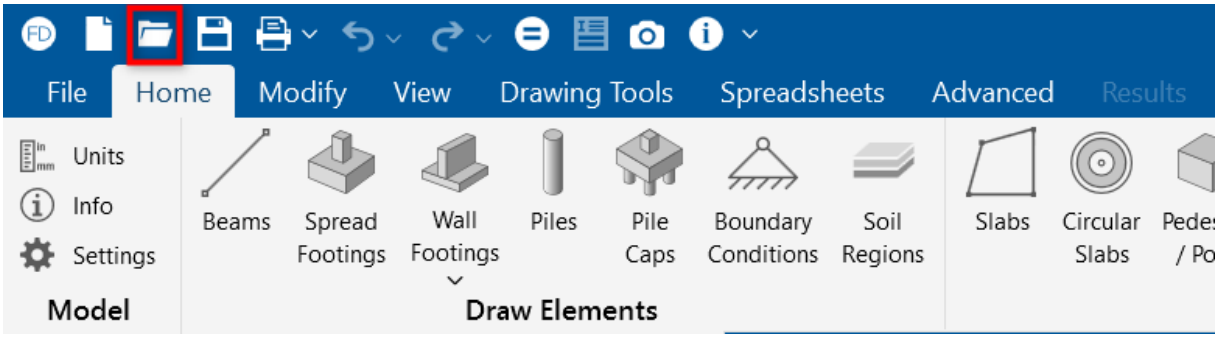
Open an Existing Model

To open an existing model:

- Click the **Open Existing** icon in the Starting a Model window.

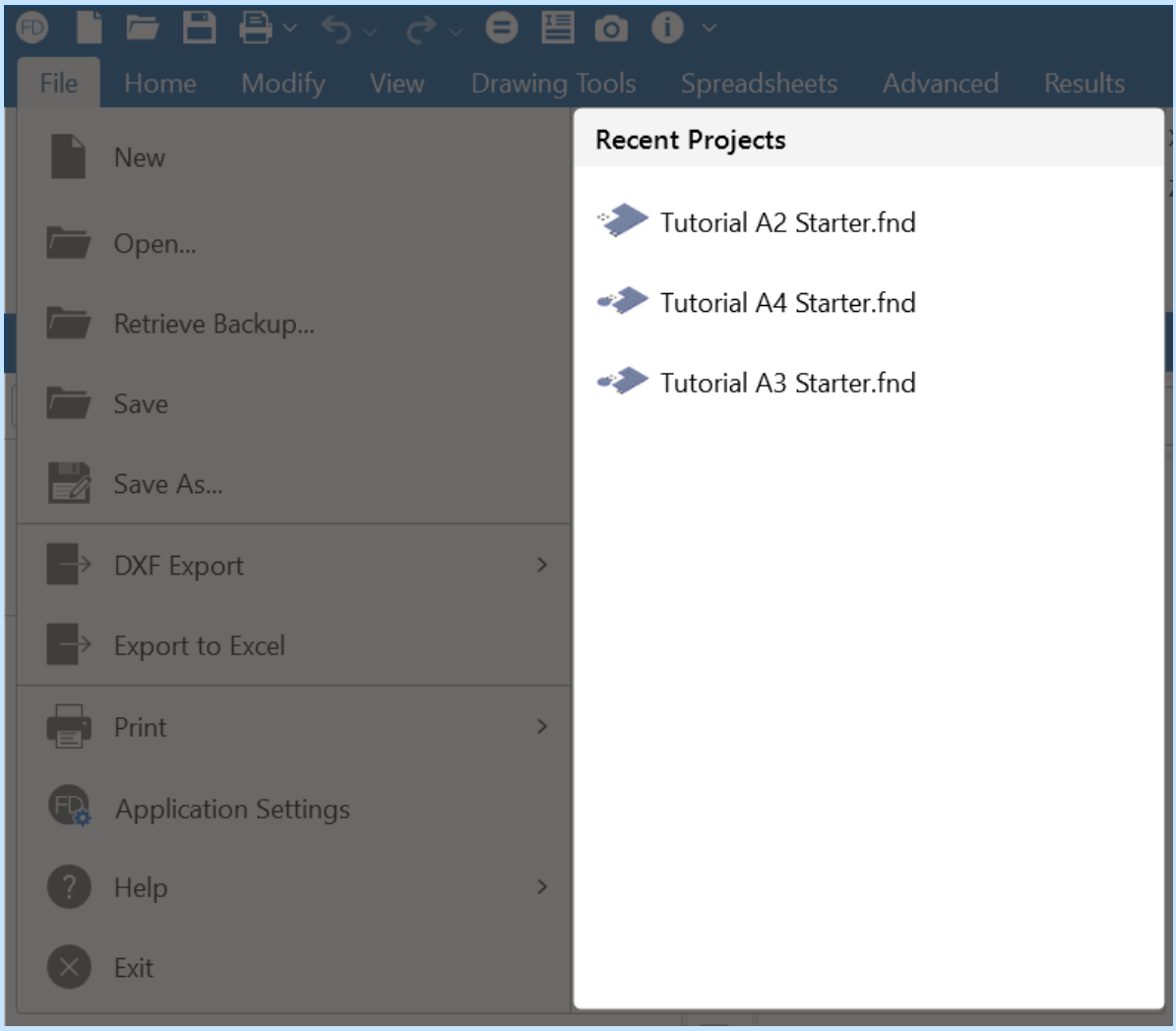
  OR

- Click the **Open Model** icon on the Quick Access Toolbar if you've already closed the 'Starting a Model' window, turned off the display of it (checked the **Don't show this again** check box) or currently have a model open in the program.



If you made any changes to the current model that has not yet been saved, you will be prompted to **Save**.

Tip:
To open a model you recently used, click the **File Menu** and select from the list of 'Recent Projects' (shown to the right of the standard menu). Each model is listed along with a preview image of the rendered structure.



Open a Sample Model

To open a sample model:

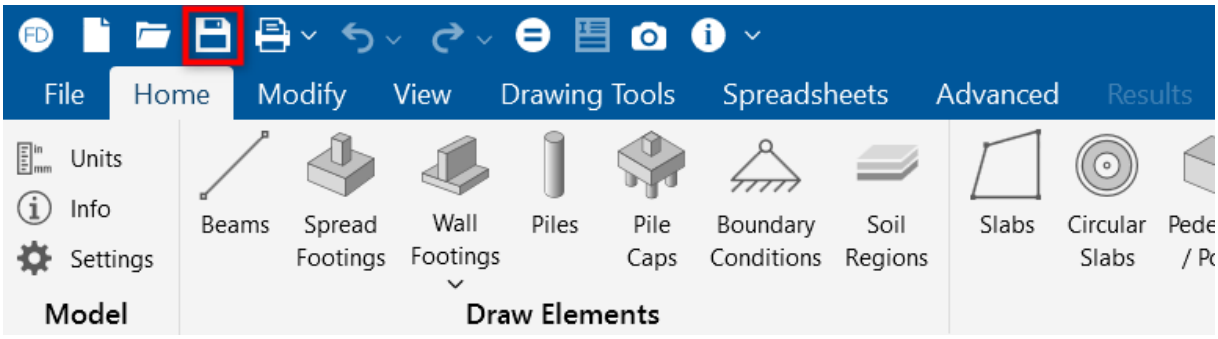
- Click the **Sample Models** icon in the Starting a Model window.

Save a Model

New Untitled Model

To save a new, untitled model:

- Click the **Save Model** icon on the Quick Access Toolbar.

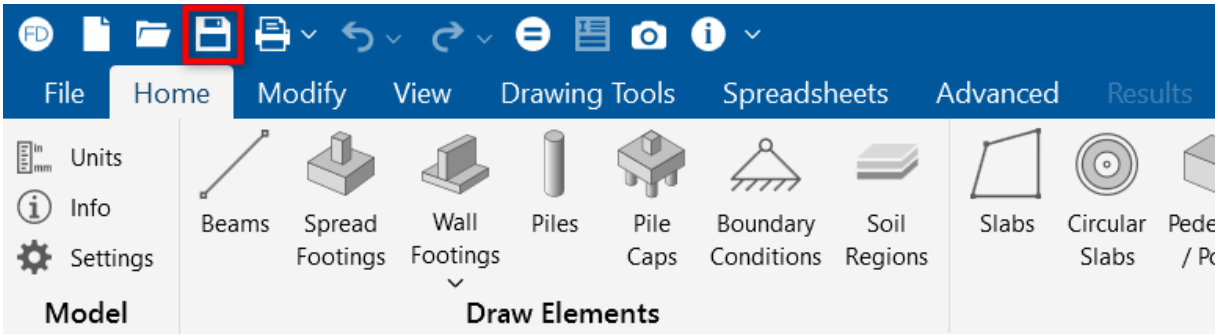


You are prompted to **Save** the model by giving it a file name and specifying a folder to save it to.

Existing Model

To save an existing model:

- Click the **Save Model** icon on the Quick Access Toolbar.



You are not prompted to save the model. Instead, the model file is simply saved with its current file name, in the folder you've previously specified.

Exporting Files

Exporting to DXF Files

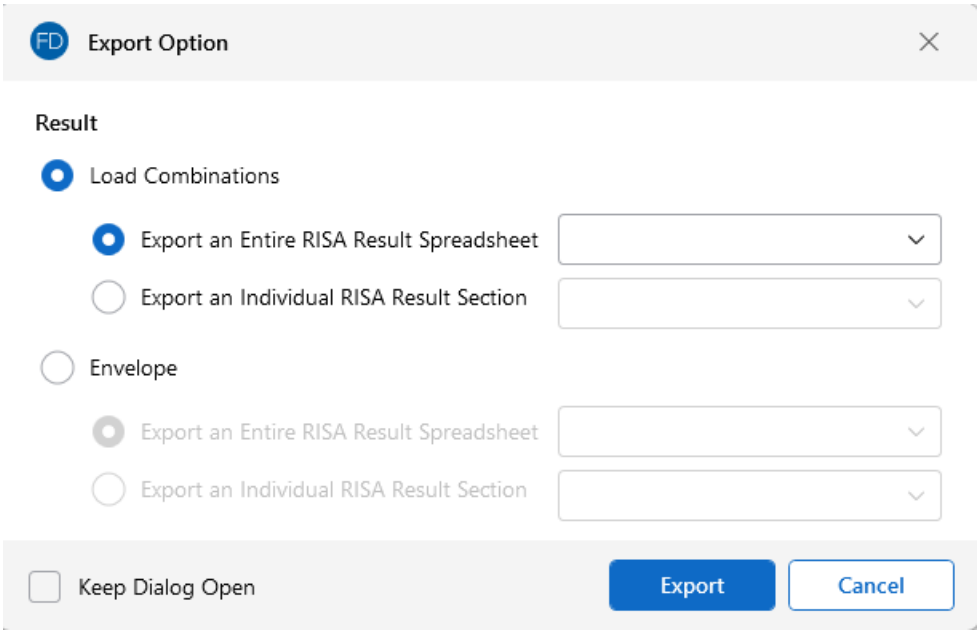
To export a .dxf file:

1. Click on the **File** menu.
2. Click on **DXF Export**.
3. Choose from the available DXF Export options.

Exporting to Excel Spreadsheet

You may export **Result Spreadsheets** into an Excel spreadsheet:

1. On the **File** menu, click **Export to Excel**.
2. Select export options, you can select to export an **Entire RISA Result Spreadsheet** or an **Individual RISA Result Section**.

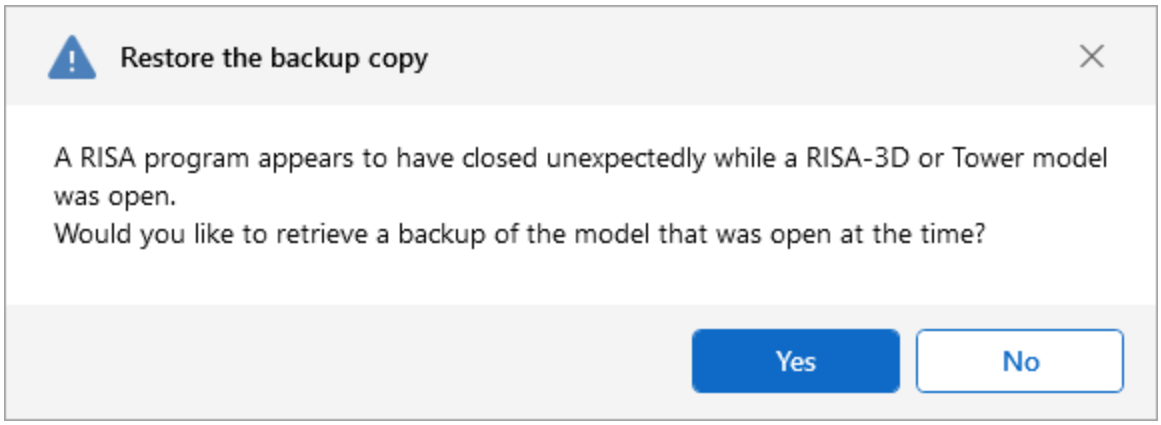


3. Select the folder to export the spreadsheet to and click **Export**.

Automatic Backup

RISAFoundation provides an automatic backup for systematically backing up your models. The purpose of this is to provide a means for you to recover your data in the event that RISAFoundation, or your computer, closes unexpectedly. As you work within RISAFoundation, your model is saved with a "_backup" suffix to a model backup folder. By default, this automatic save operation takes place every 5 minutes, but you can alter the timing and location with the **Application Settings** option in the **File Menu**. Backups for the last 100 models that you worked on are saved, although you can control how many models are backed up with the Application Settings.

If RISAFoundation does close unexpectedly, the next time you start the program the following message appears:

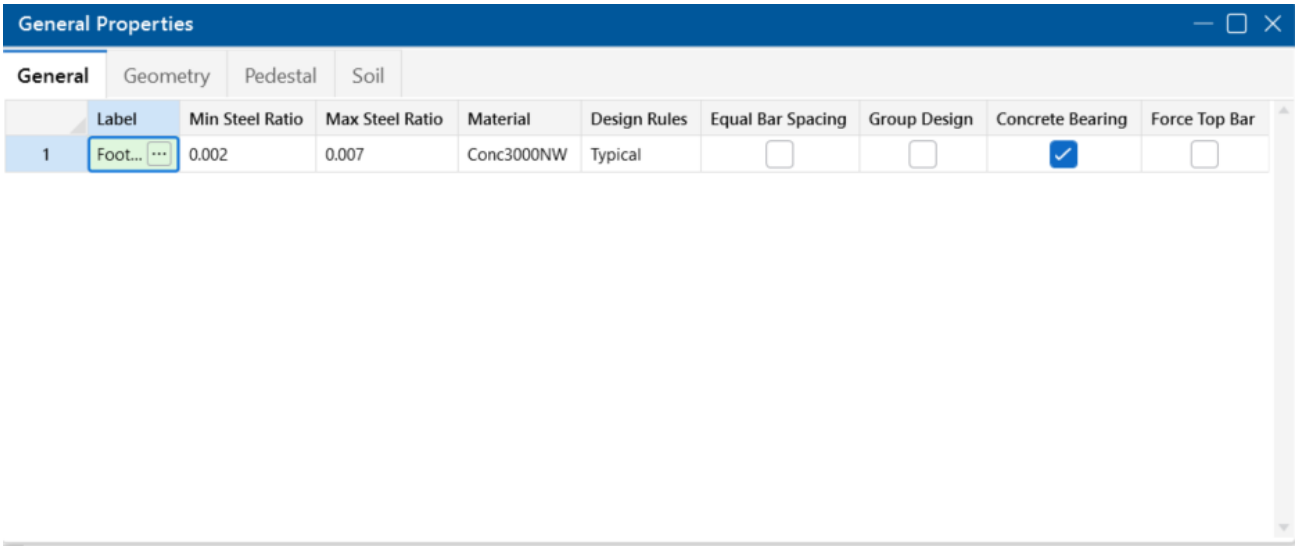

Restore the backup copy
A RISA program appears to have closed unexpectedly while a RISA-3D or Tower model was open.
Would you like to retrieve a backup of the model that was open at the time?
Yes
No

Footings - Design

Spread Footing Definitions Spreadsheet

Footing General Properties

The General tab contains material properties and other toggles specific to footing design discussed below.



Reinforcement Ratios

Acceptable steel ratios are controlled by ACI 318-14 Section 24.4.3 (ACI 318-11 Section 7.12.2.1) which stipulates that the minimum steel ratio for Grade 40 or 50 steel is .0020. For Grade 60, the ratio is 0.0018. The ratio cannot be less than 0.0014 for any grade of steel.

The required minimum flexural reinforcement under ACI 318-11 is given in Section 10.5.1 and 10.5.2. Additionally, there is a provision in Section 10.5.3 that allows you to disregard the requirements of 10.5.1 and 10.5.2 if the area of steel provided is at least 1/3 greater than what is required by analysis.

For ACI 318-14 and later, the minimum flexural reinforcement has been updated and is given in Section 8.6.1. This section states that a minimum gross concrete to steel reinforcement ratio of 0.0018 must be provided at each face where flexural steel is required. Therefore, RISA enforces this minimum for each face individually. See ACI 318-14 Sections 8.6.1. for more information.

In addition, the codes limit the maximum steel that can be used. The 1995 and 1999 codes limit the steel ratio to 75% of the balanced condition. The 2002 and newer codes instead limit the minimum strain on the reinforcing steel to 0.004. RISAFoundation doesn't specifically enforce these maximums and instead relies on this Max Steel Ratio separately for each face.

Material

This allows you to select the material from the definitions contained in the Materials Spreadsheet.

Design Rules

This allows you to select the design rule from those defined in the Design Rules Spreadsheet.

Equal Bar Spacing

Checking the **Equal Bar Spacing** option indicates that you do not want to specify banded reinforcement. If you check this box, RISAFoundation will determine the tightest spacing required per ACI 318-14 Section 13.3.3.3 (ACI 318-11 Section 15.4.4) and then use that spacing throughout the footing.

Enveloping Multiple Footing Designs

Checking the box in the **Group Design** column will cause all footings assigned to the current footing definition to use the same final size. The largest required size within the group of footings will be used for all footings within the group. The maximum required reinforcement for the footings and pedestals will also be enveloped.

Concrete Bearing Check

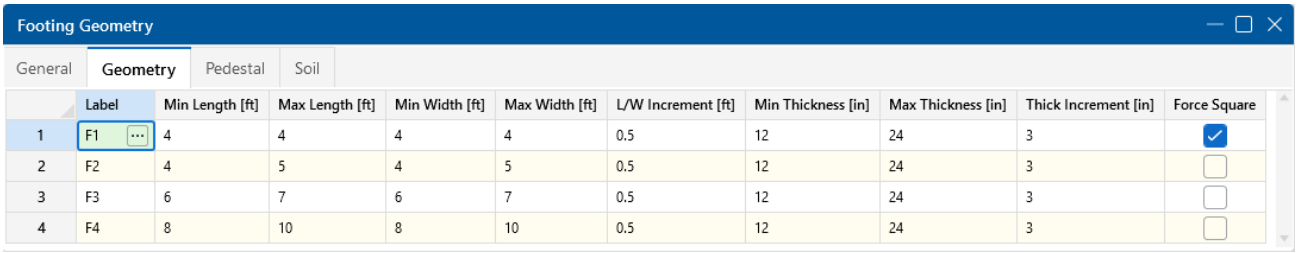
Checking the box in the **Concrete Bearing** column will force the program to check the concrete bearing failure between the footing and pedestal.

Force Top Bars

The **Force Top Bars** option allows the user to indicate that top bars are required for this footing, even for cases where the footing does not experience significant uplift or negative moment. One reason for checking this box would be to spread out the Temperature / Shrinkage reinforcement between the top and bottom layers of steel.

Footing Geometry

The **Geometry** tab contains the basic information that controls the size and thickness of the footing.



	Label	Min Length [ft]	Max Length [ft]	Min Width [ft]	Max Width [ft]	L/W Increment [ft]	Min Thickness [in]	Max Thickness [in]	Thick Increment [in]	Force Square
1	F1	4	4	4	4	0.5	12	24	3	☑
2	F2	4	5	4	5	0.5	12	24	3	☐
3	F3	6	7	6	7	0.5	12	24	3	☐
4	F4	8	10	8	10	0.5	12	24	3	☐

These parameters define the minimum and maximum length, width and thickness for the footing. When RISAFoundation sizes the footing, it optimizes for the smallest footing that fits within the

parameters and does not exceed the allowable soil bearing pressure. RISAFoundation favors footings with the lowest length/width ratio (as close to a square as possible).

If you are investigating an existing footing size that you don't want RISAFoundation to change, you may either specify the same maximum and minimum length, width and/or thickness OR specify the maximum values and set the design increments to zero.

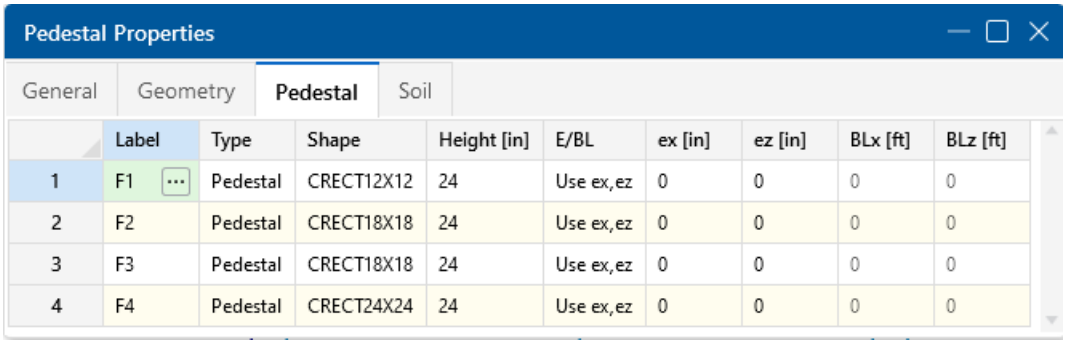
Note: The min/max length is always defined as the dimension in the local z-axis. The min/max width is always defined as the dimension in the local x-axis.

Force Footing to be Square

Checking the box in the **Force Square** column will restrict the design to square footings only.

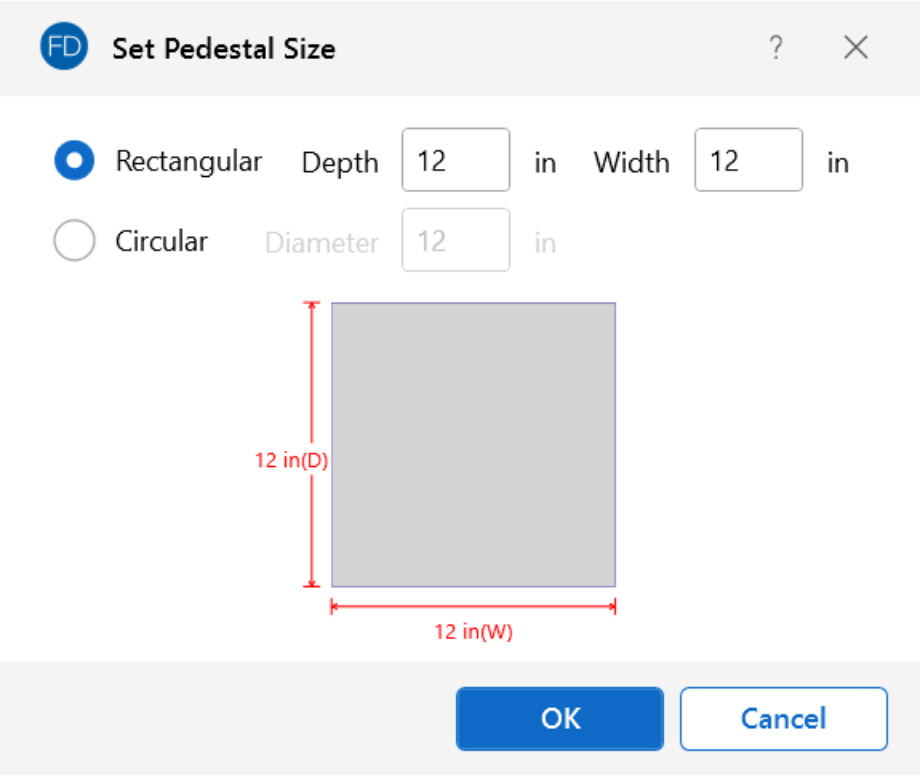
Footing Pedestal

The **Pedestal** tab contains the basic information that will control the width and height of the pedestal. It will also control the location of the pedestal if you have an eccentric footing.



	Label	Type	Shape	Height [in]	E/BL	ex [in]	ez [in]	BLx [ft]	BLz [ft]
1	F1	Pedestal	CRECT12X12	24	Use ex,ez	0	0	0	0
2	F2	Pedestal	CRECT18X18	24	Use ex,ez	0	0	0	0
3	F3	Pedestal	CRECT18X18	24	Use ex,ez	0	0	0	0
4	F4	Pedestal	CRECT24X24	24	Use ex,ez	0	0	0	0

The pedestal geometry is entered in these fields. The type can be specified as either a pedestal or post. Select the ... (ellipsis) button in the Shape cell to prompt the following dialog box where you can specify the dimensions of the pedestal:

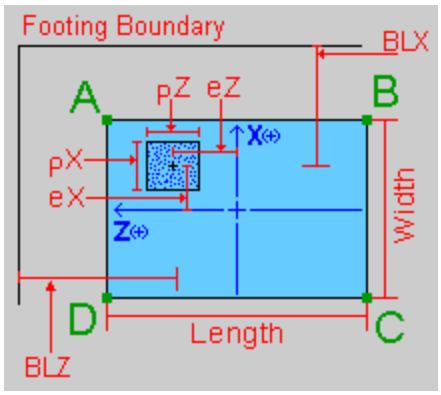


The height is the distance from the upper surface of the footing to the top of the pedestal. The pedestal location may be controlled, as discussed in the Boundaries and Eccentricities section below.

Boundaries and Eccentricities

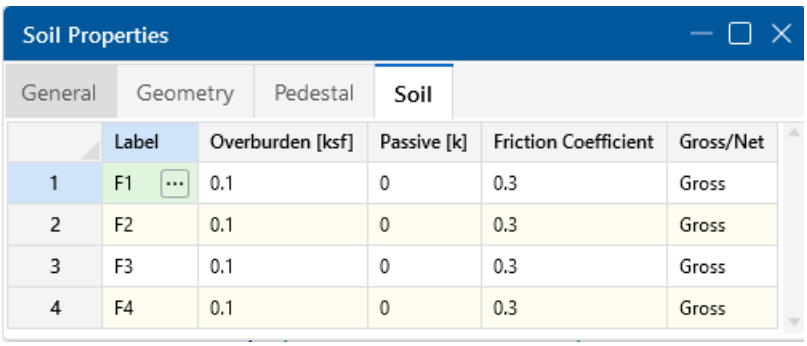
When specifying an eccentric pedestal, the **eX** and **eZ** values are the offsets from the center of the footing in the local X and Z directions, respectively. These values may be either positive or negative.

You may use the **BLx** or **BLz** entries to set boundary lines for the footing in one or two directions, or you may specify the pedestal eccentricity from the centroid of the footing. If you set boundary lines, RISAFoundation will keep the sides of the footing from violating the boundary. These values may be either positive or negative.



Soil Properties

The Soil Properties tab contains the basic information related to design for soil bearing.



Overburden

Overburden is the weight of soil or other dead loads (e.g. pavement) which exist above the footing in its final (constructed) condition. RISAFoundation applies the overburden pressure uniformly on the footing, except at the pedestal footprint.

Passive Resistance of Soil

For additional sliding resistance, you may enter the passive resistance of the soil as a lump sum value. RISAFoundation applies this value in both directions.

Coefficient of Friction

RISAFoundation multiplies this coefficient of friction by the total vertical forces to determine the soil sliding resistance.

Gross/Net

In RISAFoundation, the Allowable Soil Bearing value is controlled by the Soil Region. By default, this value is always taken as a Gross allowable soil pressure.

Allowable Soil Bearing values are typically supplied by the Geotechnical Engineer as either *Gross* or *Net* allowable values. The soil bearing check uses the same load to check soil bearing (Footing Self Weight + Overburden + Applied Load), regardless of whether Gross or Net is specified.

When a Gross soil pressure is specified, the load is compared directly against the Bearing Allowable (user-entered).

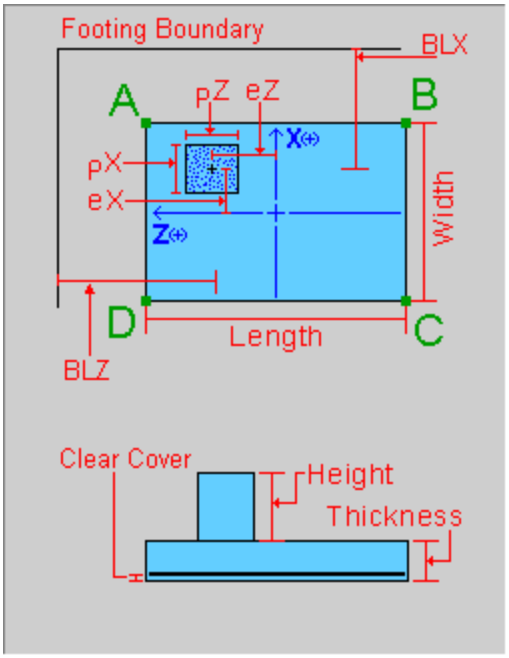
When a Net soil pressure is specified, the load is compared against a modified soil capacity, as shown below:

Soil Capacity = Bearing Allowable (user-entered) + Footing Self Weight + Overburden (user-entered).

If the sum of Footing Self Weight and Overburden *does not* equal the soil weight, which was present above the footing/soil interface strata prior to excavation, then the modified soil capacity provided by the program *is incorrect*. In this circumstance, the Net Allowable Soil Bearing value should be specified as a Gross value in the program, and the magnitude of Bearing Allowable should be increased by that pre-excavation soil weight.

Local Axes

The “A,B,C,D” notation provides a local reference system for the footing. The local z-axis is defined as positive from B towards A, and the local x-axis is defined as positive from D towards A. The origin for the local axes is the geometric center of the footing.



It is important to realize that the local footing axes do NOT necessarily correspond to the Global axes of the RISA model or the local axes of the column member.

Limitations

Non-Linear Effects

The Node Reaction results can differ from the design results used for the OTM safety factor calculation. The reason for this is that the OTM safety factor needs to determine whether each component (P, V or M) of each load creates a stabilizing or destabilizing moment.

Footing - Results

Upon completion of the analysis and footing design the results are presented in **Results Spreadsheets**, with a **Detail Report** for each footing. The Detail Report can be customized and printed. You can control what information is included in the Detail Report by clicking the **Report Options** button at the top of the **Detail Report**. Available report sections are listed for you to check those that you wish to include.

Solution Methodology

It is important to note that if the location of the load resultant is not within the footprint of the footing, then the footing is unstable. In this instance, the user is warned of the instability, which must be corrected by increasing the footing size or overturning resisting force.

The method RISAFoundation utilizes to obtain the soil pressure distribution provides an exact solution. RISAFoundation assumes the footing behaves as a rigid body and satisfies equilibrium equations. The contribution of the soil is determined through direct integration.

The calculation of the soil bearing profile for a two-way (biaxial) load state is the most complex calculation RISAFoundation makes. The soil is considered to resist compression only. If uplift is present, RISAFoundation allows that part of the soil to “let go”, so no soil tension is ever part of the soil bearing solution.

Soil bearing controls the plan dimensions of the footing; i.e. the footing is sized such that the actual maximum soil bearing pressure does not exceed the user-defined allowable soil bearing. The calculations use the service load combinations only. When an allowable bearing increase factor (ABIF) is specified, the allowable soil bearing is scaled up by the ABIF, typically a 1/3 increase. For more information, see Loads - Load Combinations.

Once the soil stress distribution has been solved, the load on the footing is known at every point. Direct integration on the contributing portion of the soil stress provides the shears and moments at the lines of interest.

Sketch and Details

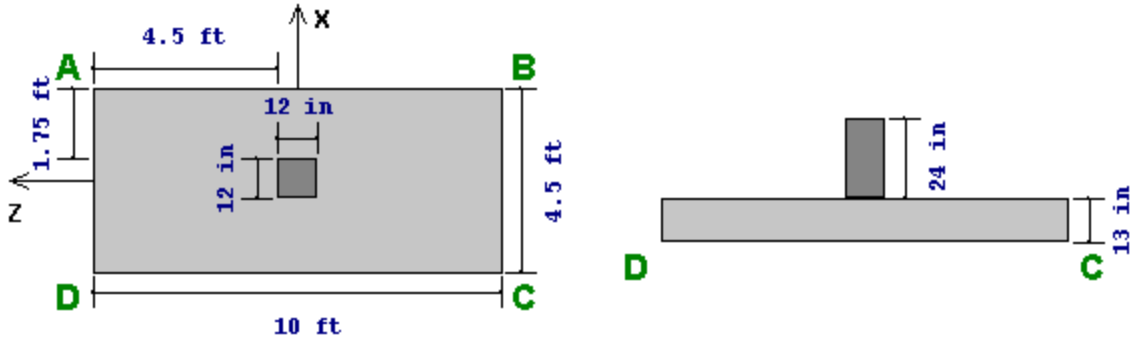
RISAFoundation will optimize the dimensions of the footing for optimum concrete volume. The final footing geometry is presented in tabular form within the **Footing Results Spreadsheet** and graphically in the **Footing Detail Report**. The orientation of the footing is based on the footing's local x- and z-axes, respectively.

Spread Footing Geometry Results

Spread Footing Code Check | Spread Footing Shear Code Check | Geometry | Reinforcement | Pedestal Code Check | Stability

	Node	Footing	Length[ft]	Width[ft]	Thickness[in]	ex[in]	ez[in]	Pedestal Ht[in]	Ped Xdim[in]	Ped Zdim[in]
1	N37	Footing 1	2	3	12	0	0	24	12	12
2	N38	Footing 1	2	3	12	0	0	24	12	12
3	N40	Footing 1	2	3.5	12	0	0	24	12	12
4	N42	Footing 1	2	3.5	12	0	0	24	12	12
5	N44	Footing 1	2	3.5	12	0	0	24	12	12
6	N43	Footing 1	2	3.5	12	0	0	24	12	12

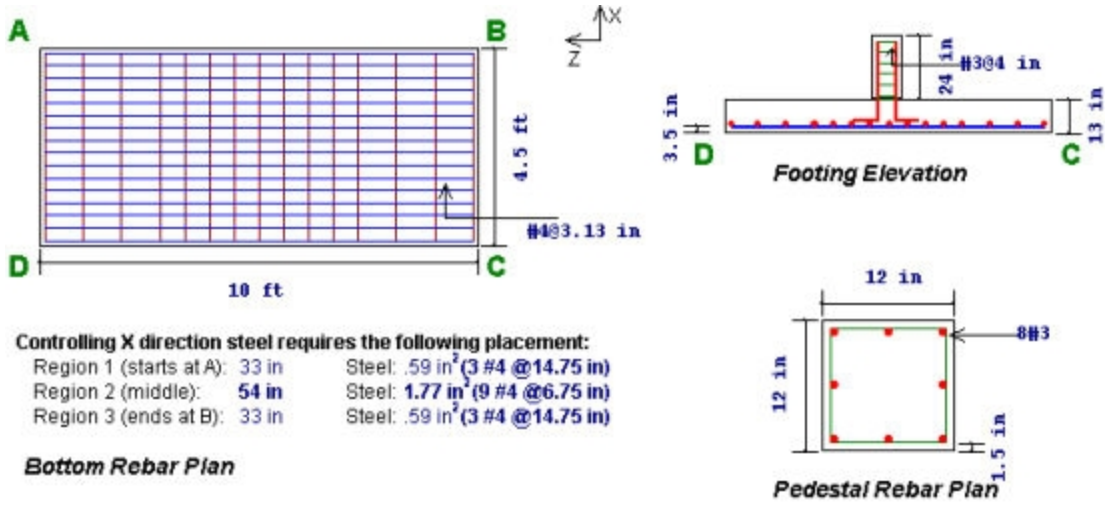
Footing Sketch



The **Detail Report** presents a sketch of the footing with dimensions (using the A,B,C,D reference system discussed in the Geometry section).

Footing Details

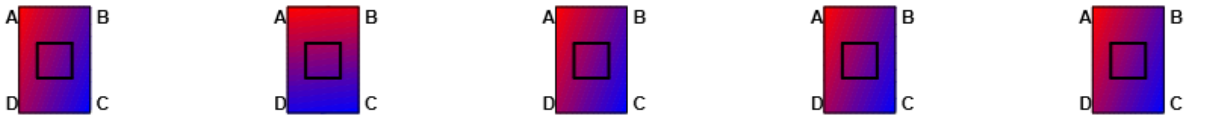
Footing details, shown below, may be included in the report. The details may also be exported to a DXF CAD file using the **Export** option on the **File** menu.



Soil Bearing Results

RISAFoundation displays the **Soil Bearing** check in a spreadsheet format on the Code Checks tab of the **Footing Results Spreadsheet**. The RISAFoundation Detail Report presents the soil bearing results for all service combinations. A soil bearing contour plot for each combination can also be displayed.

Description	Categories and Factors	Gross Allow.(ksf)	Max Bearing (ksf)	Max/Allowable Ratio
IBC 16-8	1DL	5	3.648 (A)	0.73
IBC 16-9	1DL+1HL+1LL+1LLS	5	4.78 (A)	0.956
IBC 16-10 (a)	1DL+1HL+1RLL	5	3.648 (A)	0.73
IBC 16-10 (b)	1DL+1HL+1SL+0.75SLN	5	3.648 (A)	0.73
IBC 16-10 (c)	1DL+1HL+1RL	5	3.648 (A)	0.73
IBC 16-11 (a)	1DL+1HL+0.75LL+0.75LLS..	5	4.497 (A)	0.899
IBC 16-11 (b)	1DL+1HL+0.75LL+0.75LLS..	5	4.497 (A)	0.899
IBC 16-11 (c)	1DL+1HL+0.75LL+0.75LLS..	5	4.497 (A)	0.899



1DL
QA: 3.648 ksf
QB: 2.619 ksf
QC: 2.052 ksf
QD: 3.081 ksf
NAZ: 85.092 in
NAX: 231.756 in

1DL+1HL+1LL+1..
QA: 4.78 ksf
QB: 4.658 ksf
QC: 3.33 ksf
QD: 3.452 ksf
NAZ: 942.766 in
NAX: 129.591 in

1DL+1HL+1RLL
QA: 3.648 ksf
QB: 2.619 ksf
QC: 2.052 ksf
QD: 3.081 ksf
NAZ: 85.092 in
NAX: 231.756 in

1DL+1HL+1SL+0..
QA: 3.648 ksf
QB: 2.619 ksf
QC: 2.052 ksf
QD: 3.081 ksf
NAZ: 85.092 in
NAX: 231.756 in

1DL+1HL+1RL
QA: 3.648 ksf
QB: 2.619 ksf
QC: 2.052 ksf
QD: 3.081 ksf
NAZ: 85.092 in
NAX: 231.756 in

RISAFoundation displays color contoured images of the soil bearing profile, along with the bearing pressure magnitudes at each footing corner (QA through QD). Also provided is the location of the neutral axis (the line between compressive stress and no stress due to uplift). The location is given by

NAX and NAZ values. These are the distances from the maximum stress corner to the neutral axis in the X and Z directions, respectively.

Footing Flexure Design

The flexural design results include the maximum moment and required steel for each direction for each design load combination is presented in the **Spread Footing Detail Report**. The controlling required steel values are highlighted. This information is also reported in the **Code Check** and **Reinforcement** tabs of the **Footing Results Spreadsheet**. The direction of the xx and zz moments is based on the orientation of the footing and the footing's local x- and z-axes, respectively.

Minimum flexural steel and minimum temperature and shrinkage steel are also listed in the **Detail Report**for reference.

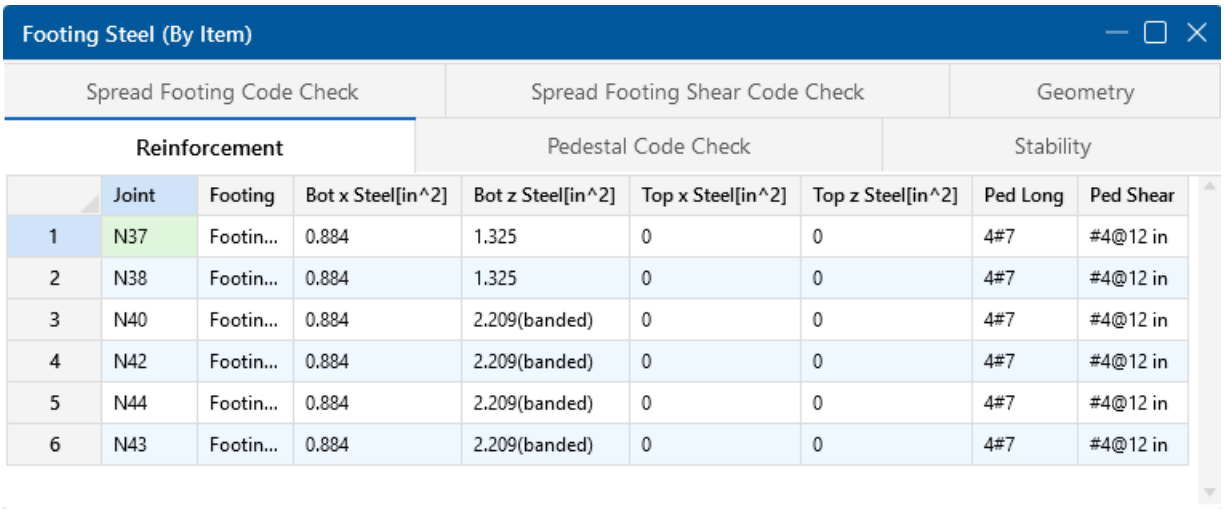
Footing Flexure Design			2	5.792 k-ft	31.081 k-ft	0.186	PASS
As-min x-dir (Top Flexure) (in^2):	0			As-min z-dir (T & S) (in^2):	0.778		
As-min z-dir (Top Flexure) (in^2):	0			As-min x-dir (Bot Flexure) (in^2):	0.518		
As-min x-dir (T & S) (in^2):	0.518			As-min z-dir (Bot Flexure) (in^2):	0.778		

Description	Categories and Factors	Mu-xx UC Max	Mu-xx (k-ft)	z-Dir As Required (in^2)	z-Dir As Provided (in^2)	Mu-zz UC Max	Mu-zz (k-ft)	x-Dir As Required (in^2)	x-Dir As Provided (in^2)
IBC 16-1	1.4DL	0.03416	1.59	0.043	1.325	0.12726	3.96	0.107	0.884
IBC 16-2 (a)	1.2DL+1.6LL+1.6L..	0.04138	1.93	0.052	1.325	0.18635	5.79	0.157	0.884
IBC 16-2 (b)	1.2DL+1.6LL+1.6L..	0.04138	1.93	0.052	1.325	0.18635	5.79	0.157	0.884
IBC 16-2 (c)	1.2DL+1.6LL+1.6L..	0.04138	1.93	0.052	1.325	0.18635	5.79	0.157	0.884
IBC 16-3 (a)	1.2DL+1.6RLL+1.6..	0.03261	1.52	0.041	1.325	0.13323	4.14	0.112	0.884
IBC 16-3 (c)	1.2DL+1.6SL+1.6S..	0.03261	1.52	0.041	1.325	0.13323	4.14	0.112	0.884
IBC 16-3 (e)	1.2DL+1.6RL+1.6H..	0.03261	1.52	0.041	1.325	0.13323	4.14	0.112	0.884

Footing Code Check (By Item)

Spread Footing Code Check | Spread Footing Shear Code Check | Geometry | Reinforcement | Pedestal Code Check | Stability

	Joint	Footing	Bearing Ratio	Bearing Pressure[ksf]	Gov LC	UC Max	Muxx[k-ft]	Gov LC	UC Max	Muzz[k-ft]	Gov LC
1	N37	Footin...	0.956	4.78	9	0.041	1.929	2	0.186	5.792	2
2	N38	Footin...	0.967	4.834	9	0.044	2.042	2	0.177	5.511	2
3	N40	Footin...	0.961	3.844	9	0.028	2.139	2	0.246	7.64	2
4	N42	Footin...	0.961	3.844	9	0.028	2.139	2	0.246	7.64	2
5	N44	Footin...	0.961	3.844	9	0.028	2.139	2	0.246	7.64	2
6	N43	Footin...	0.961	3.844	9	0.028	2.139	2	0.246	7.64	2


Footing Steel (By Item)

Spread Footing Code Check | Spread Footing Shear Code Check | Geometry

Reinforcement | Pedestal Code Check | Stability

	Joint	Footing	Bot x Steel[in^2]	Bot z Steel[in^2]	Top x Steel[in^2]	Top z Steel[in^2]	Ped Long	Ped Shear
1	N37	Footin...	0.884	1.325	0	0	4#7	#4@12 in
2	N38	Footin...	0.884	1.325	0	0	4#7	#4@12 in
3	N40	Footin...	0.884	2.209(banded)	0	0	4#7	#4@12 in
4	N42	Footin...	0.884	2.209(banded)	0	0	4#7	#4@12 in
5	N44	Footin...	0.884	2.209(banded)	0	0	4#7	#4@12 in
6	N43	Footin...	0.884	2.209(banded)	0	0	4#7	#4@12 in

To select the required flexural reinforcing steel for the footing, RISAFoundation considers moments at the face of the pedestal on all four sides. The soil bearing profile from the edge of the footing to the face of the pedestal is integrated to obtain its volume and centroid, which are in turn used to calculate the moments and shears. This moment is then used to calculate the required steel using standard flexural methods.

RISAFoundation reports the reinforcing steel as a total required area for each direction. ACI 318 also requires that short direction reinforcing in a rectangular footing be concentrated under the pedestal (this is referred to as banding), per the requirements of ACI 318-14 Section 13.3.3.3 (ACI 318-11 Section 15.4.4). RISAFoundation will report the proper steel distribution for this circumstance.

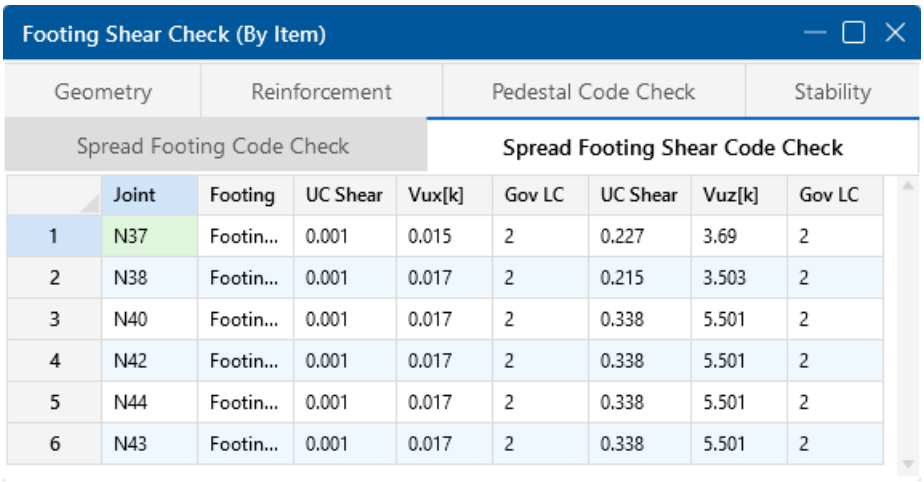
The minimum steel required for the footing is maintained based on the ratio entered on the **General** tab of the **Footing Definitions Spreadsheet**.

Top reinforcing may be required when the footing goes into partial uplift, resulting in a negative moment due to soil overburden and footing self weight. When this happens, RISAFoundation will first check to see if the plain (unreinforced) concrete section is sufficient to resist the negative moment, per ACI 318-14 Chapter 14 (ACI 318-11 Chapter 22). If not, then top reinforcing will be provided. The Load Combination shown under **Categories and Factors** in the **Detail Report** shows (in parentheses) the controlling load combination for the Z direction, then X direction for the Top bar design.

Footing Shear Check

An important part of footing design is ensuring that the footing can adequately resist the shearing stresses. Shear checks are performed for each design combination. RISAFoundation checks punching shear, as well as one-way (beam) shear in each direction. The shear check results are presented as demand vs. capacity ratios, where the actual shear is divided by the available shear capacity, thus any ratio above 1.0 would represent failure.

This information is displayed on the **Spread Footing Shear Code Check** tab, and also in the **Footing Detail Report**.



	Joint	Footing	UC Shear	Vux[k]	Gov LC	UC Shear	Vuz[k]	Gov LC
1	N37	Footin...	0.001	0.015	2	0.227	3.69	2
2	N38	Footin...	0.001	0.017	2	0.215	3.503	2
3	N40	Footin...	0.001	0.017	2	0.338	5.501	2
4	N42	Footin...	0.001	0.017	2	0.338	5.501	2
5	N44	Footin...	0.001	0.017	2	0.338	5.501	2
6	N43	Footin...	0.001	0.017	2	0.338	5.501	2

Two Way (Punching) Vc: NA **One Way (x Dir. Cut) Vc (k):** 32.535 **One Way (z Dir. Cut) Vc (k):** 21.69

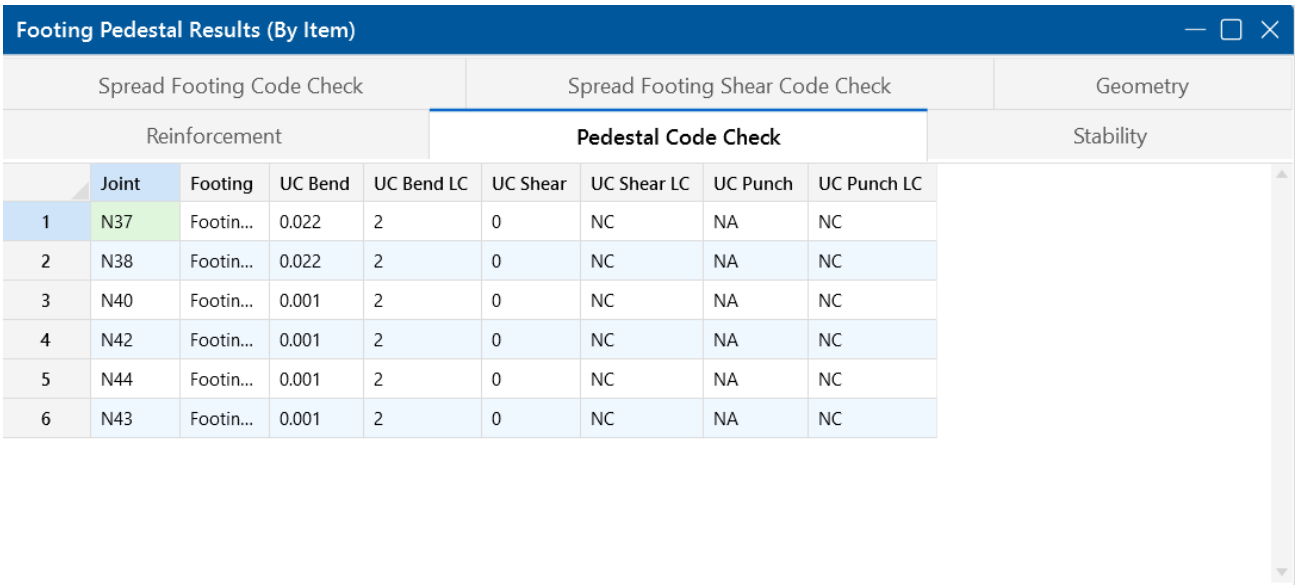
Description	Categories and Factors	Punching Vu(k)	Punching Vu/(ϕVc)	x Dir. Cut Vu(k)	x Dir. Cut Vu/(ϕVc)	z Dir. Cut Vu(k)	z Dir. Cut Vu/(ϕVc)
IBC 16-1	1.4DL	NA	NA	0.013	0	2.501	0.154
IBC 16-2 (a)	1.2DL+1.6LL+1.6LLS+1.6..	NA	NA	0.015	0	3.69	0.227
IBC 16-2 (b)	1.2DL+1.6LL+1.6LLS+1.6..	NA	NA	0.015	0	3.69	0.227
IBC 16-2 (c)	1.2DL+1.6LL+1.6LLS+1.6..	NA	NA	0.015	0	3.69	0.227

One-way and two-way shear are checked per ACI 318-14 Section 13.2.7 (ACI 318-11 Section 15.5). RISAFoundation checks shear assuming only the concrete resists the applied shear; the contribution of the reinforcing steel to shear resistance is ignored. Two-way punching shear checks for the pedestal are reported on the **Pedestal** tab of the **Spread Footing Design Spreadsheet**.

One-way shear is calculated for a “cut” through the footing a distance “d” from the face of the pedestal, where “d” is the distance from the top of the footing to the centerline of the reinforcing steel. For RISAFoundation, the “d” distance value is equal to the footing thickness minus the rebar cover and one bar diameter (since it is not known how the reinforcing is placed, the interface between the bars in each direction is used). The soil bearing profile from the edge of the footing to the cut location is integrated to obtain the total shearing force that must be resisted by the concrete. This is done for both directions (width cut and length cut) on both sides of the pedestal for a total of four one-way shear calculations.

Pedestal Design

RISAFoundation will design the longitudinal and shear reinforcement for rectangular pedestals. This information is presented in tabular form on the **Footing Pedestal Results Spreadsheet** and in the **Footing Detail Report**.

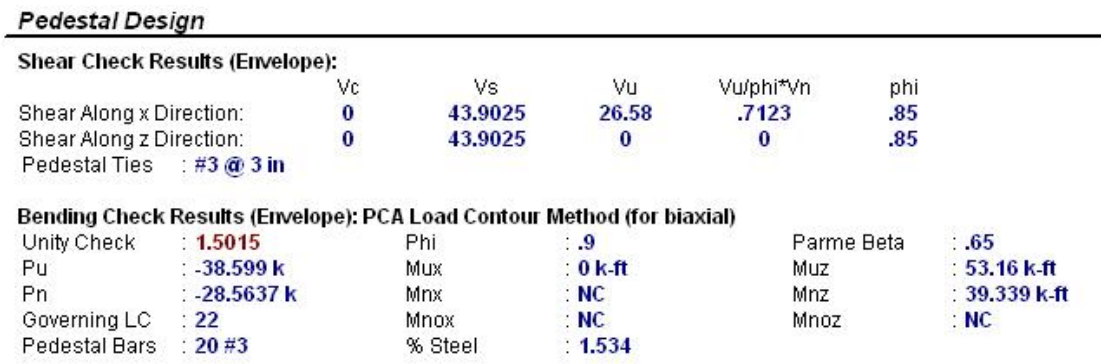


Footing Pedestal Results (By Item)

Spread Footing Code Check | Spread Footing Shear Code Check | Geometry

Reinforcement | Pedestal Code Check | Stability

	Joint	Footing	UC Bend	UC Bend LC	UC Shear	UC Shear LC	UC Punch	UC Punch LC
1	N37	Footin...	0.022	2	0	NC	NA	NC
2	N38	Footin...	0.022	2	0	NC	NA	NC
3	N40	Footin...	0.001	2	0	NC	NA	NC
4	N42	Footin...	0.001	2	0	NC	NA	NC
5	N44	Footin...	0.001	2	0	NC	NA	NC
6	N43	Footin...	0.001	2	0	NC	NA	NC



Pedestal Design

Shear Check Results (Envelope):

	Vc	Vs	Vu	Vu/phi*Vn	phi
Shear Along x Direction:	0	43.9025	26.58	.7123	.85
Shear Along z Direction:	0	43.9025	0	0	.85

Pedestal Ties : #3 @ 3 in

Bending Check Results (Envelope): PCA Load Contour Method (for biaxial)

Unity Check	: 1.5015	Phi	: .9	Parme Beta	: .65
Pu	: -38.599 k	Mux	: 0 k-ft	Muz	: 53.16 k-ft
Pn	: -28.5637 k	Mnx	: NC	Mnz	: 39.339 k-ft
Governing LC	: 22	Mnox	: NC	Mnoz	: NC
Pedestal Bars	: 20 #3	% Steel	: 1.534		

Pedestal Flexure

For flexure, RISAFoundation uses a rectangular stress block. For biaxial bending situations, the load contour method is used. RISAFoundation does not account for slenderness effects.

For flexure design, RISAFoundation presents the required longitudinal reinforcement, assuming an equal distribution of bars around the perimeter of the pedestal. Supporting information includes the axial and bending contributions to resistance Pn, Mnx and Mnz. The total factored axial and bending loads Pu, Mux and Muz are also listed. The contour method beta factor and Mnox and Mnoz are given, with the moments representing the nominal uniaxial bending capacity at the axial load Pn. Finally, the Phi factor and the resulting unity check (demand vs. capacity) ratio is given.

Note that for vertical reinforcing design, RISAFoundation will meet the provisions of the ACI 318-14 Section 16.3.4.1 (ACI 318-11 Section 15.8.2.1) which states that in pedestals you must have a minimum reinforcement of 0.5% of the pedestal cross section.

Pedestal Shear

For shear design in either direction, RISAFoundation presents the required tie spacing. Also given are the concrete and steel contributions to resisting shear Vc and Vs along with the total factored load, Vu. Finally, the Phi factor and the resulting demand vs. capacity ratios are given.

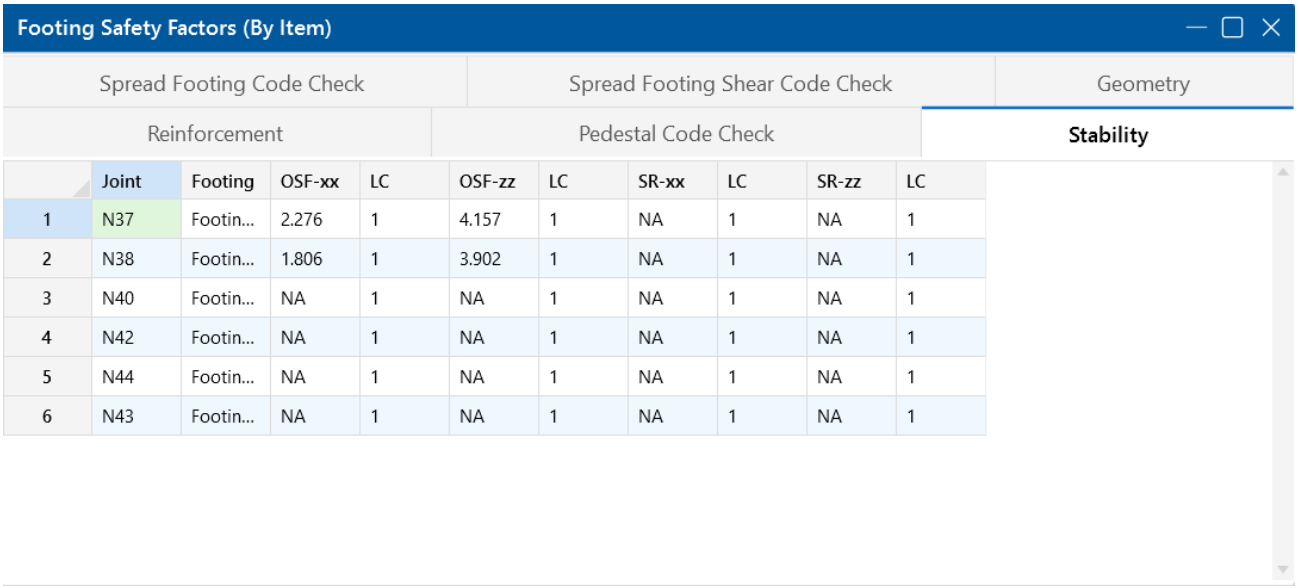
Pedestal Punching Shear

Punching shear is calculated for a perimeter at a distance d/2 from the pedestal all the way around the pedestal. The soil bearing profile is integrated, minus the volume bounded by this perimeter, to obtain the total punching force to be resisted. For cases where the pedestal is near a corner of the footing, RISAFoundation considers the punching perimeter to extend out to the footing edge if the pedestal is within a distance d from the edge of the footing.

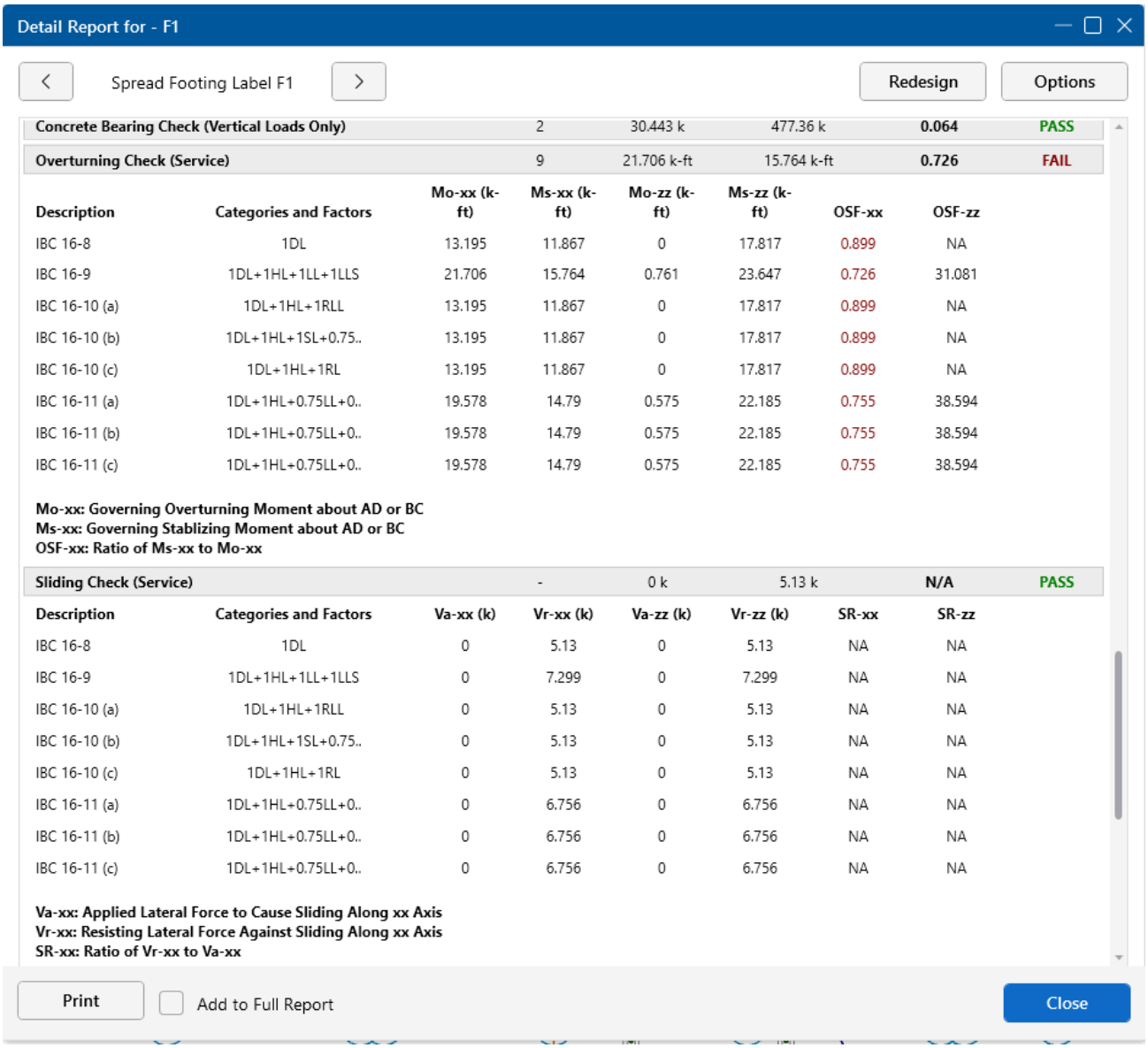
RISAFoundation presents the Vu and Vc values for punching and one-way shears, as well as the shear demand vs. capacity ratio. As long as this ratio is less than 1.0, the footing is adequate for the shear demand.

Stability Results

RISAFoundation will check the stability results for overturning and sliding. This information is presented in tabular form on the **Stability** tab of the **Footing Results Spreadsheet** and graphically in the **Footing Detail Report**.



	Joint	Footing	OSF-xx	LC	OSF-zz	LC	SR-xx	LC	SR-zz	LC
1	N37	Footin...	2.276	1	4.157	1	NA	1	NA	1
2	N38	Footin...	1.806	1	3.902	1	NA	1	NA	1
3	N40	Footin...	NA	1	NA	1	NA	1	NA	1
4	N42	Footin...	NA	1	NA	1	NA	1	NA	1
5	N44	Footin...	NA	1	NA	1	NA	1	NA	1
6	N43	Footin...	NA	1	NA	1	NA	1	NA	1



Overturning Check

Overturning calculations are presented, as shown in the figures above. For all service combinations, both the overturning moments and the stabilizing moments are given, along with the calculated safety factor.

Overturning moments are those applied moments, shears and uplift forces that seek to cause the footing to become unstable and turn over. Resisting moments are those moments that resist overturning and seek to stabilize the footing. The overturning safety factor (OSF) is the sum of resisting moments divided by the sum of overturning moments. Most codes require that this factor be

greater than 1.5. OSF calculations are based on the service load combinations only and are calculated in both the X and Z directions.

Overturning results from applied moments, applied shears and the summation of all these forces become the overall “overturning moment”.

Note:

- The OSF is calculated separately in both the X and Z directions as resisting moment / overturning moment. Overturning is evaluated about all edges of the footing and the worst case in each direction is reported.
- The width and length of the footing are only increased or optimized to prevent stability check failures if the "Optimize Footings" check box on the **Concrete** tab of the **Model Settings** has been checked.
- For Safety Factors greater than 1000, the footing is stable and the program will report 'NA' as the result.

Sliding Check

RISAFoundation provides sliding checks for all of the service combinations.

The applied forces and resisting forces, due to friction and passive soil resistance, are calculated and displayed. The final result is a sliding ratio of resistance vs. applied force that indicates a footing susceptible to sliding when the ratio is less than 1.0.

Concrete Bearing Check

The concrete bearing and demand/capacity ratios for each design combination are presented.

IMPORTANT: These concrete bearing checks are only based on the vertical applied loads, i.e. bending effects that increase or decrease bearing area and pressure are not considered.

The **Concrete Bearing Check** in RISAFoundation checks whether the footing concrete is adequate to resist the vertical loads imposed on the pedestal. These calculations are per ACI 318-14 Section 22.8 (ACI 318-11 Section 10.14). The results are presented as the demand vs. capacity ratio in the Detail Report.

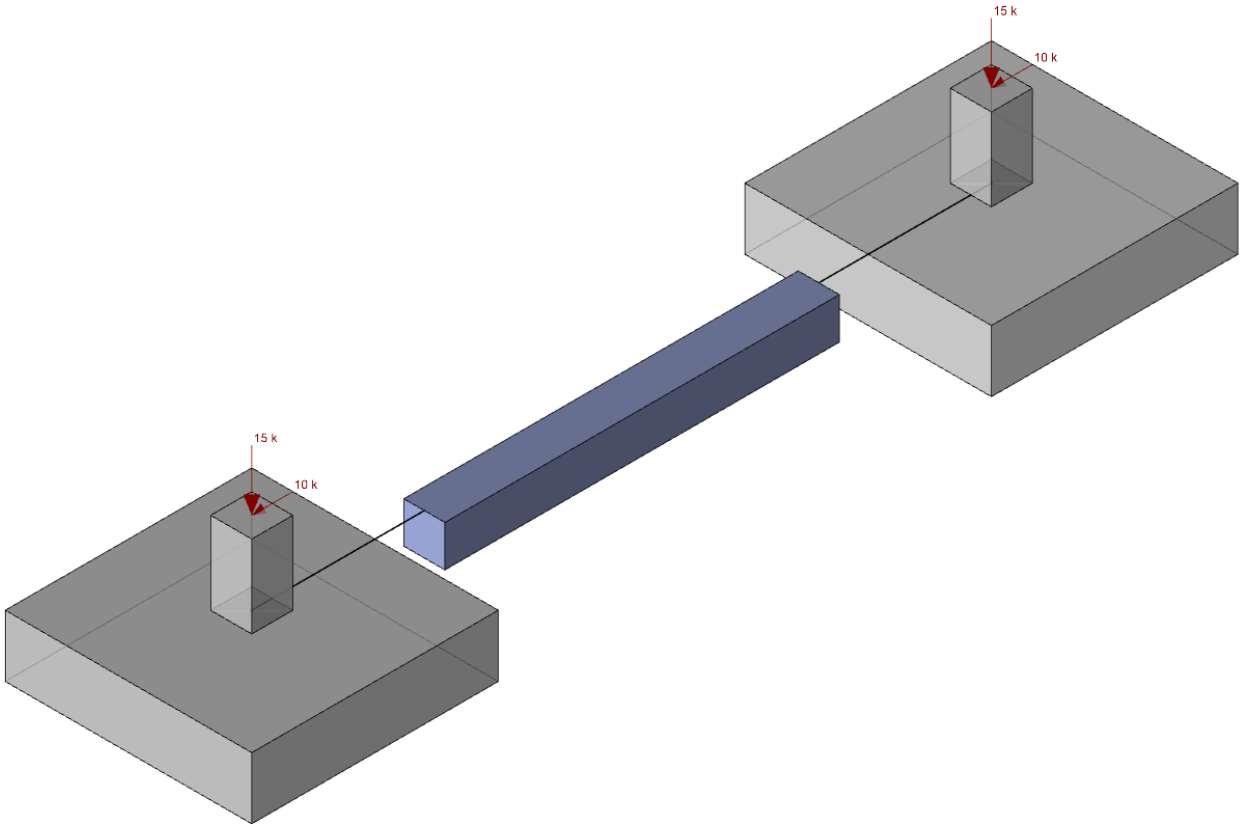
Footings - Combined

One of the strengths of the RISAFoundation program is the ability to easily create and design combined footings. There are many different types of combined foundations and this discussion is only meant to address the basic cases. However, the principals discussed here are valid for more complex arrangements as well.

Strap Footings

One of the most common types of spread footings is the simple case of multiple spread footings that are connected together with a grade beam. The beam allows the two footings to share the load

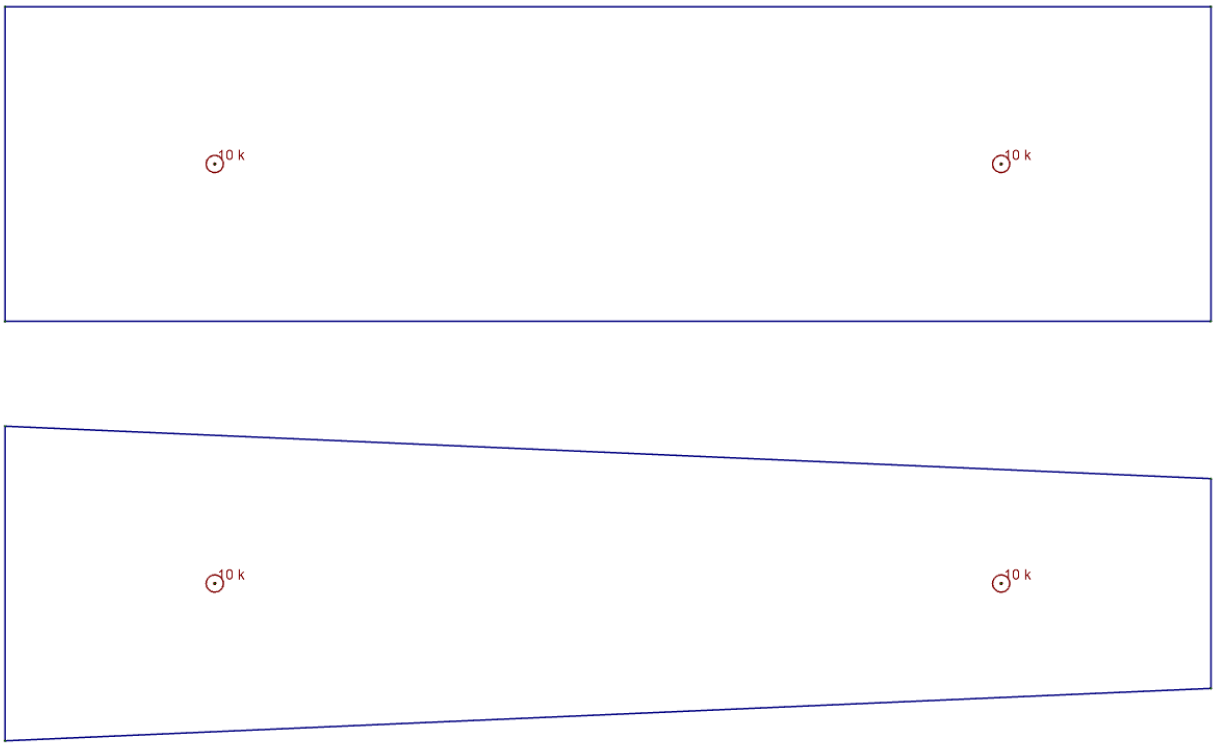
between them based on their relative stiffness and the stiffness of the connecting beam. A simple example of a **Strap Footing** is shown below:



It's important to realize that each footing will still be designed as a single spread footing. It's just that the presence of the grade beam allows a redistribution of the forces between the footings that can result in a more economical foundation. For more information, refer to the sections on individual footing design and Footing Design Results. For more information about how the program is using the soil parameters to assign vertical and rotational springs to the individual footing supports, see Soils.

Using Slab Foundations to Model Combined Foundations

Rather than using a grade beam or strap beam to connect two individual footings together, it may be preferable to use less forming and pour a small mat foundation. While many people would refer to these types of foundations as combined footings, the program will treat them as small mat foundations.



Because they are treated as mat foundations, you will have to specify **Design Strips** or **Design Cuts** to get your reinforcement design. For more information, see Slabs - Design Strips and Slabs - Design Cuts, respectfully.

Footing Stability and Overturning Calculations

Calculation of Overturning Moment (OTM) Stability Ratio

One of the important results from any footing analysis is a ratio of the stabilizing moments to the de-stabilizing moments. This is referred to as the Stability Ratio or the Safety Factor for overturning. RISAFoundation calculates this value differently than the way many engineers would traditionally approach a hand calculation. The RISA method is a bit more complicated, but will provide a more realistic representation of the true safety factor whenever uplift loads are present. A comparison of the two methods is provided below.

The Overturning Safety Factor (OSF) is the sum of resisting moments divided by the sum of overturning moments. RISAFoundation may provide default input for safety factors, but they need to be reviewed and updated (if needed) by the project EOR per code requirements, foundation types and project needs. Overturning safety factor calculations are based on the service load combinations only and are calculated in both the X and Z directions.

Traditional / Simplistic OTM Calculation

Most hand calculations are done on the assumption that all vertical loads are stabilizing loads that are only capable of producing a stabilizing moment, whereas all lateral shear and moments will act to de-stabilize the structure. As an example, consider the case where you have the following loads applied to a 4x4 footing with a pedestal that extends 2 ft above the bottom of the footing.

Force Effect	Dead Load	Wind Load
Axial	25 kips	+/- 10 kips
Shear	0	5 kips

In the traditional method, the axial forces are combined together to form a total vertical load. Since the TOTAL EFFECT of the axial loads is considered a stabilizing effect, the resisting moment is considered to be P*L/2. Since the only de-stabilizing effect is assumed to be the shear force, the de-stabilizing moment would be calculated as V*H. These calculations are summarized in the table below:

Wind Effect	Stabilizing Moment	DeStabilizing Moment	OTM Safety Factor
Downward	70 k-ft	10 k-ft	7.0
Uplift	30 k-ft	10 k-ft	3.0

True Safety Factor Method (used by RISAFoundation)

The True Safety Factor method used by RISAFoundation involves taking a look at EVERY component of every load and deciding whether it has a stabilizing or a de-stabilizing effect. Overturning moments are those applied moments, shears and uplift forces that seek to cause the footing to become unstable and turn over. Resisting moments are those moments that resist overturning and seek to stabilize the footing. These overturning checks are performed for overturning about each edge of the footing. This is important for footings with eccentric pedestals, where the vertical loads will have different stabilizing effects for each face.

By summing up all the resisting moments and comparing them to the sum of all the overturning moments we get a better sense of the true tendency of the structure to overturn. Taking the example listed above, it is clear that when the wind forces act in compression, the two method will produce identical results. That means that if the Wind shear is increased by a factor of 7.0, then the footing would be in net overturning.

However, the stabilizing moment in the uplift case would be viewed as the FULL dead load * L/2 rather than the reduced axial load used in the traditional method. Similarly, the de-stabilizing load would be viewed as the sum of the moments about the edge of the footing. This includes the de-stabilizing effect of the wind uplift and is equal to P_wind* L/2 + V*H.

The OTM Safety Factor calculations for the True Safety Factor Method are summarized in the table below:

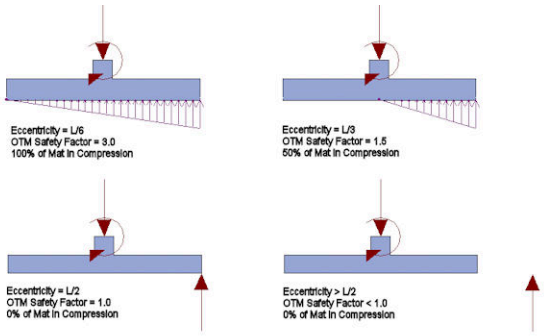
Wind Effect	Stabilizing Moment	DeStabilizing Moment	OTM Safety Factor
Downward	70 k-ft	10 k-ft	7.0
Uplift	50 k-ft	30 k-ft	1.67

Comparing the Two Methods

Which is a more accurate representation of the safety factor stability ratio for the uplift case? The best way to judge this is to multiply the wind load by a factor equal to the Safety Factor and see if the footing still works. If the wind effects (shear and uplift) are multiplied by a factor of 3.0, as given by the traditional method, then the footing will be unstable due to net uplift and net overturning. The Safety Factor would be calculated at a negative value, which has little physical meaning. If the wind effects (shear and uplift) are multiplied by a factor of 1.67, as given by the True Safety Factor method, then the stabilizing moment will equal the de-stabilizing moment and the Safety Factor becomes 1.0, as predicted.

Calculation of Moment and Shear Demand for Unstable Footings

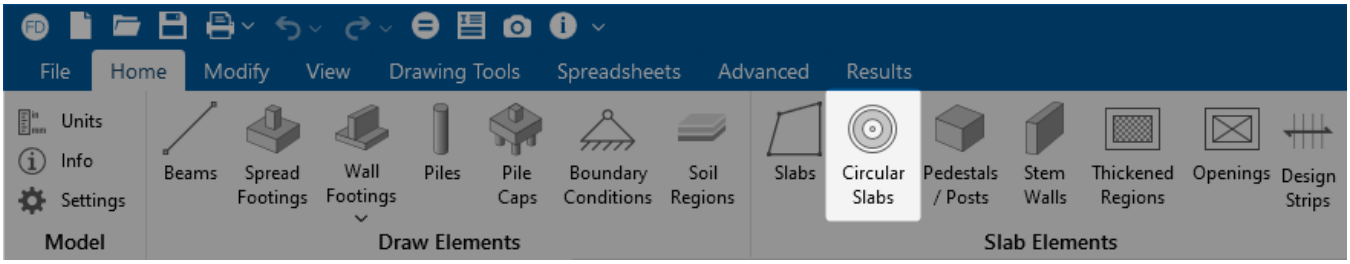
Older versions of the program could not handle situations where the Ultimate Level load combinations resulted in net overturning of the footing. This is because the previous versions always relied on the soil bearing pressure calculations to derive the moment and shear demand in the footing. For net overturning cases, the current version of the program assumes that the design shear and moment can be based on the total vertical load and the net eccentricity of that load.



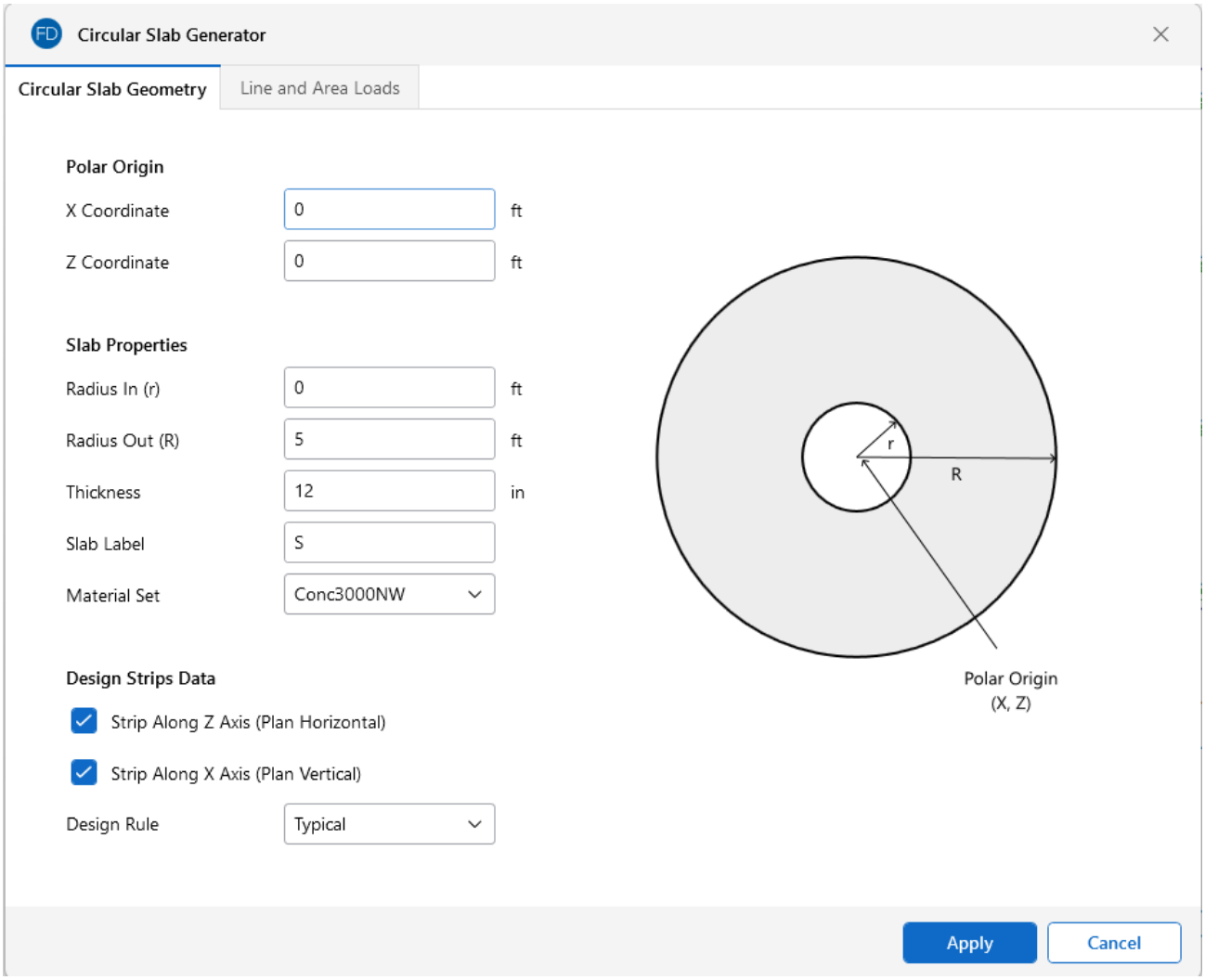
In cases where the resultant is off the footing, the design shear force in the footing will be constant and equal to the net resultant minus any effects of self weight or over burden. Also, the basis for the design moment at the face of the pedestal will be the load resultant times the distance from the resultant to the face of the pedestal. Any negative moment effect due to the self weight of the footing slab and overburden would be subtracted out when determining the final design moment.

Generation

Circular Slab Generator



The **Circular Slab Generator** allows users to quickly lay out a circular mat slab for design. There are also circular line and area load generation capabilities that allow you to automatically create typical loading for round structures; for example, tanks.



The polar origin defines the center coordinates of the slab. The inner radius, outer radius, thickness and material for the slab are also defined here.

The load generation information allows you to input the load category, magnitude and geometry of the loading.

Note: The program creates interior nodes in the slab for application of line and area loads. These nodes can be manipulated for line loads in the Line Loads Spreadsheet. The generated Area Loads will need to be deleted and redrawn to make changes.

There is also a design strip generator that allows you to define a strip over the entire slab in either of the global axes and also choose the design rule that defines the reinforcement for these strips.

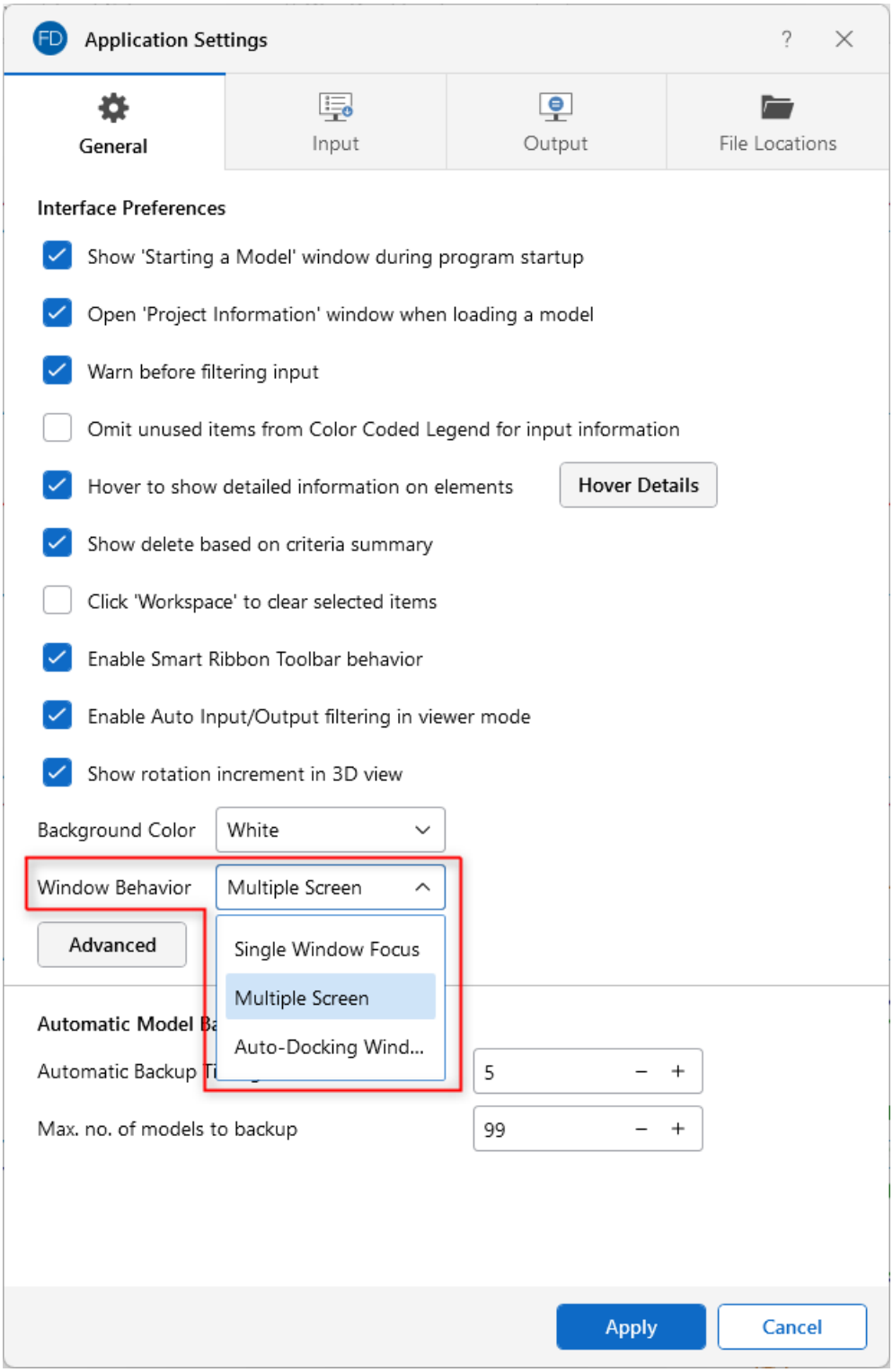
For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Circular Slab Generator**.

Graphic Display

RISAFoundation's robust graphics make building and understanding your model much easier. You can have numerous independent views open and active at the same time. Fast redraw times, responsive panning, true to scale rendering, and graphic editing tools all help you build and understand your model. You can draw the model and specify loads and supports graphically and these items can be modified graphically as well. Verification of your model is made simple with true-to-scale rendering and color coded code checks. Results such as member force diagrams, deflected shapes and animations can also be viewed for solved models.

Window Behavior

Window Behavior can be specified in the **Application Settings** window, under the **Interface Preferences** section.



There are three window behaviors to choose from:

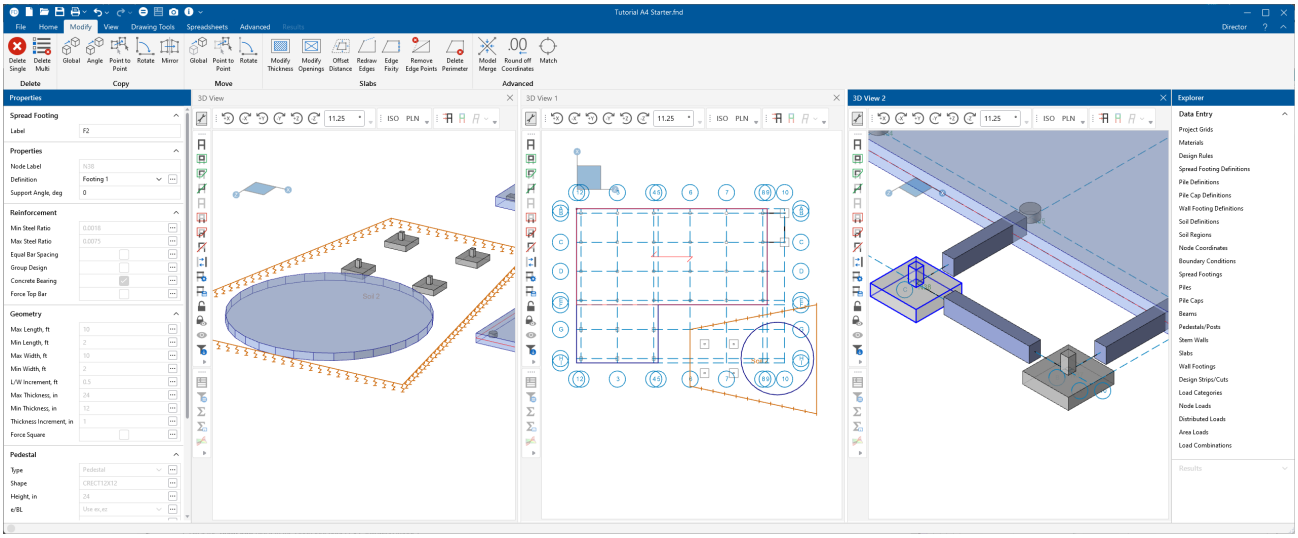
- **Single Window Focus**: With this option, when you click within the 3D view, it automatically closes all open/visible spreadsheets and detail reports.
- **Auto-Docking Windows**: With this option, when you click within the 3D view, it automatically docks all open/visible spreadsheets and detail reports as new tabs.
- **Multiple Screen**: This option lets you use two monitors. It is useful when you want to keep detail reports or spreadsheets visible on a separate monitor while navigating through the 3D view. If a detail report or spreadsheet is open and is visible on the same monitor as the active 3D view, clicking within the 3D view sends the spreadsheet behind the program window.

For additional information on working with viewpoint windows, see the Windows Behavior section under Main UI.

For information about working with multiple views, see the Multiple Windows section.

Multiple Windows

You can have multiple views of a model open at the same time, each with it's own characteristics. For example, you can simultaneously view and work with slabs in one window and footings in another. Multiple spreadsheets can be opened as well.

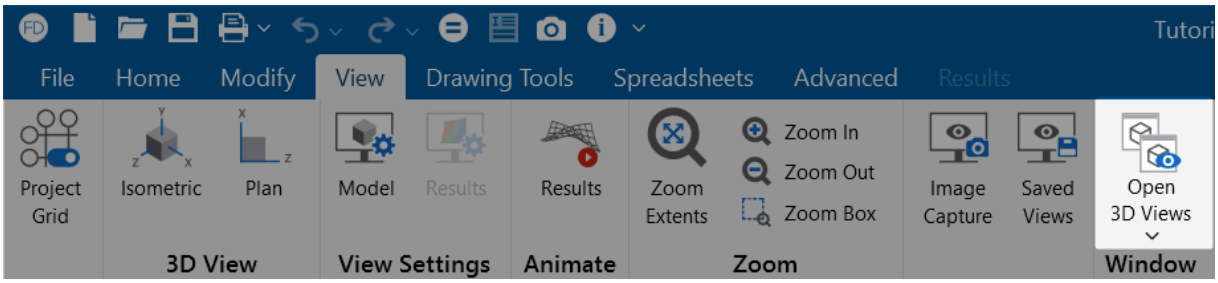


Open a New 3D View

Whether you want to open a new 3D View panel or open an additional 3D View panel, the steps are the same.

To open a new 3D View:

1. Go to the **View** ribbon.
2. Click on **Open 3D Views**



Alternatively, you can right-click in the existing 3D View and select **Add New 3D View** from the menu that appears.

A new 3D View window opens with a duplicate of the model, if you already have a model open.

3. (Optional) To adjust the view of the new model window:

   With the model view active, use the buttons (shown below) at the top of the 3D View to adjust the view of the model.



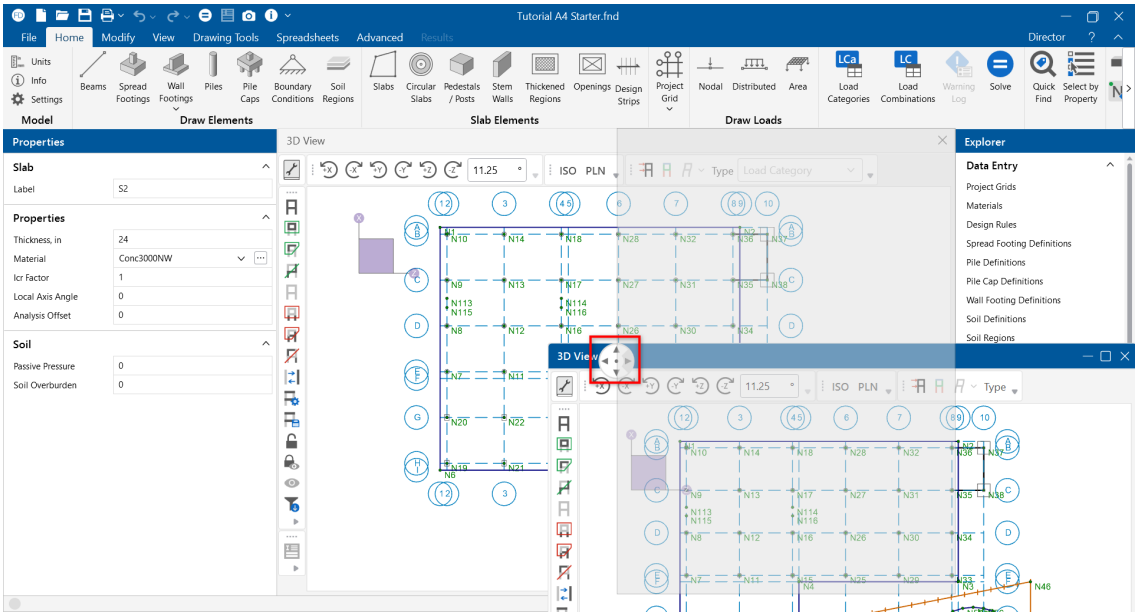
4. (Optional) To snap the 3D View panel within the model editor panel:

   a. Drag the blue title bar of any window (3D View or spreadsheet) until you see the  arrow controls appear.

   b. While still holding down the mouse button, drag over the arrow control that corresponds with where you want the view panel to be snapped.

      A gray areas appears, showing where the panel will be placed when you release the

mouse button.

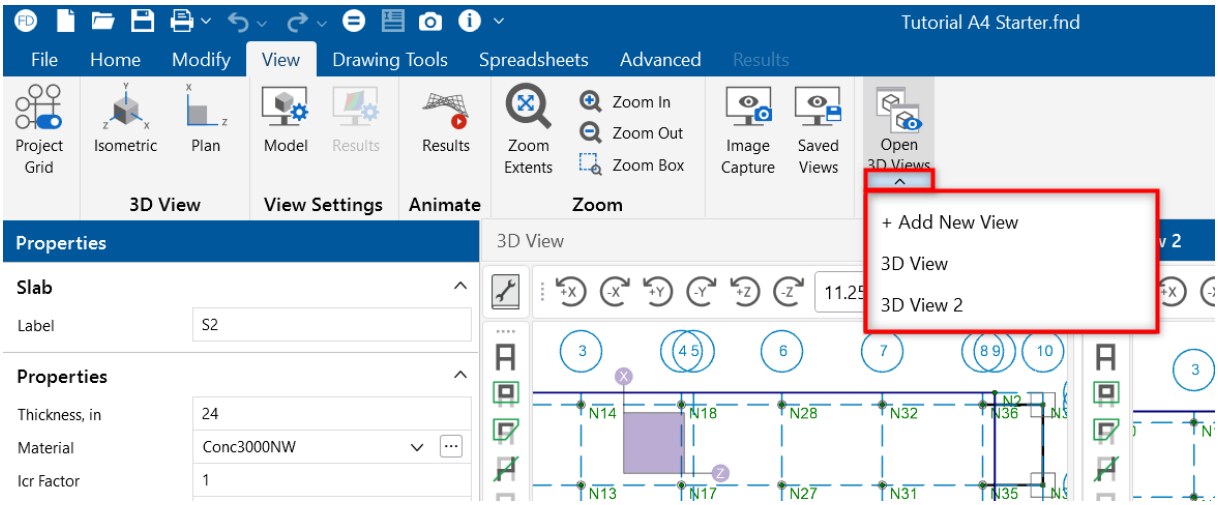


Switch Between Open 3D Views

When you have two or more 3D Views open in the view panel, you can easily switch between those views by choosing the view you want to work with.

To switch to another view:

1. Go to the **View** ribbon.
2. Click the **Open 3D Views** down arrow.



A menu appears that lists each of the views that are currently open.

3. Click on the view you want to work with.

   The view you chose is brought to the front of the view panel and becomes the active view to work with.

Adjust the Viewpoint (3D View)

Adjusting the Viewpoint refers to the ways you can modify a 3D View. An explanation of the features for adjusting the view and how to use each one is described below.

Zoom Features

There a two methods for zooming: you can simply use your mouse wheel, or use the Zoom tools on the **View** ribbon.

Use Your Mouse to Zoom

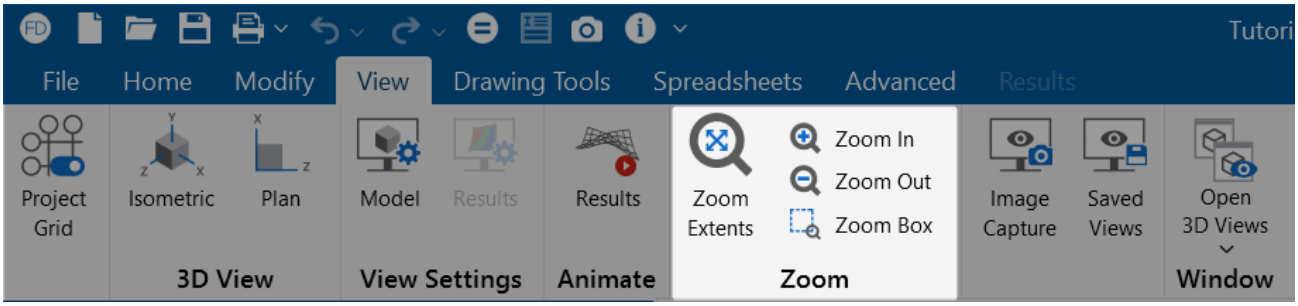
When you want to zoom in on the model to see more detail, or zoom out to view the whole model at once, it is often easiest to use your mouse wheel.

To zoom in or out using your mouse wheel:

- Rotate the wheel forward to zoom in.
- Rotate the wheel backward to zoom out.

Use the Zoom Tools

The Zoom tools found on the **View** ribbon can also be used for zooming in or out. First, choose the zoom state (In or Out) using the appropriate icon in the ribbon. Then, use your mouse to incrementally zoom. There are two additional Zoom tools: Zoom Extent and Zoom Box. These are explained below.



Zoom Extent Tool

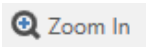
This tool resets the view to show the full extents of the model. It is especially useful for resetting the model view after you've zoomed into a specific area.

To zoom to full model view:

- Click the **Zoom Extents** icon.

Zoom In Tool

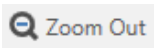
This tool sets the zoom state to “in” so that each time you click your mouse button, the model view is enlarged.

1. Click the 

Zoom In icon on the View ribbon.
2. Click anywhere in the model (3D View Window) to enlarge the model view.
3. Repeat **Step 2** until the desired view of the model is reached.

Zoom Out Tool

This tool sets the zoom state to “out” so that each time you click your mouse button, the model view is reduced.

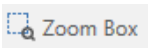
1. Click the 

Zoom Out icon on the View ribbon.
2. Click anywhere in the model (3D View Window) to reduce the model view.
3. Repeat **Step 2** until the desired view of the model is reached.

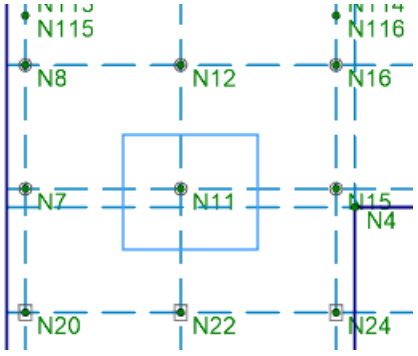
Zoom Box Tool

This tool lets you select a specific area to zoom into by drawing a box around the area.

To zoom into a specific area of the model:

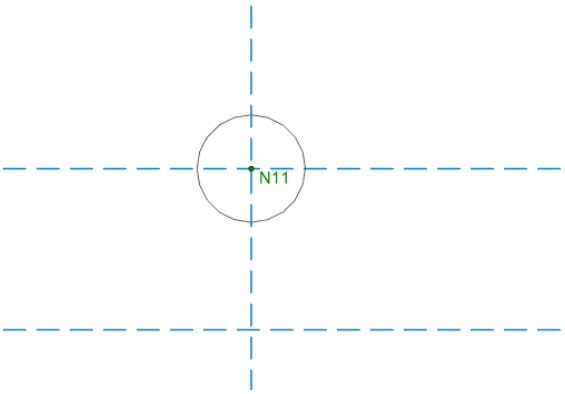
1. Click the 

Zoom Box icon on the View ribbon.
2. In the model view, click on one point and hold your mouse button down as you drag down to the opposite corner, creating a box around the area you want to zoom into.



3. Let go of the mouse button.

The view automatically zooms in to the boxed area.



Pan the View

The **Pan** feature lets you drag the model view to the edges of the 3D View window. You can accomplish this by using one of two methods: your mouse scroll wheel, or the right-click menu.

Pan Using Your Mouse Wheel

To pan the model view using your mouse wheel:

1. Click and hold the mouse wheel button to trigger the dynamic pan feature.

   You'll see a panning icon appear, indicating that you are in panning mode and can begin dragging. You can drag the model view in any direction.
2. While still holding down the mouse wheel, drag the model view in the direction you want to move it.
3. Release the mouse wheel when the model view is placed where you want it.
4. If you do not have a mouse wheel you can use the right-click menu to pan the view.

Pan Using the Right-Click Menu

To pan the model view using the right-click menu:

1. Right-click anywhere in the model view window.
2. Choose **Pan** from the menu that appears.

   You'll see the panning icon appear, indicating that you are in panning mode and can begin dragging.
3. Click and hold your left mouse button and drag the model view in the direction you want to move it.

4. Release the mouse button when the model view is placed where you want it.
5. To turn off panning mode, click the right-mouse button.

Snap to a View

To snap to an isometric or planar view of the model:

1. Click the **Snap** button at the top of the 3D View window that corresponds with the view you want.



Note that you can add your own custom snap view to the toolbar. When doing so, the current view is saved with the name you provide and added to the end of the toolbar.

Alternatively, you can right-click in the existing 3D View and select **Add New 3D View**.

(Optional) To add a custom snap view:

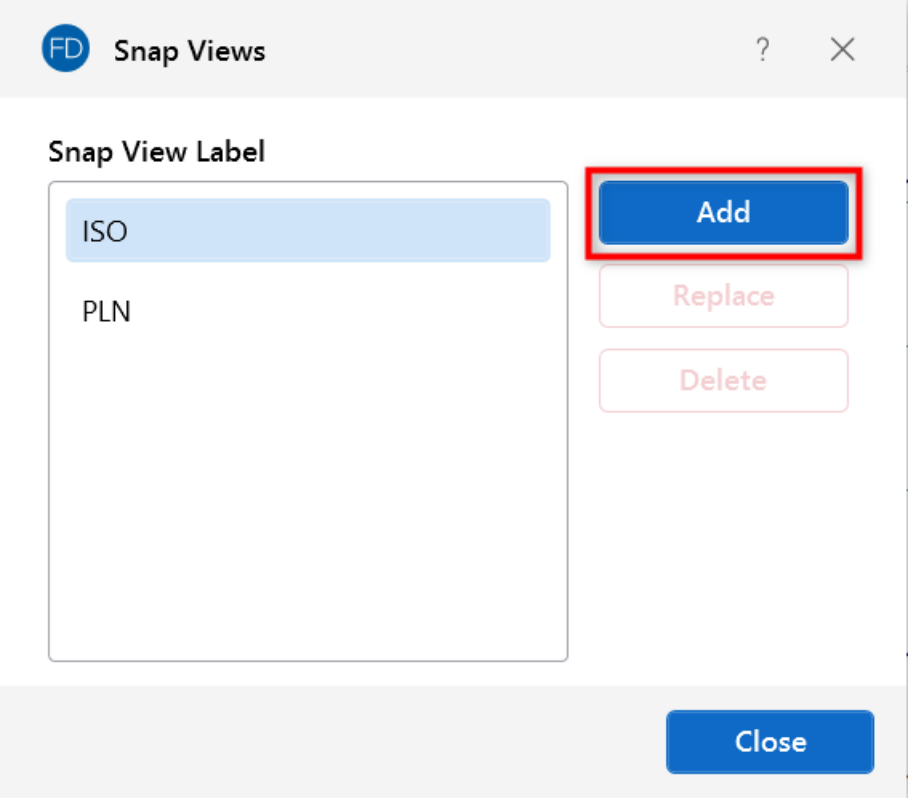
1. Click the down arrow on the Snap toolbar.



2. Click the ellipsis button that appears.

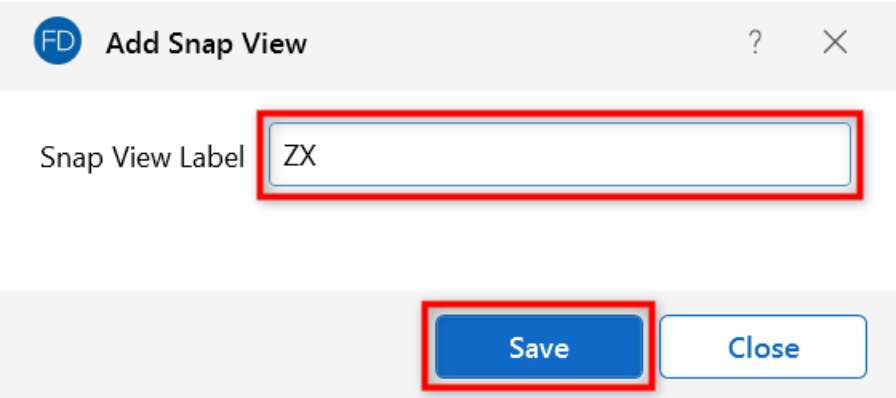
The Snap Views window opens.

3. Click **Add**.



An Add Snap View window opens.

4. In the **Snap View Label** box, type a name for the new snap view.
5. Click **Save**.



The new snap view is saved to the end of the Snap toolbar, as shown below, and you are returned to the Snap Views window.



6. Click **Close** to close the Snap Views window.

Rotate a View

There are two methods for rotating a view: click, or drag. An additional feature lets you quickly reset the model's rotation to its original, default position.

As you rotate a view, the global X-Z axis orientation is shown by the orientation of the Navigation Icon.

Click to Rotate

To use the click method to rotate a view:

1. Click the rotate icon that corresponds to the rotation direction you want to use.

   You can also change the rotation angle by entering a different degree magnitude. This affects the rotation of the model as you click to rotate.

2. (Optional) To change the rotation angle:
   a. Type the degree magnitude in the box, as shown in the following image.

   b. Click the rotate icon that corresponds to the rotation direction you want the model rotated.

      The model is set rotated by the degree you've entered and the orientation of the Navigation Icon changes to reflect the angle and degree change.

Drag to Rotate

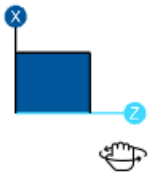
You can use one of the following drag methods to rotate the model view:

- While holding down the **SHIFT** key, press the mouse wheel to dynamically rotate the structure. For more information, see Dynamic View Controls.
- Click and drag the **Navigation Icon** to change the rotation of the model.

Default Rotation **Click and Drag**



- Use a **global axis** to rotate the view about one of the global axes by hovering over the axis until it turns blue. Then click and drag to change the rotation of the model.



Note: If you hold down the **SHIFT** key while rotating the model using the individual axis, the model only rotates at 45 degree increments around that axis.

Reset Rotation to Default

Double-click in the **Navigation icon** to reset the rotation of the model view to the default isometric view position.

3D View Display Options

Each 3D View has several display options located at the top of the view; Display Loads, Display Results and Display Deflection.



Display Loads

The **Display Loads** icon lets you display loads in the model view.

To show loads in the model view panel:

1. Click the **Display Loads** icon to toggle the display of loads.
2. Click the **Type** down arrow and choose whether you want to display loads based on Load Category or Load Combination.
3. Click the ⌄ down arrow in the second box and choose which load category or combination

is shown in the view. This box not active when you choose Envelope from the Type drop-down menu.

See Model View Settings for more control over the display of loads.

Display Results

The **Display Results** icon lets you display color-coded results in your model view.

To show results:

1. Click the **Display Results** icon.
2. Click the **Type** down arrow and choose whether you want to display color coded results based on Load Combination or Envelope.
3. When Load Combination is selected, Click the ⌄ down arrow in the second box to choose which Load Combination results are to be colored coded in the model view. This box not active when you choose Envelope from the Type drop-down menu.

   A color legend appears and color-coded results based on your choices appear in the model view.

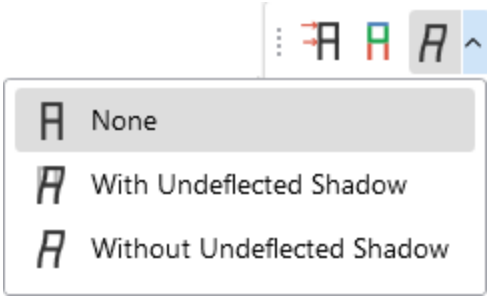
For more information on how to control the display of results, see Results View Settings.

Display Deflection

The **Display Deflection** icon lets you display the deflected shape in your model view; with or without an undeflected shadow.

To show the deflected shape:

1. Click the **Display Deflection** icon.
2. (Optional) Click the down arrow to choose whether to view the deflected shape with or without an undeflected shadow (as shown in the following image).



   The model's deflected shape is displayed.

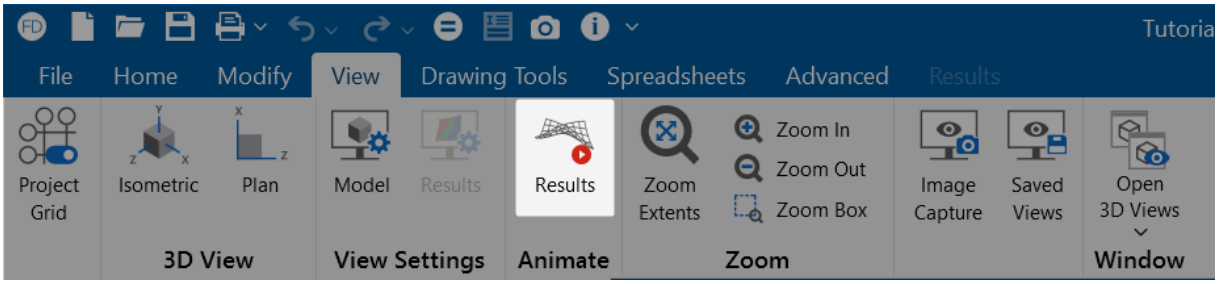
Note: Deflection results can only be shown for Load Combination load types.

Animate Results

With the Animate Results tool, you can animate a model's deflected shape.

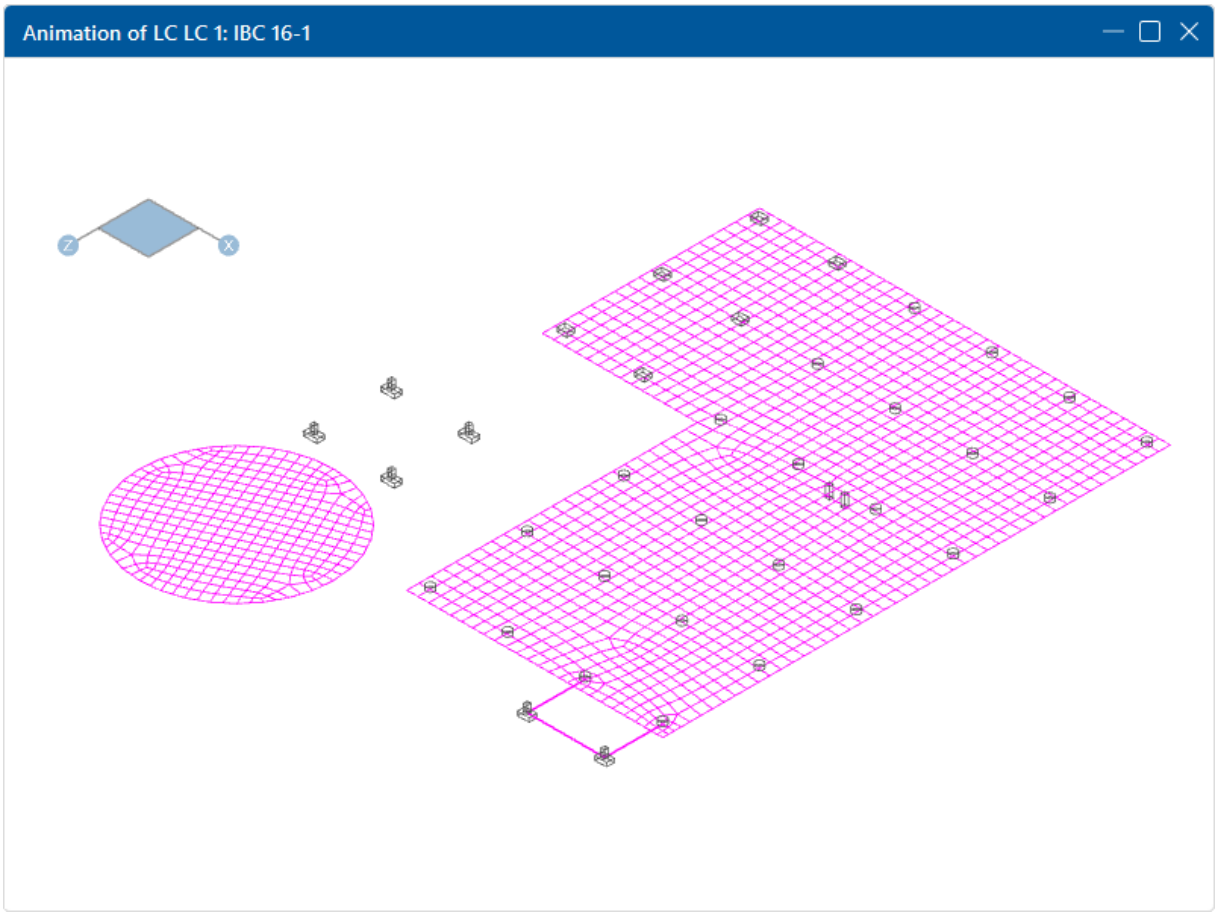
To animate results:

1. Open the **View** ribbon.

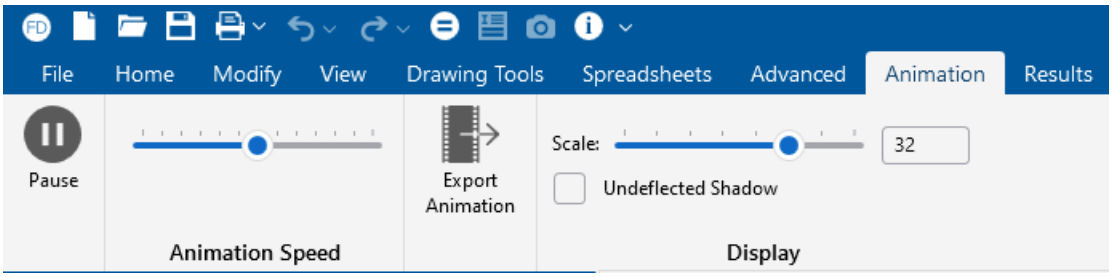


2. Click the **Results** icon in the Animate section.

   A new model view is created with the animation.



When this model view is active, an **Animation** ribbon opens providing additional animation settings.



3. (Optional) Click the **Pause** icon to pause the animation.
4. (Optional) Use the **Animation Speed** slider to control the speed of the animation.

   Dragging the marker to the right increases the speed, while dragging it to the left decreases it.
5. (Optional) Click the **Export Animation** icon to export the animation as a .wmv or .mp4 file.
6. (Optional) Use the **Scale** slider to control the scale of the animated deflection.
7. (Optional) Check or Uncheck the **Undeflected Shadow** check box to toggle it on or off.

Create a Custom Snapshot of a Model

RISAFoundation includes an **Image Capture** tool on the **View** ribbon that allows you to create a custom snapshot of a model and save it.

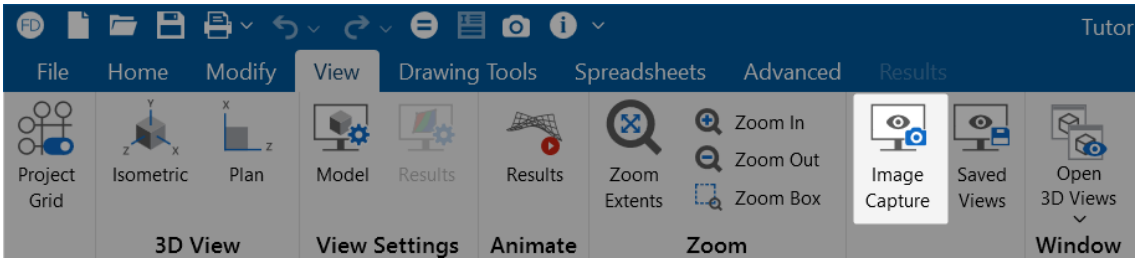
To create a custom snapshot:

1. Click the **Snapshot** icon on the Quick Access Toolbar.

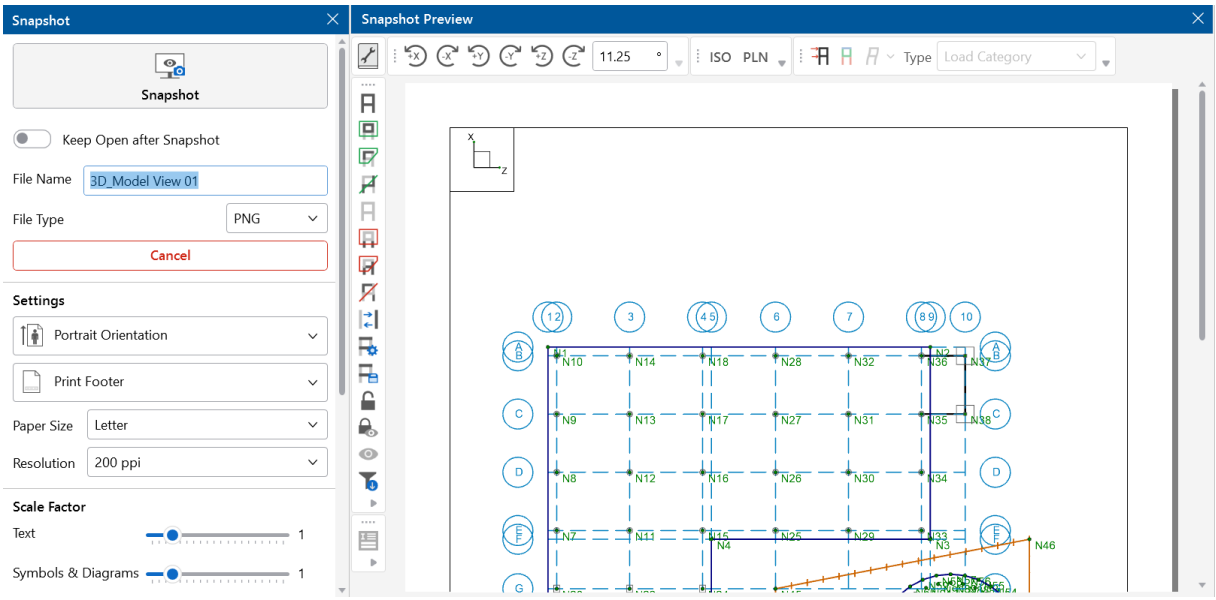
   Alternatively, you can:

   a. Open the **View** ribbon.

   b. Click the **Image Capture** icon.



A Snapshot Preview opens on the right, with a Snapshot panel to its left.



2. (Optional) For information on printing the snapshot, please see Printing.
3. To save the snapshot:
   a. Type a name for the snapshot you are saving in the **File Name** box in the Snapshot panel.
   b. Click the **File Type** drop-down arrow and choose the image format to save the snapshot in.
   c. (Optional) To keep the Snapshot and Snapshot Preview panels open after capturing the snapshot, click the **Keep Open after Snapshot** toggle (it turns blue when selected).
4. Click the **Snapshot** block in the **Snapshot** panel.



A Snapshot Created message briefly appears.

Note:
- If the model has yet to be solved, the image is created in the **Model Data Files** location from **File > Application Settings > File Locations**.
- The relative path of the image file is saved with the model. If the model file is moved, remember to move the associated **Images** folder as well so that all saved views and snapshots will still be available.

View Only Part of a Model

You may want to view only part of your model, especially if it is a big one. You may use the selection tools to give you the view that you want. You can graphically unselect parts of the model that you don't wish to see, or you can use a range of coordinates or other criteria to specify what to view.

- To select a part of the model to zoom in on, see **Zoom Box** under Adjust the Viewpoint (3D View).
- To select a part of the model to work more closely on, see **Selection Modes** under Graphic Selection.

Save / Retrieve a Model

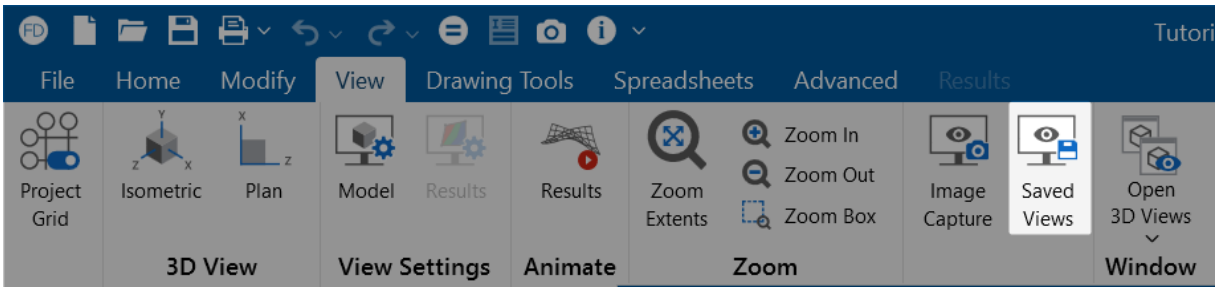
You can save and recall views for a model. Saved views are model dependent, so any views you save stay with the model. A saved view includes information, such as the current view angle, zoom state, pan location, plot option settings, etc. Saved views do NOT include the selection state for the model. You can save selection states separately (see Save Selections).

Save a Model View

If you have a view that you like to work with in the future or have created a view that took some time to set up, you can save it with this feature. All of the Model View Settings are saved with the model for later recall.

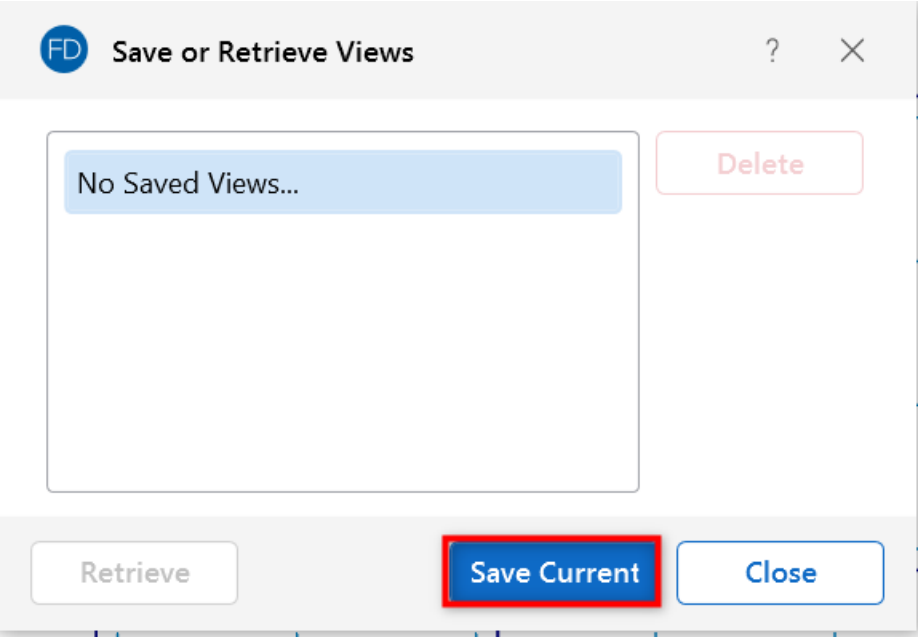
To save a model view:

1. Open the **View** ribbon.
2. Click the **Saved Views** icon.

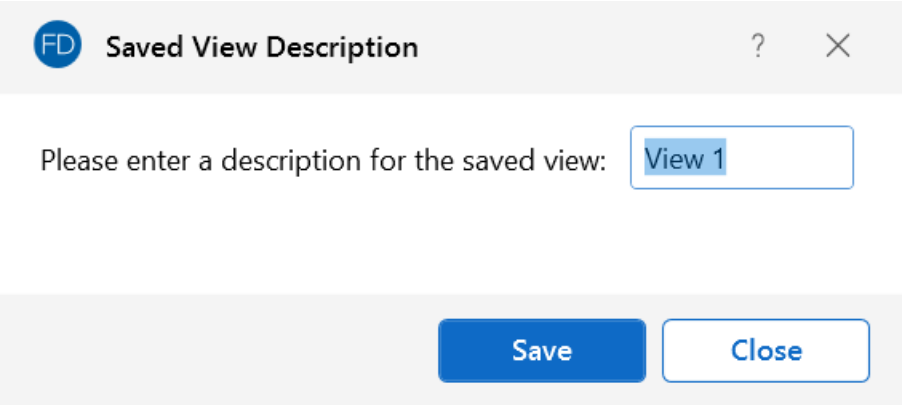


A Save or Retrieve Views window opens.

3. Click the **Save Current** button.

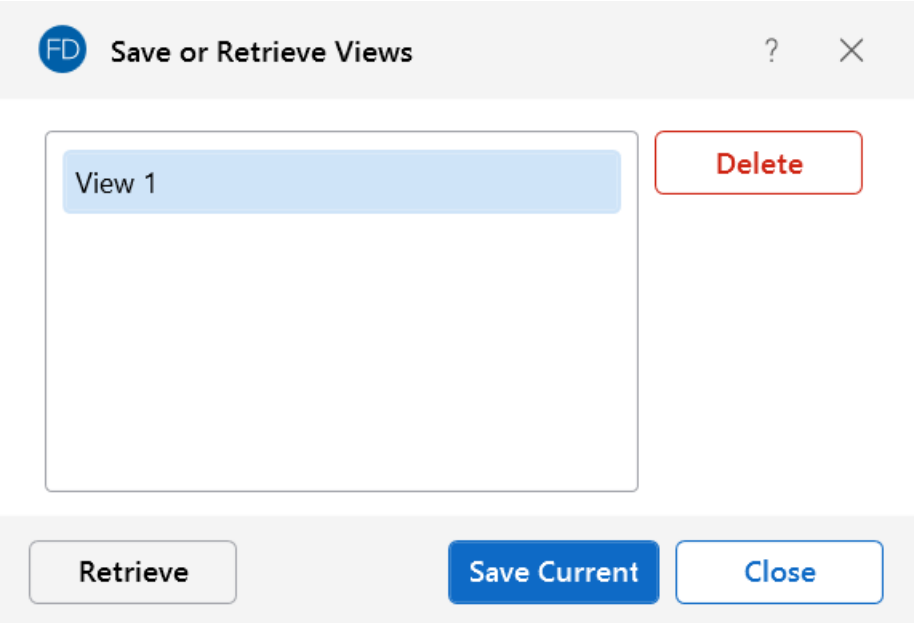


A Saved View Description window opens.



4. Enter a View description in the box.
5. Click the **Save** button.

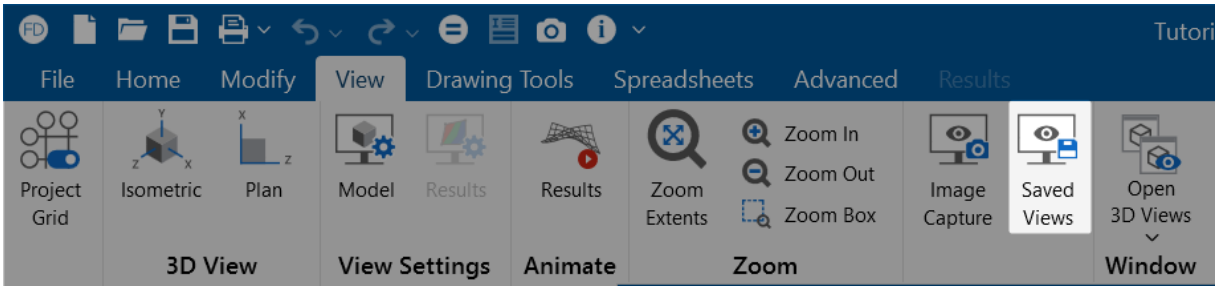
   The view is saved and the Saved View Description window is closed. You are returned to the Save or Retrieve Views window where you can see the view you just saved, along with all other saved views.



Retrieve a Saved View

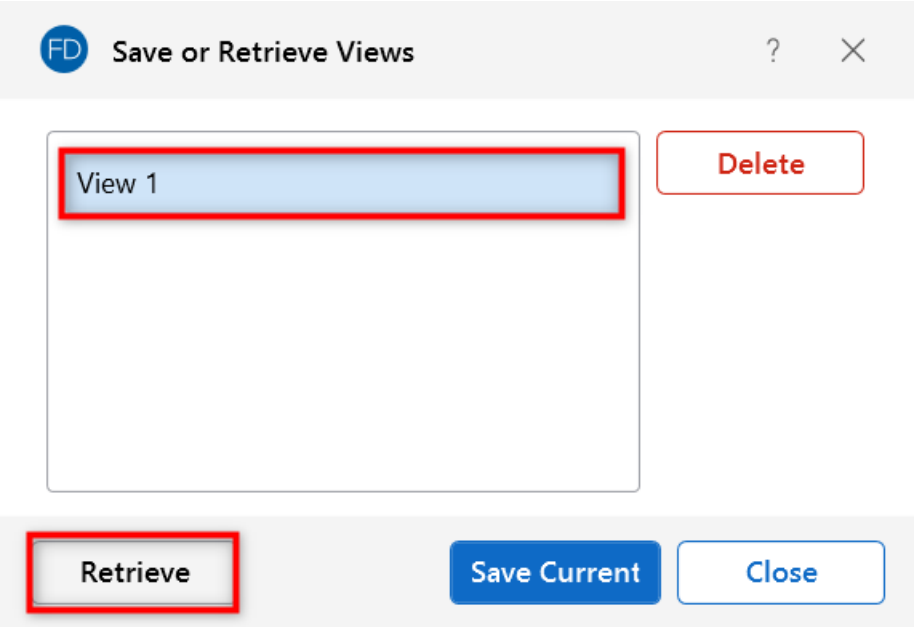
To retrieve a saved model view:

1. Open the **View** ribbon.
2. Click the **Saved Views** icon.



The Save or Retrieve Views window opens.

3. From the list, choose the view you wish to recall.
4. Click the **Retrieve** button.



The view you chose is brought forward and becomes the active view to work with.

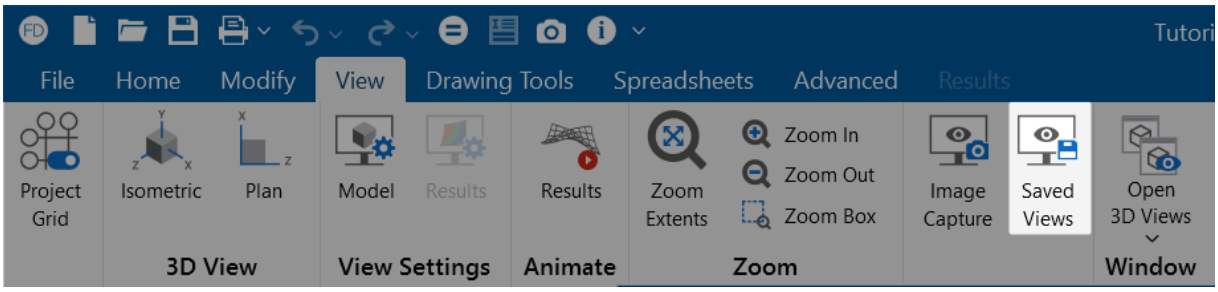
For information on deleting a view, see Delete a Model View.

Delete a Model View

If you no longer need a view that you have saved, you can simply delete it.

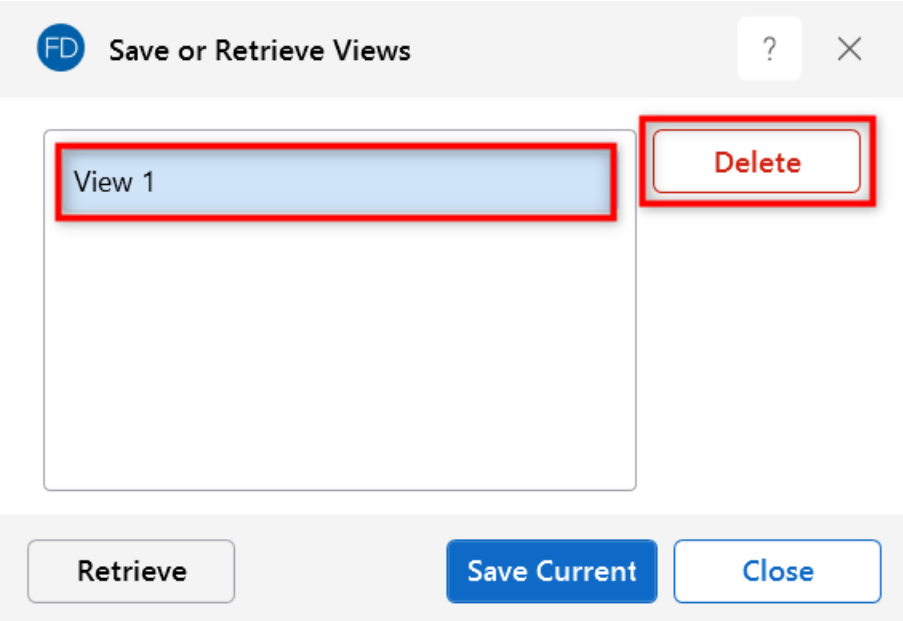
To delete a saved model view:

1. Open the **View** ribbon.
2. Click the **Saved Views** icon.



The Save or Retrieve Views window opens.

3. From the list, choose the view you wish to delete.
4. Click the **Delete** button.



You are asked to confirm the deletion.

5. Click **Yes**.

   The view you chose is removed from the saved list.

For information on saving or retrieving a model view, see Save / Retrieve a Model.

Graphic Editing

You may draw, edit and load your model directly in the model views. You can draw beams and slabs between existing nodes, or draw to grid intersections and have the nodes created automatically. You may graphically select points to be supported and beams or slabs to be loaded. All, or selected parts, of the model may be moved, copied and/or modified, allowing you to quickly model and edit your structure. To learn how to make selections, see Graphic Selection.

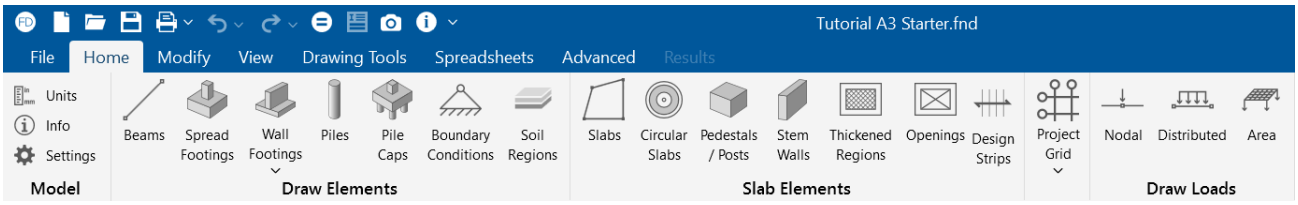
Draw / Modify Features

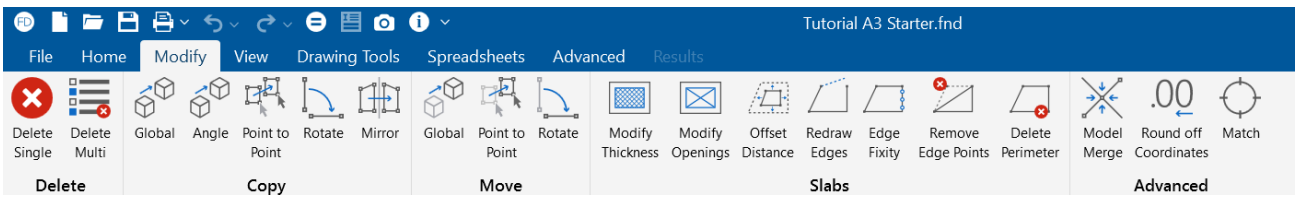
There are several graphic editing features that make the creation and modification of models quite easy. Use the **Draw Elements** tools on the **Home** ribbon and the tools on the **Modify** ribbon to utilize these features in the model view. To create new beams, slabs, footings, etc., you can use a drawing grid or draw "dot-to-dot" between existing nodes. Once you have created these items, you may use other graphic features to load the model and set boundary supports.

- See Beams for more information on drawing and modifying **Beams**.
- See Slabs for information on drawing and modifying **Slabs**.
- See Boundary Conditions for information on creating and modifying **Boundary Condition Supports**.
- See Loads - Nodal Load for information on drawing **Nodal Loads**.
- See Loads - Distributed Loads for information on drawing **Distributed Loads**.
- See Loads - Area Loads for information on drawing **Area Loads**.

All model data is automatically recorded in spreadsheets and is always in tune with the model view. As you edit a model graphically, the spreadsheets are automatically updated and as you make changes in the spreadsheets the model views reflect these changes immediately.

All of the graphical modeling tools may be found on the **Home** and **Modify** ribbons, as shown in the following images.





Where to Start

The Project Grid and Drawing Grid are often useful when you are starting a new model from scratch or adding a new section to a model. They allow you to set up grid lines, which may then be used to define the beams, slabs and footings. The differences between the two grids are simple. The Project Grid is part of the model and is limited to the model 'plan'. The Drawing Grid is independent of the model, so you may change the grid whenever it is convenient and place it anywhere without affecting the model. The Drawing Grid may also be placed in any global plane so you may draw in 'plan' or in 'elevation'.

There are also times when it is simpler to define points in the spreadsheet and then draw the members or slabs between them. This might be the case if you are only working with a few points, or if the structure is irregular and does not lend itself to a grid.

Apply Options

The **Properties Panel** makes it very easy to apply modifications to a single element, and multiple elements. For example, changing the material of a beam from 3000 psi concrete to 4000 psi can be accomplished through the **Beams Spreadsheet**. However, if you had to apply this change to 100 beams, you would not want to do that for each of them individually. Instead, you can graphically select all of the beams and then apply the changes all at once in the **Properties Panel**.

Use the tools on the **Selection Toolbar** to choose the items you want to modify and then make the changes in the Properties panel to apply the modifications to all selected items at once. For more information on the selection tools, see Graphic Selection.

Note:

- The selection and viewing tools override the graphic editing modes, so that as you are editing the model, you can alter the selected state. The **Properties Panel** will update based on the selection you make.
- You may also click any single node, beam, slab, etc. to view and edit it's properties in the **Properties Panel**.
- To correct any modeling errors, it is a good idea to run **Model Merge** before performing a solution. Becoming familiar with this feature will also allow you to take shortcuts while modeling. For more information, see Model Merge.

Undo / Redo Operations

Undo Operations

RISAFoundation provides an unlimited 'Undo' capability so that you can easily correct mistakes, or just back up to try different possibilities. To undo your previous actions, simply click the **Undo** icon on the **Quick Access Toolbar** as many times as you wish. The model view and the spreadsheets visually display the "undoing". Remember that spreadsheet edits are undone as well.



Redo Operations



RISAFoundation provides an unlimited 'Redo' capability so that you can easily redo any actions that were previously undone using the 'Undo' button. To redo actions that were previously undone, simply click the **Redo** icon on the **Quick Access Toolbar** as many times as you wish. The model view and the spreadsheets visually display the “redoing”. Remember that spreadsheet edits are redone as well.

Note: Changes made to the selection state of the model, the zoom level or rotation of the model view or to the Model Display Options settings are NOT undone or redone. Only a change to the model data can be undone or redone.

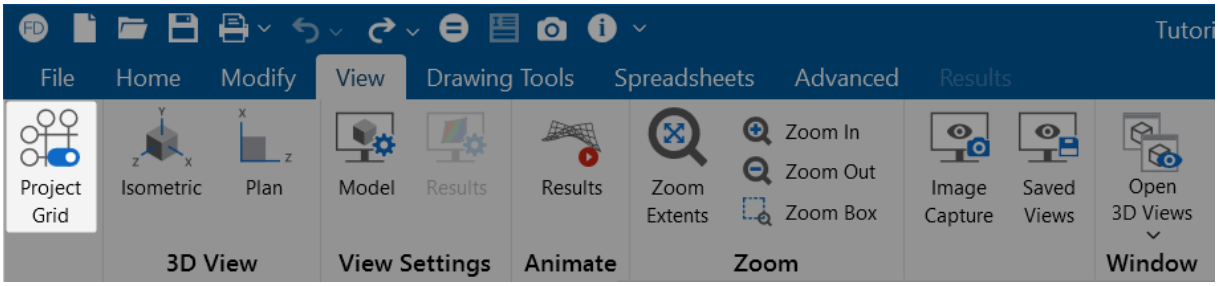
Project Grid

The Project Grid provides convenient snap points for modeling your foundation elements. It also provides convenient terminology to refer to locations in a model, such as "Grid Intersection C-4". The Project Grid is intended to be a permanent part of the model, so unlike the Drawing Grid, it is saved with the model.

Show / Hide Project Grid Lines

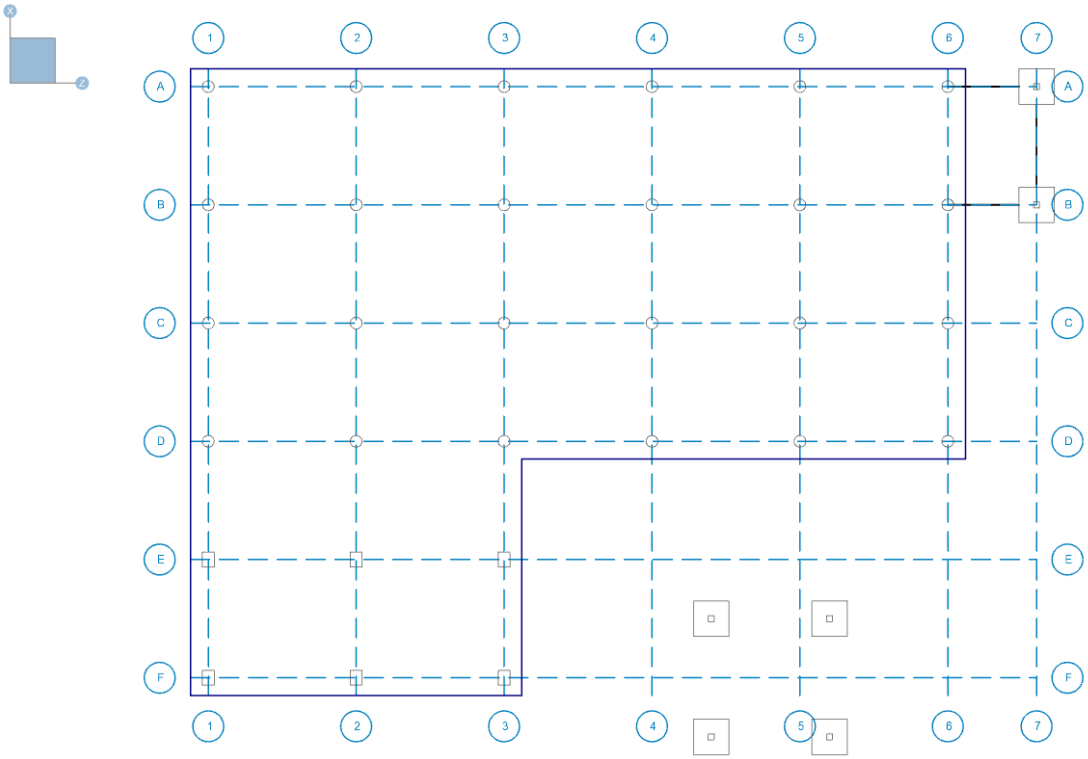
To view (or hide) Project Grid lines:

1. Go to the **View** ribbon.
2. Click the **Project Grid** icon to toggle the display of the Project Grid lines “on”.



The project grid lines appear in the model viewing panel.

3. (Optional) Click the **Project Grid** icon again to toggle the display of the lines “off”.

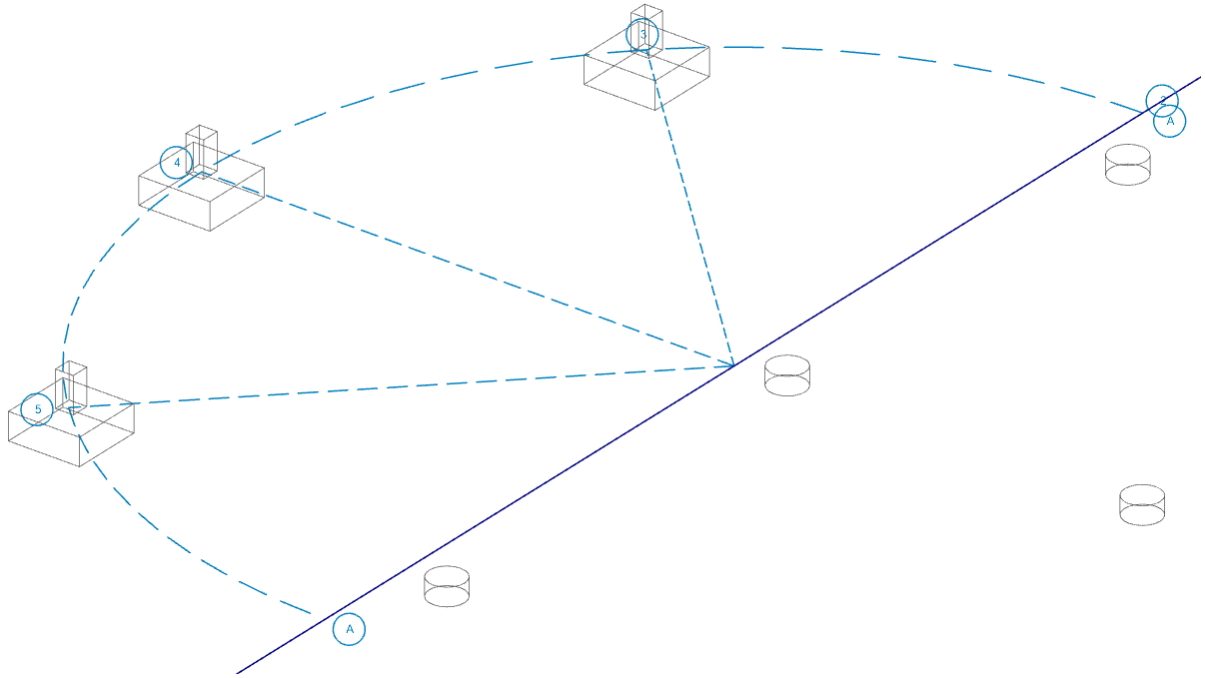


Show / Hide Project Grid Arcs

To view (or hide) the Project Grid arcs:

1. Go to the **View** ribbon.
2. Click the **Project Grid** icon to toggle the display of the arcs “on”.

   The project grid arcs appear in the model viewing panel.



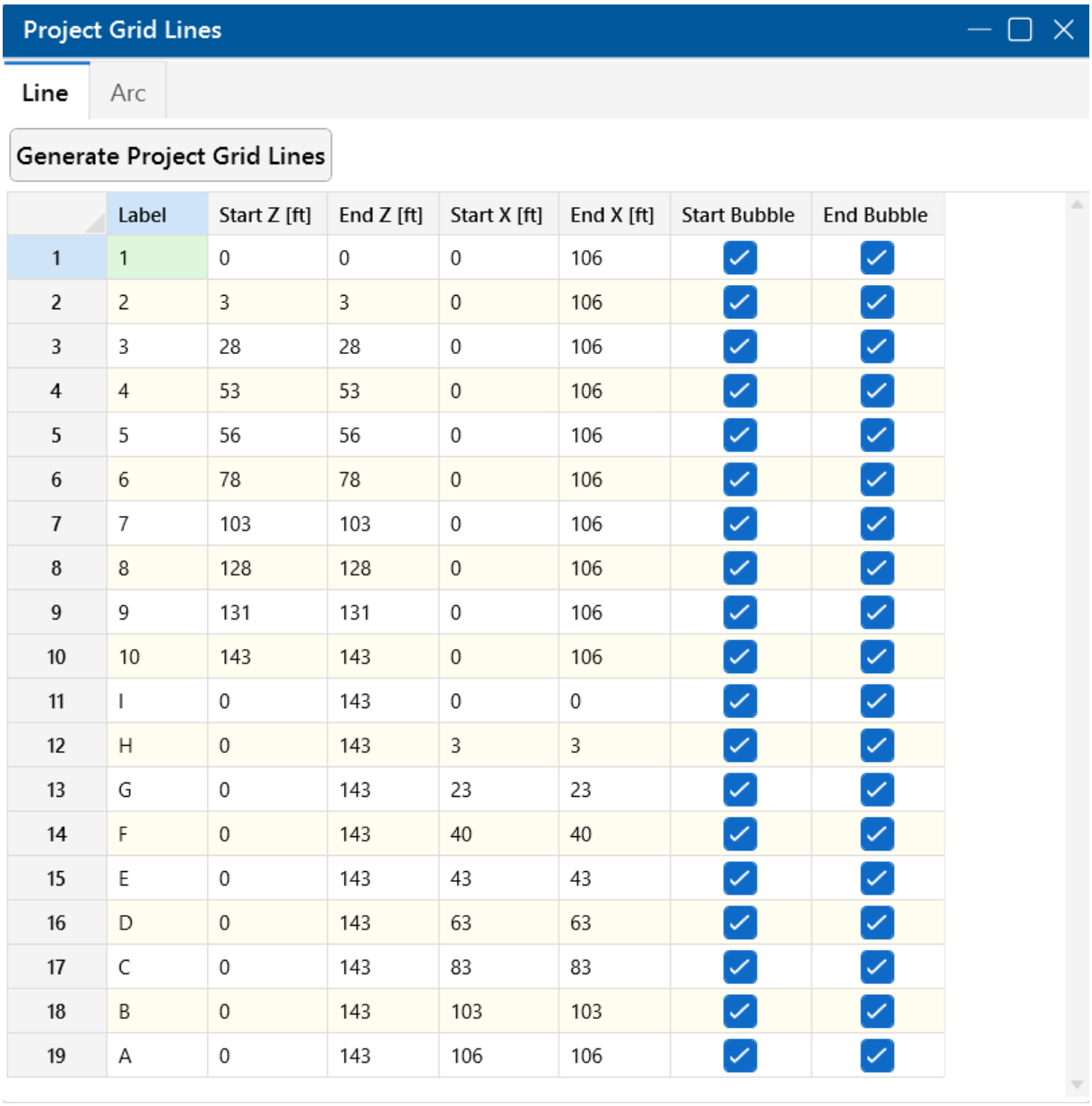
3. (Optional) Click the **Project Grid** icon again to toggle the display of the arcs “off”.

Project Grid Spreadsheet

To open the Project Grid Spreadsheet:

1. Click on **Project Grids** under Data Entry in the Explorer panel.

   The Project Grid window opens to the Line tab.



	Label	Start Z [ft]	End Z [ft]	Start X [ft]	End X [ft]	Start Bubble	End Bubble
1	1	0	0	0	106	✓	✓
2	2	3	3	0	106	✓	✓
3	3	28	28	0	106	✓	✓
4	4	53	53	0	106	✓	✓
5	5	56	56	0	106	✓	✓
6	6	78	78	0	106	✓	✓
7	7	103	103	0	106	✓	✓
8	8	128	128	0	106	✓	✓
9	9	131	131	0	106	✓	✓
10	10	143	143	0	106	✓	✓
11	I	0	143	0	0	✓	✓
12	H	0	143	3	3	✓	✓
13	G	0	143	23	23	✓	✓
14	F	0	143	40	40	✓	✓
15	E	0	143	43	43	✓	✓
16	D	0	143	63	63	✓	✓
17	C	0	143	83	83	✓	✓
18	B	0	143	103	103	✓	✓
19	A	0	143	106	106	✓	✓

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.

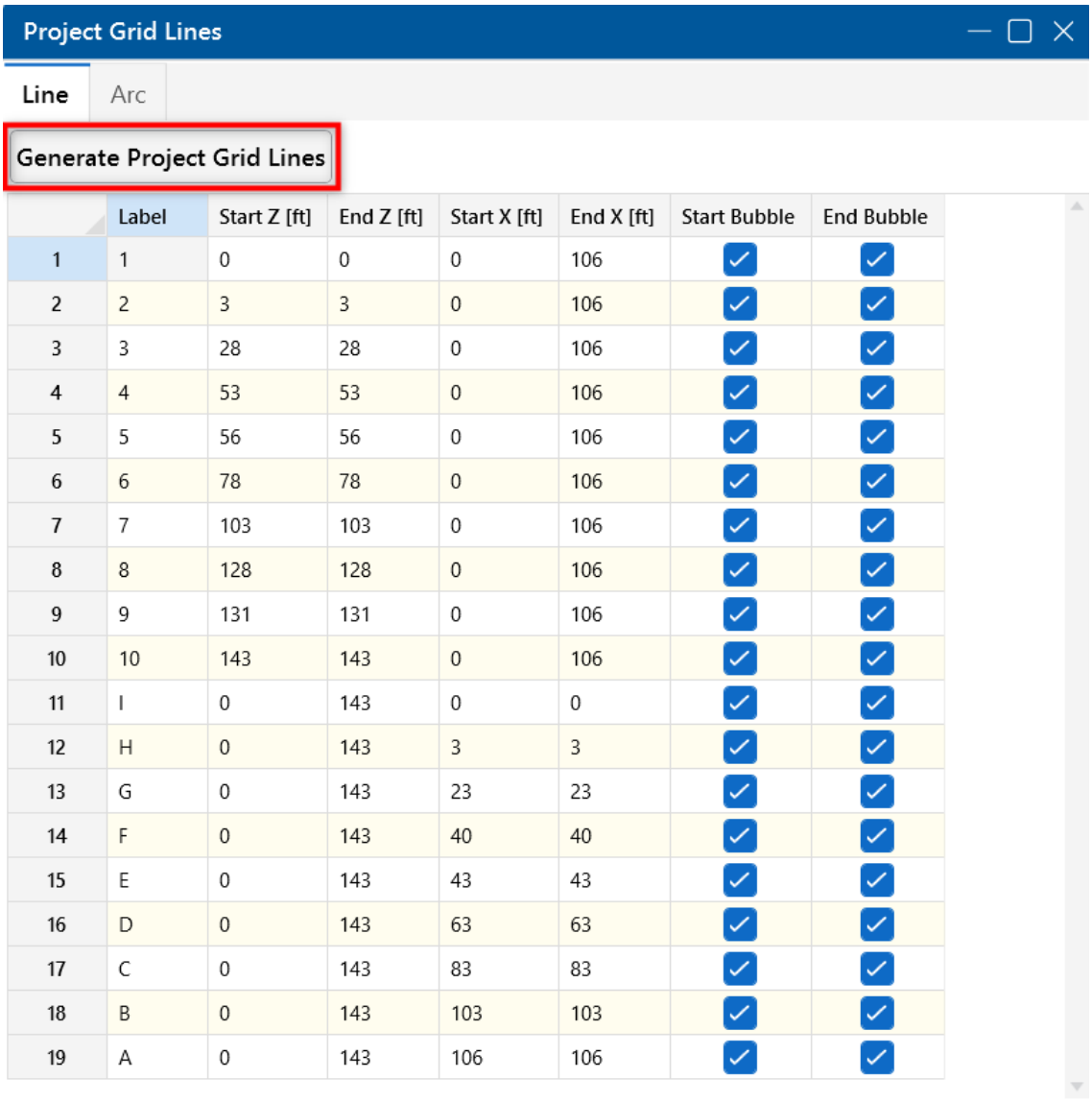
2. (Optional) Click the **Arcs** tab to view the generated arcs.

Project Grid Line Generation

Project Grid lines can be generated from the **Project Grid Spreadsheet**, or from the **Home** ribbon.

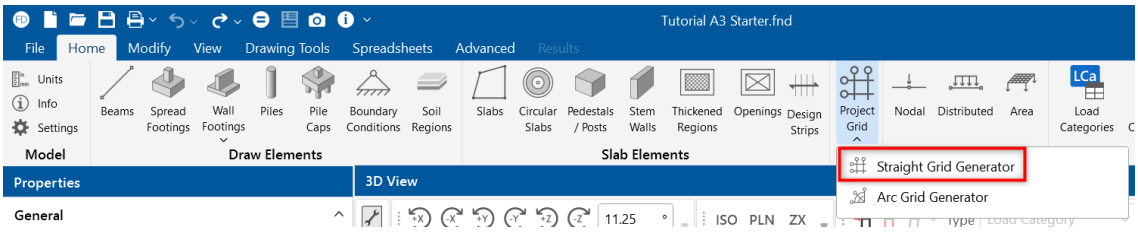
To generate Project Grid lines, do one of the following:

1. **Project Grid Spreadsheet** - if you have the spreadsheet open:
   a. If not already displayed, click on the **Line** tab.
   b. Click the **Generate Project Grid Lines** button.

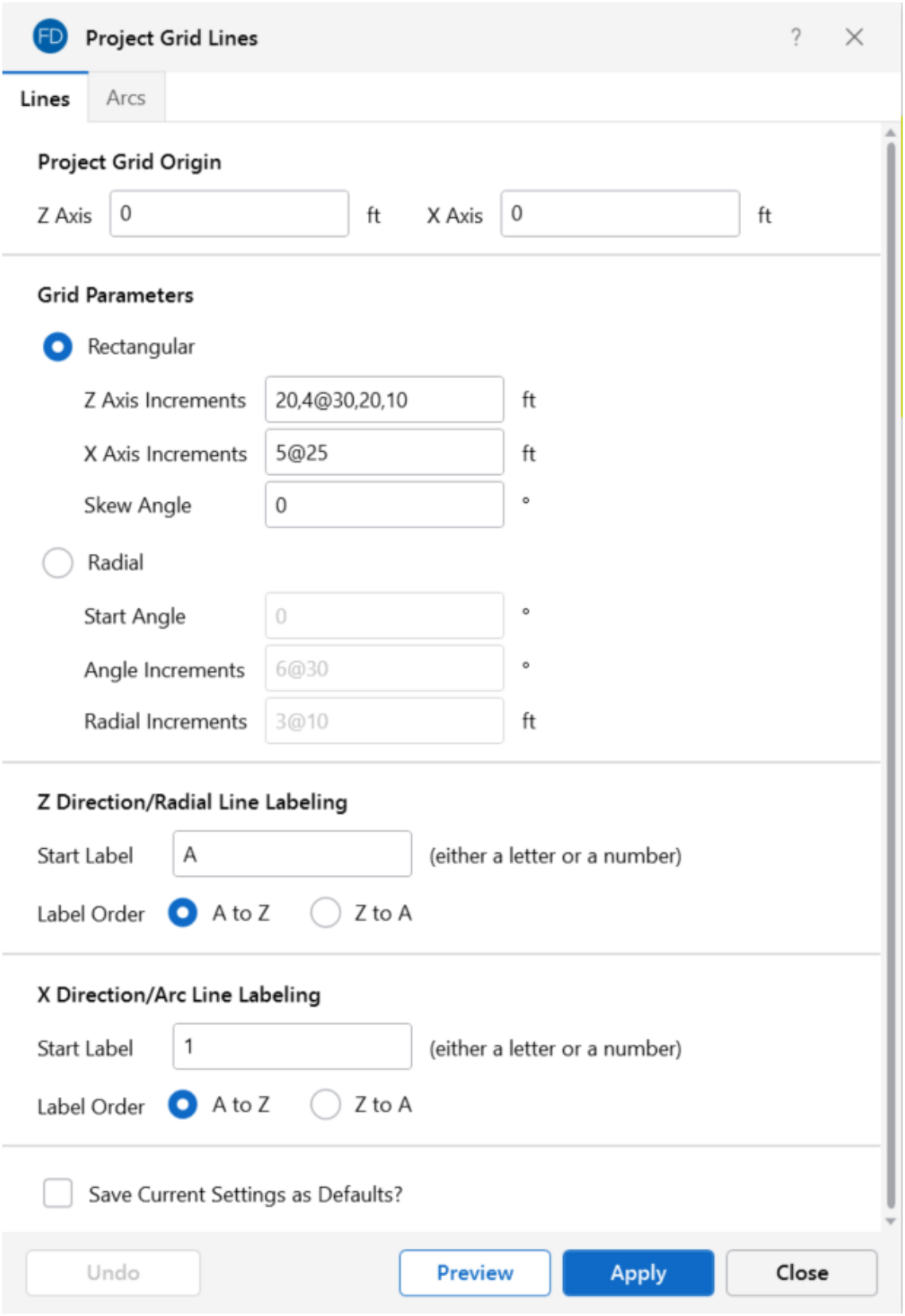


	Label	Start Z [ft]	End Z [ft]	Start X [ft]	End X [ft]	Start Bubble	End Bubble
1	1	0	0	0	106	✓	✓
2	2	3	3	0	106	✓	✓
3	3	28	28	0	106	✓	✓
4	4	53	53	0	106	✓	✓
5	5	56	56	0	106	✓	✓
6	6	78	78	0	106	✓	✓
7	7	103	103	0	106	✓	✓
8	8	128	128	0	106	✓	✓
9	9	131	131	0	106	✓	✓
10	10	143	143	0	106	✓	✓
11	I	0	143	0	0	✓	✓
12	H	0	143	3	3	✓	✓
13	G	0	143	23	23	✓	✓
14	F	0	143	40	40	✓	✓
15	E	0	143	43	43	✓	✓
16	D	0	143	63	63	✓	✓
17	C	0	143	83	83	✓	✓
18	B	0	143	103	103	✓	✓
19	A	0	143	106	106	✓	✓

2. **Home ribbon**:
   a. Click the **Project Grid** icon.
   b. Click on **Straight Grid Generator**.



The Project Grid Lines generation window opens to the Lines tab.


Project Grid Lines
Lines
Arcs
Project Grid Origin
Z Axis
0
ft
X Axis
0
ft
Grid Parameters
Rectangular
Z Axis Increments
20,4@30,20,10
ft
X Axis Increments
5@25
ft
Skew Angle
0
°
Radial
Start Angle
0
°
Angle Increments
6@30
°
Radial Increments
3@10
ft
Z Direction/Radial Line Labeling
Start Label
A
(either a letter or a number)
Label Order
A to Z
Z to A
X Direction/Arc Line Labeling
Start Label
1
(either a letter or a number)
Label Order
A to Z
Z to A
Save Current Settings as Defaults?
Undo
Preview
Apply
Close

The generator provides the ability to generate an entire grid system at once, rather than entering the grids in manually in the Project Grid Spreadsheet.

- When entering the increments, you can use symbols, such as "@", "/" and ",".
- The "@" entry can be used to specify multiple, equally spaced, grid increments.

  For example, if you wanted 7 increments at 10 units each, you would type "7@10" in the increment field.
- The "/" entry subdivides a larger increment into smaller equal increments.

  For example, the entry "12/4" would create 4 increments of 3 units each.
- Use commas (",") to enter multiple increments in the increment field.

  For example, if you wanted to define increments of 3, 4, 7 and 2 units, you could enter "3,4,7,2" in the increment field.
- The **Preview** button lets you see the Project Grid lines prior to applying them.
- The **Preview** and **Undo** buttons provide an easy way to preview a Project Grid before adding one to the model.

3. (Optional) Change the options to those you want applied.
4. Don’t forget to “Preview” your settings before you apply them.
5. Click **Apply** to generate and apply the Project Grid lines to the model.
6. Once the Project Grid is specified and displayed in the model view, it provides snap points while drawing your model.

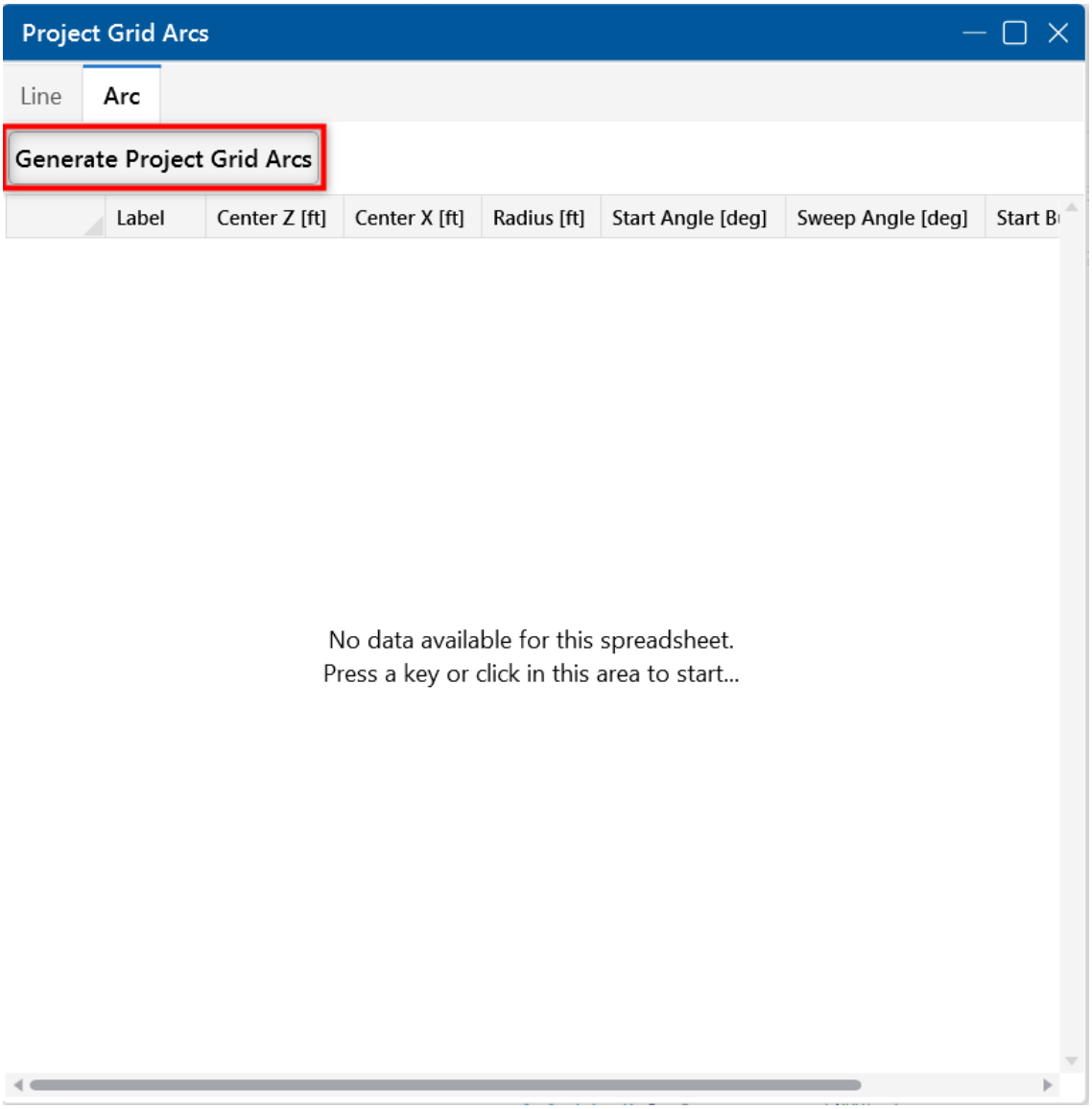
Project Grid Arc Generation

Project Grid arcs can be generated from the **Project Grid Spreadsheet**, or from the **Home** ribbon.

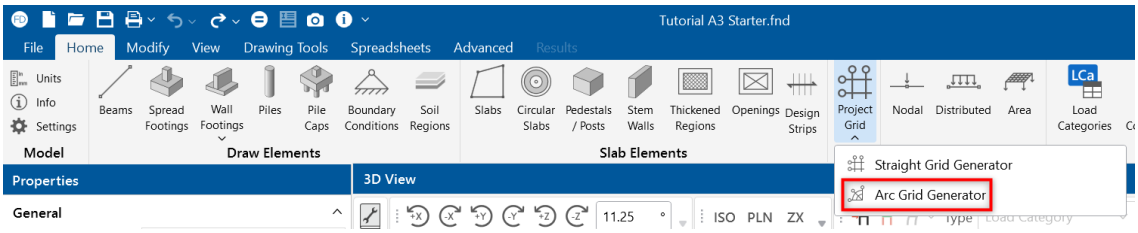
To generate Project Grid arcs, do one of the following:

1. **Project Grid spreadsheet** - if you have the spreadsheet open:
   a. If not already displayed, click on the **Arc** tab.

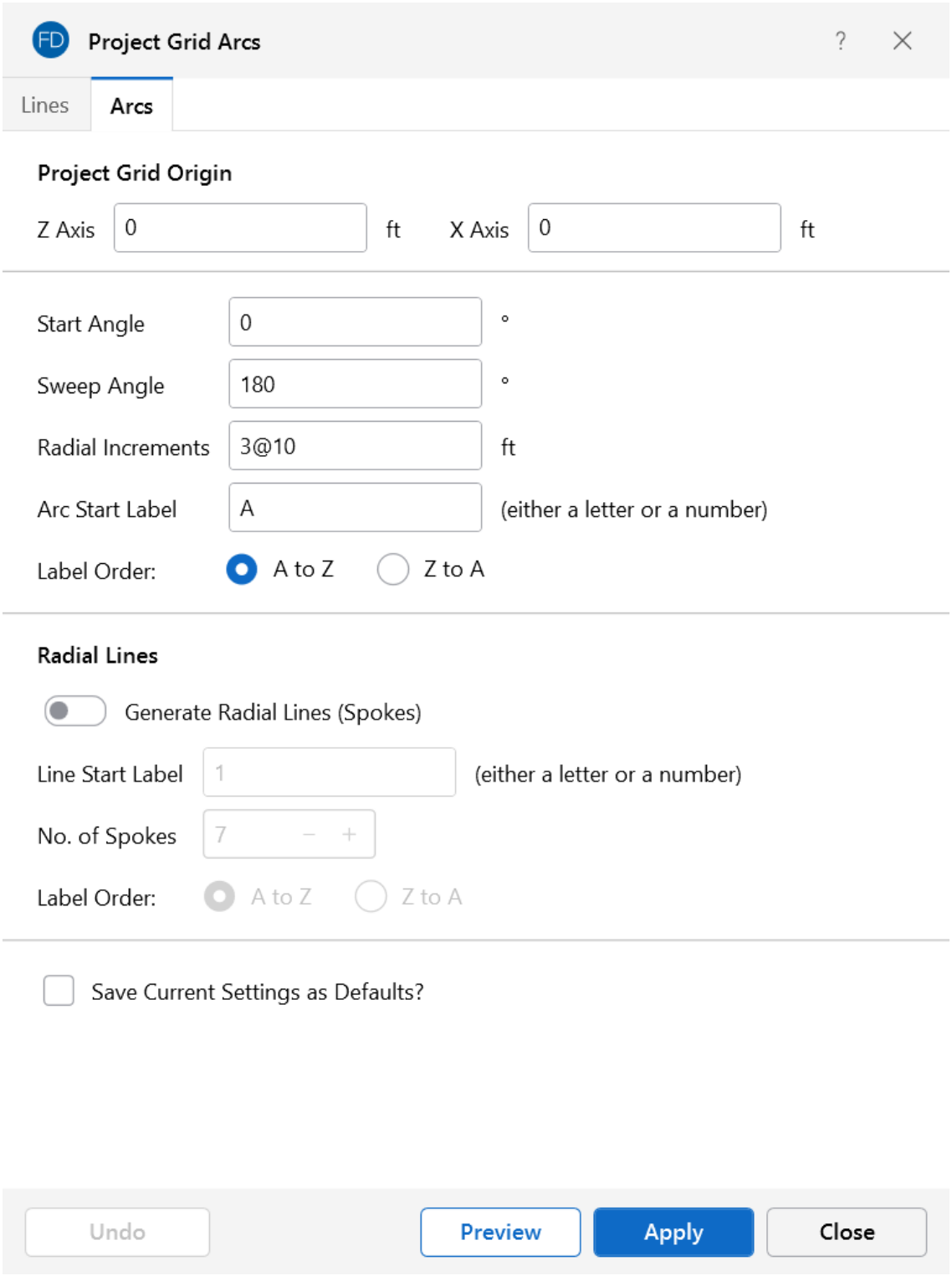
b. Click the **Generate Project Grid Arcs** button.



2. **Home ribbon**:
   a. Click the **Project Grid** icon.
   b. Click on **Arc Grid Generator**.



The Project Grid Arcs generation window opens to the Arcs tab.


Project Grid Arcs
Lines
Arcs
Project Grid Origin
Z Axis
0
ft
X Axis
0
ft
Start Angle
0
°
Sweep Angle
180
°
Radial Increments
3@10
ft
Arc Start Label
A
(either a letter or a number)
Label Order:
A to Z
Z to A
Radial Lines
Generate Radial Lines (Spokes)
Line Start Label
1
(either a letter or a number)
No. of Spokes
7
Label Order:
A to Z
Z to A
Save Current Settings as Defaults?
Undo
Preview
Apply
Close

The generator provides the ability to generate an entire grid system at once, rather than entering the grids in manually in the Project Grid Spreadsheet.

- When entering increments, you can use symbols, such as "@", "/" and ",".
- The "@" entry can be used to specify multiple, equally spaced, grid increments.

  For example, if you wanted 7 increments at 10 units each, you would type "7@10" in the increment field.
- The "/" entry subdivides a larger increment into smaller equal increments.

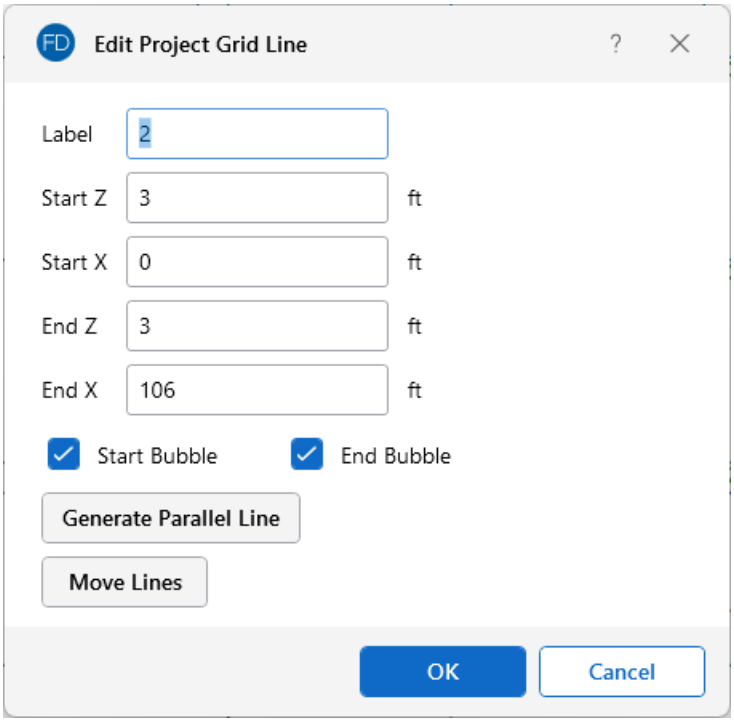
  For example, the entry "12/4" would create 4 increments of 3 units each.
- Use commas (",") to enter multiple increments in the increment field.

  For example, if you wanted to define increments of 3, 4, 7 and 2 units, you could enter "3,4,7,2" in the increment field.
- The **Preview** button lets you see the Project Grid arcs prior to applying them.
- The **Preview** and **Undo** buttons provide an easy way to preview a Project Grid before adding one to the model.

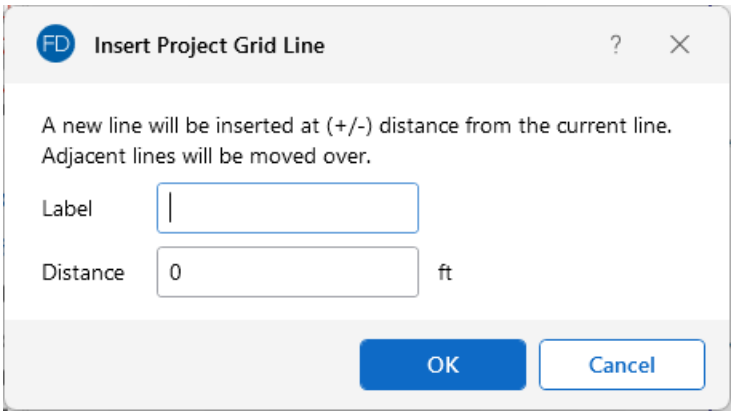
3. (Optional) Change the options to those you want applied.
4. Don’t forget to “Preview” your settings before you apply them.
5. Click **Apply** to generate and apply the Project Grid arcs to the model.
6. Once the Project Grid is specified and displayed in the model view, it provides snap points while drawing your model.

Edit the Project Grid Lines (Adding or Moving Grid Lines)

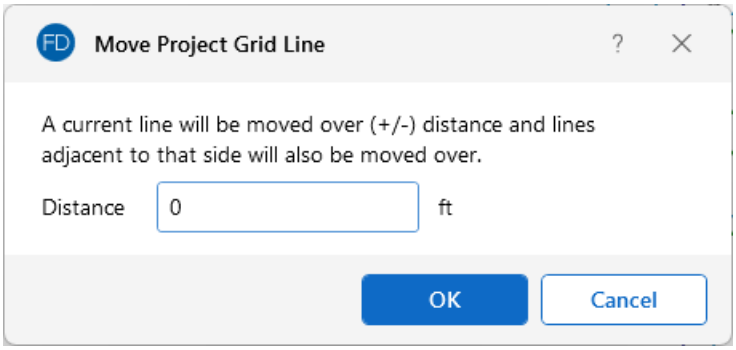
Once a Project Grid has been created, the grid lines can be moved, or additional grid lines can be created by double-clicking on a grid line in the model view.



The **Generate Parallel Line** button allows a line to be created by offsetting an existing grid line.



The **Move Lines** button allows a grid line to be moved by a specified distance. A positive distance moves the line in the positive global direction, while a negative distance moves the line in the opposite direction.

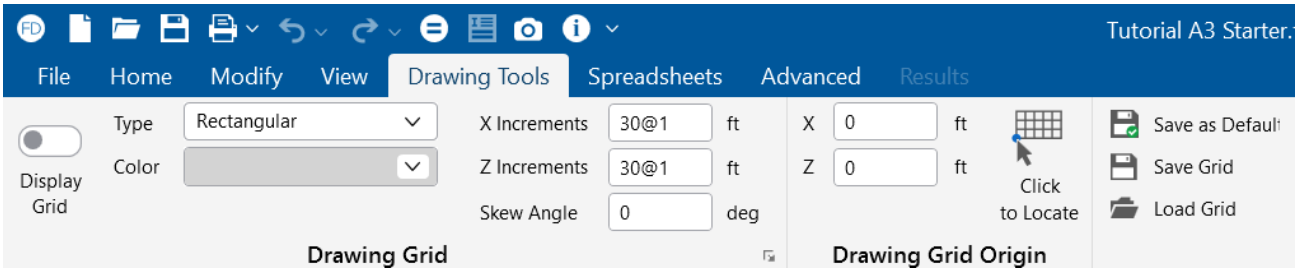


Drawing Grid

The **Drawing Grid** is a tool that helps you draw new beams, slabs and footings 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 beams, slabs and footings, the joints used to define them are created automatically. These joints remain in their locations if the grid is relocated.

Drawing Grid Features

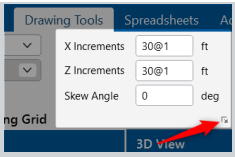
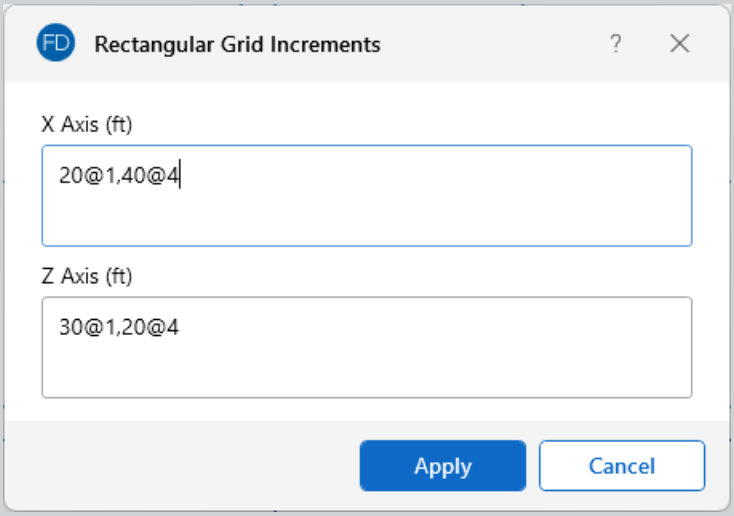
The Drawing Grid tools can be found on the **Drawing Tools** ribbon. These features allow you to define, save and reuse rectangular, radial and DXF underlay drawing grids. You can also set a specific grid as your default.

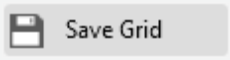


To use the Drawing Grid features refer to the following table and sections.

Drawing Grid Features

Feature	Description
Drawing Grid section	
	The **Display Grid** icon lets you turn on (show) or off (hide) the drawing grid in the current model. • **Gray slider** - indicates that the drawing grid is off (hidden) • **Blue slider** - indicates that it is on (showing) For more information, see Show / Hide the Drawing Grid.

Feature	Description
	
Type	The **Type** feature lets you choose between a rectangular drawing grid, radial drawing grid or DXF underlay.
Color	The **Color** feature lets you choose what color you would like the drawing grid lines displayed in.
X Increments	The **X Increments** input lets you set the number of and width between the drawing grid lines.
Z Increments	The **Z Increments** input lets you set the number of and height between the drawing grid lines.
Skew Angle	The **Skew Angle** input lets you skew the drawing grid perpendicular to the global axis plane.
	The icon located to the bottom-right of the Skew Angle feature lets you open the **Rectangular Grid Increments** window.  The Rectangular Grid Increments window lets you enter more X and Z incremental values.
Drawing Grid Origin section	
X	The **X** grid origin lets you set the increment for the X direction. For more information, see Rectangular Drawing Grid.
Z	The **Z** grid origin lets you set the increment for the Z direction. For more information, see Rectangular Drawing Grid.

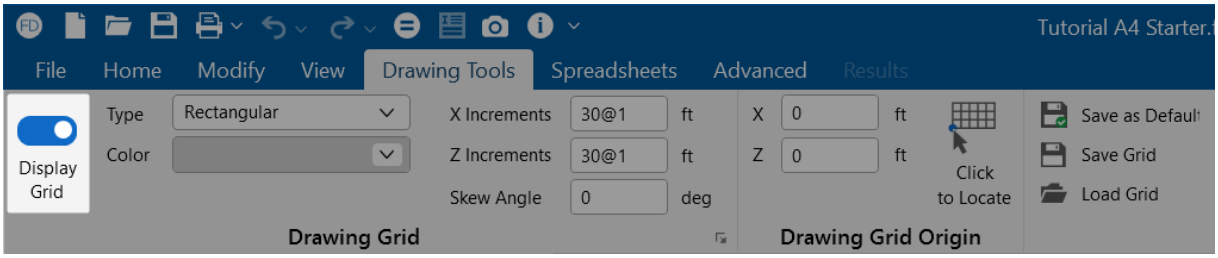
Feature	Description
	The **Click to Locate** icon lets you specify or relocate the origin of the drawing grid. For more information, see Relocating the Drawing Grid.
Save and Recall features section	
	The **Save as Default** feature lets you save any of the grid information as the default settings so that when you start a new model, that information is already there. For more information, see Save a Drawing Grid Default.
	The **Save Grid** feature lets you save a drawing grid so you can recall it for later use. Saved drawing grids are model independent, i.e. when you save a grid, you can reuse it in any other model you are working with in the future. For more information, see Save a Drawing Grid.
	The **Load Grid** feature lets you recall a saved drawing grid. For more information, see Recall a Drawing Grid.

Show / Hide the Drawing Grid

The Drawing Tools ribbon includes a toggle that allows you to show or hide the drawing grid in the current model.

To show or hide the drawing grid:

1. Open to the **Drawing Tools** ribbon.
2. Click on the **Display Grid** icon.



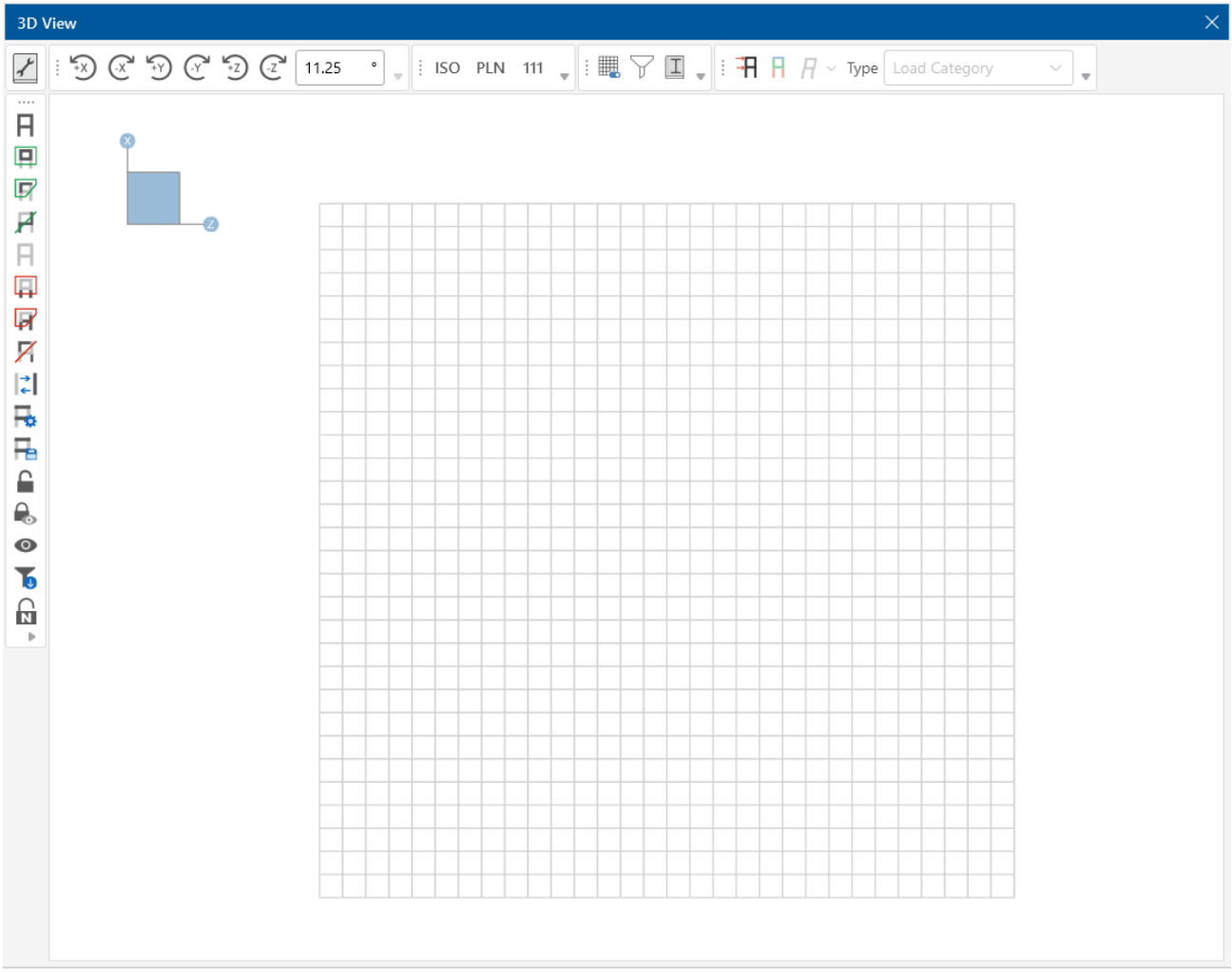
- When the Display Grid slider is **blue**, it indicates that the drawing grid is turned "on" (shown).
- When the Display Grid slider is **gray**, the drawing grid turned "off" (hidden).

Drawing Grid Type

Rectangular Drawing Grid

The rectangular drawing grid is defined by increments in two directions. The **X** and **Z Origins** are where you want the grid increments to start. The **X** and **Z Increments** are the distances between the

grid points or lines in two global directions.



You may use symbols such as "@", "/" and "," when entering the drawing grid increments.

- The "@" entry may be used to specify multiple, equally spaced, grid increments.

  For example, if you wanted 7 increments at 10 units each, you would type "7@10" in the increment field.

- The "/" entry subdivides a larger increment into smaller equal increments.

  For example, the entry "12/4" would create 4 increments of 3 units each.

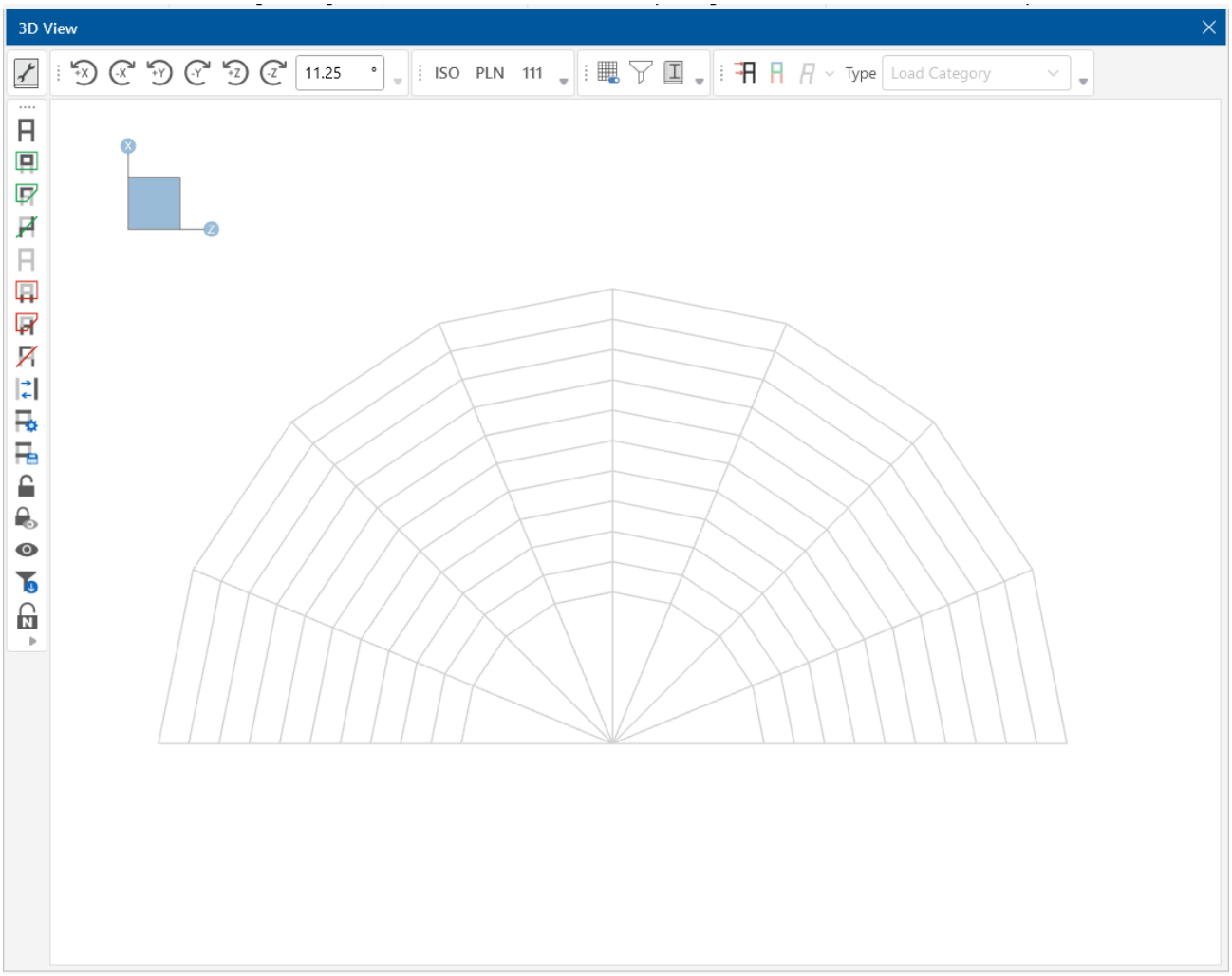
- Use commas (",") to enter multiple increments in the increment field.

  For example, if you wanted to define increments of 3, 4, 7 and 2 units, you could enter "3,4,7,2" in the increment field.

Once the drawing grid is specified and displayed in the model view, it provides snap points while drawing your model.

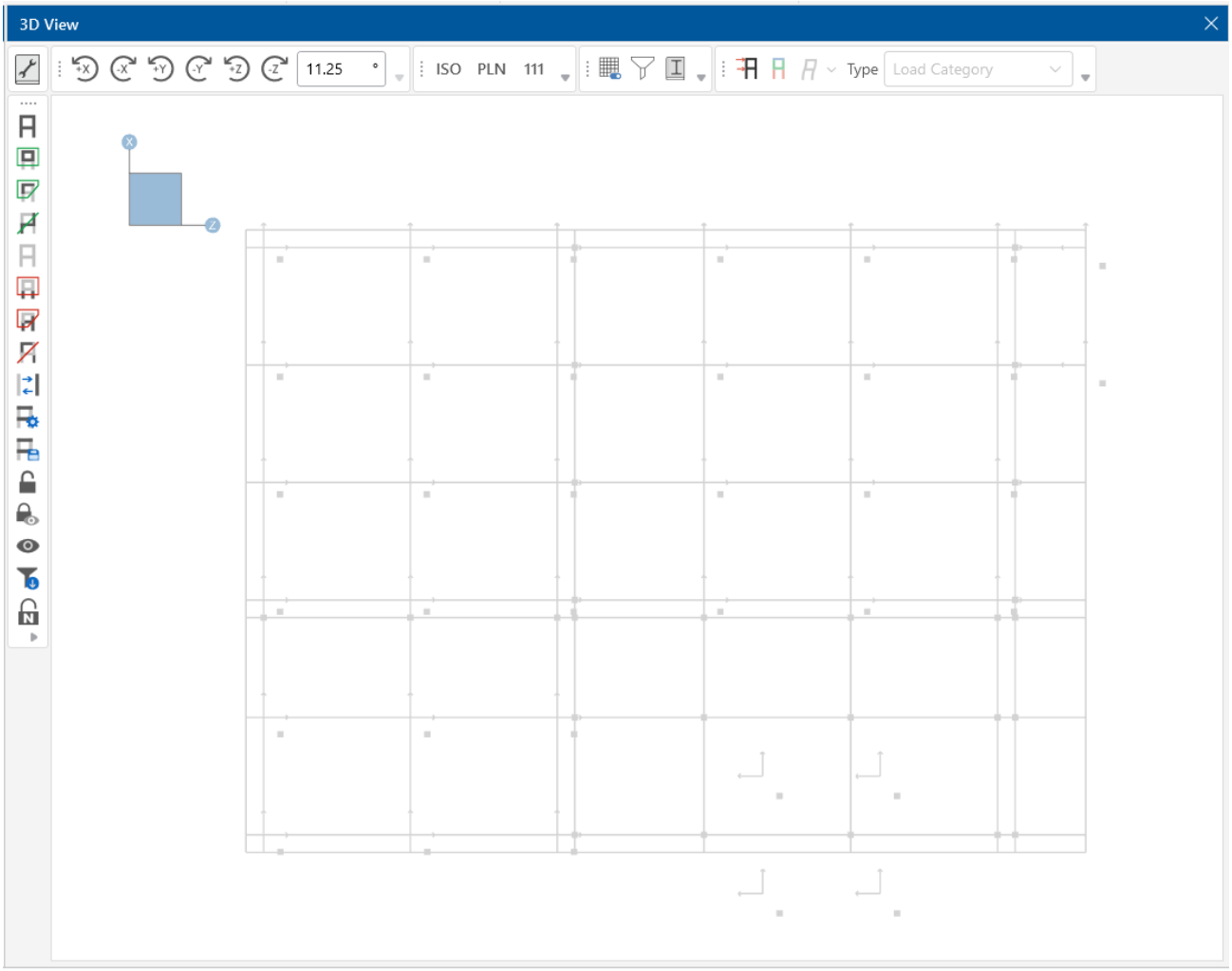
Radial Drawing Grid

Increments in two polar directions define a radial drawing grid. The **Drawing Grid Origin** is the point about which the grid will rotate. The default is at the global origin (0,0,0). The **Start Angle** defines the angle from the global axis that the first spoke will be drawn. The **Angle Increment** controls the number and angular spacing of the spokes in the grid. The **Radial Increments** controls the number and location of the rings in the grid.



DXF Underlay Grid

This type should only be chosen when you want to import a DXF Grid from a DXF file. The following is an example of an imported DXF file underlay grid. For more information, see Import DXF Grid.



Relocating the Drawing Grid

The origin of the drawing grid may be specified in two ways. The first is to enter the exact global coordinates for the origin. This can be done by entering the values in the **X Origin** and **Z Origin** boxes. The second option is to specify the origin by clicking on an existing point in the model. This option is available by clicking the **Click to Locate** button, and then clicking on the specific location in the model view where you wish the drawing grid origin to be located. The drawing grid origin is then moved to this point.

Skewed Drawing Grid

The rectangular drawing grid can be skewed by specifying a skew angle. This option is available in the **Drawing Grid Section** section of the **Drawing tools** ribbon. This skew angle allows for the creation of a regular rectangular drawing grid, but displayed in the model view at the specified angle, inclined from the global axis.

Save a Drawing Grid Default

You may save any of the grid information as the default setting so that when you start a new model, that information is already there.

To save a drawing grid as the default:

1. In the **Drawing Grid** and **Drawing Grid Origin** sections of the Drawing Tools ribbon, enter the information that you want to save.
2. Click on 

Save as Default in the Drawing Tools ribbon.

Save a Drawing Grid

To save a drawing grid:

1. In the **Drawing Grid** and **Drawing Grid Origin** sections of the Drawing Tools ribbon, enter the information that you want to save.
2. After defining the grid, click on 

Save Grid in the Drawing Tools ribbon.

   You are prompted for a name for the drawing grid. Once you provide a name, the drawing grid is saved and added to the list of grids available for recall.

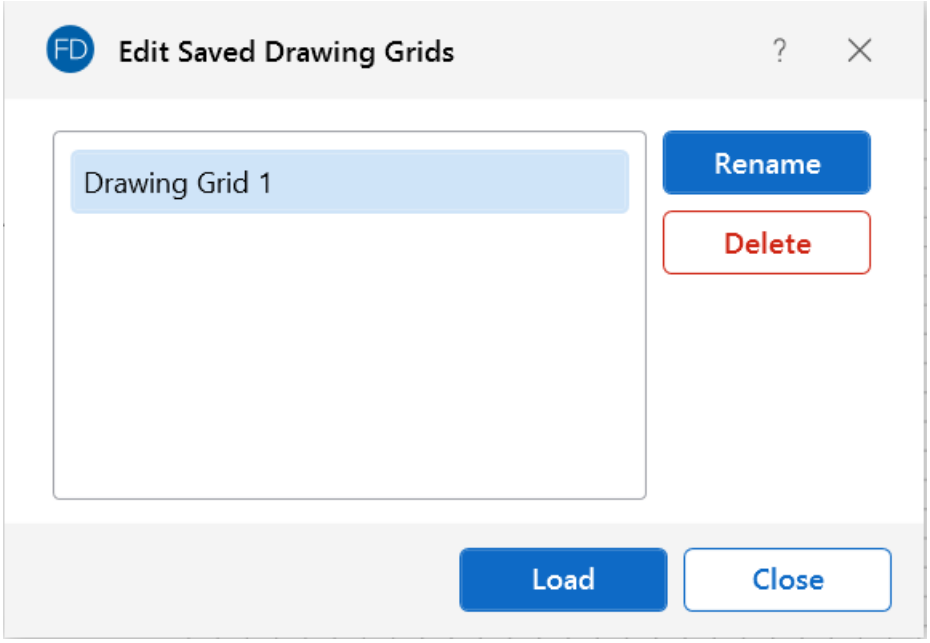
Recall a Drawing Grid

To recall a previously defined drawing grid:

1. Click on 

Load Grid in the Drawing Tools ribbon.

   The **Edit Saved Drawing Grids** window opens.



2. Click on the drawing grid you want.
3. Click the **Load** button.

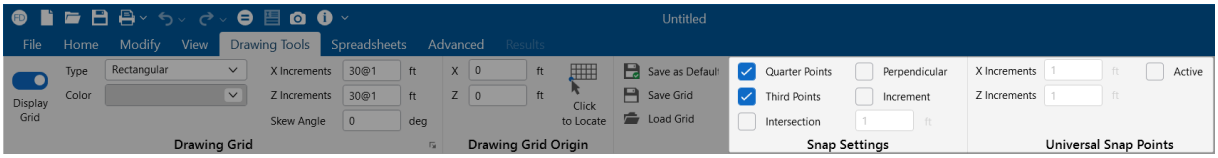
Note: You can also “Rename” drawing grids and “Delete” drawing grids from the **Edit Saved Drawing Grids** window. To choose either option, click the **Rename** or **Delete** button after choosing the drawing grid.

Snap Points

Snap Points let you draw in the model view without the use of grids.

To view or modify the snap point settings:

1. Go to the **Drawing Tools** ribbon.
2. Go to the **Snap Settings** section.



3. (Optional) Make whatever modifications you want.
    - You can set the program to automatically snap to the **Quarter Points** and/or **Third Points** and/or **Intersections** of a beam by checking the appropriate boxes.

- The **Universal Snap Points** section is used to define a snap grid for "free" drawing to any incremental location on a plane. To activate this feature, check the **Active** box.

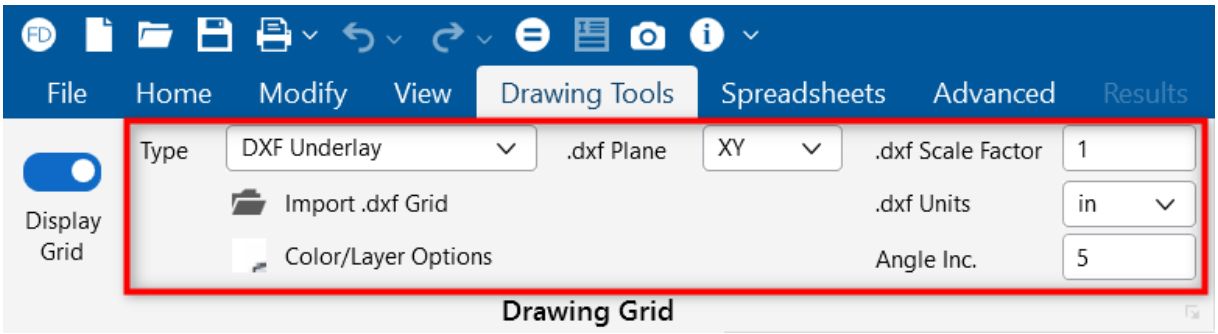
When snap points are activated, a red dot appears on your screen as you move your drawing cursor over one of these points after you have selected to draw an element. The exact coordinates of this point, and whether it is a 1/3 or 1/4 point of a beam, are reported next to the cursor.

Import DXF Grid

Import DXF lets you import a DXF drawing into the model view as a grid. The DXF image is imported so that you can snap to any point or intersection to aid in the drawing of your model. The DXF Import Drawing Grid supports Lines, Polylines, Circles, Arcs, Polylines with Arcs and Points.

To Import a DXF Drawing Grid:

1. Go to the **Drawing Tools** ribbon.
2. Click on the **Type** drop-down and choose **DXF Underlay**.



The following table provides information on the various related Drawing Grid settings.

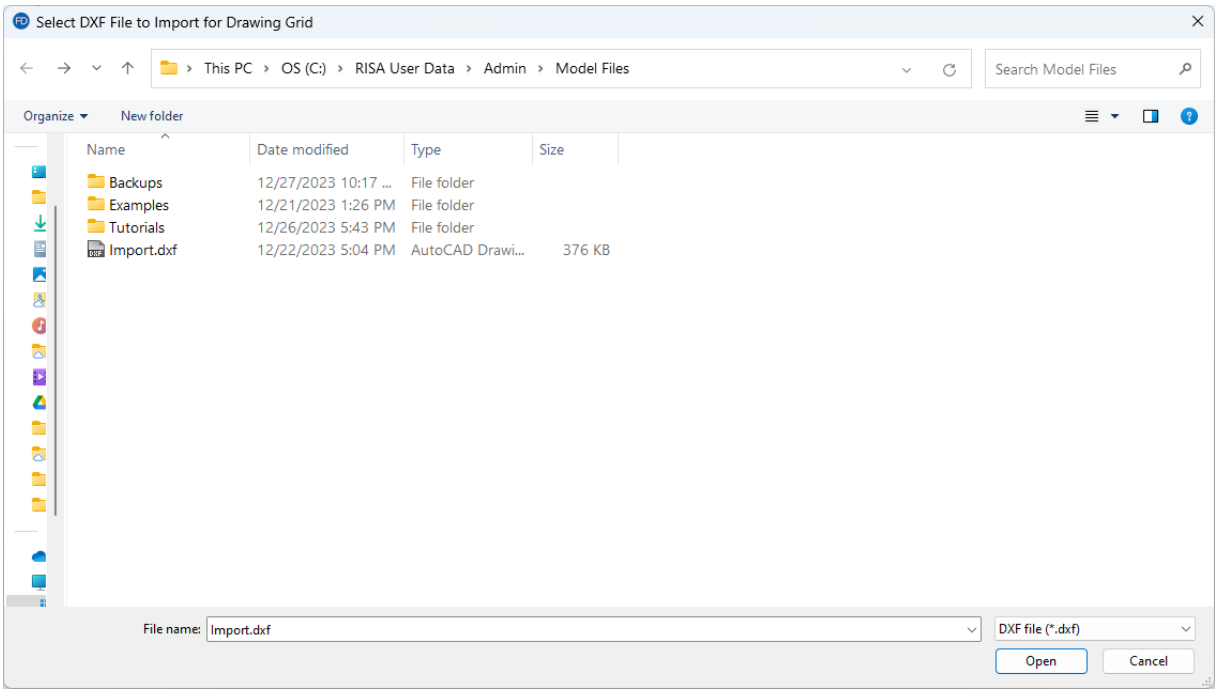
DXF Import Drawing Grid Settings

Setting	Description / Explanation
.dxf Plane	This setting lets you choose the DXF drawing plane to import from the DXF file. The DXF file will be flattened to this plane and all lines/points visible from this plane will be imported.
.dxf Scale Factor	This setting lets you enter the scale factor that will cause the DXF file to be scaled up, or down to full scale. For instance, if you have created a scaled model in AutoCAD at a scale of 1/4"=12", then the appropriate scale factor to produce a full size RISAFoundation model would be 48. The default is 1.0.
.dxf Units	This setting lets you choose the same units you used in the CAD model from which you produced the DXF file. The supported DXF units are none, inches, feet, mm, cm and

Setting	Description / Explanation
	meters.
Angle Increment	The program breaks an arc or circle into straight line segments to allow you to snap to them. This setting indicates the number of degrees that an imported arc or circle should be broken into.

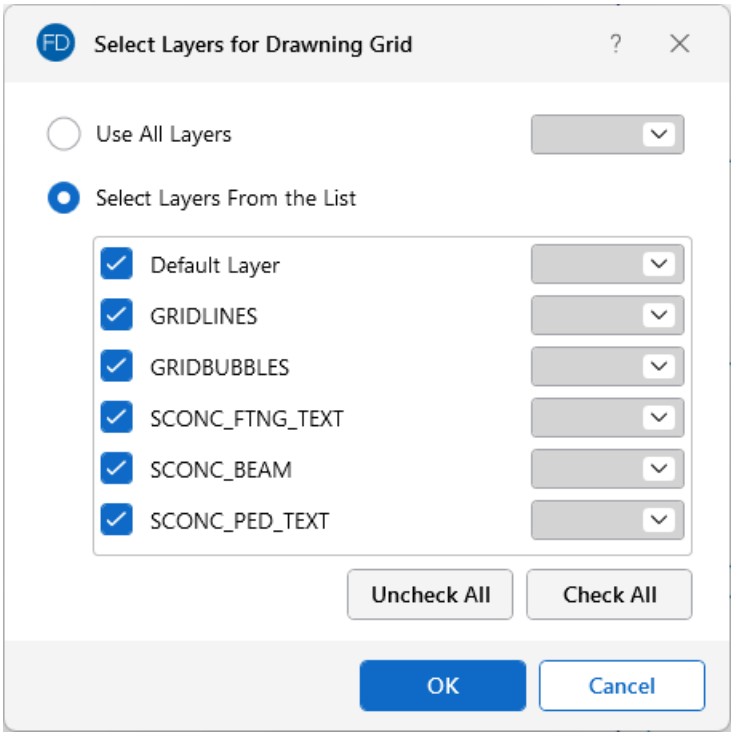
3. Assign the DXF import options like **Scale Factor**, **Units** and **Angle Inc.**, as well as the **.dxf Plane**.
4. Click on **Import .dxf Grid**.

   The Select DXF File to Import for Drawing Grid window opens.



5. Choose the DXF file to import and click **Open**.

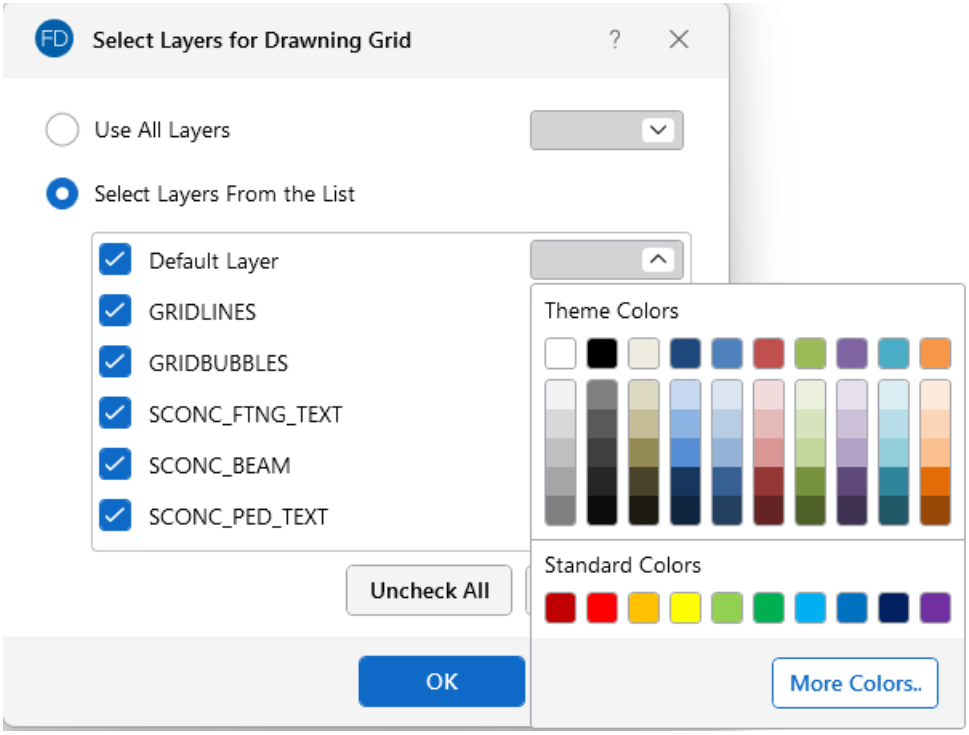
   Once the file is imported, a **Select Layers for Drawing Grid** window opens.



The default option is to **Use All Layers**.

6. Choose whether to **Use All Layers** or **Select Layers from the List**.
7. If you chose **Select Layers From the List**, check the layers you want and uncheck those you don't want.

   You can select the color of the grid lines/points from the color box next to each layer name.
8. (Optional) Click on a layer's color box arrow to select a color other than the gray default.



9. Click **OK** to save your choices and close the window.
10. Set the **Drawing Grid Origin**, or use the drop-down menu to select **Click to Locate** origin.

Locating the DXF Drawing Grid

The origin of the drawing grid may be specified in two ways. The first way to specific the origin 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 **Grid Origin** section. The second option is to specify the origin by clicking on an existing point in the model. This option is available by selecting the **Click to Locate Origin** option from the drop-down menu, clicking the **OK** button, and then clicking on the specific location in the model view where you wish the drawing grid origin to be located. After you have browsed for your file, the drawing grid origin will then be set to this point.

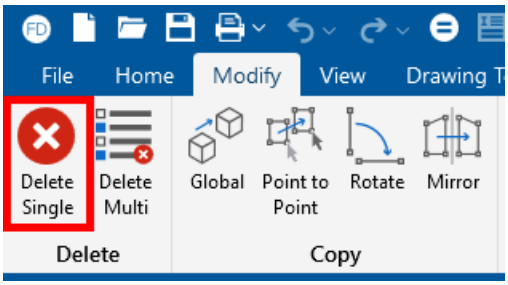
Delete Elements

You can delete parts of a model based on the current selection state, or you can click on the items you wish to delete individually.

Delete a Single Model Element

To delete a single model element, or multiple model elements, one at a time:

1. Go to the **Modify** ribbon.
2. Click the **Delete Single** icon.



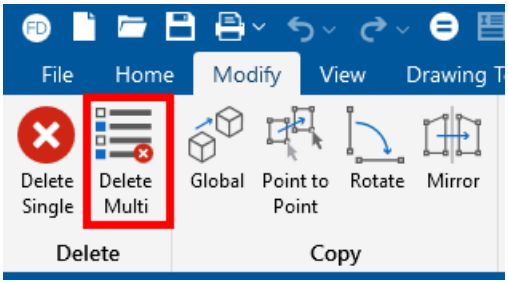
3. Click on the model element you want to remove.
4. Repeat **step 3** for each additional model element you want to remove.
5. When finished, click the **Delete Single** icon again to toggle off the single delete feature.

Delete Multiple Model Elements

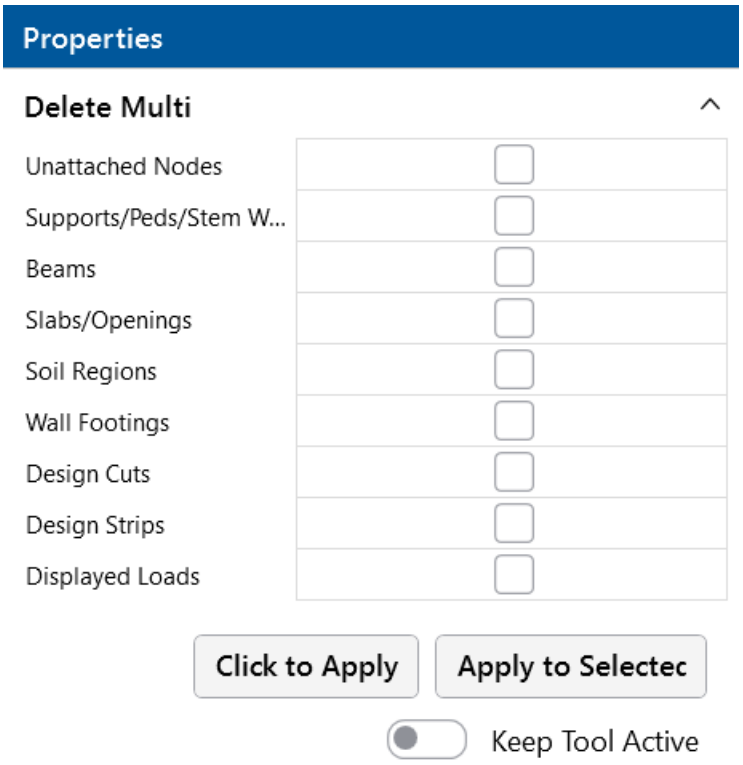
If you wish to delete based on the current selection state, you must use the check boxes to define the criteria the program will use to perform the deletion. Only items that are selected and that have their check boxes "checked" will be deleted. The choices let you delete nodes, supports, beams, slabs, soil regions, wall footings, design cuts, design strips and/or displayed loads.

To delete multiple model elements at once:

1. Go to the **Modify** ribbon.
2. Click the **Delete Multi** icon.



The Properties panel displays a list of all model elements that can be deleted.



3. Do one of the following, depending on exactly which elements you want to delete.

   To remove ALL of one or more element types:

   a. Check the box of each element type you want deleted from the model.

   b. (Optional) Click on the **Keep Tool Active** slider if you want the 'Delete Multi' list to remain open in the Properties panel, so that you can delete additional model elements.

   c. Click the **Click to Apply** button.

      This action removes all elements of the type(s) you have chosen in the Properties panel.

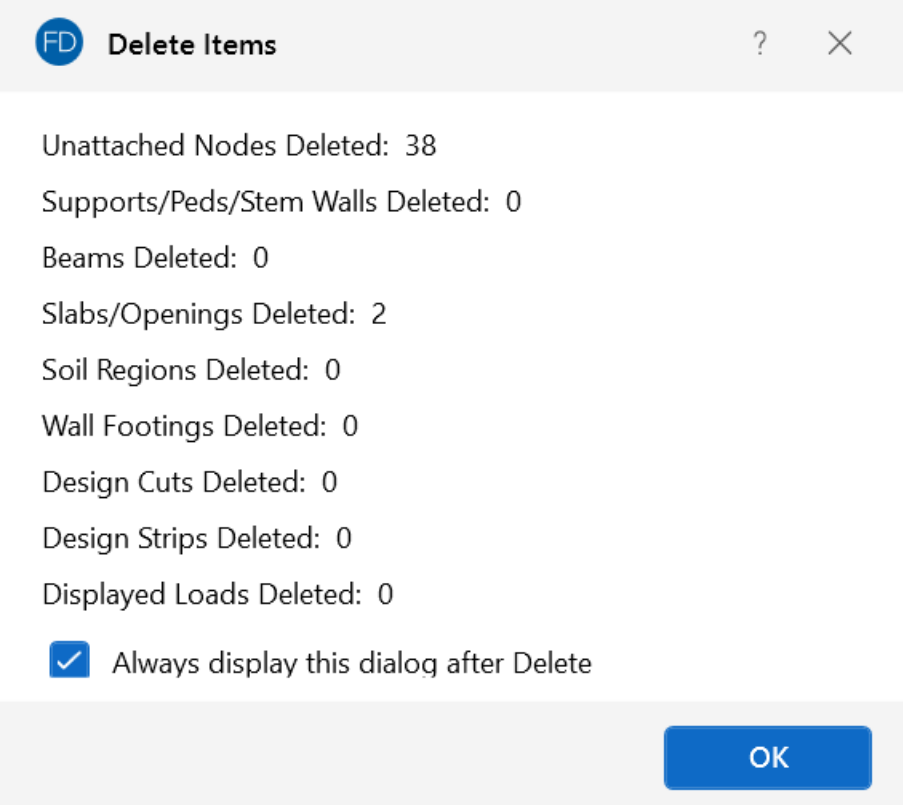
   To remove specific elements of multiple types:

   a. Check the box of each element type you want to delete from the model.

   b. Hold down the **CTRL** key (on your keyboard) while clicking on each model element you wish to delete.

   c. Click the **Apply to Selected** button.

A **Delete Items** window opens displaying the number of each type of model element that was deleted.



d. Click **OK**.

The window closes and the items are deleted from the model view.

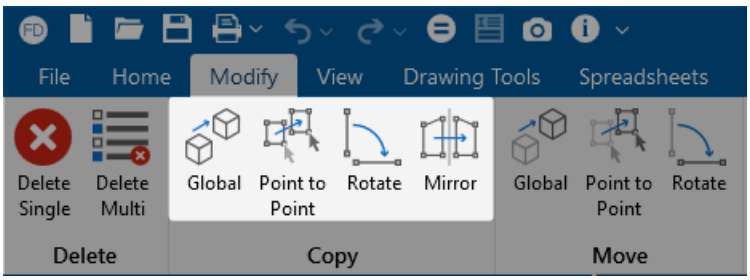
Keep in mind that if you delete nodes, any beams, slabs, footings, etc. attached to the deleted nodes MUST also be deleted, regardless of whether those elements are selected or not.

Note: If you request deletion of displayed loads, you'll get exactly that. Any load currently displayed will be deleted. By controlling what loads are displayed via the **Loads** tab in the **Model View Settings** window, you can easily delete specific types of loads for particular load category or load combination.

Note: You may undo any mistakes by clicking the **Undo** button in the Quick Access toolbar.

Copy Model Elements

The model elements copy options can be found on the **Modify** ribbon in the **Copy** section.



Copy Selected Items

To copy selected items:

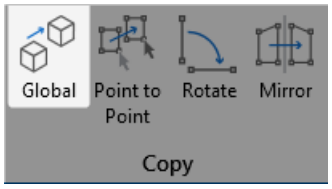
1. Open a model view, if one is not already open.
   a. Go to the **View** ribbon.
   b. Click the **Open 3D Views** icon.
2. Select the items you want to copy.
3. To *linearly* copy the selection:
   a. Go to the **Modify** ribbon.
   b. Under the **Copy** section, choose one of the following options:
      - **Global**
      - **Point to Point**
      - **Rotate**

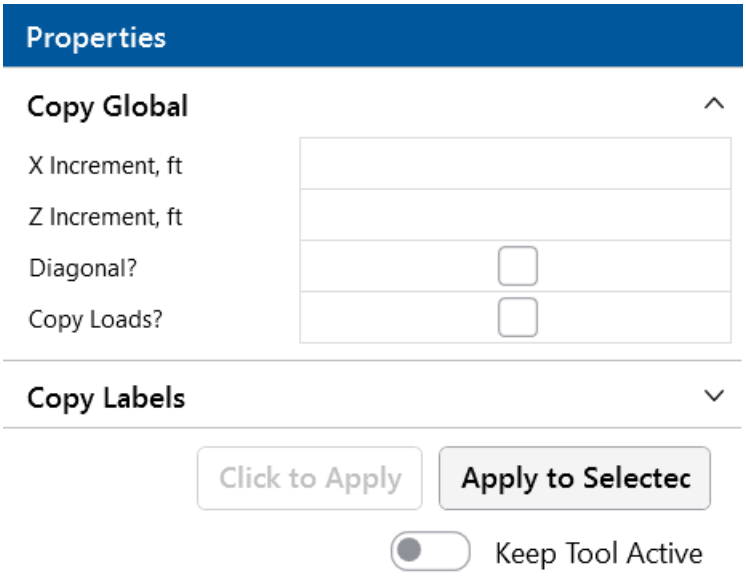
      See explanations in the following sections for more information about these choices.
4. To *Mirror* the selection, click the **Mirror** icon and specify the mirror plane.

Note:
- Use the "@" symbol to specify multiple equal increments. For example specifying "3@10" will give you 3 copies at 10 units apart.
- You can undo any mistakes by clicking the **Undo** icon in the Quick Access toolbar.

Global Copy

The **Global** icon lets you copy the selected items by entering in increments in any or all of the global directions. The selected items will copy the increment distance(s) that you have entered.



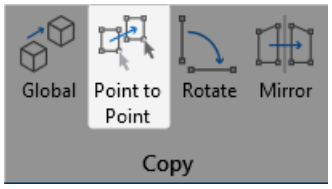


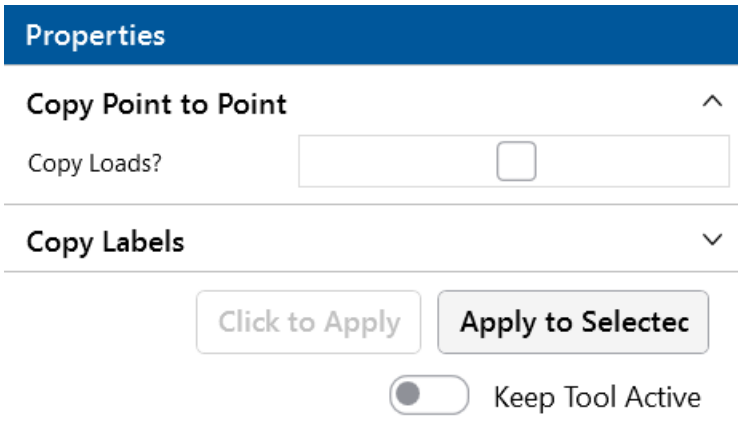
Select the **Diagonal?** check box to combine the orthogonal increments into a single resultant increment vector to copy the elements in a direction other than the three global orthogonal directions.

Checking the **Copy Loads?** box causes ALL loads associated with the original selected model elements to be copied to the corresponding newly created model elements.

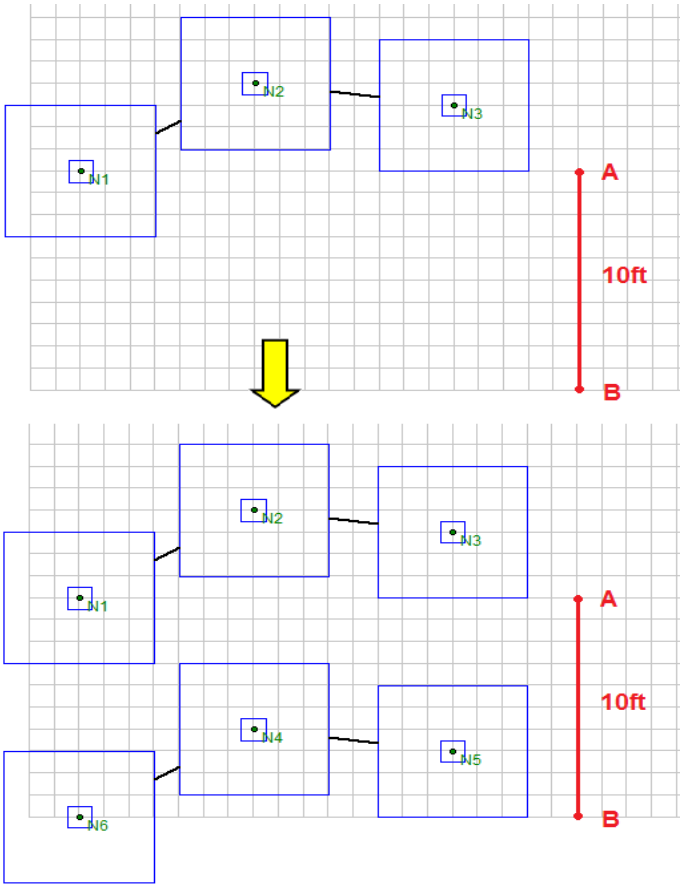
Point to Point Copy

The **Point To Point** icon lets you copy the selected items by clicking on any two points. The selected items will copy the distance between the two points in the direction of the first to second click.



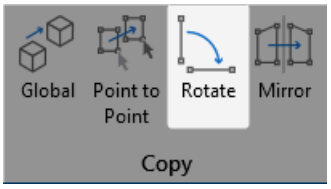


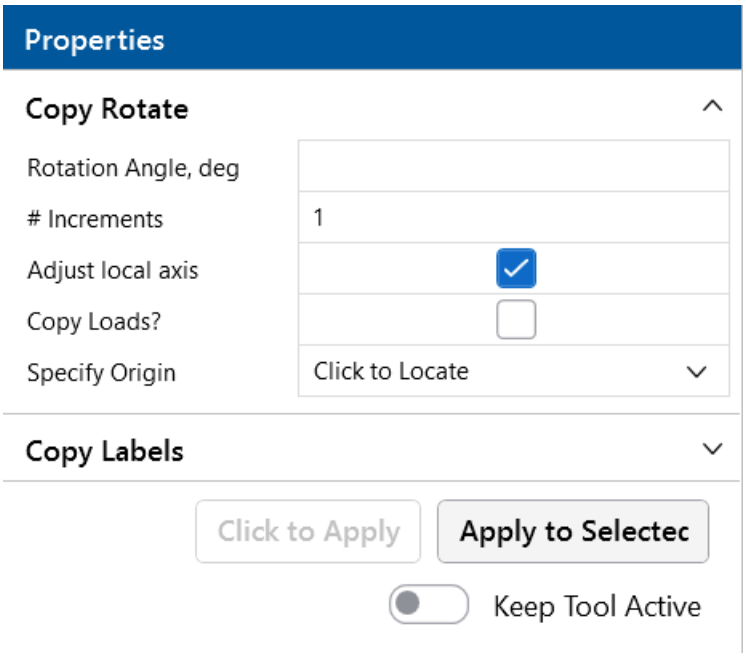
For example, if you click two points: A then B in the model below, the selected items will copy 10 feet as shown below.



Rotate Copy

The **Rotate** icon lets you copy the selected part of the model by rotating the copies about an axis. Simply enter the rotation angle, the number of increments along the rotation angle and the location of the polar origin (the point rotated about).





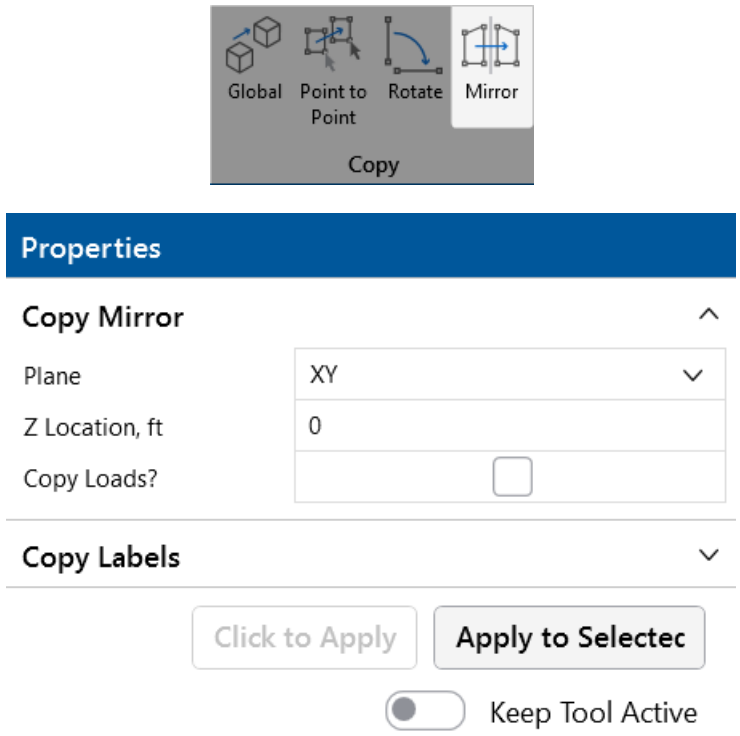
For example, suppose you wish to copy the selected part of your model in six 30° increments (for a total of 180°). You would enter "180" as the rotation angle and "6" as the number of increments. For the polar origin, enter the X and Z coordinates for the origin where you want the rotation to pass through the XZ plane.

Checking the **Adjust Local Axis** box causes the local axis of each element to be rotated with the newly created model elements. Otherwise, the member local axes will retain their original orientation with respect to the global axes.

Checking the **Copy Loads?** box causes ALL loads associated with the original selected model elements to be copied to the corresponding newly created model elements.

Mirror Copy

The **Mirror** icon lets you mirror selected parts of your model.



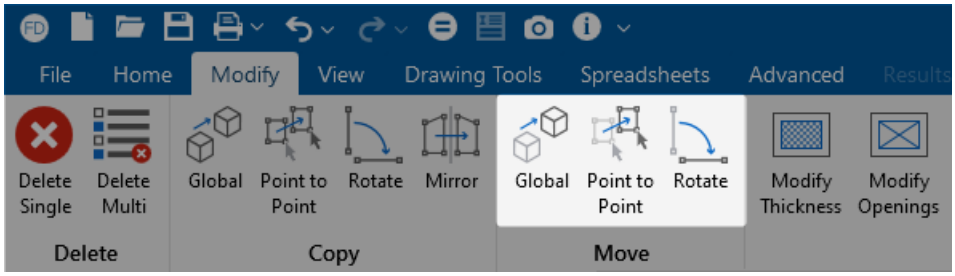
Enter the global plane that you want to mirror about and enter a location along the normal axis to move the mirror plane location. If the mirror plane location is left blank, the mirror plane is placed at the origin.

For example, suppose you want to mirror part of your model about the XY plane at a location +3.0 feet from the origin along the Z-axis. You would enter "XY" as the Mirror Plane and enter a "3.0" for the mirror plane location. (If the mirror plane is XY, the normal axis is the Z-axis).

Checking the **Copy Loads?** box causes ALL loads associated with the original selected model elements to be copied to the corresponding newly created model elements.

Move / Rotate Model Elements

The model elements move options can be found on the **Modify** ribbon in the **Move** section.



Move or Rotate Selected Items

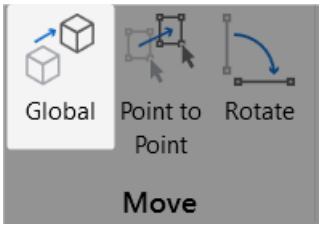
To move or rotate selected items:

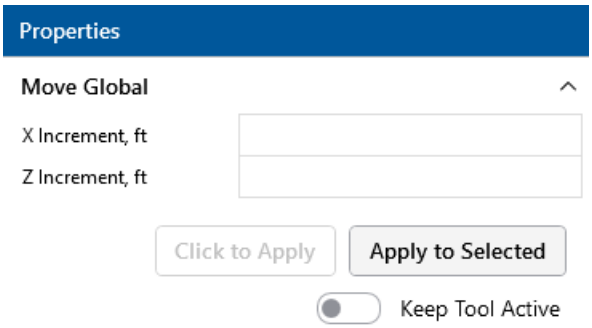
1. Open a model view, if one is not already open.
    a. Go to the **View** ribbon.
    b. Click the **Open 3D Views** icon.
2. Select the items you want to move by clicking on them or clicking and dragging over them.
3. To *linearly* move the selection:
    a. Go to the **Modify** ribbon.
    b. In the **Move** section, choose one of the following options:
        - **Global** to move the selected part of the model a specified distance in the Global Axis directions. For more information, see Global Move.
        - **Point to Point** to move the selection based upon two selected points for direction and distance. For more information, see Point to Point Move.
        - **Rotate** to move the selection in a polar fashion. For more information, see Rotate Move.

Note: You can undo any mistakes by clicking the **Undo** button in the Quick Access toolbar.

Global Move

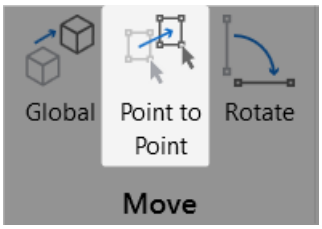
Click the **Global** icon to move the selected part of the model a specified distance in the Global Axis directions. Enter the desired translation distances into the global axes direction input fields.

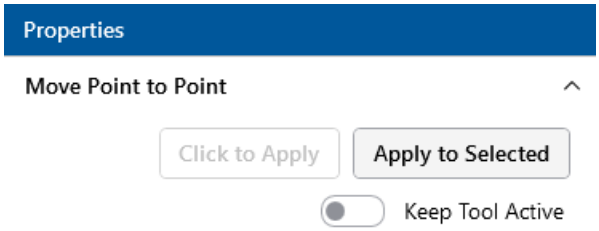




Point to Point Move

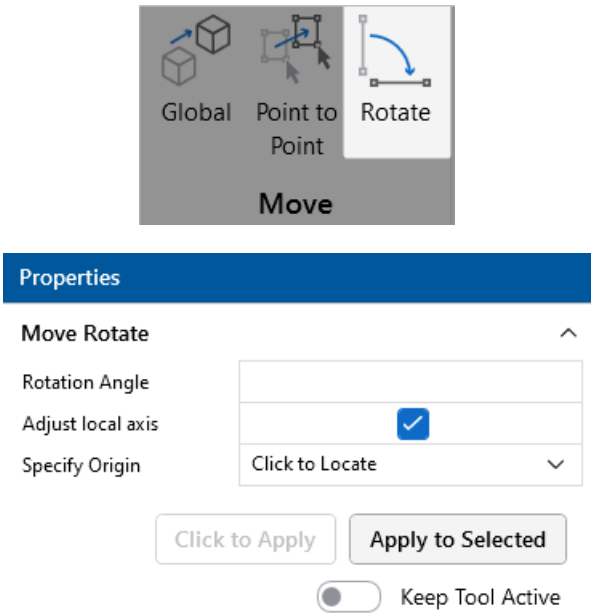
Click the **Point to Point** icon to move elements in the model by clicking on any two points. The selected items will move the distance and direction based upon the vector drawn from your first to your second points.





Rotate Move

Click the **Rotate** icon to rotate the selected parts of the model about an axis. Enter the desired rotation angle, and the coordinates of the polar origin (the point rotated about). Only two of the three polar coordinate values are used, the value corresponding to the axis of rotation is ignored. Instead of specifying coordinates for a polar origin, it may instead be chosen by clicking in the model view. To do this, select **Click to Locate**, specify the appropriate **Rotation Angle**, then click **Click to Apply** or **Apply to Selected**. The cursor will show a finger pointer, with which a polar origin may be specified by left-clicking on a node or snap point.



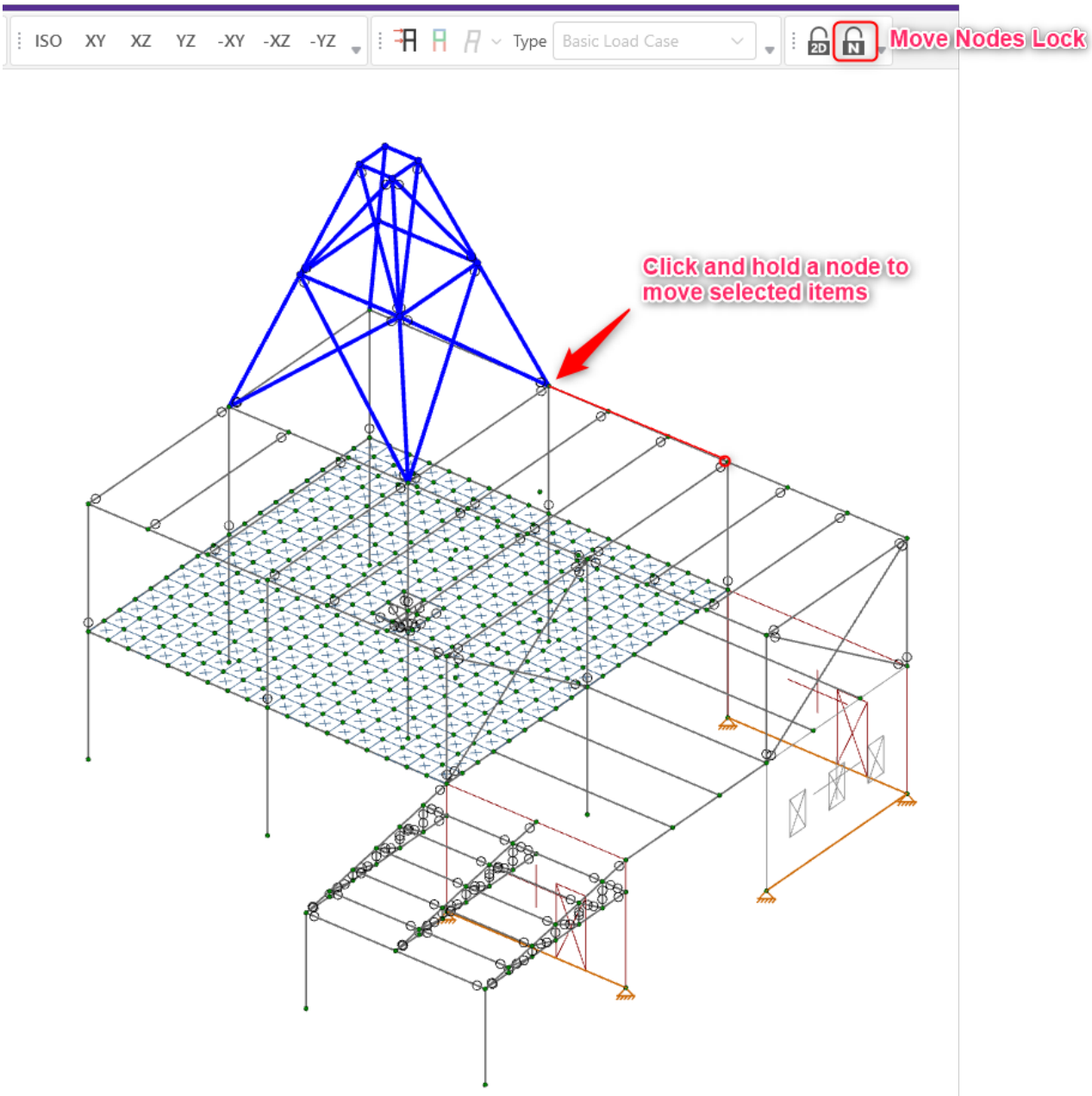
For example, say you wish to rotate the selected part of your model 90° about the Global Y-axis. You would enter "90" as the rotation angle. Under specify origin, select **Coordinates** or **Click to Locate**. If **Coordinates**is selected, enter the X and Z coordinates of where you want the Y-axis rotation to pass through the XZ plane. If **Click to Locate** is selected, press **Apply to Selected**, then click a location in the 3D view.

Checking the **Adjust Local Axis** box will cause the local axis of each member to be rotated along with the model elements. Otherwise, the member local axes will retain their original orientation with respect to the global axes.

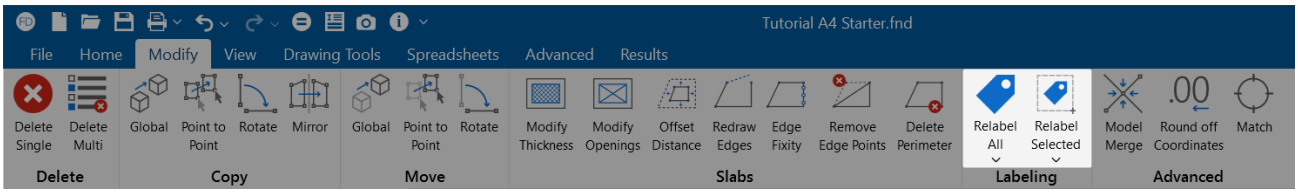
Drag and Move in 3D View

You can quickly move selected elements using the drag-and-move feature. To turn this feature on or off, click the "Move Nodes Lock" button on the ribbon toolbar. Unlocking this will enable the drag-and-move feature.

To use this feature, select the elements you wish to move, then click and hold on a reference node (similar to a Point-to-Point move). A red line will appear in the 3D view, indicating that you are in drag-and-move mode. Next, hover over the destination node and release; the selected elements will move to the new location. If you want to move the elements in the Global X, Y, or Z directions, you can hold the Shift key while performing the drag-and-move actions.



Re-Label Selection Items



You can use the **Relabel** icons on the **Modify** ribbon to re-label only selected items, or all items. This is useful when you want all members at a certain elevation to have a prefix which denotes the floor on which they are located.

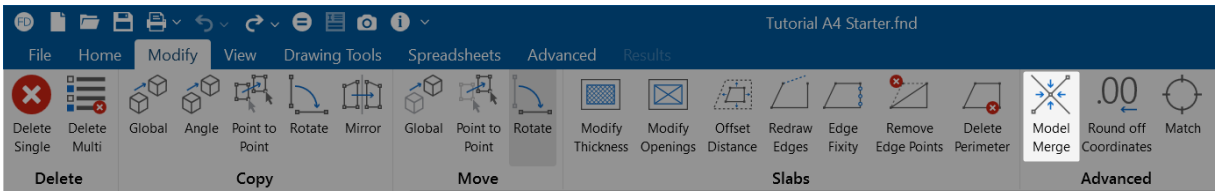
Merge Model Elements

As you build your model, you may find that you need to perform a model merge from time to time. In fact, if you count on doing this, you can generally build your models faster and let the model merge feature do a lot of the work for you. See Merge Tolerance to learn about inputting the merge tolerance.

Perform a Model Merge

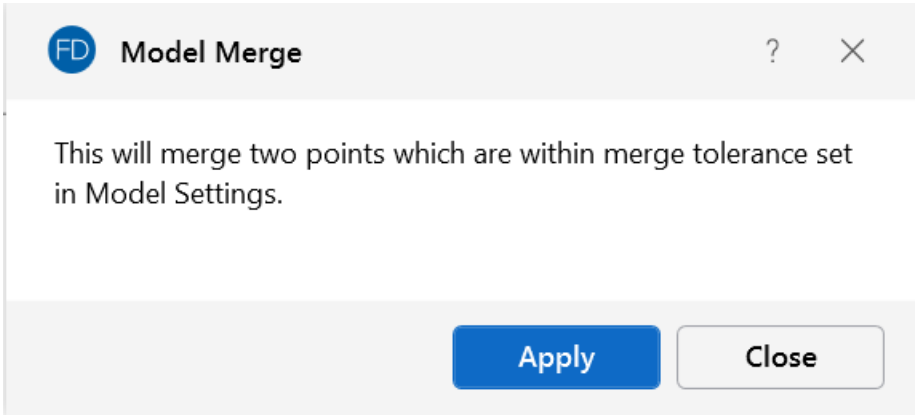
To perform a model merge:

1. Open a model view if one is not already open.
   a. Go to the **View** ribbon.
   b. Click the **Open 3D Views** icon.
2. Go to the **Modify** ribbon.
3. Click the **Model Merge** icon on the **Modify** ribbon in the Advanced section.



The Model Merge window opens.

4. Select **Apply**.

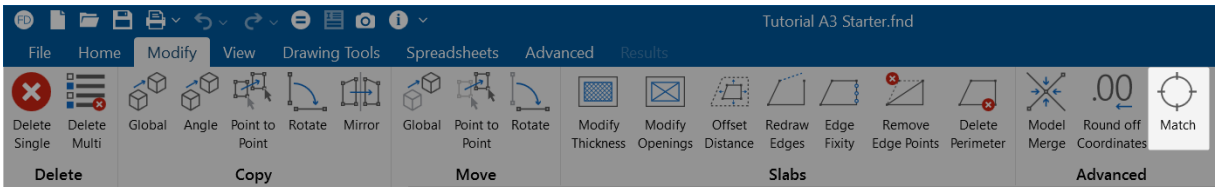


Match

Use the **Match** tool to match the properties of one element to another element.

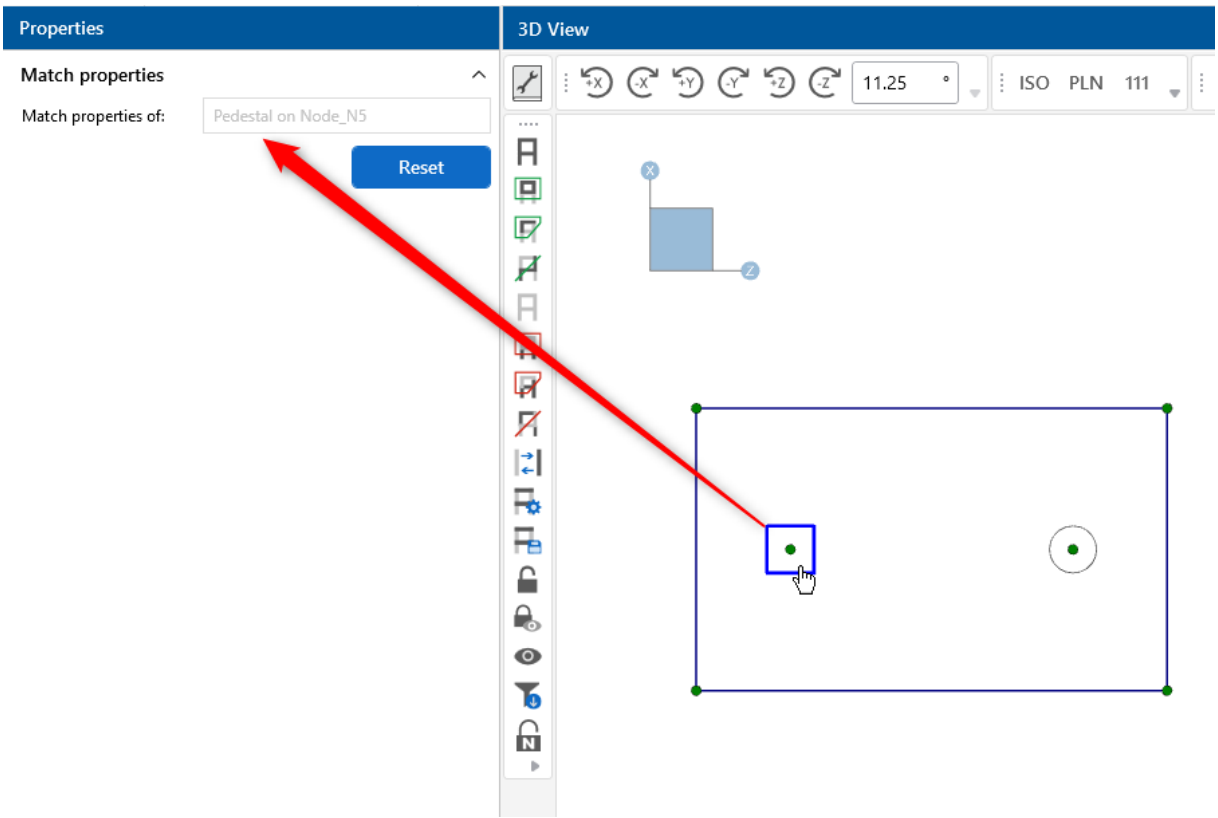
To use the Match tool:

1. Go to the **Modify** ribbon.
2. Click on the **Match** icon in the **Advanced** section.



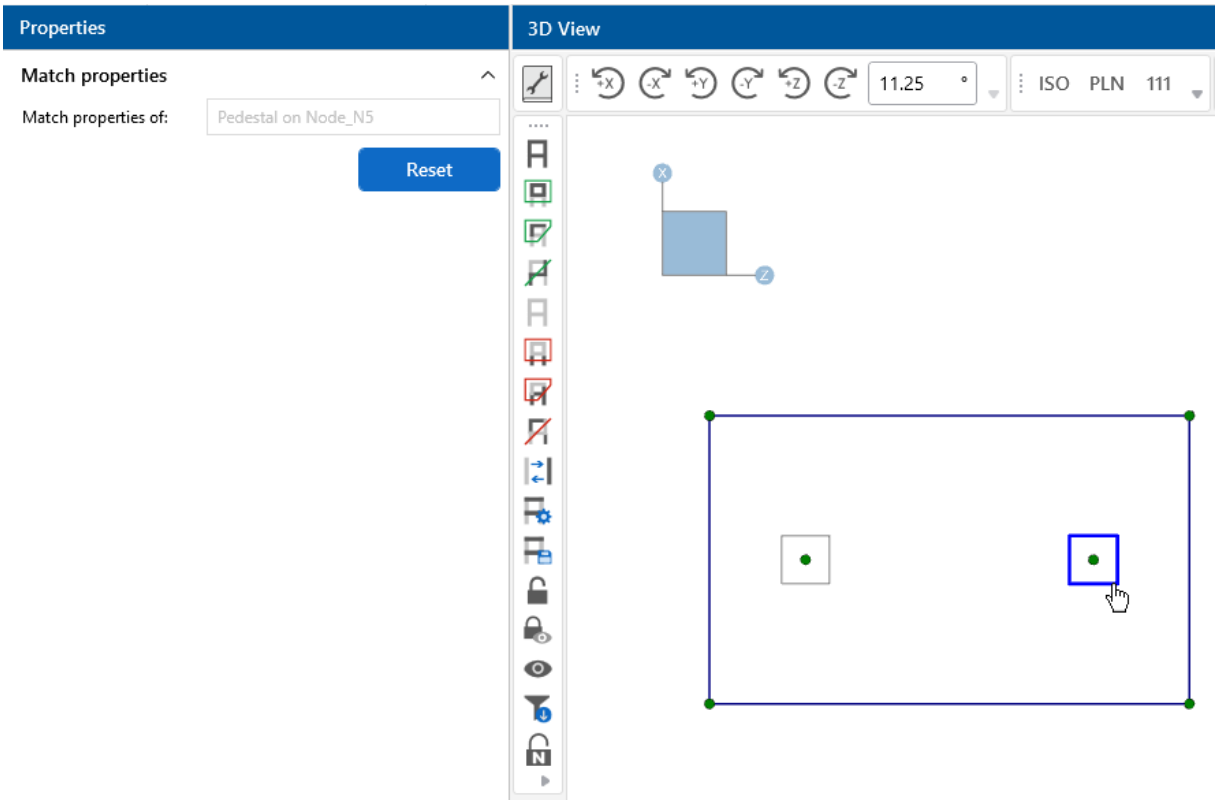
3. Select the target element you would like to copy the properties of.

   The label of this element appears in your properties panel in the **Match properties of:** section.



4. Choose the elements that you would like to match the properties to.

These elements will now share all the properties of the element that was selected in step 3.

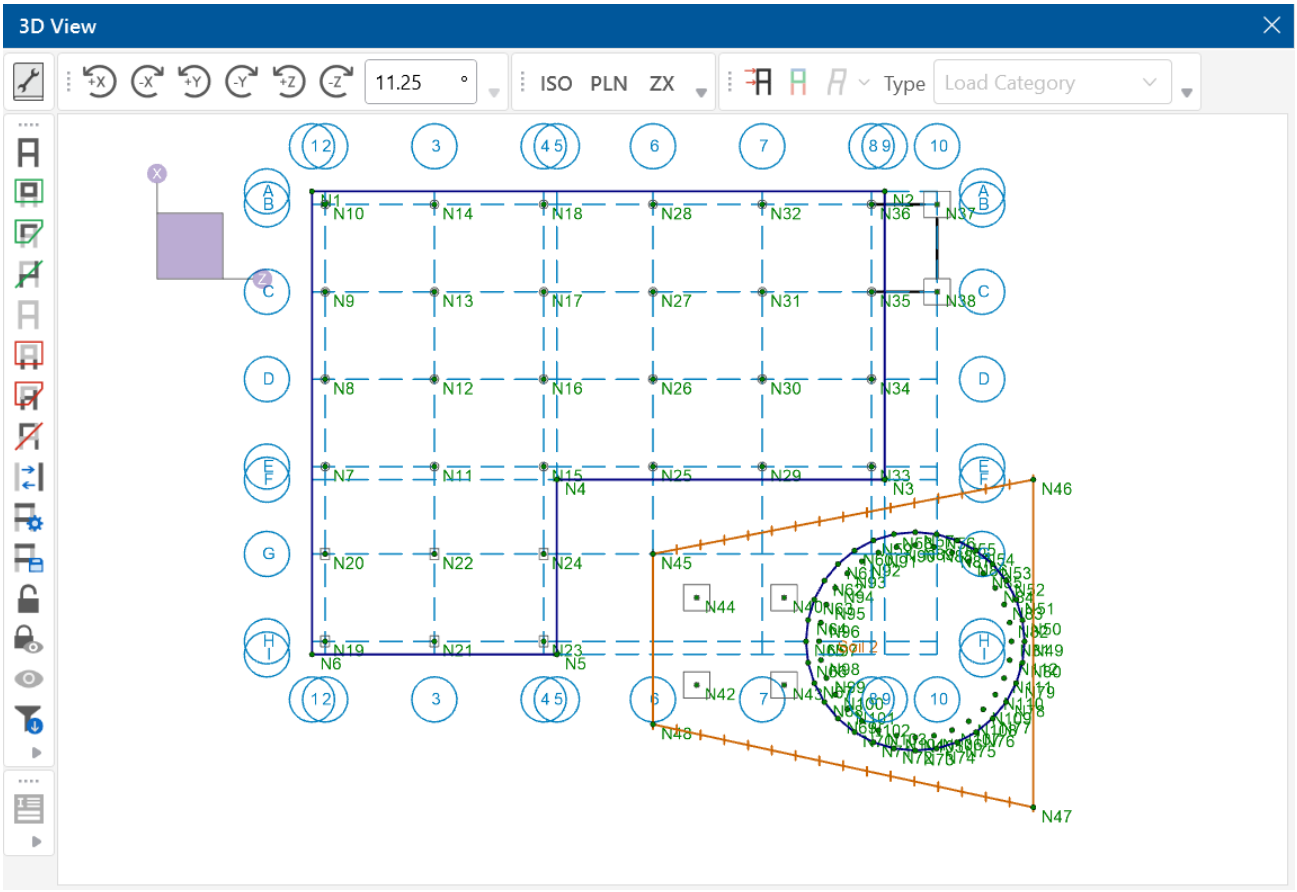


Note:

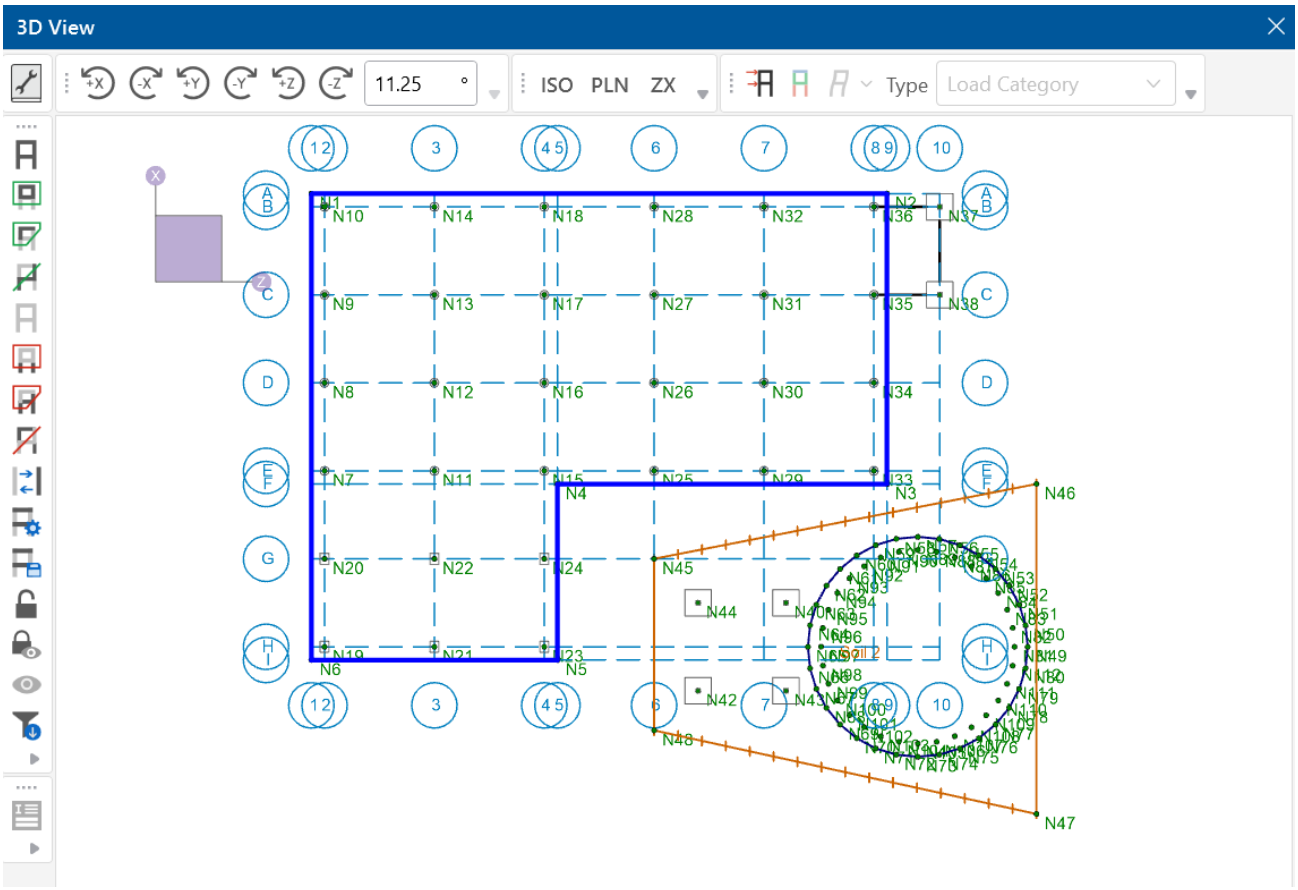
- Subsequent clicks after the first click will continue matching elements to the first member.
- If you would like to create a new target member to match the properties of, click the **Reset** button in the **Properties** panel. This makes the next element you select become the new target element.
- Currently, the program matches all of the properties of the target element. There is no way to match partial properties at this time.

Graphic Selection

You can graphically select items in order to view or modify part of the model, or to print part of the results. When used in conjunction with the graphic editing features, the selection features allow you to quickly model and make modifications. When used with results, it allows you to view and print only the input or results that you want.



The elements that you see in the model views have two possible states; selected and unselected. By default, all items are unselected, and therefore fully displayed with no selection highlight. If you select any items, they will be highlighted in blue. To select or unselect an item, simply click on it with the left mouse button. You may also use one of the other selection tools discussed below to select multiple items.



Note:

- As an alternative to using the selection tools, some operations offer a **Click to Apply** option, which lets you modify items by clicking or boxing them with the mouse.
- If you choose not to display items by turning them off in the **Model View Settings** dialog, they are removed from the 3D View and will not be able to be selected/unselected until you turn the display back on.

Selection Shortcuts

- You can *add elements to a selection state* by pressing **Ctrl** on your keyboard and clicking with the left mouse button.
- When pressing **Ctrl+Left Mouse Click**, a plus sign appears adjacent to the mouse cursor . You can now select a single element to add to the selection state, or you can select multiple elements by boxing them with the mouse.

- Similarly, you can *remove elements from a selection state* by pressing **Shift** on your keyboard and clicking with the left mouse button.
- When pressing **Shift+Left Mouse Click**, a minus sign will appear adjacent to the mouse cursor  . You can now deselect a single element from the existing selection state, or you can deselect multiple elements by boxing them with the mouse.

Cycle Selection

- To cycle through elements that are near the cursor, press the **Crtl** key while **Scrolling the mouse wheel**, either forward or backward. This can be useful if you have a model with elements blocking others that you are interested in selecting and do not want to rotate the model to a new position.

Selection Modes

The **Selection Toolbar** is the vertical toolbar located on the left side of the screen (as shown in the following image). This toolbar lets you select nodes, beams, footings, slabs etc. in model views, and is only active when a model view is active.



Some of the tools are for one-time applications, such as **Select All**. Other tools, such as **Box Select**, place you in a selection mode that remains active until you cancel it. The current mode is indicated by the mouse pointer and by the state of the selection button. While in a selection mode, the button stays depressed until you click it again, choose another button, or press the **ESC** key. You can have more than one model view open and be in different modes in each view.

Note:

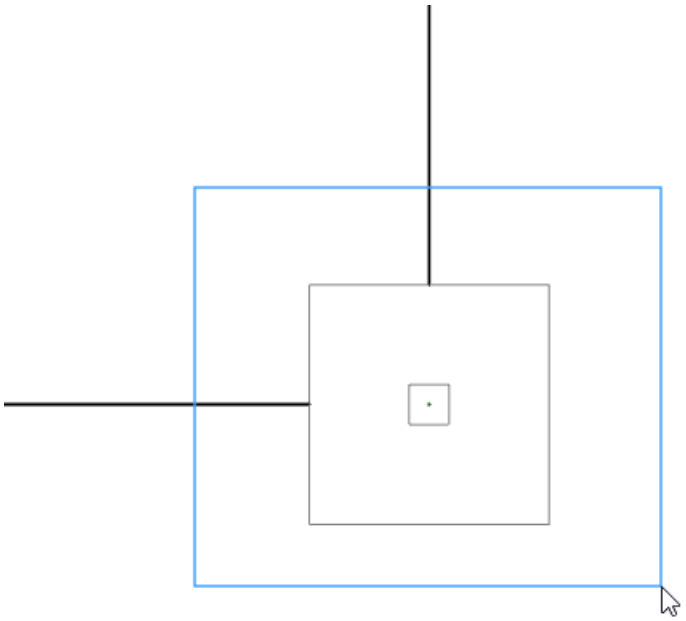
- There are other types of graphic modes, such as **editing** and **viewing** (zooming) modes. The viewing mode overrides the selection mode, which overrides the editing mode. This lets you alter the selected state while you are editing the model. You are returned to the current editing mode when you terminate a selection mode.
- To cancel a selection mode press the **ESC** key or click the right mouse button.

Select All and Unselect All

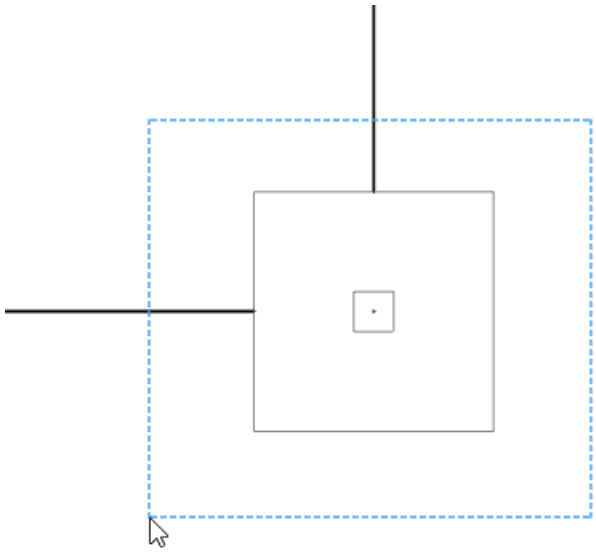
The **Select All** and **Unselect All** tools let you select, or unselect, all of the active nodes, beams, footings and slabs in the model.

Box Select and Unselect Modes

The **Box Select** and **Box Unselect** tools let you draw a box around the items that you wish to select or unselect. When selecting members by boxing, dragging your mouse from left to right will select members completely encompassed within the selection. This selection style is denoted by the solid blue line type of the selection box.



Alternatively, dragging your mouse from right to left will select all elements that you box through. This selection style is denoted with a dashed blue line type.



Polygon Select and Unselect

The **Polygon Select** and **Polygon Unselect** tools let you draw a polygon around the items that you wish to select or unselect. Beams and slabs must be entirely within the polygon in order for them to be affected.

Line Select and Unselect

The **Line Select** and **Line Unselect** tools let you draw a line through the items that you wish to select or unselect. Any element the line crosses will be affected. This is useful when choosing items between other items.

Invert Selections

The **Invert Selection** button is used to invert the selected state of the model. When clicked, all selected items are unselected and all previously unselected items are selected. For example, this can be very useful when many items are to be selected in a large model. Simply click on the undesired items to select them, then click the **Invert Selection** button to make them unselected and everything else selected.

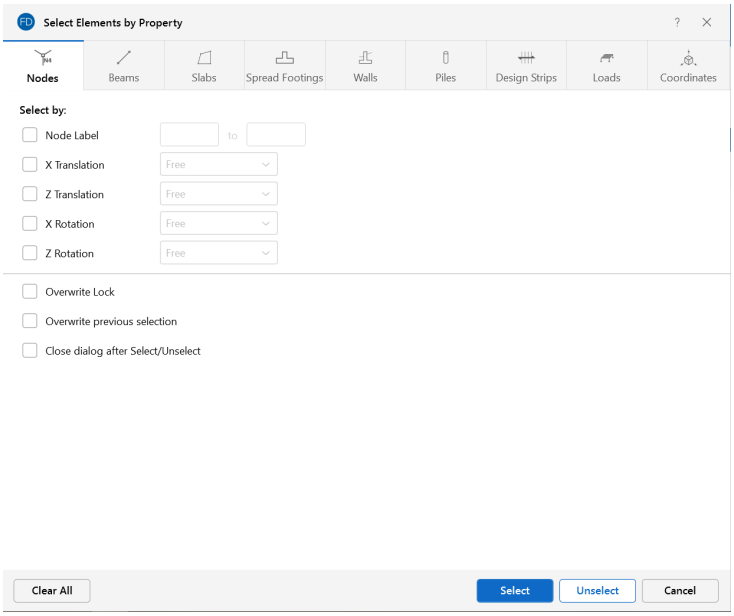
Criteria Selections

The **Criteria Selection** button lets you select items based on a wide range of criteria such as coordinates, labels and many other conditions. When the **Criteria Select** button is clicked, you will be

presented with the Criteria Selection Dialog with options grouped by tabs across the top. Each tab represents groups of criteria that you may use to refine your selection.

The options are numerous, making it easy to quickly achieve complicated selections. This is a powerful tool so it is worth taking the time to experiment with the options so that you will know how to use it to your advantage. The various tabs in the window are described in the sections that follow.

Nodes

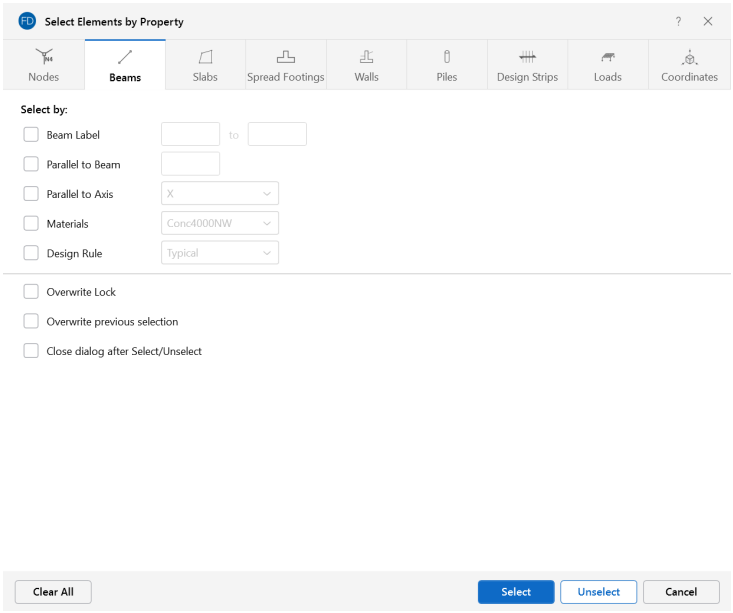


You can **Select** or **Unselect** nodes as follows:

- **Node Label** – Specify a range of node labels. Specifying only one label selects just that one node.
- **Boundary Codes** – Specify boundary condition criteria in all four degrees of freedom.
- **Overwrite Lock** – When checked, it applies the specified criteria to be selected, or unselected, regardless of any part of the model being locked.
- **Overwrite previous selection** – When checked, it overwrites the previous selection shown in the 3D View.

Note: If no criteria are specified, all nodes will be selected/unselected.

Beams

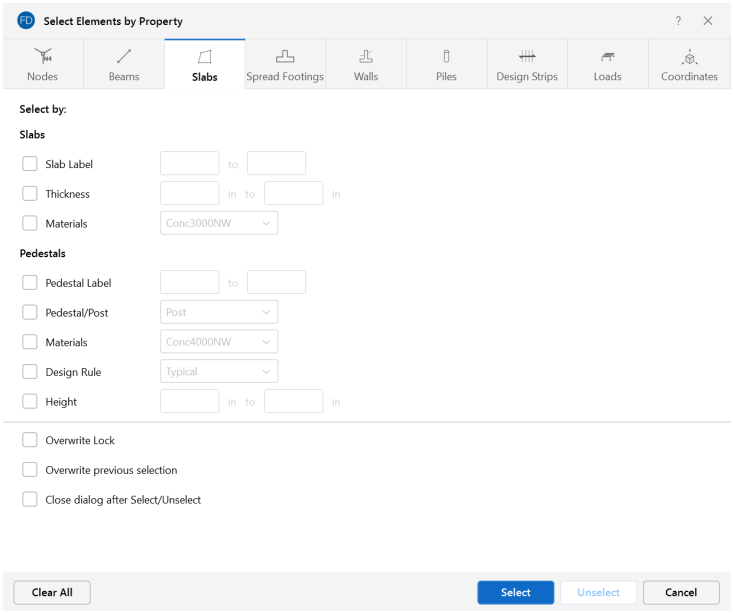


You can **Select** or **Unselect** beams as follows:

- **Beams Label** – Specify a range of beam labels. Specifying only one label selects just that one beam.
- **Parallel to Beam** – Select beams that are parallel to a certain beam, whose label you enter.
- **Parallel to Axis** – Select beams that are parallel to a specified global axis.
- **Materials** – You can select beams based on material selected from a drop-down menu.
- **Design Rule** – You can select beams based on the design rule assigned to the beams.
- **Overwrite Lock** – When checked, it applies the specified criteria to be selected, or unselected, regardless of any part of the model being locked.
- **Overwrite previous selection** – When checked, it overwrites the previous selection shown in the 3D View.

Note: If no criteria are specified, all beams will be selected/unselected.

Slabs/Pedestals

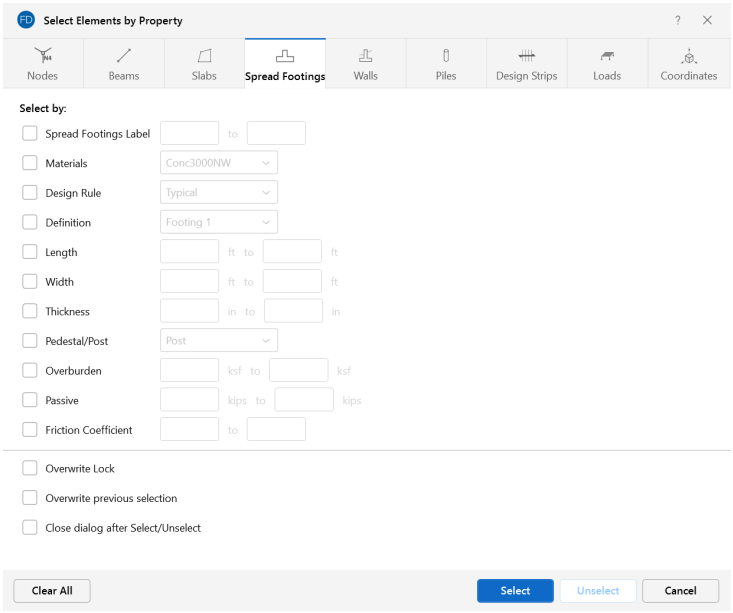


You can **Select** or **Unselect** slabs and pedestals as follows:

- **Slab Label** – Specify a range of slab labels. Specifying only one label selects just that one slab.
- **Thickness** – Specify a range of slab thicknesses to select.
- **Materials** – You can select slabs based on material selected from a drop-down menu.
- **Pedestal Label** – Specify a range of pedestal labels. Specifying only one label selects just that one pedestal.
- **Pedestal/Post** – Allows you to differentiate selections based on pedestal and post model elements.
- **Materials** – You can select pedestals based on material selected from a drop-down menu.
- **Design Rule** – You can select pedestals based on design rules.
- **Height** – You can select pedestals based on a range of heights.
- **Overwrite Lock** – When checked, it applies the specified criteria to be selected, or unselected, regardless of any part of the model being locked.
- **Overwrite previous selection** – When checked, it overwrites the previous selection shown in the 3D View.

Note: If no criteria are specified, all slabs/pedestals will be selected/unselected.

Spread Footings



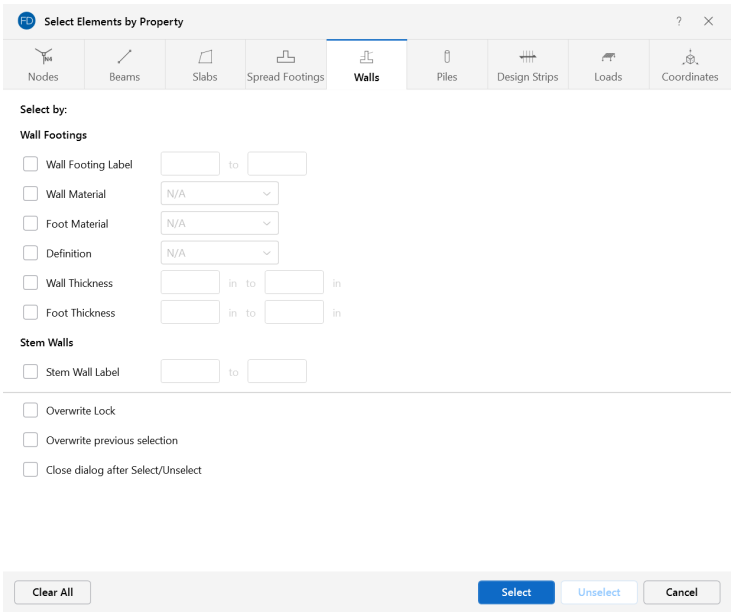
You can **Select** or **Unselect** spread footings as follows:

- **Spread Footings Label** – Specify a range of spread footing labels. Specifying only one label selects just that one spread footing.
- **Materials** – You can select spread footings based on material selected from a drop-down menu.
- **Design Rule** – You can select spread footings based on design rules.
- **Definition** – You can select spread footings based on individual footing definitions.
- **Length** – You can select spread footings based on a range of lengths.
- **Width** – You can select spread footings based on a range of widths.
- **Thickness** – Allows you to specify a range of footing thicknesses to select.
- **Pedestal/Post** – Allows you to differentiate selections based on pedestal and post model elements.
- **Overburden** – You can select spread footings based on a range of overburden pressure applied to the footing.
- **Passive** – You can select spread footings based on a range of passive resistance assigned to the footing.
- **Friction Coefficient** – You can select spread footings based on a range of friction coefficients assigned to the footing.

- **Overwrite Lock** – When checked, it applies the specified criteria to be selected, or unselected, regardless of any part of the model being locked.
- **Overwrite previous selection** – When checked, it overwrites the previous selection shown in the 3D View.

Note: If no criteria are specified, all spread footings will be selected/unselected.

Walls



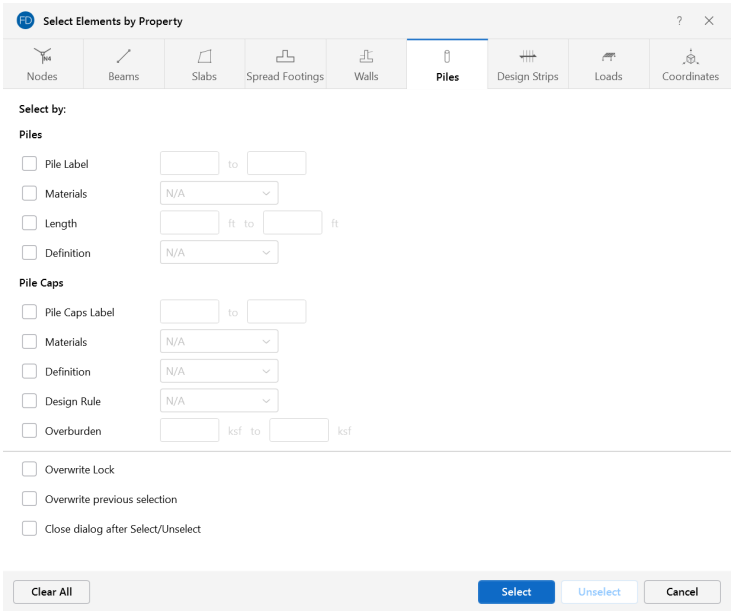
You can **Select** or **Unselect** wall footings/stem walls as follows:

- **Wall Footing Label** – Specify a range of wall footing labels. Specifying only one label selects just that one wall footing.
- **Wall Material** – You can select walls based on material selected from a drop-down menu.
- **Foot Material** – You can select footings based on material from a drop-down menu.
- **Definition** – You can select wall footings based on individual wall footing definitions.
- **Wall/Foot Thickness** – You can specify a range of wall or footing thicknesses to select.
- **Stem Wall Label** – Specify a range of stem wall labels. Specifying only one label selects just that one stem wall.
- **Overwrite Lock** – When checked, it applies the specified criteria to be selected, or unselected, regardless of any part of the model being locked.

- **Overwrite previous selection** – When checked, it overwrites the previous selection shown in the 3D View.

Note: If no criteria are specified, all wall footings/stem walls will be selected/unselected.

Piles



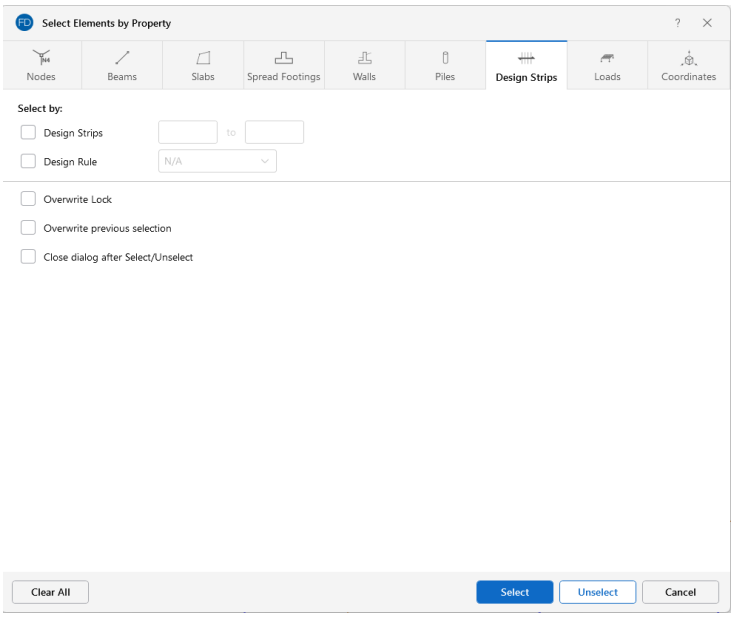
You can **Select** or **Unselect** piles/pile caps as follows:

- **Pile Label** – Specify a range of pile labels. Specifying only one label selects just that one pile.
- **Materials** – You can select piles or pile caps based on material from a drop-down menu.
- **Length** – You can select piles based on a range of lengths.
- **Definition** – You can select piles or pile caps based on individual definitions
- **Pile Caps Label** – Specify a range of pile cap labels. Specifying only one label selects just that one pile cap.
- **Design Rule** – You can select pile caps based on design rules.
- **Overburden** – You can select pile caps based on a range of overburden pressure applied to the pile cap.
- **Overwrite Lock** – When checked, it applies the specified criteria to be selected, or unselected, regardless of any part of the model being locked.

- **Overwrite previous selection** – When checked, it overwrites the previous selection shown in the 3D View.

Note: If no criteria are specified, all piles/pile caps will be selected/unselected.

Design Strips

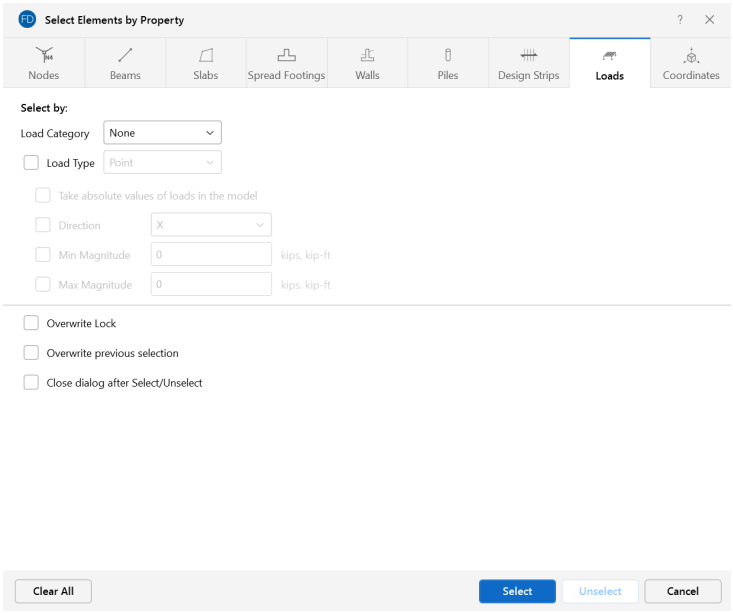


You can **Select** or **Unselect** design strips as follows:

- **Design Strips** – Specify a range of design strips using labels. Specifying only one label selects just that one design strip.
- **Design Rule** – You can select design strips based on design rules specified for the strips.
- **Overwrite Lock** – When checked, it applies the specified criteria to be selected, or unselected, regardless of any part of the model being locked.
- **Overwrite previous selection** – When checked, it overwrites the previous selection shown in the 3D View.

Note: If no criteria are specified, all design strips will be selected/unselected.

Loads

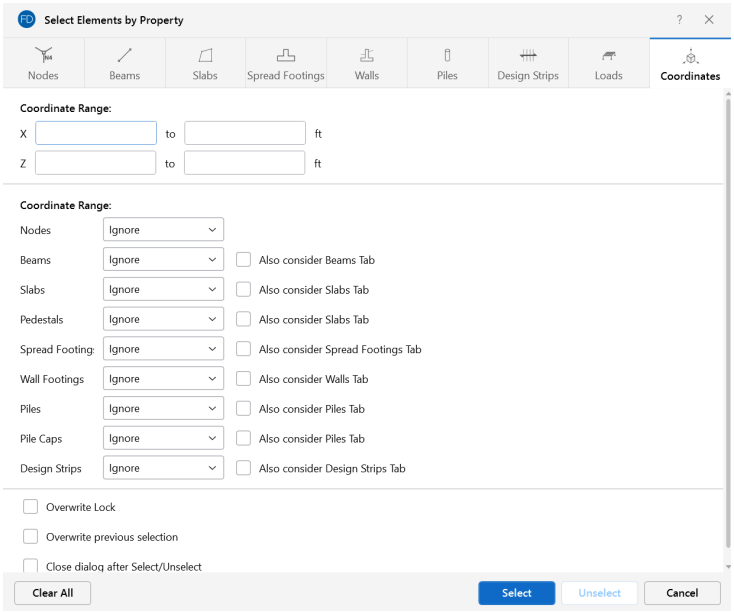


You can **Select** or **Unselect** loads as follows:

- **Load Category** – Select or unselect loads based on a specific load category.
- **Load Type** – Select or unselect loads based on the load type. You may also select a direction or min/max magnitude.
- **Overwrite Lock** – When checked, it applies the specified criteria to be selected or unselected regardless of any part of the model being locked.
- **Overwrite previous selection** – When checked, it overwrites the previous selection shown in the 3D View.

Note: If no criteria are specified, all the loads will be selected/unselected.

Coordinates



You can **Select** or **Unselect** elements based on coordinate criteria and combine this with the criteria on the other tabs as follows:

- **Coordinate Range**– Specify minimum and maximum bounds in the global directions. Items within these bounds AND meeting the criteria in the other groups will be selected/unselected.
- **Nodes** – Specify that nodes in the coordinate range be selected or not.
- **Beams** – Specify that one or both beam end nodes be within the bounds. You may also specify that no beams be selected within the range.
- **Slabs** – Specify that one or all slab corner nodes be within the bounds. You may also specify that no slabs be selected within the range.
- **Spread Footings** – Specify that spread footing nodes be within the bounds. You may also specify that no spread footings be selected within the range.
- **Wall Footings** – Specify that one or both wall footing end nodes be within the bounds. You may also specify that no wall footings be selected within the range.
- **Piles** – Specify that pile nodes be within the bounds. You may also specify that no piles be selected within the range.
- **Pile Caps** – Specify that pile cap nodes be within the bounds. You may also specify that no pile caps be selected within the range.
- **Design Strips** – Specify that one or all corners of design strips be within the bounds. You may also specify that no design strips be selected within the range.

- **Also Consider Beams/Slabs/Spread Footings/Walls/Piles/Design StripsTab** – Lets you include selection criteria from the other tabs and combine it with the coordinate range specified on this page.

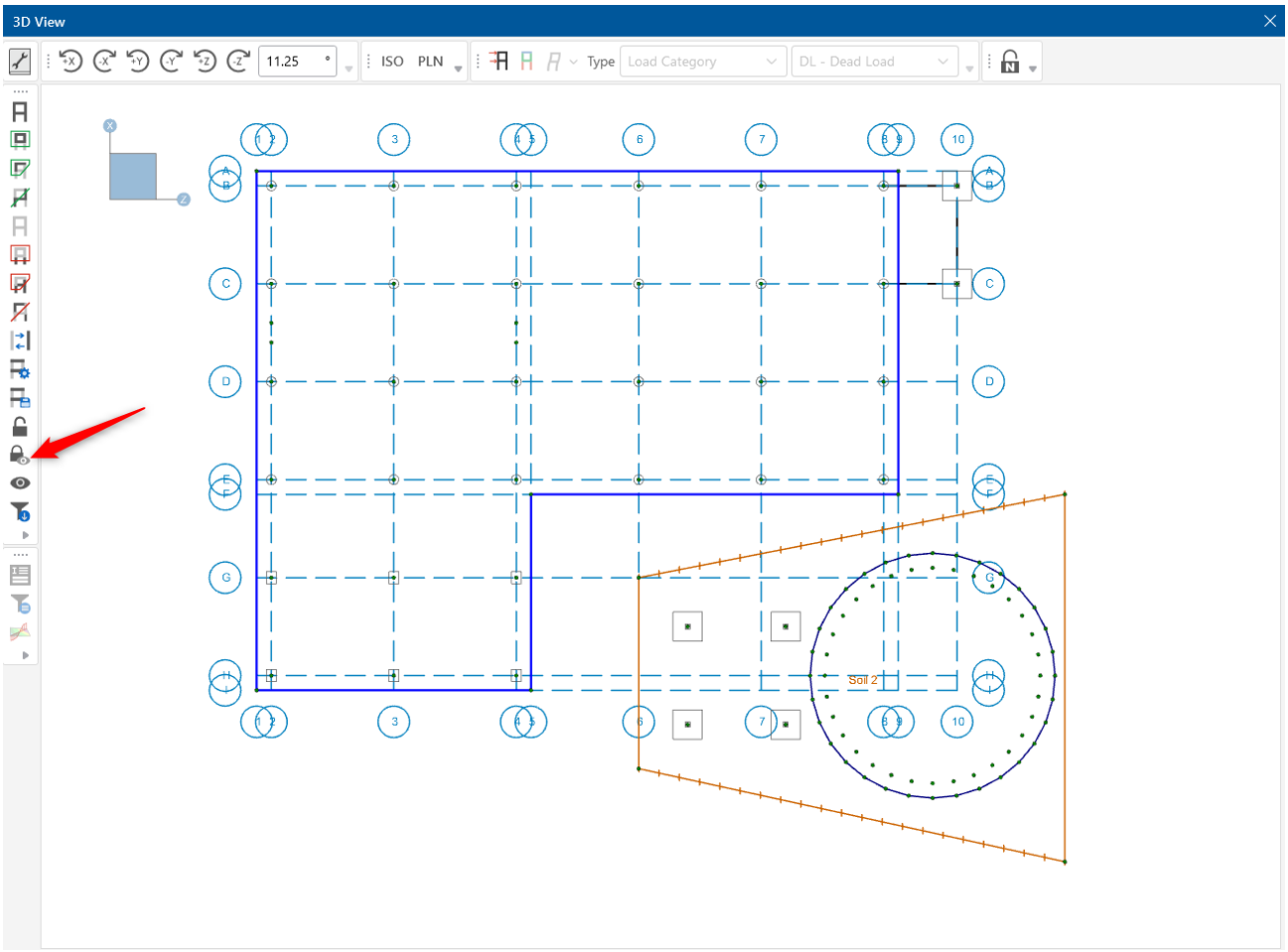
Note: If the **Also Consider ... Tab** check box is checked, all criteria will be applied together so that the affected items will meet all of the criteria. For example, if you specify a range of node labels and coordinates, only the nodes within the coordinate bounds and within the label range will be selected/unselected.

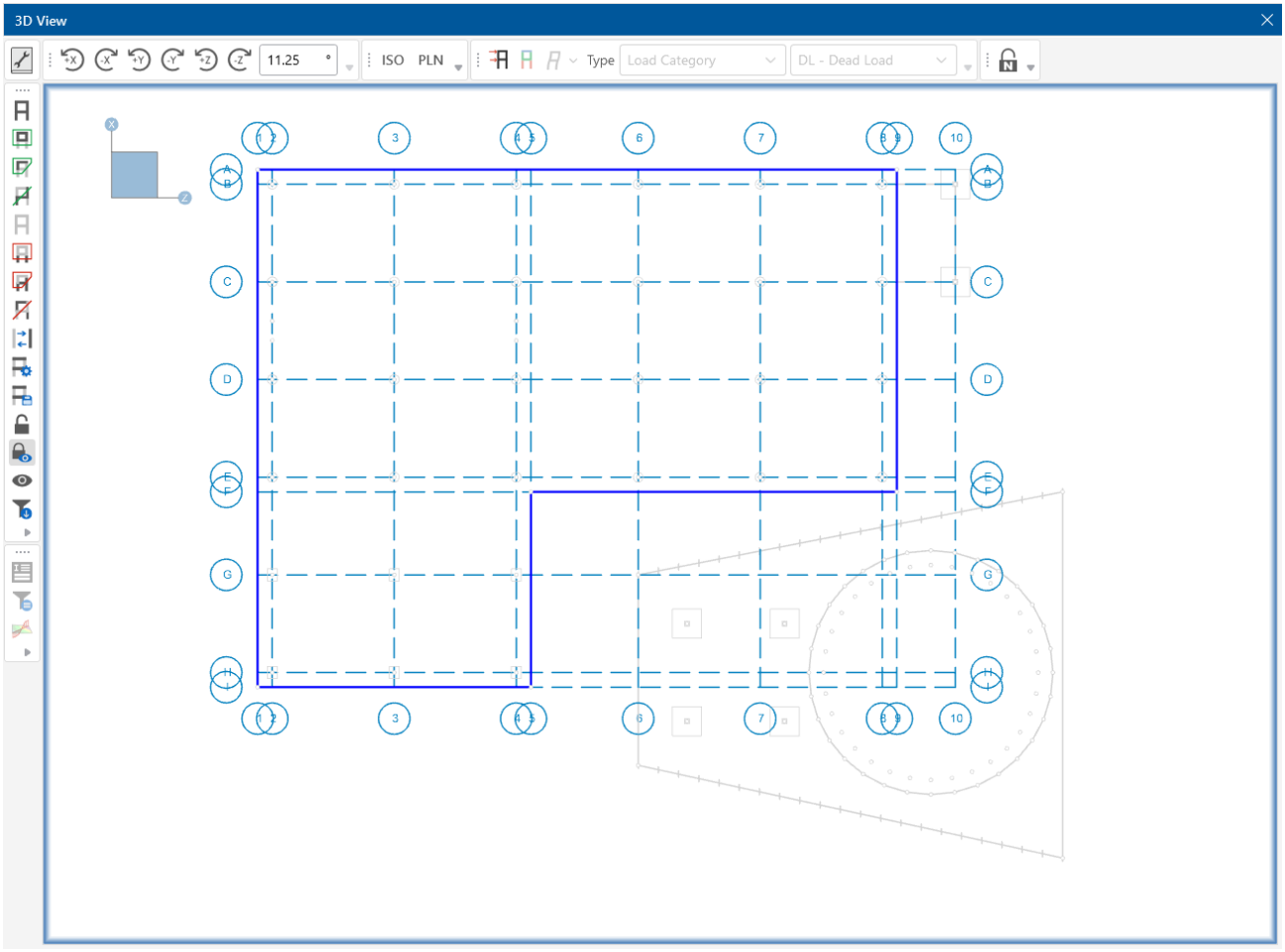
Lock & Dim Lock

Click the **Lock Selected** button to visually remove all unselected elements 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 selected model elements in the model view, click the **Unlock Selected** button and the unselected items that were previously "removed" from the model view will be returned to the view in the unselected state.

Clicking the **Dim Lock** button causes all currently unselected elements to become visually dim or grayed out. While in the dim lock state, you will not be able to click on these dimmed elements, nor will they graphically show any labels or results. This is useful for when you want to keep a portion of your structure for visual reference only but also have it stay out of your way when modeling or reporting graphical results.


3D View
11.25
ISO
PLN
Type
Load Category
DL - Dead Load
Soil 2

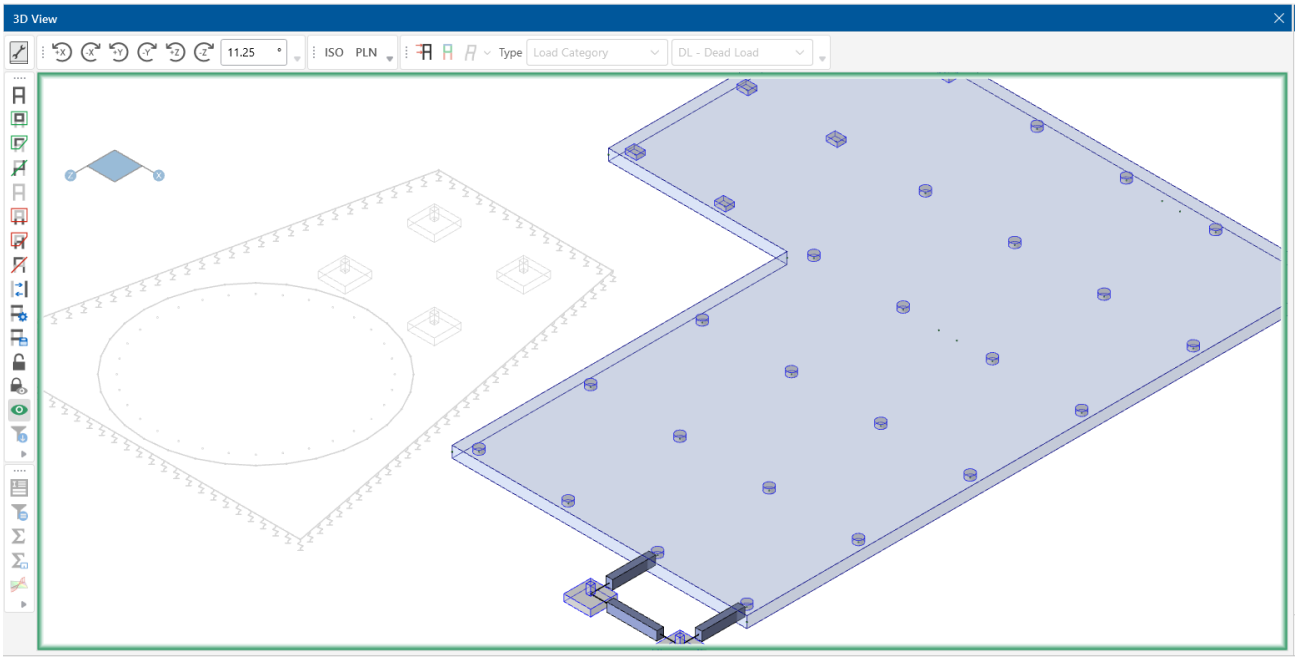


Note: When you are in the dim lock state, you see a blue border around the 3D view panel.

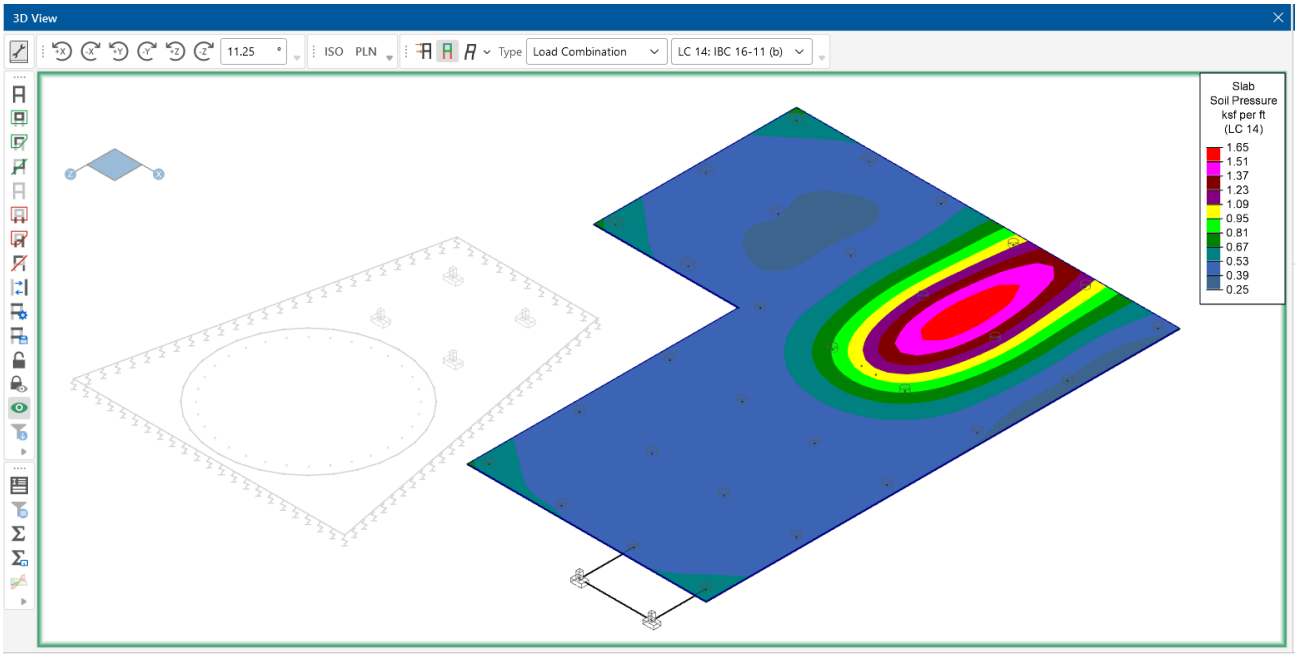
To unlock the **Dim Lock** state, simply click the button and the dimmed elements will return back to their original state.

Viewer Mode

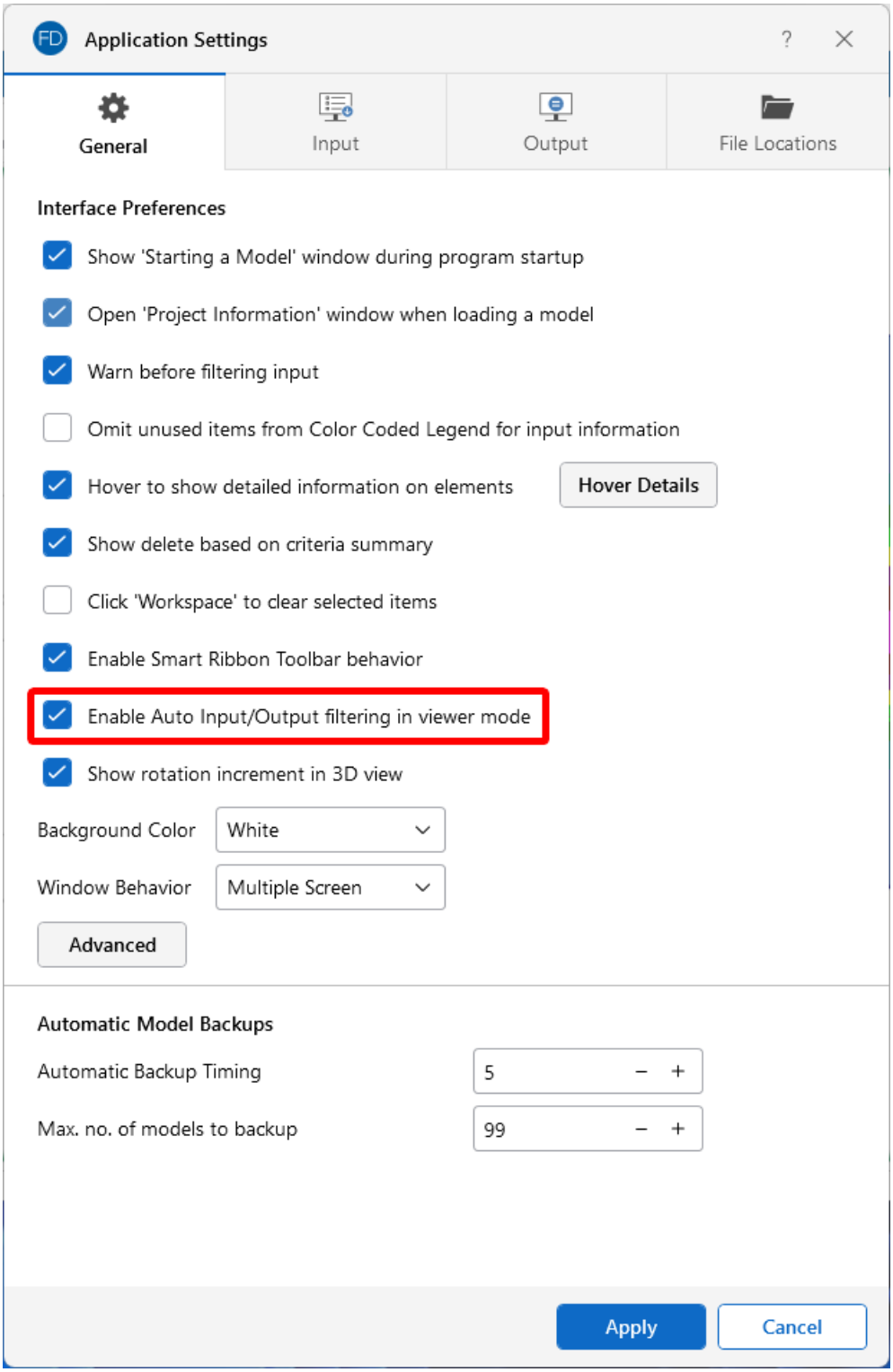
Similar to the 'Dim Lock' feature, the **Viewer Mode** icon causes all currently unselected elements to become grayed out. However, in viewer mode, you can select or unselect elements with a single click.



While in viewer mode, labels, results and other element-specific information is only displayed for elements that are selected. You can quickly click on different elements to make them visible or grayed out.



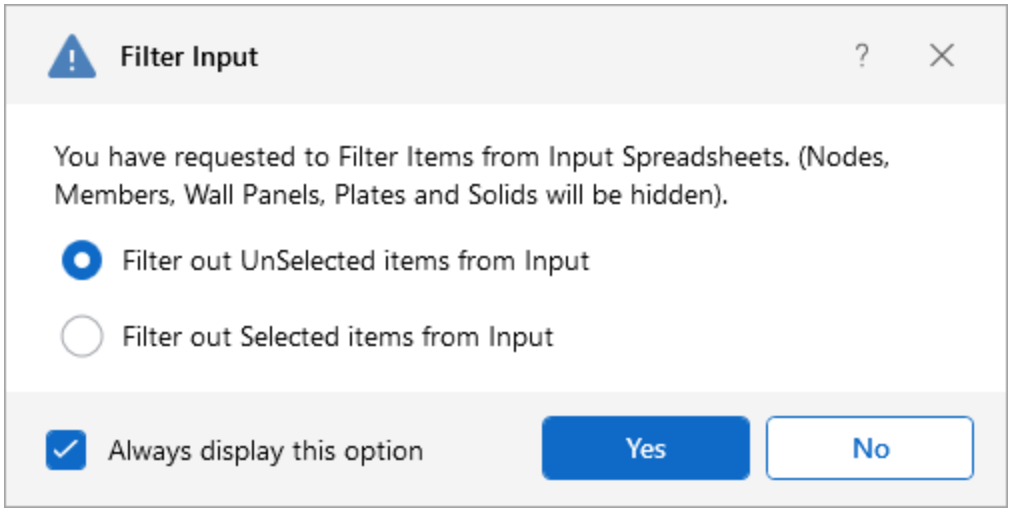
In addition to the graphic display, both the **Input** and **Result Spreadsheets** automatically filter information based on the elements that are selected in viewer mode. This automatic filtering can be turned off on the General tab of the Application Settings.


Application Settings
General
Input
Output
File Locations
Interface Preferences
Show 'Starting a Model' window during program startup
Open 'Project Information' window when loading a model
Warn before filtering input
Omit unused items from Color Coded Legend for input information
Hover to show detailed information on elements
Hover Details
Show delete based on criteria summary
Click 'Workspace' to clear selected items
Enable Smart Ribbon Toolbar behavior
Enable Auto Input/Output filtering in viewer mode
Show rotation increment in 3D view
Background Color
White
Window Behavior
Multiple Screen
Advanced
Automatic Model Backups
Automatic Backup Timing
5
Max. no. of models to backup
99
Apply
Cancel

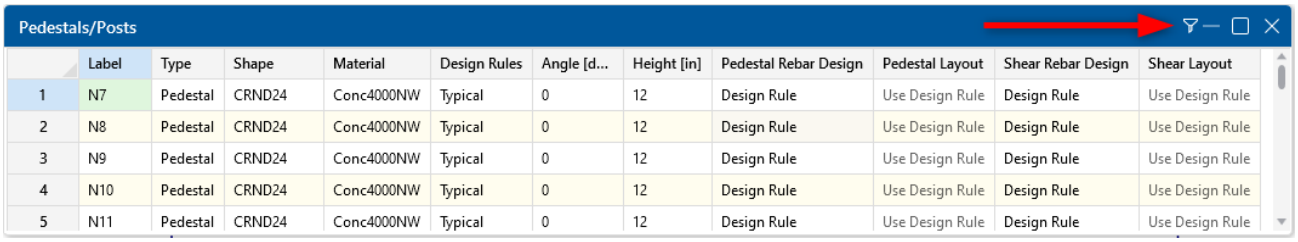
While in viewer mode, a green border appears around the 3D view. It is important to note that the viewer mode is primarily intended for 'viewing' your elements and their associated information. Therefore, it is not possible to model or make edits to your model while in the viewer mode.

Input Filter

To filter the Input Spreadsheets based on your selection, click the **Input Filter** option in the **Selection Toolbar**. You have the option to filter out the UnSelected items or the Selected items from the Input Spreadsheets.



When you open any **Input Spreadsheet**, you can tell if it has been filtered by the filter icon in the spreadsheet header.



	Label	Type	Shape	Material	Design Rules	Angle [d...	Height [in]	Pedestal Rebar Design	Pedestal Layout	Shear Rebar Design	Shear Layout
1	N7	Pedestal	CRND24	Conc4000NW	Typical	0	12	Design Rule	Use Design Rule	Design Rule	Use Design Rule
2	N8	Pedestal	CRND24	Conc4000NW	Typical	0	12	Design Rule	Use Design Rule	Design Rule	Use Design Rule
3	N9	Pedestal	CRND24	Conc4000NW	Typical	0	12	Design Rule	Use Design Rule	Design Rule	Use Design Rule
4	N10	Pedestal	CRND24	Conc4000NW	Typical	0	12	Design Rule	Use Design Rule	Design Rule	Use Design Rule
5	N11	Pedestal	CRND24	Conc4000NW	Typical	0	12	Design Rule	Use Design Rule	Design Rule	Use Design Rule

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 **Show Selected Lines in Current View** or **Hide Selected Lines in Current View** options. This graphically selects or unselects the items corresponding to the selected rows in the top most model view window. This is very useful for highlighting failing members in a spreadsheet and then having them graphically selected in a model view to see where the problems are.

Save Selections

You can save and recall various selection states for a model. If the model is altered after a selection state has been saved, the saved selection state will also be altered. Any new items (nodes, beams…) will be set to "selected" in any selection states saved prior to the creation of the new item.

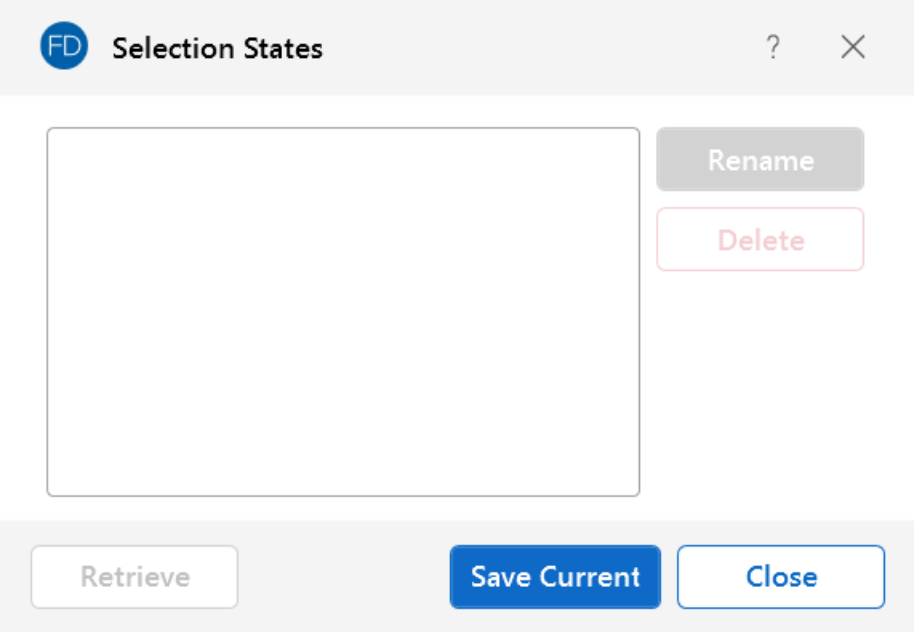
Save a Selection

You can have up to 16 different saved selections in a model.

To save a selection state:

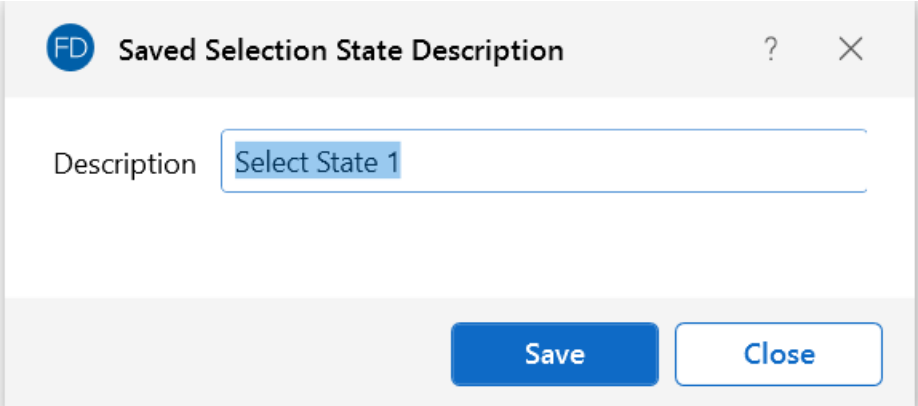
1. Click the **Saved Selections** button on the **Selection Toolbar**.

   The Selection States window opens.



2. Click **Save Current**.

   The Saved Selection State Description window opens.



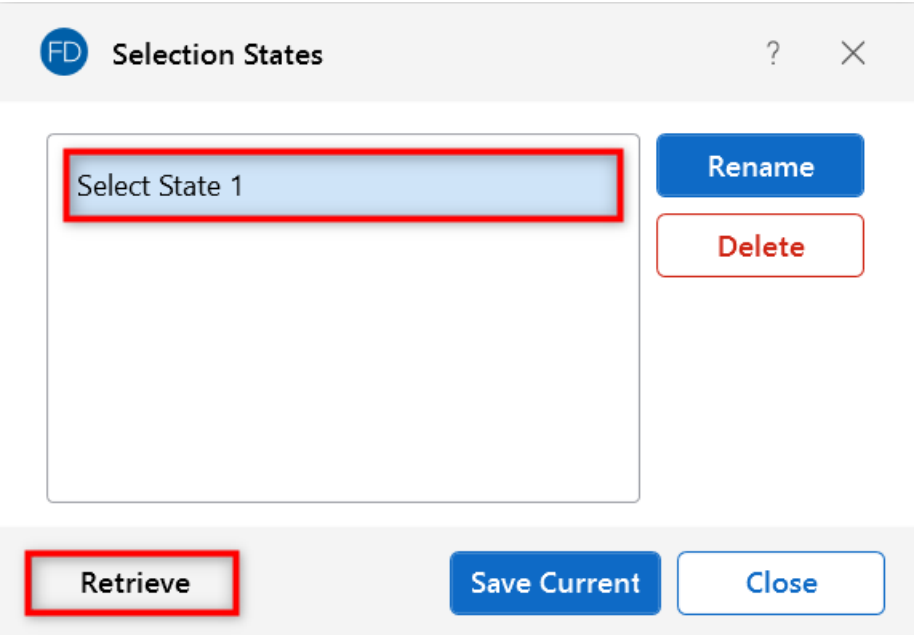
3. Type a **Description** for the selection.
4. Click **Save**.

Retrieve a Saved Selection

To retrieve a saved selection:

1. Click the **Saved Selections** button on the **Selection Toolbar**.

   The Selection States window opens.
2. Choose the selection state from the list on the left.
3. Click the **Retrieve** button



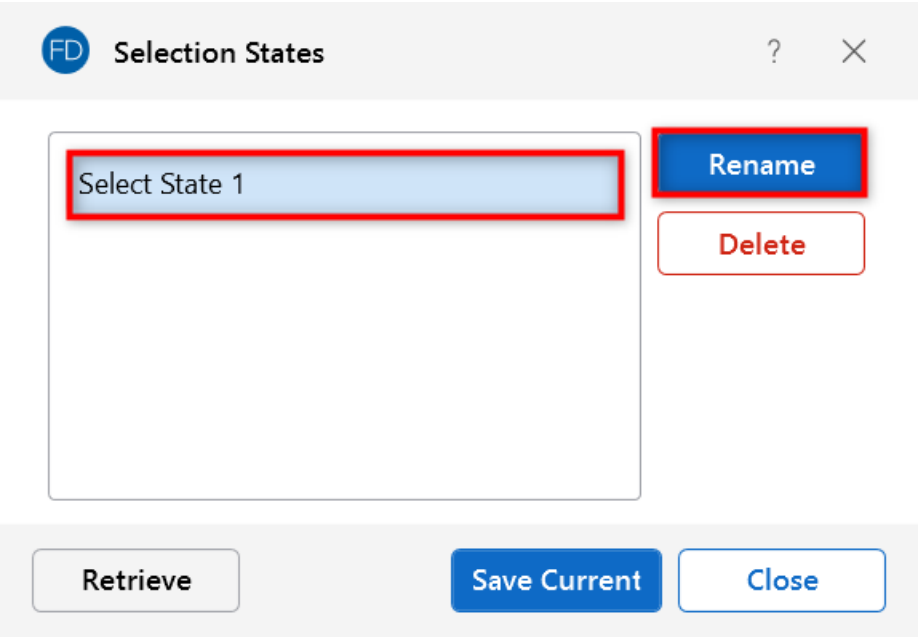
Rename a Saved Selection

To rename a saved selection:

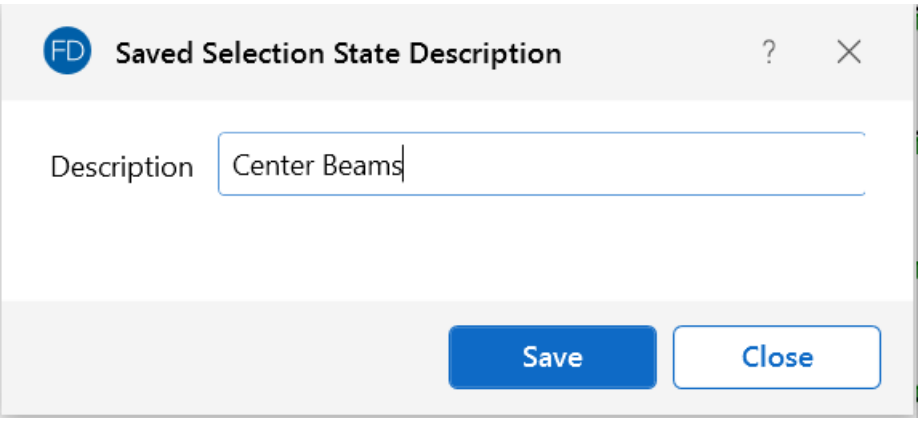
1. Click the **Saved Selections** button on the **Selection Toolbar**.

   The Selection States window opens.

2. Choose the selection state from the list on the left.
3. Click the **Rename** button.



   The Saved Selection State Description window opens.



4. In the **Description** box, type a new name for the selection.
5. Click **Save**.

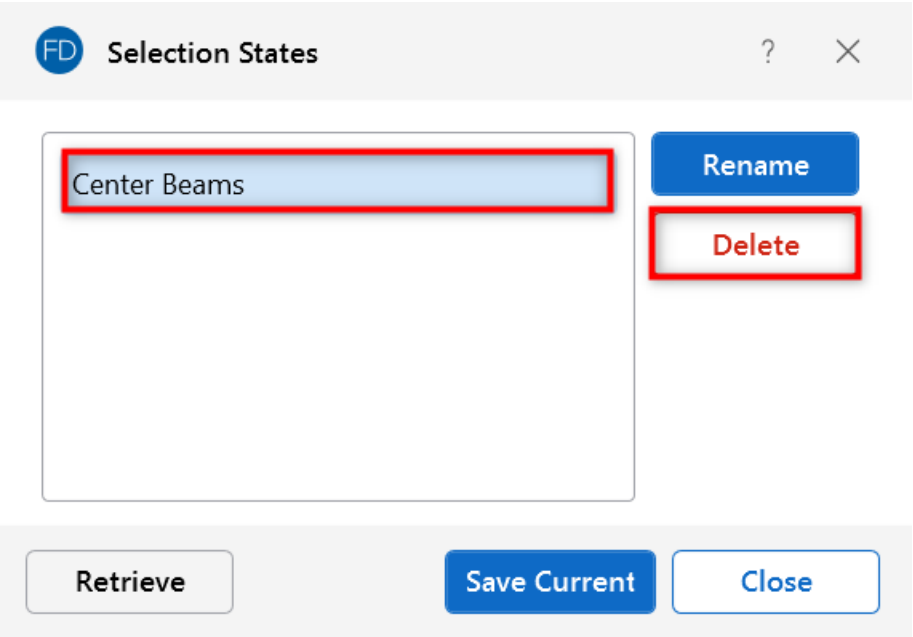
Delete a Saved Selection

To delete a saved selection:

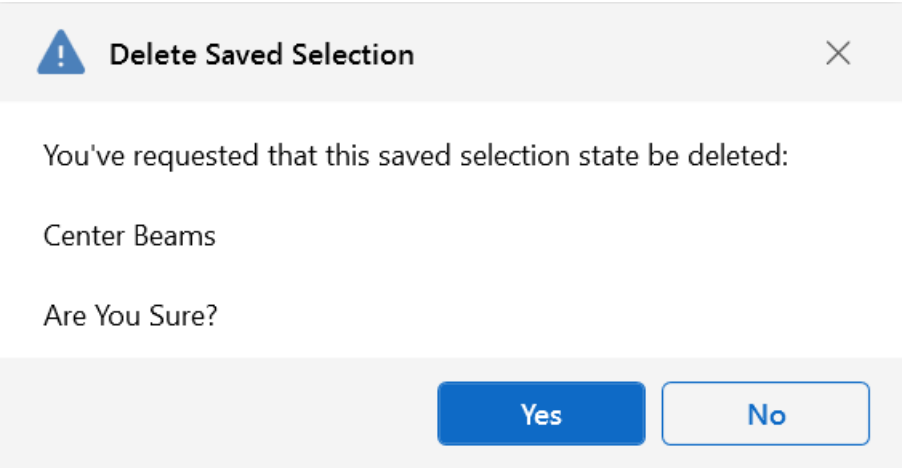
1. Click the **Saved Selections** button on the **Selection Toolbar**.

   The Selection States window opens.

2. Choose the selection state from the list on the left.
3. Click the **Delete** button.



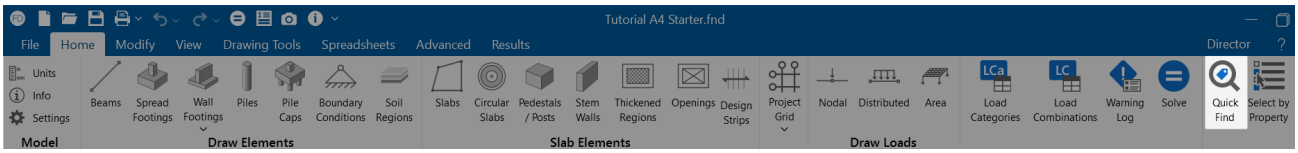
A Delete Saved Selection window opens.



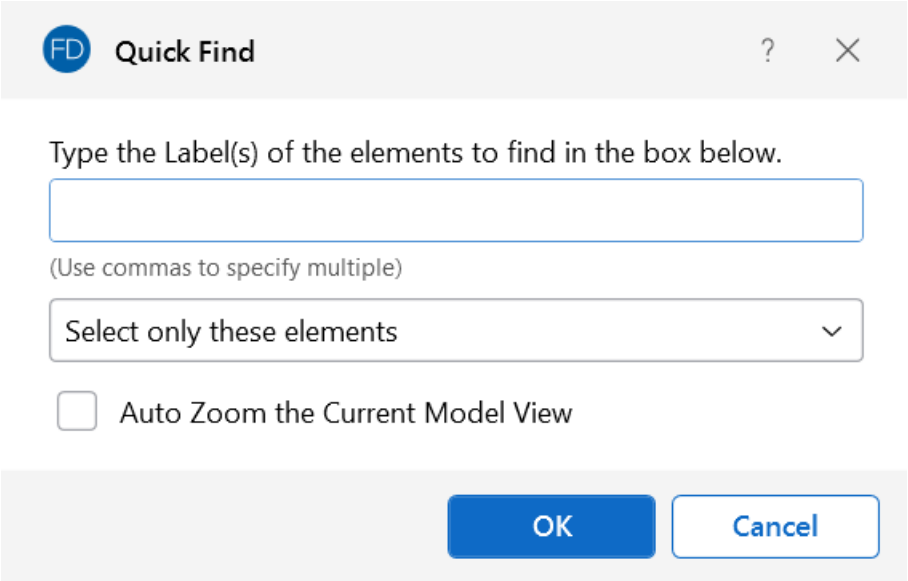
4. To confirm your deletion, select **Yes**.

Quick Find

You can quickly find elements in a model using the **Quick Find** tool located on the **Home** ribbon toolbar.



Clicking the **Quick Find** icon opens the Quick Find window.



In this window, you can type in the labels of the elements you wish to find, using commas to specify multiple elements. You can also use the drop-down to choose between **Select only these elements** or **Add these elements to the existing selection**. The Quick Find window also includes an **Auto Zoom the Current Model View** which will zoom in to the selected elements.

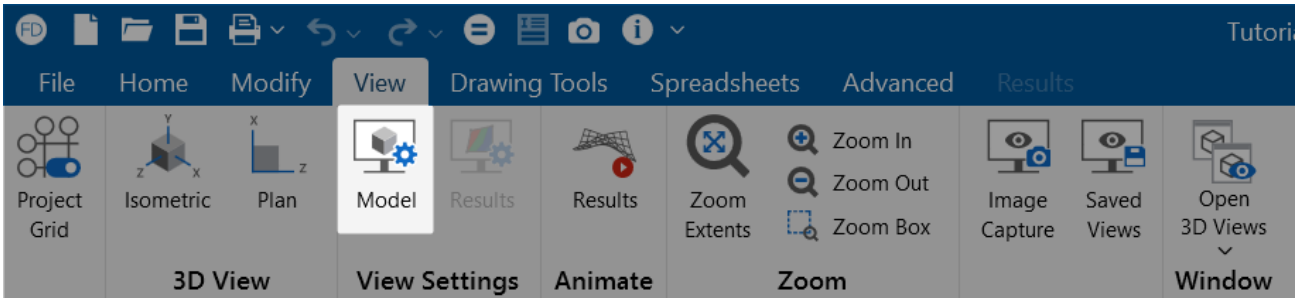
Graphic View Settings

The graphical view settings for both models and results can be found on the **View** ribbon. These settings offer the ability to define graphical plotting choices in both model and results views.

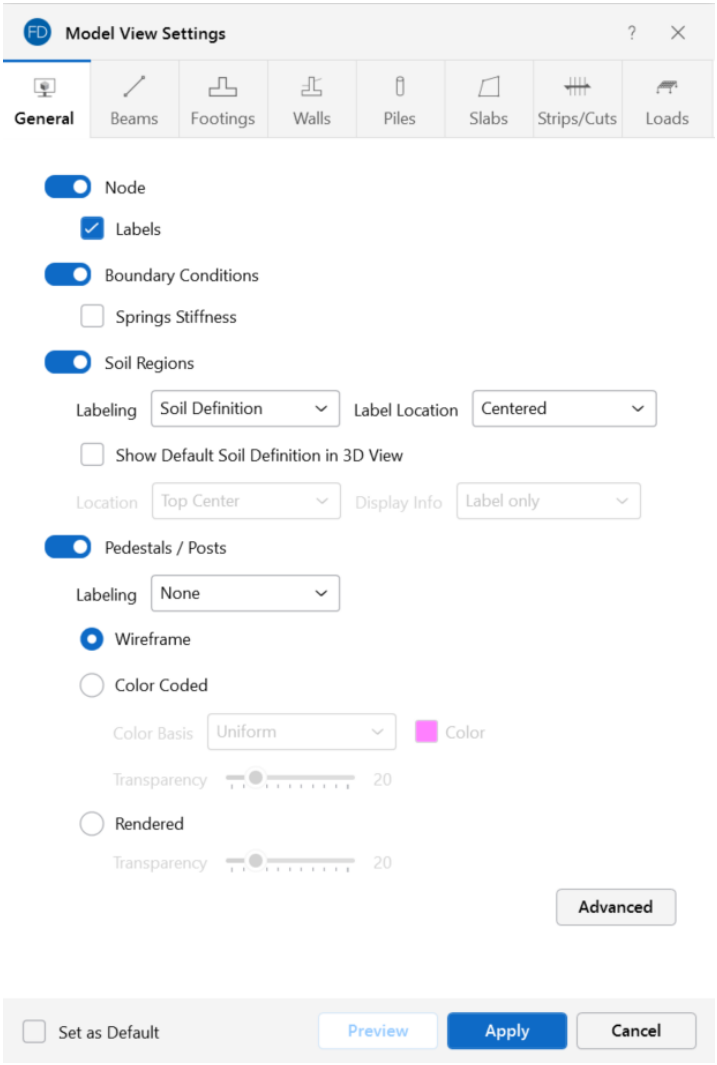
Graphics - Model View Settings

While the Quick View icons on the Home ribbon (see Graphic Display for more information) offer some common graphical plotting choices, many more options are located in the **Model View Settings** window.

The Model View Settings window can be accessed by clicking the **Model** icon in the View Settings section of the **View** ribbon.



The options in this window are organized into groups of items. Each group can be accessed by clicking on the tabs along the top of the window. Some options are mutually exclusive, while others are conditional. Indented sub-options are not available unless you choose to turn on the parent group option (click on its slider).



Each of the groups presents the option to turn off (or on) its particular items in the view. This is not the same as deselecting the items. Turning items on and off is independent from selecting and deselecting. For example, if you make a selection of beams and then turn off beams all together, the selection still stands for any modification applied to beams.

The tabs in the Model View Settings window are described in the following sections:

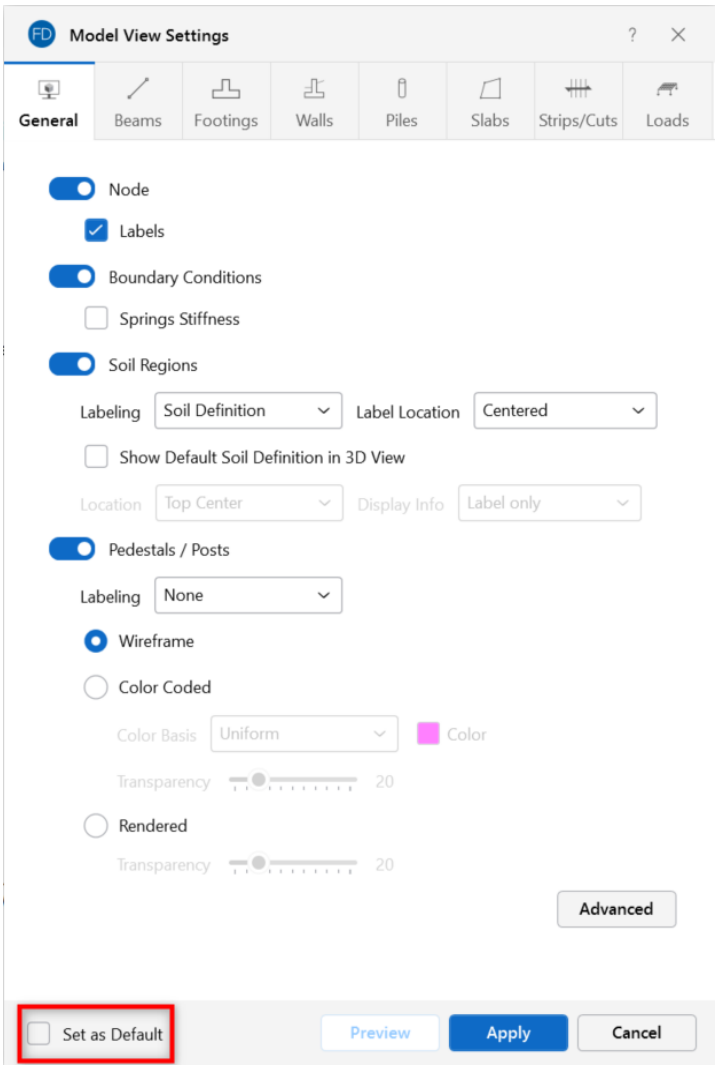
- General
- Beams
- Footings
- Walls/Stem Walls
- Piles/Pile Caps
- Slabs

- Strips/Cuts
- Loads

Set as Default

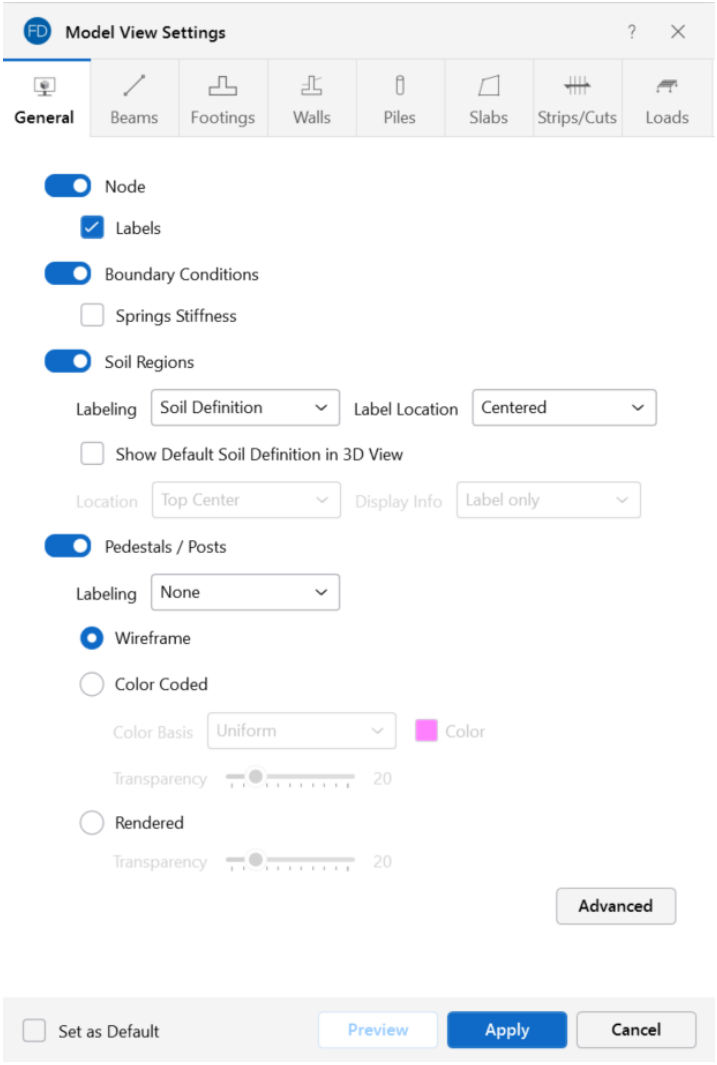
You can make any settings in the Model View Settings window the default start-up settings. To do this, go through all of the Model View Settings tabs and set the options to what you want as the start-up defaults. Once that is done, check the box at the bottom of the window next to **Set as Default**.

Note: Remember, the Set as Default check box applies to ALL tabs in the Model View Settings window.



General

Access the General graphical options by clicking the **Model** icon in the View Settings section of the **View** ribbon and selecting the **General** tab shown below.



You can specify that **Nodes** are/are not displayed, and you can include the joint label.

You can display **Boundary Conditions** and as part of this option you can display any spring stiffnesses that you have specified.

Clicking the **Soil Regions** slider will turn on the display of user-entered soil regions in the current model view. The **Labeling** drop-down can be used to turn on the display of soil region labels and to choose the labeling type. The choices are None, Soil Definition and Subgrade Modulus. Use the **Label Location** drop-down to specify the location of your label. Options include: Centered, +X, -X, +Z and -Z.

Check the **Show Default Soil Definition in 3D View** check box to display the default soil definition in the 3D View window. Then, specify the location using the **Location** drop-down. Options include: Top Center, Bottom Left and Bottom Right. The **Display Info** drop-down can be used to select the soil information displayed. Options include: Label only, or Label + Soil Data.

Clicking the **Pedestals/Posts** slider will turn on the display of pedestals/posts. The **Labeling** drop-down can be used to choose the labeling type. The choices are None, Label and Local Axes.

With the **Wireframe** option selected, you can include textual information alongside each element. The **Labeling** drop-down list provides the choices you have as to what information is displayed alongside each element.

When the **Color Coded** option is selected, pedestals/posts are plotted with different colors that are mapped in a key shown in the upper-right corner of the model view. The **Color Basis** drop-down list provides the choices you have as to how the colors are assigned to each element. Options include: Uniform, or Material. The color-coded elements may be presented with varying degrees of **Transparency**, with 100% transparency meaning completely see through, and 0% indicating completely solid. The transparency is controlled by clicking and dragging the slider bar until the desired percentage is displayed at the end of the slider bar.

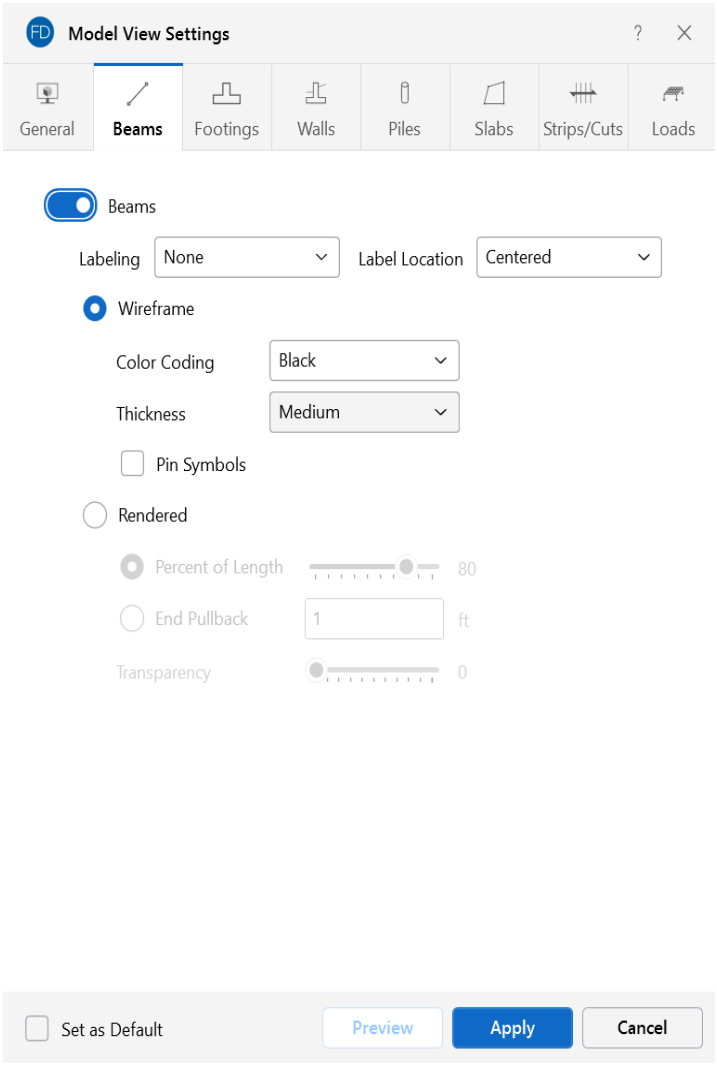
If the **Rendered** option is selected, true to scale representations of the pedestals/posts will be drawn. The **Transparency** setting described above for the Color Coded option also applies to the Rendered option.



Note: You can make any Model View Settings the default start-up settings. For more information on this feature, see Set as Default.

Beams

Access the graphical Model View Settings for beams by clicking the **Model** icon in the View Settings section of the **View** ribbon and selecting the **Beams** tab shown below.



You can specify that no beams are displayed, or you can show them with a Label, Color Code or Rendered.

Click on the **Beams** slider to not display the beams in the model at all. The **Labeling** drop-down list provides the choices you have as to what information displays alongside each beam. Options include: None, Label, Number, Shape, Material Set and Local Axes. Use the **Label Location** drop-down to specify the location of your label. Options include: Centered, Start and End.

If the **Wireframe** option is selected, you can include textual information alongside each beam. The **Labeling** drop-down list provides the choices you have as to what information will be displayed. These options are not available if the **Rendered** display option is selected. Use the **Color Coding** drop-down list to choose how the colors are assigned to each beam. Options include: None, Black, Material Set, Shape Group and Start/End. A key that defines these colors is shown in the upper-right corner of the model view. Use the **Thickness** drop-down list to select the lineweight used to represent the beams in the model view. Options include: Light, Medium and Heavy. Select the **Pin**

Symbols check box to graphically display symbols at the ends of beams assigned a pinned end release.

The **Rendered** option produces true-to-scale renderings of the beams. Beams can be rendered along the full length, a percentage of the member length, or pulled back by a specified distance. This can allow for a better understanding of complicated intersection areas and a better view of the member cross-section. This option is also very useful for verifying beam orientations and adjusting the transparency of the rendering. The rendered elements may be presented with varying **Transparency**, with 100% transparency meaning completely see through, and 0% indicating completely solid. The transparency is controlled by clicking and dragging the slider bar until the desired percentage is displayed at the end of the slider bar.

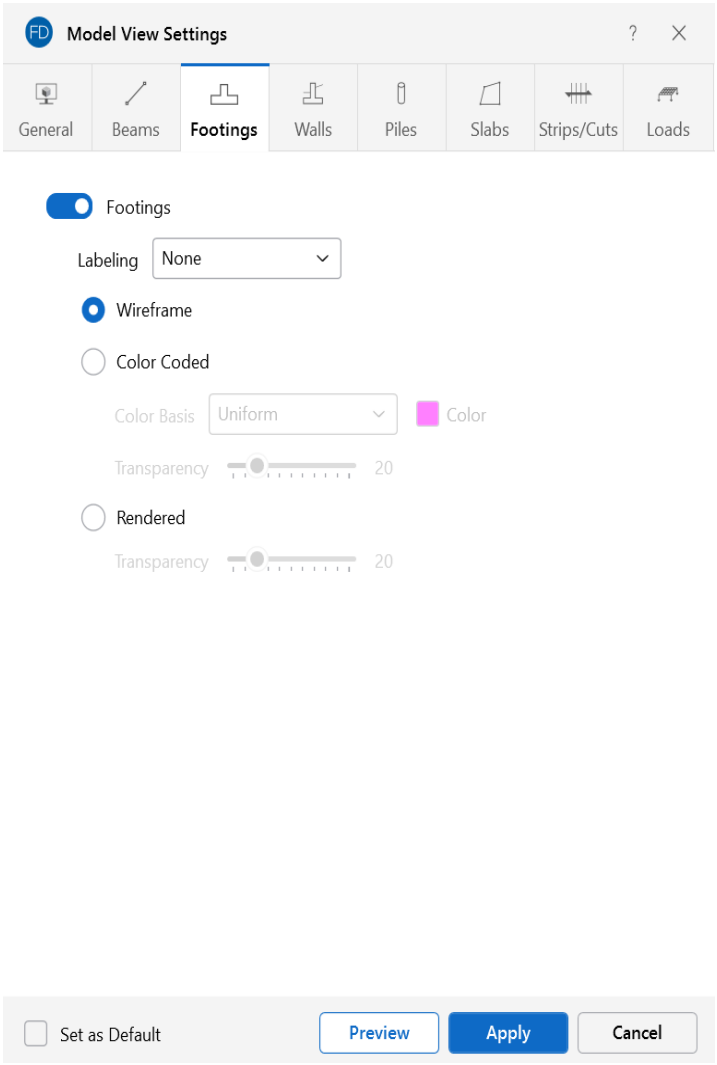
For additional advice on this topic, please see the RISA Tips & Tricks website: www.risa.com/post/support. Type in Search keyword: **Graphics**.



Note: You can make any Model View Settings the default start-up settings. For more information about this feature, see Set as Default.

Footings

Access the graphical Model View Settings for footings by clicking the **Model** icon in the View Settings section of the **View** ribbon and selecting the **Footings** tab shown below.



You can specify that no footings are displayed, or you can show them with a Label, Color Code or Rendered.

Click on the **Footings** slider to not display the footings in the model at all. The **Labeling** drop-down list provides the choices you have as to what information displays alongside each footing. Options include: None, Label and Local Axes.

When the **Wireframe** option is selected, you can include textual information on the footings when plotted.

When the **Color Coded** option is selected, footings are plotted with different colors that are mapped in a key shown in the upper-right corner of the model view. You can color the footings by material set, or specify a uniform color for all footings. The color-coded footings may be presented with varying **Transparency** with 100% transparency meaning completely see through, and 0% indicating completely solid. The transparency is controlled by clicking and dragging the slider bar until the desired percentage is displayed at the end of the bar.

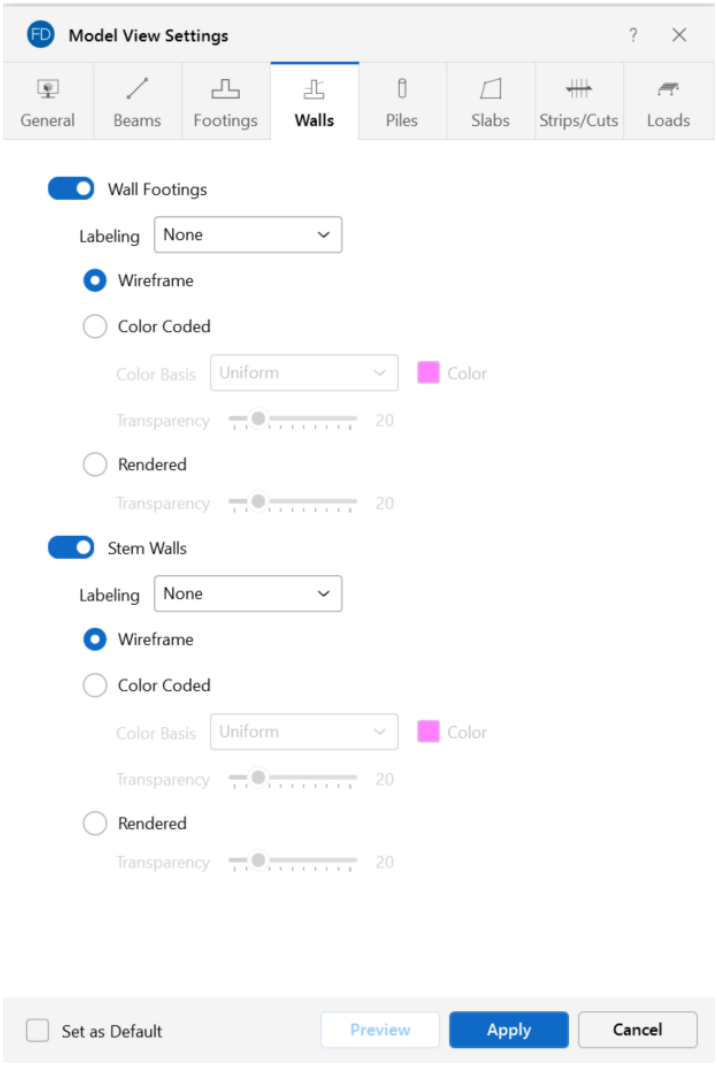
The **Rendered** option produces true-to-scale renderings of the footings. The **Transparency** setting described above for the Color Coded option also applies for the Rendered option.

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

Note: You can make any Model View Settings the default start-up settings. For more information about this feature, see Set as Default.

Walls/Stem Walls

Access the graphical Model View Settings for stem walls and wall footings by clicking the **Model** icon in the View Settings section of the **View** ribbon and selecting the **Walls** tab shown below.



You can specify that no wall footings or stem walls are displayed, or you can show them with a Label, Color Code or Rendered.

Click the **Wall Footings** and **Stem Walls** slider to not display the wall footings or stem walls in the model at all. The **Labeling** drop-down list provides the choices you have as to what information displays alongside each wall footing or stem wall. Options include: None, Label and Local Axes.

When the **Wireframe** option is selected, you can include textual information on the wall footings and stem walls when plotted.

When the **Color Coded** option is selected, wall footings and stem walls are plotted with different colors that are mapped in a key shown in the upper-right corner of the model view. You can color the wall footings and stem walls by material set, or specify a uniform color. The color-coded wall footings and stem walls may be presented with varying **Transparency**, with 100% transparency meaning completely see through, and 0% indicating completely solid. The transparency is controlled by clicking and dragging the slider bar until the desired percentage is displayed at the end of the bar.

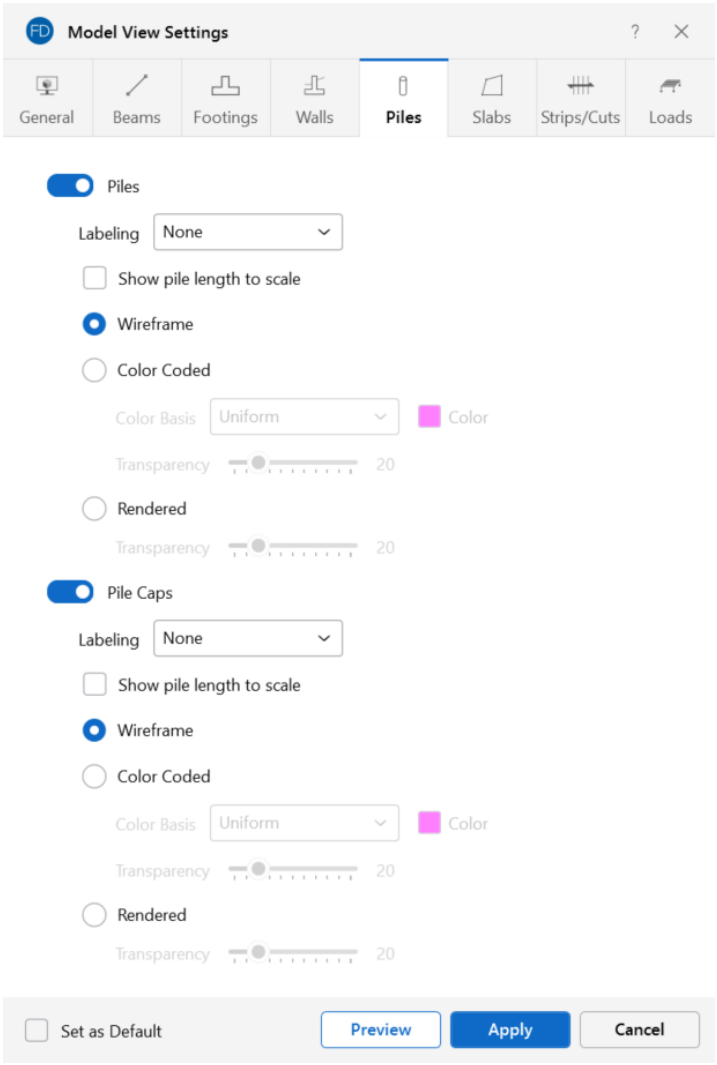
When the **Rendered** option is selected, true to scale representations of the wall footings and stem walls are displayed. The **Transparency** setting described above for the Color Coded option also applies for the Rendered option.



Note: You can make any Model View Settings the default start-up settings. For more information about this feature, see Set as Default.

Piles/Pile Caps

Access the graphical Model View Settings for piles and pile caps by clicking the **Model** icon in the View Settings section of the **View** ribbon and selecting the **Piles** tab shown below.



You can specify that no piles or pile caps are displayed, or you can show them with a Label, Color Code or Rendered.

Click on the **Piles** or **Pile Caps** sliders to not display the piles or pile caps in the model at all. The **Labeling** drop-down list provides the choices you have as to what information displays alongside each pile or pile cap. Options include: None, Label and Local Axes.

Select the **Show pile length to scale** check box to display the overall pile length true-to-scale. When this box is unchecked, the displayed length of pile elements is truncated.

When the **Wireframe** option is selected, you can include textual information on the piles when plotted.

When the **Color Coded** option is selected, piles and pile caps are plotted with different colors that are mapped in a key shown in the upper-right corner of the model view. You can color the piles or pile caps by material set, or specify a uniform color. The color-coded piles or pile caps may be presented

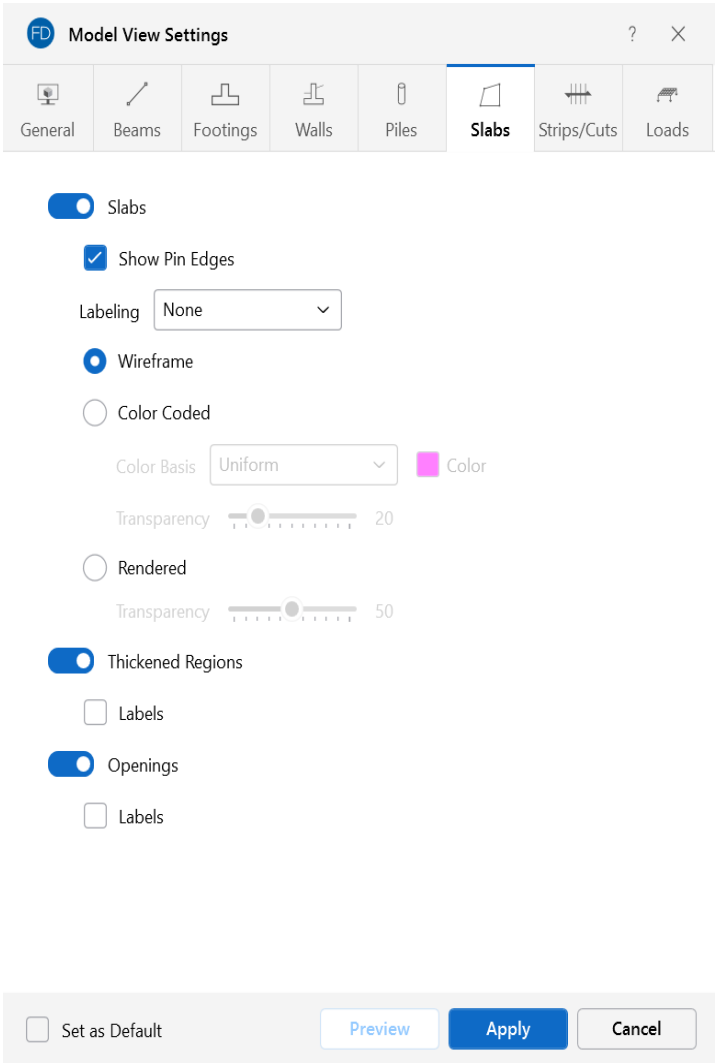
with varying **Transparency**, with 100% transparency meaning completely see through, and 0% indicating completely solid. The transparency is controlled by clicking and dragging the slider bar until the desired percentage is displayed at the end of the bar.

The **Rendered** option produces true-to-scale renderings of the piles and pile caps. The **Transparency** setting described above for the Color Coded option also applies for the Rendered option.

Note: You can make any Model View Settings the default start-up settings. For more information about this feature, see Set as Default.

Slabs

Access the graphical Model View Settings for slabs by clicking the **Model** icon in the View Settings section of the **View** ribbon and selecting the **Slabs** tab shown below.



You can specify that no slabs are displayed, or you can show them with a Label, Color Code or Rendered

Click the **Slabs**, **Thickened Regions** or **Openings** sliders to not display them in the model at all.

Select the **Show Pin Edges** check box to graphically display edge release symbols on slab elements in the model.

When the **Wireframe** option is selected, you can include textual information on the slabs when plotted. The **Labeling** drop-down list provides the choices you have as to what information displays on each slab, such as the slab label, number, thickness, material set or the local axes.

When the **Color Coded** 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 can color the slabs by material set, or specify a uniform color for all slabs. The color-coded slabs may be presented with varying **Transparency**, with 100% transparency meaning completely see through, and 0% indicating completely solid. The transparency is controlled by clicking and dragging the slider bar until the desired percentage is displayed at the end of the bar.

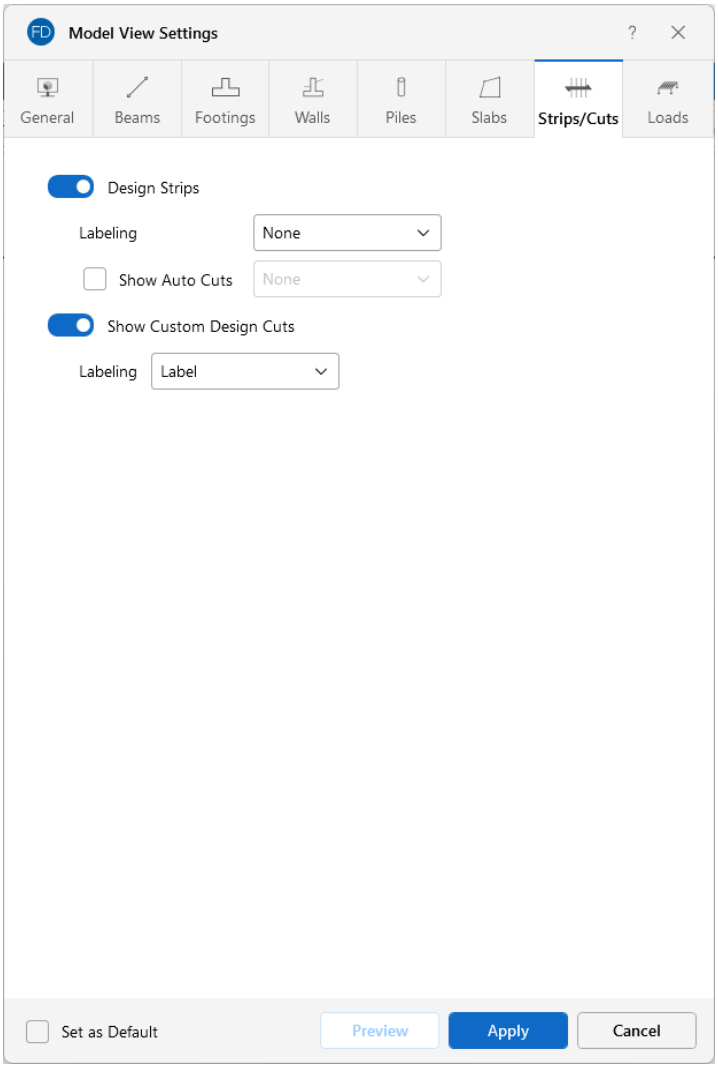
When the **Rendered** option is selected, true-to-scale representations of the slabs are drawn. The **Transparency** setting described above for Color Coded slabs also applies to Rendered slabs.



Note: You can make any Model View Settings the default start-up settings. For more information about this feature, see Set as Default.

Strips/Cuts

Access the graphical Model View Settings for design strips and cuts by clicking the **Model** icon in the View Settings section of the **View** ribbon and selecting the **Strips/Cuts** tab shown below.



Click the **Design Strips** slider to not display the design strips in the model at all.

The **Labeling** drop-down list provides the choices you have as to what information displays on each design strip. Options include: None, Label and Design Rule.

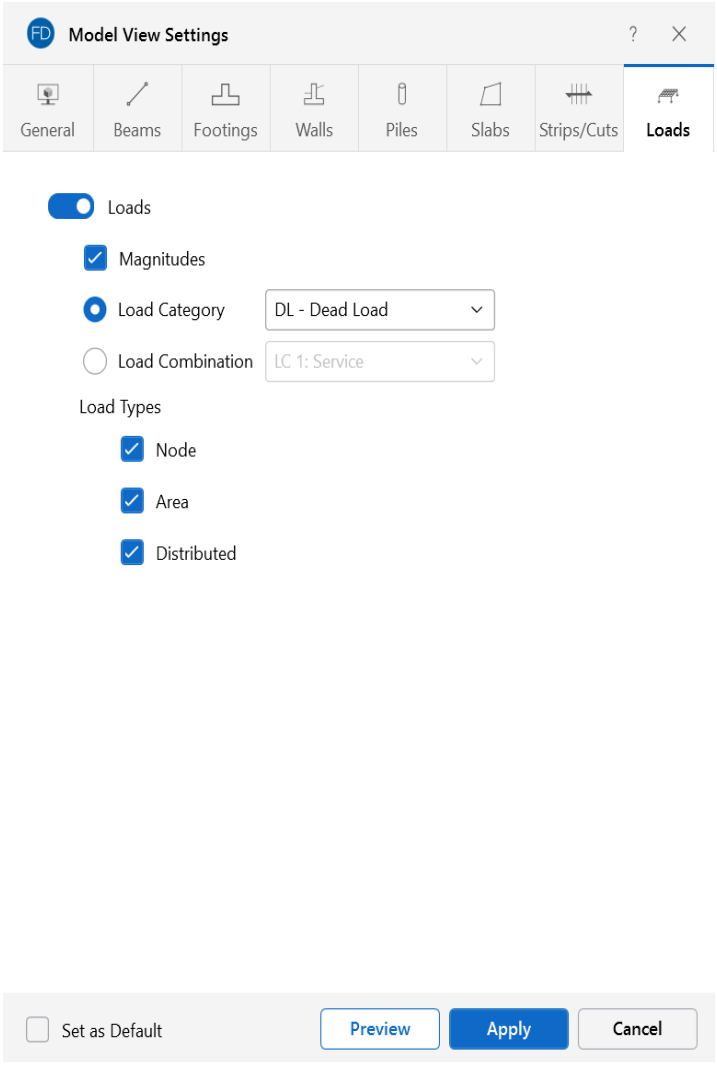
Select the **Show Auto Cuts** check box to toggle the display of cuts automatically made by the program when a model is solved. The drop-down list allows you to display design strips with the Label name or the Design Rule that is being used.

Click the **Show Custom Design Cuts** slider to not display the design cuts specified by the user. The **Labeling** drop-down list provides the choices you have as to what information displays on each custom design cut. Options include: None, Label, and Design Rule.

Note: You can make any Model View Settings the default start-up settings. For more information about this feature, see Set as Default.

Loads

Access the graphical Model View Settings for loads by clicking the **Model** icon in the View Settings section of the **View** ribbon and selecting the **Loads** tab shown below.



The **Loads** tab lets you specify that loads are to be displayed graphically, or that they are not to be shown at all.

Click the **Loads** slider to turn on and off the display of loads in the model.

Check **Magnitudes** to display the numerical magnitude of the loads. You can specify that the loads be displayed by **Load Categories** or **Load Combinations** by selecting the corresponding radio

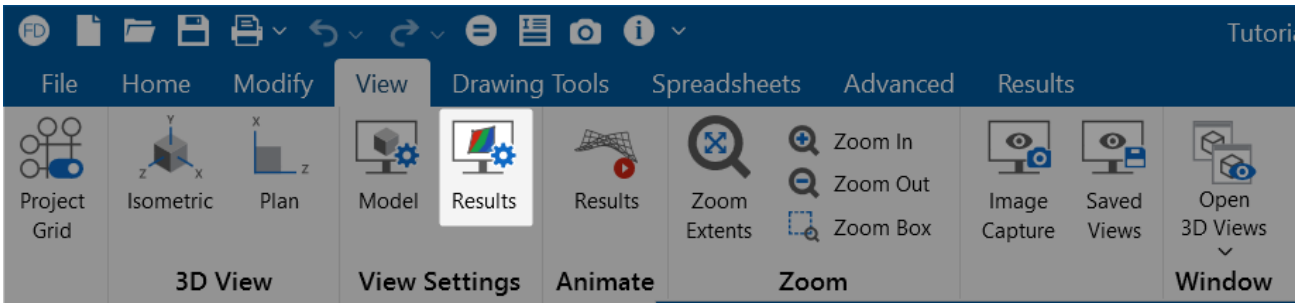
button. You can then select a specific load category or load combination to be displayed by choosing from either the drop-down list on the **Loads** tab or at the top of the **3D View**.



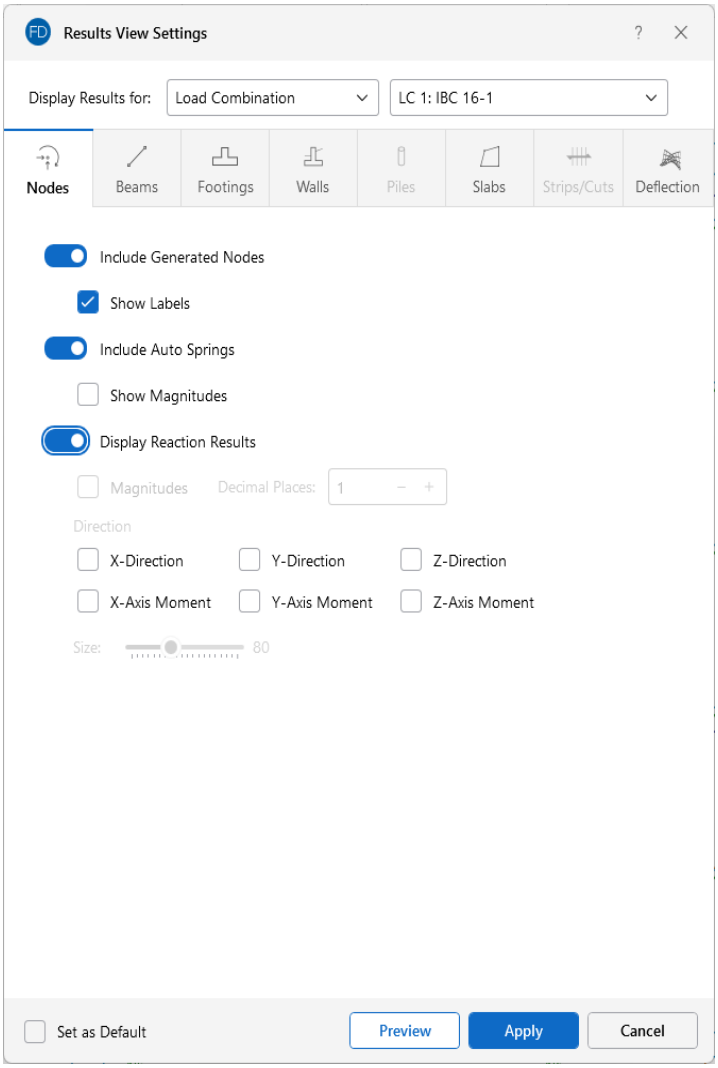
Note: You can make any Model View Settings the default start-up settings. For more information about this feature, see Set as Default.

Graphics - Results View Settings

The **Quick View** buttons on the **Home** tab (see Graphic Display for more information) offer some common graphical plotting choices, but these are just a few of the options available. Many more options related to results are located in the **Results View Settings** window which can be accessed by clicking the **Results** icon in the View Settings section of the **View** ribbon.



The options are organized into groups of items. Access each group by clicking on the tabs along the top of the window. Some of the options are mutually exclusive and others are conditional. Typically, only one radio button option may be chosen at a time, while you may be allowed to select multiple check boxes.



Each of the groups present the option to turn off the results for those particular items in the view by unchecking the Display Results option located at the top of each tab.

The displayed results are based on the selected Load Combination in the drop-down menu at the top of the Results View Settings window.

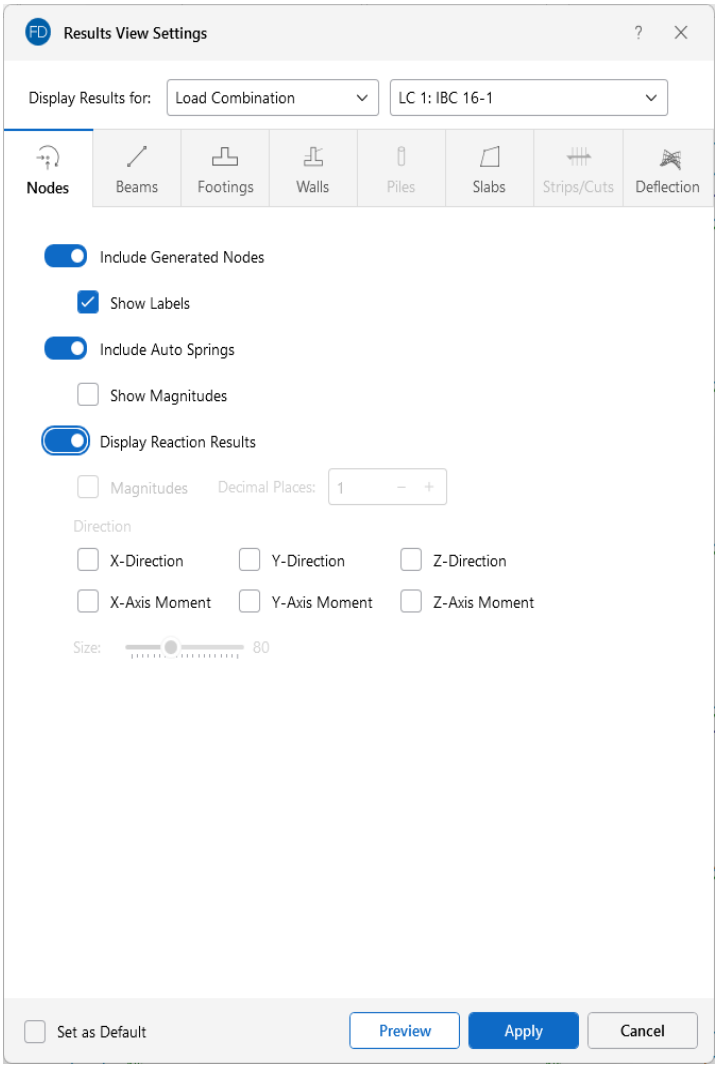
Please see the following sections for more information about the setting options in the Results View Settings window:

- Nodes
- Beams
- Footings
- Walls
- Piles

- Slabs
- Strips/Cuts
- Deflection

Nodes

Access the graphical Results View Settings for nodes by clicking the **Results** icon in the View Settings section of the **View** ribbon and selecting the **Nodes** tab shown below.



Clicking the **Include Generated Nodes** slider will display nodes generated during the automated meshing of slab elements when the model is solved. Node labels may also be turned on by checking the **Show Labels** check box.

Clicking the **Include Auto Springs** slider will turn on the display of springs automatically generated at the meshed nodes of slab elements. Note that these will only be displayed after a solution is

performed. The magnitudes of the springs may also be displayed by checking the **Show Magnitudes** check box.

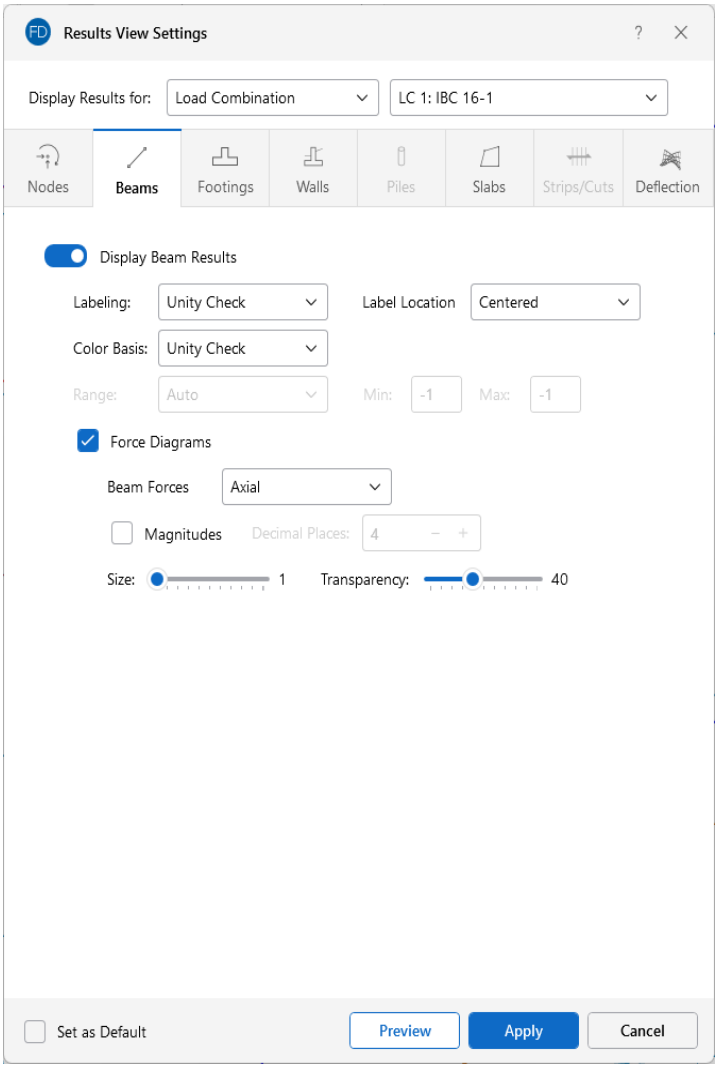
Clicking the **Display Reaction Results** slider will display reaction results. The magnitudes of the reactions may also be displayed by checking the **Magnitudes** check box. Set the decimal places displayed for the reaction magnitudes using the **Decimal Places** input field. You may specify which reactions directions are displayed in the model view using the various **Direction** check boxes. Control the graphic scaling of the force diagrams symbols for the reactions using the **Size** slider bar.

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

Note: You can make any Results View Settings the default start-up settings. For more information about this feature, see Set as Default.

Beams

Access the graphical Results View Settings for beams by clicking the **Results** icon in the View Settings section of the **View** ribbon and selecting the **Beams** tab.



Clicking the **Display Beam Results** slider (to make it turn blue) will display Beam Results in the model view.

You can specify that beam results are to be shown with labels, or using a color basis.

The **Labeling** drop-down list provides the choices you have as to what information will be displayed alongside each member.

Choosing a **Color Basis** for member results uses various colors to represent particular results, such as the unity check, stress levels, etc. The drop-down list provides the choices you have as to how the colors are to be assigned to each member. You may select which combination you want to view at the top of the Results View Settings dialog. The key that defines these colors is shown in the upper-right corner of the model view.

If the model has been solved you can plot the member force **Diagrams** along each member and include the force magnitudes by checking the **Magnitudes** check box. You can set the number of

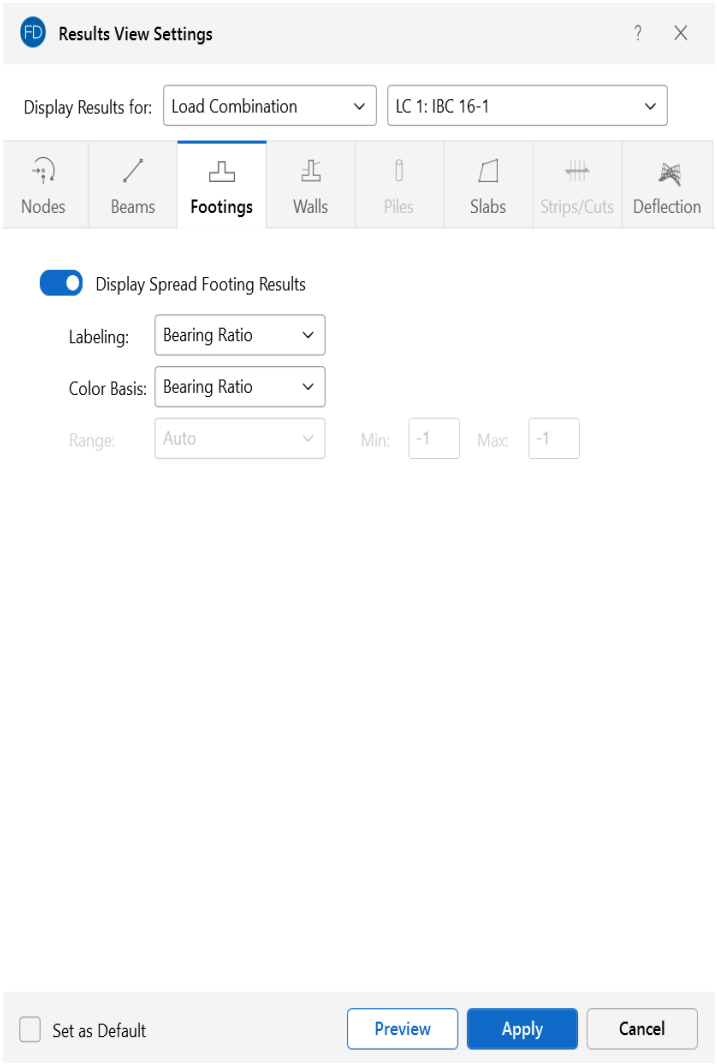
Decimal Places for the magnitudes, as well as control the graphic scaling of these force diagrams using the **Size** slider bar. The diagrams themselves can be presented with varying degrees of **Transparency**, with 100% transparency meaning only an outline of the force diagram is displayed and 0% indicating a solid fill.

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

Note: You can make any Results View Settings the default start-up settings. For more information about this feature, see Set as Default.

Footings

Access the graphical Results View Settings for spread footings by clicking the **Results** icon in the View Settings section of the **View** ribbon and selecting the **Footings** tab.



Clicking the **Display Spread Footing Results** slider (to make it turn blue) will display Spread Footing Results in the model view.

You can specify that spread footing results are to be shown with labels, or using a color basis.

The **Labeling** drop-down list provides the choices you have as to what information will be displayed alongside each spread footing.

Choosing a **Color Basis** for spread footing results uses various colors to represent particular results, such as the unity check, stress levels, etc. The drop-down list provides the choices you have as to how the colors are to be assigned to each spread footing. You may color the spread footings by the code check or the stress magnitudes. The key that defines these colors is shown in the upper-right corner of the model view.

The **Range** controls determine how the contour colors are assigned. You can either contour the full range of the results by choosing Automatic, or, if you are only interested in a specific range of values, you can choose Custom and enter your own **Min** and **Max** values.

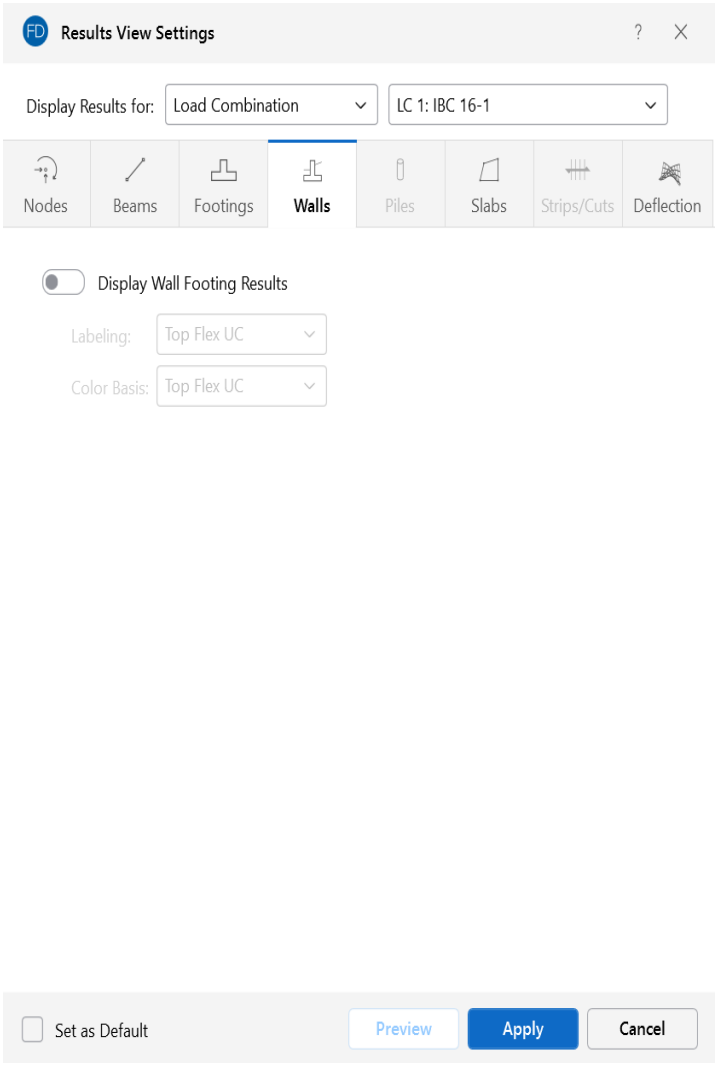
For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Graphics**.



Note: You can make any Results View Settings the default start-up settings. For more information about this feature, see Set as Default.

Walls

Access the graphical Results View Settings for wall footings by clicking the **Results** icon in the View Settings section of the View ribbon and then clicking the **Walls** tab.



Click on the **Display Wall Footing Results** slider to display any wall footing results.

You can specify that wall footing results are to be shown with labels, or using a color basis.

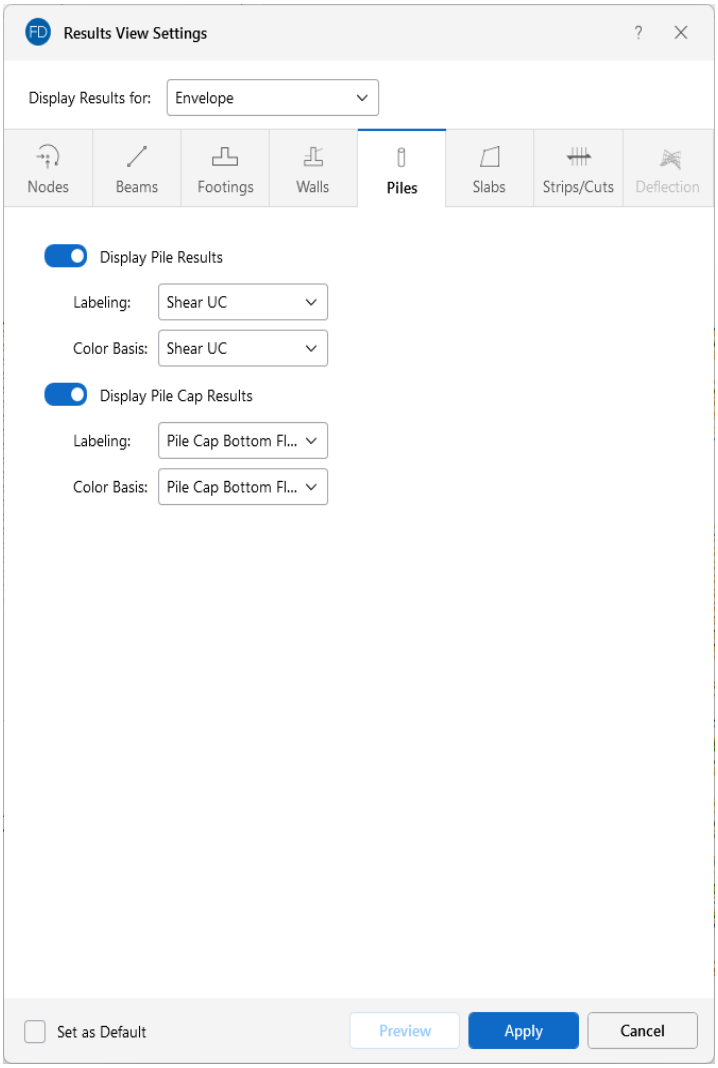
The **Labeling** drop-down list provides the choices you have as to what information will be displayed alongside each wall footing.

Choosing a **Color Basis** for wall footing results uses various colors to represent specific results. The drop-down list provides the choices you have as to how the colors are to be assigned to each wall footing.You may color the wall footing based on the unity checks for shear and flexure, sliding unity checks, max bearing pressure, etc The key that defines these colors is shown in the upper-right corner of the model view.

Note: You can make any Results View Settings the default start-up settings. For more information about this feature, see Set as Default.

Piles

Access the graphical Results View Settings for piles by clicking the **Results** icon in the View Settings section of the **View** ribbon and selecting the **Piles** tab.



Clicking the **Display Pile Results** slider (to make it turn blue) will display Pile Results in the model view.

You can specify that pile results are to be shown with labels, or using a color basis.

The **Labeling** drop-down list provides the choices you have as to what information will be displayed alongside each pile.

Choosing a **Color Basis** for pile results uses various colors to represent particular results, such as the unity check, stress levels, etc. The drop-down list provides the choices you have as to how the colors are to be assigned to each pile. You may color the piles by the code check or the stress magnitudes. The key that defines these colors is shown in the upper-right corner of the model view.

Clicking the **Display Pile Cap Results** slider (to make it turn blue) will display Pile Cap Results in the model view.

You can specify that pile cap results are to be shown with labels, or using a color basis similar to Piles as discussed above.

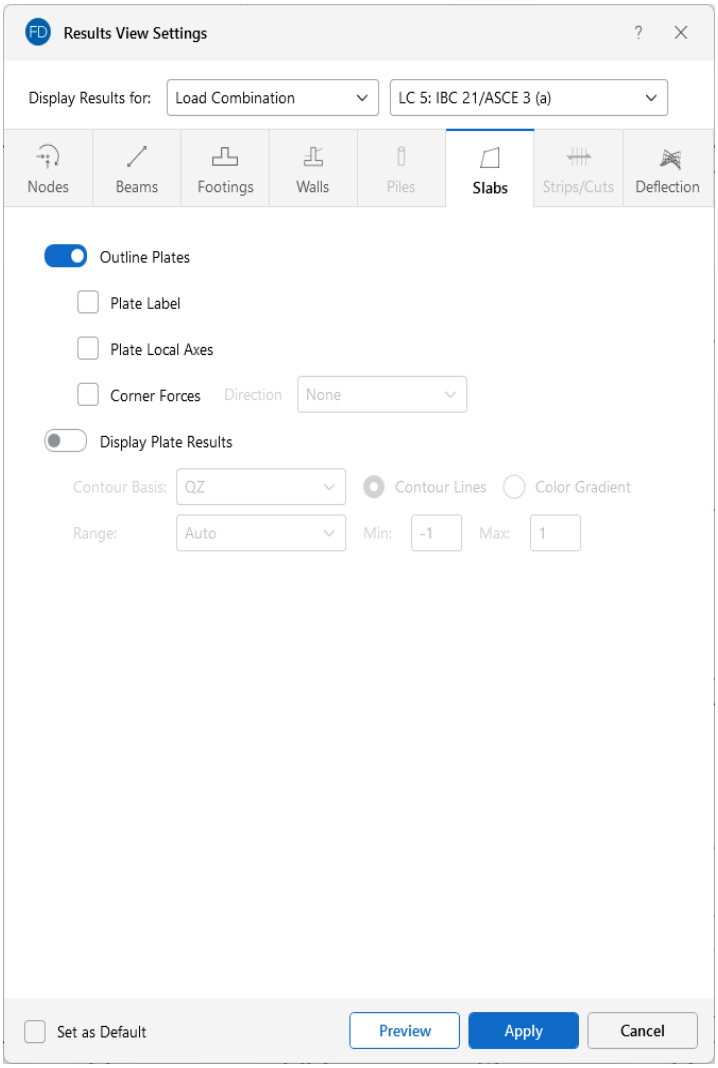
For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Graphics**.



Note: You can make any Results View Settings the default start-up settings. For more information about this feature, see Set as Default.

Slabs

Access the graphical Results View Settings for slabs by clicking the **Results** icon in the View Settings section of the View ribbon and then clicking the Slabs tab.



You can specify that slab results are to be shown as contour lines, color gradient or that they are not to be shown at all.

Click on the **Outline Plates** slider to make it turn blue if you want to display the outline of the submeshed plates automatically created when the model is solved.

Check the **Plate Label** box to display unique labels alongside each plate.

Selecting the **Plate Local Axes** check box will display the local axes on each of the submeshed plate elements making up the slab.

Click on the **Display Plates Results** slider to make it turn blue if you want to display the plate contour results.

T the contour lines represent the force or stress results for the slabs. Each color line represents a specific value. You also have the option to plot the stress and force results using filled color contours where each fill color represents a range of values.

The **Contour Basis** drop-down list is where you select the specific result to be contoured. The **Range** controls determine how the contour colors are assigned. You can either contour the full range of the results by choosing **Automatic**, or, if you are only interested in a specific range of values, you can choose **Custom** and enter your own max and min values.

The **Color Contours** are plotted with a global smoothing algorithm that allows the contour to vary across the plates within a slab. Because of this, the plotted contour results differ slightly from the tabulated results in the spreadsheets. This effect is heightened in regions with a high rate of change, such as at concentrated loads or boundaries.

Along with any of the above options, you may also plot the plate **Corner Forces**. These are the global direction forces applied to the corners of the plates within a slab that hold them in equilibrium. The **Direction** options specify which corner force (or moment) is to be displayed.



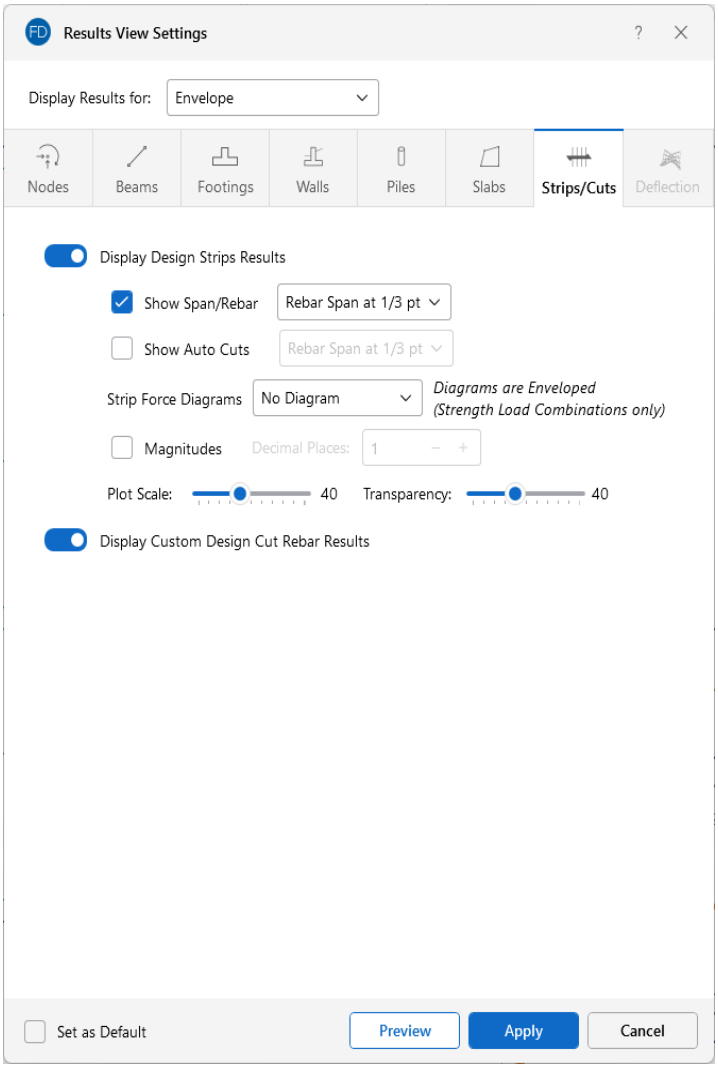
Note: You can make any Results View Settings the default start-up settings. For more information about this feature, see Set as Default.

Design Strips/Cuts

Access the graphical Results View Settings for design strips or cuts by clicking the **Results** icon in the View Settings section of the **View** ribbon and selecting the **Strips/Cuts** tab.



Note: Envelope results must be selected from the drop-down menu at the top of the dialog window in order for the Strips/Cuts tab to display.



Clicking the **Display Design Strip Results** slider (to make it turn blue) will display Design Strip Results in the model view.

Select the **Show Span/Rebar** check box to toggle on the display of the rebar layout. You can choose where you want the span/rebar label to display from the drop-down menu.

Select the **Show Auto Cuts** check box to toggle on the display of cuts automatically made by the program during solution. You can choose where you want the auto cut labels to display from the drop-down menu.

The **Strip Force Diagrams** drop-down menu lets you choose the enveloped forces to be plotted on each design strip. Include the display of force magnitudes by checking the **Magnitudes** check box and control the number of **Decimal Places** using the drop-down menu. You can also control the graphic scaling of these force diagrams using the **Plot Scale** slider bar. The diagrams can be presented with varying degrees of **Transparency**, with 100% transparency meaning only an outline of the force diagram is displayed and 0% indicating a solid fill.

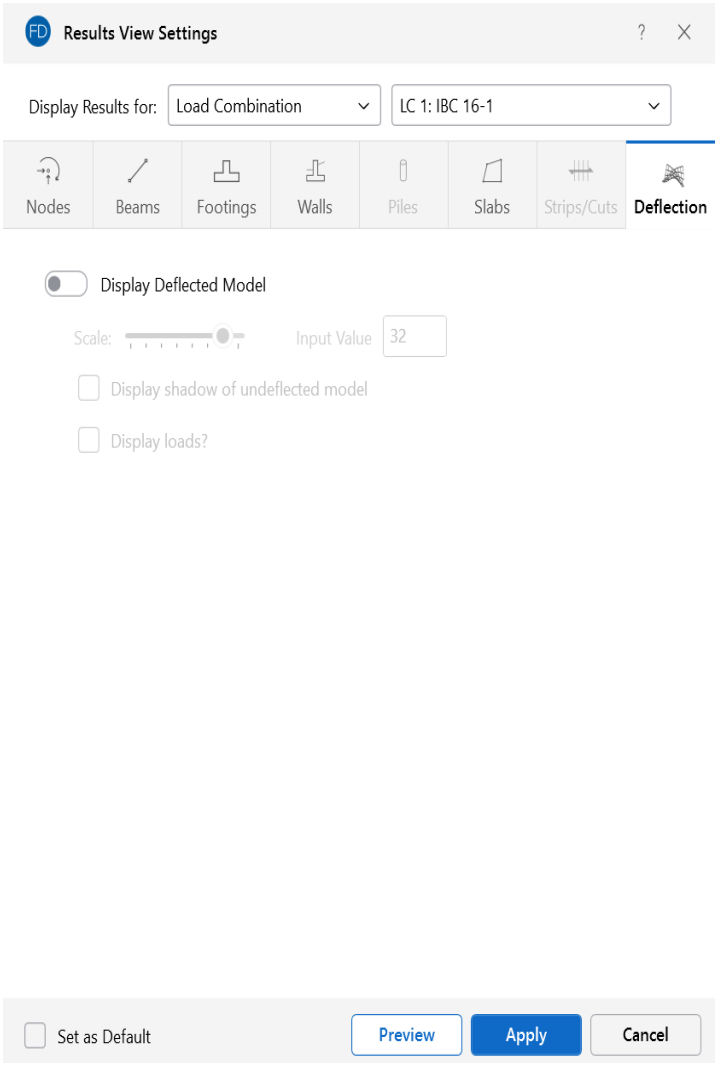
Clicking the **Display Custom Design Cut Rebar Results** slider (to make it turn blue) will display Custom Design Cut Rebar Results in the model view.

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

Note: You can make any Results View Settings the default start-up settings. For more information about this feature, see Set as Default.

Deflection

Access the Results View Settings for deflections by clicking the **Results** icon in the View Settings section of the View ribbon and selecting the **Deflection** tab.



Click on the **Display Deflected Model** slider to make it turn blue, if you want to display any deflected shape results.

You can control the magnification of the deflections with the **Scale** slider and text box. You can specify whether an undeflected shadow is to be shown by checking the **Display shadow of undeflected model** box.

To animate a particular deflected shape, first select the deflected shape (as described above) and then click the **Animate Results** icon on the **View** ribbon. For more information, see Animate Results.

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

Note: You can make any Results View Settings the default start-up settings. For more information about this feature, see Set as Default.

Help Options

RISA Tech, Inc. has, and will, put a great deal of effort into assisting you in getting your work done as quickly as possible. This includes providing several ways in which you can get help to understand the software.

Electronic Help File

The **Help File** is designed to help you get your work done as quickly as possible and is intended to provide:

- Procedures that lead users through the steps of completing tasks
- Context-sensitive Help topics that provide users with quick descriptions of items on their screens
- Troubleshooting topics that guide users through solutions to common problems
- Extensive discussions for a thorough understanding of engineering and modeling topics
- Easy access to related topics

The electronic help file can be accessed by clicking the **Help File**  button in the top right hand corner of the **Ribbon Toolbar**. A new window containing a Table of Contents opens. Click on any item in the Table of Contents for extensive information on the topic.

Context Sensitive Help

Context Sensitive Help is help that you can access while working in a program. It provides you with the information you need where and when you want it.

You can get detailed help when working in a dialog, spreadsheet or graphic view by pressing **F1**. This launches a Help File window displaying the topic that is related to the window in which you are working. The topic is explained and links to related topics may also be provided.

RISA Tech, Inc. Online

Our website, www.risa.com, provides various support information and documents.

Visit RISA Tech, Inc. on the web to:

- Download program Manuals (General Reference or Tutorial)
- Check the Customer Portal for the latest updates.
- Software Verification Problems

Tool-tips

Are you uncertain what a toolbar icon or button is for? Simply hold your mouse pointer over it without clicking. A **Tool-tip** is displayed that explains what the button does, should you decide to press it.

Tutorials

The comprehensive **Tutorials** guide you through the use of most features. They are real-world examples of building and solving models, making changes and optimizing. This is the best way to quickly get up and running.

Hot Rolled Steel - Databases

Shapes are organized by **Shape Type** and **Database / Manufacturer**. Common shapes are supported such as; wide flanges, tubes, pipes, channels, etc. You can type in the names directly, select shapes from these databases or add your own shapes.

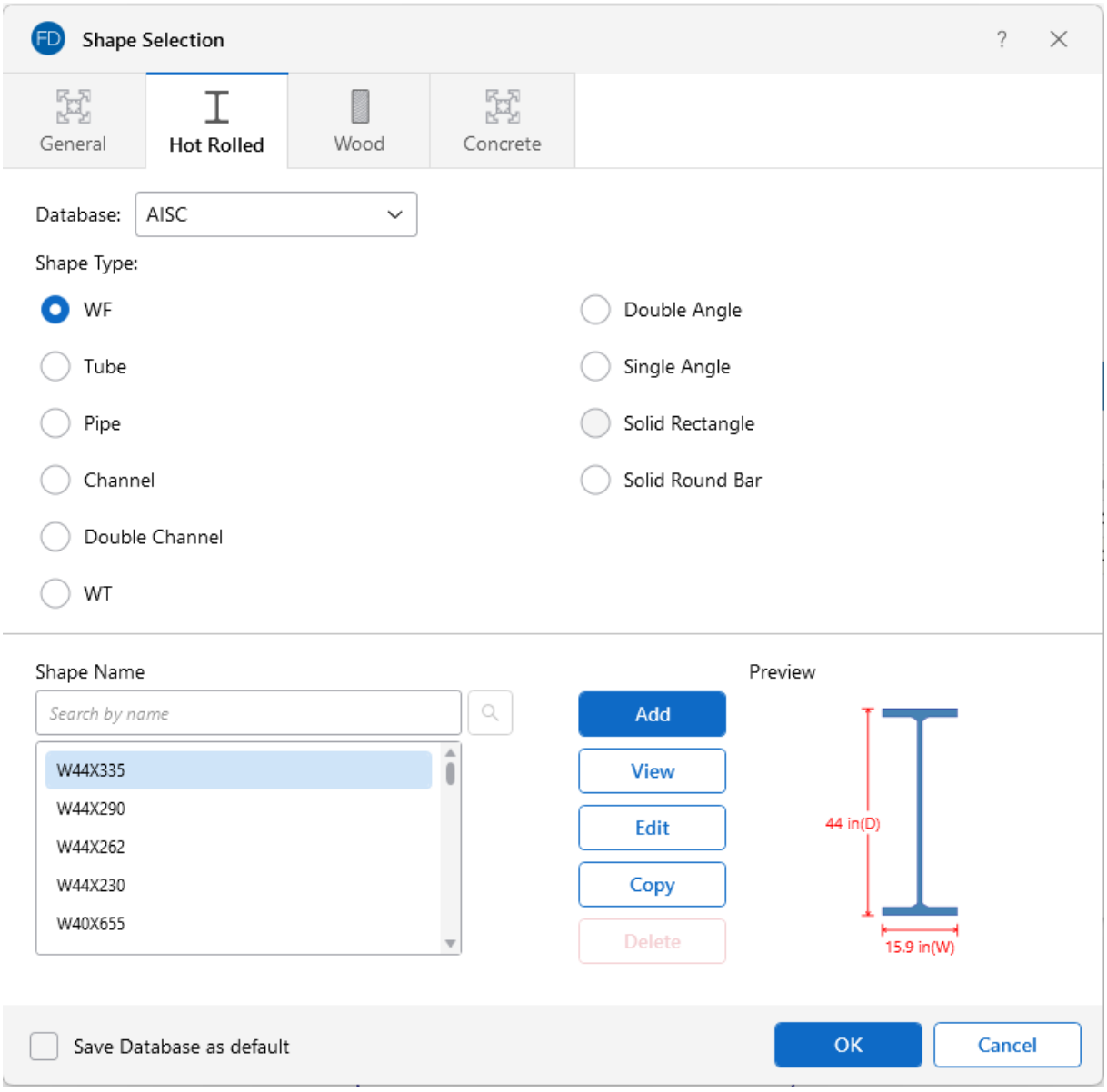
RISA currently supports the following common Hot Rolled steel databases: AISC (United States), Australian, British, Canadian, Chilean, Chinese, European, Indian, Korean and Mexican.

Note: Older AISC shapes, which are no longer part of the AISC database, are automatically moved/exported to a new AISC_backup database during installation or update.

Select a Database Shape

To select a Hot Rolled Steel database shape for pile elements:

1. Click the ellipsis button in the Shape column of the **Pile Definitions Spreadsheet** to open the **Shape Selection** dialog.
2. Click on the **Hot Rolled** tab.

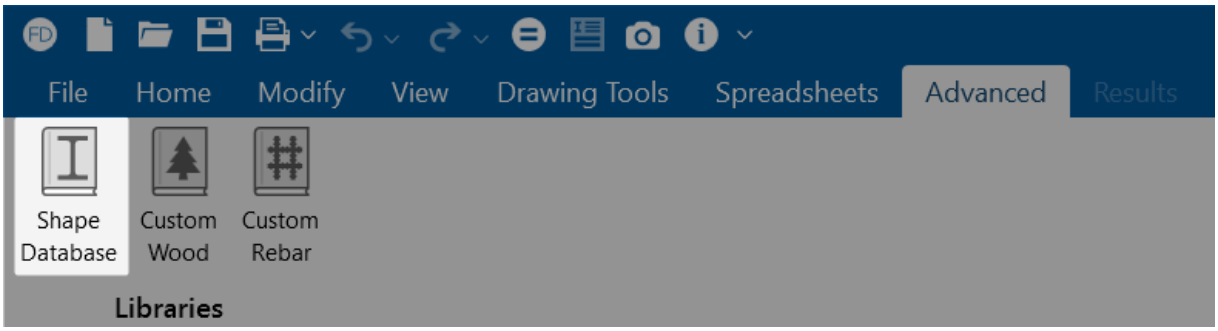


- Choose the appropriate **Database**.
- Choose the **Shape Type**. See Available Database Shapes for more information on shape types.
- In the **Shape Name** list, find and select the shape you want to use.
- Click **OK** to save your selection, close the window, and return to the spreadsheet.
- The chosen shape is displayed in the Shape cell of the spreadsheet.

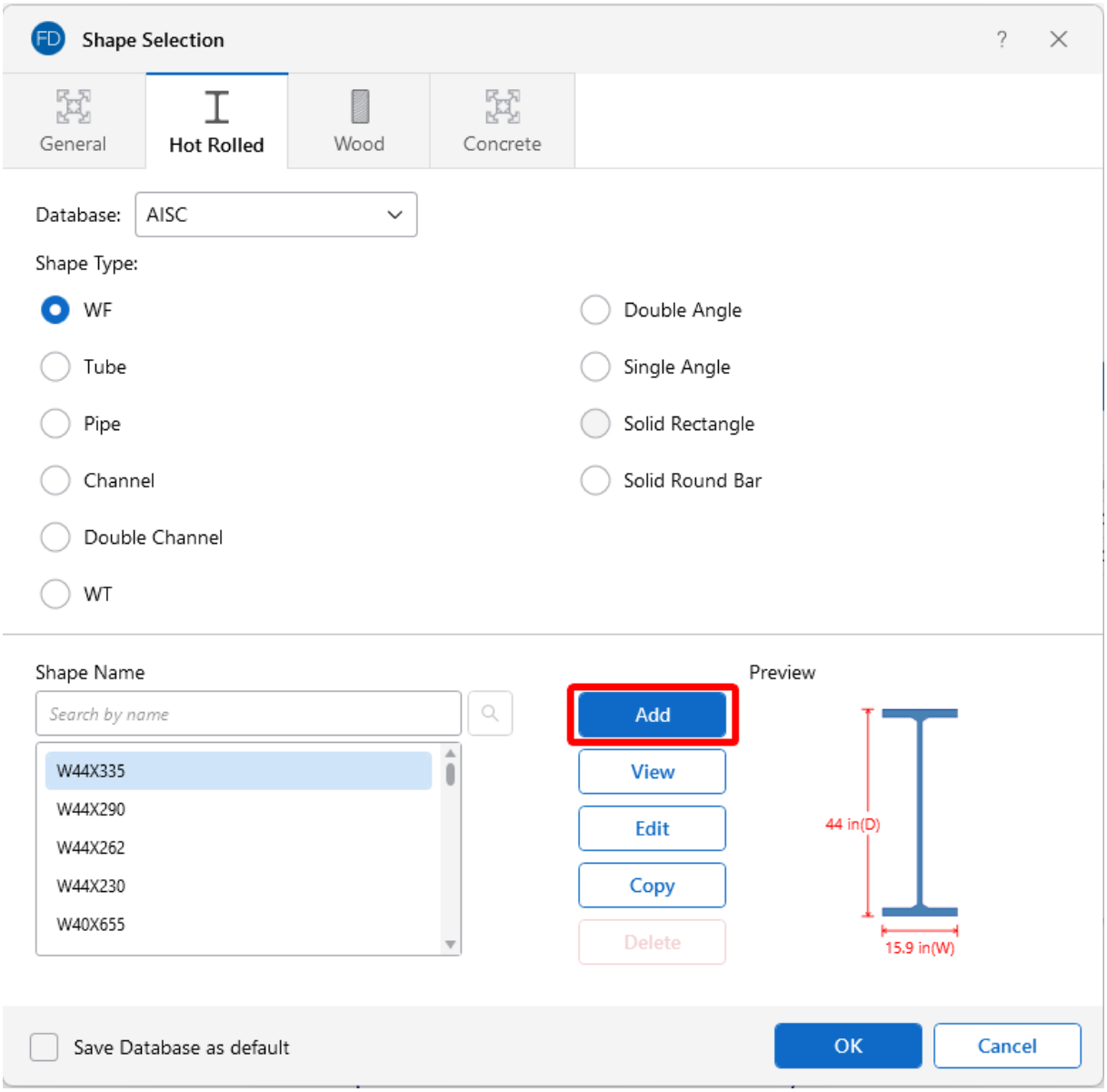
Add a Database Shape

To add a Hot Rolled Steel database shape:

1. On the **Advanced** ribbon, click the **Shape Database** icon.



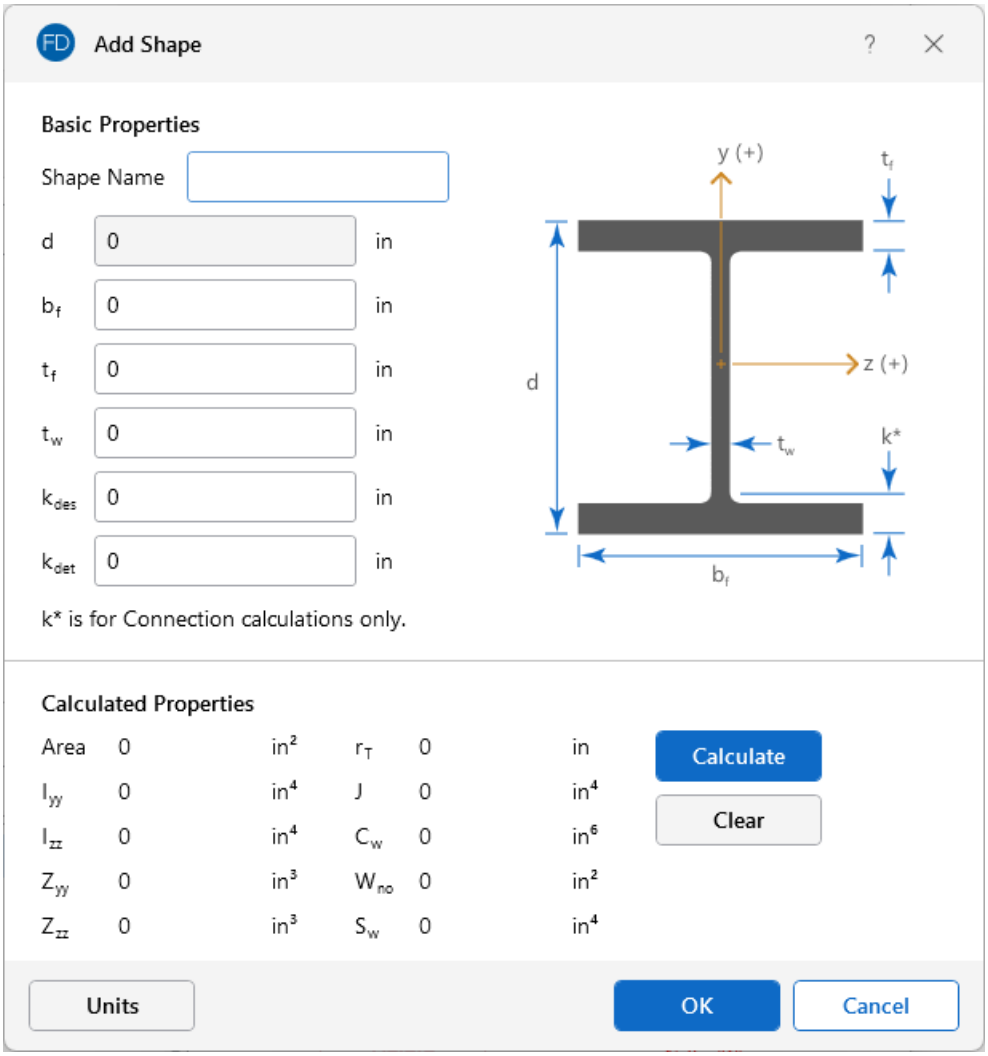
The Shape Selection window opens.



2. Click on the **Hot Rolled** tab.
3. Choose the **Database** and **Shape Type** you wish to add.

   See Available Database Shapes for more information on shape types.
4. Click the **Add** button.

   The Add Shape window opens.



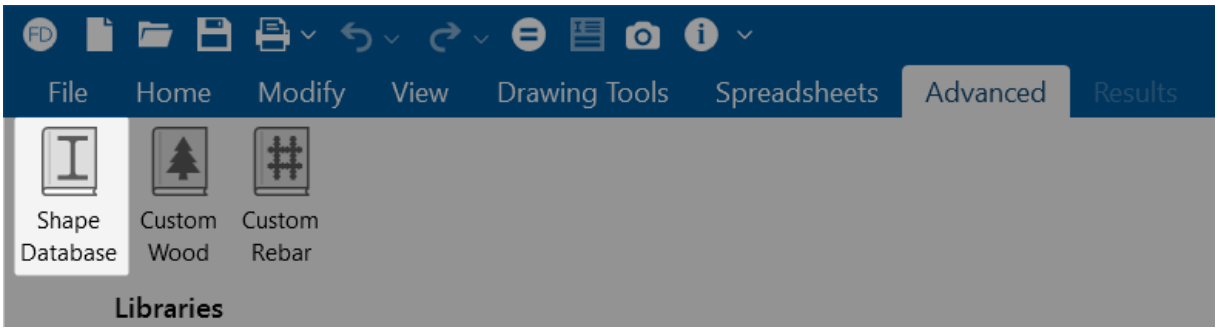
5. Under **Basic Properties**, specify a **Shape Name** for the shape and fill in the basic properties.
6. Click the **Calculate** button to determine the shape properties.
7. Click **OK** to accept the new shape and close the Add Shape window.

   The new shape is added to the bottom of the Shape Name list in the Shape Selection window.

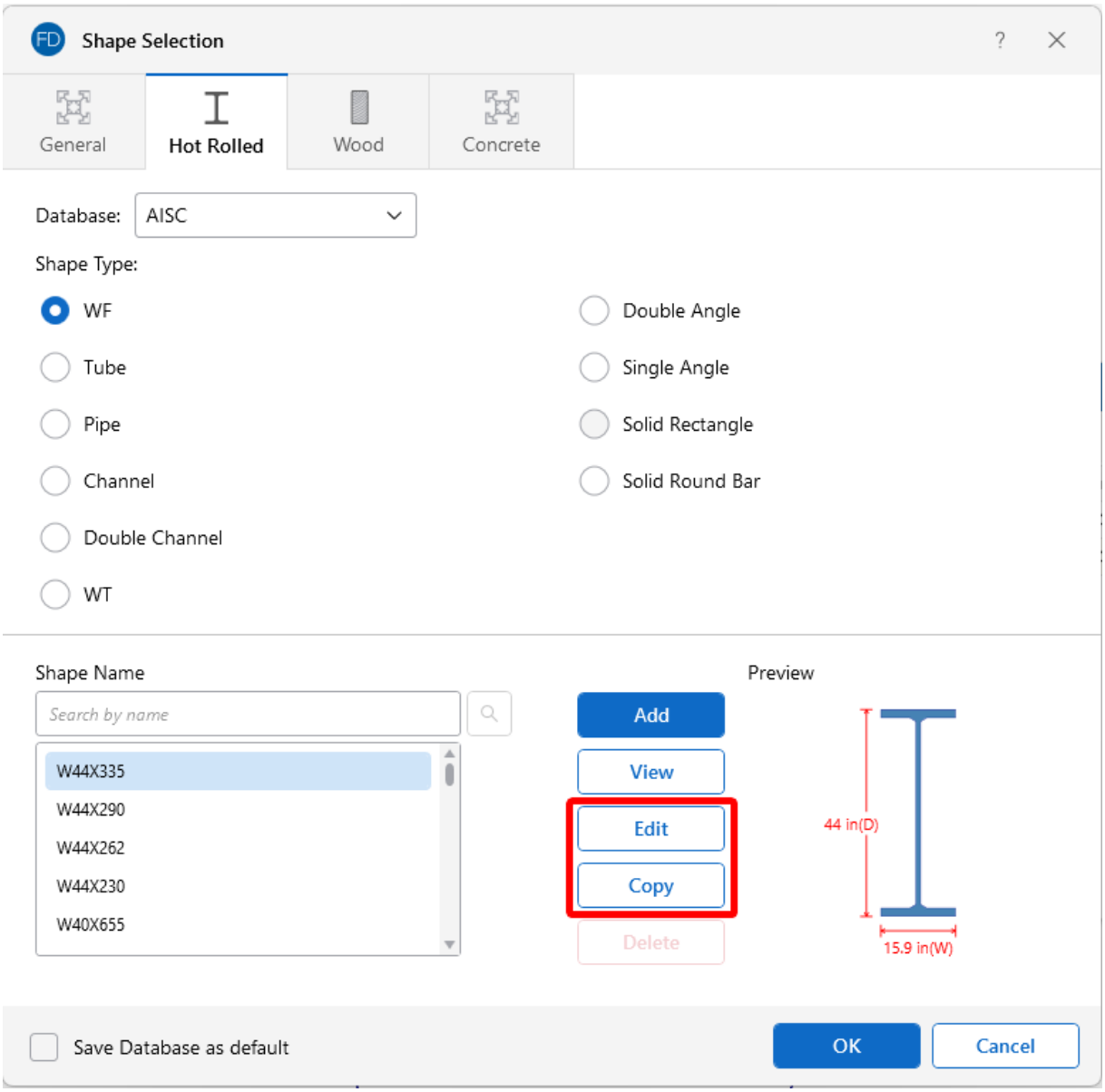
Edit a Database Shape

To edit a Hot Rolled Steel database shape:

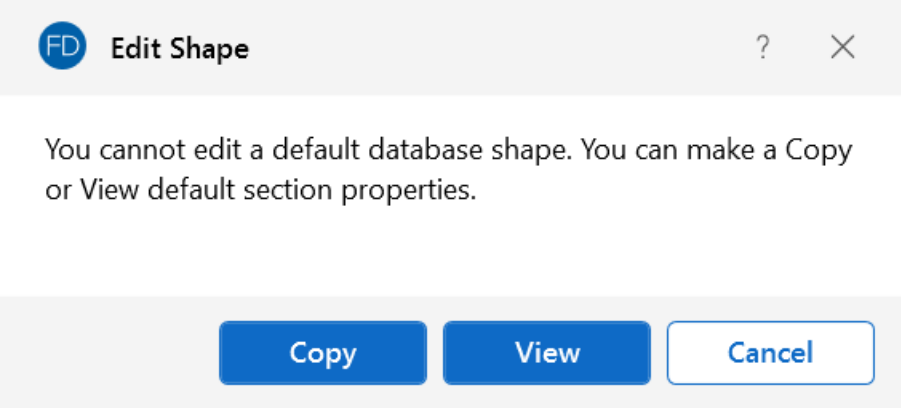
1. Go to the **Advanced** ribbon.
2. Click on the **Shape Database** icon in the **Libraries** section.



The Shape Selection window opens.



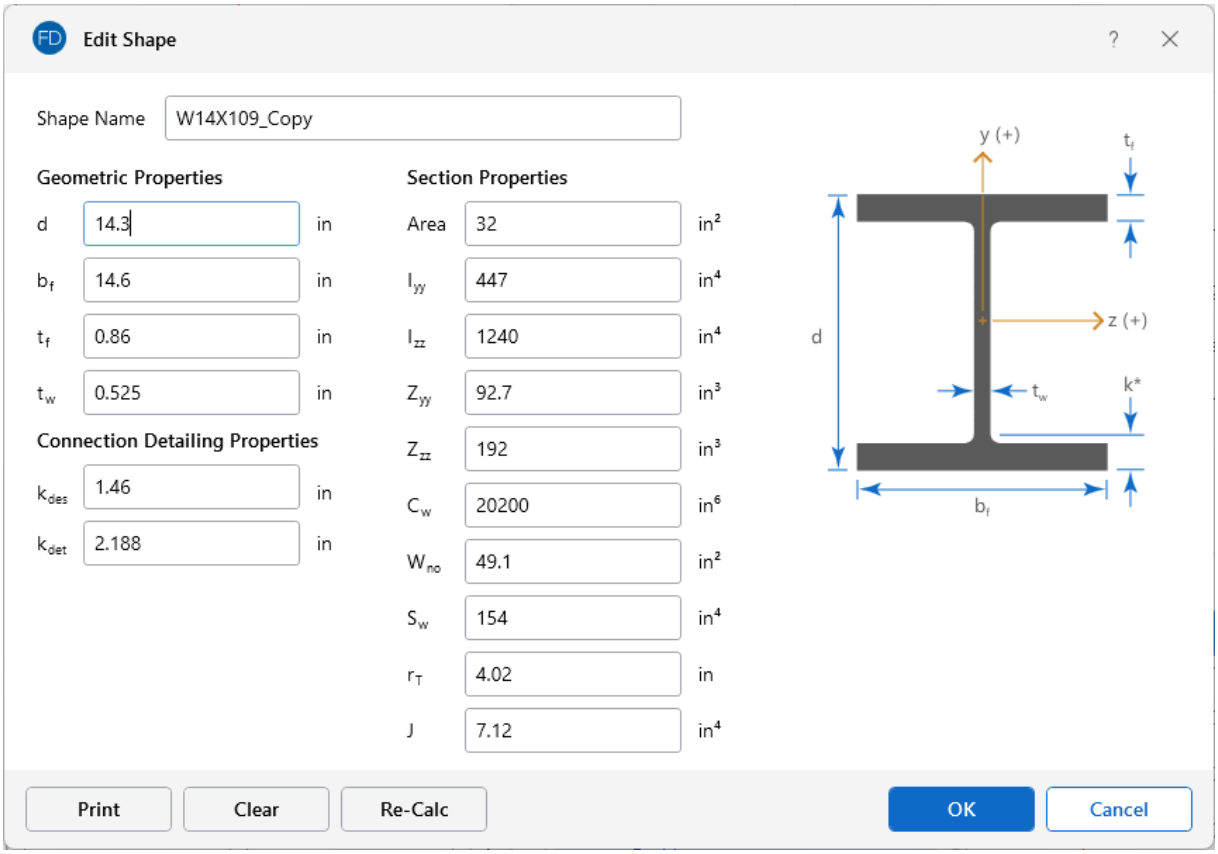
3. Click on the **Hot Rolled** tab.
4. From the **Shape Name** list, choose the shape you wish to modify and click **Edit**.
5. If the Edit Shape window opens stating that you cannot edit the default database shape, click **Copy**.



You will be able to edit the copy after you provide a new name, click the **Copy Shape** button, and answer the question as to whether or not to make the shape available for all models.

6. Select the copied shape which has been placed at the bottom of the **Shape Name** list and click the **Edit** button.

   The shape copy opens in the Edit Shape window, as shown in the following example.



7. Make whatever changes are needed by either:
   - Manually inputting the **Geometric Properties** and **Section Properties**.
   - Having the program calculate the **Section Properties** based on values you input for the **Geometric Properties**.

   Once an edit is made to the Geometric Properties or to both the properties, you'll need to run a recalculation.
8. Click the **Re-Calc** button to have the program calculate the new **Section Properties** automatically.
9. Click **OK** to save the newly calculated, edited shape.
10. Do one of the following in the Changes to Shape Database window that appears:
    - Click **Yes** if you want the changes to apply to all new models created on your computer.
    - Click **No** if you want the changes to apply ONLY to the current model.

Note:

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

Available Database Shapes

Wide Flange Database Shapes

For the AISC database, wide flange shapes are called out by the designation given to 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. To call a Trade Arbed W12X96, you 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). An 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 an "HSS" prefix and the dimensions are all called out in millimeters. British shapes use the prefix “SHS” for square tubes and “RHS” for rectangular tubes.

Note:

- The prefix for older AISC tube shapes is "TU". These shapes reflect the properties that were published with the older AISC 9th edition. As such, these shapes may only exist in the AISC_backup database.
- Tubes using the TU prefix will have a design wall thickness of the nominal wall thickness. The syntax is "TUdepthXwidthXthick", where "depth" is the tube depth, "width" is the tube width and "thick" is the tube thickness in number of 1/16ths. For example, TU16X12X8 would be a 16" deep, 12" wide tube with a thickness of 1/2" (8/16ths).

Pipe Database Shapes

Pipe shapes, which are hollow circular shapes, are entered as on-line shapes. The syntax for these shapes is "PIdiaXthick", where "dia" is the pipe outside diameter and "thick" is the pipe thickness (in inches or centimeters). For example (assuming US Standard units), PI10X.5 would be a 10" diameter pipe with a wall thickness of 1/2".

Channel and Double Channel Database Shapes

Channel shapes are entered with the "C" or "MC" prefix. For example, C15X50 would be a valid entry. For Canadian and British shapes, the depth is called out in millimeters and the mass per length is in kg/meter. Double channels are entered with the “2C” or “2MC” prefix.

Tee Database Shapes

The Tee shapes are entered with the "WT", "MT" or "ST" prefix. For example. WT15X74 would be a valid entry. For Canadian and British shapes, the depth is called out in millimeters and the mass per length is in kg/meters.

Double Angle Database Shapes

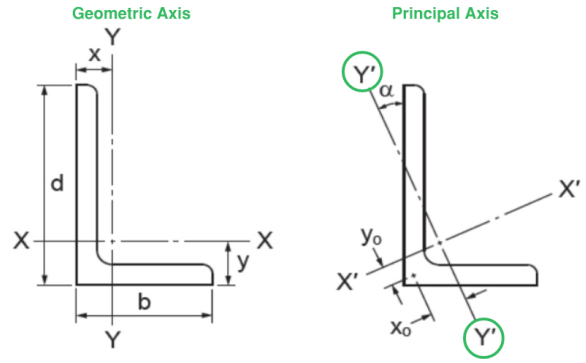
These shapes are entered with the prefix "LL". The syntax is "LLbackXflangeXthickXspace" where "back" is the back to back leg length, "flange" is the single angle flange leg length, "thick" is the angle thickness in number of 1/16ths and "space" is the space between the angles in 1/8ths. For example, "LL6X3.5X5X3" would be L6X3.5 angles that are 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

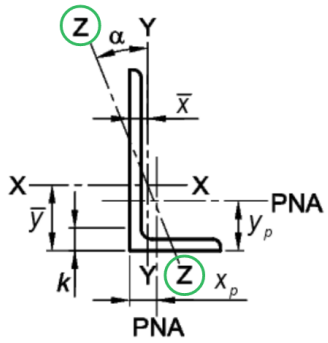
Angles are entered with an "L" prefix. The syntax is "LlongXshortXthick", where "long" is the long leg length, "short" is the short leg length and "thick" is the thickness, in number of 1/16ths. For example, L9X4X8 is a 9" by 4" angle that is 1/2" (8/16ths) thick. For the Canadian and British shapes, 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 CISC Handbook of Steel Construction - 12th Edition



Per AISC 360-22 (16th Edition)



Solid Rectangle Shapes

These shapes can be defined as on-line shapes. The syntax is "REhtXbase", where "ht" is the rectangle height and "base" is the rectangle base (in inches or cm). For example, RE10X4 would be a 10" deep, 4" width rectangular shape (assuming US Standard units). These shapes can also be defined in the Shape Editor. When defined in the Shape Editor, the depth of the solid rectangular section must always be greater than or equal to the width.

Solid Round Bar Shapes

These shapes are defined as on-line shapes. The syntax is "BARdia", where "dia" is the circle diameter. For example (assuming metric units), BAR2 would be a circular bar with a diameter of 2 cm.

Hot Rolled Steel - Databases

Shapes are organized by **Shape Type** and **Database / Manufacturer**. Common shapes are supported such as; wide flanges, tubes, pipes, channels, etc. You can type in the names directly, select shapes from these databases or add your own shapes.

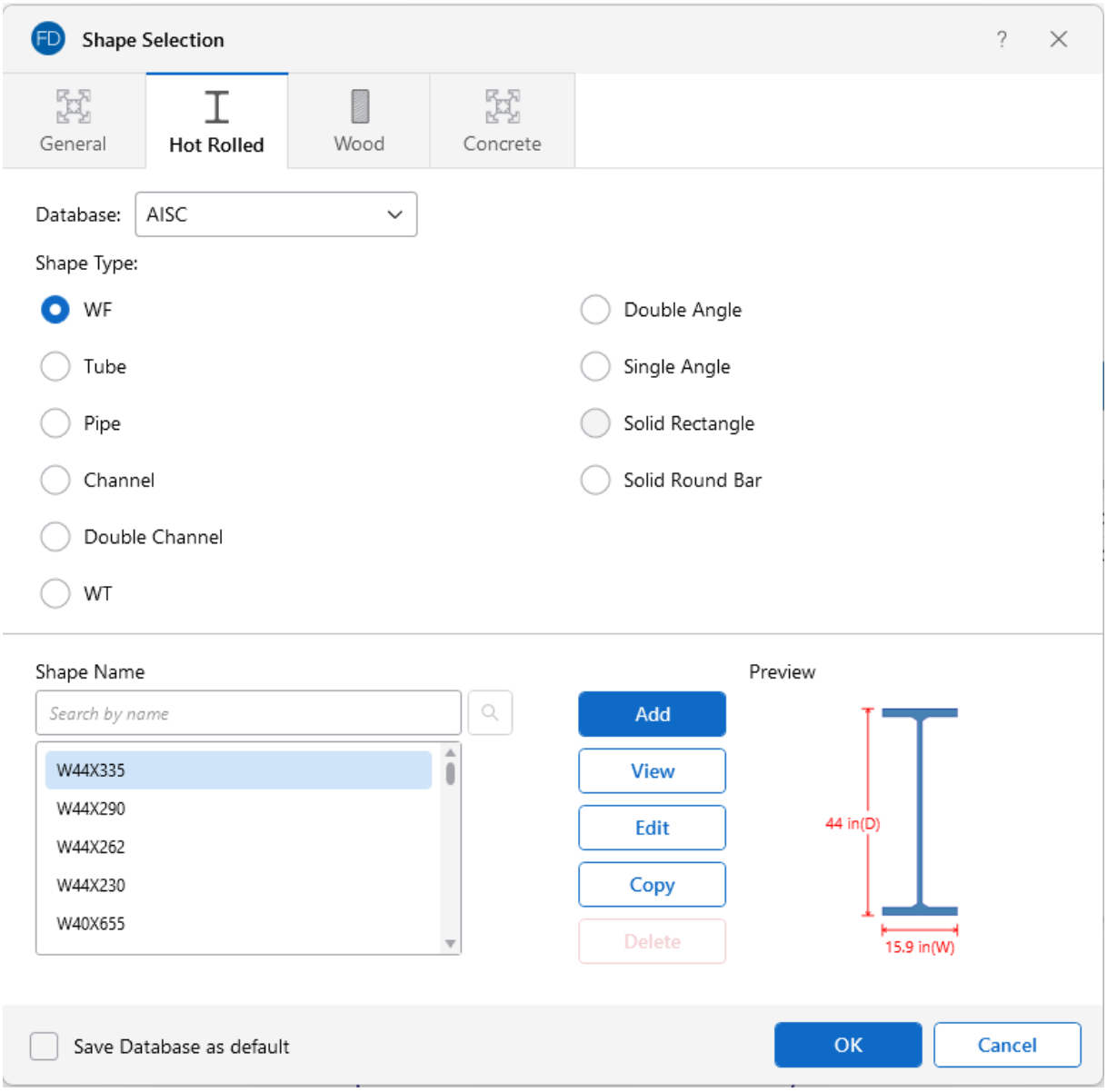
RISA currently supports the following common Hot Rolled steel databases: AISC (United States), Australian, British, Canadian, Chilean, Chinese, European, Indian, Korean and Mexican.

Note: Older AISC shapes, which are no longer part of the AISC database, are automatically moved/exported to a new AISC_backup database during installation or update.

Select a Database Shape

To select a Hot Rolled Steel database shape for pile elements:

1. Click the ellipsis button in the Shape column of the **Pile Definitions Spreadsheet** to open the **Shape Selection** dialog.
2. Click on the **Hot Rolled** tab.

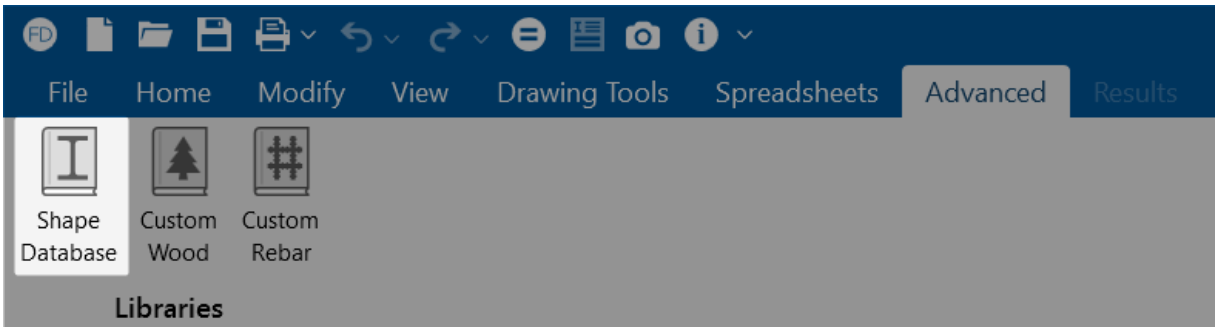


- Choose the appropriate **Database**.
- Choose the **Shape Type**. See Available Database Shapes for more information on shape types.
- In the **Shape Name** list, find and select the shape you want to use.
- Click **OK** to save your selection, close the window, and return to the spreadsheet.
- The chosen shape is displayed in the Shape cell of the spreadsheet.

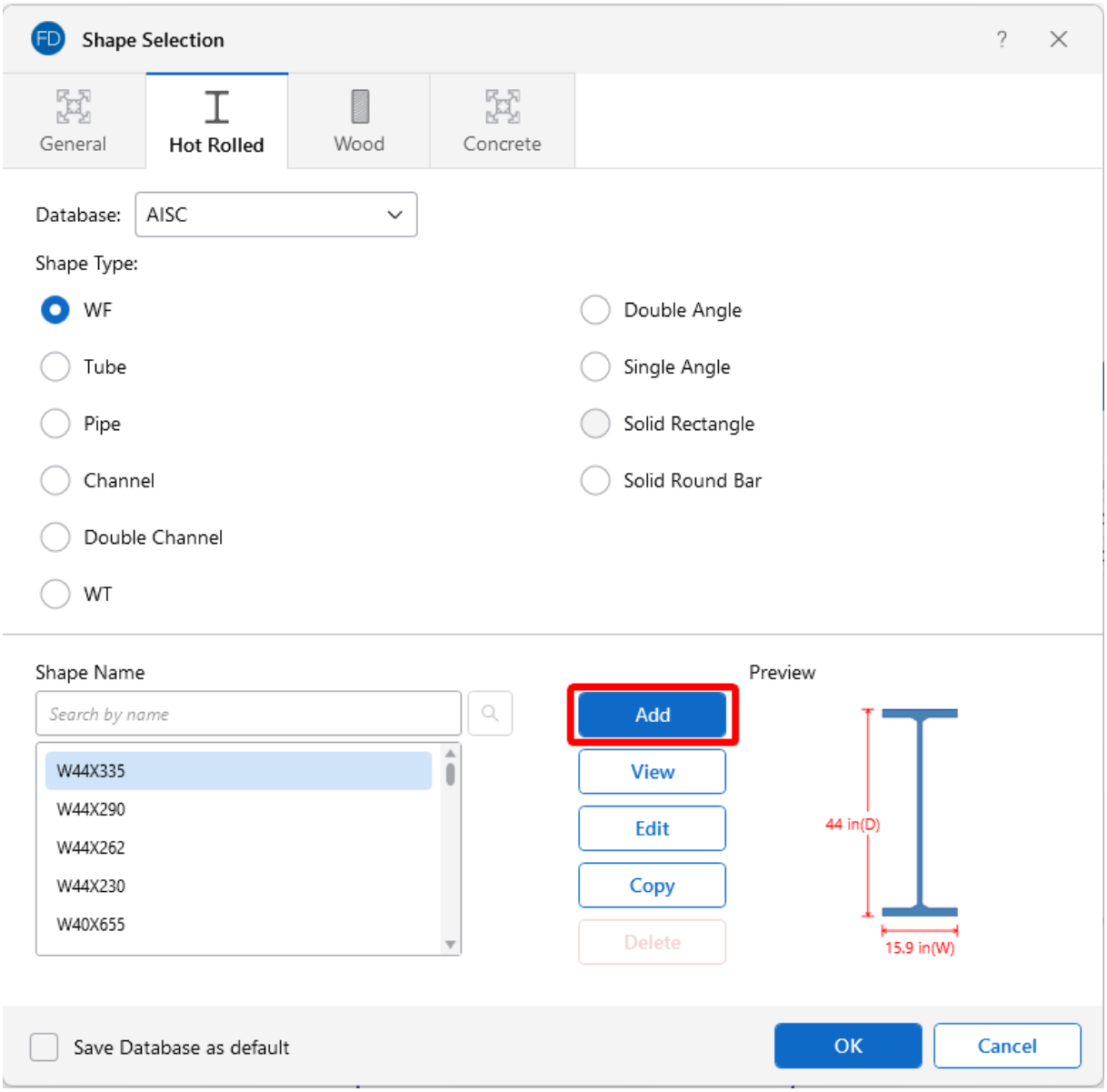
Add a Database Shape

To add a Hot Rolled Steel database shape:

1. On the **Advanced** ribbon, click the **Shape Database** icon.



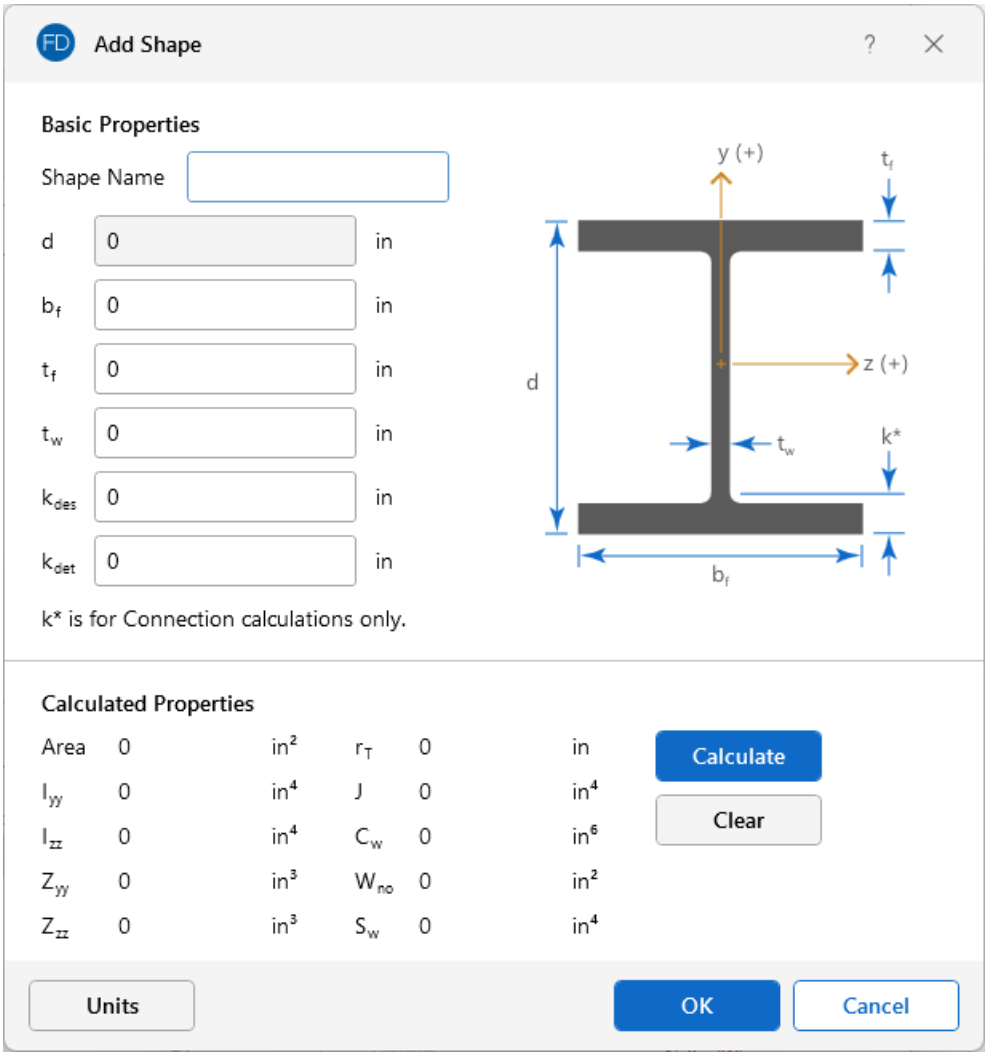
The Shape Selection window opens.



2. Click on the **Hot Rolled** tab.
3. Choose the **Database** and **Shape Type** you wish to add.

   See Available Database Shapes for more information on shape types.
4. Click the **Add** button.

   The Add Shape window opens.



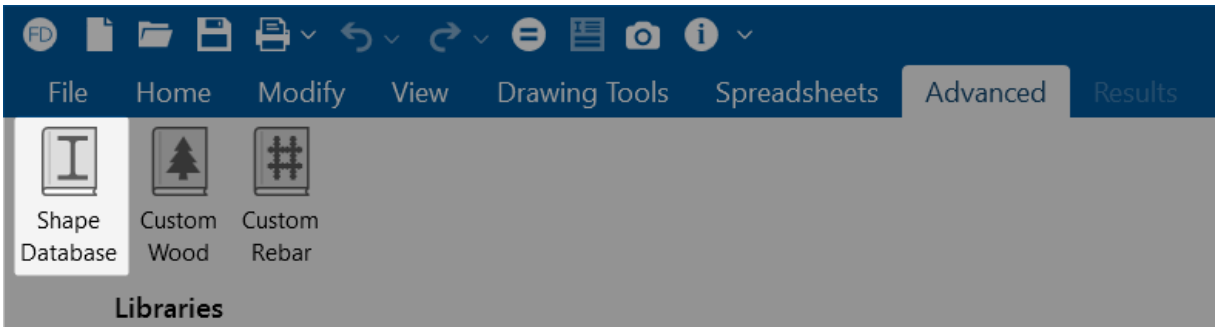
5. Under **Basic Properties**, specify a **Shape Name** for the shape and fill in the basic properties.
6. Click the **Calculate** button to determine the shape properties.
7. Click **OK** to accept the new shape and close the Add Shape window.

   The new shape is added to the bottom of the Shape Name list in the Shape Selection window.

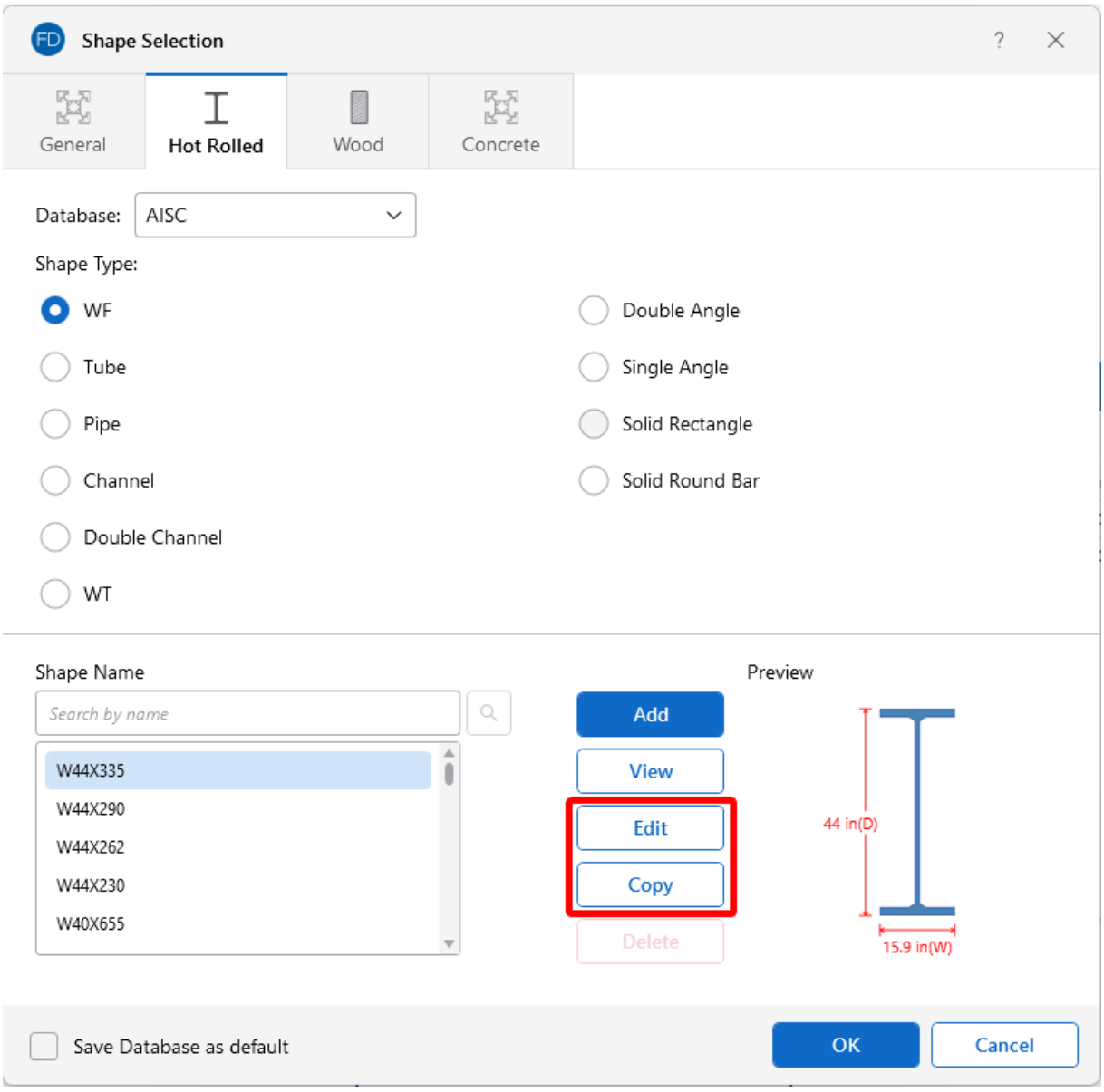
Edit a Database Shape

To edit a Hot Rolled Steel database shape:

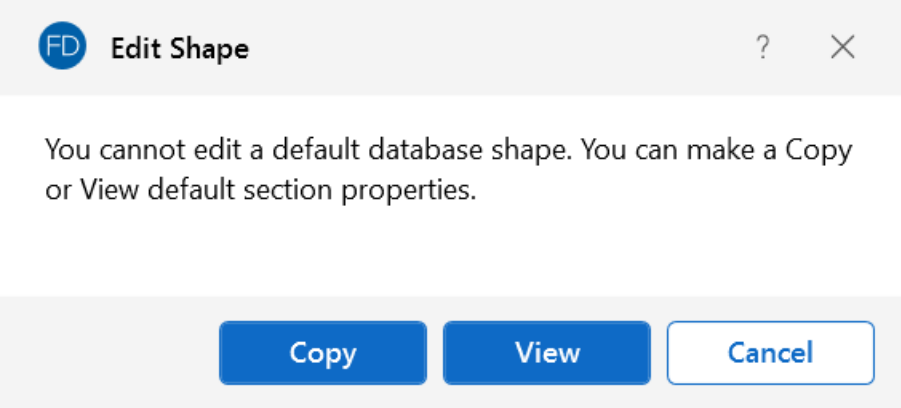
1. Go to the **Advanced** ribbon.
2. Click on the **Shape Database** icon in the **Libraries** section.



The Shape Selection window opens.



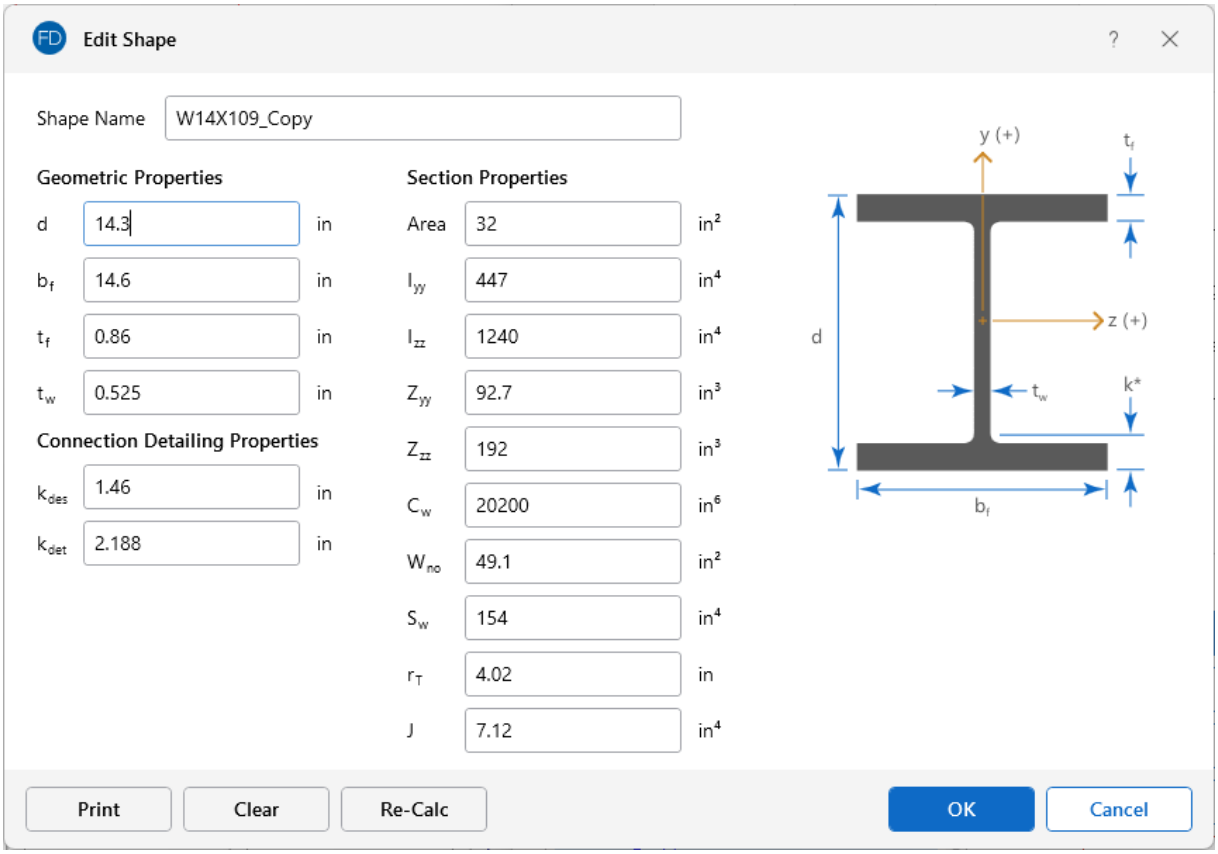
3. Click on the **Hot Rolled** tab.
4. From the **Shape Name** list, choose the shape you wish to modify and click **Edit**.
5. If the Edit Shape window opens stating that you cannot edit the default database shape, click **Copy**.



You will be able to edit the copy after you provide a new name, click the **Copy Shape** button, and answer the question as to whether or not to make the shape available for all models.

6. Select the copied shape which has been placed at the bottom of the **Shape Name** list and click the **Edit** button.

   The shape copy opens in the Edit Shape window, as shown in the following example.



7. Make whatever changes are needed by either:
   - Manually inputting the **Geometric Properties** and **Section Properties**.
   - Having the program calculate the **Section Properties** based on values you input for the **Geometric Properties**.

   Once an edit is made to the Geometric Properties or to both the properties, you'll need to run a recalculation.
8. Click the **Re-Calc** button to have the program calculate the new **Section Properties** automatically.
9. Click **OK** to save the newly calculated, edited shape.
10. Do one of the following in the Changes to Shape Database window that appears:
    - Click **Yes** if you want the changes to apply to all new models created on your computer.
    - Click **No** if you want the changes to apply ONLY to the current model.

Note:

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

Available Database Shapes

Wide Flange Database Shapes

For the AISC database, wide flange shapes are called out by the designation given to 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. To call a Trade Arbed W12X96, you 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). An 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 an "HSS" prefix and the dimensions are all called out in millimeters. British shapes use the prefix “SHS” for square tubes and “RHS” for rectangular tubes.

Note:

- The prefix for older AISC tube shapes is "TU". These shapes reflect the properties that were published with the older AISC 9th edition. As such, these shapes may only exist in the AISC_backup database.
- Tubes using the TU prefix will have a design wall thickness of the nominal wall thickness. The syntax is "TUdepthXwidthXthick", where "depth" is the tube depth, "width" is the tube width and "thick" is the tube thickness in number of 1/16ths. For example, TU16X12X8 would be a 16" deep, 12" wide tube with a thickness of 1/2" (8/16ths).

Pipe Database Shapes

Pipe shapes, which are hollow circular shapes, are entered as on-line shapes. The syntax for these shapes is "PIdiaXthick", where "dia" is the pipe outside diameter and "thick" is the pipe thickness (in inches or centimeters). For example (assuming US Standard units), PI10X.5 would be a 10" diameter pipe with a wall thickness of 1/2".

Channel and Double Channel Database Shapes

Channel shapes are entered with the "C" or "MC" prefix. For example, C15X50 would be a valid entry. For Canadian and British shapes, the depth is called out in millimeters and the mass per length is in kg/meter. Double channels are entered with the “2C” or “2MC” prefix.

Tee Database Shapes

The Tee shapes are entered with the "WT", "MT" or "ST" prefix. For example. WT15X74 would be a valid entry. For Canadian and British shapes, the depth is called out in millimeters and the mass per length is in kg/meters.

Double Angle Database Shapes

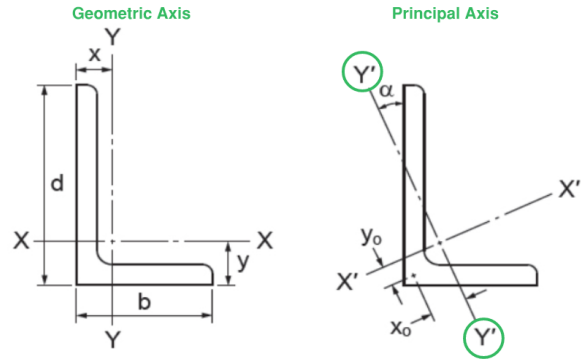
These shapes are entered with the prefix "LL". The syntax is "LLbackXflangeXthickXspace" where "back" is the back to back leg length, "flange" is the single angle flange leg length, "thick" is the angle thickness in number of 1/16ths and "space" is the space between the angles in 1/8ths. For example, "LL6X3.5X5X3" would be L6X3.5 angles that are 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

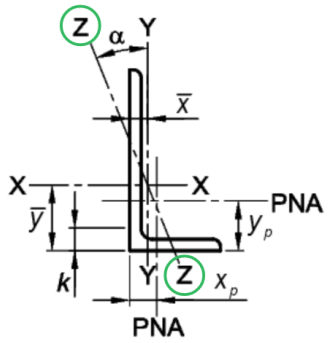
Angles are entered with an "L" prefix. The syntax is "LlongXshortXthick", where "long" is the long leg length, "short" is the short leg length and "thick" is the thickness, in number of 1/16ths. For example, L9X4X8 is a 9" by 4" angle that is 1/2" (8/16ths) thick. For the Canadian and British shapes, 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 CISC Handbook of Steel Construction - 12th Edition



Per AISC 360-22 (16th Edition)



Solid Rectangle Shapes

These shapes can be defined as on-line shapes. The syntax is "REhtXbase", where "ht" is the rectangle height and "base" is the rectangle base (in inches or cm). For example, RE10X4 would be a 10" deep, 4" width rectangular shape (assuming US Standard units). These shapes can also be defined in the Shape Editor. When defined in the Shape Editor, the depth of the solid rectangular section must always be greater than or equal to the width.

Solid Round Bar Shapes

These shapes are defined as on-line shapes. The syntax is "BARdia", where "dia" is the circle diameter. For example (assuming metric units), BAR2 would be a circular bar with a diameter of 2 cm.

Loads

As you define the loads in the model, you will notice that each individual instance of a load type must be assigned to a particular **Load Category**. You will then use these load categories to define the various **Load Combinations** that are to be solved when a solution is performed. For more information, see Loads - Load Combinations.

You may view the loads in your model for each load category or load combination. This is an excellent way to verify the loads applied to the model. See Model View Settings - Loads to learn to do this.

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 be used when loading the model. Each of the discrete load types is described in it's own section:

- Loads - Area Loads
- Loads - Distributed Loads
- Loads - Nodal Loads

When loads are defined in RISAFoundation they are automatically grouped based on their Load Categories. These are the basic building blocks of the final load combinations applied to the structure. For more information, see Load Categories.

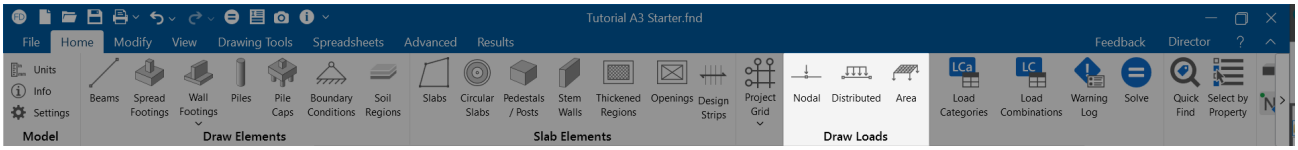
Self Weight (Gravity Load)

The **Self Weight** of slabs and beams is automatically calculated and included as part of the Dead Load Category (DL).

The 'Self Weight' is applied as a uniform distributed load along each beam and as an area load for the slab. The magnitude of the beam loads is the cross-sectional area times the material weight density.

Note: If you do not want the program to apply a member's self weight, create that member using a weightless material. For more information, see Material Properties.

Draw Loads



You can graphically apply loads to the model. You can apply one load at a time by clicking the mouse, or apply loads to entire selections at once. All of the graphical loading tools can be found in the **Draw Loads** section of the **Home** ribbon.

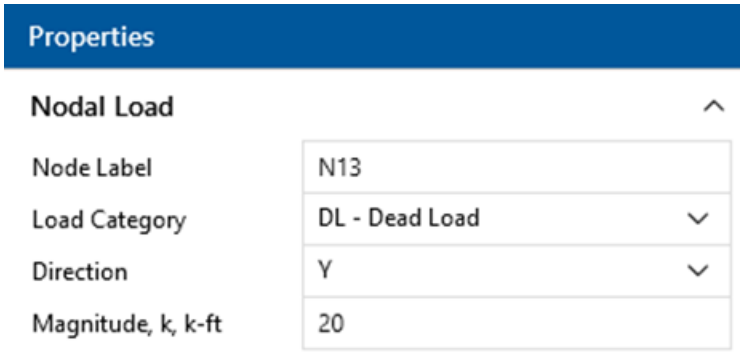
Modify Loads

Modifying loads can be done graphically or in the spreadsheets. In the **Explorer** panel, click on the spreadsheet that you wish to modify. You can also go to the **Spreadsheets** ribbon, click the **Data Entry** icon and then choose the spreadsheet you want to modify. You can then move through different Load Categories by using the drop-down list on the **Window Toolbar**.

See Spreadsheet Operations to learn how you can modify the loads in a spreadsheet.

Alternatively, you can directly click on the loads in the 3D View and edit the selected load within the **Properties Panel**.

From the **Properties Panel**, you can change the configuration of the load, as well as edit the load category by selecting the desired load category from the drop-down menu.



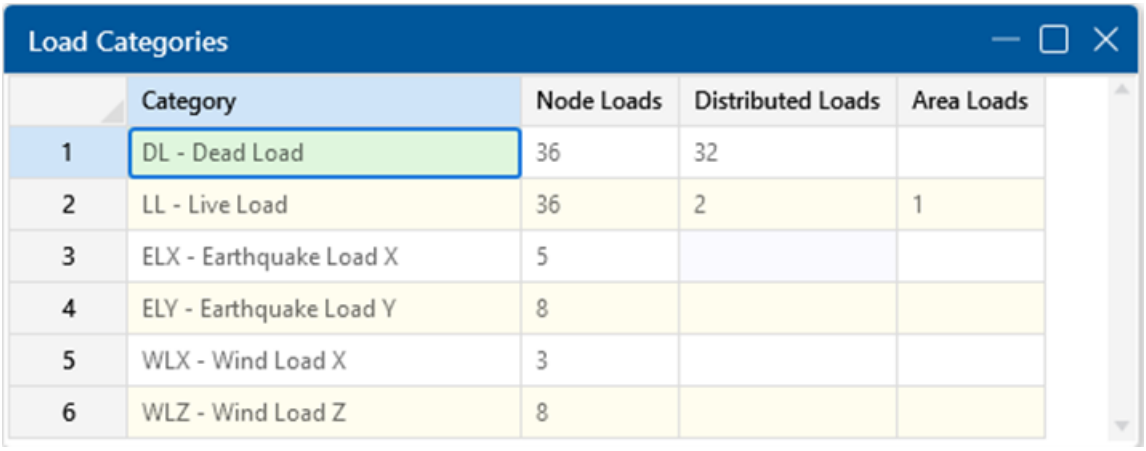
Delete Loads

Deleting loads can be done graphically or in the spreadsheets. In the **Explorer** panel, click on the spreadsheet that you wish to modify. You can also go to the **Spreadsheets** ribbon, click the **Data Entry** icon and then choose the spreadsheet you want to modify. You can then move through different Load Categories by using the drop-down list on the **Window Toolbar**. You can use spreadsheet operations to delete the loads as well.

Alternatively, you can directly click on the loads in the 3D View and press the **Delete** key.

Loads - Load Categories

Information about Basic Load Cases (BLC's) is recorded on the **Load Categories Spreadsheet** and the loads themselves are recorded in the load spreadsheets for each load type. The Load Categories Spreadsheet may be accessed by selecting **Load Categories** from the **Data Entry** portion of the **Explorer** panel.



The first column in this spreadsheet merely displays the Category for which the rest of the information in the row relates to. The remaining columns display the quantity of each type of load that is contained in that category. You may not edit these values, but you may click on the quantities to open the spreadsheet and view the loads that they represent. For example, the figure above has 36 node loads and 32 distributed loads part of the DL-Dead Load category. Clicking on the number 36 with the mouse will open the **Nodal Loads Spreadsheet,** whereas clicking on the 32 will open the **Distributed Loads Spreadsheet**.

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.

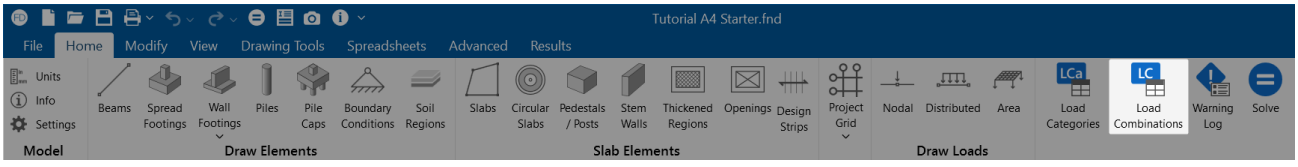
Load Category	Description
DL	Dead Load
LL	Live Load
EL	Earthquake Load
WL	Wind Load
SL	Snow Load
RLL	Roof Live Load
LLS	Live Load Special (public assembly, garage, storage, etc.)

Load Category	Description
TL	Long Term Load (creep, shrinkage, settlement, thermal, etc.)
SLN	Snow Load Non-shedding
HL	Hydrostatic Load
FL	Fluid Pressure Load
RL	Rain Load
PL	Ponding Load
EPL	Earth Pressure Load
IL	Impact Load
OL1 - OL100	Other Load 1 - 100 (generic)
ELX, ELY, ELZ	Earthquake Load along global X-axis, Y-axis, Z-axis
WLX, WLY, WLZ	Wind Load along global X-axis, Y-axis, Z-axis
WL+X, WL+Y, WL+Z	Wind Load along positive global X-axis, Y-axis, Z-axis
WL-X, WL-Y, WL-Z	Wind Load along negative global X-axis, Y-axis, Z-axis
WLXP1, WLYP1, WLZP1	Partial Wind Load 1 along global X-axis, Y-axis, Z-axis
WLXP2,	Partial Wind Load 2 along global X-axis, Y-axis, Z-

Load Category	Description
WLYP2, WLZP2	axis
ELX+Z, ELX+Y	Eccentric Earthquake Load along global X-axis shifted along positive global Z-axis, Y-axis
ELX-Z, ELX-Y	Eccentric Earthquake Load along global X-axis shifted along negative global Z-axis, Y-axis
ELZ+X, ELZ+Y	Eccentric Earthquake Load along global Z-axis shifted along positive global X-axis, Y-axis
ELZ-X, ELZ-Y	Eccentric Earthquake Load along global Z-axis shifted along negative global X-axis, Y-axis
ELY+X, ELY+Z	Eccentric Earthquake Load along global Y-axis shifted along positive global X-axis, Z-axis
ELY-X, ELY-Z	Eccentric Earthquake Load along global Y-axis shifted along negative global X-axis, Z-axis
SX, SY, SZ	Response Spectra Results (from Risa-3D)
NLX, NLY, NLZ	General notional load along global X-axis, Y-axis, Z-axis

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. To learn how to solve load combinations, see Solution.



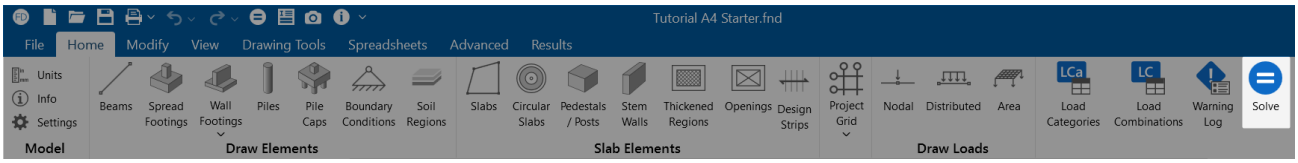
Add Load Combinations Manually

To add Load Combinations manually:

1. From the **Home** ribbon, click on **Load Combinations**.
2. Enter load combinations by pairing loads in the **Category** fields with factors in the **Factor** fields.

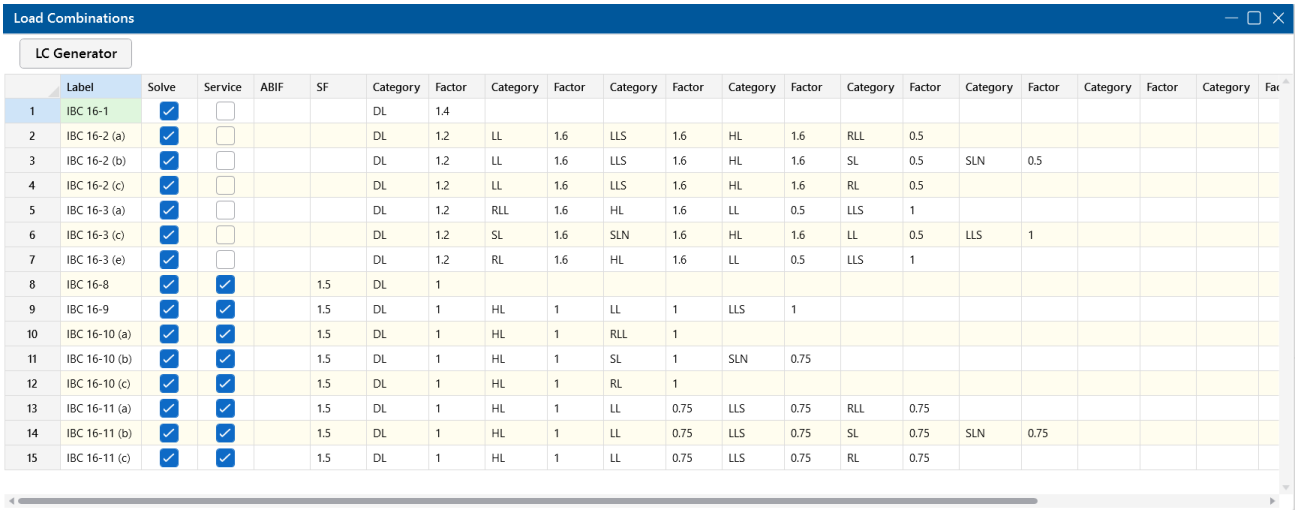
Solve Load Combinations

To solve load combinations, click the **Solve** button in the **Home** ribbon.



Load Combinations Spreadsheet

The **Load Combinations Spreadsheet** records the combinations of loads for solution. Access this spreadsheet by choosing **Load Combinations** from the **Explorer** panel under **Data Entry**.



	Label	Solve	Service	ABIF	SF	Category	Factor	Category	Factor	Category	Factor	Category	Factor	Category	Factor	Category	Factor	Category	Factor	Category	Factor	Category	Fac
1	IBC 16-1	☑	☐			DL	1.4																
2	IBC 16-2 (a)	☑	☐			DL	1.2	LL	1.6	LLS	1.6	HL	1.6	RLL	0.5								
3	IBC 16-2 (b)	☑	☐			DL	1.2	LL	1.6	LLS	1.6	HL	1.6	SL	0.5	SLN	0.5						
4	IBC 16-2 (c)	☑	☐			DL	1.2	LL	1.6	LLS	1.6	HL	1.6	RL	0.5								
5	IBC 16-3 (a)	☑	☐			DL	1.2	RLL	1.6	HL	1.6	LL	0.5	LLS	1								
6	IBC 16-3 (c)	☑	☐			DL	1.2	SL	1.6	SLN	1.6	HL	1.6	LL	0.5	LLS	1						
7	IBC 16-3 (e)	☑	☐			DL	1.2	RL	1.6	HL	1.6	LL	0.5	LLS	1								
8	IBC 16-8	☑	☑		1.5	DL	1																
9	IBC 16-9	☑	☑		1.5	DL	1	HL	1	LL	1	LLS	1										
10	IBC 16-10 (a)	☑	☑		1.5	DL	1	HL	1	RLL	1												
11	IBC 16-10 (b)	☑	☑		1.5	DL	1	HL	1	SL	1	SLN	0.75										
12	IBC 16-10 (c)	☑	☑		1.5	DL	1	HL	1	RL	1												
13	IBC 16-11 (a)	☑	☑		1.5	DL	1	HL	1	LL	0.75	LLS	0.75	RLL	0.75								
14	IBC 16-11 (b)	☑	☑		1.5	DL	1	HL	1	LL	0.75	LLS	0.75	SL	0.75	SLN	0.75						
15	IBC 16-11 (c)	☑	☑		1.5	DL	1	HL	1	LL	0.75	LLS	0.75	RL	0.75								

The first column, **Label**, is strictly for your reference. Enter any descriptive label you wish and it will be displayed with the results when the combination is solved.

The next four columns are for options that apply to each load combination.

The **Solve** check box is used to indicate which combinations should be included in the solution. For more information, see Solution.

Check the **Service** box if the load combination should be used for the calculation of service level soil bearing and footing stability checks. Load combinations for which the **Service** box is NOT checked will be used for the strength design of slab, beam and footing reinforcement.

The **Service** check box is also used for masonry wall footings, depending on which masonry code is selected. If the ASD code is selected, then only combinations with the check box checked will be used for the masonry wall design. If the Strength code is selected, then only combinations with the check box unchecked will be used for masonry wall design.

The **ABIF** column is used to set the allowable bearing increase factor used for the soil bearing check in slab, spread, and wall footing design. If this field is left blank, a default of '1.0' will be used. For more information, see RISAFoundation Design Combinations.

The **SF** column is used as the value with which overturning and sliding of spread footings and retaining walls are checked against. This value is also used in the slab overturning check. This value is only used in service load cases. If the results do not meet the safety factor, they will be denoted as failing with red text.

The next ten pairs of columns (**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. The following are the entries you can use in the **Category** field:

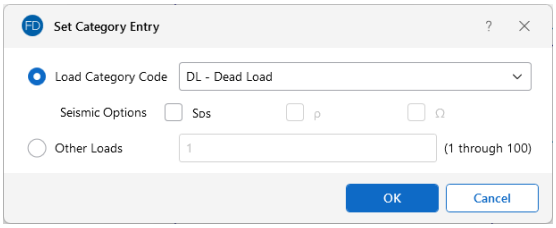
Entry	Description
Category	Enter a category code such as DL, LL, etc to include all loads in that category
Lnn	Enter “Lnn” to nest the loads from another combination, where “nn” is the number of the other combination, i.e. “L3" means include all loads from load combination 3.
Sn	Includes response spectra results for the global direction “n”, I.e. enter “SX” to include response spectra results calculated in the global X direction, “SY” for global Y , etc.

There are also a number of Category Prefixes that can be included in your load combinations. You can have these generated by the LC Generator, or you can click on the ... button in the Category cell to select them manually. These are defined in the Seismic tab of the Model Settings and are explained below:

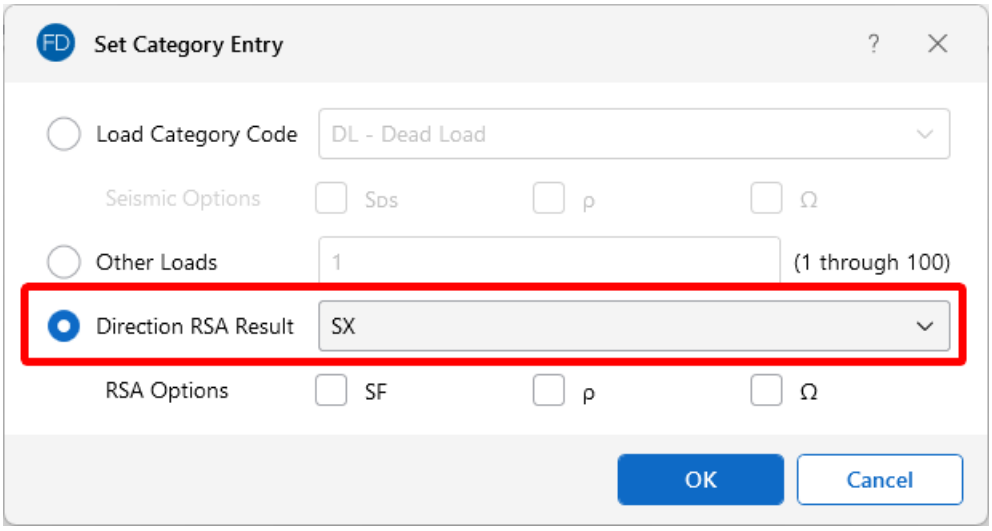
Prefix	Description
SF	Scaling Factor - applied to RSA results
ρ (Rho)	Redundancy factor - applied to EL loads for seismic load combinations (or RSA results) Note: If ELX and ELZ are used, the ρ from the X and Z direction will be used directionally. If EL is used, the ρ from the X

Prefix	Description
	direction will be used and the Z direction will be disregarded. If ELY is used, the ρ from the Z direction will be used and the X direction will be disregarded.
SDS	Spectral Response Acceleration Parameter for Short Periods - applied to DL for seismic load combinations
Ω (Om)	Overstrength Factor - applied to EL loads for seismic load combinations (or RSA results) Note: If ELX and ELZ are used, the Ω from the X and Z direction will be used directionally. If EL is used, the Ω from the X direction will be used and the Z direction will be disregarded. If ELY is used, the Ω from the Z direction will be used and the X direction will be disregarded.

You can also use the ··· (ellipsis) button in the Category cell to help you specify the loads. Choose the load category, seismic options, or you can quickly specify an Other Load category by selecting the **Other Loads** option and the number of the Other Load (i.e. 3 for OL3 or 16 for OL16).

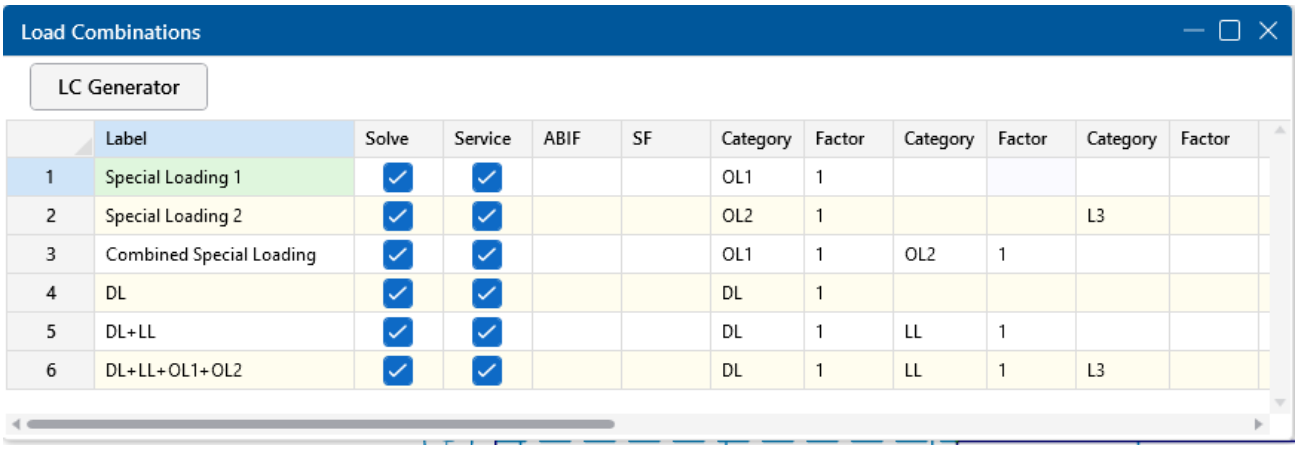


When transferring from RISA-3D with RSA results, you will have additional RSA options, as shown below:



Nesting Load Combinations

You are allowed only 10 categories per load combination, which may not always be enough. For this reason, you can define "combinations of load combinations". This means if you need more than 10 categories in a single combination, you can define the needed categories over several load combinations and then pull these combinations together into a single load combination.



	Label	Solve	Service	ABIF	SF	Category	Factor	Category	Factor	Category	Factor
1	Special Loading 1	✓	✓			OL1	1				
2	Special Loading 2	✓	✓			OL2	1			L3	
3	Combined Special Loading	✓	✓			OL1	1	OL2	1		
4	DL	✓	✓			DL	1				
5	DL+LL	✓	✓			DL	1	LL	1		
6	DL+LL+OL1+OL2	✓	✓			DL	1	LL	1	L3	

To use a nested load combination within a specific load combination, simply type in the prefix “L” with the load combination number in the category column. For example, say Load Combination 6 has “L3” entered for one of its Categories. This will include all the categories (with their factors) entered in Load Combination 3 as part of Load Combination 6. The flags for Load Combination 3 (i.e. Solve, Service, ABIF and SF entries) apply only to Load Combination 3 and will not be used when Load Combination 6 is solved.

Also, the factor we enter with the L# entry will be applied to the Category factors entered for the L#. Thus, if we enter “L1” with a factor of 0.9, we’re including 90% of the Category and corresponding factors in Load Combination 1.

Note:

- These “combinations of load combinations” can only be nested to one level; i.e. the load combinations referenced with the **L#** entries may not themselves have **L#** entries within them.
- Nested load combinations cannot contain duplicate categories within a single load combination.

RISAFoundation Design Combinations

RISAFoundation recognizes two types of combinations: **Service** and **Design**. Service combinations are used to determine if the soil bearing capacity, overturning and sliding resistance are adequate. Design combinations are used to design the slabs, beams, footings and pedestals for flexure and shear per the chosen code.

The **ABIF** column stands for "Allowable Bearing Increase Factor". For transient loads, such as wind or seismic loads, you may want to specify an allowable bearing increase factor (ABIF) to increase the allowable soil bearing pressure for that combination. The increase factor is typically 1.333 (a 1/3 increase).

Service Combinations

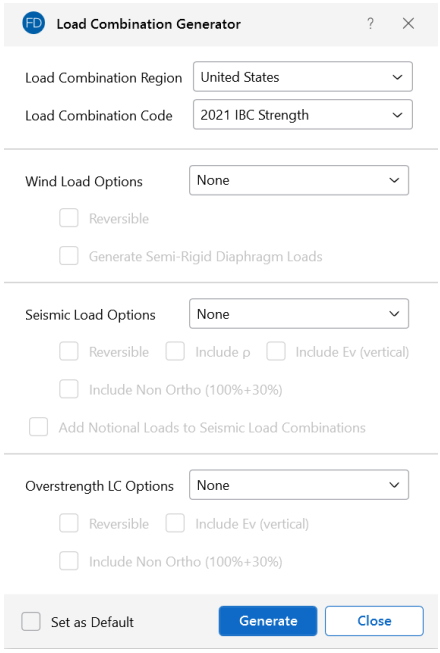
The service combinations are used to calculate soil bearing for comparison with the allowable soil bearing defined in the**Soil Regions Spreadsheet**. They are also used to calculate sliding and overturning moment safety factors.

Design Combinations

The design combinations are used to calculate required reinforcing steel and to check both shear and bearing on concrete footings and slabs. They are also used to design flexural reinforcement and perform shear checks for pile caps, retaining walls and grade beams.

Generate Building Code Combinations

Major portions of the load combinations that are specified by building codes are included and can be applied to the model for solution. These combinations can be inserted by selecting the LC Generator button from the top of the **Load Combinations Spreadsheet**. This opens the **Load Combination Generator** window, shown in the following image.



The **Region** refers to the various regions supported by the program (U.S., Canada, India, British, etc).

The **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.

Wind Load Options

The **Wind Load Options** specify how detailed the generated wind load combinations should be.

When **None** is selected as the wind load option, the program will not generate any Load Combinations that include wind load categories.

The **2D Only** option generates only the most basic wind load category (WL).

The **X and Z** option generates separate wind load combinations for each horizontal direction (WLX and WLZ when Y is set at the vertical axis).

The **X and Z w/ Ecc** option generates all possible wind load combinations that include partial / eccentric wind loading (WLX, WLXP1, WLXP2, etc.) per Case 2 from Figure 27.3-8 in the ASCE 7-22.

The **X and Z w/ Ecc, Quart** option generates all possible wind load combinations per Cases 2, 3 and 4 from Figure 27.3-8 in the ASCE 7-22. For additional advice on this topic, please see the RISA Tips & Tricks website: www.risa.com/post/support. Type in Search keyword: **Quartering**.

The **Reversible** option generates two combinations for each wind load, one with positive load factors and one with negative load factors.

The **Generate Semi-Rigid Diaphragm Loads** option generates wind loads for Load Categories using windward and leeward direction prescribed by the ASCE 7-22 (WLX+Z, WLX-Z, WLZ+X, WLZ-X).

Seismic Load Options

The **SeismicLoad Options** specify how complex the generated seismic load combinations should be.

When **None** is selected as the seismic load option, the program will not generate any Load Combinations that include the EL load category.

The **2D Only** option is used to indicate that only the most basic seismic load category (EL) will be used.

The **X and Z** option is used to indicate that the program should generate separate seismic load combinations for each horizontal direction (ELX and ELZ).

The **X and Z w/ Eccentric** option is used to indicate that the program should generate all possible seismic load combinations that include eccentricities (ELX, ELX+Z, ELX-Z, etc).

The **X and Z RSA** option will use the spectral seismic loads (SX, SY, SZ) instead of the regular earthquake loading (ELX, ELY, ELZ).

If you have selected the X and Z, X and Z w/ Eccentric, or X and Z RSA option, you will also have the ability to include non-orthogonal seismic loads. Simply check the **Include Non Ortho (100% + 30%)** check box to include these loads per the Orthogonal Combination Procedure of section 12.5.1.2 of the ASCE 7-22.

When the **Reversible** box is checked, the program will generate every seismic load combination twice. Once with a positive load factor and once with a negative load factor.

Check the **Include ρ** check box if you want to include the redundancy factor in your seismic load combinations. This factor is set in the Seismic tab of Model Settings.

Check the **Include Ev (vertical)** check box if you want to include the vertical seismic load effect in your seismic load combinations. This value is calculated per equation 12.4-4a of the ASCE 7-22 using the S_{DS} value set in the Seismic tab of Model Settings.

Check the **Add Notional Loads to Seismic Load Combinations** check box if you want to include notional loads applied as lateral seismic loads per the ASCE 7 and AISC codes.

Overstrength LC Options

The overstrength LC options are similar to the seismic load options mentioned above. You may select the option from the drop-down list that corresponds with the seismic load options for which you would like the overstrength factor applied.

Load Combination Generator - General Notes

The following loads are not generally included in the standard combinations but can be added by editing the combinations in the spreadsheet or by modifying the source document itself. For more information, see Customizing the Load Combination Generator below:

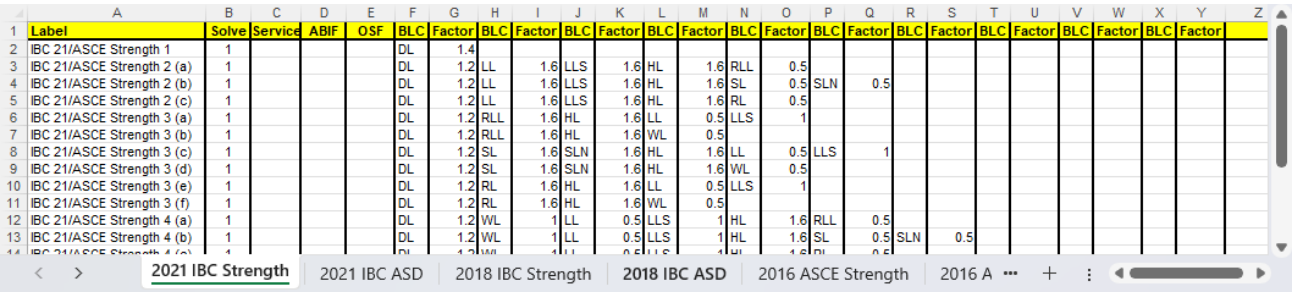
Category Code	Description
SX, SY, SZ	Response Spectra Results
TL	Long Term Load
HL	Hydrostatic Load
FL	Fluid Pressure Load
PL	Ponding Load Category
EPL	Earth Pressure Load
OL#	Other Load Categories

Note:

- The standard combinations are made up of Load Categories and Factors. Loads that are not assigned to these categories will not be included in the combinations upon solution.
- Some load categories do not occur in all of the design codes. Loads placed in categories that are not part of the standard combinations will not be included in the solution of these combinations.
- See Inserting, Deleting and Clearing Cells for quick ways to edit or add combinations.

Customize the Load Combination Generator

The Load Combinations for each region are contained in an XML file which can be opened and edited using a standard spreadsheet program. An example of 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	W	X	Y	Z
1	Label	Solve	Service	ABIF	OSF	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	
2	IBC 21/ASCE Strength 1	1				DL	1.4																			
3	IBC 21/ASCE Strength 2 (a)	1				DL	1.2	LL	1.6	LLS	1.6	HL	1.6	RLL	0.5											
4	IBC 21/ASCE Strength 2 (b)	1				DL	1.2	LL	1.6	LLS	1.6	HL	1.6	SL	0.5	SLN	0.5									
5	IBC 21/ASCE Strength 2 (c)	1				DL	1.2	LL	1.6	LLS	1.6	HL	1.6	RL	0.5											
6	IBC 21/ASCE Strength 3 (a)	1				DL	1.2	RLL	1.6	HL	1.6	LL	0.5	LLS	1											
7	IBC 21/ASCE Strength 3 (b)	1				DL	1.2	RLL	1.6	HL	1.6	WL	0.5													
8	IBC 21/ASCE Strength 3 (c)	1				DL	1.2	SL	1.6	SLN	1.6	HL	1.6	LL	0.5	LLS	1									
9	IBC 21/ASCE Strength 3 (d)	1				DL	1.2	SL	1.6	SLN	1.6	HL	1.6	WL	0.5											
10	IBC 21/ASCE Strength 3 (e)	1				DL	1.2	RL	1.6	HL	1.6	LL	0.5	LLS	1											
11	IBC 21/ASCE Strength 3 (f)	1				DL	1.2	RL	1.6	HL	1.6	WL	0.5													
12	IBC 21/ASCE Strength 4 (a)	1				DL	1.2	WL	1	LL	0.5	LLS	1	HL	1.6	RLL	0.5									
13	IBC 21/ASCE Strength 4 (b)	1				DL	1.2	WL	1	LL	0.5	LLS	1	HL	1.6	SL	0.5	SLN	0.5							

2021 IBC Strength | 2021 IBC ASD | 2018 IBC Strength | 2018 IBC ASD | 2016 ASCE Strength | 2016 A

The name of the file itself becomes the name of the Load Combination Region that appears in the Load Combination Generator. Each XML file has a series of "worksheets" and the name of each worksheet will be the name of the Load Combination Code that appears in the Load Combination Generator. You can add, edit or modify these documents to completely customize the available load combinations.

Note: When the program is updated for new versions, the existing XML files will be "backed up" and saved with a '.bak' extension, while the new XML files in the update will replace them. If any customizations were made to the XML files, they can be retrieved from the back ups.

The first row of each sheet is reserved for the column headers. The recognized column headers are as follows: Label, Solve, Service, ABIF, and OSF. In addition, there will be multiple pairs of Category & Factor headers.

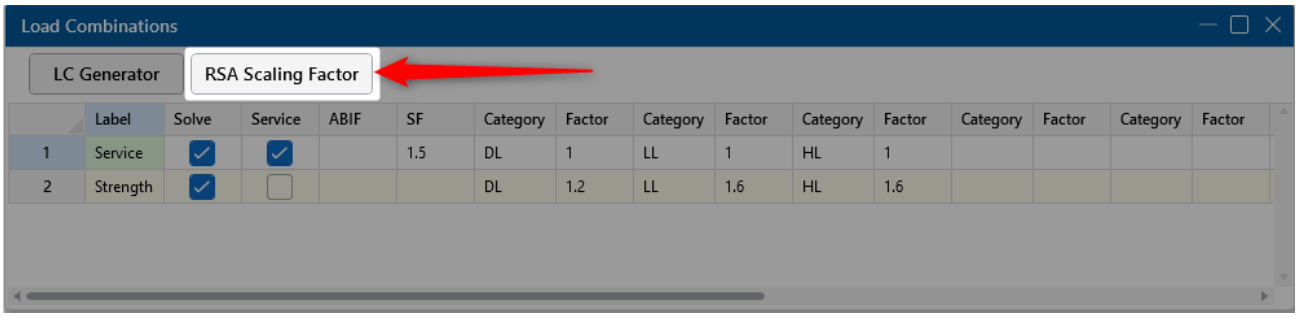
Other than Label and the Category / Factor pairs, you can omit columns. If a column is omitted, the default values will be used for those entries. The order of the column labels is optional except for pairs of Category and Factor. For the program to pair the Factor with the correct Category, the user should always provide the Category label PRIOR to the corresponding Factor. You can insert blank columns or columns with other labels than described above. In those cases, the program omits those undefined columns.

Note:

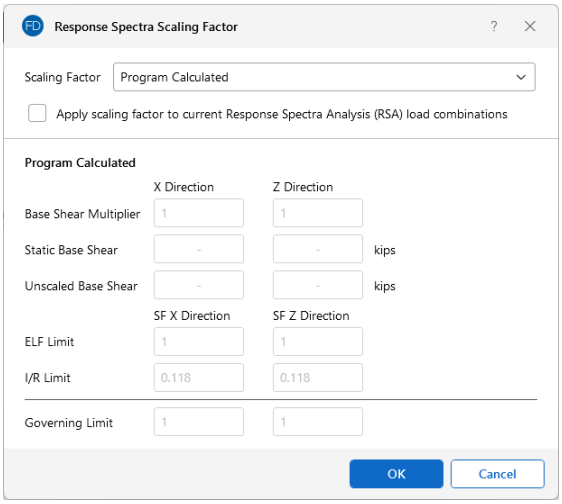
- The wind and seismic loads should be entered as WL and EL in order for them to be "expanded" using the Wind Load Options and Seismic Load Options.
- The program will read these files from the directory specified on the **File Locations** tab of the RISAFoundation **Application Settings**. For more information, see Customizing RISA.
- SX, SY and SZ are not valid Load Categories that can be used within the **Load Combination Generator Spreadsheet**. A response spectra analysis must be done first within RISA-3D.

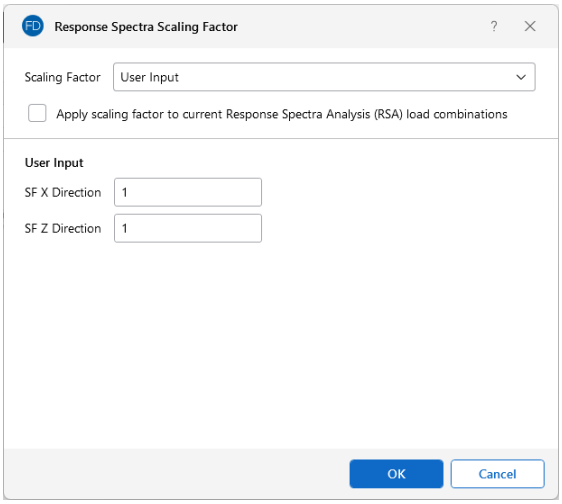
RSA Scaling Factor

When RSA reactions are transferred from RISA-3D, you will have the ability to access the **Scaling Factor** dialog. To access this dialog, first open the Load Combinations Spreadsheet and then click the **RSA Scaling Factor** button.



In this dialog, you will see the scaling factor along with corresponding unscaled base shear and static base shear that was calculated in RISA-3D.





Base Shears

The **Base Shears** section of the **Spectra Scaling Factor** dialog box reports to you the static base shear and the un-scaled RSA base shear for each direction. These values will be transferred automatically from RISA-3D and cannot be modified within RISAFoundation.

Scaling Factor

Program Calculated

This section compares the ELF (Equivalent Lateral Force) Limit, as reduced by the Base Shear Multiplier and the I/R Limit. These values are explained in detail below.

- The **Base Shear Multiplier** is used in the calculation of the ELF Limit. This multiplier is used to reduce the calculated (unscaled) base shear below the base shear value obtained by the Equivalent Lateral Force Method. For the 1997 UBC, this is covered in section 1631.5.4. For the 2000 IBC, this is covered in section 1618.7. For the 2020 NBC, it is discussed in clause 4.1.8.12 (8). In the ASCE 7-22, it is discussed in section 12.9.1.4.1.
- The **ELF Limit** is the limiting scaling factor calculated per the Equivalent Lateral Force procedure:

$$ELF\ Limit = \frac{(Base\ Shear\ Multiplier * Static\ Base\ Shear)}{Unscaled\ Base\ Shear}$$

- The **I/R Limit** simply takes the elastic dynamic base shear and reduces it to an inelastic base shear. This value is calculated per the I (Importance Factor) and R (Response Modification Coefficient). You can edit these values in the Seismic tab of Model Settings.
- The Controlling Limit is the governing (larger) value of the ELF limit and I/R limit.

It is important to note that these values are transferred automatically from the RISA-3D analysis only when RSA results are transferred. The program calculated scaling factor cannot be modified from within RISAFoundation. If you would like to update this value you will need to transfer back to RISA-3D to do so.

User Input

If you would like to use a different scaling factor you can select the **User Input** option and manually enter a custom value to override the program calculated value.

Apply Check box

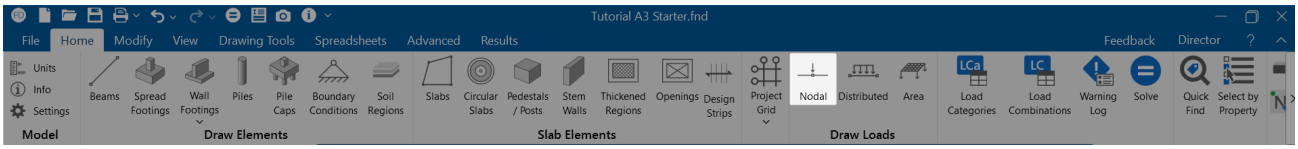
The **Apply SF to current RSA Load Combinations** check box will apply the SF scale factors to all load combinations that currently reference response spectra results.

Loads - Nodal Loads

RISAFoundation allows nodal loads to be oriented along any of the global axes or applied as a point moment about those axes. Nodal loads with a positive magnitude are applied in the direction of the positive global axis they are assigned to, and negative loads are applied in the direction of the negative global axis. The exception to this is that loads applied along the global Y-axis are reversed (i.e., positive loads are applied in the direction of the negative global Y-axis and negative loads are applied in the direction of the positive global Y-axis). All point moments are applied about the global axes according to the right-hand rule, including MY moments.

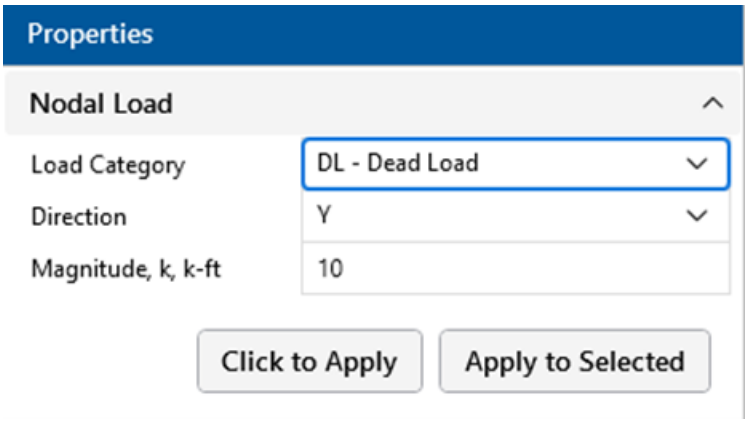
Drawing Nodal Loads

Nodal Loads and **Nodal Moments** are concentrated loads and may be placed anywhere on a slab or beam. Nodal loads may be assigned manually in the **Nodal Loads and Moments Spreadsheet** or drawn graphically in a model view.



To draw Nodal Loads:

1. If there is not a model view already open, choose the **Add 3D View** option from the **Open 3D Views** drop-down on the **View** ribbon.
2. Click the **Nodal Loads** icon on the **Home** ribbon and define the load in the **Properties** panel.

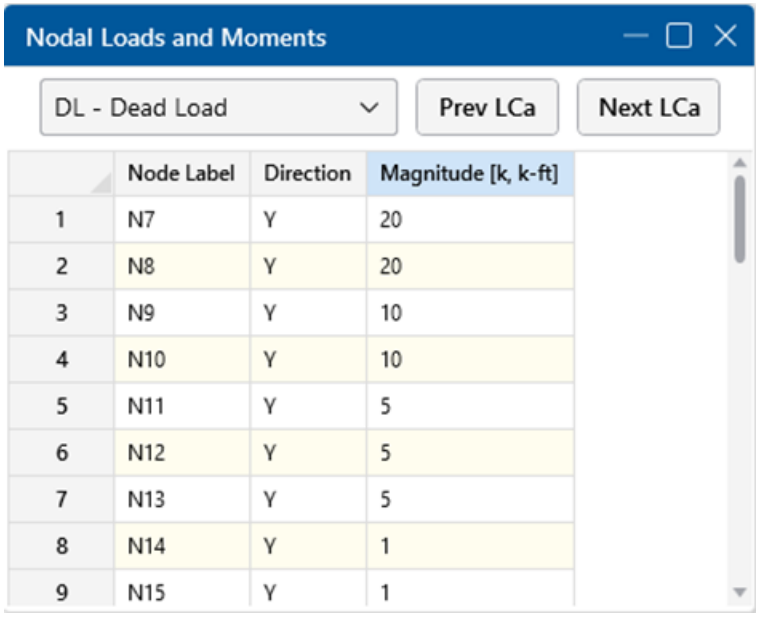


3. You can choose to apply the load to a single node at a time or to an entire selection of nodes.
    - To apply the load to just a few nodes, click the **Click to Apply** button, then click on the nodes in the 3D View with the left mouse button.
    - To apply the load to a selection, click the **Apply to Selected** button.

Note:
- To apply more loads with different parameters, change the parameters in the **Properties** panel and choose **Click to Apply** or **Apply to Selected** to apply the new loads.
- You can also specify or edit nodal loads in the **Nodal Loads Spreadsheet**.
- You can undo any mistakes by clicking the **Undo** button.

Nodal Load Spreadsheet

The **Nodal Loads Spreadsheet** records the nodal loads on all slabs and beams and may be accessed by selecting **Nodal Loads** from the **Data Entry** drop-down on the **Spreadsheets** tab of the ribbon, or by clicking **Nodal Loads** on the **Data Entry** toolbar.



	Node Label	Direction	Magnitude [k, k-ft]
1	N7	Y	20
2	N8	Y	20
3	N9	Y	10
4	N10	Y	10
5	N11	Y	5
6	N12	Y	5
7	N13	Y	5
8	N14	Y	1
9	N15	Y	1

The first column contains the **Label** of the node to which the nodal load is applied.

The **Direction** column defines the direction of the nodal load or moment relative to the Global Axes.

The load **Magnitude** is recorded in the third column. The units for the magnitude are listed at the top of the column, depending upon whether the load is a force or a moment.

Loads - Distributed Loads

RISAFoundation allows distributed loads and moments which act along/about the Global Axes. For Y-direction loads, a positive magnitude represents a downward load. Distributed Loads may be assigned manually in the **Distributed Loads Spreadsheet** or drawn graphically in the model view.

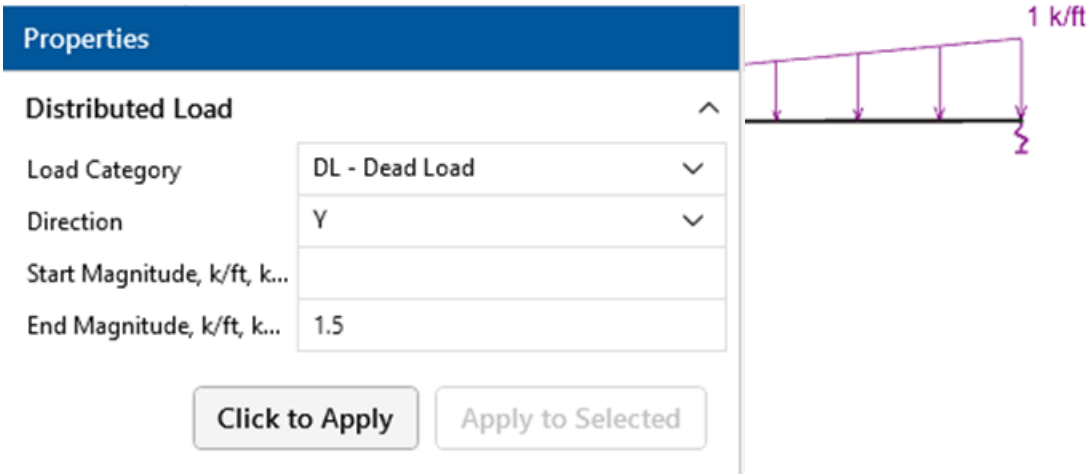
Drawing Distributed Loads

Distributed Loads are linear loads that can be placed anywhere on a slab or beam and can be of uniform, stepped, or varying magnitude. The direction of the load may be defined in the directions of the global axes. Make sure that you are careful to select the correct Load Category that you want the loads assigned to.



To draw Distributed Loads:

1. If there is not a model view already open, choose the **Add 3D View** option from the **Open 3D Views** drop-down on the **View** ribbon.
2. Click the **Distributed Load** icon on the **Home** ribbon and define the load in the **Properties** panel.



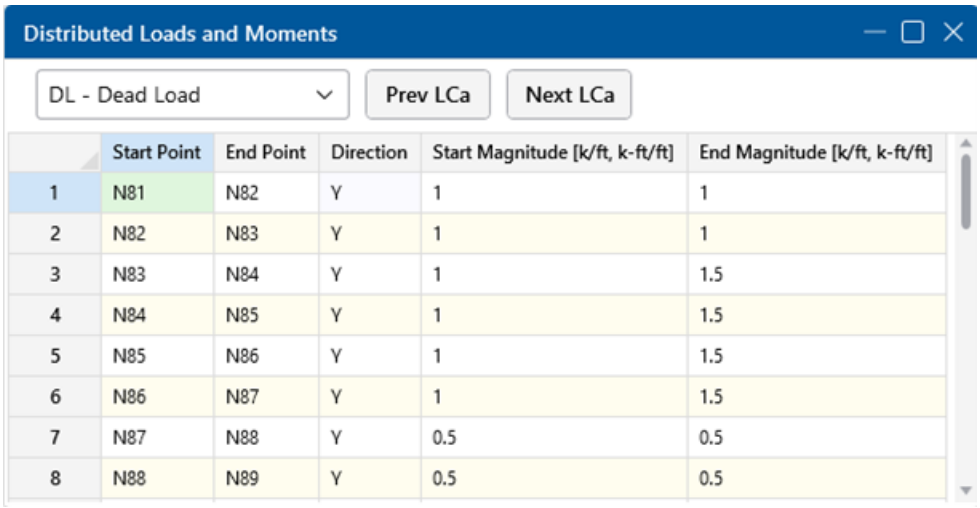
3. To begin drawing the load, choose **Click to Apply**.
4. Click on consecutive drawing grids or nodes to define the load.
    - To apply the load to a selection of beams, choose **Apply to Selected**.

Note:

- To apply more loads with different parameters, press **CTRL+ D** to recall the **Distributed Loads** tool.
- You may also specify or edit distributed loads in the **Distributed Load Spreadsheet**.

Distributed Loads Spreadsheet

The **Distributed Load Spreadsheet** records the distributed loads on all slabs and beams and may be accessed by selecting **Distributed Loads** from the **Data Entry** drop-down on the **Spreadsheets** tab of the ribbon, or by clicking **Distributed Loads** on the **Data Entry** toolbar.



	Start Point	End Point	Direction	Start Magnitude [k/ft, k-ft/ft]	End Magnitude [k/ft, k-ft/ft]
1	N81	N82	Y	1	1
2	N82	N83	Y	1	1
3	N83	N84	Y	1	1.5
4	N84	N85	Y	1	1.5
5	N85	N86	Y	1	1.5
6	N86	N87	Y	1	1.5
7	N87	N88	Y	0.5	0.5
8	N88	N89	Y	0.5	0.5

The first column contains the **Start Point** of the load, and the second column contains the **End Point** of the load.

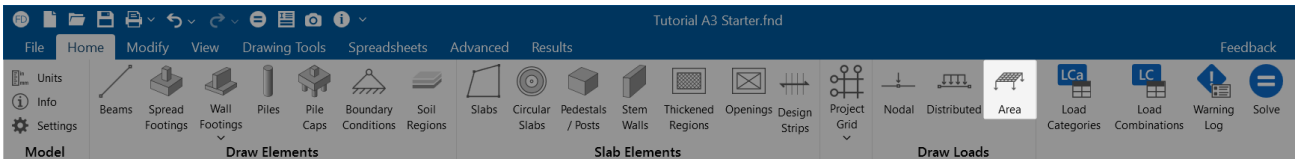
The **Direction** entry is used to select the direction of the load, relative to the Global Axes.

The **Start Magnitude** and **End Magnitude** columns are used to enter the magnitudes of the line load. If the start and end magnitudes of a line load are NOT the same, a tapered line load will be created. The units are listed in the column header.

Loads - Area Loads

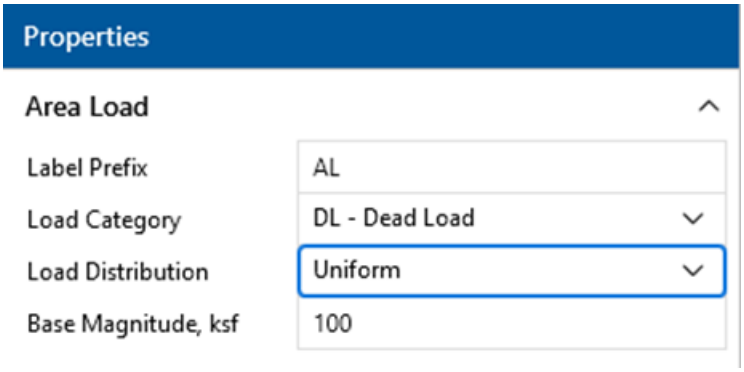
Area Loads are loads that are spread out over the surface of a slab or a portion of a slab. RISAFoundation allows area loads to be oriented along a vertical axis perpendicular to the XZ plane, with positive loads acting downward and negative loads acting upward. Loads may be input manually in the spreadsheets or drawn graphically in a model view. The area loads can be uniform or tapered.

Drawing Area Loads



To apply Area Loads:

1. Go to the **Home** ribbon.
2. Click on the **Area Load** icon to bring up the area load information in the **Properties** panel.

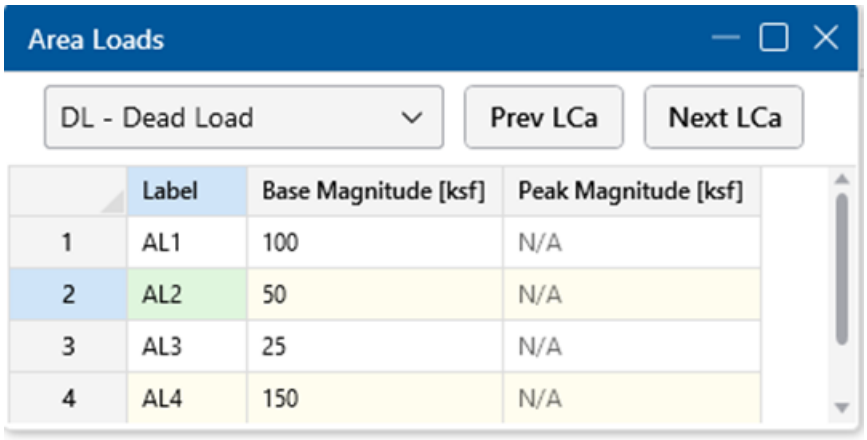


3. Define the load label and magnitude in the Area Load Properties Panel. Use the drop-down menus to select the Load Category and Load Distribution (Uniform or Tapered).
4. Click on nodes in the 3D view to define the area. Complete your area load polygon by clicking the first node a second time.

Note:
- You can edit area loads in the **Area Loads Spreadsheet**.
- You can undo any mistakes by clicking the **Undo** button on the **Quick Access Toolbar**.

Area Loads Spreadsheet

The **Area Loads Spreadsheet** records the area loads and may be accessed by selecting **Area Loads** from the **Data Entry** menu in the **Explorer** panel. Alternatively, it can be accessed by going to the **Spreadsheets** tab, selecting the **Data Entry Menu** and selecting **Area Loads**.



When you open this spreadsheet, you can view only one load category at a time. Use the drop-down list on the toolbar to specify a different load category.

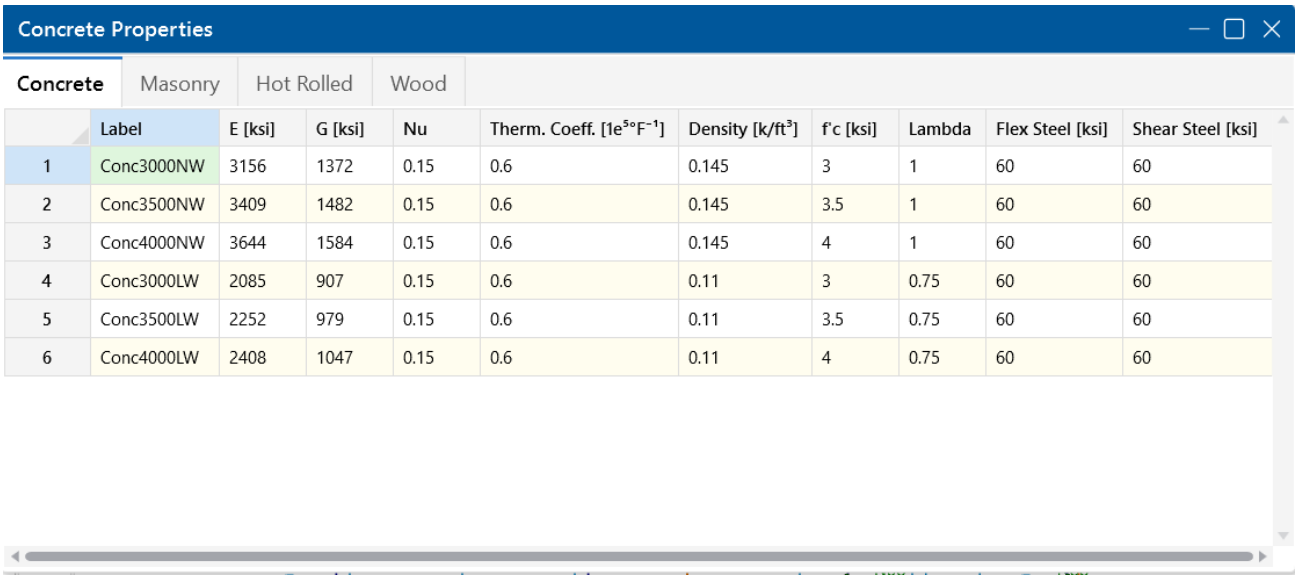
The first column contains the label for the area load. The second column defines the magnitude of the load. The third column is used to define the peak magnitude of the area load when a tapered area load has been drawn.

Material Properties

Material properties are defined on the **Material Properties Spreadsheet** and are then referred to as you build your model elements. You can use up to 500 materials in a single model.

Material Properties Spreadsheet

The **Material Properties Spreadsheet** records the material properties to be used in the model and can be accessed by selecting **Materials** from the **Data Entry** section of the **Explorer** panel. The entries are described below:



	Label	E [ksi]	G [ksi]	Nu	Therm. Coeff. [1e^{5}°F^{-1}]	Density [k/ft^{3}]	f'c [ksi]	Lambda	Flex Steel [ksi]	Shear Steel [ksi]
1	Conc3000NW	3156	1372	0.15	0.6	0.145	3	1	60	60
2	Conc3500NW	3409	1482	0.15	0.6	0.145	3.5	1	60	60
3	Conc4000NW	3644	1584	0.15	0.6	0.145	4	1	60	60
4	Conc3000LW	2085	907	0.15	0.6	0.11	3	0.75	60	60
5	Conc3500LW	2252	979	0.15	0.6	0.11	3.5	0.75	60	60
6	Conc4000LW	2408	1047	0.15	0.6	0.11	4	0.75	60	60

The materials shown are the program defaults. However, you can change these and also add as many other materials as you need. You can then save your preferred materials as the default materials by clicking the **Save as Defaults** button on the **Spreadsheets** tab.

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 model elements.

E is the Young's modulus that describes the material stiffness.

G is the shear modulus and can be left blank if you would like it calculated automatically. The equation for "G" is:

$$G = \frac{E}{2.0 * (1.0 + Poisson's\ Ratio)}$$

Note: If you enter a value for Shear Modulus that does not match the value calculated using the above equation you will be given a warning.

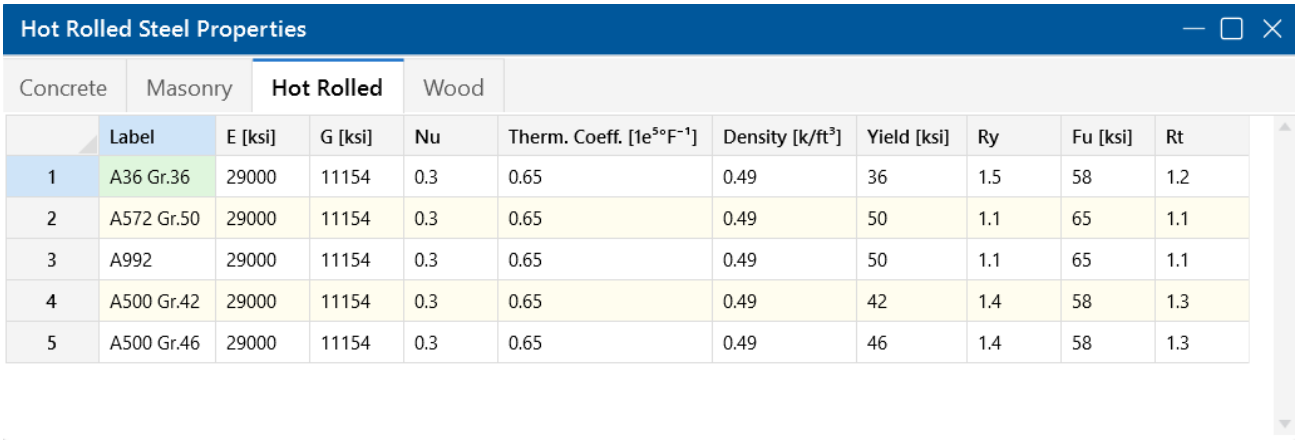
Nu is the Poisson's ratio. Besides being used for the "G" calculation, this value is also used for shear deformation calculations. The value of the Poisson's ratio may not exceed '0.5'.

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

Density is the material density and is used in the calculation for self weight. This density, times the area, gives the self weight per unit length for members; density times slab volume gives the total weight of a given slab.

Hot Rolled Specific Material Data

The **Hot Rolled** tab records a number of hot rolled steel specific material properties that do not apply for the other materials. These entries are described below:



	Label	E [ksi]	G [ksi]	Nu	Therm. Coeff. [1e^{5}°F^{-1}]	Density [k/ft^{3}]	Yield [ksi]	Ry	Fu [ksi]	Rt
1	A36 Gr.36	29000	11154	0.3	0.65	0.49	36	1.5	58	1.2
2	A572 Gr.50	29000	11154	0.3	0.65	0.49	50	1.1	65	1.1
3	A992	29000	11154	0.3	0.65	0.49	50	1.1	65	1.1
4	A500 Gr.42	29000	11154	0.3	0.65	0.49	42	1.4	58	1.3
5	A500 Gr.46	29000	11154	0.3	0.65	0.49	46	1.4	58	1.3

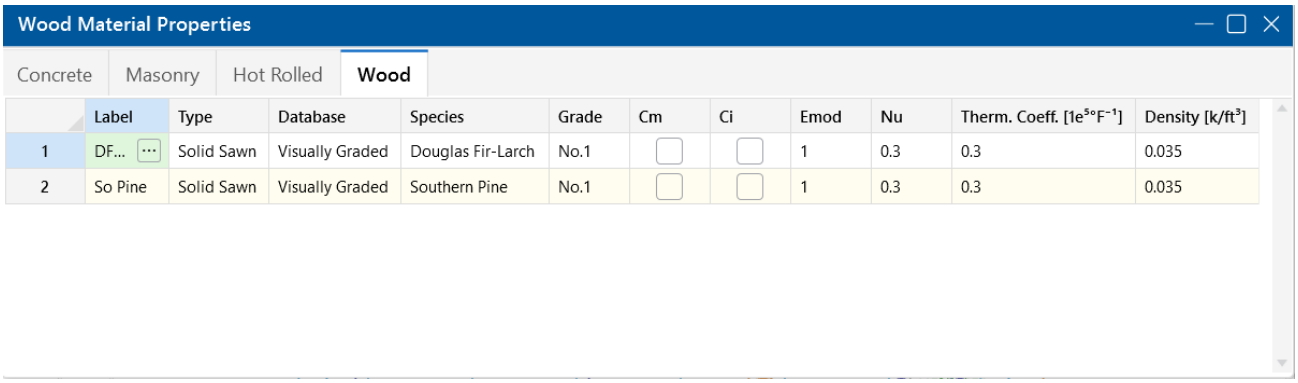
Ry is the ratio of the expected yield stress to the specified minimum yield stress.

Fu is the specified minimum tensile strength.

Rt is the ratio of the expected tensile strength to the specified minimum tensile strength Fu, as related to overstrength in material yield stress, Ry. Rt is currently not used by the program.

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:



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

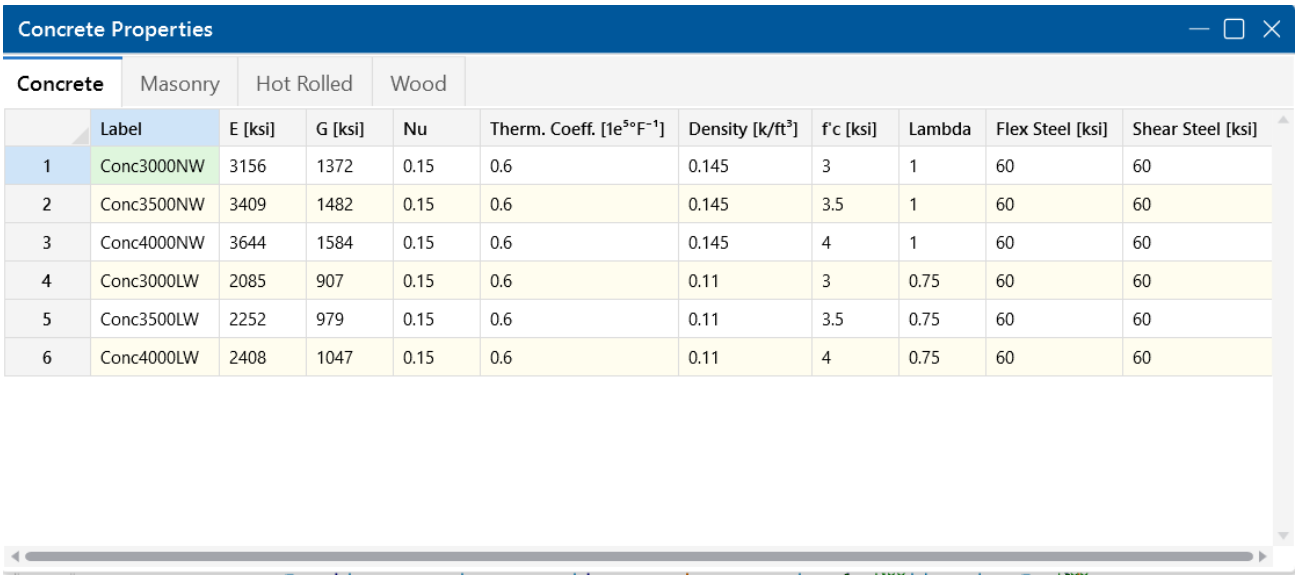
When either the NDS 05/08, 12, 15, 18 or 24 wood design code is selected, the Solid Sawn Species menu lists two new species that are not specifically shown in the NDS code. These are the *Commercial Species Group I - DF/SP* and *Commercial Species Group II - HF/SPFe Fir*. These are meant to be simplified groupings of the most commonly used wood species. It is meant to simplify the selection process for wood member design in the United States.

The design values for Group I take the NDS allowable stress values for two of the most widely used species (Doug Fir-Larch and Southern Pine) and uses the lower bound allowable stress value for each size and grade. To verify exactly what base allowable stress values were used, you can review the properties in the Shape Selection dialog.

Similarly, the design values for Group II take the allowable stress values for Hem Fir and Spruce-Pine-Fir and use the lower bound allowable stress values for each size and grade. You can also verify the base allowable stress values used for this selection in the Shape Selection dialog as well.

Concrete Specific Material Data

The **Concrete** tab records a few concrete specific material properties that do not apply for other materials. These entries are described below:



	Label	E [ksi]	G [ksi]	Nu	Therm. Coeff. [1e^{5}°F^{-1}]	Density [k/ft^{3}]	f'c [ksi]	Lambda	Flex Steel [ksi]	Shear Steel [ksi]
1	Conc3000NW	3156	1372	0.15	0.6	0.145	3	1	60	60
2	Conc3500NW	3409	1482	0.15	0.6	0.145	3.5	1	60	60
3	Conc4000NW	3644	1584	0.15	0.6	0.145	4	1	60	60
4	Conc3000LW	2085	907	0.15	0.6	0.11	3	0.75	60	60
5	Conc3500LW	2252	979	0.15	0.6	0.11	3.5	0.75	60	60
6	Conc4000LW	2408	1047	0.15	0.6	0.11	4	0.75	60	60

f'c is the concrete compressive strength used for concrete design.

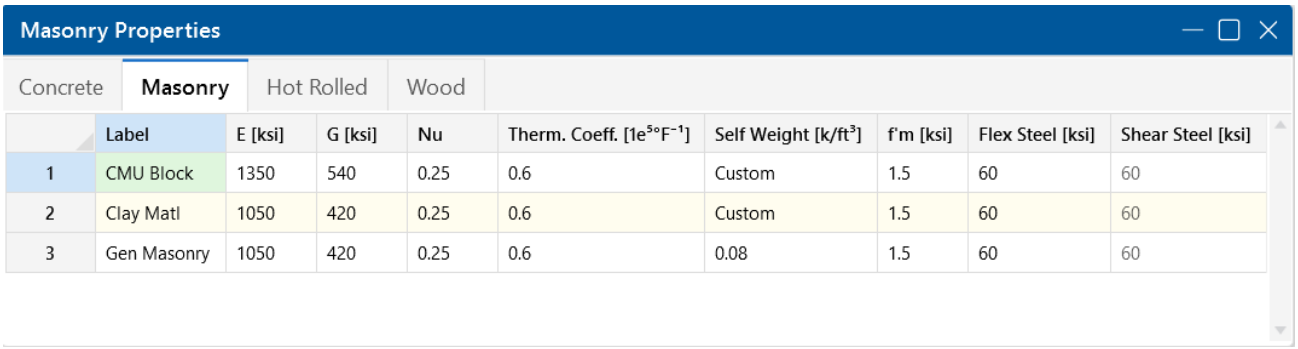
Lambda, **λ** is the lightweight concrete modification factor. This factor only applies to the *ACI 318-19* *ACI 318-14*, *ACI 318-11*, *ACI 318-08* and *CSA A23.3-04* codes. For all other codes, the Density value of the material determines any strength reduction. The program automatically calculates 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.

Shear Steel is the reinforcement yield strength for shear bars.

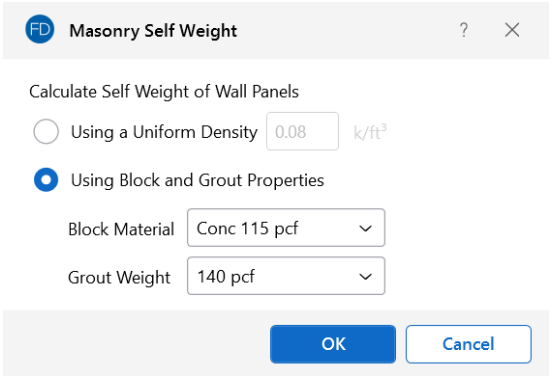
Masonry Specific Material Data

The **Masonry** tab records masonry specific material properties that do not apply for other materials.



	Label	E [ksi]	G [ksi]	Nu	Therm. Coeff. [1e^{5}°F^{-1}]	Self Weight [k/ft^{3}]	f'm [ksi]	Flex Steel [ksi]	Shear Steel [ksi]
1	CMU Block	1350	540	0.25	0.6	Custom	1.5	60	60
2	Clay Matl	1050	420	0.25	0.6	Custom	1.5	60	60
3	Gen Masonry	1050	420	0.25	0.6	0.08	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 and select ... 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 table B3 of the Reinforced Masonry Engineering Handbook (RMEH). The self weight will be listed as **Custom**.

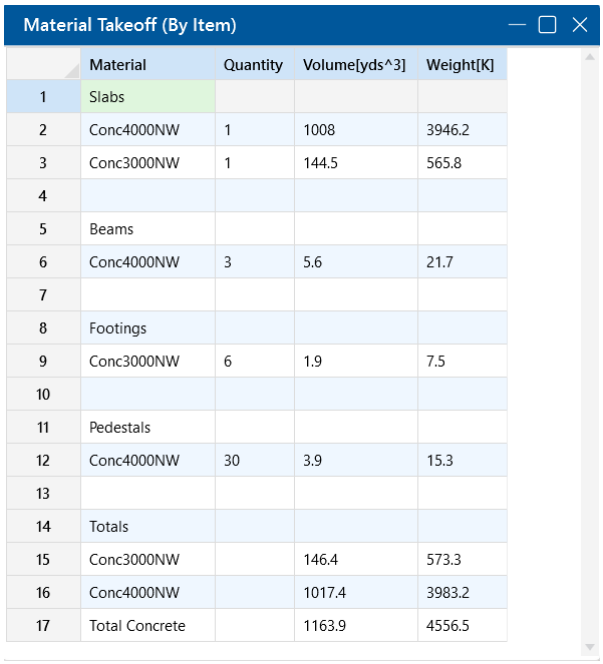
f'm is the masonry compressive strength used for masonry design.

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 only used for walls in wall footings. For more information, see the Wall Footing Definitions Spreadsheet.

Material Takeoff

Access the **Material Takeoff** spreadsheet by selecting it from the **Results** menu.



Material Takeoff (By Item)

	Material	Quantity	Volume[yds^3]	Weight[K]
1	Slabs			
2	Conc4000NW	1	1008	3946.2
3	Conc3000NW	1	144.5	565.8
4				
5	Beams			
6	Conc4000NW	3	5.6	21.7
7				
8	Footings			
9	Conc3000NW	6	1.9	7.5
10				
11	Pedestals			
12	Conc4000NW	30	3.9	15.3
13				
14	Totals			
15	Conc3000NW		146.4	573.3
16	Conc4000NW		1017.4	3983.2
17	Total Concrete		1163.9	4556.5

This spreadsheet shows material takeoff information for each material and element type in the model. For each material, the element types are listed with the total quantity, volume and weight.

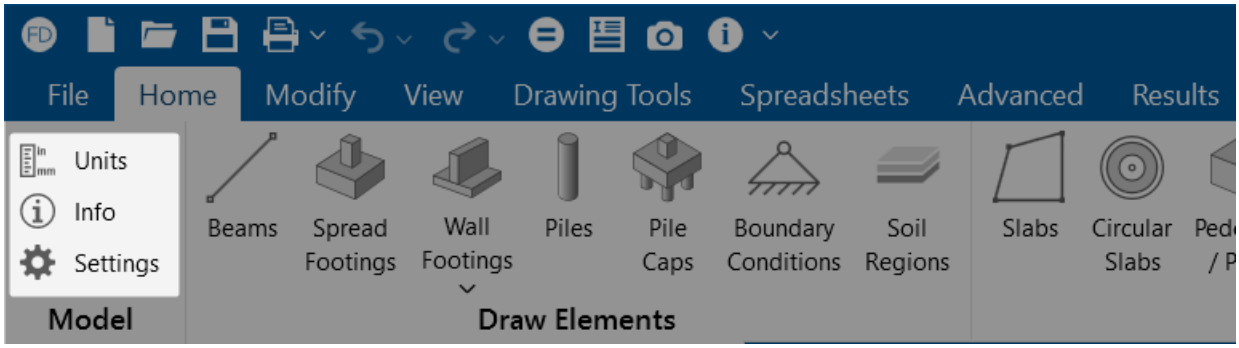
The weight is the sum of the self-weight for all elements

Note:

- This Material Takeoff report is independent of the loads applied to the model, i.e. the applied loads do not influence this report.
- Beams that frame into a footing or pile cap will show the beam stopping at the location of the element. However, for material takeoff concerns, the entire length of beam (all the way to the end node corresponding to the footing/cap pedestal) is considered.

Model Settings

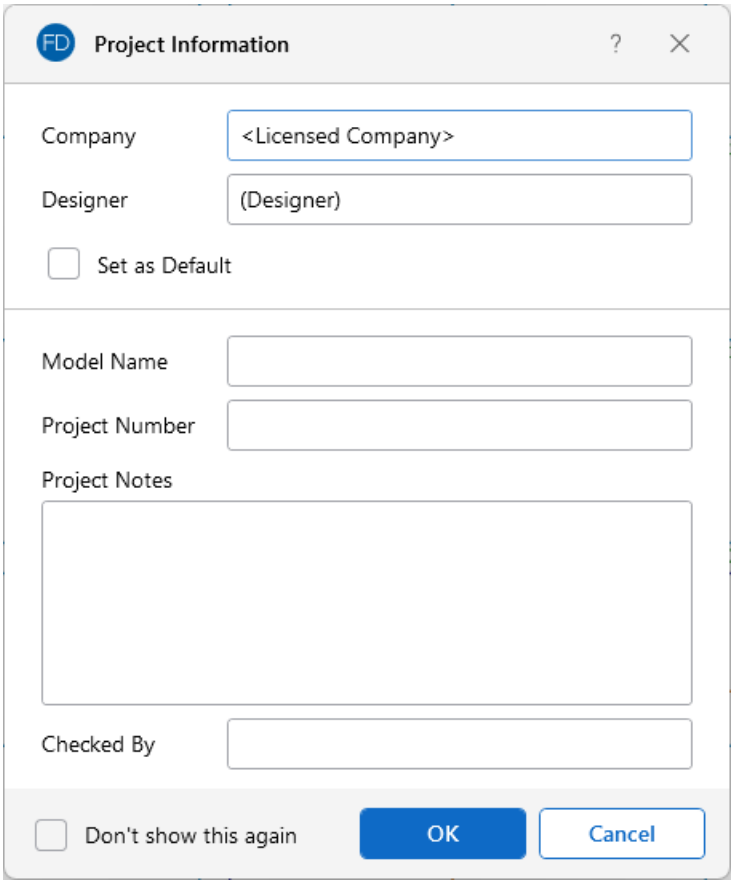
The model settings may be accessed on the **Home** ribbon in the **Model** section. The settings consist of three main options (**Units**, **Info** and **Settings**) that are highlighted in the following image.



- **Units**: The 'Units' option allows you to set the type of units (imperial, metric or both) you want to use in your model. Refer to the Units topic for more details on the units settings.
- **Info**: The 'Info' option allows you to enter descriptive information, such as a title for the particular model being defined, the name of the designer and a job number. The title may then be printed at the top of each sheet of the output, and on the graphic printouts of the model.
- **Settings**: The 'Settings' option provides control for most of the model-specific settings. These Model Settings are specific to the model that is currently open. The information stored here is retained with the model and will be retained even if the model is opened on a different computer.

  You can save any of the Model Settings as the default setting, such that new models you create will begin with the settings you saved as Defaults. To do this, simply enter the information that you want to save and check the **Set as Default** box (see the image under the Solution heading for the location of this check box).

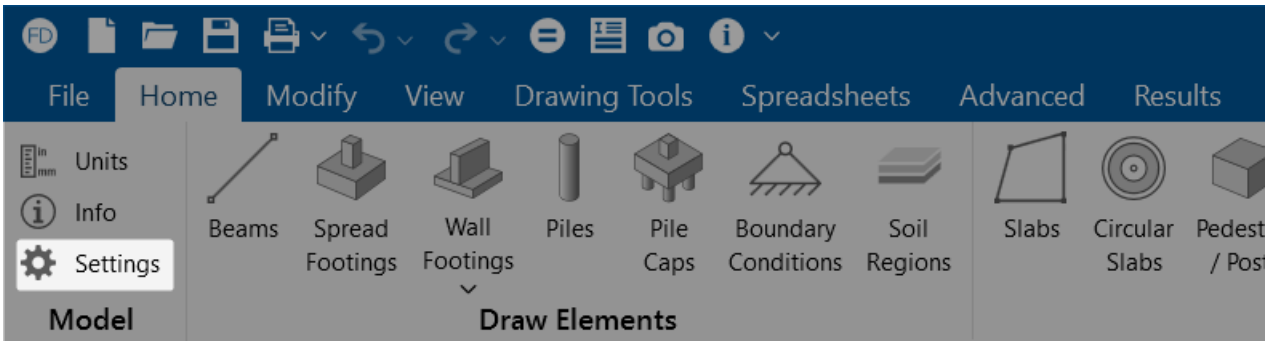
Project Information



- **Project Notes** is intended for information about the model that may be useful for the engineer or reviewer to refer to. This box is limited to 4000 characters.
- **Checked By** is intended for the model reviewer to initial upon completion of a model review. Their initials are then printed on each sheet of the output.

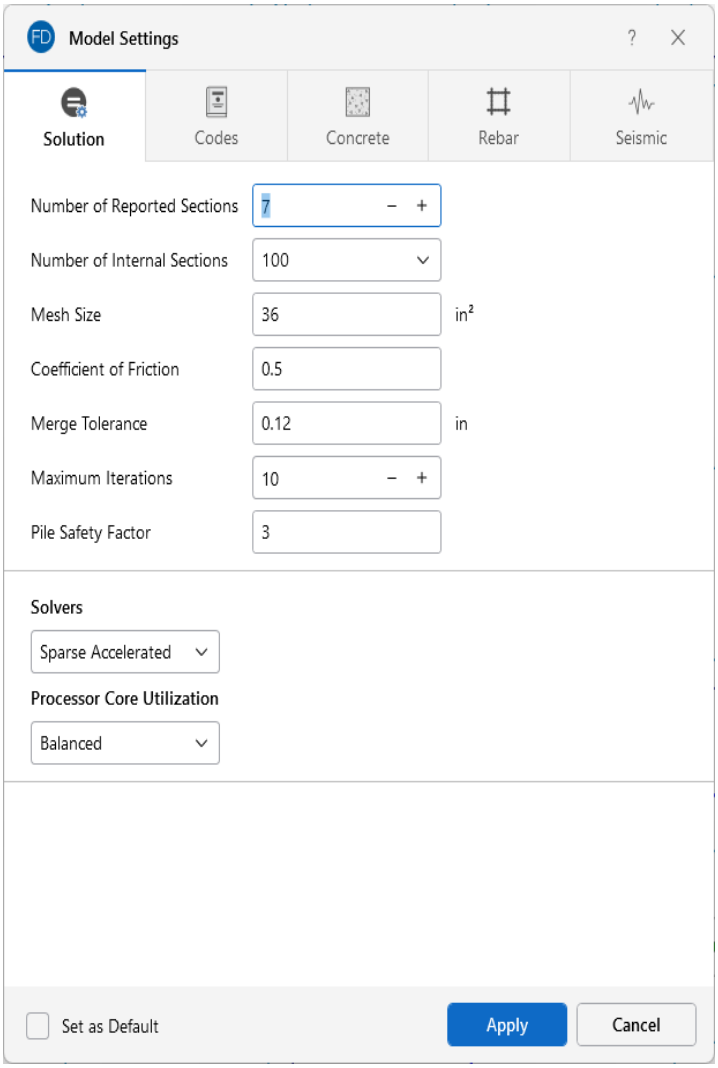
Settings

To open the **Model Settings** window, click the **Settings** option from the **Home** ribbon, as shown in the following image.



Solution

The **Solution** tab is the first tab in the Model Settings window. The entries on this tab are used to control settings that affect the general solution of the model.



Number of Reported Sections controls how many places along each beam you receive **reported** beam force, stress 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. For more information, see Printing and Beams - Forces.

Number of Internal Sections controls how many places along each beam the software calculates and stores results, such as deflections and code checks. The beam 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.

The **Mesh Size** is used to control the coarseness or fineness of the slab mesh when RISA auto-meshes slabs during solution.

Coefficient of Friction is used in the slab sliding check to calculate the resisting force against sliding.

Merge Tolerance is used as the maximum distance two 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.

Maximum Iterations defines the maximum number of iterations the program will undergo for tension and compression springs in a model. Soil regions are always modeled as compression springs, but piles may be tension springs, compression springs or both. In the first iteration, the program assumes a compression spring for every support node in the model. The program then checks the forces (tension or compression) in these springs against the soil moduli or pile stiffness. Soil springs have no tensile stiffness, and piles may or may not have tensile stiffness. If a support location sees tension, then the springs in this location are switched to tension springs of the appropriate capacity. Soil springs will have a tensile stiffness of zero and piles will have a stiffness defined in the **Pile Definitions Spreadsheet**. At this point, the model is re-run for its second iteration. The model is then checked again to make sure that tension springs are in tension and compression springs are in compression. If so, the solution is stopped. If not, the springs are updated to the proper tension or compression designation and the model is re-run again. At least three iterations are required to get acceptable behavior, though more may be needed in complicated loading scenarios.

Pile Safety Factor is used to calculate the static pile soil capacity. The user may modify this value to consider potential variances in soil and pile interaction.

Note:

- The Number of Reported Sections cannot exceed 20. Also, the 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.
- When RISAFoundation is accessed from the Director for a model originating in RISAFloor or RISA-3D, the Beam Section Options will not appear. In these cases, the Model Settings from RISA-3D will control.
- You can check the **Set as Default** check box to have your settings saved for the next time you open the program.

Processor Core Utilization

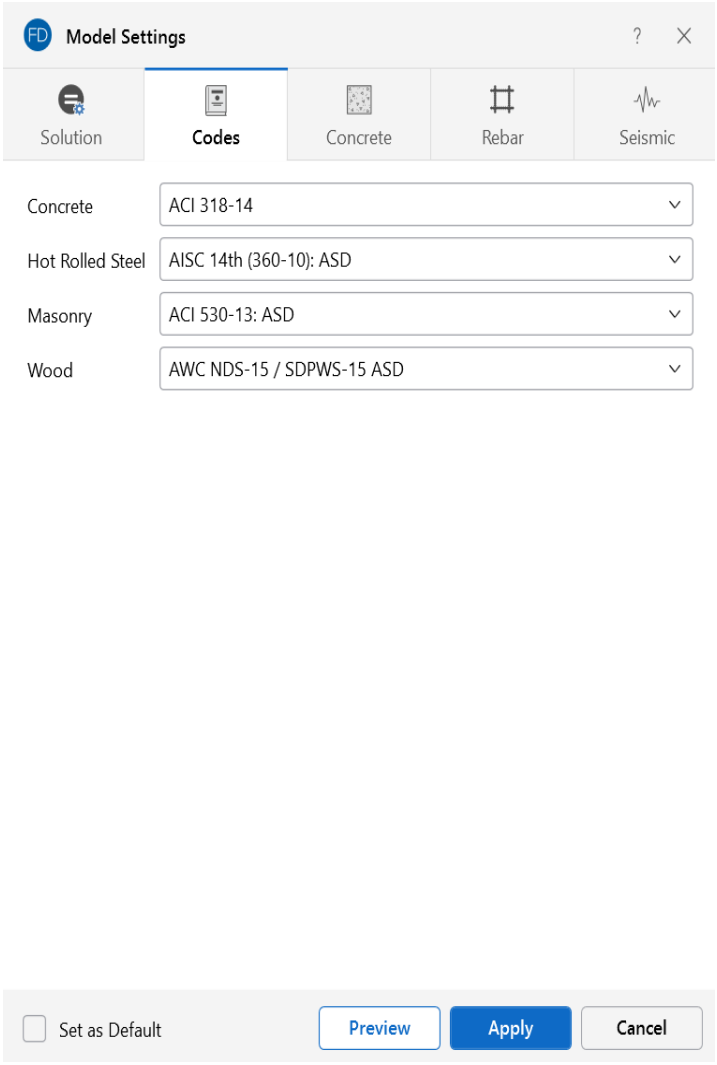
Single lets you use a single core to run the model solution. This option is useful when you have several other computer programs that are CPU intensive running in conjunction with RISAFoundation.

Balanced lets you use half of the available cores on your computer to run the solution. This option will provide a faster solution time compared to using a single core, but will still allow half of your computer's available cores to be used for other programs and computer processes.

Max lets you use all of the available cores on your computer, minus one. For instance, if you have eight available cores and select the Max option, the program uses seven cores in the processing of the solution. This option provides the fastest solution time, but still apportions a single core to be used for other programs and computer processes.

The **Static Solver** options are **Standard Skyline** or **Sparse Accelerated**. These can be selected from the drop-down menu in the **Solvers** section. See Solution for more information on these two options.

Codes



The settings under the **Codes** tab control which design code is used for code checks for each material type.

Concrete indicates which concrete code is to be used in the design of concrete elements.

Available options for concrete are:

- The 2022, 2019, 2014, 2011, 2008, 2005, 2002 and 1999 Editions of ACI 318
- The 1992 EuroCode (EC2) and the British publication of the 2014 and 2004 Eurocode (BSEN)
- The 2014 and 2004 Edition of the Canadian code (CSA-A23.3)

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

Note:
- ACI 318-08 and ACI 318-11 behave identically in the program, as there were no relevant changes between these two editions of the code.
- A significant change for the 2014 ACI code is a change to the minimum flexural reinforcement for mat 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.

Hot Rolled Steel indicates which code is to be used for the design of hot rolled steel pile elements. The design code selected is used to calculate the tensile and compressive capacity for piles in the model.

Available options for hot rolled steel are:

- AISC 360-22 (16th Edition), 360-16 (15th Edition), AISC 360-10 (14th Edition), and AISC 360-05 (13th Edition) ASD and LRFD
- AISC LRFD (2nd and 3rd Editions)
- AISC ASD (9th Edition)
- The CSA S16-19, CSA S16-14, CSA S16-09, CSA S16-05, CSA S16-01, and CSA S16.1-94 Canadian codes

Masonry indicates which masonry code is to be used for the design of masonry walls.

Masonry is only used for the wall in wall footing elements. For more information, see Wall Footing Definitions.

Available options for masonry are:

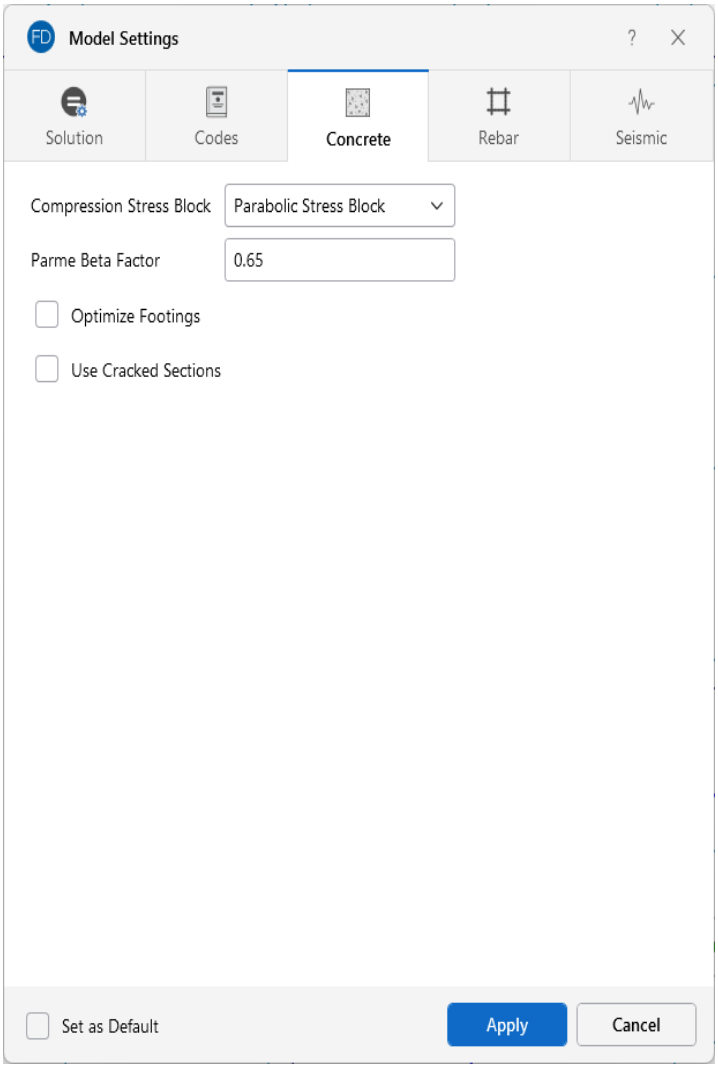
- The TMS 402-22: ASD and Strength Codes
- The TMS 402-16: ASD and Strength Codes
- The ACI 530-13: ASD and Strength Codes

Wood indicates which wood code is to be used for the design of wood pile elements. The design code selected is used to calculate the tensile and compressive capacity for piles in the model.

The available design codes for Wood are:

- The 2024, 2018, 2015, 2012 Editions of the NDS wood design code for ASD
- The 2014 and 2009 Editions of the CSA O86 Canadian wood design codes.

Concrete



The entries under the **Concrete** tab contain options related to the analysis and design of concrete elements.

Compression Stress Block lets you choose what type of stress block to consider in your analysis. The options are the constant **Rectangular Stress Block** and the **Parabolic Stress Block**. For more information, see Parabolic vs. Rectangular Stress Blocks .

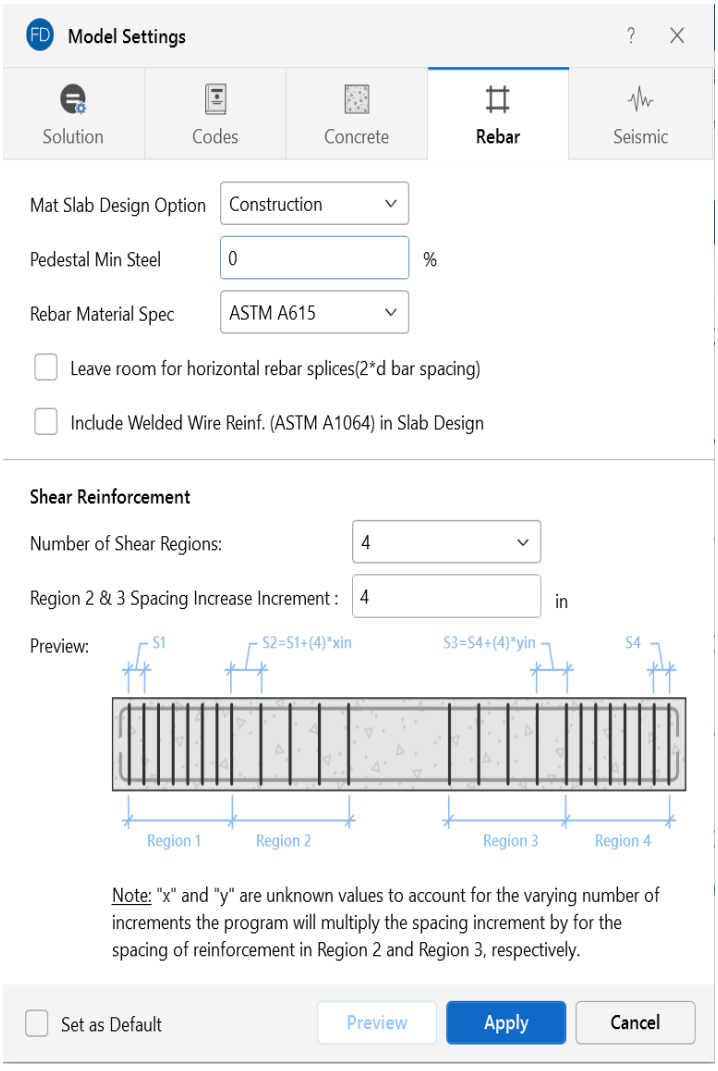
Parme Beta Factor is used to approximate the column's 3D interaction surface for the PCA Load Contour Method.

Check the **Optimize Footings** check box to get simple spread footings to be sized for overturning and sliding forces.

The **Use Cracked Sections** check box will allow the program to use the cracked moment of inertia and reduce the bending stiffness of beams and slabs. The reduction factor can either be user defined by specifying the Icr Factor directly, or the program will automatically generate it using the program defaults.

Note: Concrete cover is specified on the various tabs of the Design Rules Spreadsheet. For more information, see Design Optimization.

Rebar



Mat Slab Design Option offers a choice between two methods for the design of the mat slab rebar: namely, the **Construction method** and the **Theoretical method**. The Construction method, which has been 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 an added method 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 the 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 the Theoretical method will have no info for the total rebar quantity.

Pedestal Min Steel allows you to choose the minimum percentage of reinforcing steel to be used in a concrete pedestal or pile section. The percentage entered will be multiplied by the gross area of the pedestal or pile section to determine the minimum amount of reinforcement required in each member. If left blank, the default minimum percentage of reinforcement steel will be used based on the selected concrete design code. For example, the default minimum percentage of steel for pedestals and piles is 0.5% under ACI 318-14.

Rebar Material Spec lets you choose from the standard ASTM A615 (imperial), ASTM A615M ("hard" metric, i.e. #8M is an 8mm bar), BS 4449 (British), prENV 10080 (Euro), CSA G30.18 (Canadian), IS 1786 (Indian), and AS/NZS 4671:2001 (Australian/New Zealand) reinforcement standards.

Note:
- The rebar material specification in this section is for concrete design only. Masonry walls only support ASTM A615 rebar specification by default.

Checking the **Leave room for horizontal rebar splices (2*d bar spacing)** check 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. This option applies to beams only.

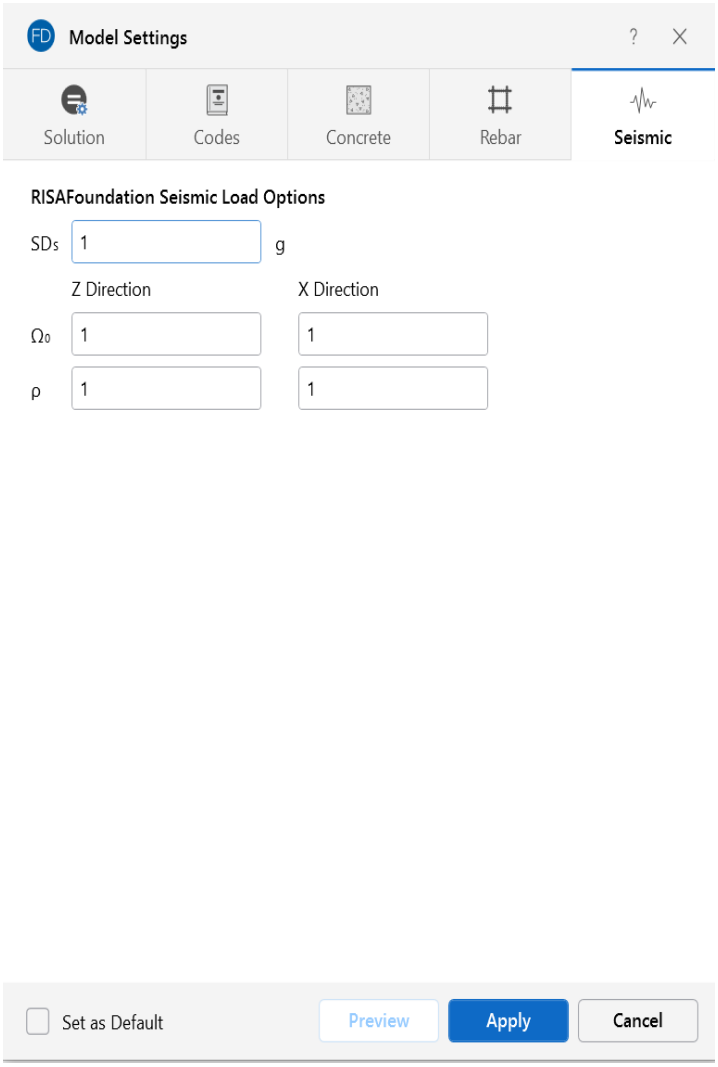
Checking the **Include Welded Wire Reinf. (ASTM A1064) in Slab Design** check box allows you to design a slab using Welded Wire Reinforcement.

Shear Reinforcement Options let you control the **Number of Shear Regions** that are used when detailing a beam span. You can also specify a **Region 2 & 3 Spacing Increase Increment** that you'd like the program to use when increasing or reducing the spacing of the shear ties.

Note:
- Welded wire reinforcement is only available for mat slabs, footings and pile caps.
- Only ASTM A1064 deformed bar sizes are available.

Seismic

The options on the **Seismic** tab list settings that are specifically related to calculation of code prescribed Seismic Loads. The input options you have depend on the Seismic Code selected.



SDs represents the 5% damped spectral response design acceleration for short-period response. This will be used if you included vertical seismic load effects in your seismic load combinations.

(Ω) is the **Overstrength Factor**.

Redundancy Factor (ρ) is based on the extent of redundancy in your structure.

Each of these inputs can be included in the seismic load combinations generated by the LC Generator.

Modeling Tips

Here are some specialty topics that can help model more complex situations.

Soil Springs for Eccentric Footings

For detailed information about modeling of combined footings with eccentric pedestals, see Soil Springs Calculation (Automatic).

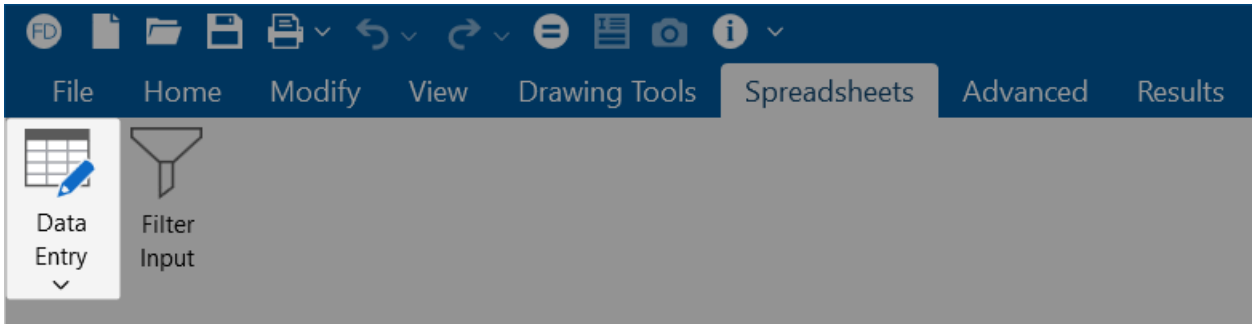
Modeling Complex Slabs

For detailed information about modeling of complicated slab geometries or slabs with varying thickness, see Slabs.

Nodes

Nodes are used to define the basic geometry of the structure or model. This includes slabs, beams, footings, piles and pile caps. Nodes are also used to define load locations such as nodal loads, line loads and area loads. Each node is a point in space defined by coordinates in the horizontal directions or axes.

Nodes are created automatically as you draw, but may be created manually from within the **Node Coordinates Spreadsheet**. Once defined, the nodes can be edited in three ways: graphically, in the **Node Properties Panel**, or in the **Node Coordinates Spreadsheet**.



Define Nodes

To define nodes:

1. Select the **Node Coordinates Spreadsheet** from the **Spreadsheets** tab ▸ **Data Entry** button, or from the **Data Entry** section of the **Explorer** panel.
2. Define the node coordinates.

Note:
- You may use cut and paste, block fill, and block math to enter and edit nodes.
- You may choose the prefix that is used to label the nodes.
- To modify one node you may click that node in the 3D View to view and edit its properties in the Properties panel.

Relabel Nodes

To relabel Nodes:

1. After sorting the **Node Coordinates Spreadsheet** into the desired order, go to the **Modify** ribbon.
2. Click the **Relabel All** or **Relabel Selected** icon.
3. Choose **Nodes**.

4. Alternatively, you can right-click in the **Node Coordinates** spreadsheet and select either **Relabel All Nodes** or **Relabel Selected Nodes**.

Round-Off Node Coordinates

To round-off Node Coordinates:

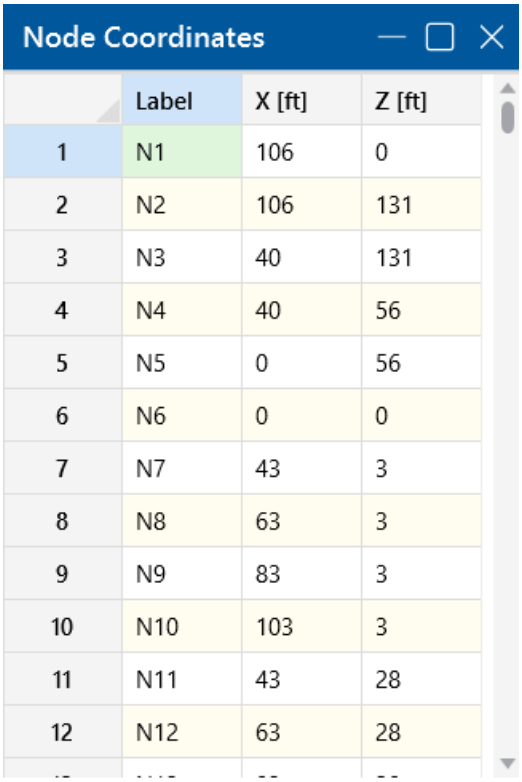
1. Go to the **Modify** ribbon.
2. Click the **Round Off Coordinates** icon.

Alternatively, you can right-click in the **Node Coordinates Spreadsheet** and select **Round Off Node Coordinates**.

Note: This utility rounds off all the node coordinates to 1, 2 or 3 decimal places (user selected). It's sometimes useful for models that have been generated or created via DXF import and have node coordinates with a large number of decimal places that aren't significant. Rounding off those coordinates makes the data more consistent and can help avoid problems such as non-coplanar model elements.

Node Coordinates Spreadsheet

The **Node Coordinates Spreadsheet** records the labels and coordinates for the nodes and may be accessed by selecting **Node Coordinates** from the **Data Entry** section in the **Explorer** panel.



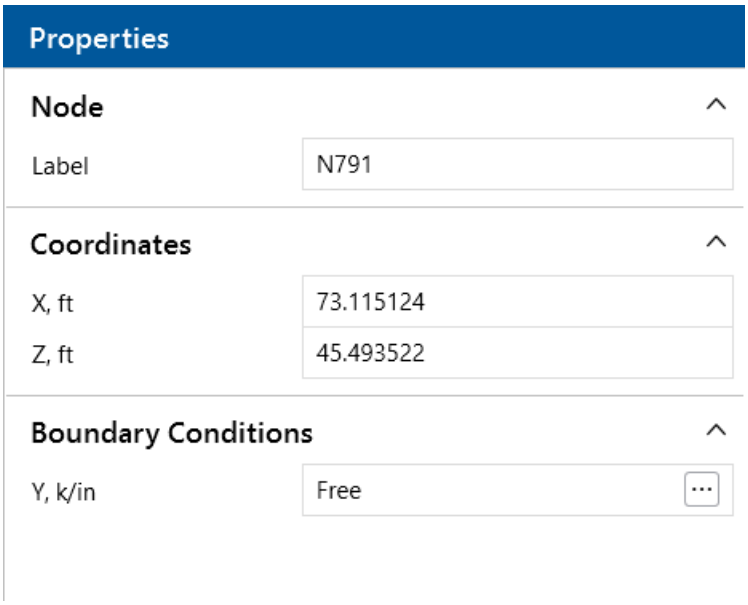
	Label	X [ft]	Z [ft]
1	N1	106	0
2	N2	106	131
3	N3	40	131
4	N4	40	56
5	N5	0	56
6	N6	0	0
7	N7	43	3
8	N8	63	3
9	N9	83	3
10	N10	103	3
11	N11	43	28
12	N12	63	28

Use the first column to assign a **Label** for each node. You can then refer to each node by its label. Each label has to be unique. If you try to enter the same label more than once, you will receive an error message. As you create new lines, the program will automatically create a new, unique label for each node.

The next two columns contain the coordinates of the node in the **X** and **Z** global directions. These represent the offsets of the nodes from the origin of the coordinate system (0, 0). The appropriate units are listed at the top of each column.

Node Information Dialog

Selecting a node will display it's properties in the **Properties** panel. All of the same information that is stored in the **Node Coordinates Spreadsheet** is displayed for the node you choose and may be edited. This is a quick way to view and change node properties. However, the spreadsheet and graphic editing tools may be a faster solution for large selections of nodes.



Label - You can view and edit the node's label.

Coordinates - You can view and edit the node coordinates.

Boundary Conditions - You can view and edit the node boundary conditions. You can enter additional data for a boundary condition, such as a fixed reaction or spring stiffness.

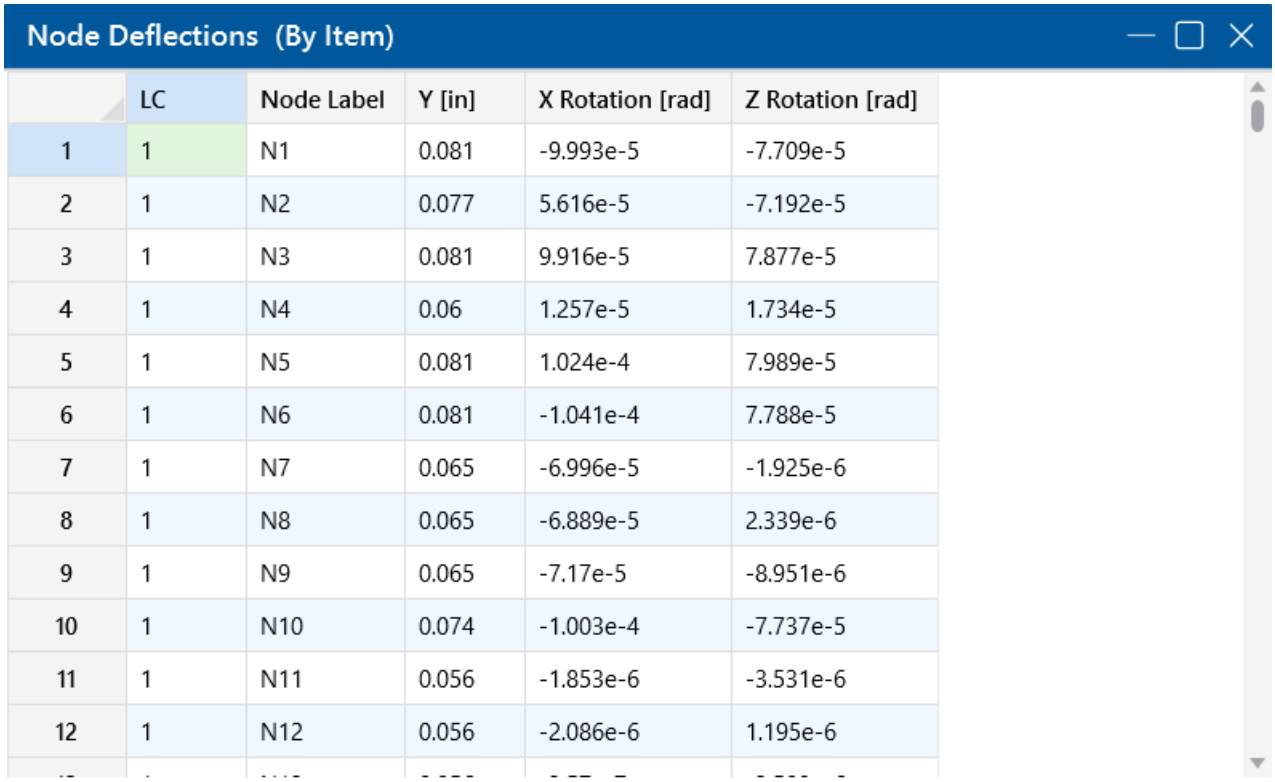
Note: It's generally more efficient to use the Graphic Editing features if you want to change the properties for many nodes at once.

Nodes - Results

When the model is solved, there are several groups of results specifically for the nodes.

Node Deflections Results

Access the **Node Deflections Spreadsheet** by selecting it from the **Results** section of the **Explorer** panel.



Node Deflections (By Item)

	LC	Node Label	Y [in]	X Rotation [rad]	Z Rotation [rad]
1	1	N1	0.081	-9.993e-5	-7.709e-5
2	1	N2	0.077	5.616e-5	-7.192e-5
3	1	N3	0.081	9.916e-5	7.877e-5
4	1	N4	0.06	1.257e-5	1.734e-5
5	1	N5	0.081	1.024e-4	7.989e-5
6	1	N6	0.081	-1.041e-4	7.788e-5
7	1	N7	0.065	-6.996e-5	-1.925e-6
8	1	N8	0.065	-6.889e-5	2.339e-6
9	1	N9	0.065	-7.17e-5	-8.951e-6
10	1	N10	0.074	-1.003e-4	-7.737e-5
11	1	N11	0.056	-1.853e-6	-3.531e-6
12	1	N12	0.056	-2.086e-6	1.195e-6

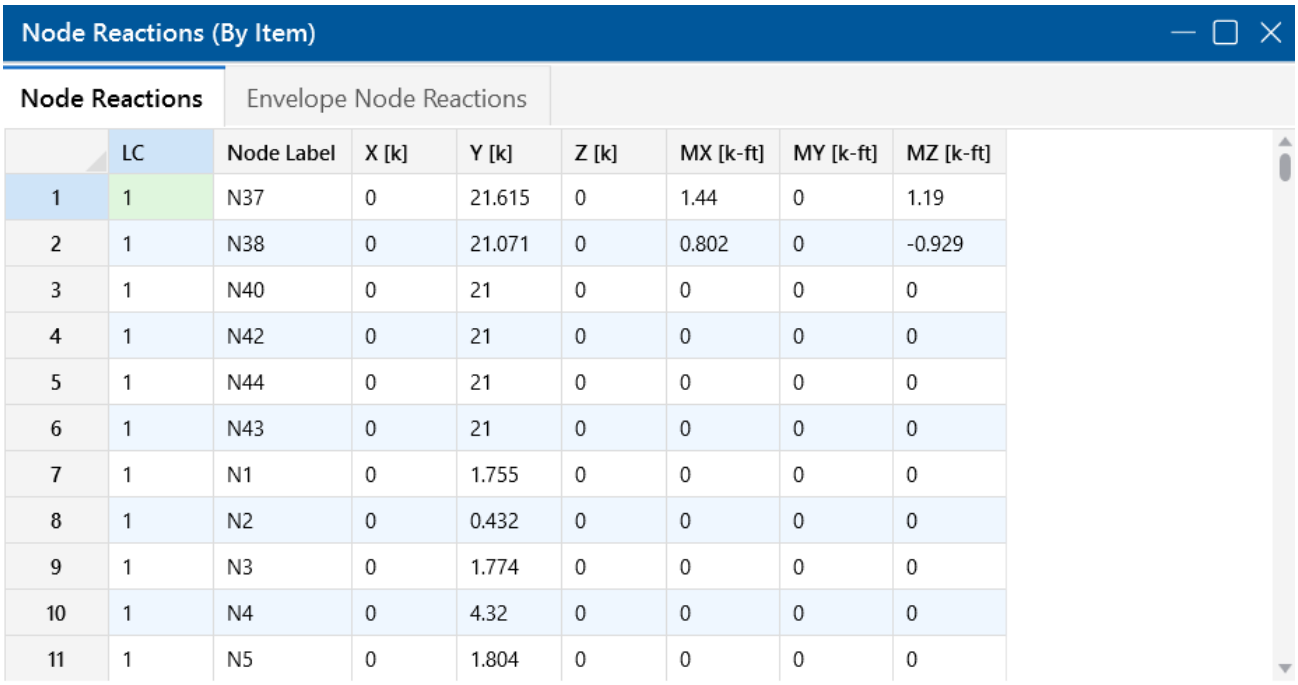
These are the deflections, or displacements, for every node in the structure. Each node has three values calculated, one for each of the three global degrees of freedom. Those being one translation and two rotations. The units for the displacements are shown at the top of each column. The rotations are shown in units of radians (1 radian = 57.3 degrees, approximately).

Note:

- See Spreadsheet Operations to learn how to use **Find**, **Sort** and other options.
- See Nodes to learn how to plot node results. You will NOT be able to plot the deflected shape for an envelope analysis. This is because the various maximum and minimum displacements probably correspond to different load combinations, so a deflected shape based on these values would be meaningless.

Node Reaction Results

Access the **Node Reactions Spreadsheet** by selecting it from the **Results** section in the **Explorer** panel.



Node Reactions (By Item)

Node Reactions | Envelope Node Reactions

	LC	Node Label	X [k]	Y [k]	Z [k]	MX [k-ft]	MY [k-ft]	MZ [k-ft]
1	1	N37	0	21.615	0	1.44	0	1.19
2	1	N38	0	21.071	0	0.802	0	-0.929
3	1	N40	0	21	0	0	0	0
4	1	N42	0	21	0	0	0	0
5	1	N44	0	21	0	0	0	0
6	1	N43	0	21	0	0	0	0
7	1	N1	0	1.755	0	0	0	0
8	1	N2	0	0.432	0	0	0	0
9	1	N3	0	1.774	0	0	0	0
10	1	N4	0	4.32	0	0	0	0
11	1	N5	0	1.804	0	0	0	0

These are the reactive forces **applied to the foundation** at its points of support. A positive reaction is applied in the upward vertical direction and a negative reaction is applied in the downward vertical direction. A positive moment is given according to the right-hand rule assuming the thumb is pointing in the positive global axis direction. Assuming a reaction has been calculated at all points of support, the total of the reactive forces in each direction should equal the total applied force in each direction. The units for the reactions are listed at the top of each column, and a total reaction for each direction is summed at the bottom of each translation column.

Retaining walls will also show up in this dialog. The total self-weight of the retaining wall and the total of any externally applied loads to the wall will be calculated here. Keep in mind that there are soil loadings implicit in the design of the wall as well, but those are not shown here.

For enveloped results the maximum and minimum reaction value is listed. The load combination producing the maximum or minimum is also listed, in the "**LC**" column.

Pedestals/Posts & Stem Walls

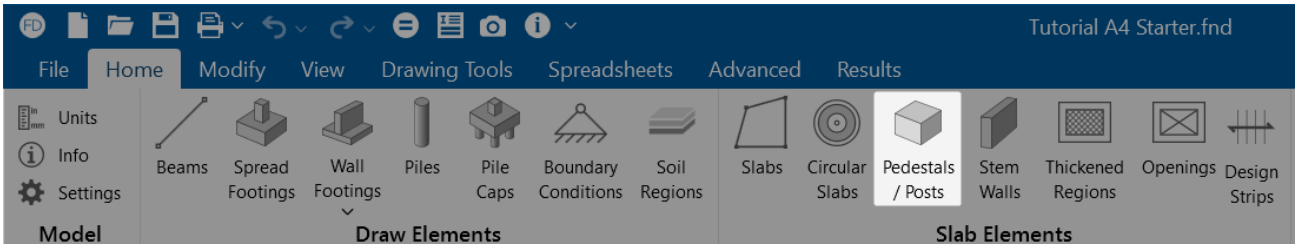
Aside from applying loads directly to the slab as a point or line load, loading can be applied to the slab with a pedestal, post or a stem wall. Pedestals consist of cast-in-place concrete, and are designed (complete with code check and rebar) similar to concrete columns. Posts allow you to define the outline of a post or column baseplate which bears directly on the slab. No design is performed for the post or baseplate itself, but the slab is checked for punching shear based on the post's outline. Stem walls are wall entities whose only purpose is a load transfer mechanism to a slab. There is no design of the stem wall itself, but its self-weight will be considered.

Pedestal, post and stem wall data may be viewed and edited in two ways: graphically in a model view or through the **Pedestals/Posts Spreadsheet** and **Stem Walls Spreadsheet**.

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

Draw Pedestals/Posts

To create new pedestals/posts, you can draw them using a Drawing Grid, Project Grid, or by clicking on existing nodes. You can set your pedestal/post properties up front, or you can modify these properties after drawing them.



To draw new Pedestals/Posts:

1. If there is not a model view already open, go to the **View** ribbon.
2. Click the **Open 3D Views** icon in the **Window** section to open a new view.
3. If you are not drawing on existing nodes, you need to create a drawing grid or define nodes in the **Node Coordinates Spreadsheet**.
4. Go to the **Home** ribbon.
5. Click the **Pedestals/Posts** icon in the **Draw Elements** section.
6. In the **Properties** panel, set the pedestal/post properties.
7. To start drawing pedestals/posts, click on nodes or grid intersections with the left mouse button. The coordinates of the closest point or grid intersection to your cursor are displayed on the screen near your cursor.

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

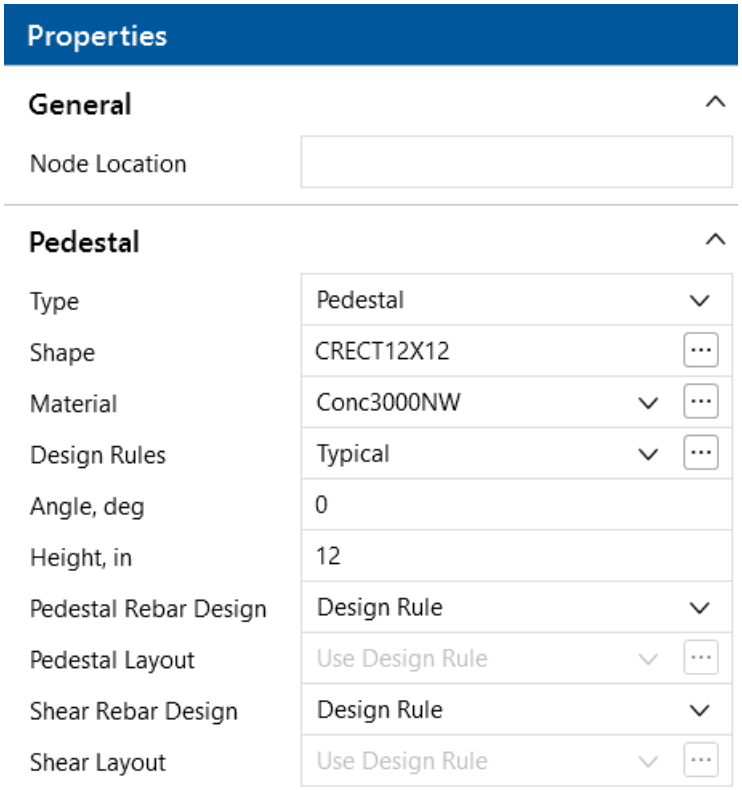
Note:

- To draw more pedestals/posts with different properties, press CTRL-D to recall the **Pedestals/Posts** settings.
- You may also specify or edit pedestals in the Pedestals/Posts Spreadsheet.
- You may undo any mistakes by clicking the Undo button.

Modify Pedestals/Posts Properties

There are a number of ways to modify pedestals/posts. You can view and edit the pedestal/post data in the **Pedestals/Posts Spreadsheet**, which can be accessed by selecting **Pedestals/Posts** from the **Data Entry** drop-down on the **Spreadsheets** tab of the ribbon, or by clicking **Pedestals/Posts** from the **Data Entry** section of the **Explorer** panel. You may also select a single pedestal/post (or multiple pedestals/posts) to view and edit the properties in the **Properties** panel.

The parameters shown are the same as those used to define new pedestals/posts.



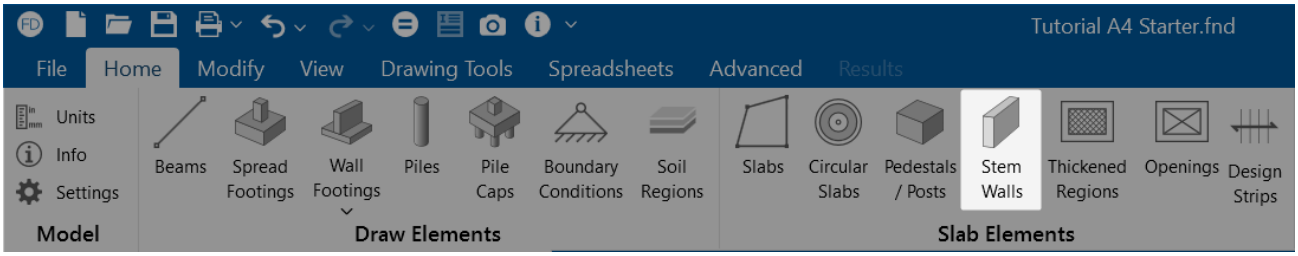
To modify Pedestals/Posts:

1. If there is not a model view already open, go to the **View** ribbon.
2. Click the **Open 3D Views** icon to open a new view.

3. Click a single pedestal/post, or select multiple pedestals/posts that you wish to modify.
4. If you have multiple elements selected, specify which element type to view properties for using the **Selection Properties** drop-down in the **Properties** panel.
5. Set the parameters for the pedestals/posts within the **Properties** panel.

Draw Stem Walls

To create new stem walls, you can draw them using a Drawing Grid, Project Grid, or by clicking on existing nodes. You can set your stem wall properties up front, or you can modify these properties after drawing them.



To draw new Stem Walls:

1. If there is not a model view already open, go to the **View** ribbon.
2. Click the **Open 3D Views** icon in the **Window** section to open a new view.
3. If you are not drawing on existing nodes, you need to create a drawing grid or define nodes in the **Node Coordinates Spreadsheet**.
4. Go to the **Home** ribbon.
5. Click the **Stem Walls** icon in the **Draw Elements** section.
6. In the **Properties** panel, set the stem wall properties.
7. To start drawing stem walls, click on nodes or grid intersections with the left mouse button to define the start point of the wall. The coordinates of the closest point or grid intersection to your cursor are displayed on the screen near your cursor.
8. The drawing tool will remain active after defining the start point of the stem wall. Continue clicking nodes or grid intersections with the left mouse button to further define the stem wall.
9. To stop drawing, click the **right mouse button** or press the **Esc** key.

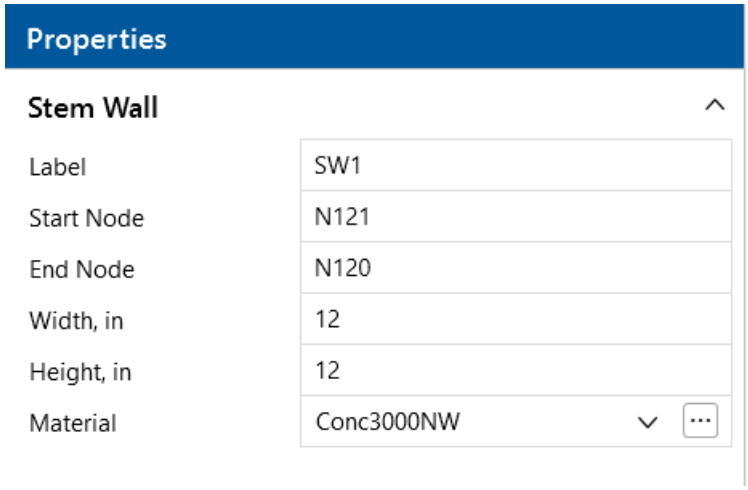
Note:

- To draw more stem walls with different parameters, press **CTRL-D** to recall the **Stem Walls** settings.
- You may also modify stem walls in the Stem Walls Spreadsheet.
- You may undo any mistakes by clicking the Undo button.

Modify Stem Walls

There are a number of ways to modify stem walls. You can view and edit the stem wall data in the **Stem Walls Spreadsheet**, which can be accessed by selecting **Stem Walls** from the **Data Entry** drop-down on the **Spreadsheets** tab of the ribbon, or by clicking **Stem Walls** from the **Data Entry** section of the **Explorer** panel. You may also select a single stem wall (or multiple stem walls) to view and edit the properties in the **Properties** panel.

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

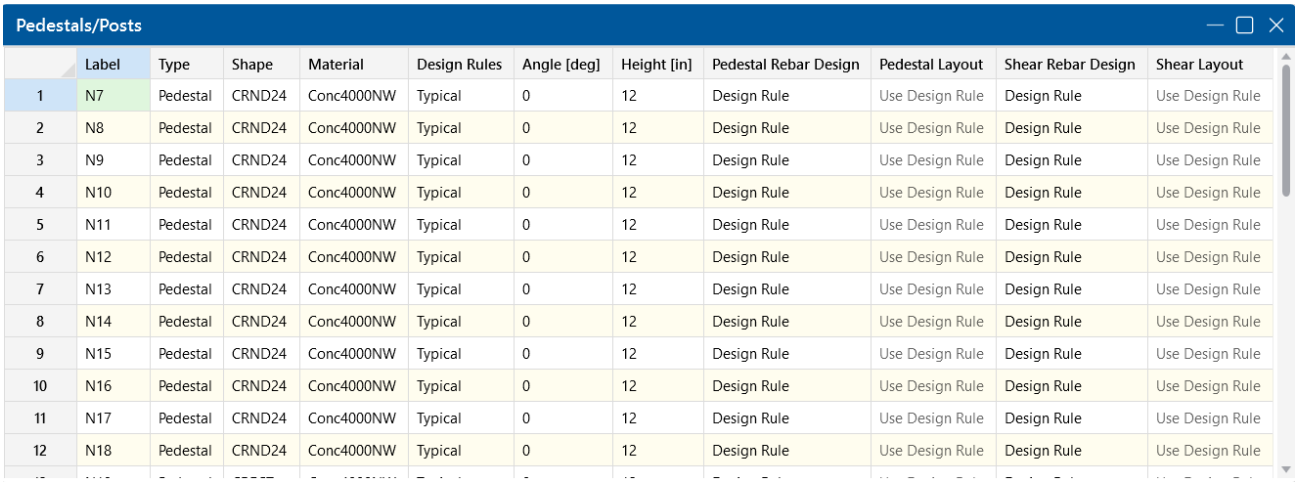


To modify Stem Walls:

1. If there is not a model view already open, go to the **View** ribbon.
2. Click the **Open 3D Views** icon to open a new view.
3. Click a single stem wall, or select multiple stem walls that you wish to modify.
4. If you have multiple elements selected, specify which element type to view properties for using the **Selection Properties** drop-down in the **Properties** panel.
5. Set the parameters for the stem walls within the **Properties** panel.

Pedestals/Posts Spreadsheet

Another way of adding or editing pedestals/posts is through the **Pedestals/Posts Spreadsheet**, which may be accessed by selecting **Pedestals/Posts** from the **Data Entry** drop-down on the **Spreadsheets** tab of the ribbon, or by clicking **Pedstals/Posts** from the **Data Entry** section of the **Explorer** panel.



	Label	Type	Shape	Material	Design Rules	Angle [deg]	Height [in]	Pedestal Rebar Design	Pedestal Layout	Shear Rebar Design	Shear Layout
1	N7	Pedestal	CRND24	Conc4000NW	Typical	0	12	Design Rule	Use Design Rule	Design Rule	Use Design Rule
2	N8	Pedestal	CRND24	Conc4000NW	Typical	0	12	Design Rule	Use Design Rule	Design Rule	Use Design Rule
3	N9	Pedestal	CRND24	Conc4000NW	Typical	0	12	Design Rule	Use Design Rule	Design Rule	Use Design Rule
4	N10	Pedestal	CRND24	Conc4000NW	Typical	0	12	Design Rule	Use Design Rule	Design Rule	Use Design Rule
5	N11	Pedestal	CRND24	Conc4000NW	Typical	0	12	Design Rule	Use Design Rule	Design Rule	Use Design Rule
6	N12	Pedestal	CRND24	Conc4000NW	Typical	0	12	Design Rule	Use Design Rule	Design Rule	Use Design Rule
7	N13	Pedestal	CRND24	Conc4000NW	Typical	0	12	Design Rule	Use Design Rule	Design Rule	Use Design Rule
8	N14	Pedestal	CRND24	Conc4000NW	Typical	0	12	Design Rule	Use Design Rule	Design Rule	Use Design Rule
9	N15	Pedestal	CRND24	Conc4000NW	Typical	0	12	Design Rule	Use Design Rule	Design Rule	Use Design Rule
10	N16	Pedestal	CRND24	Conc4000NW	Typical	0	12	Design Rule	Use Design Rule	Design Rule	Use Design Rule
11	N17	Pedestal	CRND24	Conc4000NW	Typical	0	12	Design Rule	Use Design Rule	Design Rule	Use Design Rule
12	N18	Pedestal	CRND24	Conc4000NW	Typical	0	12	Design Rule	Use Design Rule	Design Rule	Use Design Rule

The following data columns hold the primary data for the pedestals/posts:

Label

You may assign a unique label to any or all of the pedestals/posts. You can then refer to the pedestal/post element 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.

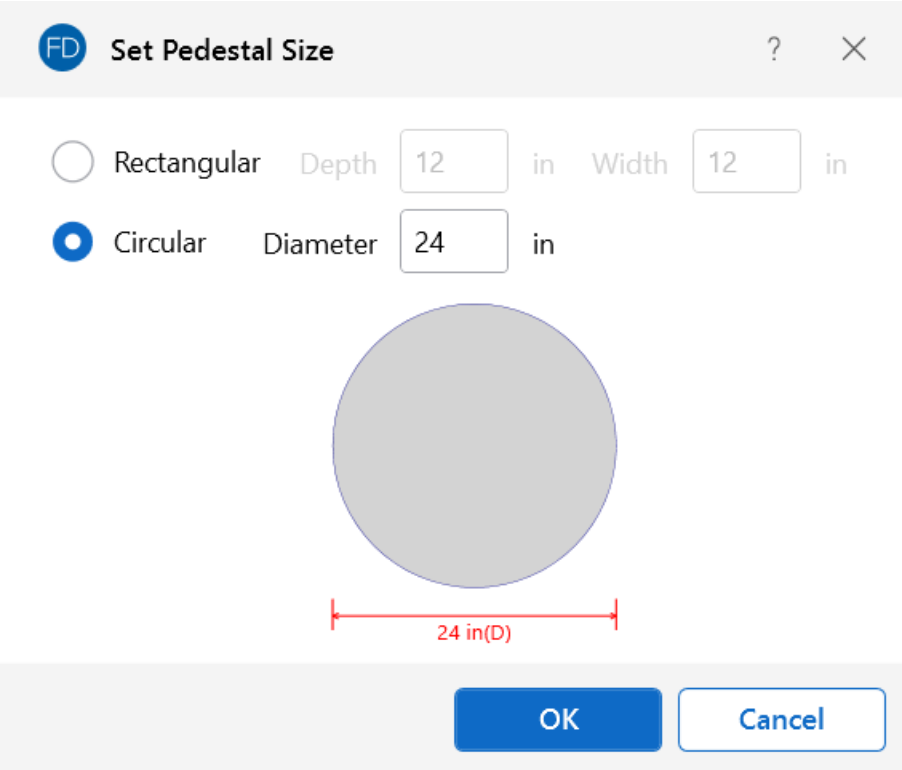
Type

You may specify whether this is a pedestal or a post. Posts do not have a material or design rule and their height is always set to zero since posts are only used for punching shear checks.

Shape

You may assign the size of your pedestals/posts in two ways:

1. You can select the ... (ellipsis) button to open the **Set Pedestal Size** window. This window allows you to input shape information. Once your parameters are set, choose **OK** and you will return to the **Pedestals/Posts Spreadsheet** with the defined shape/size added to the spreadsheet.



2. You can also enter the size by typing it directly into the cell. Use CRNDXX (where XX is the diameter of the pedestal) if you want a circular pedestal/post, or use CRECTYYxZZ (where YY is the depth and ZZ is the width of your pedestal) if you want a rectangular pedestal.

Material

This column allows you to assign the material definition that will be used in the analysis and design of the pedestal.

Design Rules

This allows you to choose the design rule specific to this pedestal. The Design Rule defines the pedestal reinforcing and cover dimensions.

Angle

This column allows you to enter an angle (in degrees) if you want your rectangular pedestal/post rotated about its local y-axis.

Height

This entry allows you to set the height of your pedestal from the top of slab.

Pedestal Rebar Design

This entry allows you to select whether the program will design the flexural reinforcement using the pedestal design rules or a custom rebar layout selected from the **Custom Rebar Database**.

Pedestal Layout

This column allows you to select a custom flexural rebar layout. For more information on how to create a custom layout, see Pedestal Rebar Layouts. If you do not select a custom layout, the program will design the reinforcement for you based on the information in your Design Rules.

Shear Rebar Design

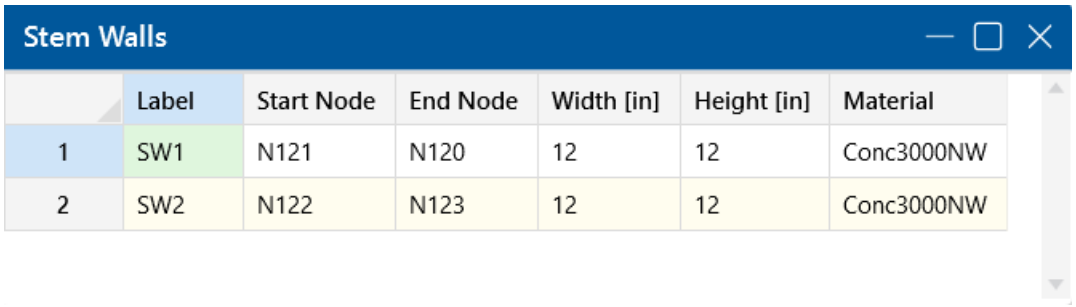
This entry allows you to select whether the program will design the shear reinforcement using the pedestal design rules or a custom rebar layout selected from the **Custom Rebar Database**.

Shear Layout

This column allows you to select a custom shear rebar layout. For more information on how to create a custom layout, see Shear Rebar Layouts. If you do not select a custom layout, the program will design the reinforcement for you based on the information in your Design Rules.

Stem Walls Spreadsheet

Another way of adding or editing stem walls is through the **Stem Walls Spreadsheet**, which may be accessed by selecting **Stem Walls** from the **Data Entry** drop-down on the **Spreadsheets** tab of the ribbon, or by clicking **Stem Walls** from the **Data Entry** section of the **Explorer** panel. This spreadsheet includes the primary data for stem walls, including end points, width, height and material.



	Label	Start Node	End Node	Width [in]	Height [in]	Material
1	SW1	N121	N120	12	12	Conc3000NW
2	SW2	N122	N123	12	12	Conc3000NW

The following data columns hold the primary data for the stem walls:

Label

You may assign a unique label to any or all of the stem walls and then refer to it by that 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.

Start/End Node

This shows which two nodes define the stem wall. These nodes can be edited directly in the spreadsheet.

Width/Height

These fields allow you to define the geometry of the stem wall. The height may be used for overturning calculations. The volume will be used for self-weight calculations.

Material

This column allows you to define the material properties assigned to a stem wall. The density for the material will be used for self-weight considerations.

Pedestal/Post - Design and Punching Shear Results

The **Pedestals/Posts Design and Punching Shear Spreadsheet** reports pedestal/post design information, as well as the two-way/punching shear code checks for the defined Pedestal/Post, as described in the following sections.

Note:

- This spreadsheet does not apply to beam design or individual spread footing/pedestal design.
- For detailed information on punching shear calculations, see Punching Shear - Design.
- There are no results for stem wall design or punching shear. A stem wall is only used as an analysis tool in the program.
- For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: **Pedestals**.

Pedestals/Posts Code Checks

Pedestals/Posts Design Values (By Item)

Pedestal Code Checks | Punching Shear Values | Punching Shear Geometries

	Label	UC	Gov LC	Shear UC	Gov LC	Dir	Phi Used	Vertical Reinf	Shear Reinf
1	N7	0.001	2	0	N/A	x	0.65	6#6	#4@12 in
2	N8	0.001	2	0	N/A	x	0.65	6#6	#4@12 in
3	N9	0.001	2	0	N/A	x	0.65	6#6	#4@12 in
4	N10	0.001	2	0	N/A	x	0.65	6#6	#4@12 in
5	N11	0.001	2	0	N/A	x	0.65	6#6	#4@12 in
6	N12	0.001	2	0	N/A	x	0.65	6#6	#4@12 in
7	N13	0.001	2	0	N/A	x	0.65	6#6	#4@12 in
8	N14	0.001	2	0	N/A	x	0.65	6#6	#4@12 in
9	N15	0.001	2	0	N/A	x	0.65	6#6	#4@12 in
10	N16	0.001	2	0	N/A	x	0.65	6#6	#4@12 in
11	N17	0.001	2	0	N/A	x	0.65	6#6	#4@12 in

The **Pedestal Code Checks** tab of the spreadsheet reports pedestal code checks and the reinforcing for the pedestals.

Note: Only reinforcement bars selected by the program are listed in this spreadsheet. If a custom rebar layout is used for a particular pedestal, the reinforcement section entries will report "CUSTOM" as a reminder that the user applied a custom rebar layout.

UC (Unity Check) and Governing LC

The **UC** value is the governing code check for the pedestal combined axial and bending check using the PCA Load Contour method. The **Gov LC** value is the load combination that resulted in the maximum code check.

Shear UC (Unity Check), Governing LC and Direction

The **Shear UC** value is the governing code check for the pedestal beam shear check. The **Gov LC** value is the load combination that resulted in the maximum code check. The **Direction** value is the local axis direction of the governing shear force in the pedestal.

The shear check is an envelope check. Therefore, the program will find the largest shear force (Vu) in each lateral direction and then keep track of the associated axial force (Pu) in the governing load combination(s). This means that there can be different axial forces for the check in each direction since the governing **Vux** and **Vuz** may come from different load combinations.

ACI 318-19 requires the consideration of shear interaction in y and z directions when both Vux/φVnx and Vuz/φVnz are larger than 0.5. In this case, the program recalculates the shear UC based on Equation 22.5.1.11. If the UC considering shear interaction happens to govern, this UC will be reported.

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 equations (a) and (b) in the table. 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 longitudinal bars on the tension face.

When ACI 318-14 and earlier codes are selected, Vc will be taken as zero if the load combination includes a tensile axial force. Since the shear check is an envelope check, any load combination resulting in a tensile axial force will cause Vc to be taken as zero. The program will not attempt to determine if the tensile load is "significant". In ACI 318-14 Section 22.5.7.1 the language was modified but RISA still considers Vc = 0 if there is tension present.

Aside from determining if the pedestal/post is in tension or not, the shear check does not consider any axial load in the pedestal.

Phi Used

This value is based on the strain in the tension steel in the pedestal and code provisions in ACI 318-14 Section 21.2 (ACI 318-11 Section 9.3.2).

Pedestal Reinforcement

The **Vertical Reinf** and **Shear Reinf** values are reinforcement requirements for the pedestal considering spacing and reinforcement requirements per the ACI code and the Design Rules provided for pedestals. If these sections report "CUSTOM" it means that the user has applied a custom rebar layout.

Punching Shear Values

Pedestals/Posts Punching Shear Values (By Item)

Pedestal Code Checks | **Punching Shear Values** | Punching Shear Geometries

	Label	UC	Gov LC	Location	Vuy[k]	Muz[k-ft]	Mux[k-ft]	Total Stress[ksi]	Phi*Vny[ksi]
1	N7	0.001	2	EDGE	0.375	0	0	0	0.164
2	N8	0.001	2	EDGE	0.358	0	0	0	0.164
3	N9	0.001	2	EDGE	0.343	0	0	0	0.164
4	N10	0.001	2	CORNER	0.265	0	0	0	0.164
5	N11	0	2	INTERI...	0.389	0	0	0	0.164
6	N12	0.001	2	INTERI...	0.468	0	0	0	0.164
7	N13	0.001	2	INTERI...	0.418	0	0	0	0.164
8	N14	0.001	2	EDGE	0.472	0	0	0	0.164
9	N15	0.001	2	INTERI...	0.721	0	0	0	0.164
10	N16	0.003	2	INTERI...	2.56	0	0	0.001	0.164
11	N17	0.006	2	INTERI...	4.967	0	0	0.001	0.164

The **Punching Shear Values** tab of the spreadsheet reports the basic / summary data for the punching shear code 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 more information on how this works, see Punching Shear - Design.

Vuy

This value represents the factored load transferred from the pedestal into the slab for the governing load combination. This value corresponds to the axial force in the Pedestal/Post after subtracting the

counteracting bearing pressure force that occurs within the punching shear perimeter.

Muz and Mux

These values represent the factored moments about the local axes of the Pedestal/Post.

Total Stress

The **Total Stress** is the direct shear stress plus the shear stress due to moment. For more information, see Punching Shear - Design.

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. For more information, refer to ACI 318-14 Section 22.6.5 (ACI 318-11 Section 11.11.7.2). 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 CSA A23.3-04 code provision 13.3.4.3 is used.
- For 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. 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 sections to account for this code requirement. RISA provides the calculations of As,min per Equation (8.6.1.2) in the Detail Report - Punching Shear Results. Please note that the As,min calculated is based on area of steel per foot, i.e., b_slab was taken as 12". The calculated As,min shall be multiplied by b_slab (specified in 8.4.2.2.3) to obtain the total area of steel required.

Punching Shear Geometries

Effective depth	4 in				
L1 along zz	10 in	Polar Moment Ixx	2773.333 in^4	Gamma xx	0.4
L2 along xx	10 in	Polar Moment Izz	2773.333 in^4	Gamma zz	0.4

Minimum Slab Reinforcement Requirements (per foot) provided over the effective slab width (bslab) per ACI318-19 Sec8.6.1.2

v_{uv}	0.36 ksi	alpha	20
lambda_s	1	bo	40 in
As_min	0.959 in^2/ft		

Punching Shear Geometries

Pedestals/Posts Punching Shear Geometries (By Item)

Pedestal Code Checks | Punching Shear Values | **Punching Shear Geometries**

	Label	Shape	Location	Effective Depth[in]	L1[in]	L2[in]	Polar Moment-xx[in^4]	Polar Moment-zz[in^4]	Gamma-x	Gamma-z
1	N7	CRND24	EDGE	26.5	58.675	45.35	1.82019e+6	1.87516e+6	0.431	0.37
2	N8	CRND24	EDGE	26.5	58.675	45.35	1.82019e+6	1.87516e+6	0.431	0.37
3	N9	CRND24	EDGE	26.5	58.675	45.35	1.82019e+6	1.87516e+6	0.431	0.37
4	N10	CRND24	CORNER	26.5	58.675	58.675	1.20621e+6	1.20621e+6	0.4	0.4
5	N11	CRND24	INTERI...	26.5	45.35	45.35	1.78834e+6	1.78834e+6	0.4	0.4
6	N12	CRND24	INTERI...	26.5	45.35	45.35	1.78834e+6	1.78834e+6	0.4	0.4
7	N13	CRND24	INTERI...	26.5	45.35	45.35	1.78834e+6	1.78834e+6	0.4	0.4
8	N14	CRND24	EDGE	26.5	45.35	58.675	1.87516e+6	1.82019e+6	0.37	0.431
9	N15	CRND24	INTERI...	26.5	45.35	45.35	1.78834e+6	1.78834e+6	0.4	0.4
10	N16	CRND24	INTERI...	26.5	45.35	45.35	1.78834e+6	1.78834e+6	0.4	0.4
11	N17	CRND24	INTERI...	26.5	45.35	45.35	1.78834e+6	1.78834e+6	0.4	0.4

The **Punching Shear Geometries** tab of the spreadsheet reports the basic geometry data used for the punching shear code check.

Note: For more information on many of these values, see Punching Shear - Design.

Shape

The **Shape** column shows the pedestal/post shape description assigned at the corresponding node location.

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 more information on how this works, see Punching Shear - Design.

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

The **L1** and **L2** variables represent the length of the critical section along the local axes.

Note: For a skewed Edge condition, L1 or L2 may have two values. For more information, see Punching Shear - Design.

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.

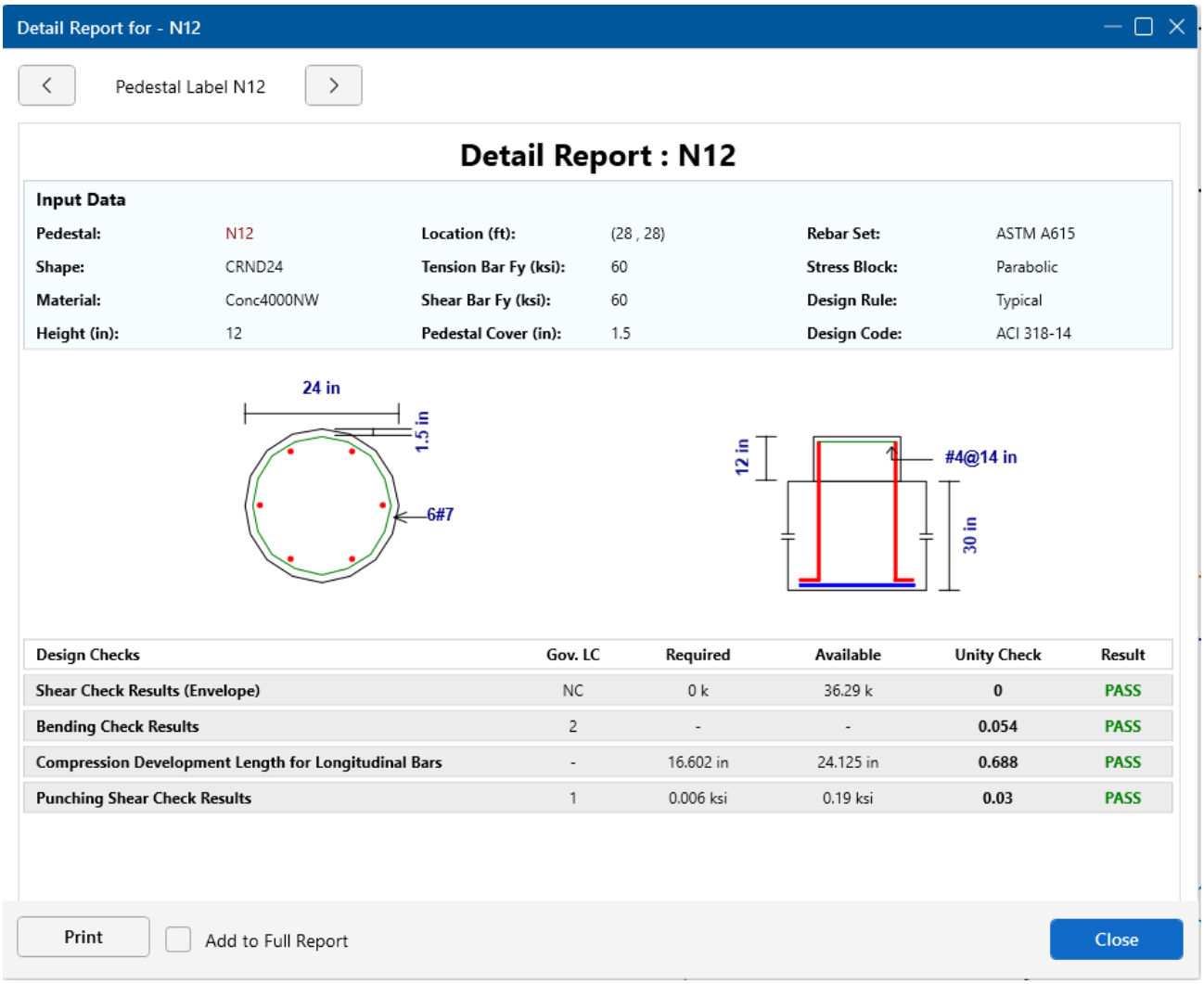
Gamma x and Gamma z

This value 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.

Pedestal/Post Detail Report

The **Detail Reports** for pedestals/posts show the overall geometry, materials, codes and analysis/design values used in coming up with the reinforcing and code check values. Below, we will walk through each section of the report.

Input Data and Graphics



The **Input Data** section lists the point label of the pedestal/post, material properties, reinforcement properties and design requirements, as applicable. The graphics in this section show a plan and section view of the geometry for the pedestal/post along with the actual pedestal reinforcement designed by the program.

Note: The program bases the number and spacing of bars on requirements per the applicable code regarding spacing and reinforcement limits.

Design Checks

Shear Check Results (Envelope)			NC	0 k	18.899 k	0	PASS
Along xx Vcx (k):	12.652	Along zz Vcz (k):	12.652		Shear Ties:	#4 @ 12 in	
Vsx (k):	18.899	Vsz (k):	18.899				
Vux (k):	0	Vuz (k):	0				
Vux/Φ*Vnx:	0	Vuz/Φ*Vnz:	0				
Gov LC:	NC	Gov LC:	NC				

The **Shear Check Results** summarize the shear capacities and demands along with the shear ties considered. The **Vc** values are the shear capacities of concrete only in each of the horizontal directions. The **Vs** values are the shear reinforcement capacities in each of the horizontal directions. The **Vu** values are the demand requirements in each of the horizontal directions. The code check value comes from the equation below:

$$\text{Interaction} := \frac{V_u}{\phi \cdot (V_c + V_s)}$$

The **Gov LC** value is the load combination that results in the governing code check, based on the highest shear force. The shear check is an envelope check. Therefore, the program will find the largest shear force (Vu) in each lateral direction and then keeps track of the associated axial force (Pu) in the governing load combination(s). This means that there can be different axial forces for the check in each direction since the governing Vux and Vuz may come from different load combinations.

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. 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 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). Per ACI 318-11 section 11.2.1.3, Vc will be taken as zero if the load combination includes a tensile axial force. Since the shear check is an envelope check, any load combination resulting in a tensile axial force will cause Vc to be taken as zero. The program will not attempt to determine if the tensile load is "significant". In ACI 318-14 Section 22.5.7.1 the language was modified but RISA still considers Vc = 0 if there is tension present.

Note: ACI 318-19 requires the consideration of shear interaction in x and z directions when both Vux/φVnx and Vuz/φVnz are larger than 0.5. In this case, the program recalculates the shear UC based on Equation 22.5.1.11. If the UC considering shear interaction happens to govern, a new section is added in the Detail Report to present this check.

Bending Check Results			2	-	-	**0.0007139**	**PASS**
Unity Check:	0.0007139	**Φ:**	0.65		**Parme β:**	0.65	
Pu (k):	0.174	**Mux (k-ft):**	0		**Muz (k-ft):**	0	
Pn (k):	374.978	**Mnx (k-ft):**	NA		**Mnz (k-ft):**	NA	
Gov LC:	2	**Mnox (k-ft):**	NA		**Mnoz (k-ft):**	NA	
Long. Bars:	4 #6	**% Steel:**	1.227				

Aside from determining if the pedestal/post is in tension or not, the shear check does not consider any axial load in the pedestal.

The **Bending Check Results** summarize the moment capacities and demands along with the longitudinal reinforcing considered. The **Pu** and **Mu** values are the axial and moment demand requirements. The **Pn** and **Mn** values are the axial and moment capacities based on the PCA Load Contour method. The **Mnox** and **Mnoz** are the uniaxial moment capacities. For more information on this method, see the Notes on the ACI 318 Building Code Requirements for Structural Concrete, a document provided by the PCA. The **Parme Beta** factor defaults to 0.65. The **Gov LC** is the load combination that produced the maximum unity check. The **Long. Bars** and the **% Steel** give the actual amount of steel and the number of bars in the pedestal. The **Unity Check** is a combined axial and bending unity check based on the interaction diagram for the pedestal.

The **Compression Development Length** section gives the values for longitudinal bar compression development length checks. The required length is the compression development length required per the ACI 318 code. The available length is the actual maximum value considering the thickness of slab and the bottom cover requirements. The equation for determining the available length is:

Lavailable = concrete thickness – cover – 2*reinforcing bar diameters – dowel bar diameter

The program uses a default value of the diameter of a #5 bar as the footing bar diameter for this calculation (0.625 in).

Lrequired/Lavailable gives the code check ratio.

Note: The compression development length check may not be the most essential check for current building codes but it is a criteria that may be applicable in specific conditions.

Punching Shear Check Results

Punching Shear Check Results			2	9.109e-5 ksi	0.164 ksi	0.001	PASS
Unity Check:	0.001	**Vuy (k):**		0.372	**Total Stress (ksi):**	9.109e-5	
Location:	INTERIOR	**Mux (k-ft):**		0	**Φ*vny (ksi):**	0.164	
Gov LC:	2	**Muz (k-ft):**		0			
Punching Shear Geometries							
L1 along zz (in):	38.5	**Polar I_{xx} (in^4):**		1.128e+6	**γ_{xx}:**	0.4	
L2 along xx (in):	38.5	**Polar I_{zz} (in^4):**		1.128e+6	**γ_{zz}:**	0.4	
Effective depth (in):	26.5						

The **Unity Check** gives a ratio of demand to capacity, while the **Gov LC** gives the load combination that produced this maximum unity check.

For an explanation of the rest of these values see both the spreadsheet result explanations above, as well as the Punching Shear - Design topic.

Design Considerations

Moment and Axial Force Thresholds

The program will ignore axial forces and moments that are below a certain threshold. If the moment or axial force is deemed to be inconsequential to the code check, the program will not include the interaction of that force. There are two thresholds that are considered:

- Axial Force Threshold: If **Pu < 0.01*f'c*Ag** for a LC, the axial force in the wall will be ignored for code checks for that LC.
- Moment Threshold: If **Mu < 0.01*d*Pu** for a LC, the moment in the wall will be ignored for code checks for that LC.

These two thresholds allow the concrete solver to work much more efficiently while having little to no effect on code check values.

Piles and Pile Caps

Piles and **Pile Caps** can be defined in RISAFoundation. Singular **piles** are used *when individual pile locations need to be defined to support slab elements*. These are best utilized for mat foundations where the mats are continuously supported by piles.

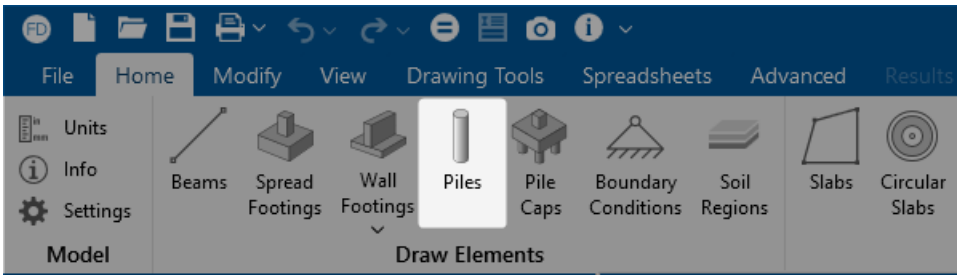
A **pile cap** is an element consisting of *multiple piles, a cap and a pedestal* that are defined at a single node in the model. A pile cap is *defined at a single point location*, typically where a column reaction is located.

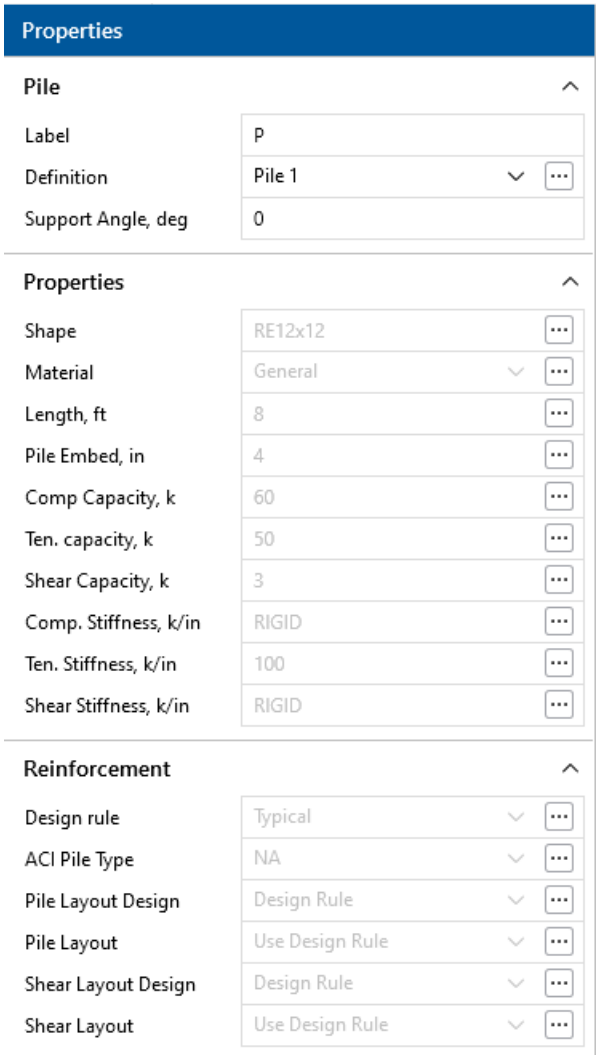
Here, we will explain the pile and pile cap interface and modeling techniques.

For more information on individual pile results, see Pile Results. For more information on pile cap results, see Pile Cap Results.

Draw Piles

To create new piles, you can draw them using a Drawing Grid, Project Grid, or by clicking on existing nodes. You can set your pile properties up front, or you can modify these properties after drawing them.





To draw Piles:

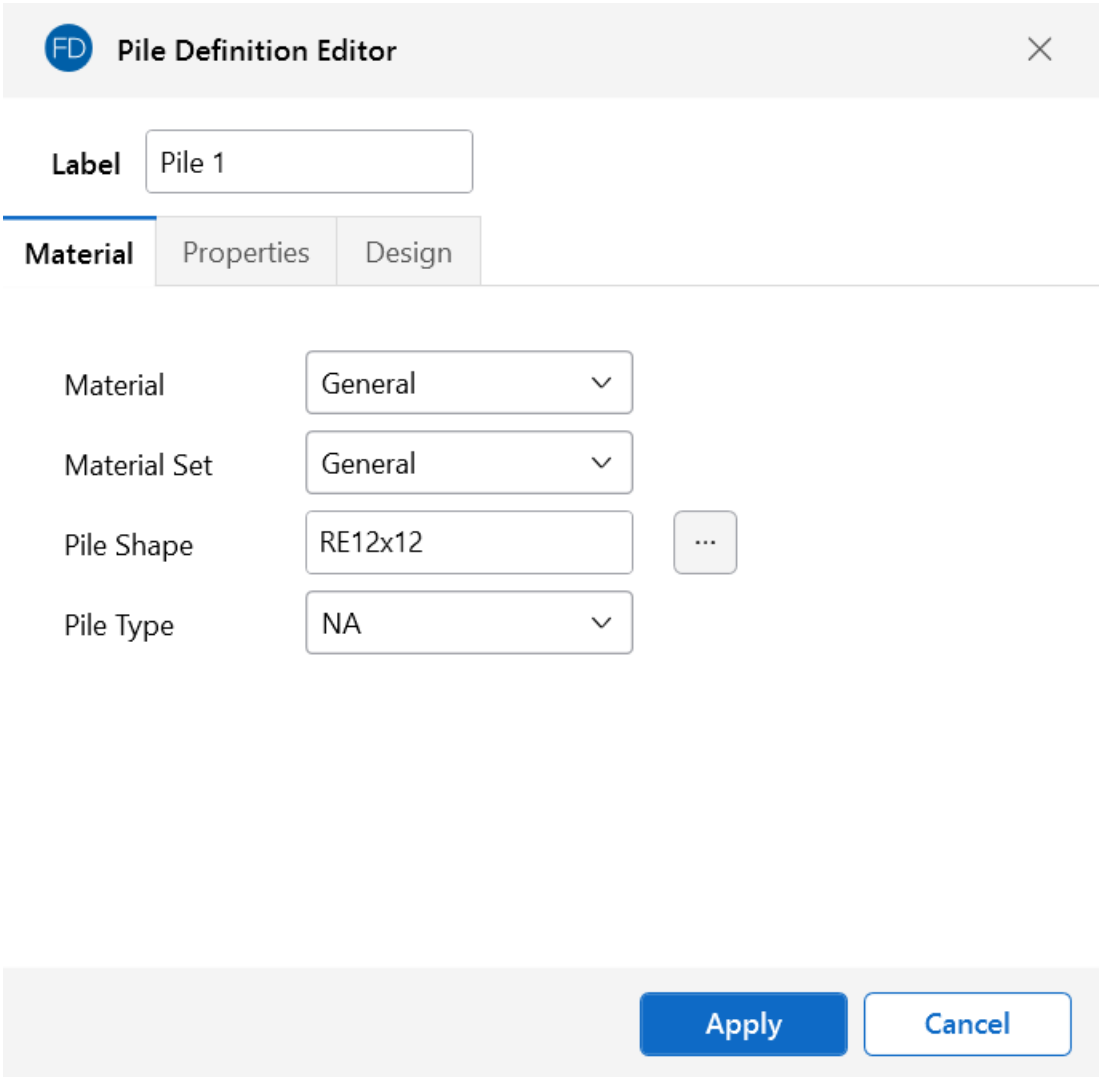
1. If there is not a model view already open, go to the **View** ribbon.
2. Click the **Open 3D Views** icon in the **Window** section to open a new view.
3. If you are not drawing piles at existing nodes, you need to create a drawing grid or define nodes in the **Node Coordinates Spreadsheet**.
4. Go to the **Home** ribbon.
5. Click the **Piles** icon in the **Draw Elements** section.
6. In the **Properties** panel, set the pile properties.
7. To start drawing piles, click on nodes or grid intersections with the left mouse button. The coordinates of the closest point or grid intersection to your cursor are displayed on the

screen near your cursor.

8. To stop drawing, click the **right mouse button** or press the **Esc** key.

Modify Pile Properties

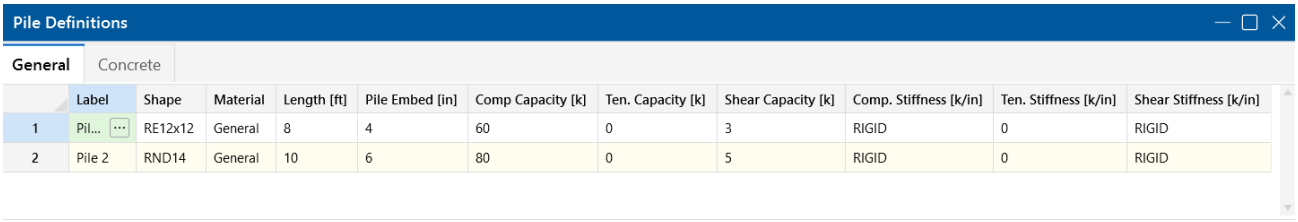
There are a number of ways to modify piles. You can view and edit the pile data in the **Pile Definitions Spreadsheet**, which can be accessed by selecting **Pile Definitions** from the **Data Entry** drop-down on the **Spreadsheets** tab of the ribbon, or by clicking **Pile Definitions** from the **Data Entry** section of the **Explorer** panel. You may also select a single pile (or multiple piles) to view and edit the properties in the **Properties** panel. Clicking the ··· (ellipses) button within the Definition input of the **Properties** panel or within the Label input of the **Pile Definitions Spreadsheet** will open the **Pile Definition Editor**.



This dialog provides a quick way to build Pile Definitions. Any updates made from within this dialog will automatically update the **Pile Definitions Spreadsheet**. Each of the items available in this dialog are explained in the following section.

Pile Definitions Spreadsheet

General Tab



Label

The **Label** provides a unique name for the pile definition.

Shape

For **General** piles, two shape options are provided: **Rectangle** and **Circular**. For other pile shapes, you can approximate them to one of these shapes (H-piles would just be added as a rectangle pile with the side dimension equal to either the depth or width). This shape will affect punching shear calculations.

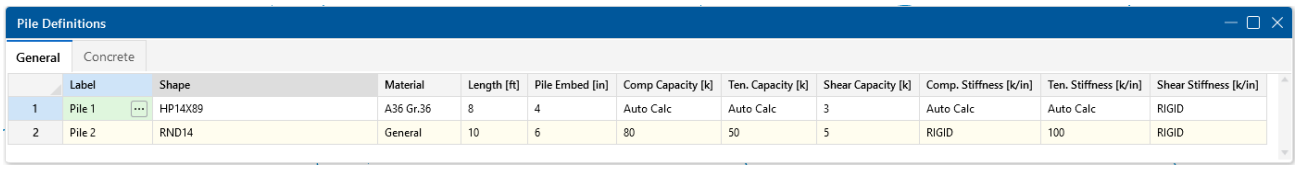
The **Hot Rolled Steel** shape database is provided when a hot rolled material is selected. Although the full HR Steel shape database is provided, steel piles are typically pipe and H-section shapes.

The **Wood** shapes are available when a wood material is selected. The program assumes a constant cross section along the pile's length in the soil.

The **Concrete** shapes are available when a concrete material is selected. The program assumes a constant cross section along the pile's length in the soil.

Material

The pile materials available are hot rolled steel, concrete and wood. The program will use the material properties to calculate the pile tensile and compressive capacities and stiffnesses.



Note: Selecting a general pile material and shape is consistent with the analysis and pile checks prior to RISAFoundation version 9.0.

Length

The total length of the pile in soil. The pile length is used for hot rolled steel, concrete and wood pile design calculations.

Pile Embedment

This is the depth that the pile is embedded in the pile cap. This value affects the effective depth "d" for pile cap design and punching shear calculations.

Note: The **Embedment Depth** field is only used to calculate "d" with Pile Cap entities. If you are using slabs with pile supports then this embedment depth is not considered for the slab design - though it is still considered for the punching shear check. For pile cap entities d = t_{slab} - pile embed - bot cover - $d_{bot\ bar}$. For slab entities d = t_{slab} - bot cover - $d_{bot\ bar}$. Thus, the bottom cover must include the embedment depth for slabs supported on piles but not for pile cap entities.

Pile Compressive and Tensile Capacities

For hot rolled steel, concrete and wood piles, the program will calculate these values based on the material properties and display *Auto Calc*. For general piles, the user will specify the pile allowable capacities. The program will use these capacities to do a design check of your general material piles.

Pile Shear Capacity

For all pile shapes and materials, the user will specify the pile allowable shear capacity. The program will use the user input value of the shear capacity to do a design check.

Note: The program currently calculates the static bearing capacity however it does not consider lateral pile design at this time.

Pile Compressive and Tensile Stiffness

For hot rolled steel, concrete and wood piles, the program will calculate the stiffness values based on AE/L and displays *Auto Calc*. The *Auto Calc* stiffness entry may be overridden by a user entered value.

For general piles, this entry allows you to define the stiffnesses for your piles. By default, the compression entry is set to completely RIGID. The tension entry is set to 0, meaning there is no resistance to tension.

Note:
- The default entries for general piles are typical for most pile cap designs. To define the piles as a spring support simply enter the pile stiffnesses.
- To get an approximate vertical stiffness of an individual pile, simply take the pile capacity and divide that by the maximum possible settlement.
- When Auto Calc is displayed for the tensile stiffness of concrete piles, the program assumes only the steel reinforcement will contribute to the tensile stiffness. However, because the reinforcement is designed after the analysis, the program makes an assumption for the analysis that the tensile stiffness is 5% of the compressive capacity.

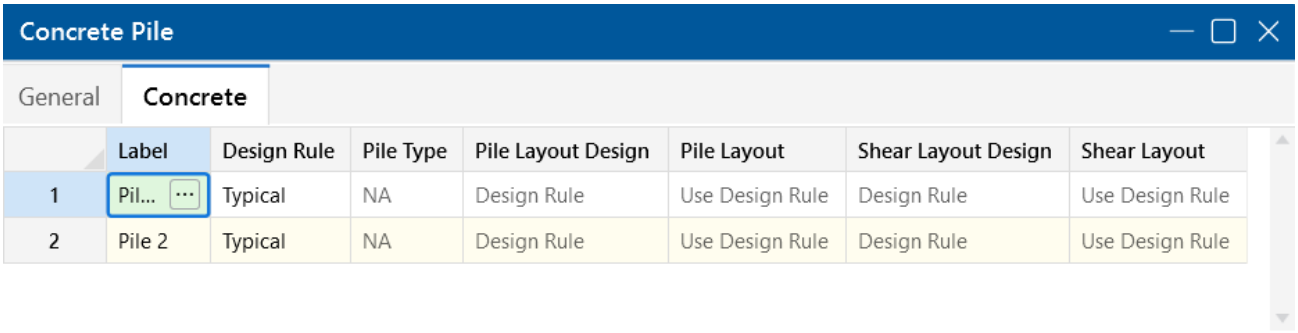
Pile Shear Stiffness

For all pile shapes and materials, this entry allows you to define the shear stiffness for your piles. By default, the shear stiffness entry is set to completely RIGID.

Note: To get an approximate lateral stiffness of an individual pile, simply take the pile capacity and divide that by the maximum possible lateral deflection.

Concrete Tab

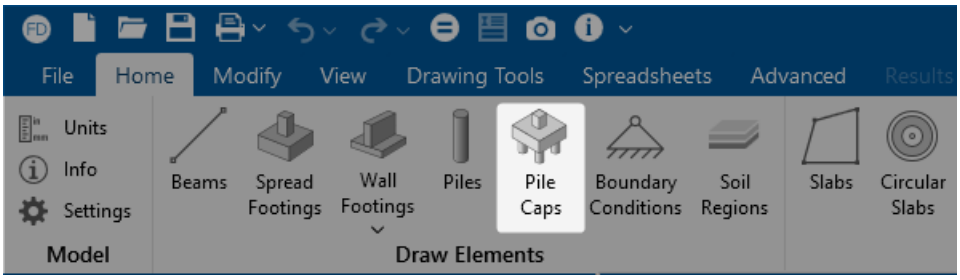
The concrete reinforcement can be specified in the **Concrete** tab of the **Pile Definitions Spreadsheet**. You can either use the reinforcement specified in the **Design Rules Spreadsheet**, or create a custom rebar layout to apply to the concrete pile.



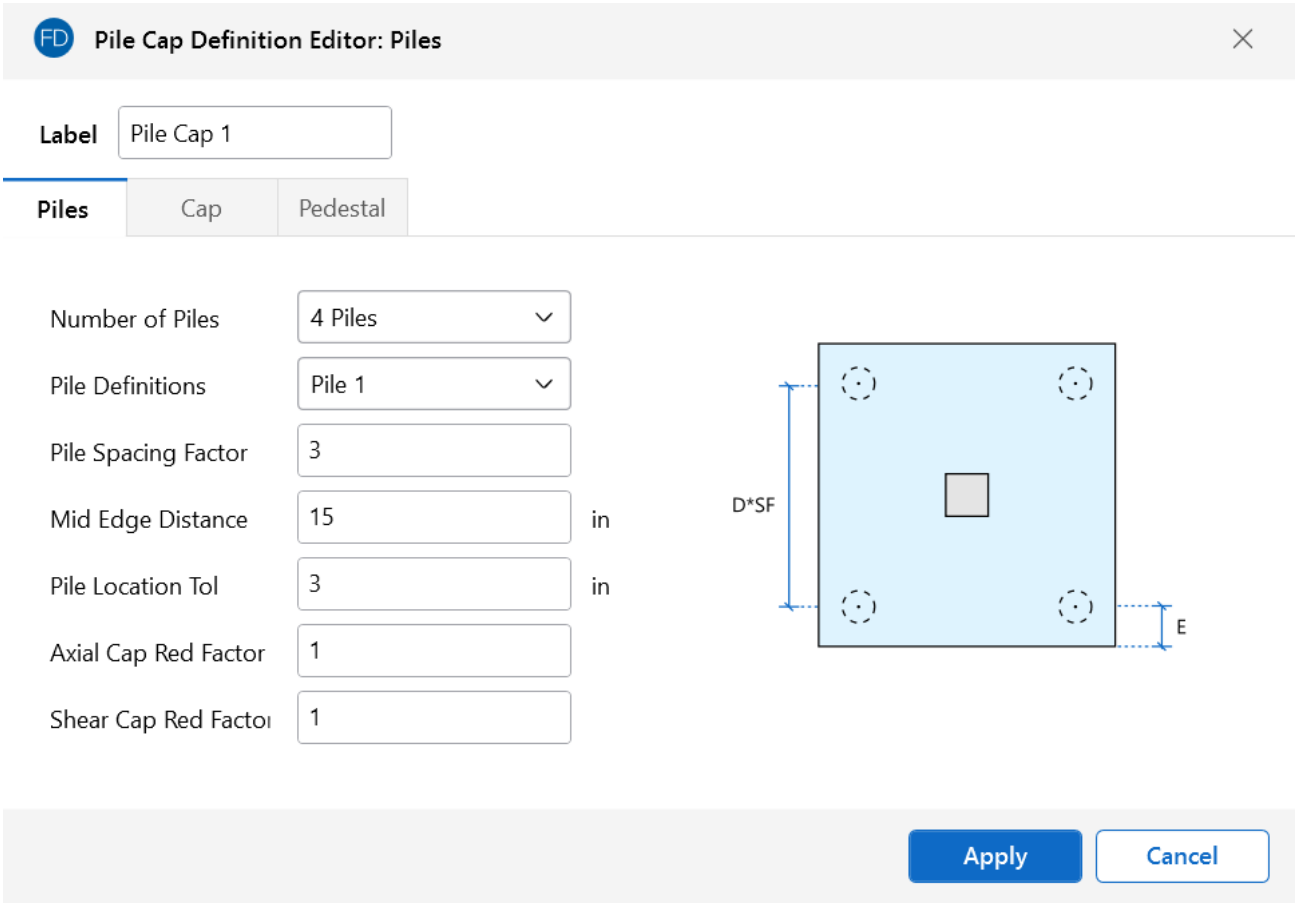


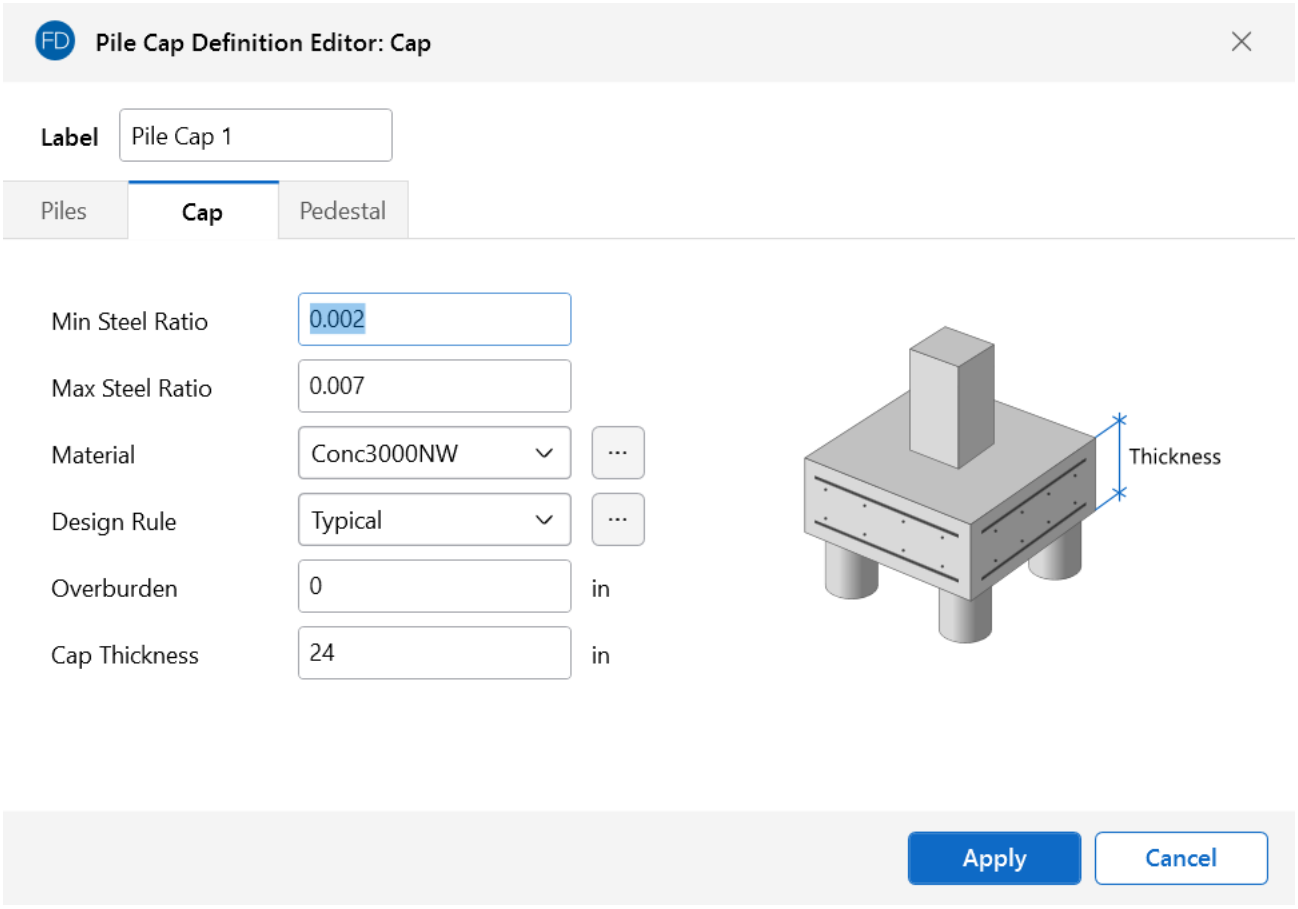
ACI 318-19 provides explicit equations in Table 13.4.2.1 on allowable compressive strength for piles depending on the pile types. When ACI 318-19 code is selected, the pile type can be modified in **Concrete** tab. Currently, RISAFoundation supports pile types (a)-(d) in this table.

Pile Caps



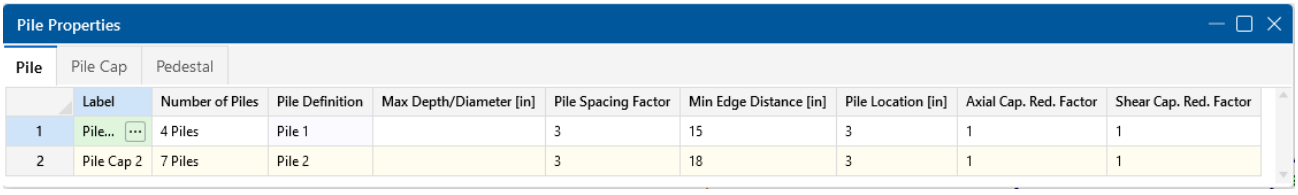
Pile caps can be drawn and modified similar to piles as discussed above. They can be defined directly within the **Pile Cap Definitions Spreadsheet**, which can be access by selecting **Pile Cap Definitions** from the **Data Entry** drop-down on the **Spreadsheets** tab of the ribbon, or by clicking **Pile Cap Definitions** from the **Data Entry** section of the **Explorer** panel. You may also define pile caps through the **Pile Cap Definition Editor**, which can be accessed by clicking the ··· (ellipses) button within the Definition input field of the**Properties** panel, or within the Label input field of the **Pile Cap Definitions Spreadsheet**


Pile Cap Definition Editor: Piles
Label
Pile Cap 1
Piles
Cap
Pedestal
Number of Piles
4 Piles
Pile Definitions
Pile 1
Pile Spacing Factor
3
Mid Edge Distance
15
in
Pile Location Tol
3
in
Axial Cap Red Factor
1
Shear Cap Red Factor
1
D*SF
E
Apply
Cancel



This dialog provides a quick input selection screen for building Pile Cap Definitions. Any updates made from within this dialog will automatically update the **Pile Cap Definitions Spreadsheet**. Each of the items here is explained below.

Pile Tab



Label

This label provides a unique name for the pile cap definition. When clicking in this column, a ... (ellipsis) button will become available. This will open the **Pile Cap Definition Editor** and allow you to update all spreadsheet entries via a single dialog.

Number of Piles

This selection defines the number and layout of piles. These pile layouts were derived from the CRSI manual on pile caps. Here are the standard layouts from the CRSI manual.

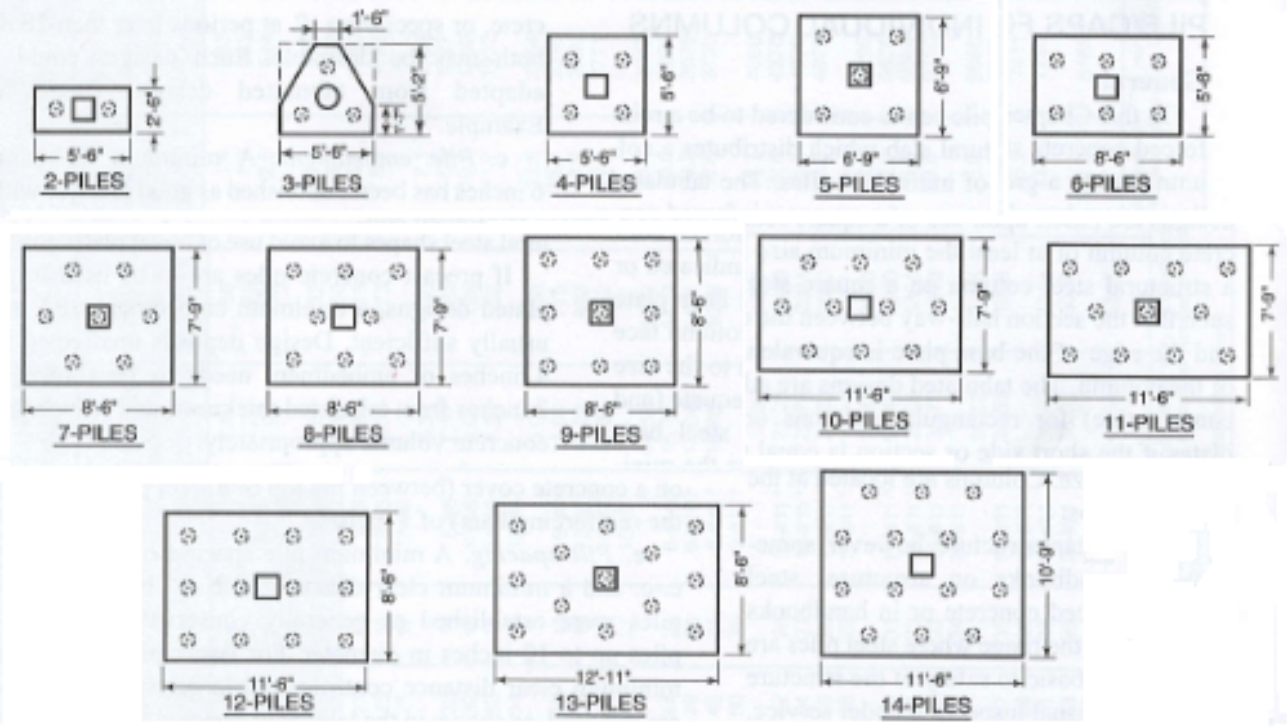


Figure 13-2 from CRSI Design Handbook 2008

Note:

- RISAFoundation uses the pile layouts from CRSI, but allows you to change the spacing, edge distance and other properties to customize your pile cap design.
- Currently, the program only includes pile cap design up to 14 piles. For custom pile caps and pile caps that have greater than 14 piles, you can use a slab with singular pile elements.
- Once you have defined your pile caps and applied them to your model, solve the model and look at the Pile Cap Detail Report for exact geometry and locations of piles.

Pile Definition

This allows you to choose a pile type for your pile cap from the Pile Definitions Spreadsheet.

Max Depth/Diameter

This entry allows you to specify the maximum depth/diameter to use for piles when considering potential shapes for the pile design.

Pile Spacing Factor

This value defines the center-to-center spacing of piles. This factor multiplied by the diameter/side dimension of the pile defines the spacing. This factor is typically used to help define capacity reduction factors for closely spaced piles.

Min Edge Distance

This value defines the minimum distance from the centerline of pile to the edge of pile cap.

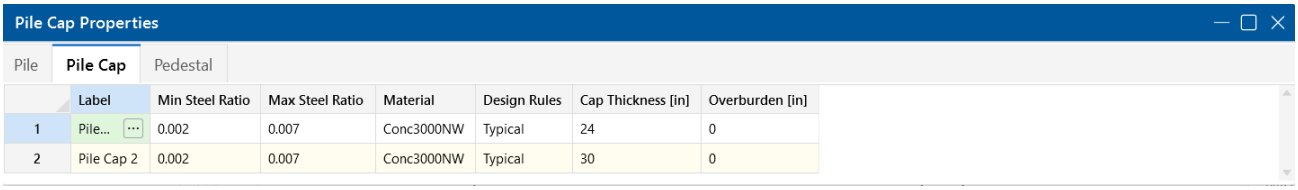
Pile Location

This entry allows you to define a field placement tolerance that is acceptable for your pile caps. This affects two checks in the program: the moment demand calculation and the one way shear capacity calculation.

Axial and Shear Capacity Reduction Factors

These factors will reduce the pile capacities from the Pile Definitions Spreadsheet. The value in the Pile Definitions Spreadsheet will be divided by this value to produce a reduced capacity based on closely spaced pile groups.

Pile Cap Tab



Reinforcement Ratios

When ACI 318-19 is selected for the design code, the ratio of deformed shrinkage and temperature reinforcement area to gross concrete area must be greater than or equal to 0.0018 following the stipulations of ACI 318-19 Section 24.4.3.

When other ACI 318 editions are selected, acceptable steel ratios are controlled by ACI 318-14 Section 24.4.3 (ACI 318-11 section 7.12.2.1), which stipulates that the minimum steel ratio for Grade 40 or 50 steel is .0020. For Grade 60 or greater the ratio is 0.0018.

Note: When you are only viewing three decimal places, the 0.0018 appears as 0.002, though we are really using 0.0018. To change the number of decimal places displayed, go to **File** > **Application Settings** > **Input** tab.

In addition, the codes limit the maximum steel that can be used. For example, ACI 318-14 limits the minimum strain on the reinforcing steel to 0.004.

Note: If the Max Steel Ratio you define does not meet minimum requirements or strength requirements, we will override your ratio and display a message indicating this in the detail report.

Material

This allows you to specify the pile cap material using the definitions from the **Materials Spreadsheet**.

Design Rules

This allows you to specify the design rule considered for the pile cap from the **Design Rules Spreadsheet**. For pile caps, we take information from the **General**, **Footing/Pile Cap** and the **Pedestal** tabs of the **Design Rules Spreadsheet**.

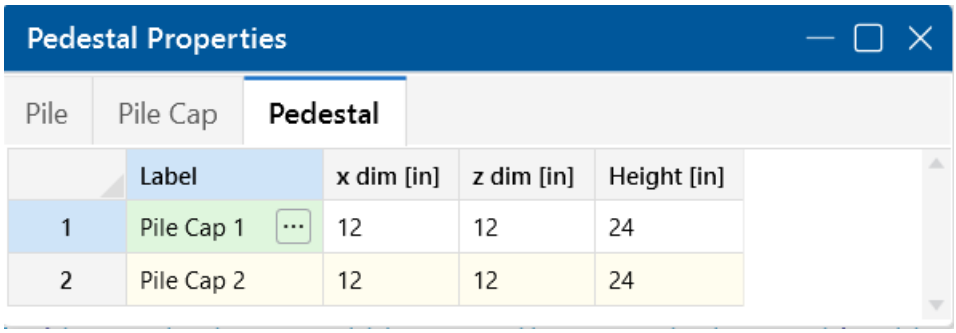
Cap Thickness

This value defines the total thickness of your pile cap.

Overburden

This entry allows you to define a soil overburden on your pile cap. This overburden weight is added to the weight of the slab for pile demand calculations. These pile forces affect flexural and shear design for the pile cap.

Pedestal Tab



The pedestal dimensions are entered in the fields on this tab of the spreadsheet. The **x dim** and **z dim** values are the pedestal dimensions parallel to the local x- and z-axes of the pile cap, respectively. The **Height** is the distance from the upper surface of the pile cap to the top of the pedestal.

Note: Pedestals are always located at the centroid of the pile group.

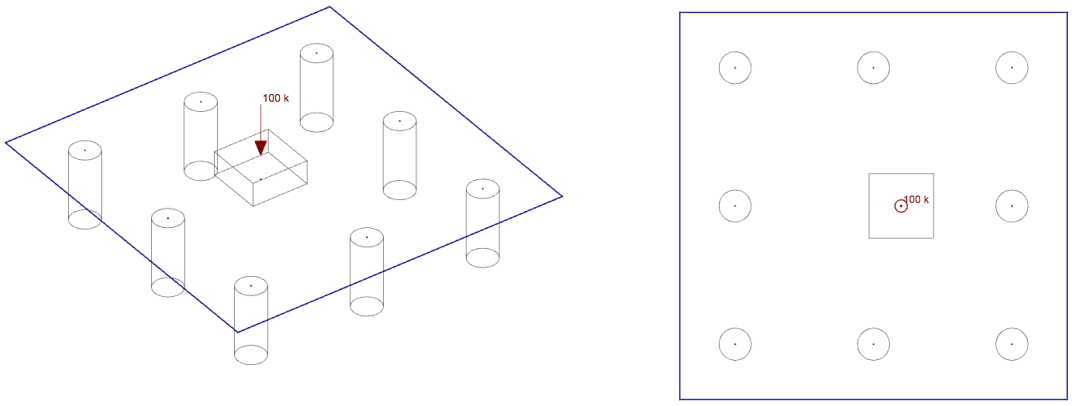
Pile and Pile Cap Modeling Considerations

Modeling with Piles, Slabs and Pedestals vs Pile Caps

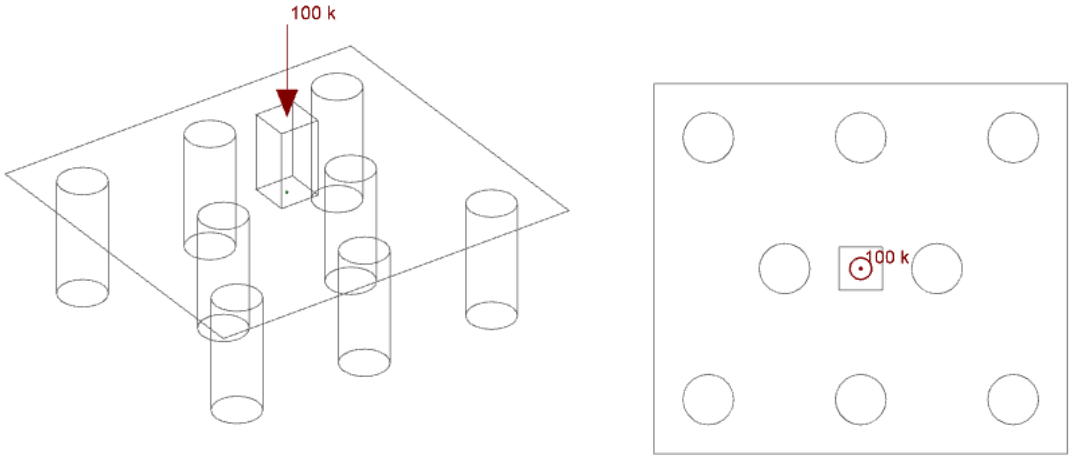
As discussed previously, the general functionality of the program is that piles would be used in scenarios where a mat slab was drawn and piles are located at specific locations under the slab. Pile caps, on the other hand, would be used at a specific point location to support a reaction.

The **Pile Cap** feature has some limitations and can not be applied to all situations. When such a situation is encountered, using singular piles, slabs and pedestals within the program will almost always allow you to overcome those limitations. The piles, slabs and pedestal option allows you to create almost any custom geometries you need to consider.

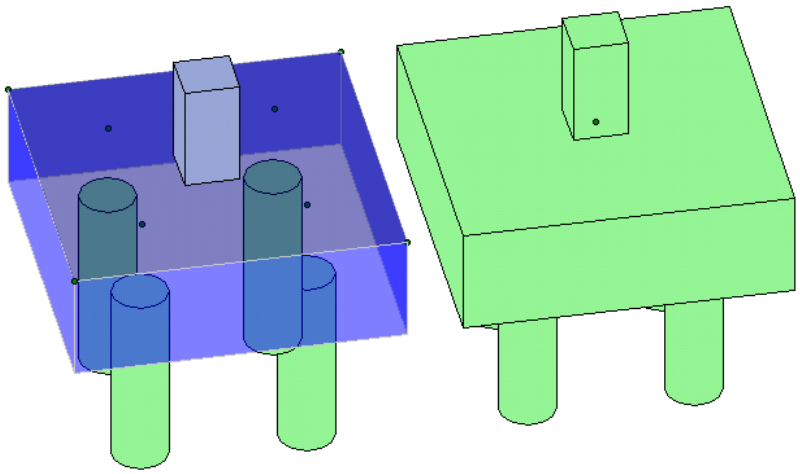
The pile cap pile layouts are configured based on the CRSI design manual layouts. Thus, these configurations can not be manipulated. However, it is possible to simply create a pile cap for a concentrated force using singular piles, a slab and a pedestal. Simply draw your pile cap as a slab element, add the pedestal using the draw pedestals tool, and locate/draw your piles according to your specific pile configuration. Below is an example where this was done.



Compare this with CRSI layout for eight piles:



From this example, we can also see that locating a pedestal at an eccentric location to the pile group is possible. It is also possible to model identical configurations using piles, slabs and pedestals compared to a singular pile cap element. Doing this considering identical loading may lead to nearly identical results. Differences in results can often be explained by differences in rigidity assumptions between these two approaches for modeling a pile cap.



Rigidity Assumptions

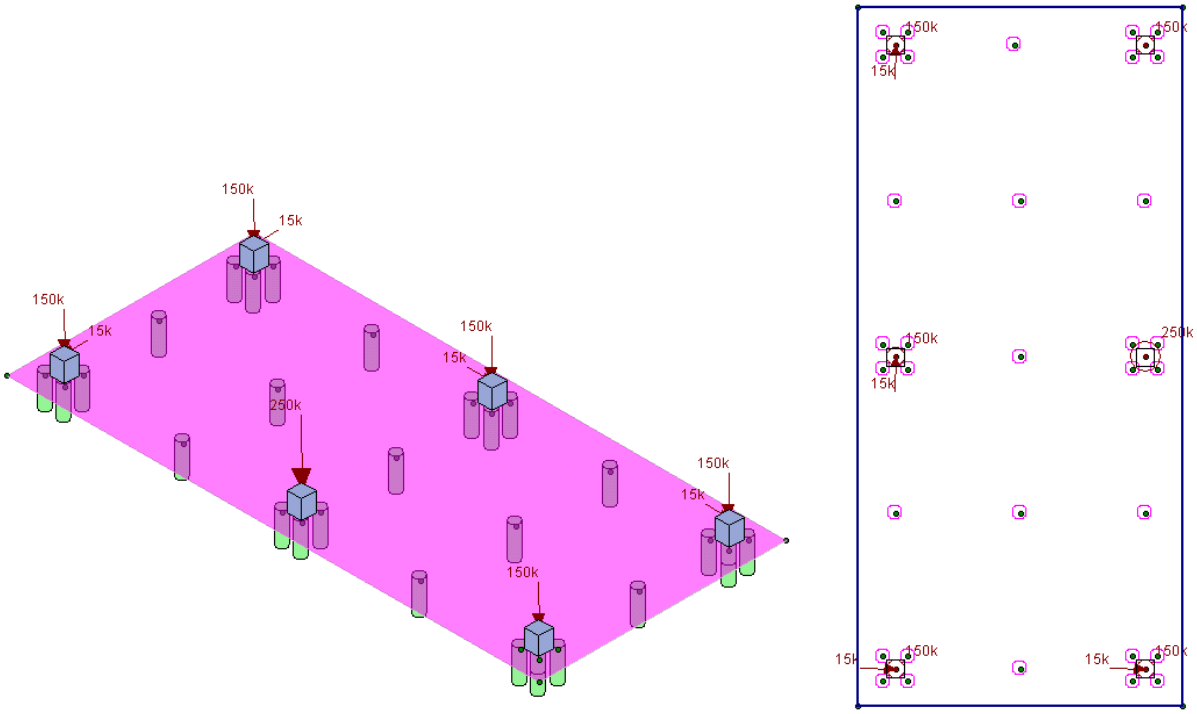
Keep in mind that pile cap entities assume the cap is fully rigid. Slabs supported with piles assume semi-rigid behavior. Thus, the slab entity will place load in the piles based on the rigidity of the cap which can alter the force distribution into the piles.

Also, the default stiffness for general piles is completely rigid. Thus, you can get very different distributions of force to these two seemingly identical entities. However, if you lower the pile stiffness so that it is not completely rigid, your pile force distribution should be much more similar.

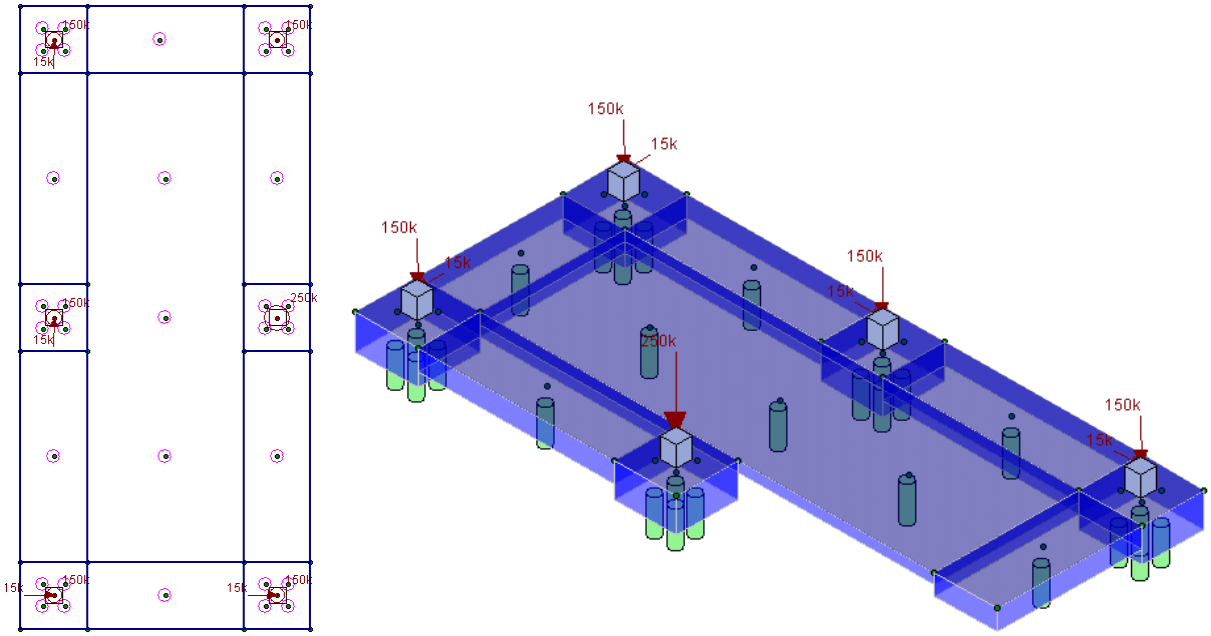
If the slab is relatively thick such that the slab pile cap behaves in a rigid manner, and the pile stiffness values are not set to be completely rigid, you will likely get similar pile force distributions between the two pile cap entities.

Pile Groups and Thickened Portions of a Slab within a Mat

Many times there are regions within a mat foundation where large loads are coming down and a pile group is required. The program does not allow you to define a pile cap within a slab, but you can simply draw in closely spaced individual piles within the slab element near the reaction. See the example below where large column loads are coming down on a mat foundation.



In this scenario, the slab is one continuous thickness over the entire mat. If the slab was thickened at the pile groups and around the exterior, we would simply need to draw multiple slab elements adjacent to each other with varying thickness. If two slab elements are adjacent to each other, they will have full fixity to each other. Alternatively, we can use the Modify Thickness tool on the Modify Ribbon Toolbar to adjust the slab thickness at certain locations within the slab element. Drawing multiple slab elements adjacent to each other may look something like this:



The image on the right shows a rendered view with part of the slab cut away to see the differences in thickness.

Pile and Pile Cap Limitations

The program does not consider battered piles.

Pile - Results

Results for individual piles can be found in the **Pile Design Results Spreadsheet**, which may be accessed by clicking **Results Browser** from the **Results** toolbar and clicking **Pile Design**. Or, click **Pile Design** from the **Results** section of the **Explorer** panel.

Pile Design Checks

Pile Design Checks (By Item)

Pile Code Checks | Punching Shear Values | Punching Shear Geometries | Pile Reinforcement (Concrete)

	Label	Pile	Shear UC	Shear[k]	Shear LC	Soil Comp UC	Pile Comp UC	Comp[k]	Comp LC	Soil Tens UC	Pile Tens UC	Tens[k]	Tens LC
1	N49	Pile 1	0	0	NC	NC	0.418	71.376	1	NC	0	0	NC
2	N50	Pile 1	0	0	NC	NC	0.399	68.178	1	NC	0	0	NC
3	N51	Pile 1	0	0	NC	NC	0.301	51.372	1	NC	0	0	NC

Pile UC Checks

The Shear, Compression and Tension UC values give the ratio of demand forces to allowable forces.

- For general piles, the allowable forces are pulled directly from the Pile Definitions Spreadsheet. The UC ratios are given for the load combination that produces the worst case UC.
- For hot rolled steel, wood and concrete piles, the compressive and tensile capacities are calculated by the program based on the material, cross sectional area and design code selected in the Model Settings.
- For concrete piles, only the reinforcement is considered in the tensile capacity.

Forces

The shear, compression and tension forces are given for the load combination that produces the worst case UC.

Note:

- The program does not consider the self-weight of the piles as part of the pile's compression/tension forces. Therefore, users should account for pile self-weight by manually applying a force to account for the pile's self-weight. For piles part of a pile cap system, the user should manually apply a force on the pile cap, which then would distribute to the supporting piles. For piles supporting slabs, the pile self-weight can be applied as a point load directly on top of each respective pile.
- The shear value reported is directionless and taken as:

$$V = \sqrt{{V_X}^2 + {V_Z}^2}$$

Gov LCs

These are the governing load combinations for each type of force which produce the highest unity check.

Soil UC Checks

The soil unity checks are the static pile capacities for tension and compression.

The **Soil Comp UC** is the ratio of the applied compressive force to the compressive capacity. The compressive capacity is equivalent to the sum of the skin friction at each layer and the tip bearing capacity at the layer of soil where the pile terminates. The capacity is divided by the pile factor of safety set in the Model Settings. When determining the soil compressive capacity, the program uses the following equation (Equation reference: Joseph E. Bowles, *Foundation Analysis and Design, 5th Ed.*):

$$Q_u = \frac{Q_p + \sum Q_s}{FS}$$

- Qu : Ultimate pile capacity in compression
- Qp : Bearing capacity in the respective soil layer, $Q_p = A_p \times q_p$
  - Ap : Area of pile including the soil plug
  - qp : Unit tip-bearing
- ∑ : Summation process over the soil layers making up the soil profile along the length of the pile shaft
- Qs : Skin Friction at each soil layer, $Q_s = p \times \Delta L \times q_s$
  - p : Perimeter of pile cross section including soil plug
  - ΔL: Length of pile in soil layer
  - qs : Unit skin friction
- FS : Pile Factor of Safety input in the Model Settings

Note: The program does not consider the self-weight of the pile as part of the axial force in the pile. See the Forces section for information on how to account for pile self-weight.

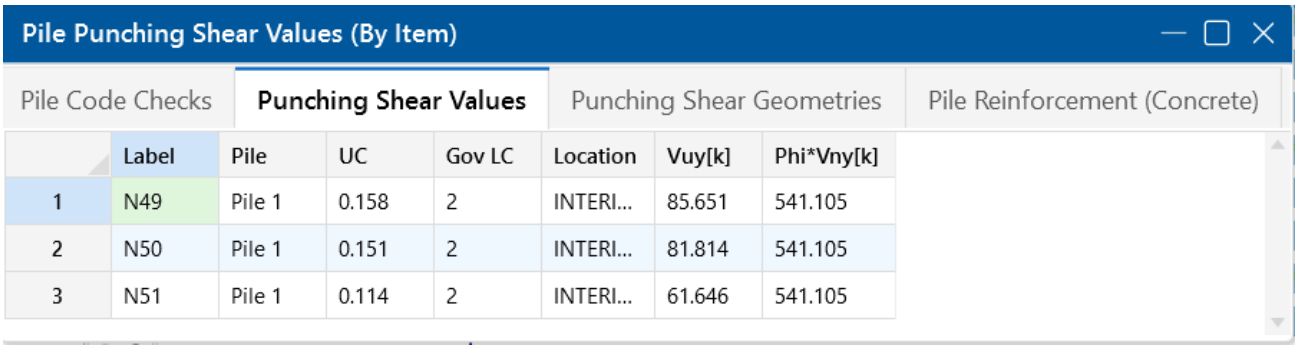
The **Soil Tens UC** is the ratio of the applied tensile force to the tensile capacity. The tensile capacity is equivalent to the weight of the pile plus the sum of the skin friction at each soil layer. The Soil Tensile UC is divided by the pile factor of safety specified in the Model Settings. When determining the soil tensile capacity, the program uses the following equation (Equation reference: Joseph E. Bowles, *Foundation Analysis and Design, 5th Ed.*):

$$Q_u = \frac{W + \sum Q_s}{FS}$$

- Qu : Ultimate pile capacity in tension
- W : Weight of the pile, $W = \gamma \times A_p \times L$
  - γ : unit weight of pile
  - Ap : Area of pile
  - L : Length of pile
- ∑ : Summation process over the soil layers making up the soil profile along the length of the pile shaft
- Qs : Skin Friction at each soil layer, $Q_s = p \times \Delta L \times q_s$
- p : Perimeter of pile cross section
- ΔL: Length of pile in soil layer

- qs : Unit skin friction
- FS : Pile Factor of Safety set in Model Settings

Punching Shear Values



	Label	Pile	UC	Gov LC	Location	Vuy[k]	Phi*Vny[k]
1	N49	Pile 1	0.158	2	INTERI...	85.651	541.105
2	N50	Pile 1	0.151	2	INTERI...	81.814	541.105
3	N51	Pile 1	0.114	2	INTERI...	61.646	541.105

UC Check

This is the ratio of the punching shear demand to the punching shear capacity for the worst case load combination.

Gov LC

This is the load combination that produces the worst case punching shear for each individual pile.

Location

This helps to define the geometry for the shear perimeter. The program calculates a shear perimeter assuming a corner failure, an edge failure and an interior failure. The governing shear perimeter is given here.

Vuy

This is the factored punching shear demand.

Phi*Vny

This is the punching shear capacity of the slab/pile cap at this pile location. For more information on this calculation, see Pile Caps - Design.

Punching Shear Geometries

Pile Punching Shear Geometries (By Item)

Pile Code Checks | Punching Shear Values | **Punching Shear Geometries** | Pile Reinforcement (Concrete)

	Label	Pile	Shape	Location	Effective Depth[in]	L1[in]	L2[in]
1	N49	Pile 1	Rect	INTERI...	24.125	34.125	34.125
2	N50	Pile 1	Rect	INTERI...	24.125	34.125	34.125
3	N51	Pile 1	Rect	INTERI...	24.125	34.125	34.125

Effective Depth

The effective depth, d, of the slab/pile cap is used to define the punching shear perimeter at a distance of d/2 beyond the edge of the pile.

$d = t_{slab}$ - pile embed - top cover - $d_{top\ bar}/2$

The effective depth is based on the smallest depth (to top reinforcing) for the Design Strips that encompass the pile. In cases where no design strips are defined at the pile 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. For more information, refer to the image in Using Equivalent Square Piles in the Pile Caps - Design topic.

Note: For further discussion, refer to ACI 318-14 Sections 22.6.4.1 and 8.4.2.3.1 (ACI 318-11 Sections 11.11.1.2 and 13.5.3).

Pile Reinforcement (Concrete)

Concrete Pile Rebar (By Item)

Pile Code Checks | Punching Shear Values | Punching Shear Geometries | **Pile Reinforcement (Concrete)**

	Label	Phi Used	Vertical Reinf	Shear Reinf
1	N49	0.65	CUSTOM	CUSTOM
2	N50	0.65	CUSTOM	CUSTOM
3	N51	0.65	CUSTOM	CUSTOM

Phi Used

This value is based on the strain in the tension steel in the pile and code provisions in ACI 318-14 Section 21.2 (ACI 318-11 Section 9.3.2).

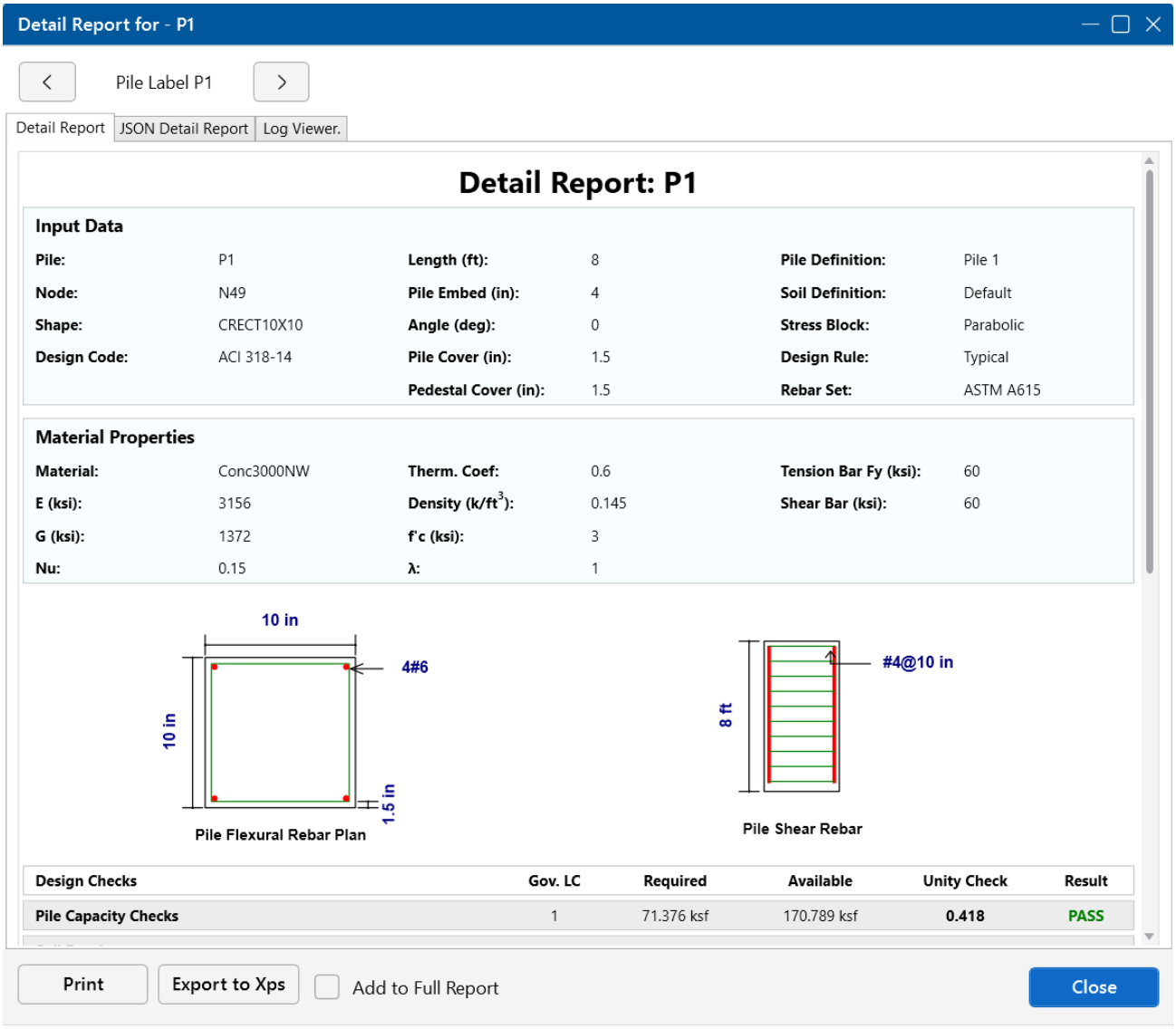
Pile Reinforcement

The **Vertical Reinf** and **Shear Reinf** values are reinforcement requirements for the pile considering spacing and minimum reinforcement requirements, per the ACI code and the Design Rules - Pedestal/ Pile tab. If these sections report "CUSTOM", it means that the user has applied a custom rebar layout.

Pile Detail Report

To see more detailed information for the pile results, select **Detailed Report** from the **Results** ribbon (when in a graphical view) and click on the node that defines the pile. The pile **Detail Report** shows the overall geometry, materials, codes and analysis/design inputs used for coming up with the reinforcing and code check values. The report is split into sections which include: input data/graphics, material properties, design checks and static soil capacity diagrams.

Input Data and Graphics



The **Input Data** section provides the point label of the pile and echos various input data used to define the pile element such as the pile geometry, embedment, concrete cover, pile definition label and selected rebar set.

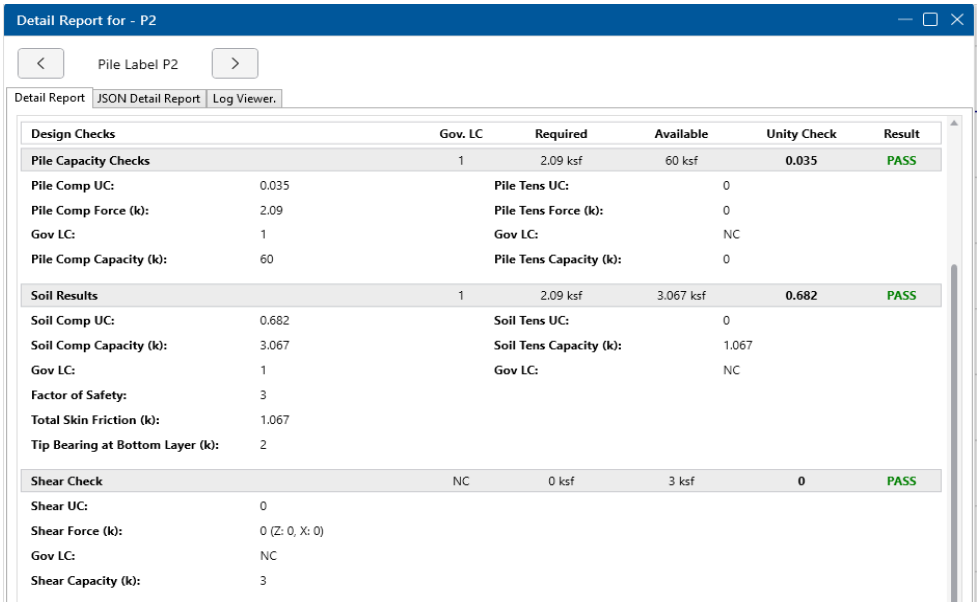
The **Material Properties** section contains the material properties used for the analysis and design of the pile based on the materials specified in the Material Properties Spreadsheet.

A graphical representation of the pile geometry and reinforcement design by the program is given below the material properties section in the form of a flexural rebar plan and shear rebar elevation.

Note:

- The program bases the number and spacing of bars on the requirements per the applicable code regarding spacing and reinforcement limits.
- If a custom rebar layout is used, the program will display the reinforcement as input by the user.

Design Results



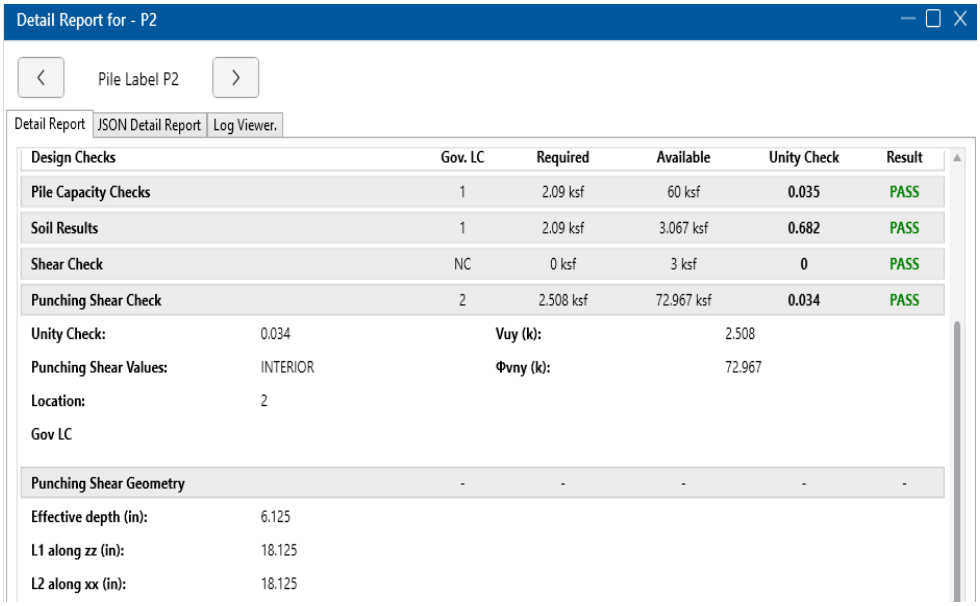
Design Checks		Gov. LC	Required	Available	Unity Check	Result
Pile Capacity Checks		1	2.09 ksf	60 ksf	**0.035**	PASS
Pile Comp UC:	0.035		Pile Tens UC:	0		
Pile Comp Force (k):	2.09		Pile Tens Force (k):	0		
Gov LC:	1		Gov LC:	NC		
Pile Comp Capacity (k):	60		Pile Tens Capacity (k):	0		
Soil Results		1	2.09 ksf	3.067 ksf	**0.682**	PASS
Soil Comp UC:	0.682		Soil Tens UC:	0		
Soil Comp Capacity (k):	3.067		Soil Tens Capacity (k):	1.067		
Gov LC:	1		Gov LC:	NC		
Factor of Safety:	3					
Total Skin Friction (k):	1.067					
Tip Bearing at Bottom Layer (k):	2					
Shear Check		NC	0 ksf	3 ksf	**0**	PASS
Shear UC:	0					
Shear Force (k):	0 (Z: 0, X: 0)					
Gov LC:	NC					
Shear Capacity (k):	3					

The **Pile Capacity Checks** summarize the code checks for the pile axial capacity based on the pile shape and material properties. Pile Comp and Pile Tens UC values give the ratio of demand forces to allowable forces.

The **The Soil Results** show the static pile capacities for compression and tension. This section also gives more detail about the total skin friction and tip bearing values used to calculate the capacities. The factor of safety input by the user in the Model Settings is also shown in this section of the report.

The **Shear Check** section shows the total shear demand and its force components in the Z and X directions. The shear capacity is a user-defined value in the Pile Definitions Spreadsheet and is compared against the demand for the unity check.

Punching Shear Check Results

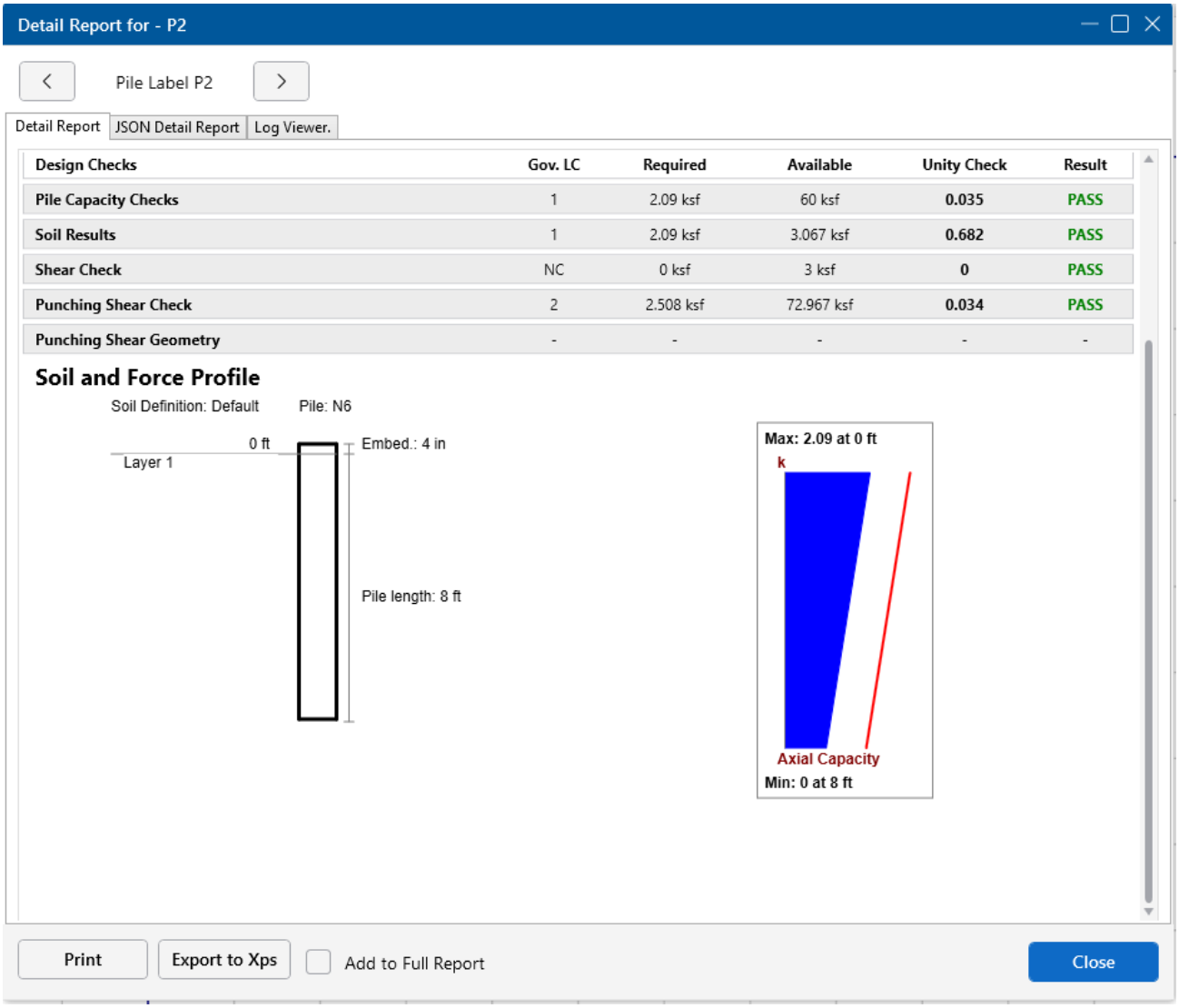


The **Punching Shear Check** section displays the unity check for the ratio of the punching shear demand to capacity. The governing LC is the load combination which produces the maximum unity check.

The **Punching Shear Geometry** section provides the dimensions used to calculate the punching shear capacity.

For an in depth explanation of the these values, see the Punching Shear Values and Punching Shear Geometries sections at the top of this topic.

Soil and Force Profile



The soil profile based on the applicable soil definition is displayed on the left side of the report. The axial capacity diagram on the right side of the report shows the force demand in blue versus the static pile capacity line in red. The capacity line represents the skin friction as a function over the length of the pile. At the bottom of the pile, the tip bearing capacity is reflected by an offset. The factor of safety is considered in both the skin friction and tip bearing for the capacity line.

Pile Caps - Design

In this section, we will talk specifically about the calculations for pile caps. These design results are NOT available when using piles and a slab to create your pile cap. The only portion of this section that

is applicable to individual piles is the Pile Punching Shear section. For slab/pile cap design, see Slabs.

Pile Cap Design Philosophy

Pile cap design within RISAFoundation considers beam philosophy, as opposed to truss philosophy. Both are acceptable by code (ACI 318-14 Section 13.4.2/ACI 318-11 Section 15.5.3).

Pile Capacity Checks

These checks allow the engineer to see if any of the piles in the model are exceeding the allowable pile forces.

Allowable Pile Loads

The allowable pile loads are defined by the user for general piles in the **Pile Definitions Spreadsheet**. For hot rolled steel, concrete and wood piles, the program calculates the axial capacity based on the pile and material properties. If the piles are part of a pile cap element, these allowable loads are reduced by the reduction factors entered on the **Pile** tab of the Pile Cap Definitions Spreadsheet.

Loads Developed in Piles in Pile Cap Applications

The loads developed in the piles within the pile cap elements are found using the equations below. Some things to note on the computation of these forces:

- Loads are only applied to the cap at the pedestal location, which is placed at the centroid of the cap.
- The axial pile stiffness (given in the Pile Definitions Spreadsheet) is considered the same for both compression and tension and is taken from the **Comp. Stiffness** entry.
- The pile cap is assumed to be rigid.
- The centroid location for force calculations of the piles is from the mid-height of the pile cap.

$$P_{pile} = \left(\frac{P}{N}\right) + \left[\frac{\left[M_x \cdot z_p + V_z \cdot \left(h_{ped} + \frac{d}{2}\right) \cdot z_p\right]}{\Sigma z_p^2}\right] + \left[\frac{\left[M_z \cdot x_p + V_x \cdot \left(h_{ped} + \frac{d}{2}\right) \cdot x_p\right]}{\Sigma x_p^2}\right]$$

$$Vpile_x := \frac{V_x}{N} + \frac{T \cdot \sqrt[2]{x_p^2 + z_p^2}}{\Sigma x_p^2 + \Sigma z_p^2} \cdot sin(\alpha) \qquad Vpile_z := \frac{V_z}{N} + \frac{T \cdot \sqrt[2]{x_p^2 + z_p^2}}{\Sigma x_p^2 + \Sigma z_p^2} \cdot cos(\alpha)$$

$$V_{pile} := \sqrt{Vpile_x^{\,2} + Vpile_z^{\,2}}$$

where:

P_{pile} = Total axial load on an individual pile

P = Column load, self-weight of the pile cap, pedestal and overburden

N = Total number of piles in pile cap

$M_{x,z}$ = Applied moment to pedestal

x_p, z_p = Distance from centroid of pile cap to centroid of individual pile parallel to the respective axis

$V_{x,z}$ = Applied shear to pedestal

h_{ped} = Height of pedestal from the top of the pile cap

d = Thickness of pile cap

T = Applied torque to pedestal

α = Angle between the centroid of the column and the centroid of the pile location, taken from the negative local x direction counter-clockwise

$Vpile_x$: Shear in pile in local x direction

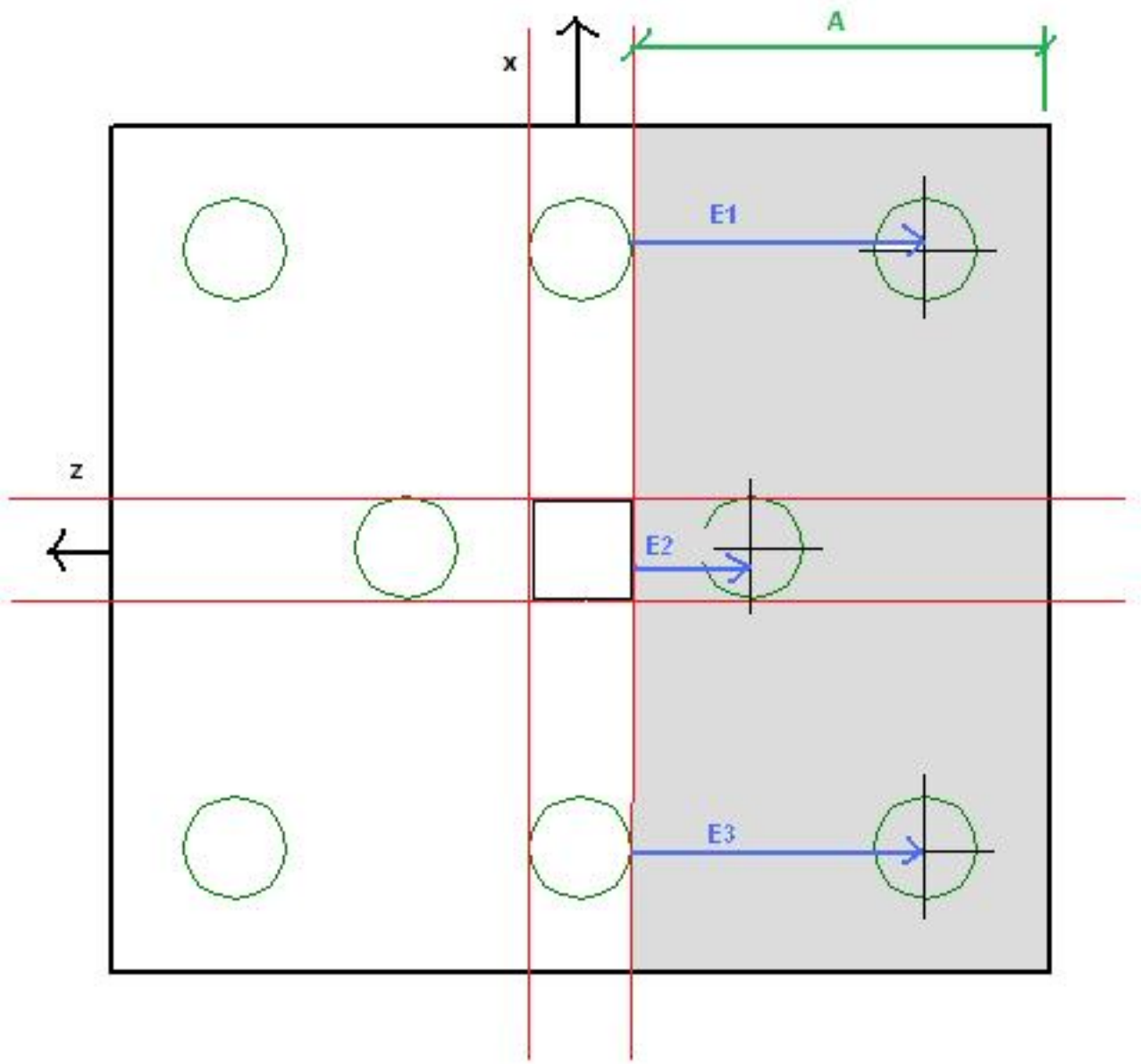
$Vpile_z$: Shear in pile in local z direction

V_{pile}: Total demand shear in pile (in any direction)

Note: The program does not consider the self-weight of the pile as part of the axial force in the pile. See the Forces section in Pile - Results for information on how to account for pile self-weight.

Pile Cap Flexural Design

The flexural design of pile caps is based on the moment at the face of the pedestals in each direction. The image below explains how the design moment, Mu, is evaluated.



The red lines in the image show the locations where we compute the moments in the pile cap. The calculation of the moment at the face of pedestal is as follows:

$$M_u := P_{u1} \cdot E_1 + P_{u2} \cdot E_2 + P_{u3} \cdot E_3 - \left(\frac{A}{2}\right) \cdot (A \cdot \text{Width}) \cdot (\rho_{slab} \cdot t_{slab} + \text{Over})$$

Thus, the program uses the factored pile forces multiplied by their horizontal distance to the face of pedestal and subtracts the self-weight of the pile cap and overburden in this area multiplied by its moment arm. In the example shown above, this would be the Muxx moment at the right face of the pedestal. This calculation would then be done for the left face of the pedestal and the enveloped maximum positive and negative moments are given.

This calculation is then carried out identically in the Muzz direction.

Flexural Reinforcement Design

The design of reinforcement runs through a design routine, which is described below, to come up with the area of steel required in both directions. This routine checks the area of steel based on flexure and then checks minimum requirements. The routine also considers the short side amplification factor, as well.

$A_{s\ req'd}$ = area of steel required for the design moment, using typical bending equations.

1. If $A_{stop\ req'd} > .0033bd$ use $A_{stop\ req'd}$. This is exactly the same for A_{sbot}.
2. If $A_{stop\ req'd} < .0033bd < 4/3A_s$, use 0.0033bd. This is exactly the same for A_{sbot}.
3. If $4/3A_{s\ top\ req'd} < .0033bd$, use$4/3A_s$. This is exactly the same for A_{sbot}.
4. Check for amplification of A_s in the short direction considering the β factor. See below for more information.
5. After these steps compare:
    1. ACI 318-14: $A_{s\ top}$ to 0.0018bh and $A_{s\ bot}$ to 0.0018bh. If both are greater than 0.0018bh then you are done. If less than 0.0018bh, steel is added to meet this requirement for each face.
    2. ACI 318-11: $A_{s\ top} + A_{s\ bot}$ to 0.0018bh. If greater than 0.0018bh then you are done. If less than 0.0018bh, we simply take the area of steel that needs to be added and add half to the top and half to the bottom.
6. The final check would be to compare the steel ratios to the min and max ratios from the Pile Cap Definitions - General tab.

Note:

- If the user-defined steel ratios do not meet code requirements we will use the code requirements and provide a note in the Detail Report.
- **β Factor consideration**
    - β = ratio of long dimension of pile cap to short dimension of pile cap.
    - $A_{s\beta} = A_{s\ req'd}*(2\beta\ )/(1+\beta)$
    - For pile caps with clipped corners we are taking the full dimension for the β calculation.
- If no flexural reinforcement is required in a given direction, all of the shrinkage and temperature steel in that direction will be added to the bottom of the slab.
- If top reinforcement is required in one direction and bottom reinforcement is required in both directions, then all provisions may be satisfied without giving top reinforcement in the direction where reinforcement is not required. We currently do not reinforce this opposite direction, though it is recommended to do so.

d Calculation

For bottom bar reinforcement design:

$d = t_{slab}$ - pile embed - bot cover - $d_{bot\ bar}$

For top bar reinforcement design:

$d = t_{slab}$ - top cover - $d_{top\ bar}$.

Note:

- Because we do not know exactly how the reinforcement will be placed, we take the interface of the two directions of reinforcement as the location for the d calculation.
- The **Embedment Depth** field is only used to calculate "d" with Pile Cap entities. If you are using slabs with pile supports, this embedment depth is not considered for the slab design - though it is still considered for the punching shear check. For slab entities $d = t_{slab}$ - bot cover - $d_{bot\ bar}$. Therefore, the bottom cover must include the embedment depth of piles for models with slabs supported on piles, but not for pile cap entities.

Pile Location Tolerance Consideration

If there is a pile location tolerance defined, we simply modify the Mu equation above by adding the tolerance to the moment arm for each individual pile. This will nearly always increase the moment demand.

$$M_u := P_{u1}\cdot(E_1 + Pile_{Tol}) + P_{u2}\cdot(E_2 + Pile_{Tol}) + P_{u3}\cdot(E_3 + Pile_{Tol}) - \left(\frac{A}{2}\right)\cdot(A\cdot Width)\cdot(\rho_{slab}\cdot t_{slab} + Over)$$

One Way (Deep Beam) Shear Check

This check considers the entire pile cap as a beam element and checks the overall shear across the pile either at a distance **d** from the face of the pedestal, or at the face of the pedestal. The location of this check depends on the distance, w, where w = the distance along the local axes from the face of pedestal to the centerline of the nearest pile in that direction.

If **d < w**, the critical location where shear is checked is taken at a distance d from the face of the pedestal. In this case, the equation for the shear stress capacity is found by:

$$v_c = 1.9\sqrt{f'_c} + 2500\rho_w\left(\frac{V_u d}{M_u}\right) \geq 2\sqrt{f'_c} \quad \text{(ACI Eq. 11-5)}$$

ACI 318-14 Table 22.5.5.1/ACI 318-11 Equation 11-5

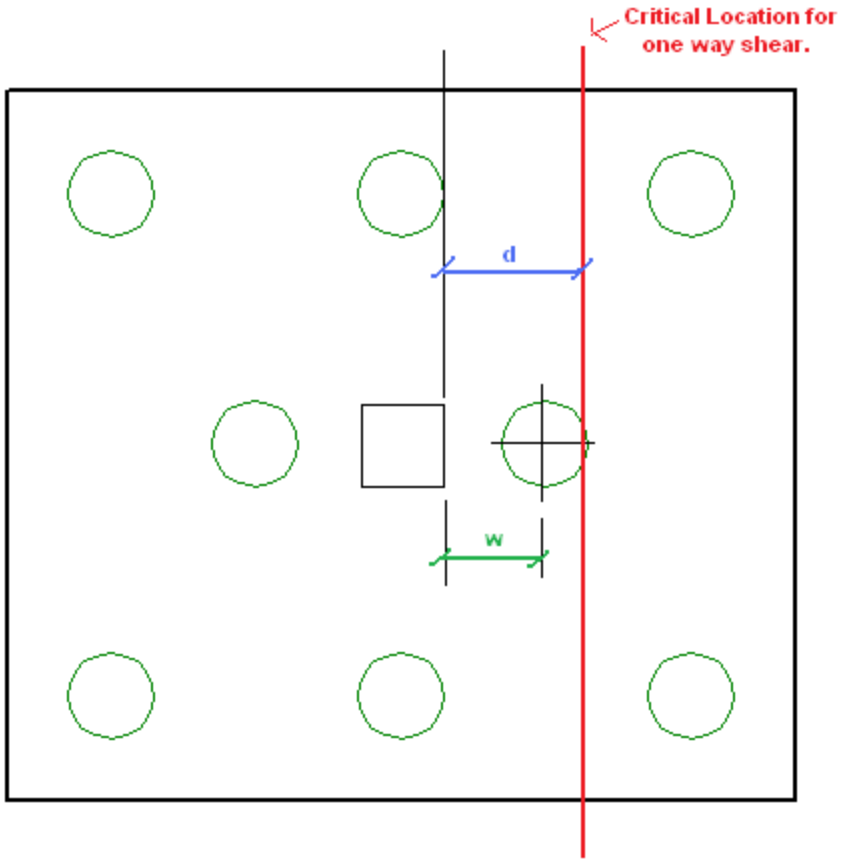
If **d > w**, the critical location where shear is checked is taken at the face of support.

$$v_c = \left(\frac{d}{w}\right)\left[3.5 - 2.5\left(\frac{M_u}{V_u d}\right)\right]\left[1.9\sqrt{f_c'} + 2500\rho_w\left(\frac{V_u d}{M_u}\right)\right] \leq 10\sqrt{f_c'}$$; $1.0 > M_u/V_u d > 0;\ \infty > V_u d/M_u \geq 1.0$

CRSI Design Handbook 2008 Eqn 13-2 on P.13-26

Note: The Vc equations were changed in ACI 318-19. When this is selected for the design code, Vc is calculated per Table 22.5.5.1. Since the pile cap has no shear reinforcement, Vc is calculated using Equation (c) in the table. Ignoring axial loads in pile cap, the vc term 1.9√fc+2500*ρw*(Vu*d/Mu) in the two equations above is changed into the equation shown below. Note that the size effect factor, λs, is a function of effective depth of the slab, d, and is calculated per Equation (22.5.5.1.3).

$$v_c = 8\lambda_s\lambda(\rho_w)^{1/3}\sqrt{f_c'}$$



The equation to calculate the shear capacity is:

$$\phi V_n := \phi \cdot v_c \cdot b \cdot d$$

$$\phi V_{nmax} := \phi \cdot 10 \cdot \sqrt{f_c} \cdot b \cdot d$$

(φVnmax only applies for eqn 13-2)

where:

M_u = enveloped moment at critical location.

V_u = enveloped shear at critical location

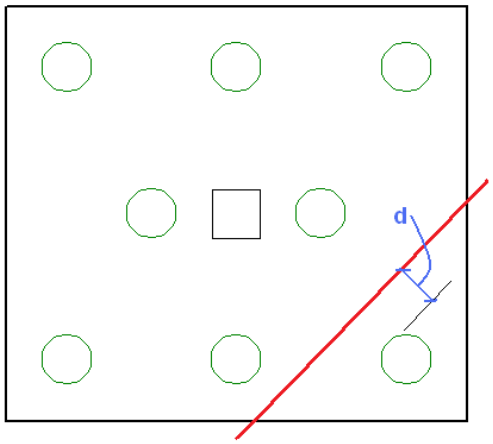
$\rho_w = A_{sprovided}/bd$

$d = t_{slab}$ - pile embed - cover - d_{bar}

We do these calculations for both sides of the pedestal in both directions and envelope the worst code check in each direction.

Considerations

- Mu and Vu are the enveloped moment and shear values. The maximum moment and shear occur at the same location for the majority of cases.
- We are NOT considering the one-way beam shear failures that can occur in other locations within the pile cap. An example of this type of failure would be a shear force that might shear off a corner of the pile cap.



- For lightweight concrete, the lambda reduction factor specified in the Materials Spreadsheet is applied to the capacity per ACI 318-14 Section 22.5.6.1 (ACI 318-11 Section 11.2.1.2).

Pile Location Tolerance Considerations

When a pile location tolerance is defined, that tolerance is simply added to the 'w' term in each direction. An increased value for 'w' will decrease the one way shear capacity of the pile cap.

Code Considerations

For the Saudi Code, ACI 318-05 is used. This is noted in the Detail Report. The Mexican and Canadian codes are simply using their standard beam and slab one-way bending equations.

Pile Punching Shear Checks

The punching shear in a pile is simply the axial compression force in a pile considering strength load combinations.

The punching shear capacity is obtained via the ACI code equations. The code requires this value to be the minimum from the equations below.

$$V_c = \left(2 + \frac{4}{\beta}\right)\sqrt{f_c'}\,b_o d \qquad (11\text{-}33)$$

$$V_c = \left(\frac{\alpha_s d}{b_o} + 2\right)\sqrt{f_c'}\,b_o d \qquad (11\text{-}34)$$

$$V_c = 4\sqrt{f_c'}\,b_o d \qquad (11\text{-}35)$$

Note:

- When ACI 318-19 is selected, an additional size effect factor, λs, is added in the above equations. λs is a function of the effective depth of the slab 'd', and is calculated per Equation (22.5.5.1.3).
- For lightweight concrete, the lambda reduction factor specified in the Materials Spreadsheet is applied to the capacity, per ACI 318-14 Section 22.5.6.1 (ACI 318-11 Section 11.2.1.2).

b_o Calculations

The calculation of b_o is done considering an interior, edge and corner condition. The program calculates a b_o value for each of these conditions for each pile and then uses the minimum value. The interior, edge or corner condition that resulted in the minimum value is also reported.

When there is no reinforcement at the top face of the pile cap (in either direction)

If there is no reinforcement in the top of the pile cap, the slab is considered unreinforced for pile punching shear. Because of this, we use the plain concrete provisions from ACI 318.

For ACI 318-19 and newer editions, RISAFoundation uses the structural plain concrete punching shear provisions directly (Table 14.5.5.1).

ACI 318-14 (ACI 318-11 Chapter 22) and earlier editions indirectly consider the plain concrete provisions. From ACI 318-14 Section 21.2.1 (ACI 318-11 Section 9.3.5), the ϕ factor for shear is 0.60 (as compared to 0.75 for reinforced concrete). The punching shear equation for structural plain concrete (ACI 318-14 Table 14.5.5.1/ACI 318-11 Equation 22-10) is the reinforced punching shear equations with a 2/3 factor applied. Thus, the ratio of using structural plain concrete to reinforced concrete is: 2/3*(0.55/0.75) = .488. RISAFoundation rounds this to 0.5 and takes a **50% reduction in capacity** for punching shear if there is no reinforcement in the top of the slab.

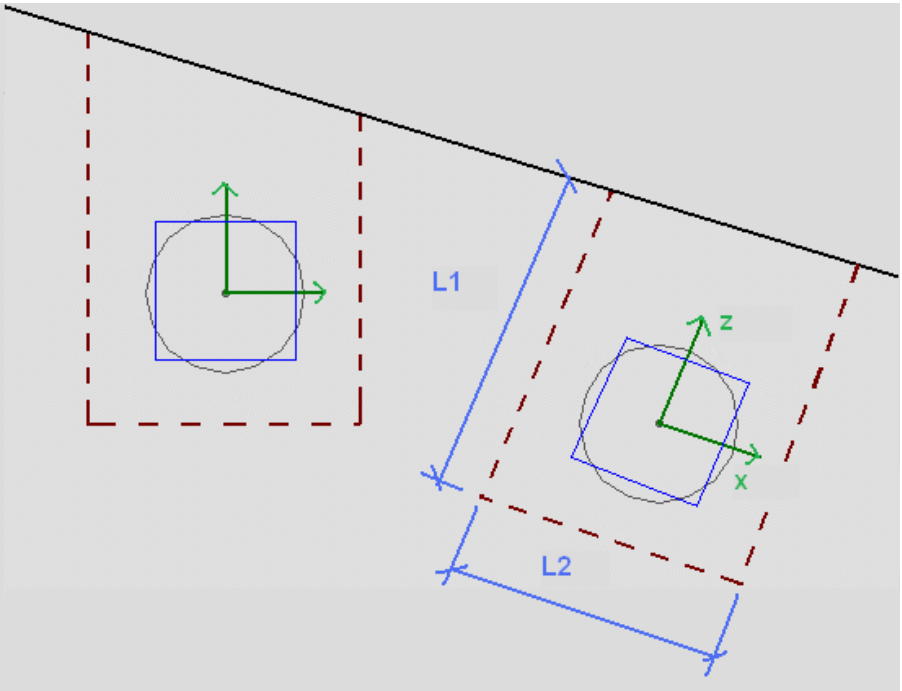
Note: If there is reinforcement in ONE direction we do not use this reduction.

When the Pile's Local Axes do not correspond to the slab geometry

The punching shear perimeters are always investigated with respect to the pile local axes. As shown below, there may be cases where these local axes do not correspond to the cap edge or corner geometry adjacent to that pile. In cases like these it is possible for this to result in an artificially large assumed punching shear perimeter.

Using Equivalent Square Piles

When calculating the punching shear perimeter for a round pile, the program always converts the pile into an equivalent square pile of equal perimeter (similar to the super-imposed square in the image below). This greatly simplifies the calculation of punching perimeters when close to an edge or corner.



d Calculation

$d = t_{slab}$ - pile embed - top cover - $d_{top\ bar}$

Group Punching for Piles

We do not currently calculate a group punching check. At solution we will check to see if closely spaced piles have shear perimeters that overlap. If so, we will give a warning in the Warning Log noting this. We will still give the capacities assuming the total shear perimeter for the pile not considering group action.

Beams/Pedestal Intersecting the Shear Perimeter of a Pile

- When a beam is co-linear with the pedestal joint, the program will not provide a punching shear check on the pedestal. In these cases, a code check of "NC" will be listed.
- The program does not specifically check for instances where the pedestal for the pile cap is located within the pile's shear perimeter. Thus, the punching shear capacity will be calculated as if there is no pedestal above.

Pedestal Punching Shear Checks

Pedestal punching shear checks are handled in much the same way that pile punching shear checks are considered. A major difference is that pedestals can transfer moment into the pile cap. Thus, the

pile cap punching shear capacity is reported as a stress and the maximum stress induced due to axial load and moment is checked. For more information, see Pile Cap Results.

To calculate the shear demand at the pedestal, simply sum the factored axial forces in the piles outside of the pedestal shear perimeter and add to that the factored self-weight of the pedestal and a ratio of the factored self-weight of the pile cap.

$$V_{punch} := \Sigma P_{uOutsideShearPerim} + SelfWeight_{Pedestal} + SelfWeight_{Slab} \cdot \left(\frac{No_{PilesOutsidePerim}}{TotalNo_{Piles}} \right)$$

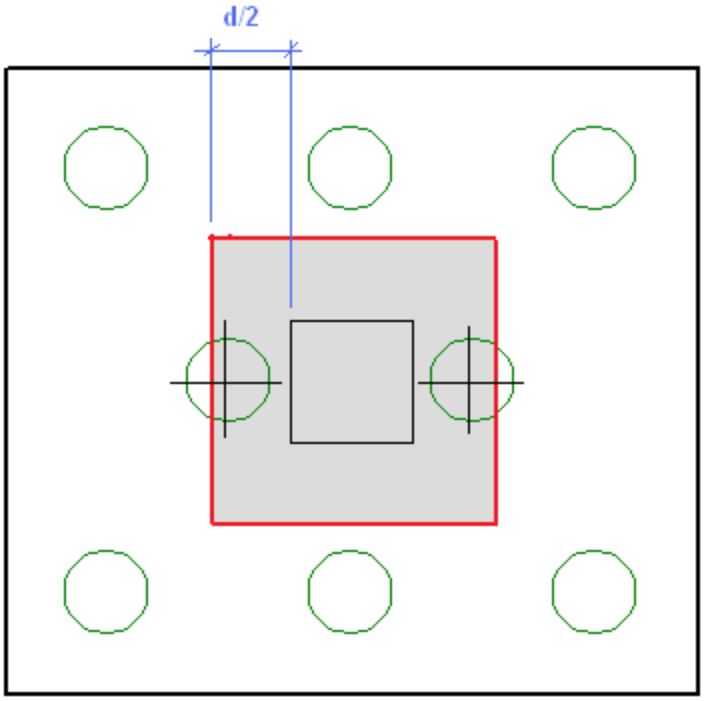
Note: For lightweight concrete, the lambda reduction factor specified in the Materials Spreadsheet is applied to the capacity, per ACI 318-14 Section 22.5.6.1 (ACI 318-11 Section 11.2.1.2).

d Calculation:

$d = t_{slab}$ - pile embed - bot cover - $d_{bot\ bar}$

When Piles intersect the Shear Perimeter of the Pedestal

At solution the program checks to see whether any of the piles in the pile cap have their **centerlines** located within the shear perimeter of the pedestal. If so, these piles do not contribute to the pedestal punching shear demand in the cap. If this centerline falls outside of the shear perimeter the entire force of the pile adds to the pedestal punching shear demand.



Note: ACI 318-19 Section 13.4.6.5 (ACI 318-14 Section 13.4.2.5, ACI 318-11 Section 15.5.4) discusses the cases where piles are very near the shear perimeter location. They recommend proportioning load according to how much of the pile is within the pedestal shear perimeter, however, RISAFoundation only considers the centerline of the pile.

Pedestal Design

Pedestal design for pile caps is identical to what is done for pedestals on slabs. For more information, see Pedestals-Design Results.

Pile Design

Pile design for pile caps is identical to what is done for individual piles. For more information, see Pile Results.

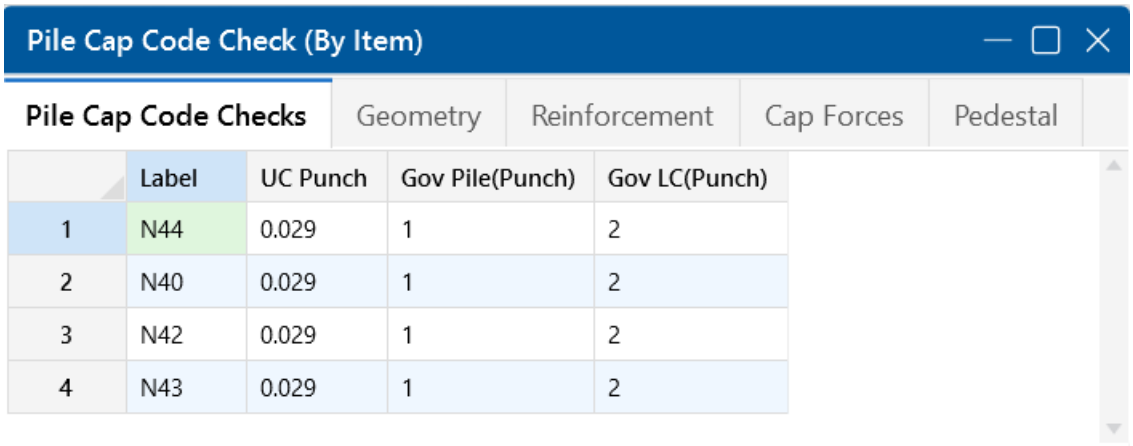
Pile Cap - Results

The locations for output for pile caps are the **Pile Cap Design Results Spreadsheet** and the Pile Cap Detail Report.

Pile Cap Design Spreadsheet

To view the **Pile Cap Design Spreadsheet**, click **Results Browser** from the **Results** toolbar and click **Pile Cap Design**. Or, click **Pile Cap Design** from the **Results** section of the **Explorer** panel. These output browsers are meant to summarize the results for the pile cap. For more information on the pile cap results, select **Detailed Report** from the **Results** ribbon (when in a graphical view) and click on the pile cap you want to review.

Pile Cap Code Checks

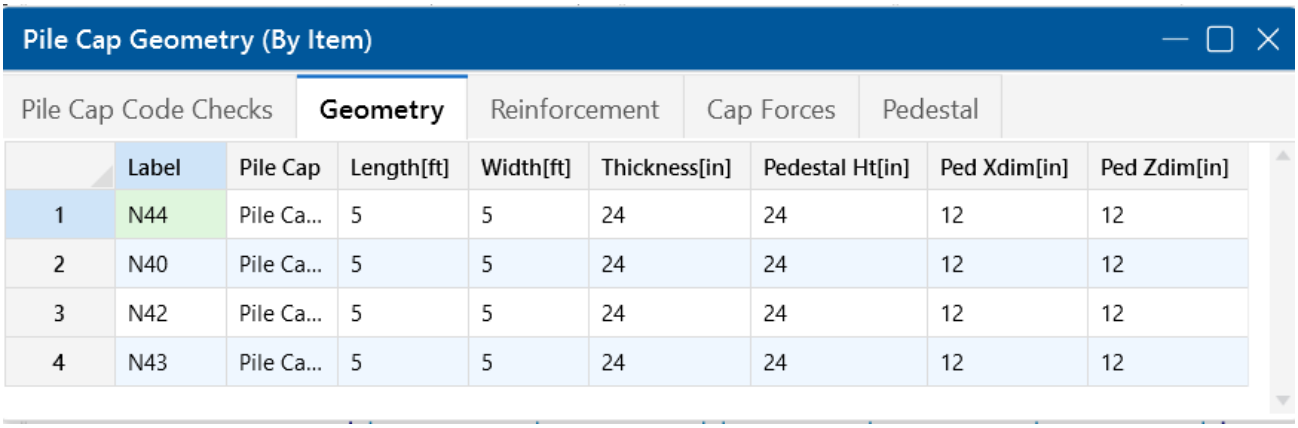


	Label	UC Punch	Gov Pile(Punch)	Gov LC(Punch)
1	N44	0.029	1	2
2	N40	0.029	1	2
3	N42	0.029	1	2
4	N43	0.029	1	2

The **Pile Cap Code Checks** tab gives the worst case pile for the worst case load combination for the punching shear code check.

The rest of the pile design checks including shear, compression, tension and punching shear are in the Pile Design Spreadsheet.

Pile Cap Geometry



	Label	Pile Cap	Length[ft]	Width[ft]	Thickness[in]	Pedestal Ht[in]	Ped Xdim[in]	Ped Zdim[in]
1	N44	Pile Ca...	5	5	24	24	12	12
2	N40	Pile Ca...	5	5	24	24	12	12
3	N42	Pile Ca...	5	5	24	24	12	12
4	N43	Pile Ca...	5	5	24	24	12	12

The **Geometry** tab of the spreadsheet provides dimensions for the pile cap and pedestal. To get a detailed sketch of this information, see the Detail Report.

Note: Values given in parentheses for the length and width are dimensions for non-rectangular sides of the cap. Review a 3 pile layout for an example of this.

Pile Cap Steel Reinforcement

Pile Cap Steel (By Item)

Pile Cap Code Checks | Geometry | **Reinforcement** | Cap Forces | Pedestal

	Label	Pile Cap	Bot z Dir. Steel[in^2]	Bot x Dir. Steel[in^2]	Top z Dir. Steel[in^2]	Top x Dir. Steel[in^2]	Ped Long	Ped Shear
1	N44	Pile Ca...	2.651	2.651	0	0	4#6	#4@12 in
2	N40	Pile Ca...	2.651	2.651	0	0	4#6	#4@12 in
3	N42	Pile Ca...	2.651	2.651	0	0	4#6	#4@12 in
4	N43	Pile Ca...	2.651	2.651	0	0	4#6	#4@12 in

The **Reinforcement** tab of the spreadsheet gives the required steel for both the pile cap and pedestal. For more information on the design of the pile cap reinforcement, see Pile Cap - Design and the Pile Cap Flexural Design portion of the Detail Report.

Pile Cap Design Forces

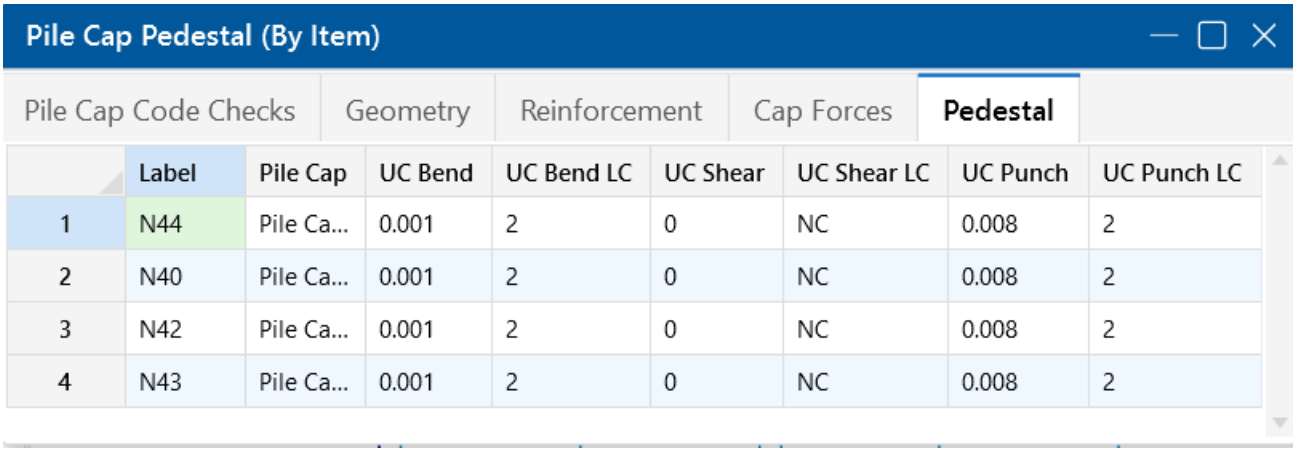
Pile Cap Maximum Forces (By Item)

Pile Cap Code Checks | Geometry | Reinforcement | **Cap Forces** | Pedestal

	Label	Pile Cap	Muxx[k-ft]	Muxx LC	Muzz[k-ft]	Muzz LC	Vu-p[k]	Vu-p LC	Vu-x[k]	Vu-x LC	Vu-z[k]	Vu-z LC
1	N44	Pile Ca...	1.044	2	1.044	2	2.277	2	1.044	2	1.044	2
2	N40	Pile Ca...	1.044	2	1.044	2	2.277	2	1.044	2	1.044	2
3	N42	Pile Ca...	1.044	2	1.044	2	2.277	2	1.044	2	1.044	2
4	N43	Pile Ca...	1.044	2	1.044	2	2.277	2	1.044	2	1.044	2

The **Cap Forces** tab of the spreadsheet provides a summary of the forces in the pile cap for the worst case load combinations. The **Muxx** and **Muzz** moments are the worst case moments from the positive and negative faces of the pedestal in both directions. They are based on the local axis of the pile cap, which can be displayed using the options in the Model View Settings. For more information on these design values, see Pile Cap - Design and the Pile Cap Flexural Design portion of the Detail Report. **Vu-p**is the pedestal punching shear demand. For more information, see Pile Cap - Design and the Pedestal Punching Shear portion of the Detail Report. The **Vu-x** and **Vu-z** shears are the worst case shears from the positive and negative faces of the pedestal in both directions. For more information, see Pile Cap - Design and the One Way Shear portion of the Detail Report.

Pedestal Design Checks



	Label	Pile Cap	UC Bend	UC Bend LC	UC Shear	UC Shear LC	UC Punch	UC Punch LC
1	N44	Pile Ca...	0.001	2	0	NC	0.008	2
2	N40	Pile Ca...	0.001	2	0	NC	0.008	2
3	N42	Pile Ca...	0.001	2	0	NC	0.008	2
4	N43	Pile Ca...	0.001	2	0	NC	0.008	2

The **Pedestal** tab provides a summary of the design checks for the pedestal for the worst case load combination. For more information, see Pedestal - Design and Punching and the Pedestal Design portion of the Detail Report.

Pile Cap Detail Report

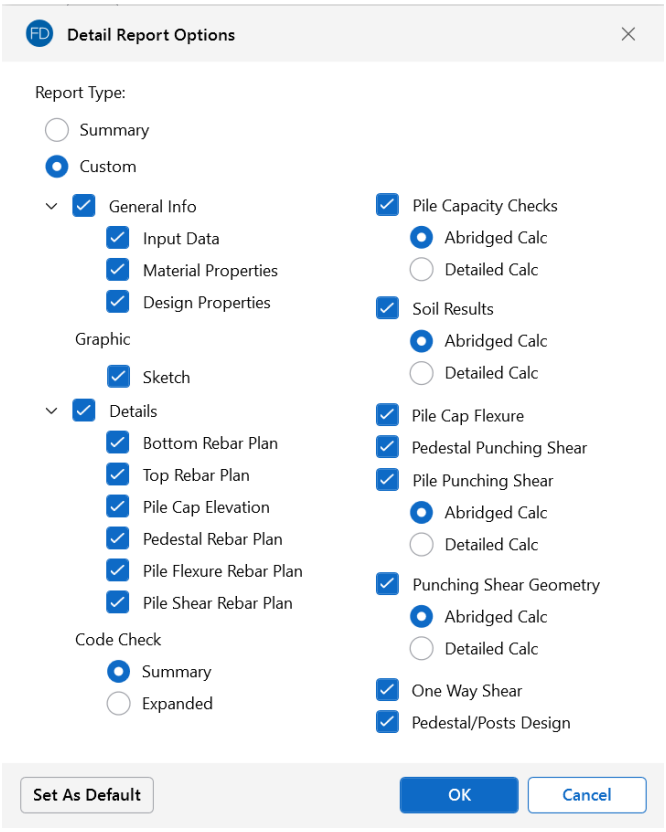
The **Pile Cap Detail Report** provides detailed information about geometry, materials, criteria, design checks and sketches associated with this pile cap. Below, we will walk through each section of the report.

Header

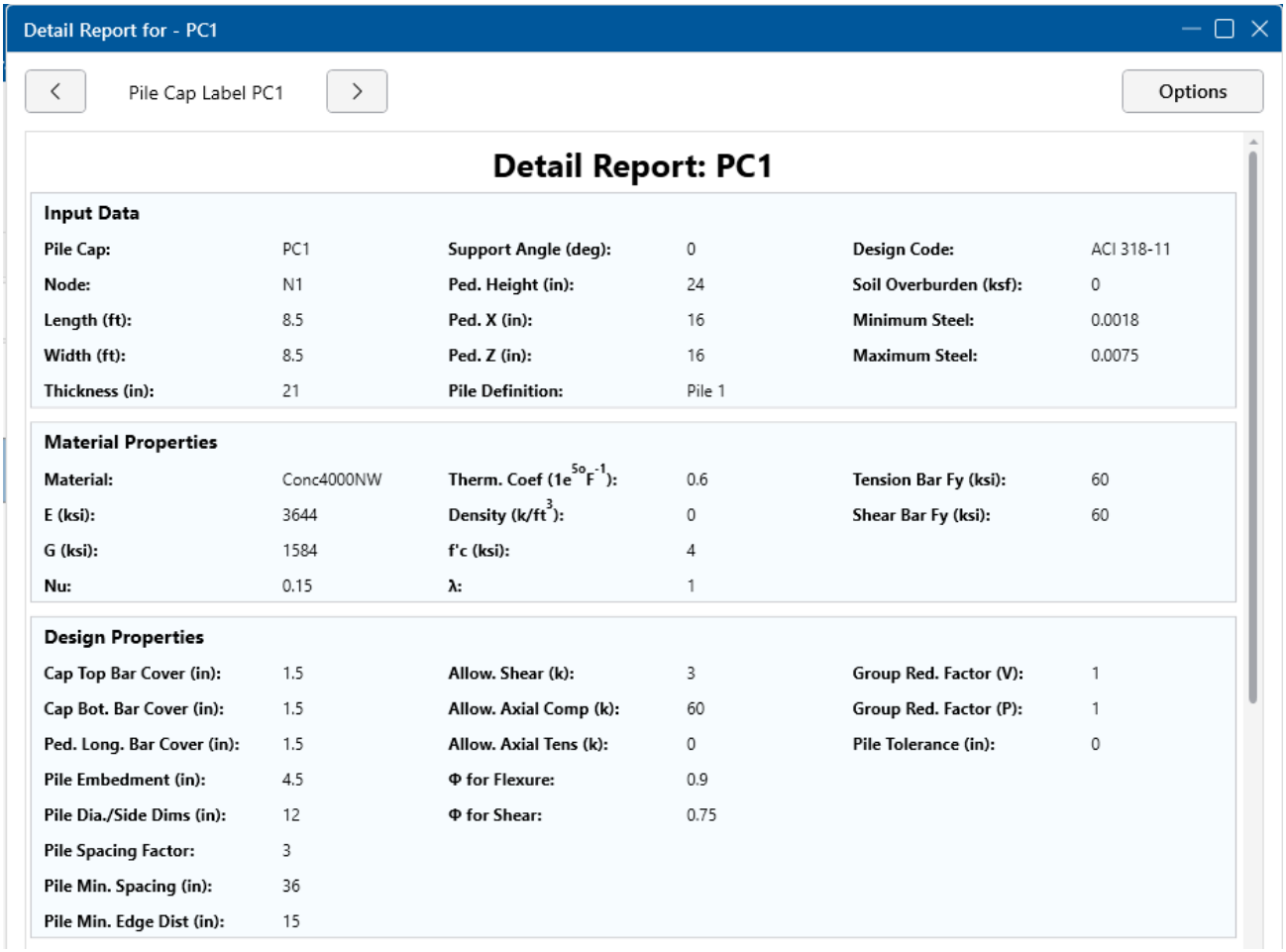


The header of the Detail Report is useful for navigating and viewing information in the pile cap reports.

- The **arrows** allow you to click through detail reports for the different pile caps in the model.
- The **Options** button opens up the **Detail Report Options** dialog. This allows you to customize exactly what you see in the Detail Report. Some Code Checks have an option to show either the Abridged Calc or the Detailed Calc. We will talk more about this in the following sections.

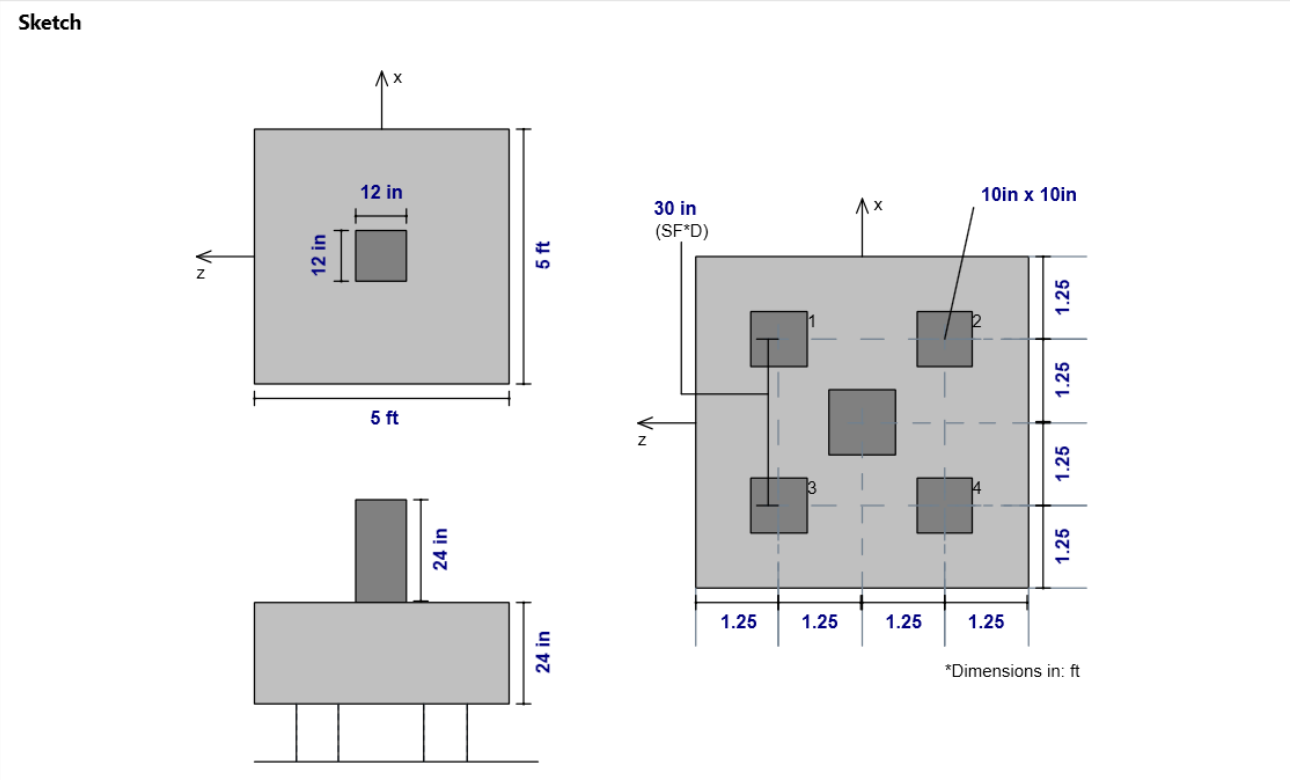

Detail Report Options
Report Type:
Summary
Custom
General Info
Input Data
Material Properties
Design Properties
Graphic
Sketch
Details
Bottom Rebar Plan
Top Rebar Plan
Pile Cap Elevation
Pedestal Rebar Plan
Pile Flexure Rebar Plan
Pile Shear Rebar Plan
Code Check
Summary
Expanded
Pile Capacity Checks
Abridged Calc
Detailed Calc
Soil Results
Abridged Calc
Detailed Calc
Pile Cap Flexure
Pedestal Punching Shear
Pile Punching Shear
Abridged Calc
Detailed Calc
Punching Shear Geometry
Abridged Calc
Detailed Calc
One Way Shear
Pedestal/Posts Design
Set As Default
OK
Cancel

Input Data, Material Properties and Design Properties



The **Input Data** lists off the point label of the pile, material properties, reinforcement properties and design requirements, as applicable from the Design Rules, Pile Definitions, Pile Cap Definitions and Pile Caps Spreadsheets.

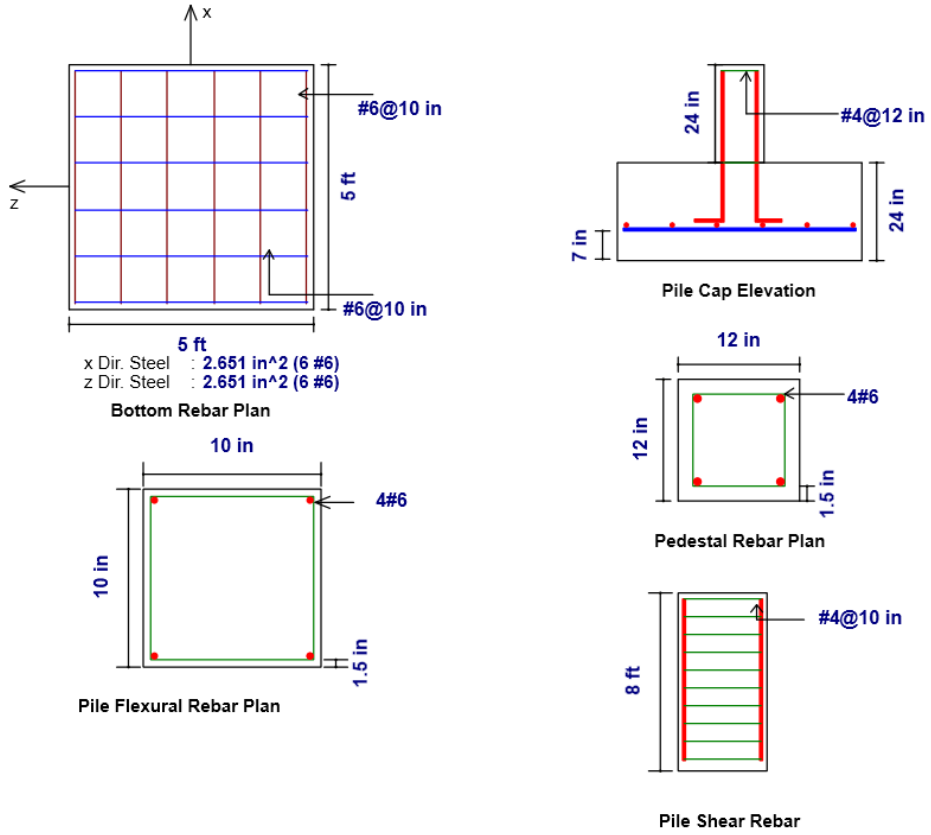
Sketch



The **Sketch** section provides a full sketch of pedestal and pile locations. The dimensions are given using the CRSI pile layouts, edge distance, pile spacing factor and the pile diameter/side dimension. The minimum distance between piles is defined by the Spacing Factor x Pile Diameter/Side Dimension and is shown on the sketch as well (SF*D).

Reinforcement Details

Details



The **Reinforcement Details** section lists bar size, spacing and total reinforcement in each direction of the pile cap. If welded wire reinforcement is designed, the output result will be preceded by a “D” instead of “#” to indicate the wire bar size. This gives you a to scale visual of the reinforcement in your pile cap, pedestal, and concrete piles. For more information on the required steel calculation, see Pile Cap - Design and the Pile Cap Flexural Design portion of the Detail Report.

Pile Capacity Checks

Pile Capacity Checks				1	0.111 k	3 k	0.037	PASS
Pile Allow Tens (k):	0		Pile Allow Comp (k):	175.904		Pile Allow Shear (k):	3	

LC	Pile	Shear Force (k)	Shear UC	Axial Comp Force (k)	Axial Comp UC	Axial Tens Force (k)	Axial Tens UC
1	1	0.111	0.037	5.396	0.031	0	0
	2	0.111	0.037	5.556	0.032	0	0
	3	0.111	0.037	5.715	0.032	0	0
	4	0.111	0.037	5.396	0.031	0	0
	5	0.111	0.037	5.556	0.032	0	0
	6	0.111	0.037	5.715	0.032	0	0
	7	0.111	0.037	5.396	0.031	0	0
	8	0.111	0.037	5.556	0.032	0	0
	9	0.111	0.037	5.715	0.032	0	0

Note: Pile capacity checks are only given for service LCs

Pile Capacity Checks				1	-	-	0.037	PASS
Pile Allow Tens (k):	0		Pile Allow Comp (k):	175.904		Pile Allow Shear (k):	3	

Shear UC Max	Gov Pile for Shear	Gov Shear LC	Axial Comp UC Max	Gov Pile for Axial Comp	Gov Axial Comp LC	Axial Tens UC Max	Gov Pile for Axial Tens	Gov Axial Tens LC
0.037	1	1	0.032	3	1	0	NC	NC

Note: See Expanded View for information for each individual pile for each load combination
Pile capacity checks are only given for Service LC's

The **Pile Capacity Checks** section can be shown in two ways by selecting between the **Abridged Calc** and **Detailed Calc** options in the **Detail Report Options** dialog. The detailed view (top) gives forces and code checks in each pile in the pile layout for each service load combination run. The abridged view (bottom) simply gives the maximum code check for the worst case pile for the worst case LC. To learn more about how these forces are calculated and how the allowable forces are modified, see Pile Caps - Design.

Soil Results

Soil Results				1	5.675 k	5.926 k	0.958	PASS
LC	Pile	Soil Comp Force (k)	Soil Comp UC	Soil Tens Force (k)	Soil Tens UC	Total Skin Friction (k)	Tip Bearing (k)	
1	1	5.436	0.917	0	0	0	5.926	
	2	5.556	0.937	0	0	0	5.926	
	3	5.675	0.958	0	0	0	5.926	
	4	5.436	0.917	0	0	0	5.926	
	5	5.556	0.937	0	0	0	5.926	
	6	5.675	0.958	0	0	0	5.926	
	7	5.436	0.917	0	0	0	5.926	
	8	5.556	0.937	0	0	0	5.926	
	9	5.675	0.958	0	0	0	5.926	

Note: Pile capacity checks are only given for service LCs

Soil Results				1	-	-	0.958	PASS
Soil Comp UC Max	Gov Pile for Soil Comp	Gov Soil Comp LC	Soil Tens UC Max	Gov Pile for Soil Tens	Gov Soil Tens LC	Total Skin Friction (k)	Tip Bearing (k)	
0.958	3	1	0	NC	NC	0	5.926	

Note: See Expanded View for information for each individual pile for each load combination
Pile capacity checks are only given for Service LC's

The **Soil Results** section can be shown in two ways by selecting between the **Abridged Calc** and **Detailed Calc** options in the **Detail Report Options** dialog. The detailed view (top) gives forces and code checks in each pile in the pile layout for each service load combination run. The abridged view (bottom) simply gives the maximum code check for the worst case pile for the worst case LC.

Pile Cap Flexural Design

Pile Cap Flexural Design				2	1.044 k-ft	187.634 k-ft	0.006	PASS
As-min x-dir (Top Flexure) (in^2):	2.592	As-min x-dir (Bot Flexure) (in^2):	2.592			As-min x-dir (T & S) (in^2):	2.592	
As-min z-dir (Top Flexure) (in^2):	2.592	As-min z-dir (Bot Flexure) (in^2):	2.592			As-min z-dir (T & S) (in^2):	2.592	

Bottom Bar Reinforcement

Mu-xx UC Max	Gov Mu-xx LC	Mu-xx (k-ft)	z-Dir As Required (in^2)	z-Dir As Provided (in^2)	Mu-zz UC Max	Gov Mu-zz LC	Mu-zz (k-ft)	x-Dir As Required (in^2)	x-Dir As Provided (in^2)
0.006	2	1.044	0.014	2.651	0.006	2	1.044	0.014	2.651

Top Bar Reinforcement

Mu-xx UC Max	Gov Mu-xx LC	Mu-xx (k-ft)	z-Dir As Required (in^2)	z-Dir As Provided (in^2)	Mu-zz UC Max	Gov Mu-zz LC	Mu-zz (k-ft)	x-Dir As Required (in^2)	x-Dir As Provided (in^2)
0	NC	0	0	0	0	NC	0	0	0

The **Pile Cap Flexural Design** section starts by giving minimum steel requirements for both flexure and shrinkage and temperature. Then there is a bottom and top bar reinforcement section where the max moments at the face of the pedestal is given in both orthogonal directions. The **Muxx** moment (moment causing bending about the x-axis) is the maximum moment computed considering all strength load combinations at both the positive and negative z faces of the pedestal. Top reinforcement is only added if there is negative bending in the pile cap. For more information on reinforcement calculations, see Pile Caps - Design.

Pedestal Punching Shear Check

Pedestal Punching Shear Check				2	0.001 ksi	0.164 ksi	0.008	PASS
Φ:	0.75							
Gov LC	UC	Punching Shear Location	Punching Shear (k)	Muz (k-ft)	Mux (k-ft)	Total Stress (ksi)	Φ*Vny (ksi)	
2	0.008	INTERIOR	2.277	0	0	0.001	0.164	

The **Pedestal Punching Shear Check** section provides the demand punching shear and moments at the base of the pedestal. These forces are then converted into an equivalent total stress and this stress is reported.

Phi*Vny 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. This value does NOT include any additional capacity due to shear reinforcing in the slab. For information on the calculation of the punching shear demand, see Pile Caps - Design. For information regarding geometry, see the Punching Shear Geometry section of the Detail Report.

Pile Punching Shear Check

Pile Punching Shear Check				2	7.255 k	159.379 k	0.046	PASS
Φ:	0.75							
LC	Pile	Punching Shear UC	Punching Shear Location	Punching Shear Cap (k)	Punching Shear (k)			
2	1	0.044	CORNER	159.379	6.967			
	2	0.03	EDGE	239.068	7.111			
	3	0.046	CORNER	159.379	7.255			
	4	0.029	EDGE	239.068	6.967			
	5	0.022	INTERIOR	318.758	7.111			
	6	0.03	EDGE	239.068	7.255			
	7	0.044	CORNER	159.379	6.967			
	8	0.03	EDGE	239.068	7.111			
	9	0.046	CORNER	159.379	7.255			
3	1	0	CORNER	159.379	0			
	2	0	EDGE	239.068	0			
	3	0	CORNER	159.379	0			
	4	0	EDGE	239.068	0			
	5	0	INTERIOR	318.758	0			
	6	0	EDGE	239.068	0			
	7	0	CORNER	159.379	0			
	8	0	EDGE	239.068	0			
	9	0	CORNER	159.379	0			

Note: Punching shear checks are only given for strength LCs

Pile Punching Shear Check			2	7.255 k	159.379 k	0.046	PASS
Φ:	0.75						
Gov LC	Gov Pile	Punching Shear Location	Punching Shear Cap (k)	Punching Shear (k)	Punching Shear UC		
2	3	CORNER	159.379	7.255	0.046		

Note: See Expanded View for information for each individual pile for each load combination
Punching shear checks are only given for Strength LC's

The **Pile Punching Shear Check**section can be shown in two ways by selecting between the **Abridged Calc** and **Detailed Calc** options in the **Detail Report Options** dialog. The detailed view (top) gives forces and code checks in each pile in the pile layout for each strength load combination run. The abridged view simply gives the maximum code check for the worst case pile for the worst case LC. To learn more about how these forces are calculated and how the allowable forces are modified, see Pile Caps - Design.

Punching Shear Geometry

Punching Shear Geometry

Pedestal Shape: CRECT16X16 **Pile Shape:** CRECT16X16

Pile/ Pedestal	Location	Effective Depth (in)	L1 (in)	L2 (in)	Polar Moment-xx (in^4)	Polar Moment-zz (in^4)	γ-x	γ-z
Pedestal	INTERIOR	14	30	30	2.657e+5	2.657e+5	0.4	0.4
1	CORNER	14	30	30	0	0	0	0
2	EDGE	14	30	30	0	0	0	0
3	CORNER	14	30	30	0	0	0	0
4	EDGE	14	30	30	0	0	0	0
5	INTERIOR	14	30	30	0	0	0	0
6	EDGE	14	30	30	0	0	0	0
7	CORNER	14	30	30	0	0	0	0
8	EDGE	14	30	30	0	0	0	0
9	CORNER	14	30	30	0	0	0	0

Punching Shear Geometry

Pedestal Shape: CRECT16X16 **Pile Shape:** CRECT16X16

Gov Pile/ Pedestal	Location	Effective Depth (in)	L1 (in)	L2 (in)	Polar Moment-xx (in^4)	Polar Moment-zz (in^4)	γ-x	γ-z
Pedestal	INTERIOR	14	30	30	2.657e+5	2.657e+5	0.4	0.4
3	CORNER	14	30	30	0	0	0	0

Note: See Expanded View for information for each individual pile for each load combination

The **Punching Shear Geometry**section can be shown in two ways by selecting between the **Abridged Calc** and **Detailed Calc** options in the **Detail Report Options** dialog. The detailed view (top) gives geometry information for every pile. The abridged view (bottom) simply gives the geometry information for the worst case pile.

Location

The pile 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.

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 318-18 sections 11.12.1.2 and 13.5.3.

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 local xx and zz axes. This value is used to determine the shear stress in the critical section which is due to unbalanced moment.

Note: For further discussion, refer to ACI 318-14 Sections 22.6.4.1 and 8.4.2.3.1 (ACI 318-11 Sections 11.11.1.2 and 13.5.3).

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.

Note: For further discussion, refer to ACI 318-14 Sections 8.4.4.2 and 8.4.2.3.1 (ACI Sections 11.11.7.1 and 13.5.3).

See Pile Caps - Design for more information about geometry considerations in the Code Checks.

One Way Shear Check

One Way (Deep Beam) Shear Check				2	1.044 k	185.504 k	0.006	PASS
w (in):	12	w (in):	12			Φ:	0.75	
d (in):	16.25	d (in):	16.25					
Vu (k):	1.044	Vu (k):	1.044					
Mu-xx (k-ft):	1.044	Mu-zz (k-ft):	1.044					
Vc(x-Dir Cut) (k):	247.338	Vc(z-Dir Cut) (k):	247.338					

UC	x-Dir Cut Vu (k)	Gov LC	UC	z-Dir Cut Vu (k)	Gov LC
0.006	1.044	2	0.006	1.044	2

Note: One way (beam) shear failures associated with the pile locations are not considered.

The **One Way Shear Check** section provides a results summary for beam shear across the pile cap. The **x-Dir Cut Vu** displays the worst case shear force at the positive and negative x sides of the pedestal for all strength LCs. The **Vc (x-Dir Cut)** and **Vc (z-Dir Cut)** gives the shear capacity about both local axis directions. The **w**, **d**, **Vu** and **Mu-xx** and **Mu-zz** values are given to verify the calculation of shear capacity. For more information on these calculations, see Pile Cap - Design.

Pedestal Design Checks

Pedestal Design Shear	NC	-	-	**0**	**PASS**

Shear Check Results (Envelope):

	Vc (k)	Vs (k)	Vu (k)	Vu/Φ*Vn	Φ	Gov LC
Shear Along x Direction:	12.652	18.899	0	0	0.75	NC
Shear Along z Direction:	12.652	18.899	0	0	0.75	NC

Pedestal Ties: #4@12 in

Pedestal Design Bending	2	-	-	**0.001**	**PASS**

Bending Check Results (Envelope): PCA Load Contour Method (for biaxial)

Unity Check:	0.001	**Φ:**	0.65	**Parme β:**	0.65
Pu (k):	0.348	**Mux (k-ft):**	0	**Muz (k-ft):**	0
Pn (k):	374.978	**Mnx (k-ft):**	NC	**Mnz (k-ft):**	NC
Governing LC:	2	**Mnox (k-ft):**	NC	**Mnoz (k-ft):**	NC
Pedestal Bars:	4 #6	**% Steel:**	1.227		

The **Pedestal Design Shear** and **Pedestal Design Bending** sections provide design check results for both shear and combined bending and axial forces. For more information on these checks, see Pedestals-Design Results.

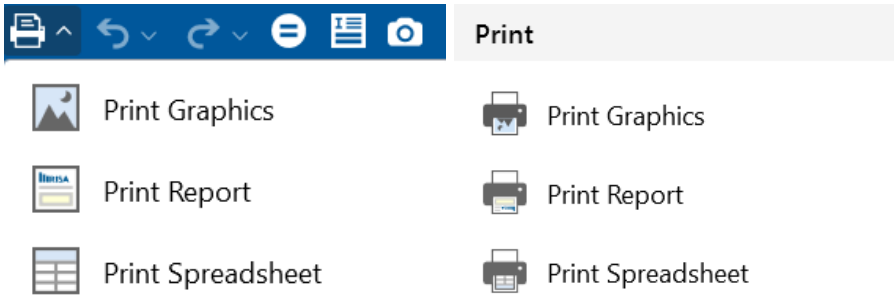
Printing

You can print graphic or tabulated information from your current model.

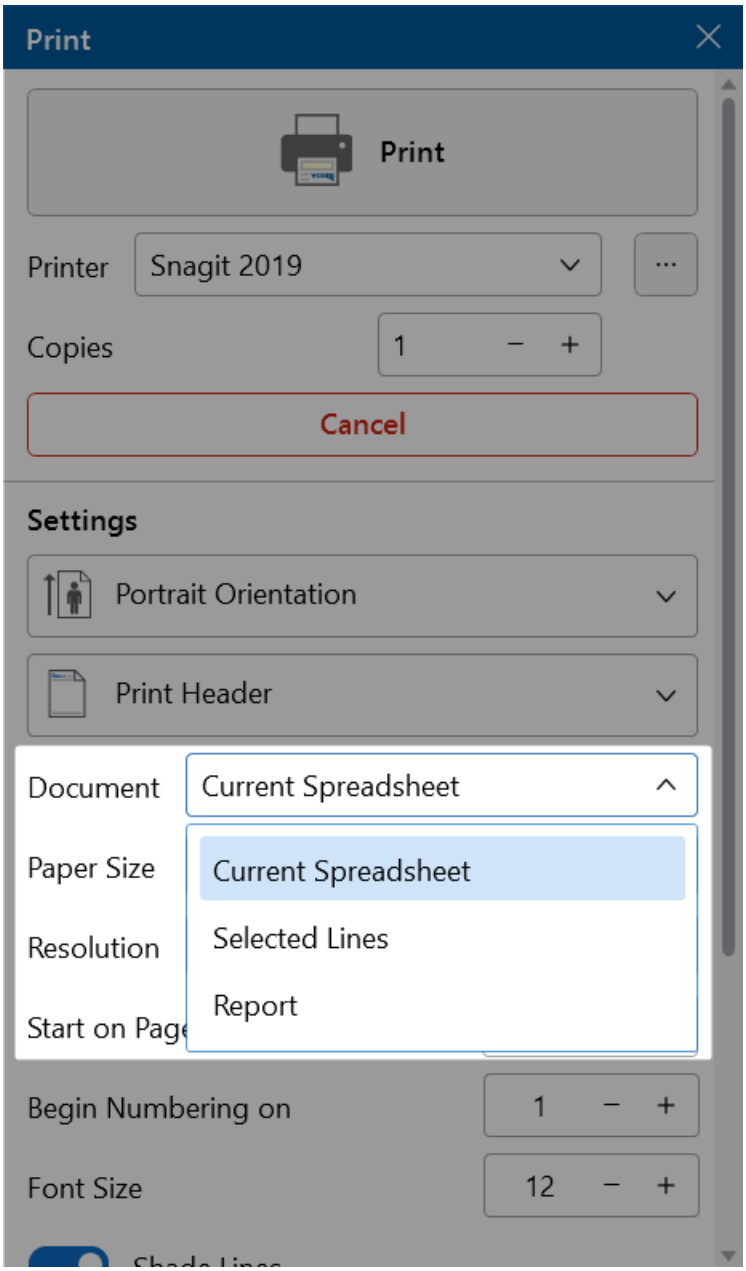
Print a Report or Spreadsheet

If a spreadsheet is currently active, you can click **Print** in the **Quick Access** toolbar or from the **File** menu. You are given the option to **Print Report** or **Print Spreadsheet**.

Print - Quick Access Toolbar **Print - File menu**



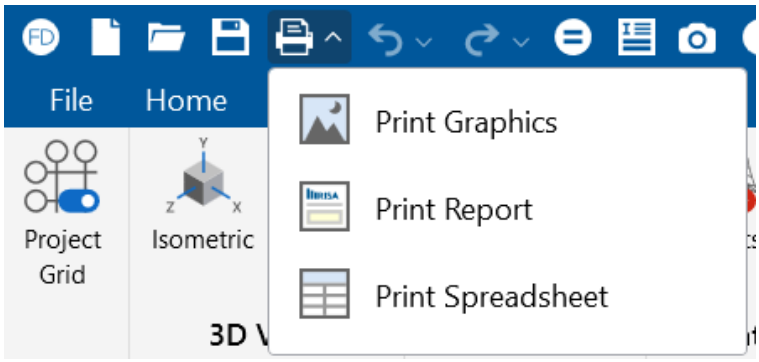
In the Print Settings for spreadsheets, you have the option to print the **Current Spreadsheet**, the **Selected Line**(s) of the spreadsheet or switch over to **Report Printing**.



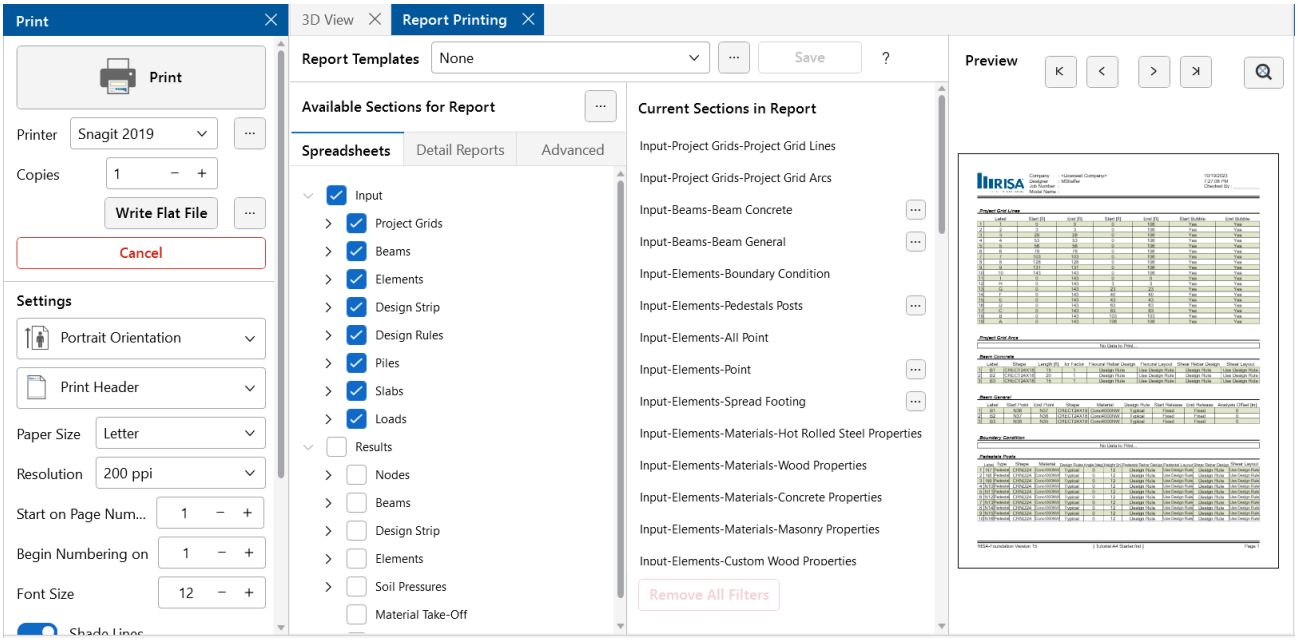
If a 3D View is currently active, you will be given the option to **Print Graphics** or **Print Report**.

Print Graphics

While in a 3D View, select **Print** from the **Quick Access** toolbar, or the **File** menu, and you will be given the option to **Print Graphics**, **Print Report** or **Print Spreadsheet**.



Printing Reports

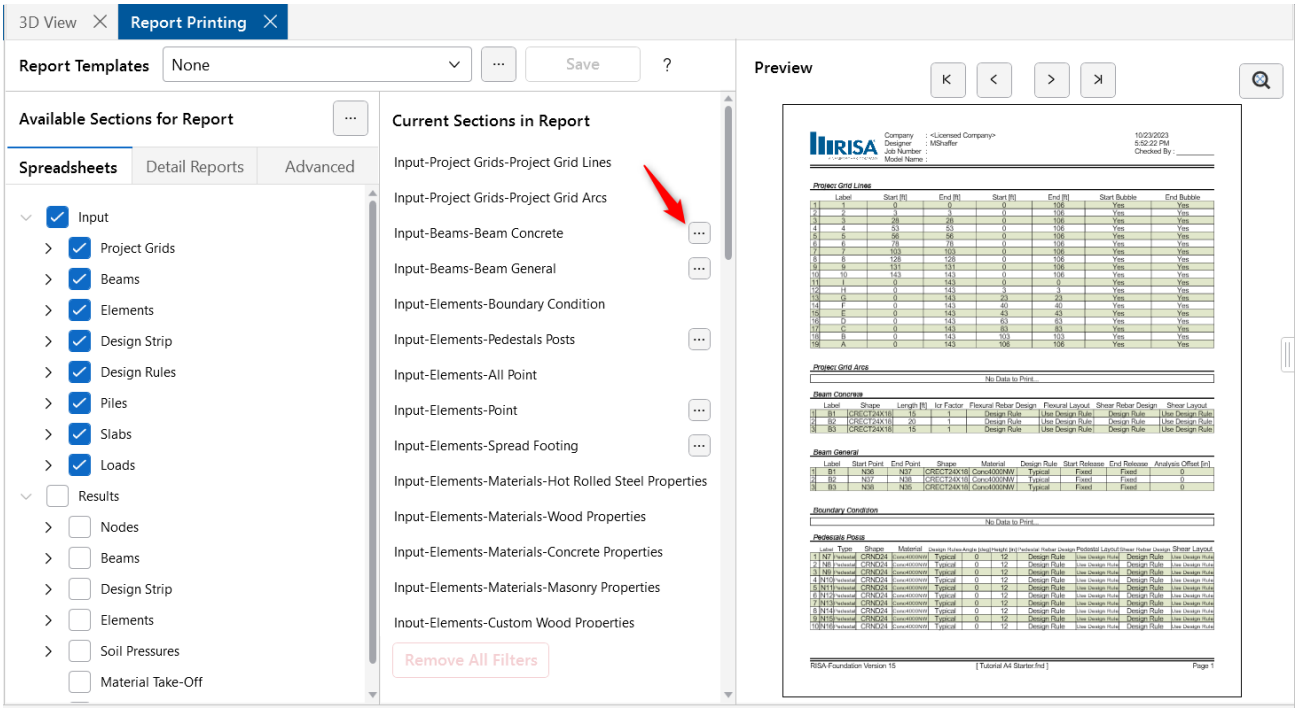


The **Report Printing** window options help you build your reports with spreadsheets, graphics, Detail Reports, etc. There are standard reports for you to choose from and you can also name and save any report format you custom build as your own template. To choose a standard report, simply pick it from the **Report Templates** drop-down list.

Building a Report

To build your own custom report you can select any of the spreadsheets, Detail Reports, or miscellaneous sections found in the **Available Sections for Report** and they will be populated in the **Current Sections in Report** region. You can add entire categories by selecting the heading of a drop-down section in the **Spreadsheets** tab. You can re-order the report by dragging and dropping the sections in your report you would like to move. You can also remove sections from your report by deselecting them, similar to how they were selected, or you can drag the item out of the **Current Sections in Report** region.

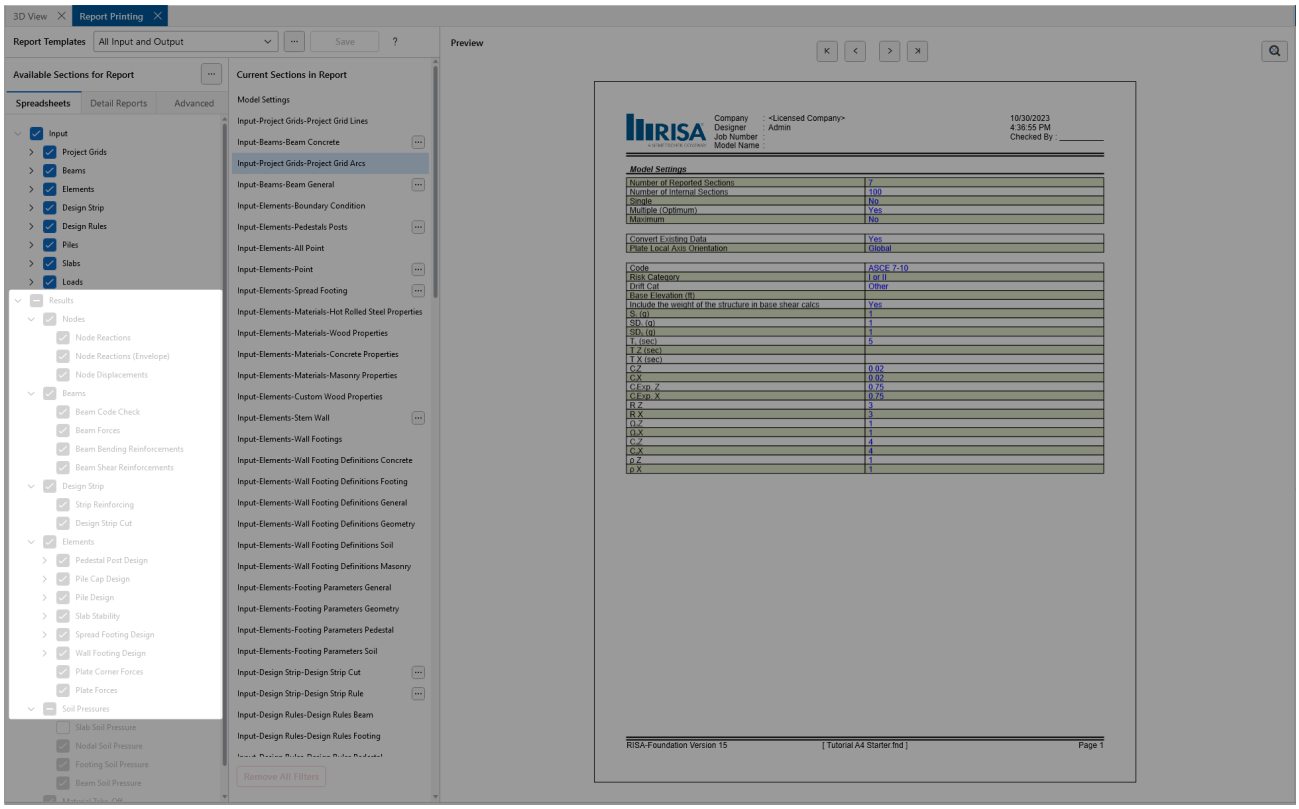
The **Filter Elements by Property** option will filter the sections included in the report based on the selected property. This applies to both spreadsheets and Detail Reports. Access the Filter Elements by Property dialog by selecting the (ellipsis) button next to the section, where applicable.



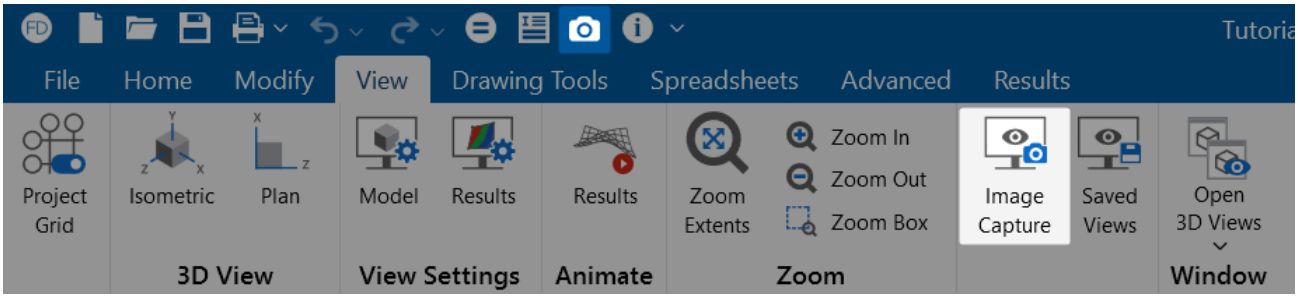
The **Current Sections in Report** portion of the dialog shows which sections and the order the printed report will be built. The **Advanced** tab of the **Available Sections for Report** contains all images that were added to the project, as well as the other miscellaneous items not in the **Spreadsheets** or **Detail Reports** tab. Images can be added using the "Snapshot" option, or by adding external images using the **Add Image** button.

Note:

- Any saved reports from the **Report Templates** drop-down will currently only save spreadsheet information. Any snapshots, Detail Reports or added items from the **Advanced** tab will need to be manually added to any saved reports.
- The solution spreadsheets will be grayed out in the **Available Sections for Report** region if a solution has not been run and specific results that would be contained in the spreadsheet are not available.



Snapshot (Capture) Images



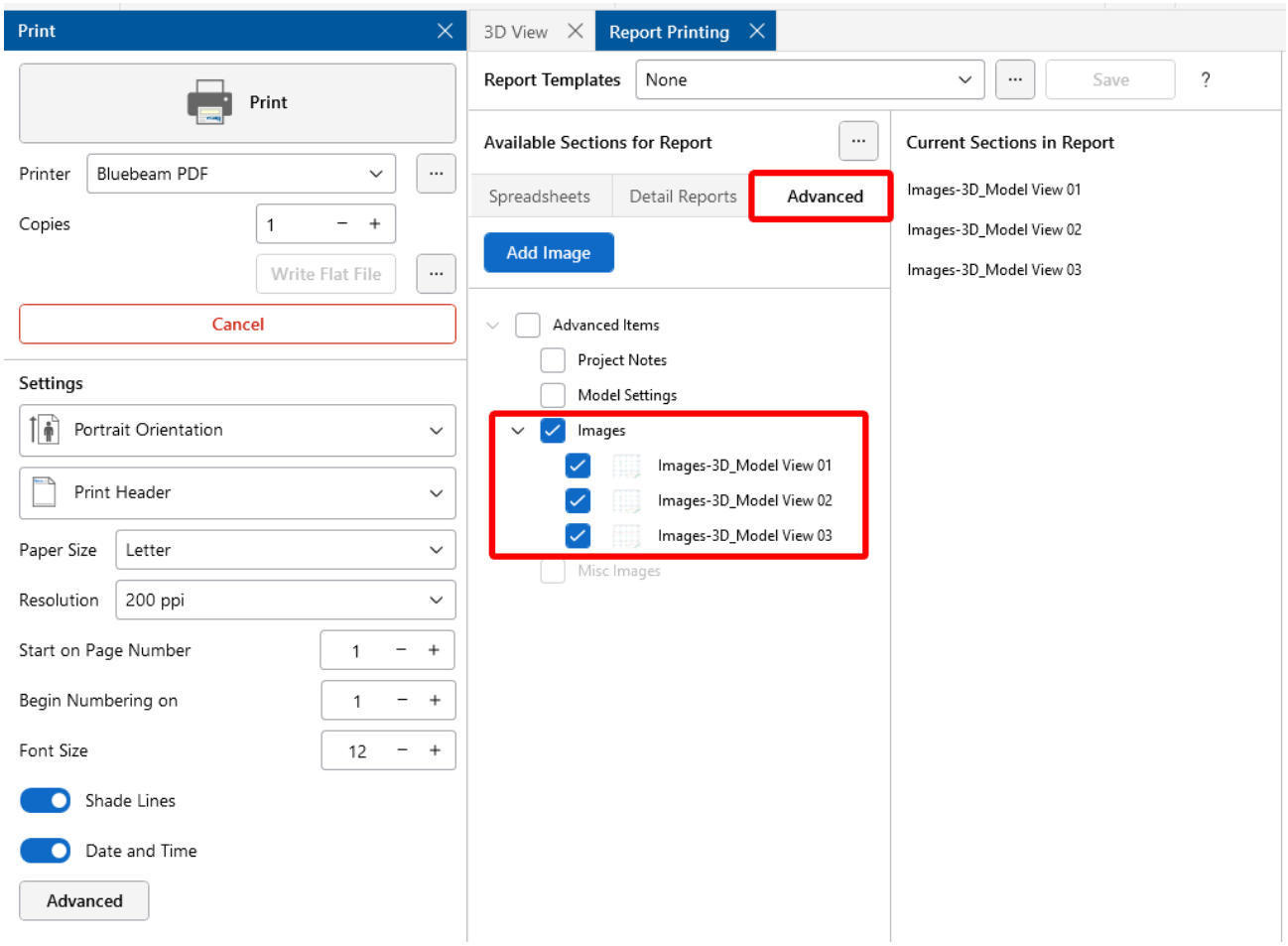
A snapshot image button is located on the **Quick Access Toolbar** and the **View** ribbon.

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 **File** > **Application Settings**> **File Locations**.The relative path of these image files will be saved with the model. If the model file is moved, move the associated **Images** folder as well and then all images will still be available.

This snapshot is a great way to get images of model views and Detail Reports that you want to add to your report. Once a snapshot is taken, it will show up in the **Images** section in the **Advanced** tab of the **Report Printing** dialog and can then be added to the **Current Sections in Report**.



Adding Images

Any additional images that you wish to add to your report that are not from RISA can be added using the **Add Images** button. Press this button and navigate to any PNG, BMP or JPEG file. Once you select it, it will then show up in the **Misc Images** section in the **Advanced** tab and can then be added to the **Current Sections in Report**.

Note: The complete path of these additional images are saved with the file. If the model file is moved, it will still find these additional images.

Report Print Settings

The **Settings** section allows you to specify many different formatting options:

Portrait or **Landscape Orientation** can be selected from the first drop-down menu.

Print Header – All reports have a footer with version information, the file name and path, and the page number. The header option will include the Model Title specified in the **Model Settings**, company, designer, job number and a place to initial any checking.

Start on Page Number – 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.

Begin Numbering On – 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.

Shade Lines – Allows you to shade every other line to enhance readability.

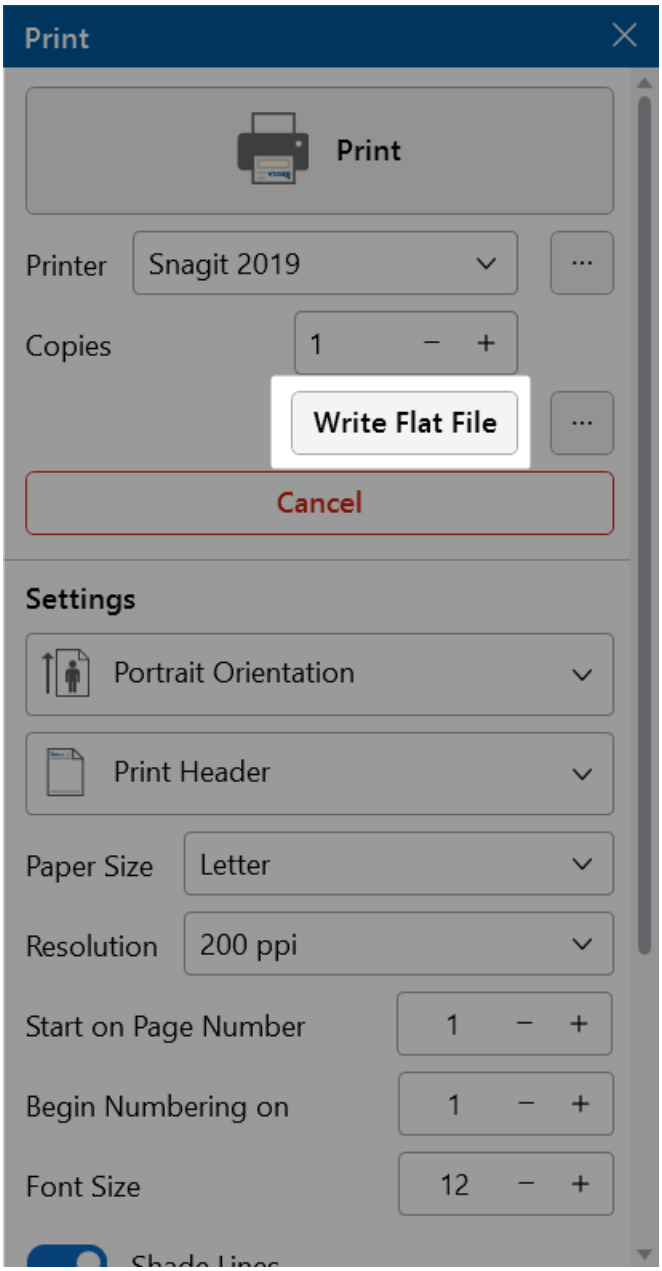
Date and Time – Allows you to turn off the Date and Time stamp so that they will not appear on your printed reports.

Item Options – Allows you to select member and plate related options. You can specify that you want member results to be listed for each member section (specified in the **Model Settings**), or just for the member ends. For plate elements, you can specify that only top stresses will be listed for the plate results.

Note: For more information about printing, including the ability to print out a RISA or custom company logo, see Customizing RISAFoundation.

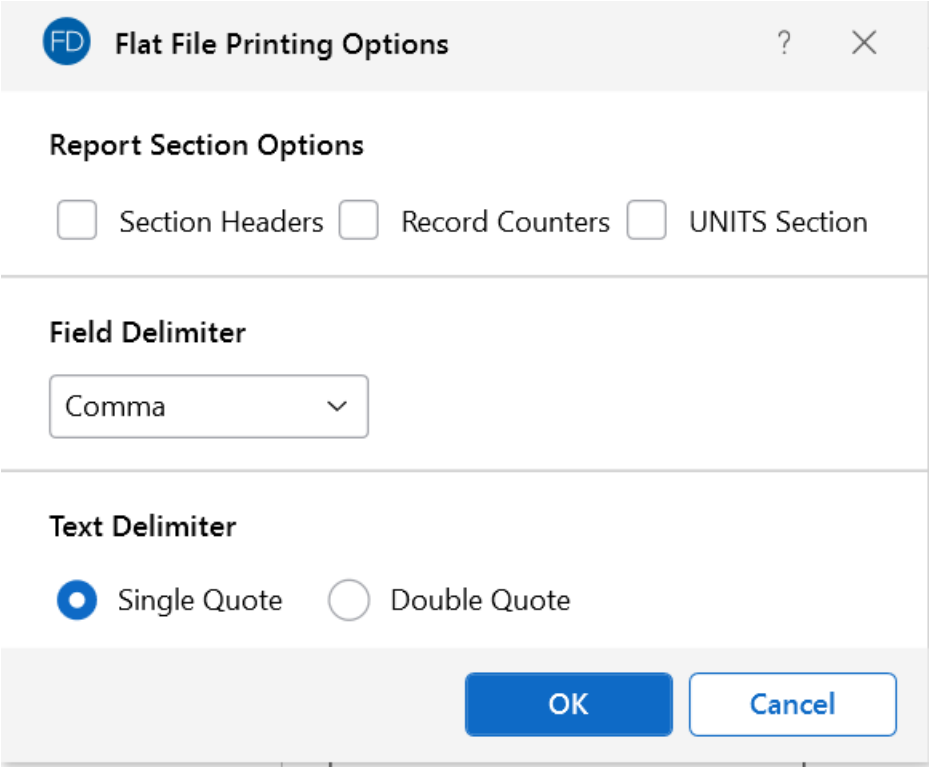
Printing to a Flat File

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. To print to a flat file, select **Write Flat File** in the **Print Report** dialog.



Flat File Printing Options

There are several options available to make the flat file easier to parse. Access these options by selecting the ··· button next to **Write Flat File** in the **Print Report** dialog. 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 the option to include the **Section Headers**, which will print a text description of each block of data. For example, the node data would be preceded by a [NODE] header on it's own line before the data. You can also include **Record Counters**, which are 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 print out a special **UNITS Section**, which will give you the units used for all the values in the program.

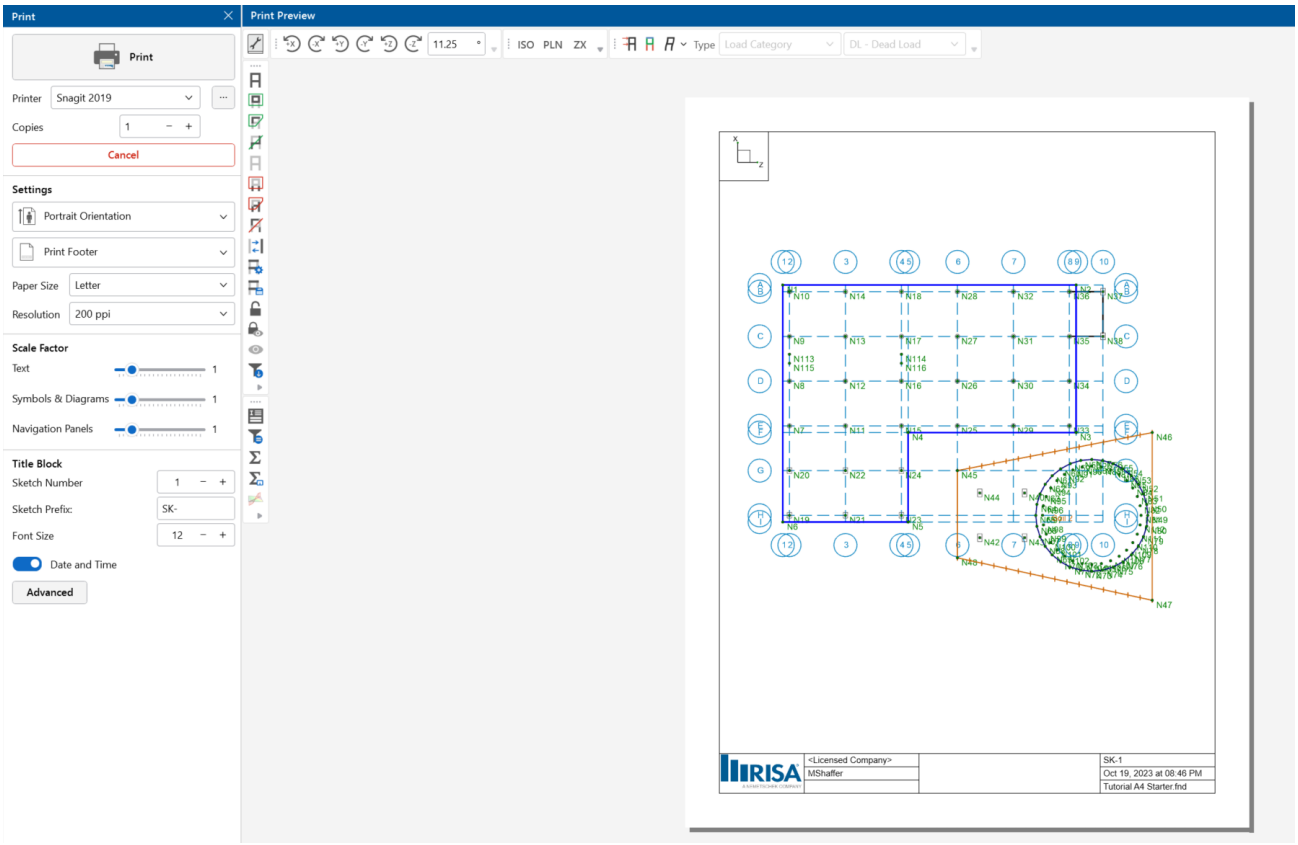
The **Field Delimiter** option lets you choose what character will be used to separate each field of data within record. For example, each coordinate value in the node data would be separated by the character selected. Programs that you might be importing the file into, like MS Excel or MS Access, often have options to select what the field delimiter will be for records of data.

The **Text Delimiter** works like the field delimiter, except it's used to set apart text labels. All text in the flat file will be enclosed at the beginning and end by the selected text delimiter character. This is very useful when trying to read in label strings that contain embedded spaces. As an example, the node labels in the node data would each be enclosed by a single or double quote.

The current flat file options are saved each time the program is closed.

Graphics Printing

At any point, while in a 3D View, you can print a graphic by choosing **Print Graphics** from any of the **Print** menus. The current window will change into a Print Preview and will allow all the same view settings to be adjusted, even when in the preview. Rotating, zooming and rendering are only a few of the settings that can be altered dynamically in the preview.

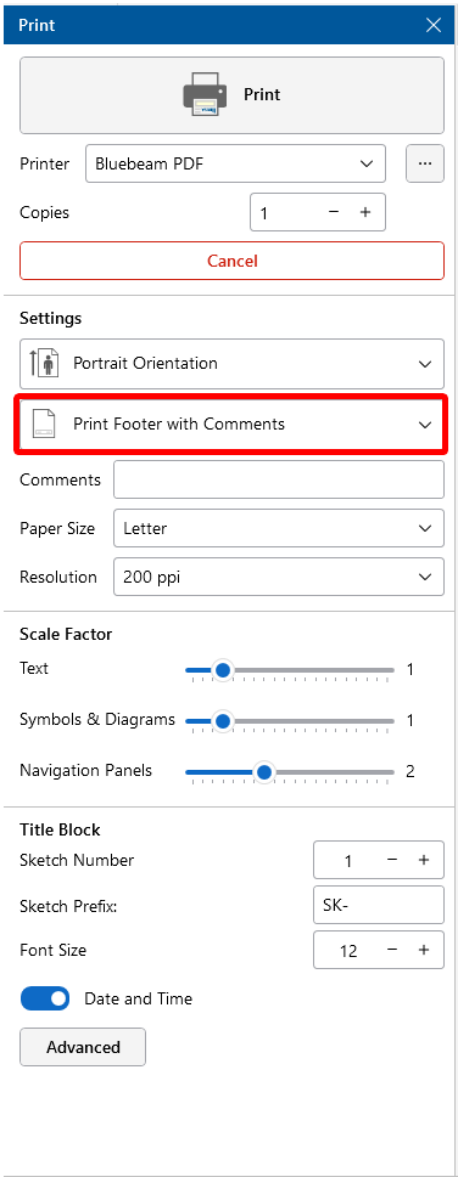


Many of the same options that exist in **Report Printing** are present in **Graphic Printing**, but some unique options are available for the **Scale Factor** and **Title Block**.

Three different **Scale Factors** are defined. These scale factors are used to make the text, symbols, Diagrams and Navigation Panel displayed as part of the graphic, larger or smaller. A higher scale factor makes the item bigger in the graphic.

You can include **Title Block** information at the bottom of the graphic to display the Model Title, Designer, Company Name, Date and Time (all of which are defined in your Project Information). Clicking the **Advanced Settings** button will open the **Advanced Graphic Print Settings** dialog where you can set the logo displayed in the report title block and make adjustments to the report margins.

Choosing **Print Footer with Comments** allows you to enter a comment that will be included in the Title Block. In addition to this, you can specify a **Sketch #** and a **Sketch Prefix** to label your graphic prints as well.



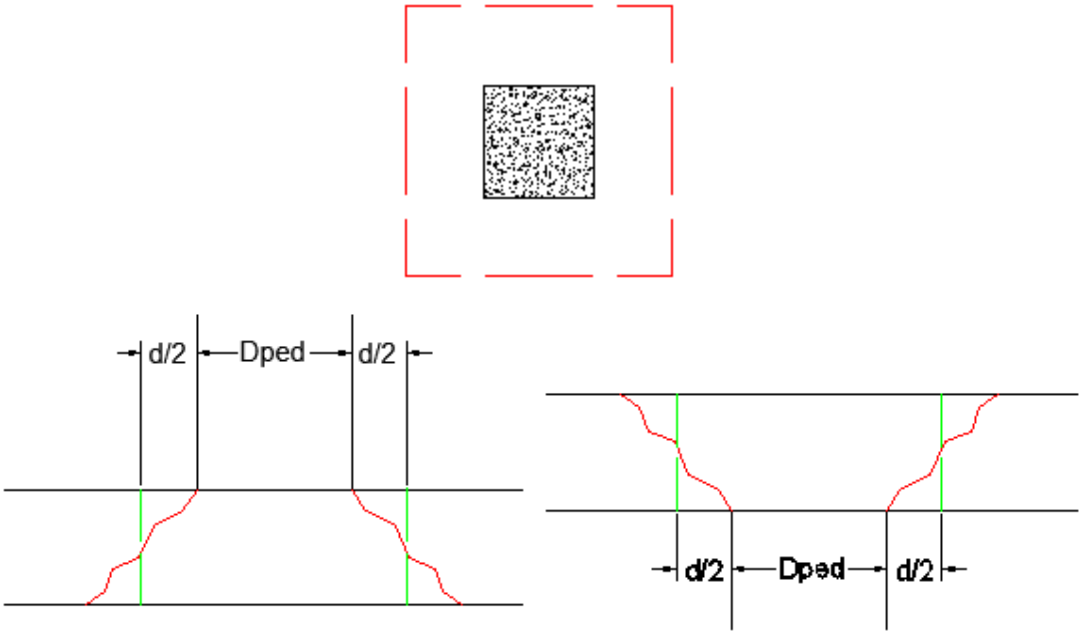
Choose **Print** once you have everything the way you want it. This will bring you to the print settings specific to your printer. The available choices will be different for each printer, but generally allow you to choose the printer, orientation, quality and quantity.

Punching Shear Design

Punching shear calculations are provided for columns/pedestals that land on concrete slabs. The following contains more detailed information on those calculations in RISAFoundation.

Punching Shear Basics

Punching shear is a phenomenon where a concentrated force on a slab causes a shear failure cone that "punches" through the slab.



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. 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

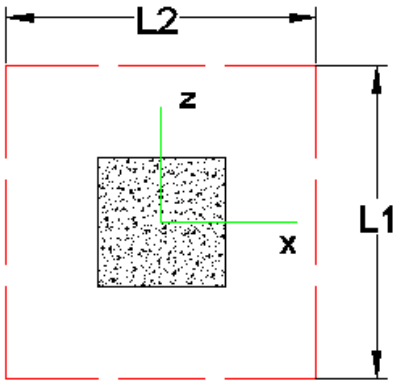
The effective depth of the slab is used to define the punching shear perimeter at a distance of d/2 beyond the edge of the column/pedestal.

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

L1 and **L2** represent the length of the critical section along the local axes. This value is used to define the punching shear perimeter as well as 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 318-14 section 22.6.4 and 8.4.2.3.

Polar Moment Ixx and Polar Moment Izz

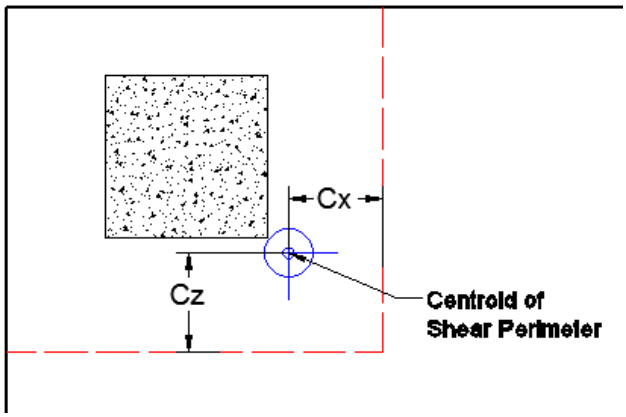
The **polar Moment** is a property (Jc) of the critical section which is analogous to polar moment of inertia about the local yy axes. This value is used to determine the shear stress in the critical location due to unbalanced moment.

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 8.4.4.2.3 (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

The **centroid** gives 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: For more information, see Punching Shear Loading.

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. For more information, refer to ACI 318-14 Section 22.6 (ACI 318-11 Section 11.11.7.2). 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 the 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 $v_{uv}>\varphi 2\lambda_s\lambda\sqrt{f'c}$ per Section 8.6.1.2. As,min is calculated and presented in the punching shear result of columns under this condition. Please note that the reinforcement design in RISA programs for slabs does NOT consider this requirement. Users need to consider separately the required additional slab reinforcement at punching shear critical sections to account for this code requirement. RISA provides the calculations of As,min per Equation (8.6.1.2) in the Detail Report.

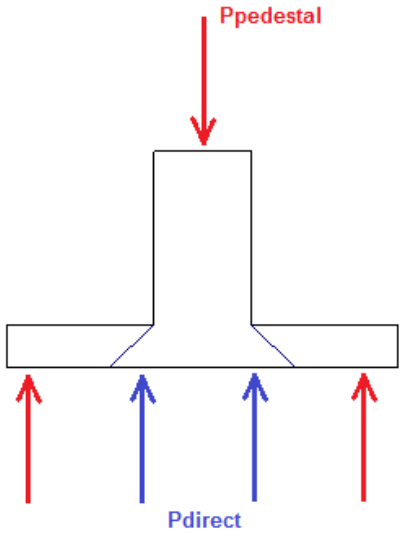
Punching Shear Loading

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 Pedestal/Post. The area within the punching shear perimeter loading is assumed to pass directly into the support, so that force must be discounted.

This includes soil pressure resistance that occurs within the punching shear perimeter.

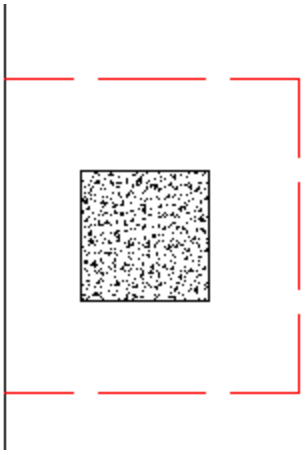


In the image above, the Vu demand equals $P_{pedestal} - P_{direct}$

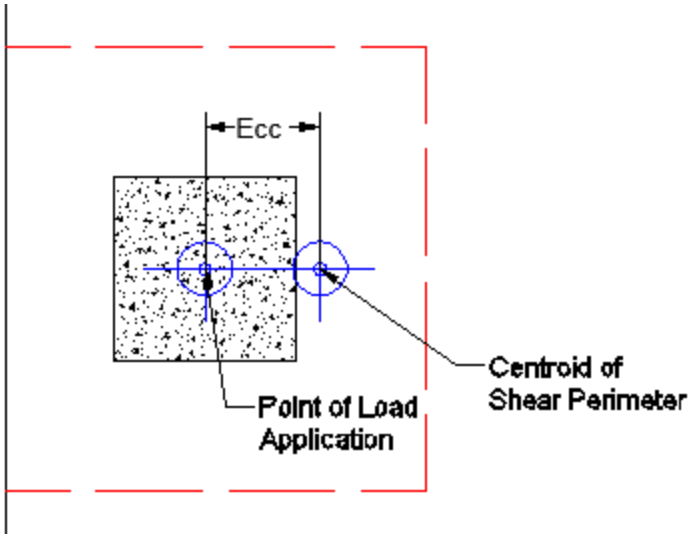
Moment Consideration

The moments used are also not necessarily just the moments directly from the Pedestal/Post. If the centroid of punching shear area is not the same as the centerline of the Pedestal/Post, then there will be an eccentricity that also adds/subtracts moment.

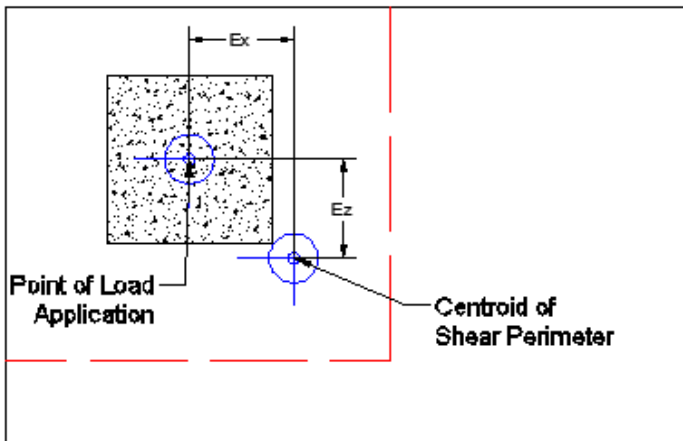
Take the edge punching shear situation, shown below:



The punching shear perimeter is to the right of the center of the Pedestal/Post location. This causes a moment in the punching shear calculation, even if a moment is not explicitly applied on the Pedestal/Post. This eccentricity is automatically accounted for in the punching shear stress calculation.



This same inherent eccentricity occurs in corner punching shear conditions. However, in this case, you get an induced moment in both local axis 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.

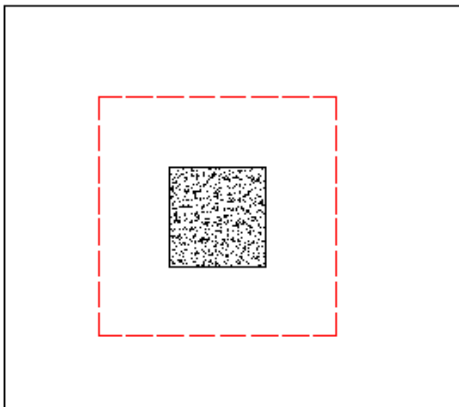


Note: The moments due to eccentricity are not accounted for in the detail report moment output. The moments shown there are taken directly from the Pedestal/Post.

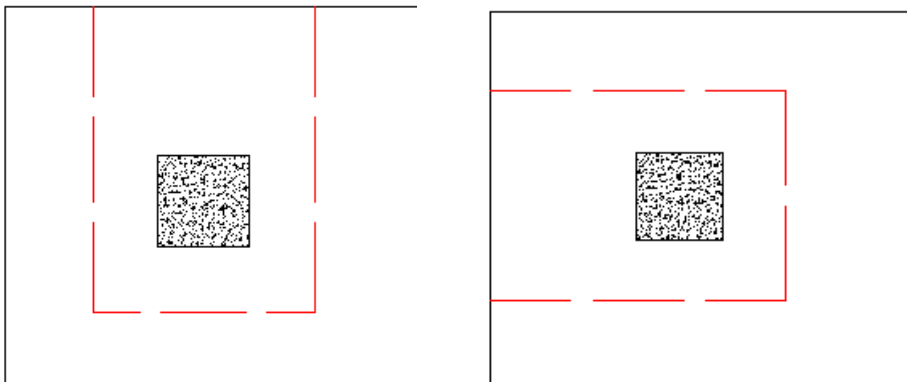
Finding the Controlling Case (Interior, Edge or Corner)

The program considers as many as nine different punching shear perimeters for each column in the model. 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.

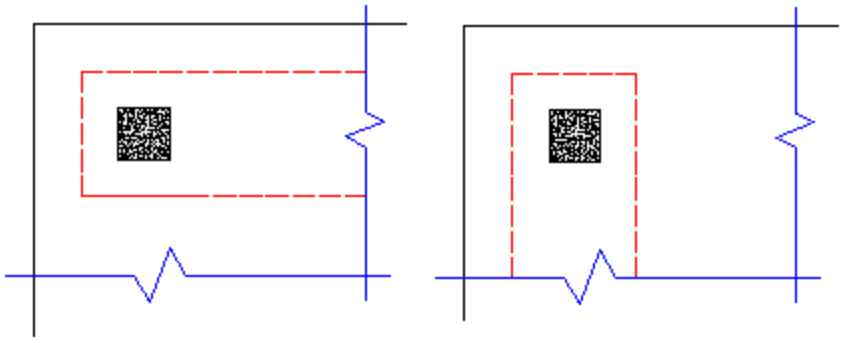
For example, let's consider a Pedestal/Post that has an interior punching shear perimeter close to a corner.



Here, the program will also check out any EDGE punching shear perimeter considerations. The two examples below would be the most likely ones for this given geometry.

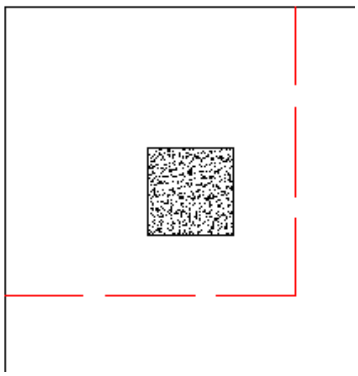


However, these other two edge conditions would also be checked:

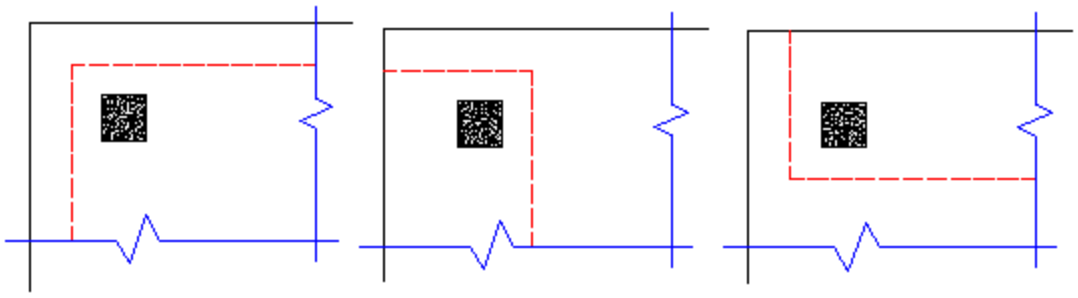


In most cases, these odd situations will not govern. However, large moments in a given direction can cause punching shear perimeters with larger areas to govern in specific cases.

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:



Again, 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 some specific cases.

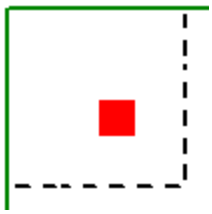
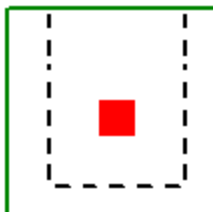
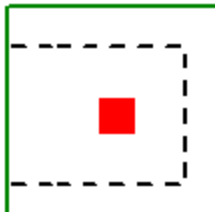
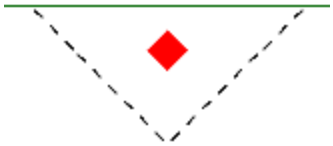
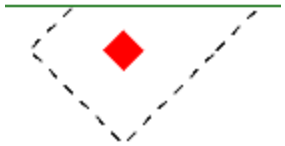
Whichever configuration produces a maximum code check will be reported in the punching shear results.

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 instances where an edge case is governing that by inspection you might think would be governed by a corner case. See the images below:

Corner Case	Edge Case 1	Edge Case 2
		
		

In the 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 can occur in some instances.

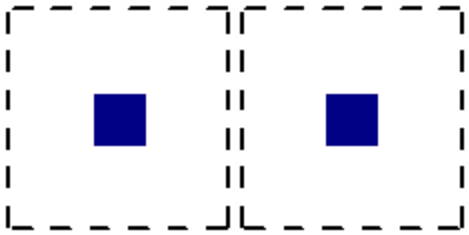
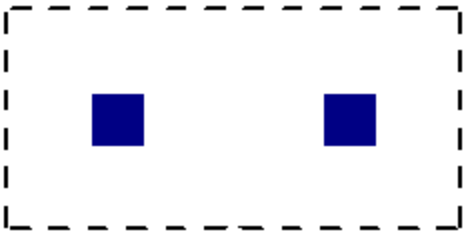
Other Punching Shear Considerations/Limitations

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. Also, there will be a warning message in the **Detail Report** when there is a different thickness detected. The L1 and L2 values used when calculating b_o will use the depth at the location of the pedestal/column.

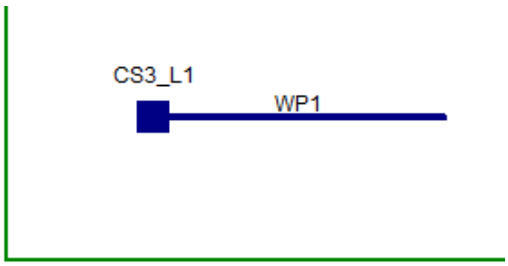
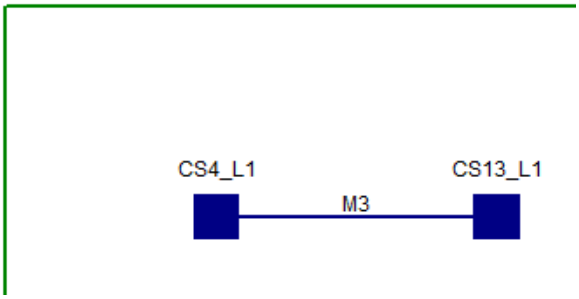
Pedestal/Post Groups

It may be possible for multiple, closely spaced pedestals/posts, to create a punching shear perimeter around the GROUP at a lower force level than would be observed by investigating one of the individual pedestals/posts. This is NOT currently considered in the punching shear code checks.

Individual Punching Shear Perimeter	Group Punching Shear Perimeter
	

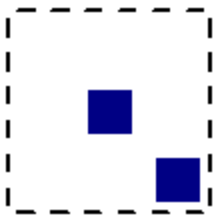
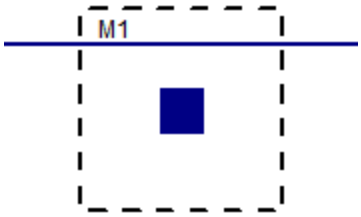
Beams, Columns and Walls intersecting the influence area of a Pedestal/Post

When a beam or wall is co-linear with the Pedestal/Post joint, the program will not provide a punching shear check on the Pedestal/Post. 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 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 column or beam.

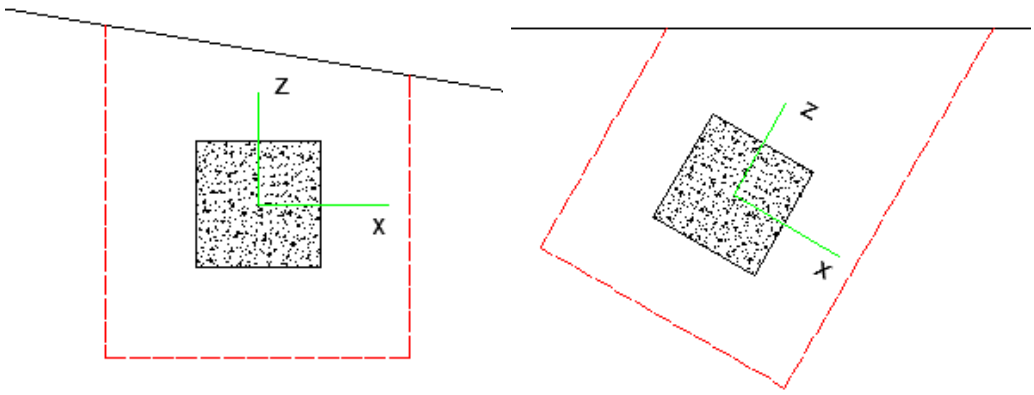
	

When the Pedestal/Post Local Axes Do Not Correspond to the Slab Geometry

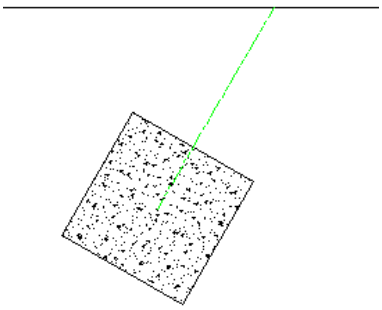
The punching shear perimeters are always investigated with respect to the Pedestal/Post local axes. As shown below, there may be cases where these local axes do not correspond to the slab edge or corner geometry adjacent to that Pedestal/Post. In cases like these, it is possible for this to result in an artificially large assumed punching shear perimeter.

Edge Conditions

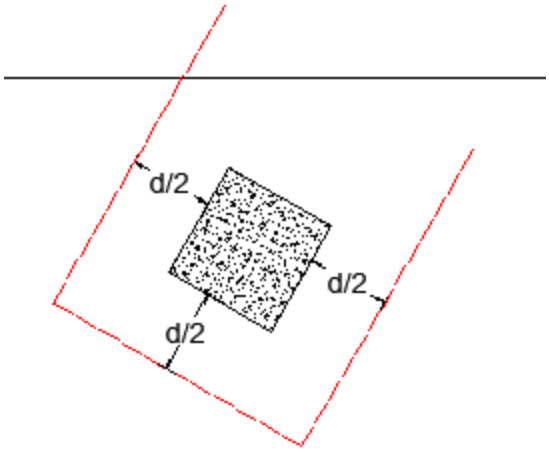
It is possible to get edge shear perimeters if either the slab edge is sloping, or a column local axis is rotated, as shown below:



The program calculates this perimeter by drawing a line in the local axis direction along the centerline of the Pedestal/Post, as shown below:



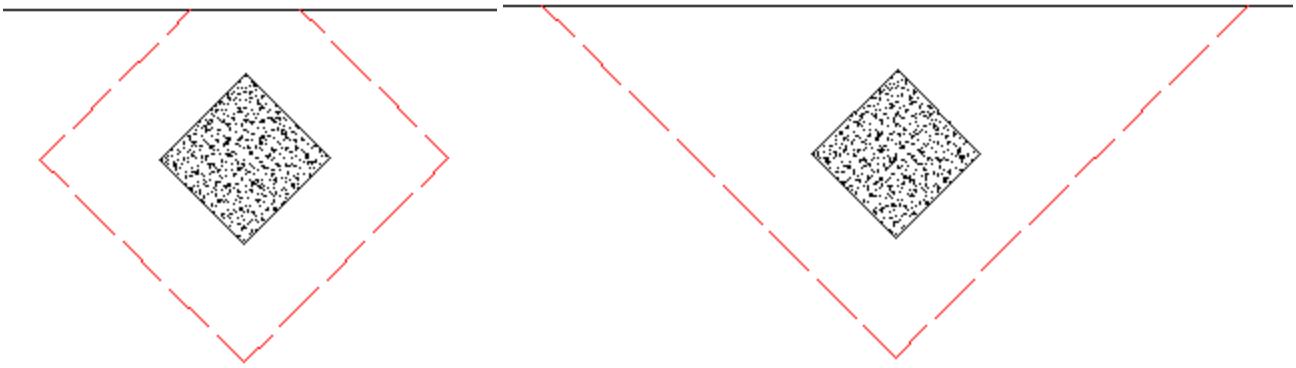
From here, the program offsets this line on either side of the Pedestal/Post at a distance d/2 from the face. This produces a shear perimeter that looks like this:



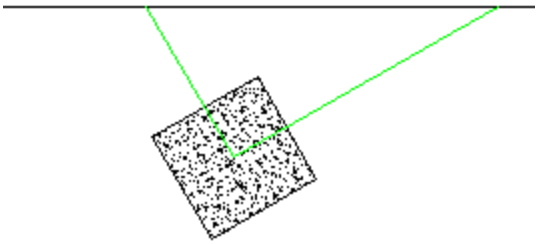
Though this shear perimeter is not technically correct, it typically produces a nearly identical shear perimeter to the actual shear perimeter shown above.

Corner Conditions

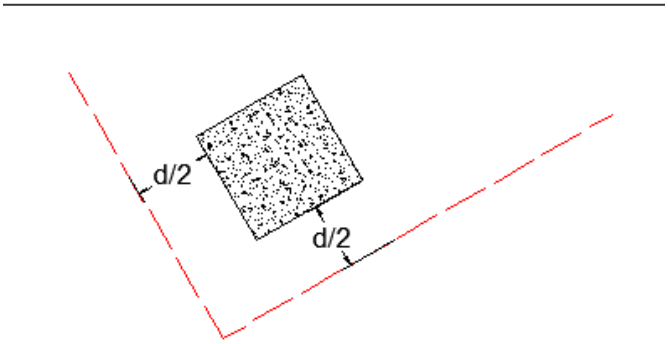
If either the slab edge is sloping, or the pedestal local axis is rotated, it is possible to get a corner shear perimeter. The program starts by trying to draw the full shear perimeter. In this case, we find that the shear perimeter has two sides that run into a slab edge. When this occurs, the program considers this a corner. There are two extremes for what the punching shear perimeter might look like for this case.



In reality, the punching shear is likely somewhere between these two extremes. What the program does here is similar to what is done for the edge case, except now this is done for both axes on one side only. Lines are drawn along the local axes of the Pedestal/Post until they reach the edge:

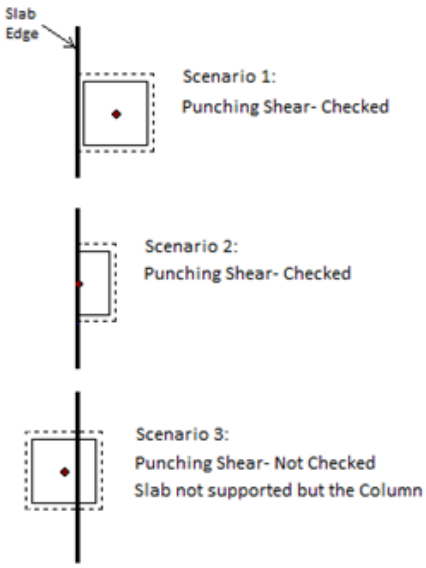


From here, the program offsets this line on the side away from the edge at a distance d/2 from the face. This produces a shear perimeter that looks like this:



The red line represents the L1 and L2 values the program uses to calculate punching shear perimeter. This perimeter doesn't actually hit the edge of the slab in our calculation, but this puts us in a situation where we are between the two extremes shown above and is a conservative approximation.

Note: Pedestals/posts are defined at a single node in the model. If this node lands inside of the slab, then a punching shear check will be performed. If the node does NOT land on the slab, then the Pedestal/Post will not be considered attached to the slab and no punching shear check will be done.



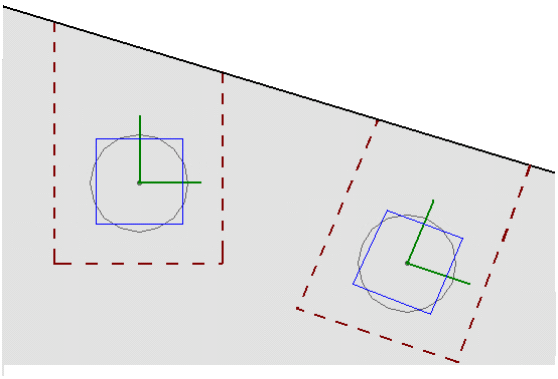
Using Equivalent Square Pedestal/Post

When calculating the punching shear perimeter for a round Pedestal/Post, 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 cross-section. 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 Pedestal/Post would be considered a 9.424" x 9.424" square for use in 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.

Results

You may work with the results of a solution by viewing and sorting data in the spreadsheets, graphically plotting them within the model, or by viewing detail reports for individual model elements. 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** menu in the **Explorer** panel and the **Node Reactions Spreadsheet**. If any warnings have been written to the **Warning Log**, it is displayed as well. You may specify that other results be displayed automatically in the Application Settings. You can then proceed to view any results and make any needed changes for further analysis.

Any changes you make to the model will void the results, such as moving nodes or adding beams. Once the results have been cleared, another solution will be required before the results can be viewed again.

Each of the result types is described in it's own section:

Results Spreadsheets

- Node Deflection Results
- Node Reaction Results
- Soil Pressure Results
- Beam Force Results
- Beam Design Results
- Pedestal/Post Design Results
- Pile Design Results
- Pile Cap Design Results
- Plate Force Results
- Plate Corner Force Results
- Design Cut Results
- Strip Reinforcing Results
- Wall Footing Design Results
- Spread Footing Design Results
- Slab Stability Results

Detail Reports

- Beam Detail Reports
- Spread Footing Detail Reports

- Pedestals/Posts Detail Reports
- Pile Detail Reports
- Pile Cap Detail Reports

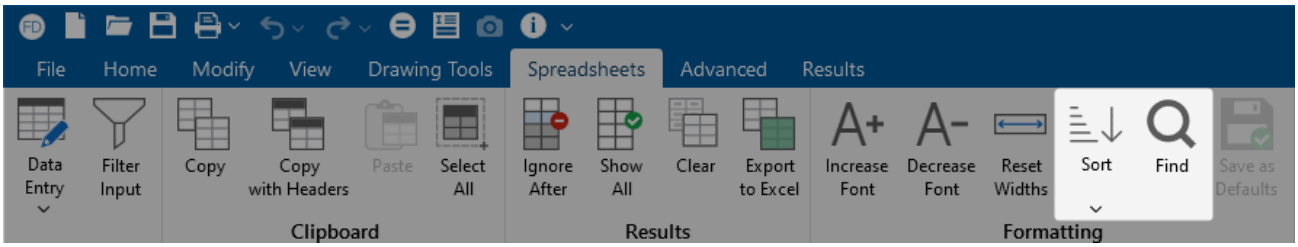
Saving Results

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 open as well. You may use the **Application Settings** in the **File** tab of the ribbon toolbar 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 in the **Explorer** panel, or from the **Results** tab in the ribbon toolbar. You may use the **Find**, **Sort** and **Exclude** features to find the results you are interested in. For example, you might sort the beam stresses from high to low, bringing all of the highly stressed members to the top. You might then exclude members that do not have significant axial stresses so that they do not distract you or so that they are not printed.



Finding Results

To go to a certain item while in a spreadsheet, click the **Find** icon on the **Spreadsheets** tab, or select **Find** from the right-click menu and type in the label for the model element.

Sort Results

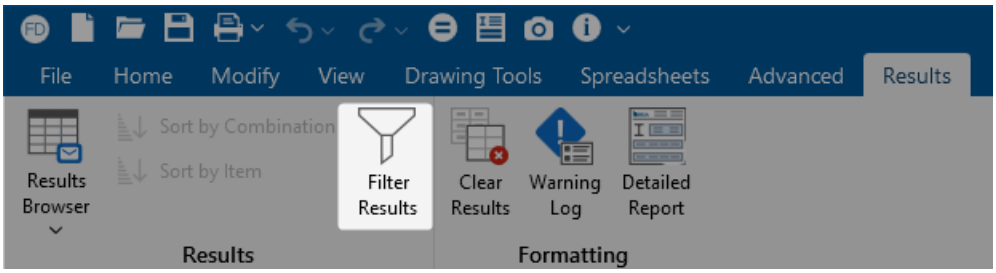
To sort the results, click on the column of results you wish to sort and then click the **Sort** icon on the **Spreadsheets** tab, or select **Sort** from the right-click menu to specify sorting settings. You can sort based on maximum, minimum, absolute maximum or input order.

Exclude Results

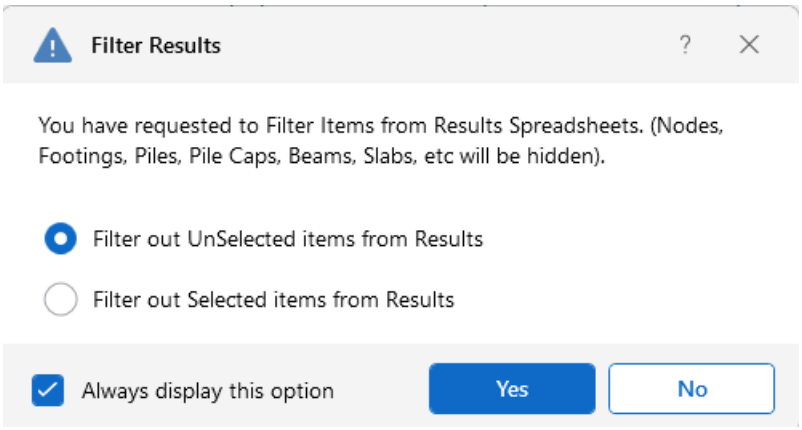
You can exclude results so that you can work with only the results that are important to you.

Excluding Results After the Solution

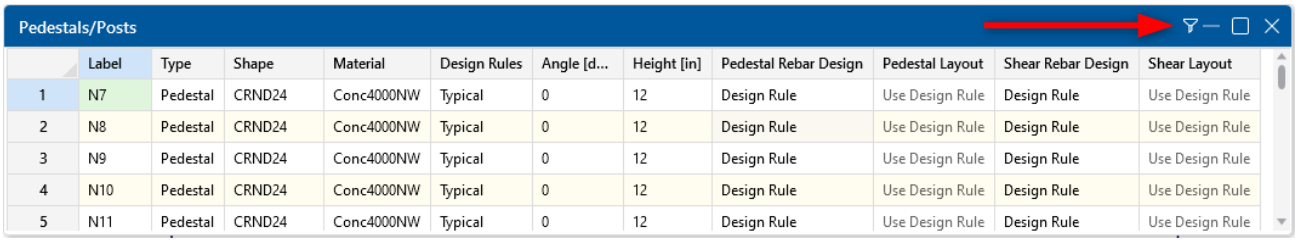
You can run the solution and then graphically select foundation elements that are of interest. Clicking the **Filter Results** button will open a pop-up dialog where you can select to **Filter Out UnSelected items from Results** or **Filter out Selected items from Results**. You can then update the spreadsheets and printed reports so that they will not have results for the selected or unselected items. All of the spreadsheets and reports will be controlled by this selection and you can adjust this at any time.



This option is also available by clicking the **Filter Results** button from the **Selection Toolbar** in the 3D View.



When you open any results spreadsheet, you can tell if it has been filtered by the filter icon in the spreadsheet header, shown below:

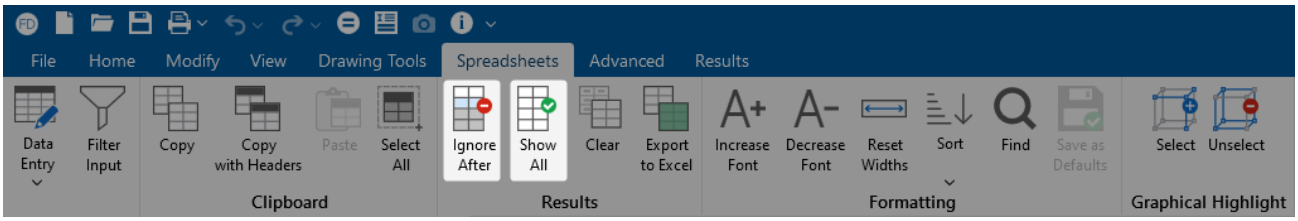


	Label	Type	Shape	Material	Design Rules	Angle [d...	Height [in]	Pedestal Rebar Design	Pedestal Layout	Shear Rebar Design	Shear Layout
1	N7	Pedestal	CRND24	Conc4000NW	Typical	0	12	Design Rule	Use Design Rule	Design Rule	Use Design Rule
2	N8	Pedestal	CRND24	Conc4000NW	Typical	0	12	Design Rule	Use Design Rule	Design Rule	Use Design Rule
3	N9	Pedestal	CRND24	Conc4000NW	Typical	0	12	Design Rule	Use Design Rule	Design Rule	Use Design Rule
4	N10	Pedestal	CRND24	Conc4000NW	Typical	0	12	Design Rule	Use Design Rule	Design Rule	Use Design Rule
5	N11	Pedestal	CRND24	Conc4000NW	Typical	0	12	Design Rule	Use Design Rule	Design Rule	Use Design Rule

Excluding Results While in Spreadsheets

You can also select the results of interest on each spreadsheet. While viewing spreadsheet results, you can select the last line of interest and exclude the rest. Exclusions are applied independently for each results spreadsheet. Excluding items graphically will reset **all** of the spreadsheets to match the graphic selection. If you wish to combine these two features to fine tune the results, perform the graphic selection first.

To exclude results, click on the last line of results you wish to keep and then click the **Ignore After** button on the **Spreadsheets** tab. You can bring the results back by clicking on the **Show All** button. Alternatively, you can select these features from the right-click menu in any result spreadsheets.



Note: It might be best to first sort the results before excluding. As an example, say that you only wanted to view beams with code checks greater than or equal to 0.7. To do this, you would open the **Beam 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 beams after that one.

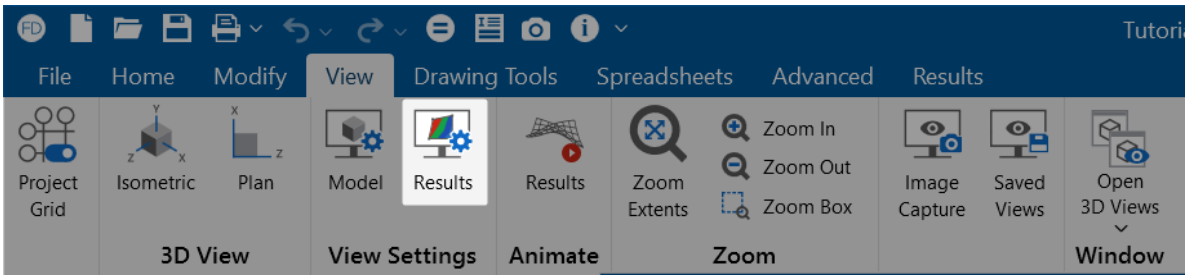
Graphic Results

Most of the analysis results may be viewed graphically, as well as in the spreadsheets. You can plot reactions for nodes in the model. For beams, you can plot force diagrams, as well as color-code the beams by code check results. Slab stress contours and plate corner forces may also be viewed graphically. Deflected shapes may be viewed and animated, as well. For more information, see Graphic View Settings.

Plot Results Graphically

To plot results graphically:

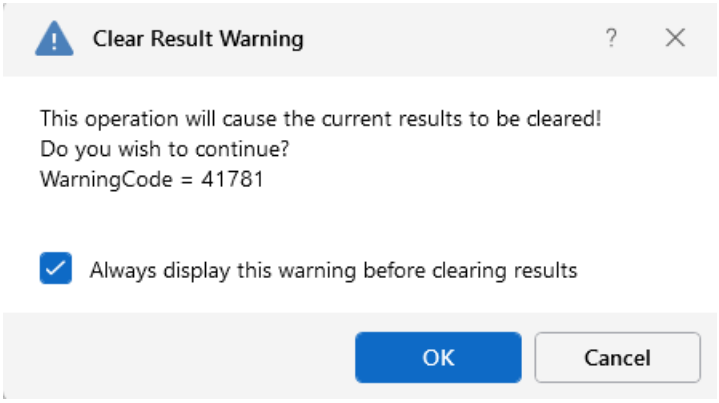
1. If you do not already have a model view open, click the **Open 3D Views** button under the **View** Ribbon.



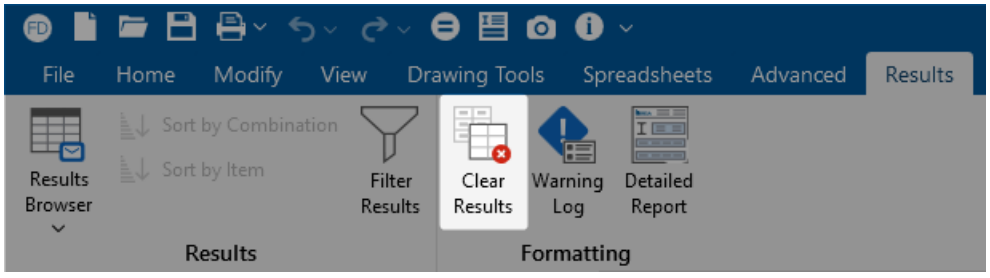
2. Click the **Results** icon for graphic view settings.
3. Select the options you would like to view.
4. Click **Apply** or **OK**.

Clear Results

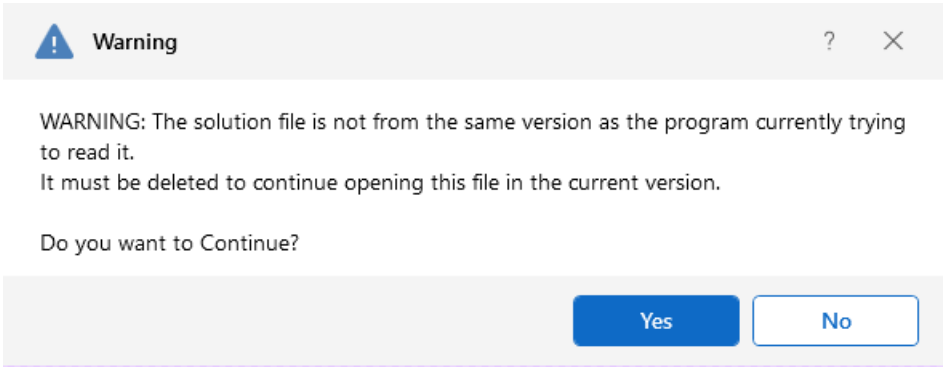
You will be provided with a warning if changes are made to the model that might invalidate the current results. Should you decide to proceed, the results will automatically be cleared and you will have to re-solve the model to get results once you are finished making changes. This warning may be disabled in the **Application Settings**.



To manually clear results, click the **Clear Results** icon on the **Results** ribbon.



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.


Warning
WARNING: The solution file is not from the same version as the program currently trying to read it.
It must be deleted to continue opening this file in the current version.
Do you want to Continue?
Yes
No

RISAFoundation Integration

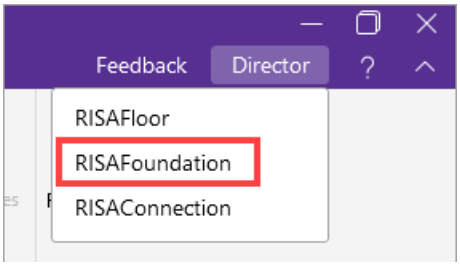
RISAFoundation has the ability to transfer load information from RISA-3D and RISAFloor and use this information to design foundations. The following will describe how these loads come across the design modules so you can accurately use RISAFoundation for your foundation design.

RISAFoundation Integration with RISA-3D

RISAFoundation transfers joint reactions from RISA-3D into RISAFoundation by way of load categories. This is not the same as load combinations. Load categories are defined in the Basic Load Cases spreadsheet of RISA-3D.

To get your reactions to transfer from RISA-3D to RISAFoundation:

1. Loads must be split into specific load categories (DL, LL, etc.) under the Basic Load Cases spreadsheet.
2. The model must be solved.
3. Click on **Director** and choose **RISAFoundation**, to transfer to RISAFoundation.



Note:

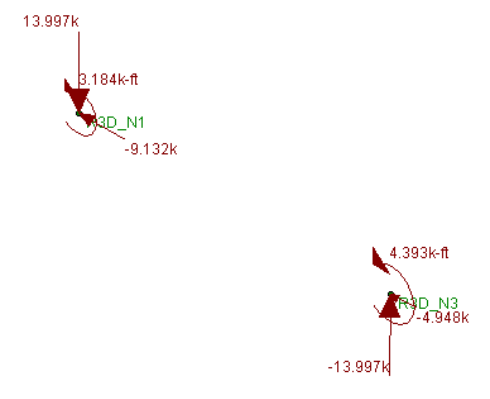
- If you do not define a load category for a basic load case, the loads from that case will not be transferred into RISAFoundation.
- You must solve at least one single load combination.
- When you are working between RISA-3D and RISAFoundation, Model Settings will only be brought over to RISAFoundation the first time you transfer from RISA-3D. After this first transfer, any time you change something in the RISA-3D Model Settings that would affect RISAFoundation, you would also have to make the change in RISAFoundation.
- 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 points at the base of the wall that define the wall mesh in the RISA-3D model.

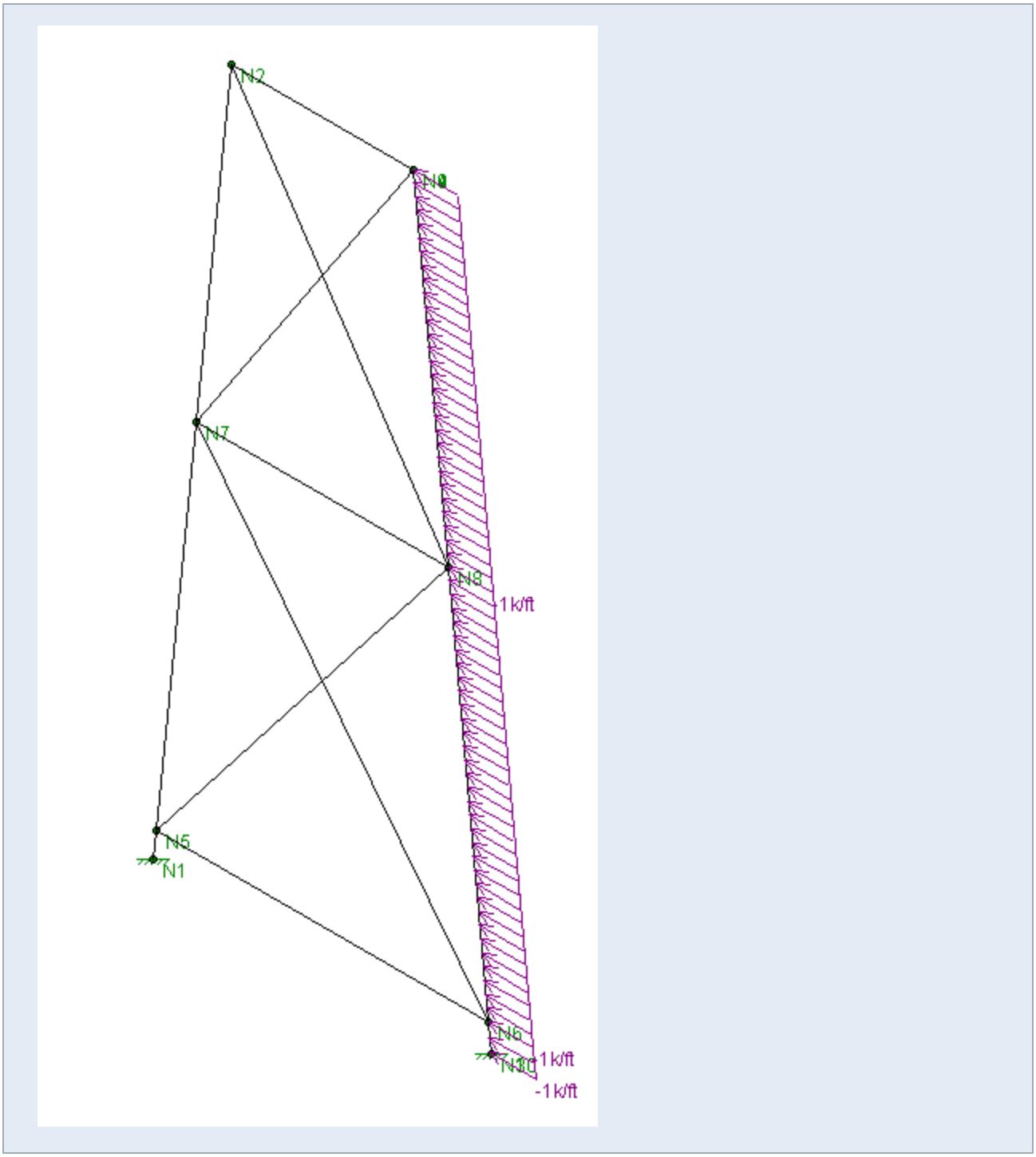
Example of Interaction

A simple example would be the braced frame structure seen below on the left. This frame 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.

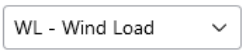
RISA-3D Wind Loads Graphical Representation

RISAFoundation Wind Loads Graphical Representation





First, a load combination is run (note that this does not have to have the wind load 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 nodes in a plan view of your structure are displayed. Clicking the ISO button will then give an orientation similar to the one shown above. Next, choose



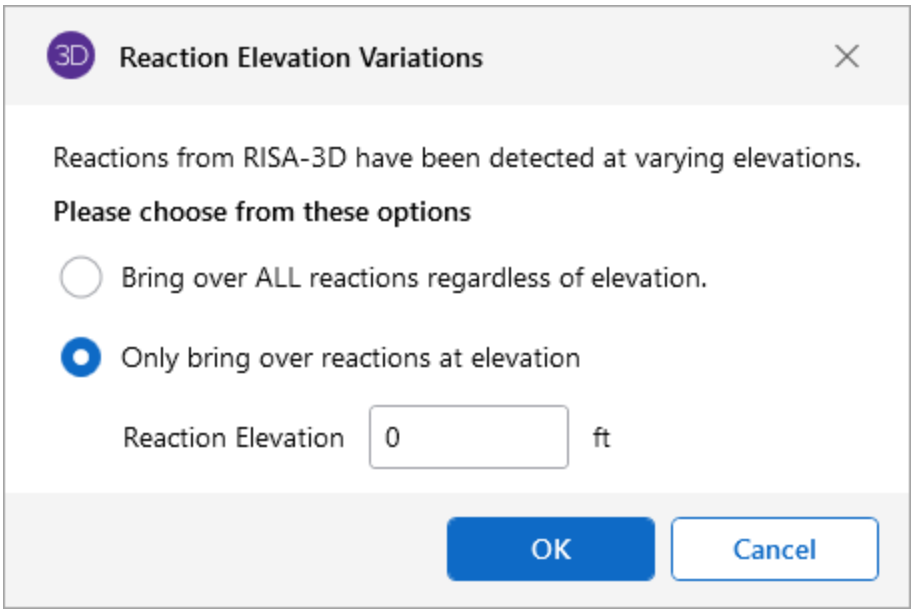
WL-Wind Load from the drop down list. The loads shown above on the right will be displayed.

Note:

- The nodes are relabeled in RISAFoundation, though labeled in a way that can be easily compared to the original RISA-3D model nodes. You can scroll through the different types of loads and compare them to RISA-3D to verify accuracy.
- 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 there are boundary conditions that are at different elevations when moving into RISAFoundation from RISA-3D, a window opens that looks like this:



You can choose to bring over reactions at a single elevation of your choice, which may be helpful if there are some reactions that are due to an adjacent structure or support that is not a foundation.

You can also 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 Integration with RISAFloor

To transfer from RISAFloor to RISAFoundation, you must first pass through RISA-3D.

Gravity Elements in RISAFloor

Elements (columns, walls) defined with a gravity function 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 to the overall moment, M, and axial force, P in the 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 and brought over into RISA-3D. Thus, RISAFoundation does not bring any information in from RISAFloor for lateral members. Instead this information is brought into RISAFoundation from RISA-3D as discussed above.

Limitations

RISAFoundation can not recognize moving loads

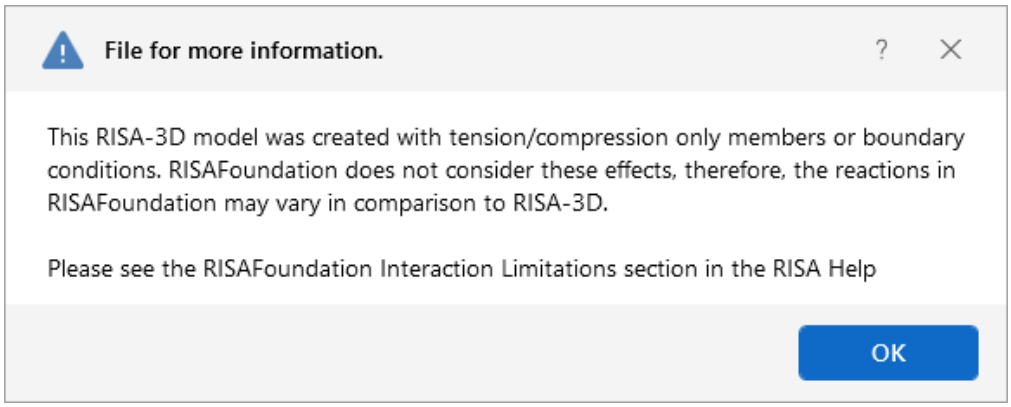
Moving Loads are not described as a load category in RISA-3D. Therefore, moving load reactions can't be read as a loading within RISAFoundation. Thus, these loads will need to be applied manually to account for them in the foundation design.

RISAFoundation requires your Y-axis to be vertical

Models coming from RISA-3D need to have the Global Y-axis set as the vertical axis to be brought into RISAFoundation. If the Y-axis is not the vertical axis, a dialog box will pop up indicating this when you try to use the director to move to RISAFoundation.

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 Load Combination was used to create them since different members may be active for different Load Combinations. Because of this, the Load Category reactions that make their way into RISAFoundation are imperfect since 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 Load Combination 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 the other load categories present in a particular Load Combination.

Shape Databases

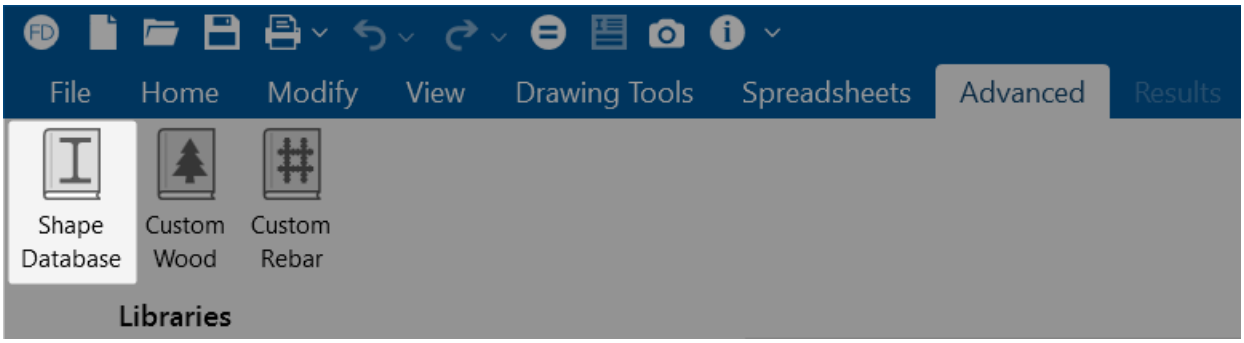
For each material, there are several databases of common structural shapes such as Hot Rolled Steel, Wood, and Concrete. You may type in the shape names directly, select shapes from these databases or add your own shapes.

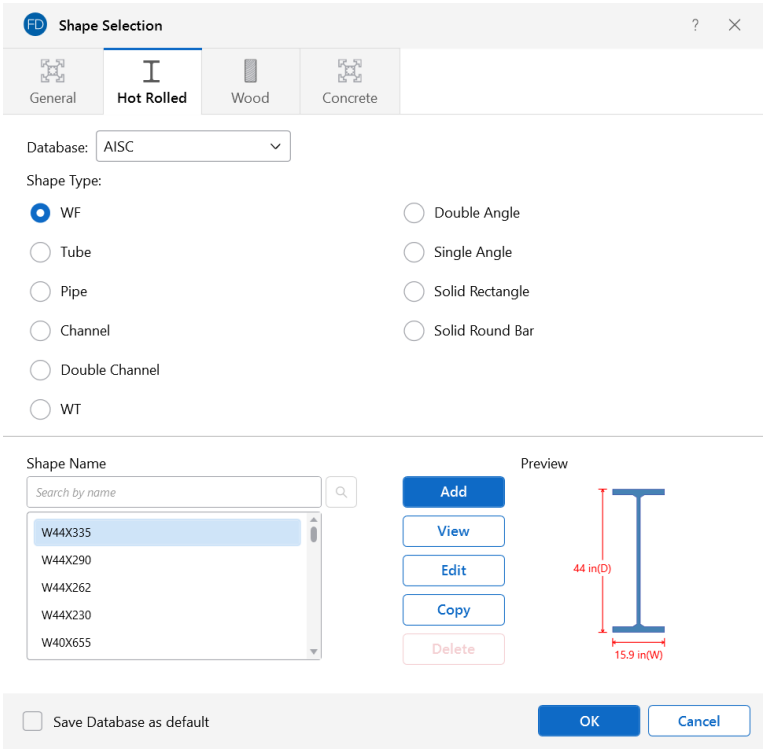
Database Shape Types

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** fields in the **Beams**and **Pile Definitions Spreadsheets**. Alternately, you may click the ... (ellipsis) button to open the **Shape Selection** dialog to assist with selecting or creating shapes.

Hot Rolled Shapes

Hot Rolled shapes are accessed by clicking the **Shape Database** button from the **Advanced** tab of the ribbon, and then clicking the **Hot Rolled** tab within the **Shape Database Editor** dialog.



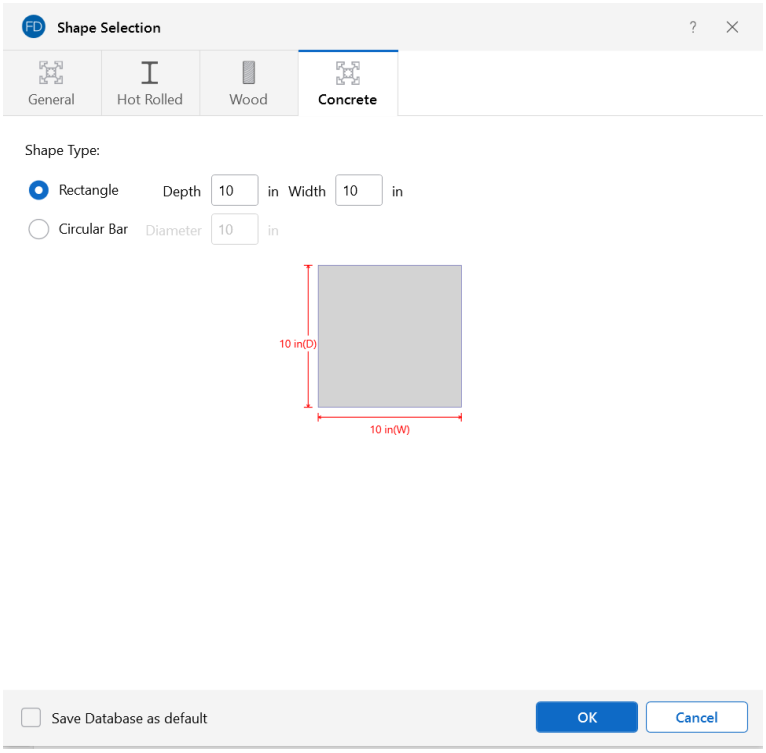


The hot rolled shapes and databases are more fully described in the Hot Rolled Steel Design section.

Note: You can check the **Save Database as Default** check box to have your settings saved for the next time you open the program.

Concrete Shapes

Concrete shapes do not have a predefined database like hot rolled steel and wood. 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. For more information, see Beams - Database.

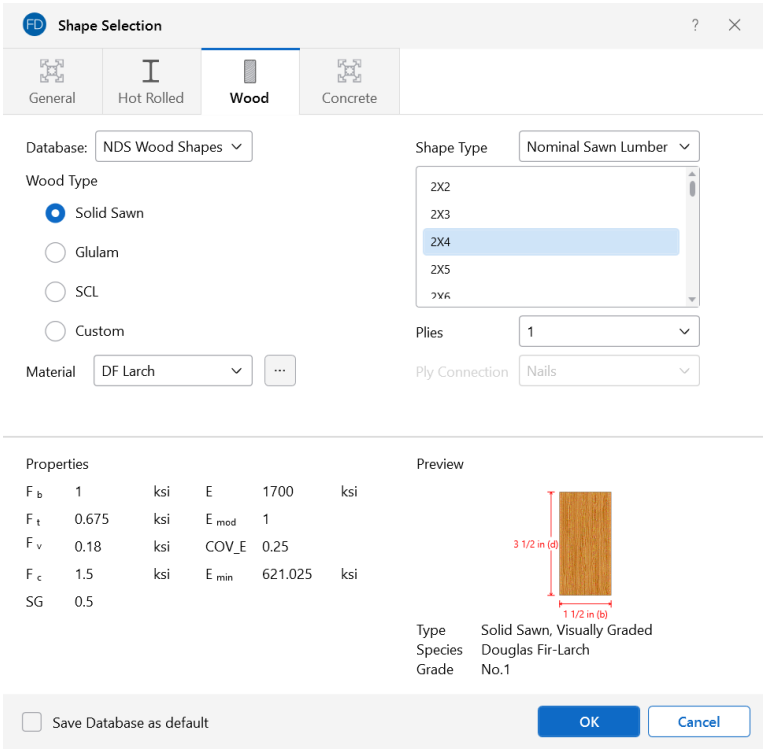


Note: You can check the **Save Database as Default** check box to have your settings saved for the next time you open the program.

Wood Shapes

The available wood shapes are based on the dimensional lumber and post and timber shapes given in the NDS or CSA O86, depending on which database you have chosen. You may also design for multiple plies of these shapes. Note that the NDS dimensional 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 specified for the selected **Material**. For more information, see Wood Database.



Note: You can check the **Save Database as Default** check box to have your settings saved for the next time you open the program.

General Shapes

The **General Shape** type is provided so that additional shapes can be added to the shape database that do not fall within the other available categories.

Code checks are not calculated for general shapes since their place in the relevant specifications is unknown. Everything else will be calculated for them (forces, deflections, stresses).

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.
- 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.

Pipe Database Shapes

Pipe shapes, which are hollow circular shapes, are entered as on-line shapes. The syntax for these shapes is "PIdiaXthick", where "dia" is the pipe outside diameter and "thick" is the pipe thickness (in inches or centimeters). For example (assuming US Standard units), PI10X.5 would be a 10" diameter pipe with a wall thickness of 1/2".

Solid Rectangular Shapes

These shapes can be defined as on-line shapes. The syntax is "REhtXbase", where "ht" is the rectangle height and "base" is the rectangle base (in inches or cm). For example, RE10X4 would be a 10" deep, 4" width rectangular shape (assuming US Standard units). These shapes can also be defined in the **Shape Editor**. When defined in the **Shape Editor**, the depth of the solid rectangular section must always be greater than or equal to the width.

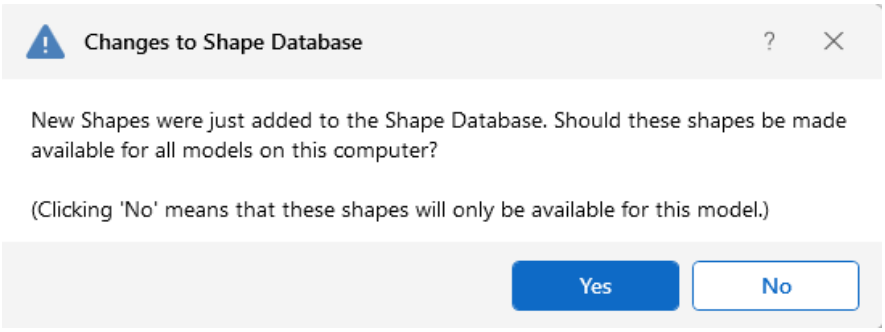
Solid Circular Shapes

These shapes are defined as on-line shapes. The syntax is "BARdia", where "dia" is the circle diameter. For example (assuming metric units), BAR2 would be a circular bar with a diameter of 2 cm.

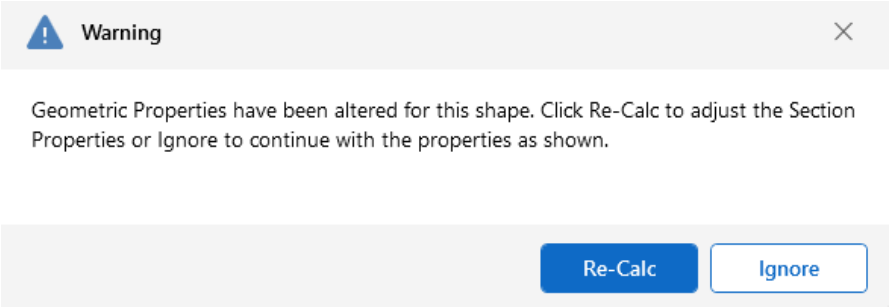
Database Files

The shape databases are stored in the database files (*.FIL). These files may only be edited through the program. The path to these files is set in the **File Locations** tab of **Application Settings**, which is found in the **File** menu.

After adding a new shape in your model, you will be prompted with the **Changes to Shape Database** dialog, shown below. 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.



Note:

- Alterations to the Shape Databases are not permanent unless you agree to save them. Changes that are not saved only remain valid for the current session and will not be present the next time you start RISA.
- New shapes are always added to the bottom of the database in blue text.
- To delete a shape, specify the database and shape type you wish to delete and then click the **Delete** button.
- To edit a shape, click the **Edit** button and edit the shape properties. Only shapes that are not part of the default database are able to be edited. If the shape you would like to edit is a default database shape, the program will ask if you would like to make a copy of the shape. Geometric values can be edited here, and will allow you to recalculate the section properties if edits have been made.

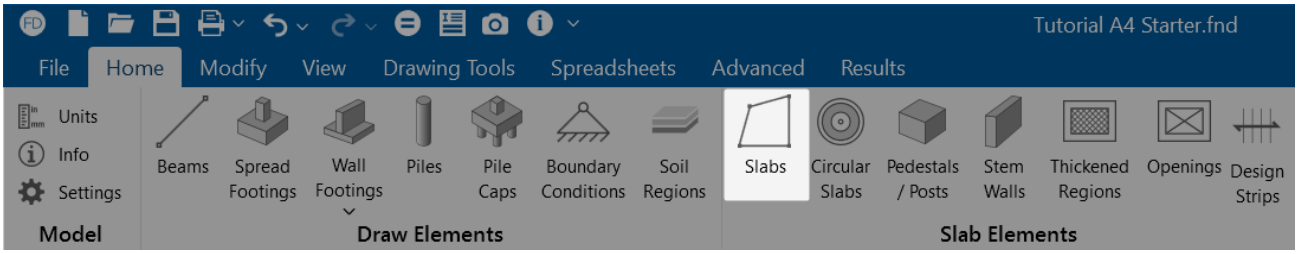
Slabs

In this section, we will describe how to draw a slab and how RISAFoundation handles the automatic generation of finite element plates. For information about reinforcement design for slabs, see Slabs - Design Strips and Slabs - Design Cuts.

Draw Slabs

You can draw slabs using a Drawing Grid, Project Grid or by clicking on existing nodes. You can set your slab properties up front, or you can modify these properties after drawing a slab.

You may define the slab by drawing a polygon that defines the entire slab. Or, you may draw multiple polygons that each define a smaller portion of the slab. If the border of your slab region aligns with another slab, both will be fully connected as if they were a single slab element. See the Automatic Slab Meshing section for a discussion on how the slabs are meshed.



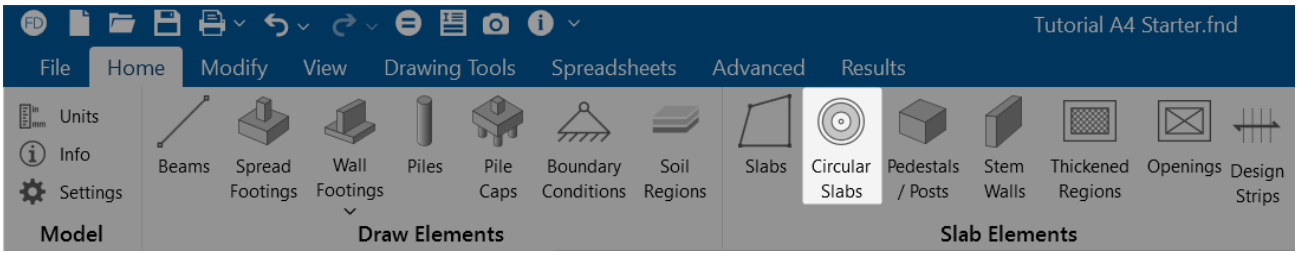
To create a Slab:

1. If there is not a model view already open, go to the **View** ribbon.
2. Click the **Open 3D Views** icon in the **Window** section to open a new view.
3. Go to the **Home** ribbon.
4. Click the **Slabs** icon in the **Slab Elements** section.
5. In the **Properties** panel, set the slab properties.
6. Start drawing a polygon to define slab region. To indicate that you have finished drawing the region, double-click on the last node of the polygon.

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

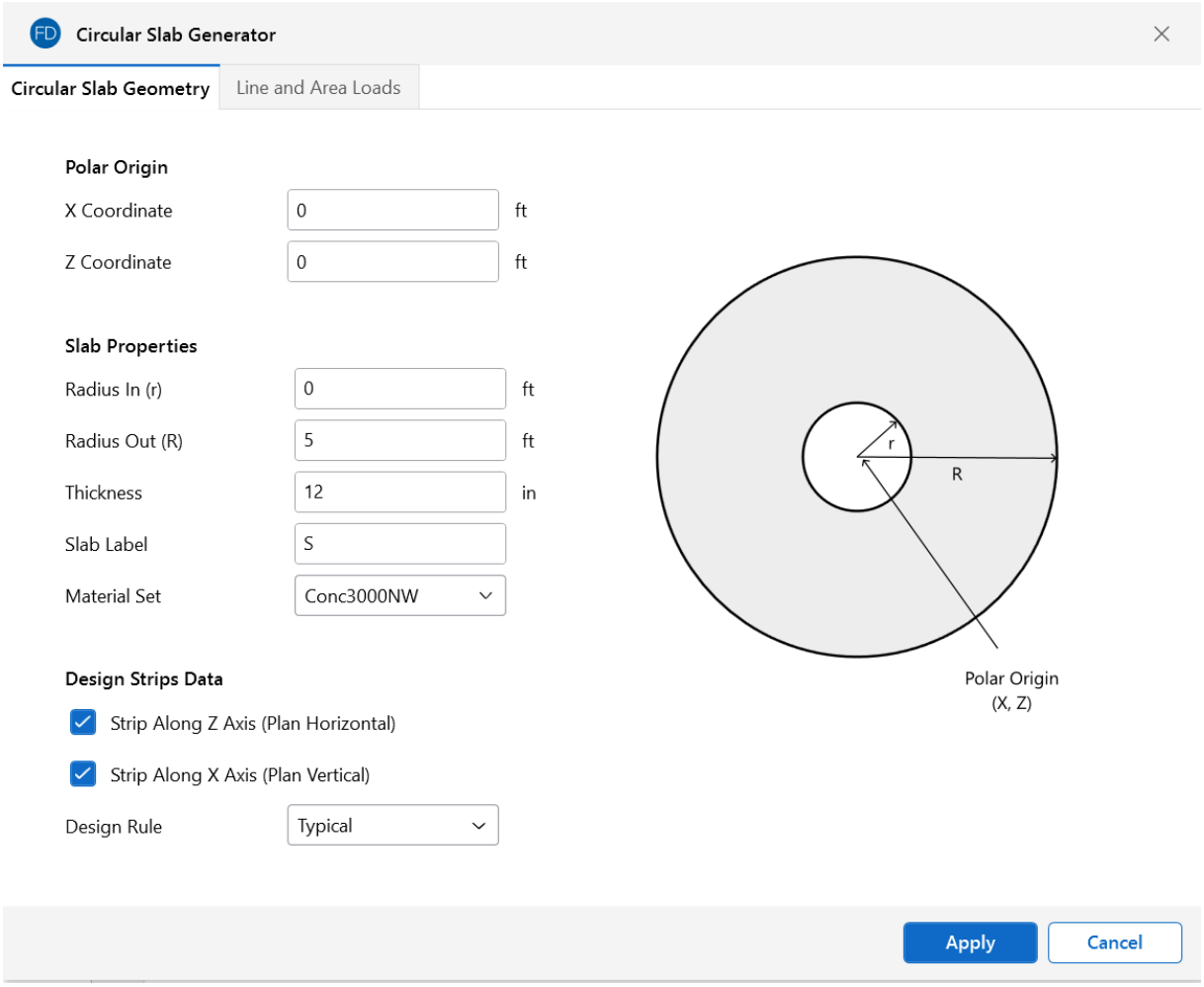
Circular Slab Generator

The Circular Slab Generator allows users to quickly lay out a circular mat slab for design. Within the generator are options to include circular line loads and area loads so that you can automatically create your typical loading for round structure such as tanks.



To create a Circular Slab:

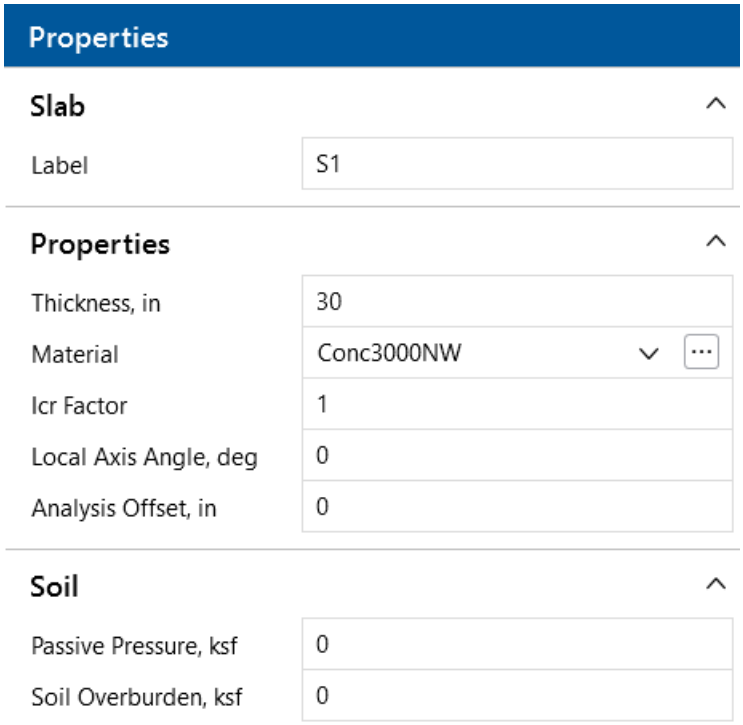
1. If there is not a model view already open, go to the **View** ribbon.
2. Click the **Open 3D Views** icon in the **Window** section to open a new view.
3. Go to the **Home** ribbon.
4. Click the **Circular Slabs** icon in the **Slab Elements** section.
   The **Circular Slab Generator** dialog displays, shown below:



5. Enter the **Polar Origin** and **Slab Properties** information.
6. Select the **Strip Along Z Axis (Plan Horizontal)** and/or **Strip Along X Axis (Plan Vertical)** check boxes to have Design Strips automatically added to the slab for each respective direction.
7. Click **Apply** and the dialog closes. The circular slab will display in your 3D View.
8. Adjust properties as needed in the **Properties** panel.

Modify Slab Properties

There are a number of ways to modify slabs. You can view and edit the slab data in the **Slabs Spreadsheet**, which can be accessed by selecting **Slabs** from the **Data Entry** drop-down on the **Spreadsheets** tab of the ribbon, or by clicking **Slabs** from the **Data Entry** section of the **Explorer** panel. You may also select a slab (or multiple slabs) to view and edit the properties in the **Properties** panel. Here you can adjust the material, thickness, local axis angle, analysis offset, passive pressure, soil overburden and Icr factor. Clicking the ··· (ellipses) button within the Material input of the **Properties** panel or **Slabs Spreadsheet** will open the **Review or Change Material** dialog, where you can modify the element's material properties.



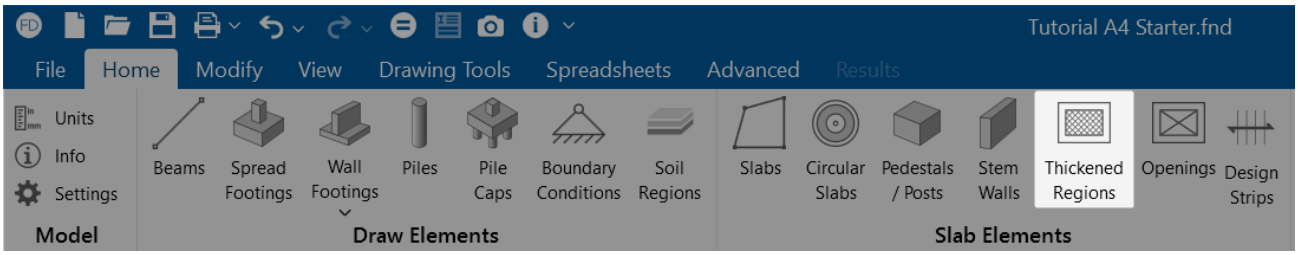
To modify a Slab:

1. If there is not a model view already open, go to the **View** ribbon.
2. Click the **Open 3D Views** icon to open a new view.
3. Click a single slab, or select multiple slabs that you wish to modify.
4. If you have multiple elements selected, specify which element type to view properties for by using the **Selection Properties** drop-down in the **Properties** panel.
5. Set the parameters for the slab(s) within the **Properties** panel.

Note:
- For more information on the Local Axis Angle, see the Local Axis Angle section.
- For more information on the Passive Pressure, see the Passive Pressure section.
- For more information on Soil Overburden, see the Soil Overburden section.

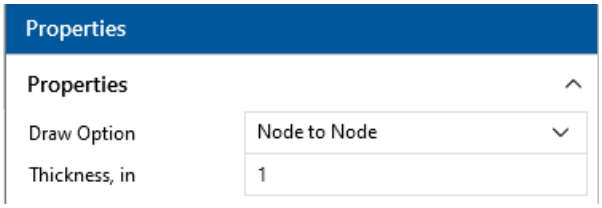
Draw Thickened Regions

You can draw **Thickened Regions** within the interior of an existing slab element to capture changes in slab thickness.



To draw a **Thickened Region** within a slab:

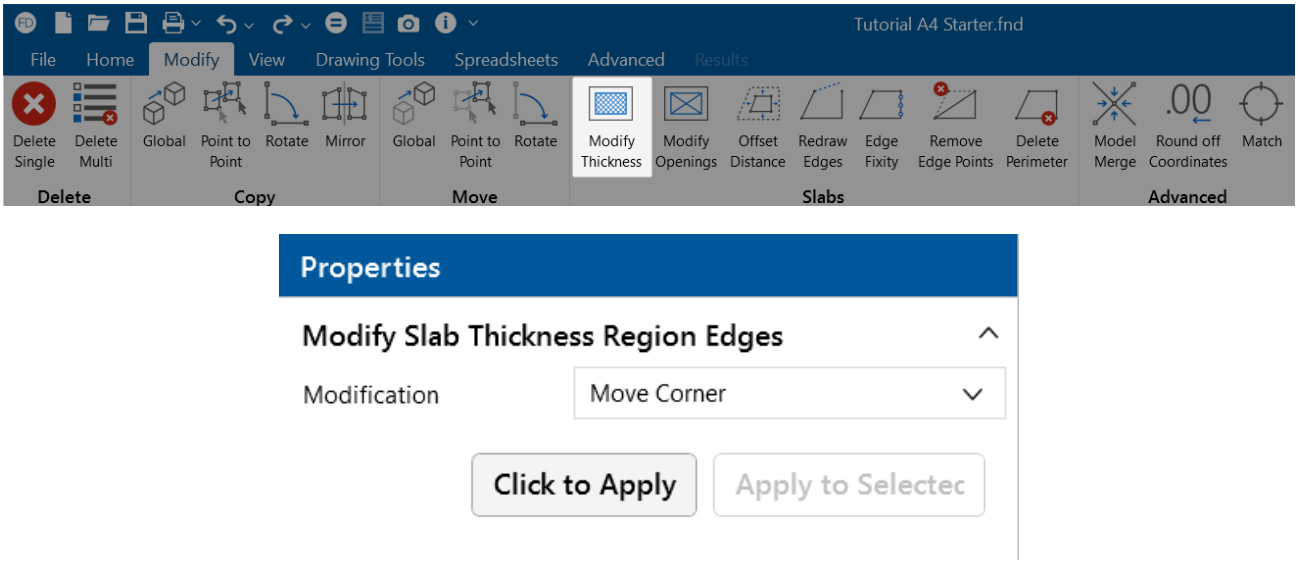
1. If there is not a model view already open, go to the **View** ribbon.
2. Click the **Open 3D Views** icon in the **Window** section to open a new view.
3. Go to the **Home** ribbon.
4. Click the **Thickened Regions** icon in the **Slab Elements** section.
5. In the **Properties** panel, set the slab region thickness and choose a drawing option for creating the region.



6. Start drawing a polygon to define a slab region. If drawing using the Node to Node option, double-click on the last node to indicate that you have finished drawing the region.

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

Modify a Thickened Region



To modify Slab Thickness Region Edges:

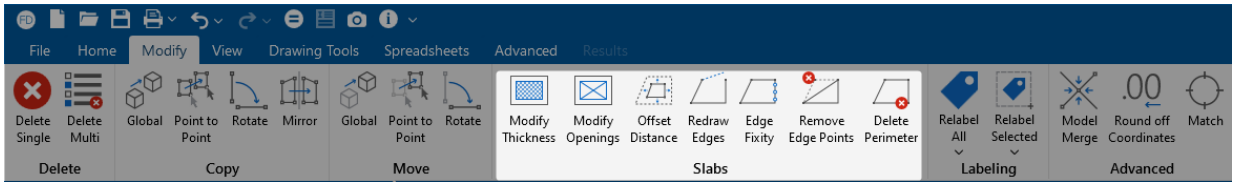
1. Go to the **Modify** ribbon.
2. Click the **Modify Thickness** icon in the **Slabs** section.
3. In the **Properties** panel, select the **Modification** drop-down and choose between **Move Corner**, **Merge Points** and **Delete the Entire Perimeter**.
4. Click on the corners of the thickened region you wish to modify in the 3D View window, or click anywhere along the thickened region perimeter if you selected the Delete Entire Perimeter Option.

Note: The edges of the slab cannot be modified. Therefore, in order to create a thickened edge, you'll want to use the thicker dimension on the entire slab and use this tool on the interior of the slab to reduce the thickness.

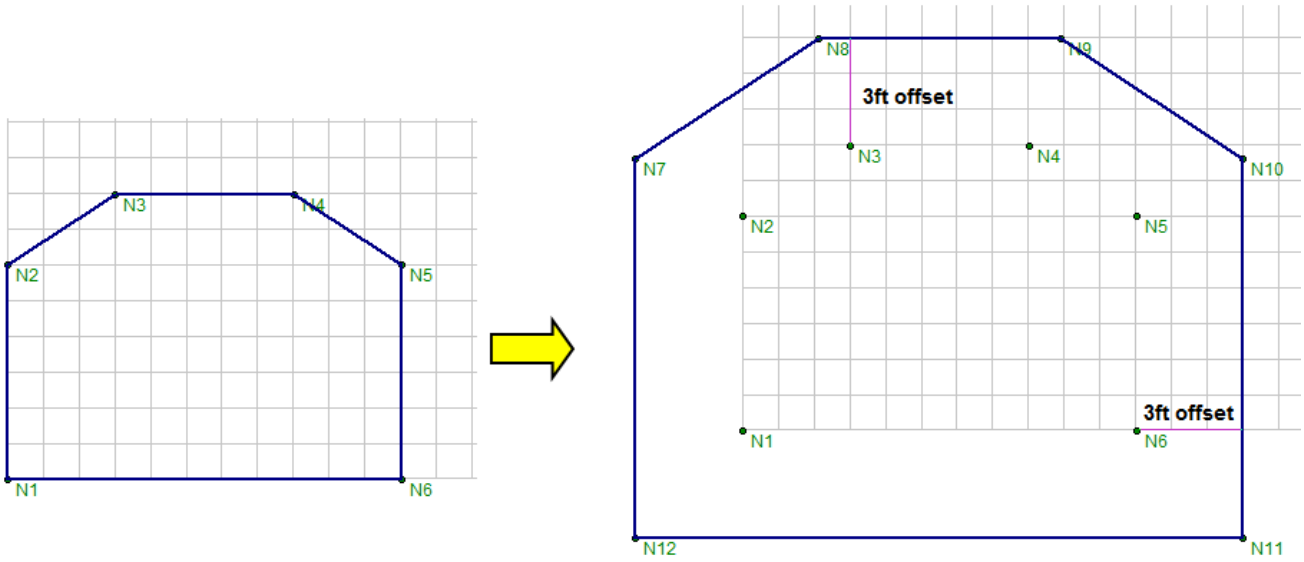


Modify Edges

There are several options available for modifying your overall slab geometry in the Slabs section of the **Modify** Ribbon.

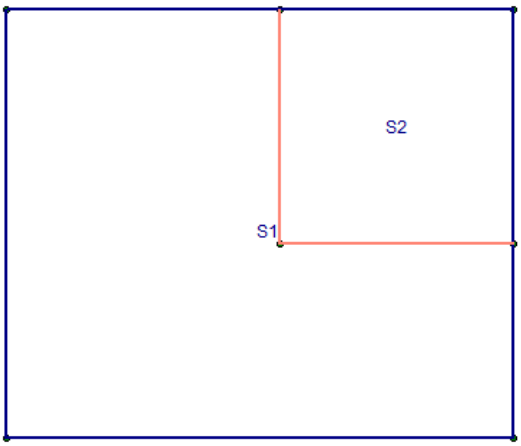


Offset Distance - Allows you to increase or decrease your slab edges by a fixed distance. Enter a positive number to enlarge the slab and a negative number to shrink the slab footprint. For example, if you indicate a 3 ft offset distance, the slab edges will be offset linearly by 3 ft on all sides.



Redraw Edges-Allows you to redefine the locations of individual slab edges.

Edge Fixity - Adjusts the slab edge fixity to an adjacent slab element. This is useful for slabs that are not monolithically placed, where a shear key or dowels are used to transfer forces out-of-plane to the slab. By default, slab edges are fixed to each other when two adjacent slabs touch each other. Pinned slab edges are graphically indicated with a salmon color, as opposed to the typical blue slab edge. In the example below slab S2 is pinned along both edges that it shares with slab S1:

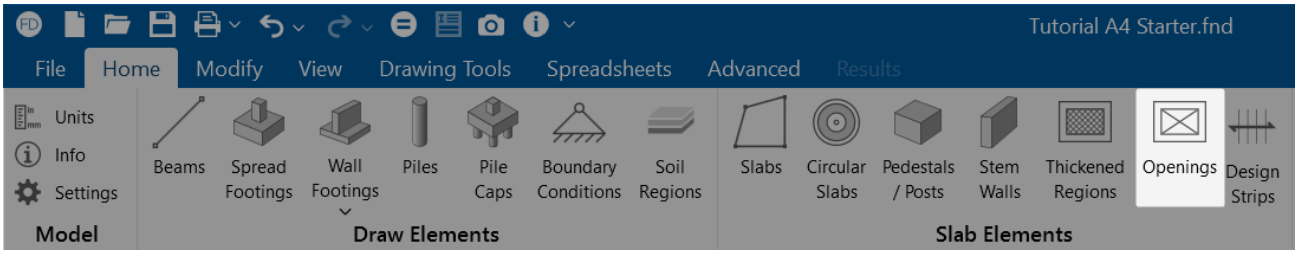


Remove Edge Points - Deletes the corner of a slab and connects the two adjacent points at the slab perimeter.

Delete Perimeter - Allows you to deletes the entire slab perimeter from the model.

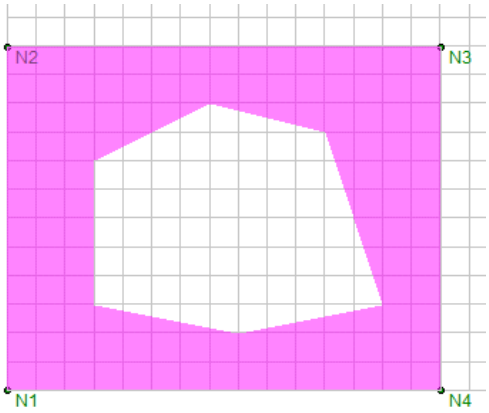
Create Slab Openings

The Slab Openings tool can be used to create an opening in an existing slab in your model.

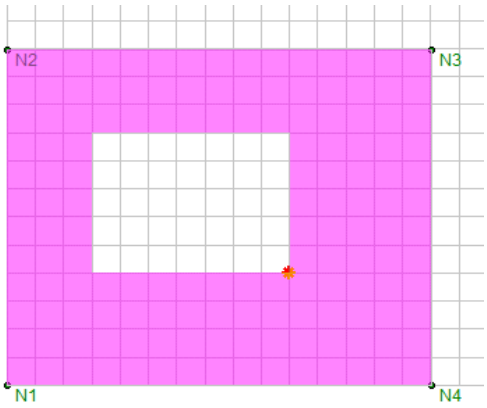


To create Slab Openings:

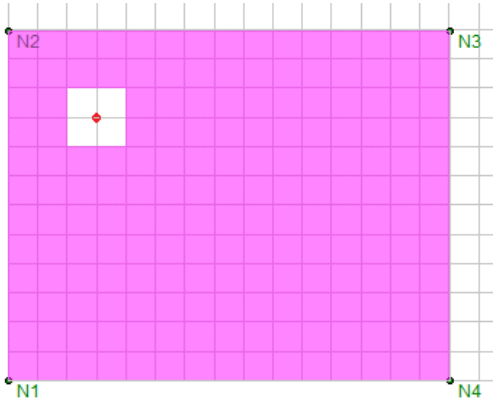
1. Go to the **Home** ribbon.
2. Click the **Openings** icon in the **Slab Elements** section.
3. Select the **Draw Option** in the **Properties** panel from the following:
    - **Node to Node** - Click individual nodes to define an opening of any shape. double-click on the last node of the polygon to indicate that you have finished drawing



    - **Rectangular Node to Node** - Create a rectangular opening by clicking to define opposite corners of the opening.



- **Rectangular Center Node** - Enter the overall dimensions for the length and width of a rectangular opening and then click to define the center of the opening. The image below shows the center point (red dot) with Z-axis dimension of 24 in and X-axis dimension of 24 in.

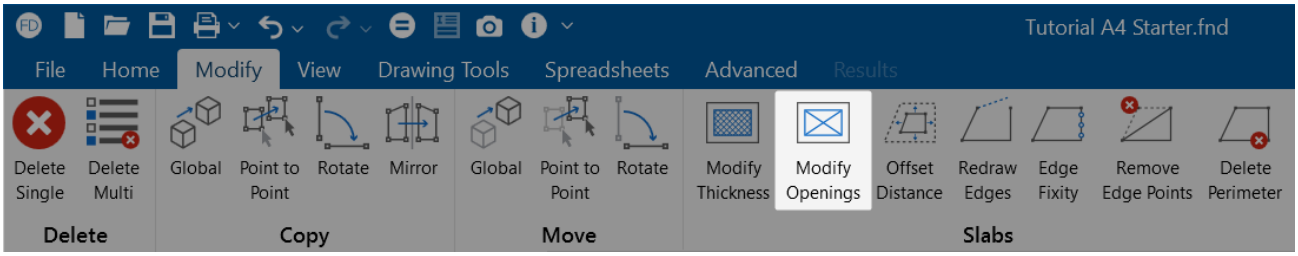


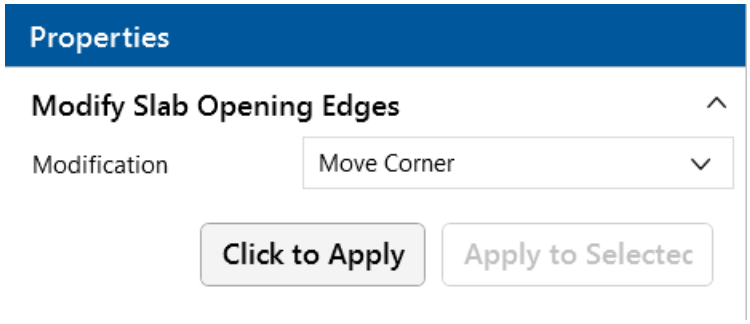
4. Draw the opening based on the selected draw option. If Node to Node is selected, your opening.

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

Note:
- Slab openings can be drawn in contact with the edge of the slab.
- Line loads and area loads will not be considered in the analysis of the slab if they are within the perimeter of an opening.

Modify Slab Opening Edges



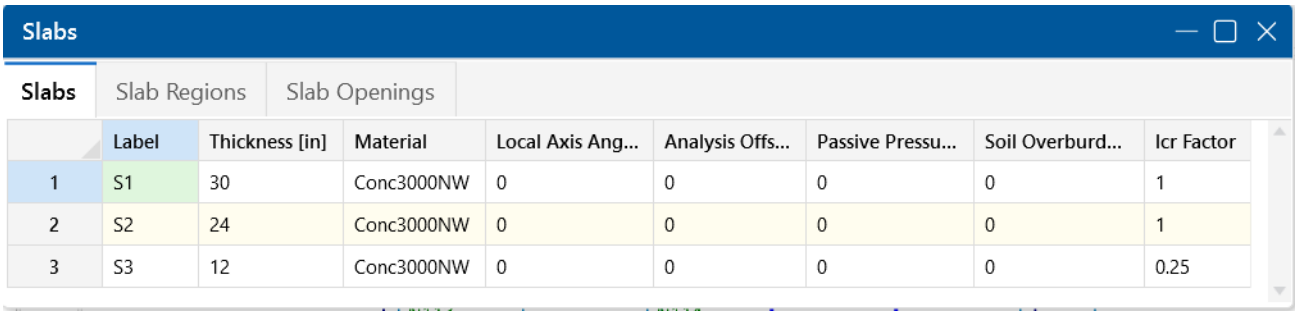


To modify Slab Opening Edges:

1. Go to the **Modify** ribbon.
2. Click the **Modify Openings** icon in the **Slabs** section.
3. In the **Properties** panel, select the **Modification** drop-down and choose between **Move Corner**, **Merge Points** and **Delete the Entire Perimeter**.
4. Click on the corners of the slab opening you wish to modify in the 3D window, or click anywhere along the opening perimeter if you selected the Delete Entire Perimeter Option.

Slabs Spreadsheet

The **Slabs Spreadsheet** records the properties of each slab and may be accessed by selecting **Slabs** from the **Data Entry** drop-down on the **Spreadsheets** tab of the ribbon, or by clicking **Slabs** from the **Data Entry** section of the **Explorer** panel.



	Label	Thickness [in]	Material	Local Axis Ang...	Analysis Offs...	Passive Pressu...	Soil Overburd...	Icr Factor
1	S1	30	Conc3000NW	0	0	0	0	1
2	S2	24	Conc3000NW	0	0	0	0	1
3	S3	12	Conc3000NW	0	0	0	0	0.25

The following data columns hold the primary data for the slabs:

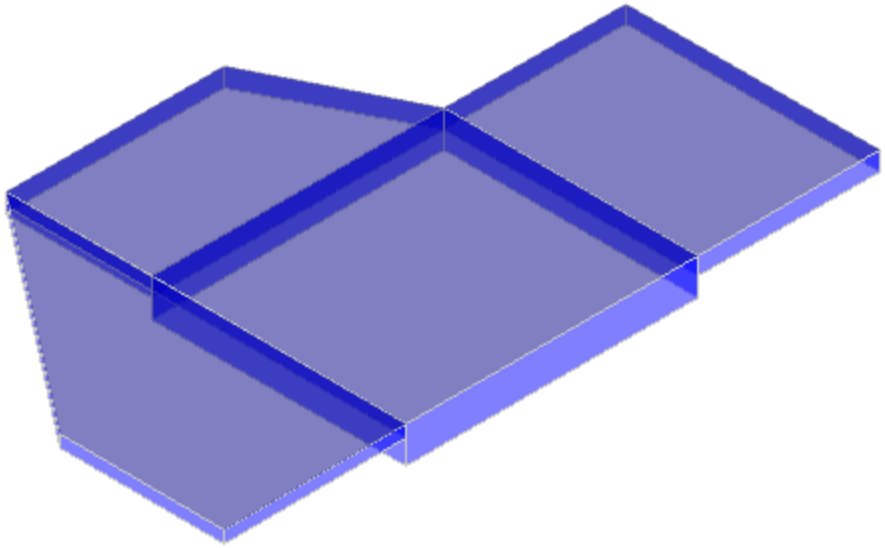
Label

You may assign a unique label to each slab. Each label has to be unique, so if you try to enter the same label more than once, you will get an error message.

Thickness

This entry designates the thickness of the slab.

Note: Slabs with varying thickness may be created by defining adjoining slabs that share a border but have a different thickness, as shown in the image below. The thickened areas underneath each column are modeled as separate slabs that share edges with the main slab.

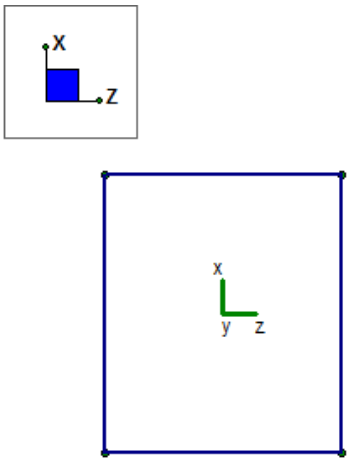


Material

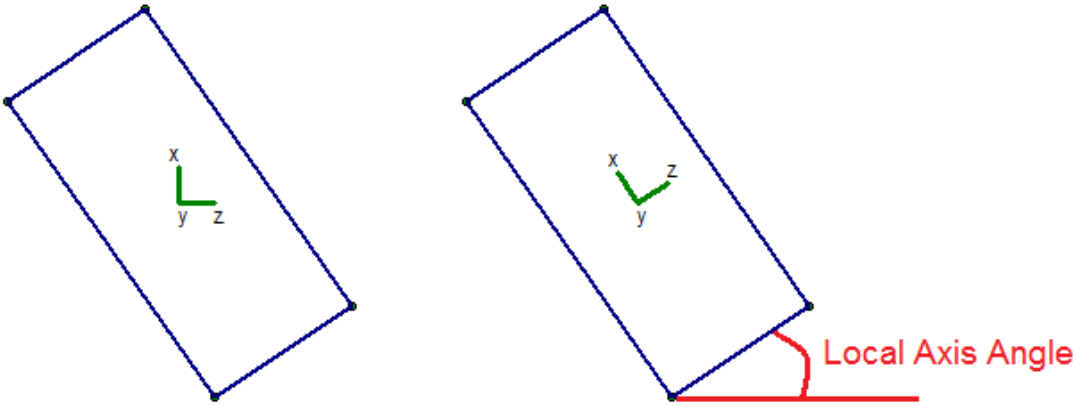
This entry defines the concrete material that will be used for the properties of the slab. You can select the ... (ellipsis) button to open the **Review or Change Material** dialog, where you can modify the element's physical properties.

Local Axis Angle

This angle defines the orientation for sliding and overturning checks for slabs. By default, this value is set to 0 and will provide sliding and overturning checks about the global axes.



However, if you have a slab that is skewed from the global axes, then you may wish to have the overturning and sliding checks taken about those different axes.



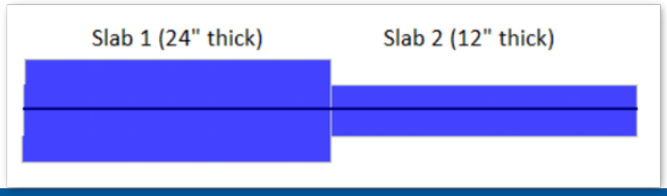
The results for sliding and overturning for slabs are then based on these local axes coordinates. For example, Mo-xx is the demand overturning moment about a slab's local x axis.

Note:
- The Local Axis Angle must be between 0 and 90.
- For output information, see Slabs - Sliding & Overturning.

Analysis Offset

This is a positive value that will move the slab downward in relation to any other element in the model. This not just a graphics change. The program is internally adjusting the location of the slab and connecting it to surrounding elements with rigid links. You can expect to see different distribution of forces due to the offset.

Example 1: This is an example of two slabs with no Analysis offset. The slabs are connected at their centerlines.

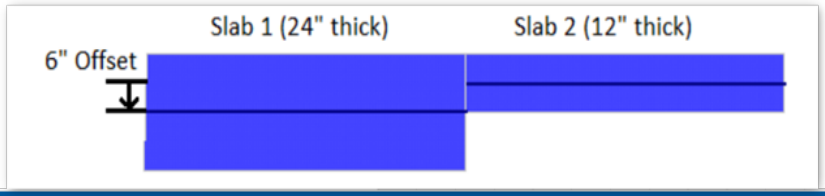


Slabs

Slabs | Slab Regions | Slab Openings

	Label	Thickness [in]	Material	Local Axis Angle [deg]	Analysis Offset [in]	Passive Pressure [ksf]	Soil Overburden [ksf]	Icr Factor
1	S1	24	Conc3000NW	0	0	0	0	0.25
2	S2	12	Conc3000NW	0	0	0	0	0.25

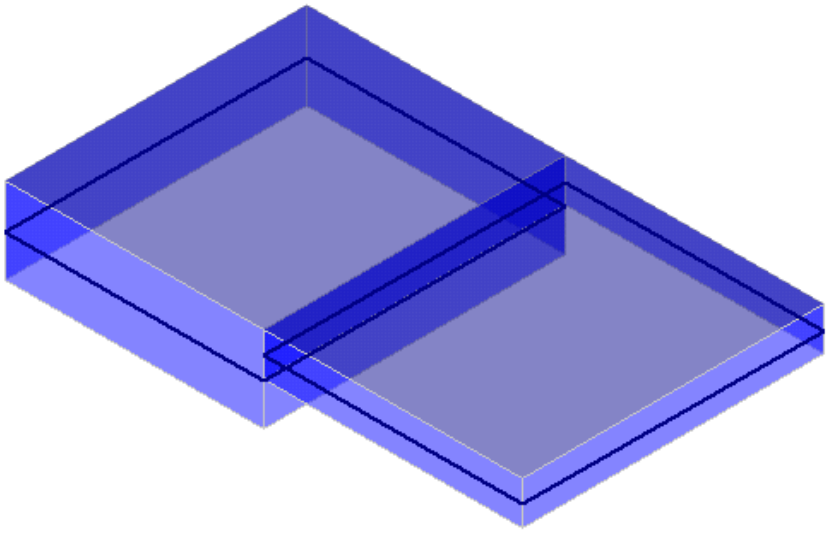
Example 2: This is an example of the slab offset in order to align the tops of each slab.



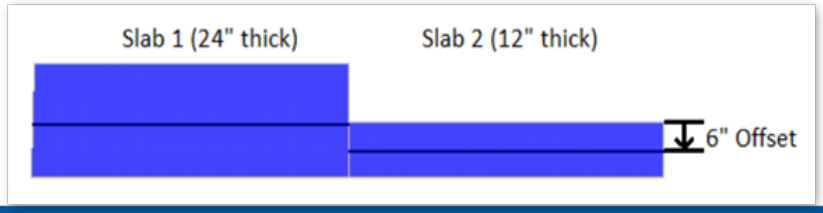
Slabs

Slabs | Slab Regions | Slab Openings

	Label	Thickness [in]	Material	Local Axis Angle [deg]	Analysis Offset [in]	Passive Pressure [ksf]	Soil Overburden [ksf]	Icr Factor
1	S1	24	Conc3000NW	0	6	0	0	0.25
2	S2	12	Conc3000NW	0	0	0	0	0.25



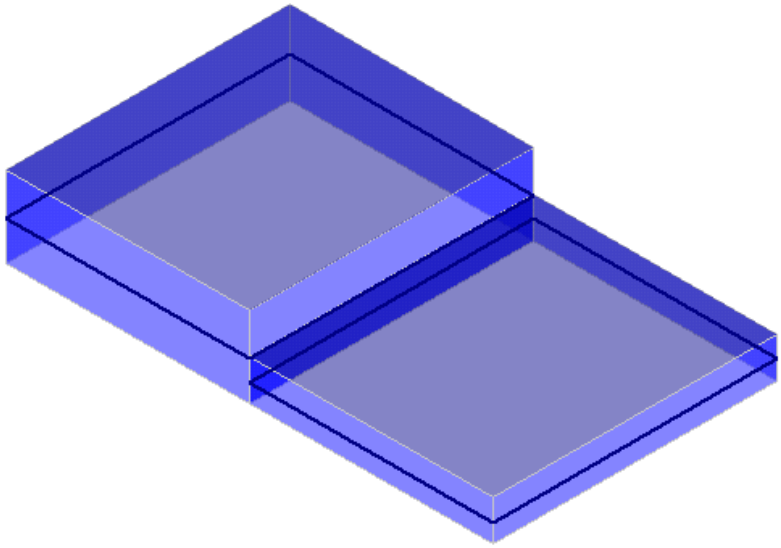
Example 3: This is an example of the slab offset in order to align the bottom of each slab.



Slabs

Slabs | Slab Regions | Slab Openings

	Label	Thickness [in]	Material	Local Axis Angle [deg]	Analysis Offset [in]	Passive Pressure [ksf]	Soil Overburden [ksf]	Icr Factor
1	S1	24	Conc3000NW	0	0	0	0	0.25
2	S2	12	Conc3000NW	0	6	0	0	0.25



Passive Pressure

The **Passive Pressure** represents a soil pressure used to calculate sliding resistance. The passive pressure is multiplied by the vertical area of the slab in the slab's local z and local x directions. This total force is added to the sliding resistance from friction in the **Slab Stability Results Spreadsheet**.

Note:

- The passive resistance only applies to sliding resistance and has no effect on overturning.
- The passive resistance applies to each individual slab and the presence of adjacent slabs has no effect.
- Thickened regions are not considered when computing the force from passive pressure.

Soil Overburden

The **Soil Overburden** represents a soil pressure on top of the slab which is also considered to calculate resistance to overturning and sliding. The soil overburden pressure is multiplied by the area of the slab and is scaled to reduce the magnitude for the areas of pedestals, stem walls, columns and post. This total force is added to the overturning resistance in the **Slab Stability Results Spreadsheet**.

Note: The soil overburden magnitude is scaled to account for a reduction from pedestals and stem walls; however, the pressure is applied uniformly over the slab. This is because the slab plate mesh constrains to only a pedestal center node and stem wall center line and not the perimeter. This assumption is typically valid when the areas of the pedestals and stem walls are small relative to the entire area of the mat slab. Although, if the pedestals or stem walls are relatively large when compared to the slab area, this assumption may be unconservative and the soil burden load should be applied manually.

Icr Factor (Cracked Moment of Inertia Factor)

The **Icr Factor** is used to reduce the bending stiffness of slabs. If this entry is left blank, the Icr factor will default to a value of 0.25.

Note: The **Icr Factor** will be ignored if the **Use Cracked Sections** box is not checked on the **Concrete** tab of the (Global) Model Settings dialog.

Service Level Stiffness

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

For typical applications, ACI 318-14 Section 6.6.3.1 (ACI 318-11 Section 10.10.4) requires that member stiffness be reduced to account for the cracking that occurs when a member is subjected to ultimate level loads. As described in the previous section, RISA uses the **Icr Factor** to account for this stiffness reduction. However, for service level analysis, the level of cracking will be significantly less. Therefore, the stiffness used in your analysis should be representative of the reduced loading and reduced cracking. Per ACI 318-14 Section R6.6.3.1.1 (ACI 318-11 Section R10.10.4.1), the program 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"** flag checked in the **Load Combinations Spreadsheet.**

Automatic Slab Meshing

RISAFoundation uses a Finite Element based approach to solving a Slab foundation model. As such, each slab will be automatically sub-meshed into a series of plate elements. This plate element mesh is very important for determining the quality of the structural model.

The **Mesh Size**, set on the **Solution** tab of the Model Settings, is the most important entry for controlling the size of your model. This is the starting size of plate element grid. This entry can be adjusted larger or smaller depending on the size of your slab. Generally speaking, a very large mat would need a larger mesh size, whereas a smaller mat would require a smaller mesh size.

Control Points

In addition to the mesh size, any points on the boundary of the slab and any support points within the slab will be considered **control points**. These control points are "fixed" points within the mesh and will dictate the layout of individual plates surrounding them. Any control points that are closer than the mesh distance specified in the Model Settings will cause an error message to occur during solution.

Note:

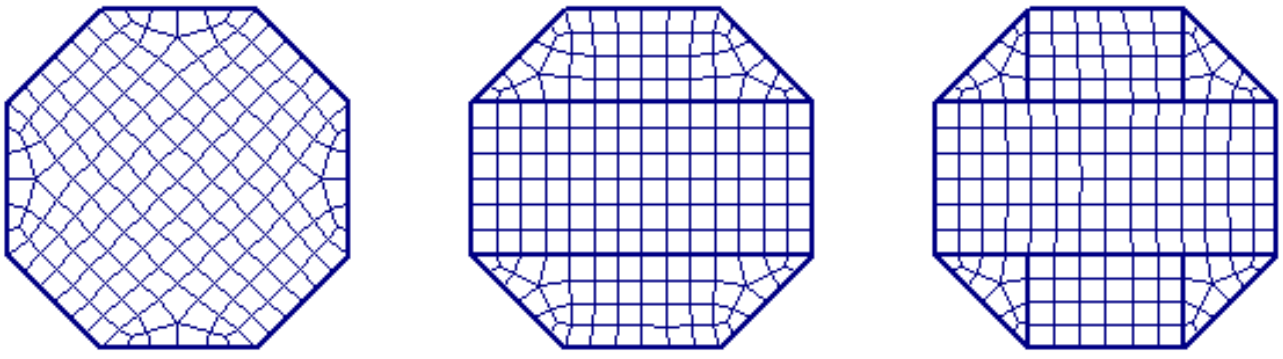
- Points which are used to apply loads are NOT automatically considered part of the plate mesh. These loads will be attributed to the surrounding nodes based on the local element stiffness and the location of the applied loading.
- Points which are not automatically meshed into the slab will have their soil pressure and deflection reported as a linearly interpolated value between the nearest meshed slab plate corner values.

Meshing Adjacent Slabs

When two slabs share a common border, each slab will initially be meshed individually. However, the mesh will then be modified so that all plate nodes line up at the slab borders. This is done to ensure perfect force transfer across the slab connection.

Modeling Complex Slabs

The automatic meshing routine can produce odd meshes for complex geometries. Therefore, it can be beneficial to model the slab as multiple adjacent slabs instead. This will force a more rectangular meshing pattern at borders between slabs. The image below shows what kind of an effect modeling an octagon foundation out of multiple slabs can have on the final plate mesh.



It should be stressed that ALL of the meshes shown above are good meshes that will produce accurate results. However, the image clearly shows how modeling adjacent slabs can force a more uniform and symmetric mesh.

Limitations

Foundation slabs do not consider any long term deflections effects from creep.

Slabs - Design Cuts

Design cuts are used to look at areas of interest or concern within a slab, or to investigate specific critical locations that may be difficult to define with a design strip. They can also be used to double-check design strip results, as design strip results typically show the envelope of the worst case moments in the whole strip. A design cut is defined by drawing the cut plane from a start point to an end point. The total moment about the cut is computed from the underlying finite element results. This moment is then used to calculate the required area of steel and the number of bars required to satisfy that area of steel. The bars used to resist the moment will be perpendicular to the design cut. Care should be taken when drawing a design cut to only include the area of interest.

Create Design Cuts

Design cuts may be defined either before or after a solution is performed.

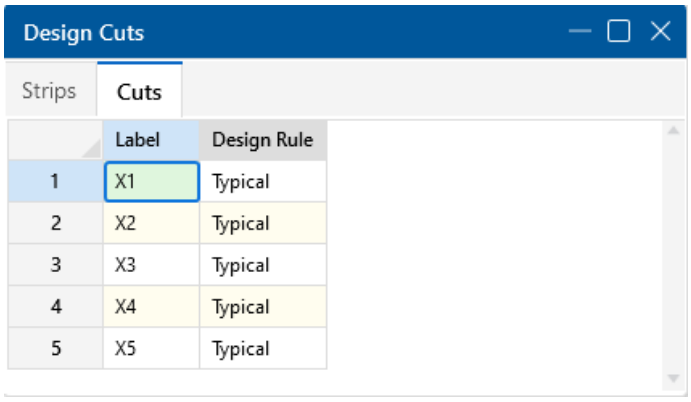
To create a Design Cut:

1. Select the Design Cut tool from the context menu by right-clicking within a 3D Model View.
2. Click two points to define the start and end of your design cut.
3. A design cut will be shown on the screen as a green dashed line, and you will see a new Design Cut definition show up in the Design Cut spreadsheet.

Note: A partial arrow is also shown in the Design Cut to indicate the bar direction (which is always perpendicular to the cut).

Design Cuts Spreadsheet

As you draw design cuts, they are automatically entered into the **Design Cut Spreadsheet**, which can be accessed by selecting **Design Strips/Cuts** from the **Data Entry** drop-down on the **Spreadsheets** tab of the ribbon, or by clicking **Design Strips/Cuts** from the **Data Entry** section of the **Explorer** panel.



You can modify the **Label** and the **Design Rule** used to compute the reinforcing bars. You cannot modify the Start and End points of a design cut. If you need to move a design cut, you will need to delete it and redraw it in the new location.

Label

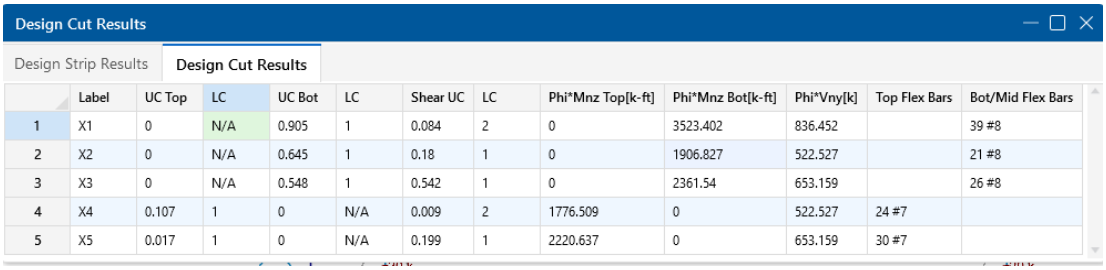
You may assign a unique label to each design cut. Entering a duplicate label will result in an error message.

Design Rule

The Design Rule used for each design cut defines the parameters used to calculate the required area of steel and the reinforcing bars. For more information on Design Rules, see Design Optimization.

Design Cut Results Spreadsheet

The summary results for all your Design Cuts can be seen in the **Design Cut Results Spreadsheet**. To view the design cuts results spreadsheet, click **Results Browser** from the **Results** toolbar and select **Design Strip/Cut Results**. Or, click **Design Strip/Cut Results** in the **Results** section of the **Explorer** panel.



	Label	UC Top	LC	UC Bot	LC	Shear UC	LC	Phi*Mnz Top[k-ft]	Phi*Mnz Bot[k-ft]	Phi*Vny[k]	Top Flex Bars	Bot/Mid Flex Bars
1	X1	0	N/A	0.905	1	0.084	2	0	3523.402	836.452		39 #8
2	X2	0	N/A	0.645	1	0.18	1	0	1906.827	522.527		21 #8
3	X3	0	N/A	0.548	1	0.542	1	0	2361.54	653.159		26 #8
4	X4	0.107	1	0	N/A	0.009	2	1776.509	0	522.527	24 #7	
5	X5	0.017	1	0	N/A	0.199	1	2220.637	0	653.159	30 #7	

The spreadsheet results let you quickly see the Unity Checks and Shear Checks along with the moment capacity and number of bars required in the top and bottom of the slab for your design cuts.

Design Cut Detail Report

Additional results information can be obtained by viewing the **Detail Report** for a design cut. To access this report, select **Detailed Report** from the **Results** toolbar and then click the design cut you want to review. The Detail Report gives additional information about the design cut properties and provides many of the supplementary values to help check the results. Information in the Detail Report will vary depending on whether the design cut was auto-generated for a design strip or was a custom design cut created by the user. The information in a user created design cut will be limited. Below, we will walk through each section of the report for a design cut generated for a design strip.

Header



The header of the Detail Report is useful for navigating and viewing information in the design cut reports.

- The **arrows** allow you to click through detail reports for the different design cuts in the model.

Input Data

Input Data

Cut:	DS1-X1	**Max Top bar Spac. (in):**	18	**Stress Block:**	Parabolic
Material:	Conc3000NW	**Min Top bar Spac. (in):**	3	**Slab Design Option:**	Construction
Start (ft):	(4.02 , 82.18)	**Max Bot bar Spac. (in):**	18	**Rebar Spacing Inc (in):**	2
End (ft):	(4.02 , 83.82)	**Min Bot bar Spac. (in):**	3	**Design Rule:**	Typical
Tension Bar Fy (ksi):	60	**Concrete Weight (k/ft^3):**	0.145	**Top Cover (in):**	1.5 in
Shear Bar Fy (ksi):	60	**λ:**	1	**Bottom Cover (in):**	3
F'c (ksi):	3	**$E_{Concrete}$ (ksi):**	3156	**Side Cover (in):**	0
Flex. Rebar Set:	ASTM A615				

The first section of the Detail Report merely echoes back basic input data, such as the starting and ending locations of the design cut, the basic Design Rule parameter, the type of stress block used and the material type for the slab.

Code Check Data

ACI 318-14 Code Check

Top Bending Check:	0.000	**Bot Bending Check:**	0.000	**1 Way Shear Check:**	0.002
Gov Mu Top (k-ft):	0	**Gov Mu Bot (k-ft):**	-0.045	**Gov Vu (k):**	0.064
phi*Mn Top (k-ft):	162.994	**phi*Mn Bot (k-ft):**	180.65	**phi*Vn (k):**	42.655
Governing LC:	N/A	**Governing LC:**	2	**Governing LC:**	2
Top As Reqd:		**Bot As Reqd:**			
Analysis (in^2):	NA	**Analysis (in^2):**	0		
4/3 (in^2):	NA	**4/3 (in^2):**	NA	**As Reqd(T/S) (in^2):**	1.05796
Min Flex (in^2):	NA	**Min Flex (in^2):**	1.058	**Rho Reqd(T/S):**	0.00180
Top As Prvd (in^2):	1.325	**Bot As Prvd (in^2):**	1.571	**Rho Prvd(Gross):**	0.00493
	(3 #6)		(2 #8)		

The information in this section is intended to give the basic data used in the code checking of the concrete section. This includes the demand moment (Mu) and capacity (Mn) when the top or bottom bars are in tension. This section also reports the one-way shear demand (Vu) and capacity (Vn).

Note: The program only performs a one-way shear checks on the design strip. The two way/punching shear code check is discussed in Pedestals - Punching Shear.

Area of Steel Required

The program calculates the area steel required based on the steps listed below.

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 8.6.1 (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, and Eurocode 2 do not use the 4/3 As for slabs, the results will show NA.

Reinforcement Design Routine

Since 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 an equivalent number of bars if **Theoretical** Method is selected in the **Rebar** tab in Model Setting) 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. Report As from Step 1.

Once this is complete, we have a rebar spacing and As for each cut.

Then report the smallest value of Spacing from all of the design cuts for each design strip.

Single Layer Rebar Design

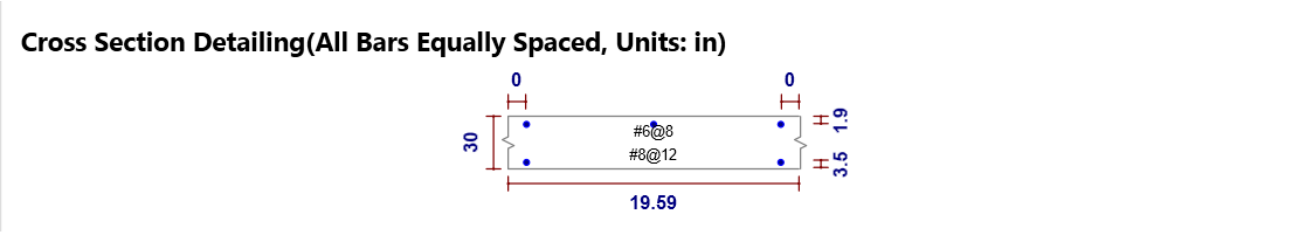
The single layer rebar option allows the user to specify one layer of rebar at any depth in the slab. The location of the rebar in the slab is controlled by the top cover input in the Design Rules. This option especially benefits the design of thinner and lightly loaded slabs on grade where two layers of reinforcement may be too complex and/ or unnecessary. The single layer rebar uses the design routine above to determine how much steel is required, with the addition of the following assumptions:

1. If the single layer is placed at or above mid-depth, the one layer of rebar is assumed to resist both the positive and negative bending.
2. If the single layer is placed below mid-depth, the layer of rebar is only assumed to resist the bottom moment. Any top moment will be compared against the plain concrete capacity according to ACI 318-14 Equation 14.5.2.1a (ACI 318-11 equation 22-2).

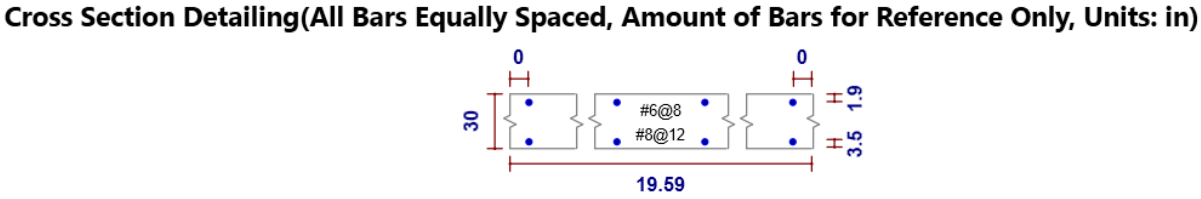
Cross Section Detailing

This section provides a visual verification of the bar arrangement, slab thickness and cover. Depending on the **Mat Slab Design Option** selected in the **Concrete** tab of Model Settings, 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:



Slabs - Design Strips

Design Strips are used to create regions within a slab where design cuts will automatically be created and flexural reinforcing bars will be designed. If the moment varies along the length of a

design strip, then the individual design cuts will show a varying amount of moment demand at each section. The results for the entire design strip will be governed by the maximum moment demand at an individual design cut. The same number and size of reinforcing bars will be used for the entire design strip, so this should be considered when defining the boundary for a particular strip. 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.

The moment demand at an individual design cut is defined as the total moment over the width of the design cut. No attempt is made to design for local stress or force risers within the width of the design cut. Therefore, it is critical that good engineering judgment is used to determine an appropriate width for setting the design strips/design cuts. You can also see a Detail Report for each of these design cuts.

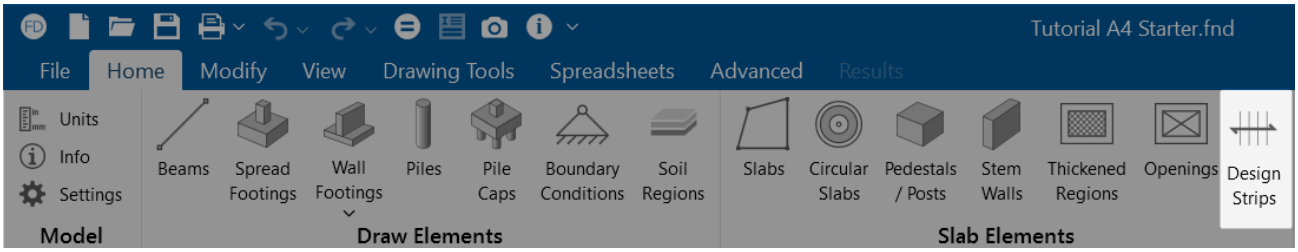
Slabs may be analyzed and designed using traditional reinforcement bars or welded deformed wire reinforcement (WWR). WWR can be enabled by going to the Global Model Settings - Rebar and checking the “Include Welded Wire Reinf.” option.

Welded Wire Reinforcement Limitations:

- WWR design is only available for Mat Slab, Footing and Pile Caps.
- Only ASTM A1064 deformed bars sizes are available.
- Development length is conservatively calculated in a similar fashion to traditional reinforcement bars. The Ψw factor is conservatively ignored, which reduces the development length due to contribution of “welded intersections”.

Create Design Strips

You can draw design strips using a Drawing Grid, Project Grid or by clicking on existing nodes. You can set your design strip properties up front, or you can modify these properties after drawing a design strip.



To create a Design Strip:

1. If there is not a model view already open, go to the **View** ribbon.
2. Click the **Open 3D Views** icon in the **Window** section to open a new view.
3. Go to the **Home** ribbon.
4. Click the **Design Strips** icon in the **Slab Elements** section.

5. In the **Properties** panel, set the design strip properties. You can modify the direction of the reinforcing bars within the strip. You may choose to increase or decrease the number of design cuts that will be created automatically within the design strip, and you can choose which Design Rule to apply to all the design cuts within the strip. The Design Rule sets the various parameters needed to calculate the required area of steel and the reinforcing bar size and number.
6. Start clicking nodes to define the boundary of your design strip. To indicate that you have finished drawing the region, double-click on the last node.

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

A design strip will be shown graphically with a red boundary line. Design cuts that are automatically created will be shown as orange dashed lines (as opposed to green if you draw a design cut manually).

Setting the Width of Design Strips

One of the most important design considerations is how wide to set the design strip. If the width is set too large, the program will average out the moments and shears over too wide of a region. This would result in unconservative design moments and shears. Similarly, if the strip is set too small, the effects of stress risers in the FEM analysis will be over estimated and the design will be overly conservative.

The setting of the design strip width is truly a matter of engineering judgment. RISA Tech, Inc. makes no endorsement on which methods would be most appropriate.

ACI Definition of Strips (ACI 318-14 Section 8.4.1/ACI 318-11 Section 13.2)

This section of the ACI code is really intended for elevated slabs. But, the concepts can be extended into mat foundations as well. The requirement for "column strips" is that the width on each side should be set to 25% of the span length or width, whichever is smaller. Then the "middle strip" is defined to span between the edges of the column strips.

This method requires engineering judgment for column grids that are not perfectly aligned and rectangular. In addition, when the column strip becomes very small then the middle strip may become very wide so that the entire slab is included in either a column strip or a middle strip.

The ACI strip method listed above is based on essentially 1/2 of the mid-span tributary lines. The hand calculation methods would have you design for the full tributary moments over this smaller width, which should be conservative. Computer methods (like RISAFoundation) will design for the average moment over the assumed design width, which should result in a more efficient design.

Zero Shear Transfer Method

The Zero Shear Transfer method uses the shear force contours perpendicular to the span of the slab to set the design width. This should provide a result very similar to using the mid-span tributary lines, but is a bit more theoretically derived for non-rectangular column layouts. This method is described in greater detail in the PTI publication Design Fundamentals of Post-Tensioned Concrete Floors. Ideally, this method should give design strips of similar width to the ACI strip method. However, it is

more rationally derived and should work better for cases where uneven column spacing makes the strip method difficult to apply.

Zero Moment Method

In a similar fashion to the zero shear transfer method, the Zero Moment method uses the moment contours to identify where the moment changes sign. This can be used to set the design strip width approximately equal to the distance between zero moments.

Shear Perimeter Method

Another basis would be to set the design width equal to the pedestal width plus a distance 'd' or 'd/2' on each side. This will end up being a more conservative assumption for flexure than the other methods listed. As such, it would be more appropriate for situations where shear or punching shear failures are a primary concern. Examples would also include cases where the pedestal is very large, such as for a vertical vessel or grain silo. This is similar, though not identical, to a method given in the NEHRP document GCR 12-917-22 (Seismic Design of Reinforced Concrete Mat Foundations).

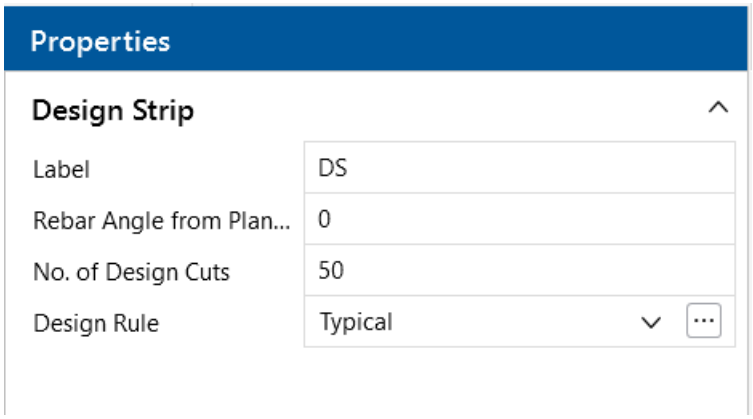
Hybrid Method / Engineering Judgment

A variation on these methods would be to start off setting the column strip using the ACI strip method. Then, if necessary, the width could be modified based on considering the other methods. This is especially true for situations where the column grid is not aligned or rectangular.

In addition, when the middle strip widths get too large, they could be set to values closer to the column strip width. The middle strip would normally be centered on the area with the highest mid-span moments. This would neglect lower moment regions between the column and middle strips. Hence, the strips would designed for a higher moment per unit width. This reinforcement could then be extended into the lower moment regions between strips. Or, the user could set up another design strip for these lower moment regions.

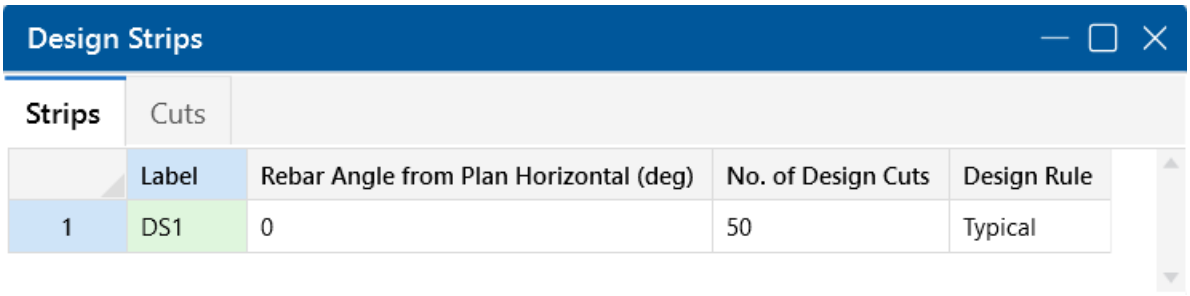
Modify Design Strips

There are a number of ways to modify design strips. You can view and edit the design strip data in the **Design Strips/Cuts Spreadsheet**, which can be accessed by selecting **Design Strips/Cuts** from the **Data Entry** drop-down on the **Spreadsheets** tab of the ribbon, or by clicking **Design Strips/Cuts** from the **Data Entry** section of the **Explorer** panel. You may also select a design strip (or multiple design strips) to view and edit the properties in the **Properties** panel. Here, you can modify the direction of the reinforcing bars within the strip, the number of cuts or the Design Rules associated with the strip. Clicking the ··· (ellipses) button within the Design Rules input of the **Properties** panel or **Design Strips/Cuts Spreadsheet** will open the **Review or Change Design Rules** dialog, where you can modify the element's Design Rules.



Design Strips Spreadsheet

As you create design strips, they are automatically entered into the **Design Strips/Cuts Spreadsheet**.



As shown in the figure, you can modify the Label, the angle of the reinforcing bars, the number of automatically generated design cuts and the Design Rule used compute the reinforcing bars. You cannot modify the design strip boundary. If you need to modify the boundary of a design strip, you'll need to delete it and redraw it with the new modification.

Label

The labels can be shown on the model view and it's useful to change them to something more descriptive.

Rebar Angle from Plan Horizontal (deg)

The rebar angle lets you adjust the angle of the reinforcement to handle curved and other types of custom layouts.

No. of Design Cuts

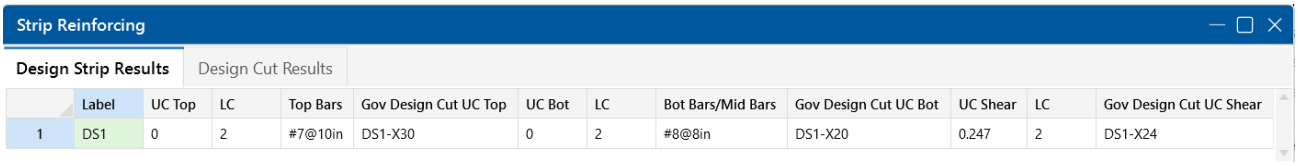
The number of design cuts can be increased for larger design strips to make sure your capturing the representative loads within the design strip.

Design Rule

The Design Rule is used for each design cut to define the parameters used to calculate the required area of steel and the reinforcing bars (the same way Design Rules are used for beam design).

Design Strip Results

Results for your Design Strips can be found in the **Design Strip/Cut Results Spreadsheet**, which may be accessed by clicking **Results Browser** from the **Results** toolbar and selecting **Design Strip/Cut Results**. Or, click **Design Strip/Cut Results** from the **Results** section of the **Explorer** panel.



	Label	UC Top	LC	Top Bars	Gov Design Cut UC Top	UC Bot	LC	Bot Bars/Mid Bars	Gov Design Cut UC Bot	UC Shear	LC	Gov Design Cut UC Shear
1	DS1	0	2	#7@10in	DS1-X30	0	2	#8@8in	DS1-X20	0.247	2	DS1-X24

This spreadsheet lets you quickly see required top and bottom reinforcing bars for each design strip and the code check produced for that strip. If welded wire bars are designed, the output result will be preceded by a “D” instead of “#” to indicate wire bar size. Additionally, the governing design cut for the top and bottom reinforcing bars are shown so you can investigate the way the reinforcing bars were calculated. Also given is the shear code check and which design cut governed for that shear check. The results for each strip are the envelope of all the design cuts within the design strip. Top and bottom results are enveloped independently.

Design Strip Detail Report

To see more detailed information for the design strip results, select **Detailed Report** from the **Results** ribbon (when in a graphical view) and click on the design strip you want to review. You can also access the Detail Report for a selected design strip from the right-click menu when working in the spreadsheet. The design strip **Detail Report** gives additional information about the Design Cut properties, and also gives many of the supplementary values to help check the results. The Detail Report will always show the **Envelope** force diagrams. The report is split into sections which include: input data, diagrams and code checks. Below, we will walk through each section of the report.

Header



The header of the Detail Report is useful for navigating and viewing information in the design strip reports.

- The **arrow** buttons allow the user to quickly browse through numerous design strip Detail Reports.
- The **Cut** button will switch between the Detail Reports for individual design cuts and the Detail Reports for the overall design strip.

Input Data

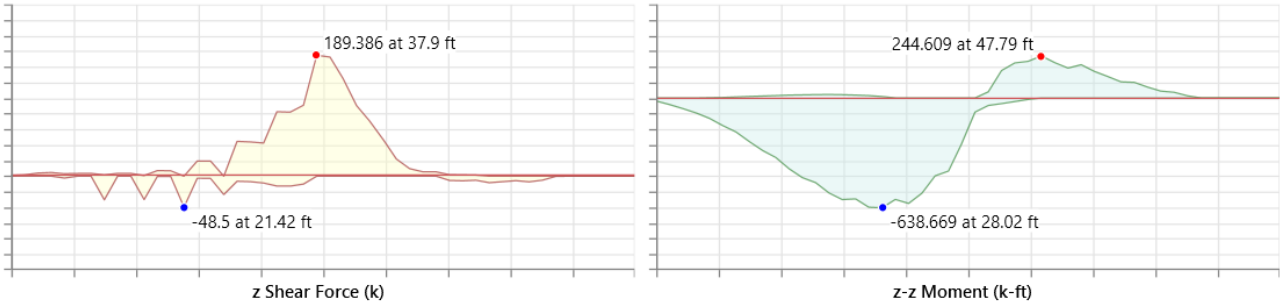
Input Data

Material:	Conc4000NW	**Max Top Bar Spac. (in):**	12	**Stress Block:**	Parabolic
Strip Width (in):	240	**Min Top Bar Spac. (in):**	10	**Rebar Orientation:**	0
Total Cuts:	50	**Max Bot Bar Spac. (in):**	12	**Rebar Spacing Inc (in):**	2
Tension Bar Fy (ksi):	60	**Min Bot Bar Spac. (in):**	8	**Design Rule:**	Typical
Shear Bar Fy (ksi):	60	**Concrete Weight (k/ft^3):**	0.145	**Top Cover (in):**	1.5
F'c (ksi):	4	**λ:**	1	**Bottom Cover (in):**	3
Flex. Rebar Set:	ASTM A615	**$E_{Concrete}$ (ksi):**	3644	**Side Cover (in):**	0
		Rho Bot Prvd:	0.00259	**Rho Top Prvd:**	0.00223
		Prvd Bot Bar Spac.:	#8@12in	**Prvd Top Bar Spac.:**	#7@10in

The first section of the Detail Report merely echoes back basic input data, such as the strip width, number of cuts, basic Design Rule parameters, the type of stress block used and the material type for the slab.

Envelope Shear and Moment Diagrams

Diagrams



This section shows the shear and bending moment envelope diagrams for every cut along the length of the design strip.

Note: Since the width of a design strip can vary along its length, the controlling code check location (which will normally be based on the maximum force per unit width) does not necessarily correspond to maximum shear and moment locations reported in the diagrams.

Code Check Data

ACI 318-14 Code Check

Top Bending Check:	0.015	**Bot Bending Check:**	0.344	**1 Way Shear Check:**	0.328
Gov Mu Top (k-ft):	27.573	**Gov Mu Bot (k-ft):**	-661.008	**Gov Vu (k):**	198.14
phi*Mn Top (k-ft):	1860.999	**phi*Mn Bot (k-ft):**	1921.828	**phi*Vn (k):**	603.363
Governing Cut:	DS1-X24	**Governing Cut:**	DS1-X35	**Governing Cut:**	DS1-X48
Governing LC:	1	**Governing LC:**	2	**Governing LC:**	2

Bending Steel Reqd/Prvd, Units: in^2

Cut Label	Top As Reqd	Top As Prvd	Bot As Reqd	Bot As Prvd	Rho Reqd(T/S)	Rho Reqd(Flex)	Rho Prvd(Gross)
DS1-X24	0.218	15.033	5.224	16.493	0.00180	0.00180	0.00438
DS1-X35	0.107	15.033	5.586	16.493	0.00180	0.00180	0.00438

The information in this section is intended to give the basic data used in the code checking of the concrete section and the cut locations which governed those code checks. This includes the demand moment (Mu) and capacity (Mn) when the top or bottom bars are in tension. This section also reports the one way shear demand (Vu) and capacity (Vn).

The Bending Steel details give the amount of steel "required by analysis" compared to the amount provided. In addition, the temperature and shrinkage (T/S) requirements are compared to the gross reinforcement ratio provided (which includes both the top and bottom reinforcement).

Note:

- The program only performs a one-way shear check on the design strip. Note that when ACI 318-19 is selected, Vc is calculated based on Equation (c) in Table 22.5.5.1. The size effect factor, λs, is not considered for foundation slabs as allowed in Section 13.2.6.2. The two way / punching shear code check is discussed in Pedestals - Punching Shear.
- If the design moments are too large or the Design Rules are too constrained, it is possible to fail the reinforcement checks for the design strip. These checks include temperature and shrinkage reinforcement minimums, flexural minimums, maximum spacing requirements and strength requirements. If any of these checks fail, a warning log message will be given.

Slabs - Results

When the model is solved, slabs are sub-meshed into plate elements and there are a couple of results spreadsheets specifically for these plates.

Plate Force Results

Access the **Plate Forces Spreadsheet** by selecting **Results Browser** from the **Results** toolbar and clicking **Plate Forces**. Or, select **Plate Forces** from the **Results** section of the **Explorer** panel.

Plate Forces (per ft) (By Combination)

	LC	Plate Label	Qx[k]	Qy[k]	Mx[k-ft]	My[k-ft]	Mxy[k-ft]
1	1	P1	0.442	-0.023	-0.59	1.055	0.211
2	1	P2	0.062	-0.106	-1.41	-1.136	0.305
3	1	P3	0.089	-0.017	-0.666	-3.023	0.546
4	1	P4	0.115	0.496	-1.125	-3.861	-0.293
5	1	P5	0.056	0.031	-0.31	-1.252	0.067
6	1	P6	0.107	-0.144	-0.506	-3.187	0.386
7	1	P7	0.019	0.372	-0.71	-2.349	-0.28
8	1	P8	0.033	0.213	-0.395	-2.983	-0.099
9	1	P9	-0.46	0.266	-0.624	0.643	0.339
10	1	P10	0.058	0.417	0.141	-2.271	-0.271
11	1	P11	-0.055	0.031	-0.245	-3.57	-0.118
12	1	P12	-0.036	0.045	-0.929	-3.598	0.214

The **Plate Forces** are listed in the spreadsheet for each plate element. Interpretation of output results is perhaps the most challenging aspect in using the plate/shell element. The results for the plates are shown for the geometric center of the plate.

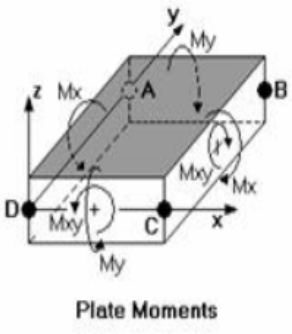
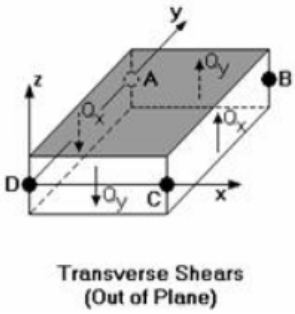


Plate Moments



Transverse Shears (Out of Plane)

The forces (Q_x and Q_y) are the out-of-plane (also called “transverse”) shears that occur through the thickness of the element. The Q_x shear occurs on the element faces that are perpendicular to the local x-axis, and the Q_y shear occurs on the element faces that are perpendicular to the local y-axis. Q_x is positive in the z-direction on the element face whose normal vector is in the positive x-direction. Q_y is positive in the z-direction on the element face whose normal vector is in the positive y-direction. The total transverse shear on an element face is found by multiplying the given force by the width of the element face.

The plate bending moments (M_x, M_y and M_{xy}) are the plate forces that induce linearly varying bending stresses through the thickness of the element. M_x is the moment that causes stresses in the positive x-direction on the top of the element. Likewise, M_y is the moment that causes stresses in the positive y-direction on the top of the element. M_x can then be thought of as occurring on element faces that are perpendicular to the local x-axis, and the M_y moment occurs on faces that are perpendicular to the local y-axis. To calculate the total M_x or M_y on the face of an element, multiply the given value by the length of the element that is parallel to the axis of the moment. For example, looking at the 'Plate Moments' figure above, the total M_x moment could be obtained by multiplying the

given M_x force by the length of side BC (the distance from joint B to joint C). The total M_y force can be calculated in the same way by instead using the length of side DC.

The M_{xy} moment is the out-of-plane twist or warp in the element. This moment can be added to the M_x or M_y moment to obtain the 'total' M_x or M_y moment in the element for design purposes. This direct addition is valid since on either the top or bottom surface, the bending stresses from M_{xy} will be going in the same direction as the M_x and M_y moments.

Note:

- For the placement of concrete reinforcement, it is helpful to realize that laying reinforcement parallel to the local x-axis will resist the M_x moment.
- A positive M_x or M_y moment will put the top fiber of the plate in tension.

Note that the plate bending (Q_x, Q_y, M_x, M_y, M_{xy}) results are forces per unit length. For example, a rectangular element with a B to C length of 10 feet showing a M_x force of 20K-ft would have a total force on the B-C face of the element of 20K-ft (per foot) times 10 feet, or 200K-ft.

Plate Corner Force Results

Access the **Plate Corner Forces Spreadsheet** by selecting **Results Browser** from the **Results** toolbar and clicking **Plate Corner Forces**, or select **Plate Corner Forces** from the **Results** section of the **Explorer** panel.

Plate Corner Forces (By Combination)

	LC	Plate Label	Point	Y[k]	MX[k-ft]	MZ[k-ft]
1	1	P1	N119	0.667	1.72	1.715
2			N120	0.69	-0.981	2.092
3			N121	-0.645	-2.044	0.539
4			N122	-0.712	1.513	-0.331
5	1	P2	N127	0.057	1.485	-1.709
6			N128	-0.259	-1.966	-2.569
7			N129	-0.058	-0.944	2.054
8			N130	0.26	2.47	2.835
9	1	P3	N131	0.119	-3.74	0.474
10			N132	0.158	5.336	1.978
11			N133	-0.112	3.507	-0.007
12			N134	-0.166	-4.952	-1.637

The plate corner forces and moments are those calculated at the corners of each plate in the GLOBAL directions. These values are obtained by multiplying the plate's corner displacements with the global stiffness matrix. Unlike the local forces, which are very accurate approximations, these corner forces represent EXACT results based on linear elastic theory. Also, the local plate forces are listed on a 'per unit length' basis where these global direction corner forces represent the total force on the plate at the corner in the given direction. This is similar to the way that joint reactions are

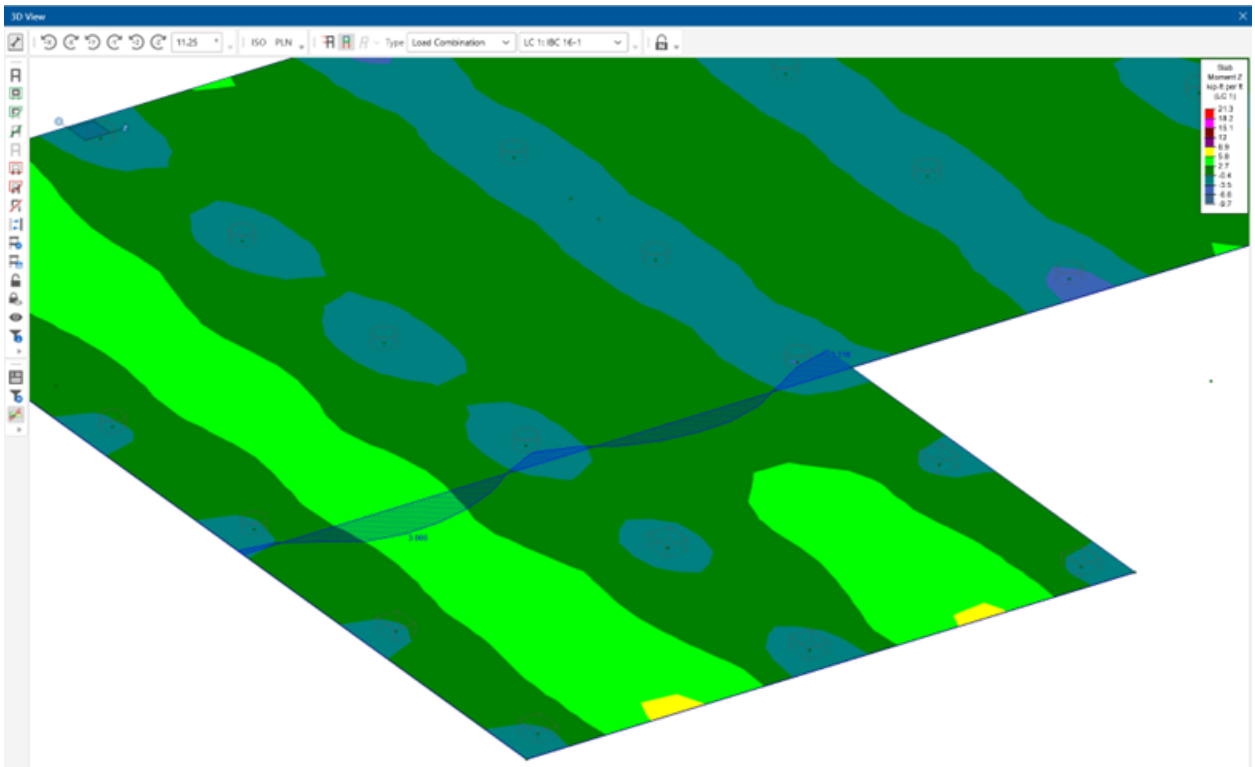
reported. At any given joint, the corner forces for all plates connected to that joint should sum to zero (a requirement of equilibrium), assuming no beams or supports are also present at the joint.

Note:

- See Spreadsheet Operations to learn how to use **Find**, **Sort**, and other options.
- See Model View Settings - Slabs to learn how to plot joint results.
- See Results View Settings - Slabs to learn how to plot plate force results.

Slab Contour Display Details

The slab **Contour Diagram** tool may be activated by clicking the  icon below the **Selection Toolbar** or by using clicking the icon on the **Results** toolbar. With this tool active, you may select two points within your slab to create a cut plane for which you want to review the slab analysis forces.



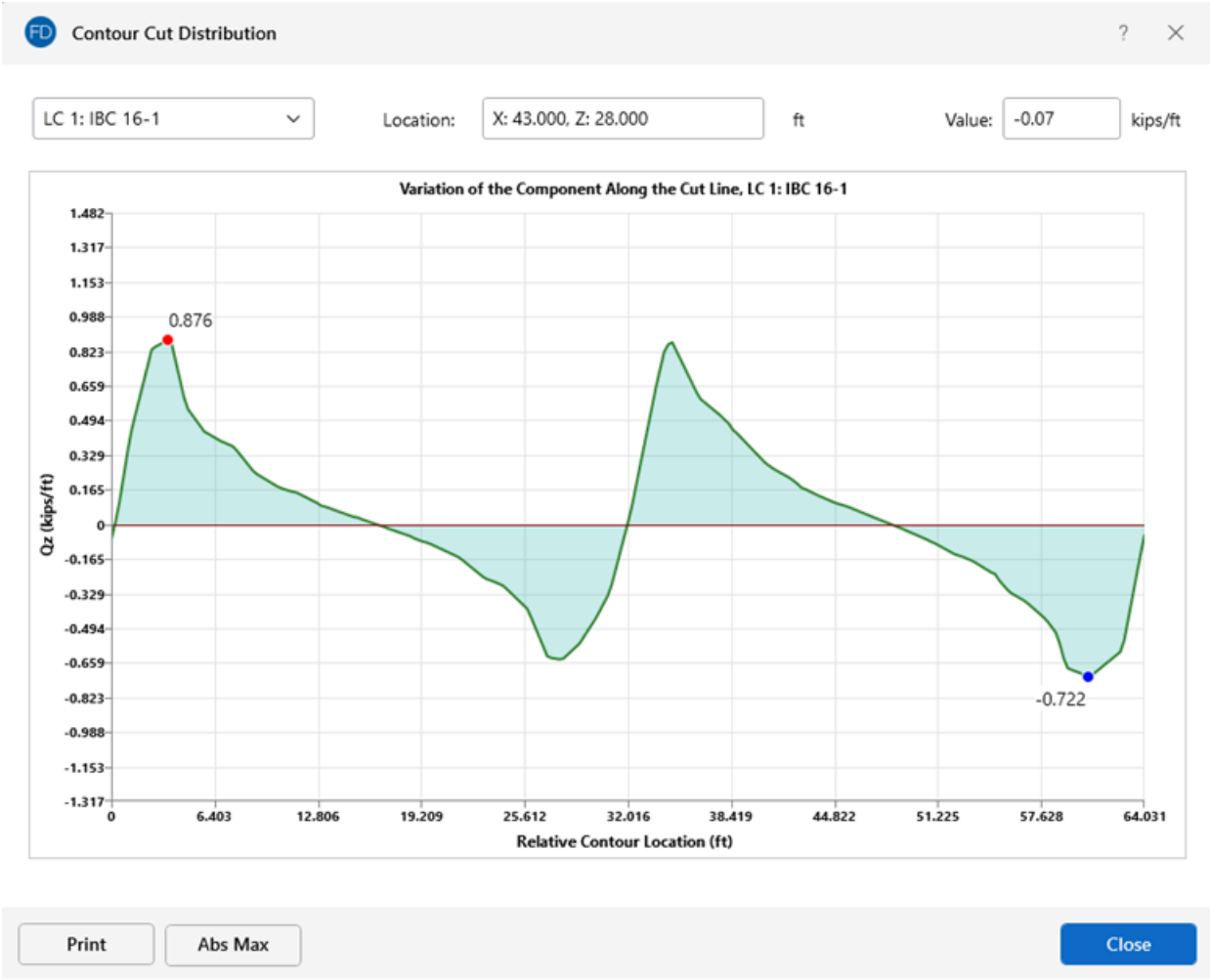
The integral of the force diagram at your cut is displayed in the bottom portion of the Properties Panel.

Contour Diagram

Total Integral: 51.925 k-ft
Positive Integral: 73.978 k-ft
Negative Integral: -22.052 k-ft

The **Contour Diagram Report** is used to turn a visual plate contour into a more traditional shear or moment diagram. To open a report, click the **Diagram Report** 

icon on the **Results** toolbar after you've made a cut through your slab using the Contour Diagram tool. The tool merely transforms the visual contour display into an actual diagram. In the image below, the contour diagram shows the numerical values associated with the contour at the cutting line.



This detailed diagram will show you all the results for every section along the length of the diagram. This information may even be copied to the clipboard for use in a spreadsheet program. This can be done by selecting **Copy Data** from the right-click menu.

Note: Because this display is just for the currently displayed plate contours, the units for the detailed diagrams are always the same as for the displayed contours.

Slabs - Sliding & Overturning

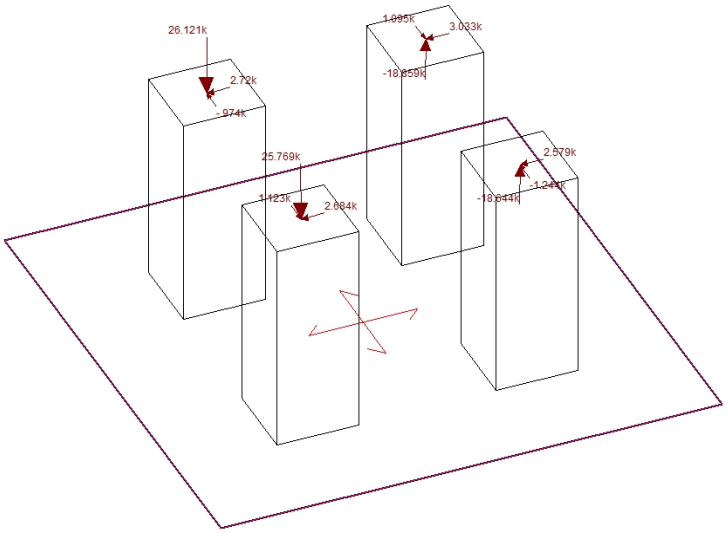
Slab sliding and overturning checks are done automatically for all service load combinations in RISAFoundation. These checks are similar to the sliding and overturning checks for footing elements, but differ in that each load category is considered as stabilizing or destabilizing (instead of each component of each load category as currently used by footing elements). For additional advice on this topic, please see the RISA Tips & Tricks webpage at risa.com/post/support. Type in Search keywords: Mat Slab Overturning.

Note: You can access these spreadsheets by selecting **Slab Stability** from the **Results Browser** drop-down on the **Results** tab of the ribbon, or by clicking **Slab Stability** from the **Results** section of the **Explorer** panel.

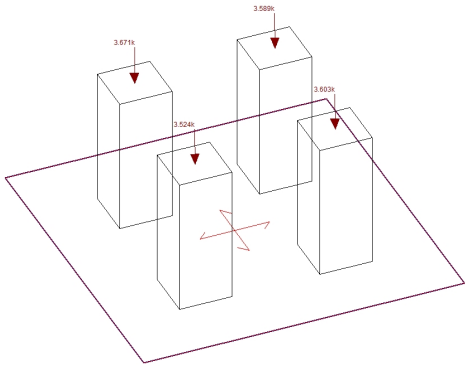
Demand vs Resisting Forces

Forces (overturning or sliding) are considered based on load category, not on the overall load combination loads. For more information, see the Footing Stability and Overturning sections in Footings Results. Footing Results specifically talk about footings, but the same considerations hold for slabs as well.

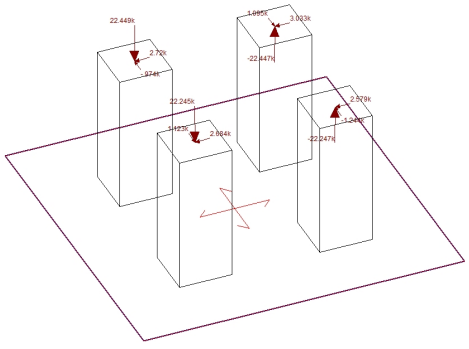
Let's show an example. Below, we have a slab that supports a four leg tower. The tower has both dead loads and wind loads applied. If we view a load combination with both dead load and wind load, it may look something like this:



Next, if we look at the model by load category, we'll see something like this:



DL Category



WL Category

When coming up with overturning and sliding demand/resistance, the contribution of *EACH category is taken separately*. This is even more important when you have more load categories. Here, if

attempting to verify the program, you would take all of the loads in the DL category and sum their effects for both overturning and sliding. From this, you can see whether there is a net overturning/sliding force or a resistance force. This would then determine whether this value ends up in the demand or resistance column. You would do the same process for all load categories in a given load combination. Once you've done that, you sum up each column (demand vs resistance) and these are the values you'll find in the program.

Overturning

The slab overturning checks are presented in the **Slab StabilitySpreadsheet**, shown below.

Slab Overturning Safety Factors (By Combination)

Overturning | Sliding | Overturning by Category | Sliding by Category

	LC	Slab	Angle[deg]	Mo-xx[k-ft]	Ms-xx[k-ft]	Mo-zz[k-ft]	Ms-zz[k-ft]	Ms-xx/Mo-xx	Ms-zz/Mo-zz
1	8	S1	0	0	263662.357	0	201025.106	9.99+	9.99+
2	8	S2	0	0	41745.675	0	41745.675	9.99+	9.99+
3	9	S1	0	0	345762.357	0	287275.106	9.99+	9.99+
4	9	S2	0	0	56098.08	0	56098.08	9.99+	9.99+
5	10	S1	0	0	263662.357	0	201025.106	9.99+	9.99+
6	10	S2	0	0	41745.675	0	41745.675	9.99+	9.99+
7	11	S1	0	0	263662.357	0	201025.106	9.99+	9.99+
8	11	S2	0	0	41745.675	0	41745.675	9.99+	9.99+
9	12	S1	0	0	263662.357	0	201025.106	9.99+	9.99+
10	12	S2	0	0	41745.675	0	41745.675	9.99+	9.99+
11	13	S1	0	0	325237.357	0	265712.606	9.99+	9.99+
12	13	S2	0	0	52509.978	0	52509.978	9.99+	9.99+
13	14	S1	0	0	325237.357	0	265712.606	9.99+	9.99+
14	14	S2	0	0	52509.978	0	52509.978	9.99+	9.99+
15	15	S1	0	0	325237.357	0	265712.606	9.99+	9.99+
16	15	S2	0	0	52509.978	0	52509.978	9.99+	9.99+

This spreadsheet lists the overturning checks for each slab under each service load combination.

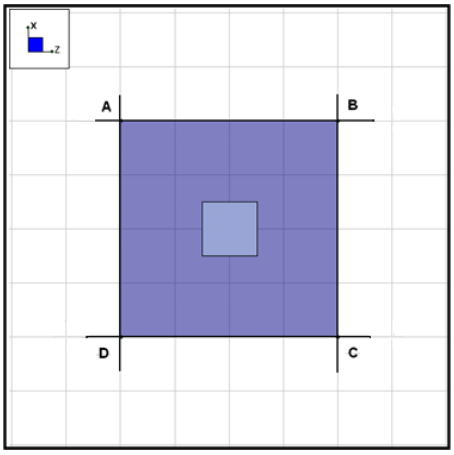
Angle

This is the **Local Axis Angle** from the Slabs Spreadsheet. This just gives the local axis of the slab, which defines the orientation of the sliding and overturning checks.

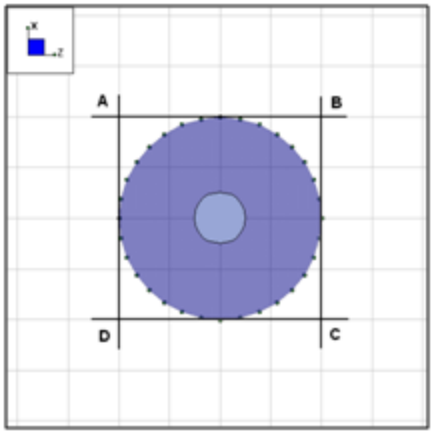
Overturning Moments

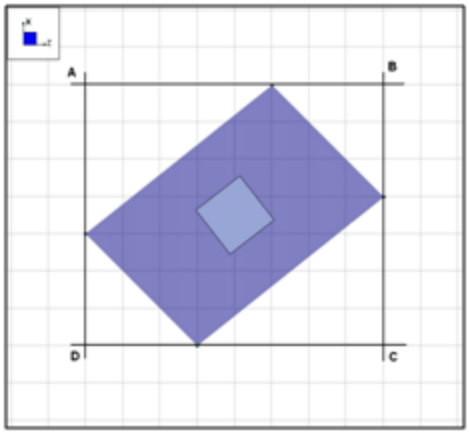
The **Mo-xx** and **Mo-zz** columns represent the overturning moments about the slab x and z local axes, respectively. The program will consider overturning about all edges of the slab (see image below); the

worst case in each direction is reported in the spreadsheet.



RISAFoundation calculates the moment about each of the four orthogonal lines set at the extreme points of the slab (AB, BC, CD, AD). **Mo-xx** reported in the spreadsheet is the largest moment taken about either AD or BC. **Mo-zz** reported in the spreadsheet is the largest moment taken about either AB or CD.





For more complex slabs, RISAFoundation sets four orthogonal lines at the four extreme points of the slab along the xx- and zz-axes. The program then calculates the moment about each of the four lines (AB, BC, CD, AD). **Mo-xx** reported in the spreadsheet is the largest moment taken about either AD or BC. **Mo-zz** reported in the spreadsheet is the largest moment taken about either AB or CD. These moments are calculated about the bottom face of the slab.

Note:
- For slabs in RISAFoundation, the loading is applied to the mid-height of the slab for finite element calculations. However, for slab overturning checks the program will assume the loading is applied at the top of the slab. This affects the overturning when column/wall shear loads are applied to the slab.
- If the **Local Axis Angle** is non-zero, the images above would be skewed at that angle.

Resisting Moments

The **Ms-xx** and **Ms-zz** columns represent the resisting moments about the slab local x- and z-axes, respectively. The program will consider all point loads, line loads and area loads in addition to the self weight in the calculation of the moments. The soil overburden pressure in the Slabs Spreadsheet is also considered to resist overturning. These moments are calculated about the bottom face of the slab.

Note: If there are piles defined on the slab, they will not affect these overturning calculations.

Ratio

The **Ms-xx/(Mo-xx)** and **Ms-zz/(Mo-zz)** columns represent the ratio of resisting moment to overturning moment. This is then compared against the Overturning Safety Factor, **SF**, value set in the Load Combinations Spreadsheet. If this value is lower than the SF, then the value in the spreadsheet will be in red and your slab fails the overturning check.

Limitations

- Currently, the program does not consider whether an individual slab is connected to other elements in the model. The program simply calculates the overturning and resisting forces it is seeing on that slab to determine the safety factor. The slab may, however, be connected to grade beams, pile caps, footings or other slabs that would prevent overturning of the slab from actually occurring.
- The overturning is reported for each individual slab element. If a system of slabs is connected to one another, each overturning check will occur individually and the overturning of the overall group is not considered.

Sliding

The sliding check is done along both the slab local x- and z-axes and the results are presented in the **Slab Stability Spreadsheet**, shown below.

Slab Sliding Safety Factors (By Combination)

Overturning | Sliding | Overturning by Category | Sliding by Category

	LC	Slab	Angle[deg]	Va-xx[k]	Vr-xx[k]	Va-zz[k]	Vr-zz[k]	SR-xx	SR-zz
1	8	S1	0	0	2280.7...	0	2280.7...	9.99+	9.99+
2	8	S2	0	0	834.913	0	834.913	9.99+	9.99+
3	9	S1	0	0	3505.7...	0	3505.7...	9.99+	9.99+
4	9	S2	0	0	1121.9...	0	1121.9...	9.99+	9.99+
5	10	S1	0	0	2280.7...	0	2280.7...	9.99+	9.99+
6	10	S2	0	0	834.913	0	834.913	9.99+	9.99+
7	11	S1	0	0	2280.7...	0	2280.7...	9.99+	9.99+
8	11	S2	0	0	834.913	0	834.913	9.99+	9.99+
9	12	S1	0	0	2280.7...	0	2280.7...	9.99+	9.99+
10	12	S2	0	0	834.913	0	834.913	9.99+	9.99+
11	13	S1	0	0	3199.4...	0	3199.4...	9.99+	9.99+
12	13	S2	0	0	1050.2	0	1050.2	9.99+	9.99+
13	14	S1	0	0	3199.4...	0	3199.4...	9.99+	9.99+
14	14	S2	0	0	1050.2	0	1050.2	9.99+	9.99+
15	15	S1	0	0	3199.4...	0	3199.4...	9.99+	9.99+
16	15	S2	0	0	1050.2	0	1050.2	9.99+	9.99+

This spreadsheet lists the sliding checks for each slab under each service load combination.

Angle

This is the **Local Axis Angle** from the Slabs Spreadsheet. This just gives the local axis of the slab, which defines the orientation of the sliding and overturning checks.

Applied Forces

The **Va-xx** and **Va-zz** columns represent the applied lateral force that causes sliding along the slab local x- and z-axes, respectively.

Resisting Forces

The **Vr-xx** and **Vr-zz** columns represent the resisting lateral forces that counteract sliding along the x- and z-axes, respectively. These loads include applied normal loads and self weight, and are multiplied by the **Coefficient of Friction** set in the **Solution** tab of **Model Settings**. The passive pressure is also considered as a sliding resisting force in both the slab's local z and x directions.

Note: The active and resisting sliding forces will be shown as an absolute value.

Sliding Ratio (SR)

The **SR-xx** and **SR-zz** columns represent the ratio of resisting forces to applied forces. If this value is lower than 1.0, then your slab is subject to sliding.

Limitations

- Currently, the program does not consider whether an individual slab is connected to other elements in the model. The program simply calculates the sliding and resisting forces it is seeing on that slab to determine the safety factor. The slab may, however, be connected to grade beams, pile caps, footings or other slabs that would prevent sliding of the slab from actually occurring.
- The sliding is reported for each individual slab element. If a system of slabs is connected to one another, each sliding check will occur individually and the sliding of the overall group is not considered.

Overturning by Category

The **Overturning by Category** tab of the **Slab Stability Spreadsheet** shows a breakdown of the overturning and resisting categories for each slab and load combination. Each category's x and z components are considered to act as either an overturning or resisting value. This category tab shows how RISA comes up with the *net* category value for each load combination.

RISAFoundation takes the net sum of the overturning and resisting values for each category per the True Safety Factor Method. Each category will either net a resisting or an overturning value in the slab's local xx and zz directions. The net overturning or resisting value for each category is combined with all load categories in a given load combination to come up with the Safety Factor result in each direction. It is important to assign the categories properly for your model to get correct safety factor results in the program.

Slab Overturning by Category (By Combination)

Overturning | Sliding | **Overturning by Category** | Sliding by Category

	LC	Slab	Category	Mo-xx[k-ft]	Ms-xx[k-ft]	Mo-zz[k-ft]	Ms-zz[k-ft]
1	8	S1	DL	0	263662.357	0	201025.106
2	8	S2	DL	0	41745.675	0	41745.675
3	9	S1	DL	0	263662.357	0	201025.106
4	9	S1	LL	0	82100	0	86250
5	9	S2	DL	0	41745.675	0	41745.675
6	9	S2	LL	0	14352.405	0	14352.405
7	10	S1	DL	0	263662.357	0	201025.106
8	10	S2	DL	0	41745.675	0	41745.675
9	11	S1	DL	0	263662.357	0	201025.106
10	11	S2	DL	0	41745.675	0	41745.675
11	12	S1	DL	0	263662.357	0	201025.106
12	12	S2	DL	0	41745.675	0	41745.675
13	13	S1	DL	0	263662.357	0	201025.106
14	13	S1	LL	0	61575	0	64687.5
15	13	S2	DL	0	41745.675	0	41745.675
16	13	S2	LL	0	10764.304	0	10764.304
17	14	S1	DL	0	263662.357	0	201025.106
18	14	S1	LL	0	61575	0	64687.5
19	14	S2	DL	0	41745.675	0	41745.675
20	14	S2	LL	0	10764.304	0	10764.304
21	15	S1	DL	0	263662.357	0	201025.106
22	15	S1	LL	0	61575	0	64687.5
23	15	S2	DL	0	41745.675	0	41745.675
24	15	S2	LL	0	10764.304	0	10764.304

Note: The results in the Overturning by Category tab gives the user the ability to take the overturning and resisting components of each category and calculate the results outside of the program using a traditional/simplistic method if they choose.

Sliding by Category

The **Sliding by Category** tab of the **Slab Stability Spreadsheet** shows a breakdown of the sliding and resisting categories for each slab and load combination. Each category's x and z components are considered to act as either a sliding or resisting value. This category tab shows how RISA comes up with the *net* category value for each load combination.

RISAFoundation takes the net sum of the sliding and resisting values for each category per the True Safety Factor Method. Each category will either net a resisting or an sliding value in the slab's local xx and zz directions. The net sliding or resiting values for each category is combined with all load categories in a given load combination to come up with the Safety Factor result. It is important to assign the categories properly for your model to get correct safety factor results in the program.

Slab Sliding by Category (By Combination)

Overturning | Sliding | Overturning by Category | **Sliding by Category**

	LC	Slab	Category	Va-xx[k]	Vr-xx[k]	Va-zz[k]	Vr-zz[k]
1	8	S1	DL	0	2280.7...	0	2280.7...
2	8	S2	DL	0	834.913	0	834.913
3	9	S1	DL	0	2280.7...	0	2280.7...
4	9	S1	LL	0	1225	0	1225
5	9	S2	DL	0	834.913	0	834.913
6	9	S2	LL	0	287.048	0	287.048
7	10	S1	DL	0	2280.7...	0	2280.7...
8	10	S2	DL	0	834.913	0	834.913
9	11	S1	DL	0	2280.7...	0	2280.7...
10	11	S2	DL	0	834.913	0	834.913
11	12	S1	DL	0	2280.7...	0	2280.7...
12	12	S2	DL	0	834.913	0	834.913
13	13	S1	DL	0	2280.7...	0	2280.7...
14	13	S1	LL	0	918.75	0	918.75
15	13	S2	DL	0	834.913	0	834.913
16	13	S2	LL	0	215.286	0	215.286
17	14	S1	DL	0	2280.7...	0	2280.7...
18	14	S1	LL	0	918.75	0	918.75
19	14	S2	DL	0	834.913	0	834.913
20	14	S2	LL	0	215.286	0	215.286
21	15	S1	DL	0	2280.7...	0	2280.7...
22	15	S1	LL	0	918.75	0	918.75
23	15	S2	DL	0	834.913	0	834.913
24	15	S2	LL	0	215.286	0	215.286

Note: The results in the Sliding by Category tab give the user the ability to take the sliding and resisting components of each category and calculate the results outside of the program using a traditional/simplistic method if they choose.

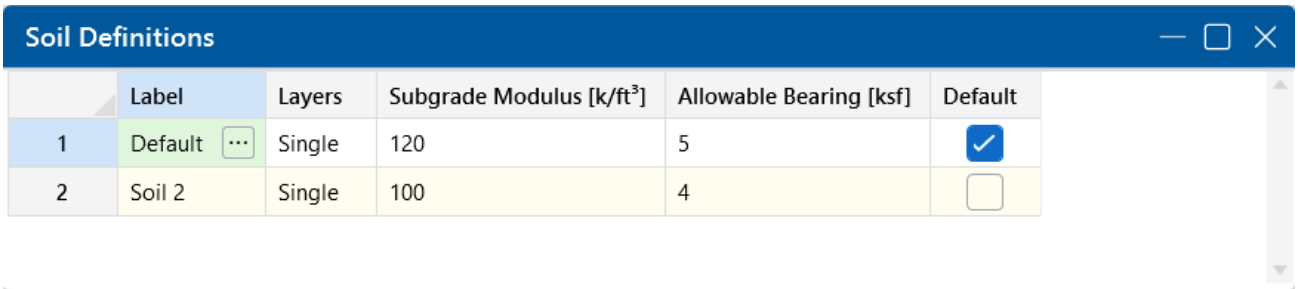
Soils

Assigning Soil Properties

The default **Subgrade Modulus** and **Allowable Bearing Pressure** are entered in the **Soil Definitions Spreadsheet**. There is a check box in the Soil Definitions Spreadsheet to determine which soil definition will be used as the default. The default soil definition will be used for the soil properties in the whole model, except where another soil region is drawn. It is not necessary to draw a soil region for the default soil definition.

Soil Definitions Spreadsheet

The **Soil Definitions Spreadsheet** records the properties of each region and may be accessed by selecting **Soil Definitions** from the **Data Entry** drop-down on the **Spreadsheets** tab of the ribbon, or by clicking **Soil Definitions** from the **Data Entry** section of the **Explorer** panel. The entries are explained below.



Label

You may assign a unique label to each soil definition. The soil definition with the **Default** box checked will be used for the soil properties in the model, except where another soil region using a different soil definition is drawn.

Layers

Soil properties may be defined for a single soil layer, or multiple layers. Clicking the (ellipsis) button in the Layers column will open the **Soil Layer Profile Spreadsheet**, where you can further define properties for a multi-layer soil definition. These properties are used for static pile design of Concrete, Hot Rolled Steel and Wood piles. For more information, see Soil Layer Profile.

Subgrade Modulus

The subgrade modulus is used to establish the elastic stiffness of the soil. For more information, refer to the section on automatically generating soil springs.

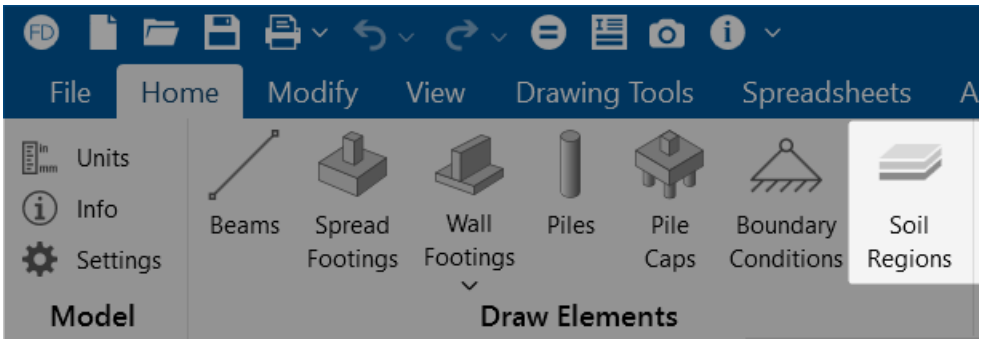
Allowable Bearing Pressure

Enter the gross allowable soil bearing pressure for the soil definition in this column.

Draw Soil Regions

To create a new soil region, click the **Soil Regions** button in the **Draw Elements** section of the **Home** ribbon.

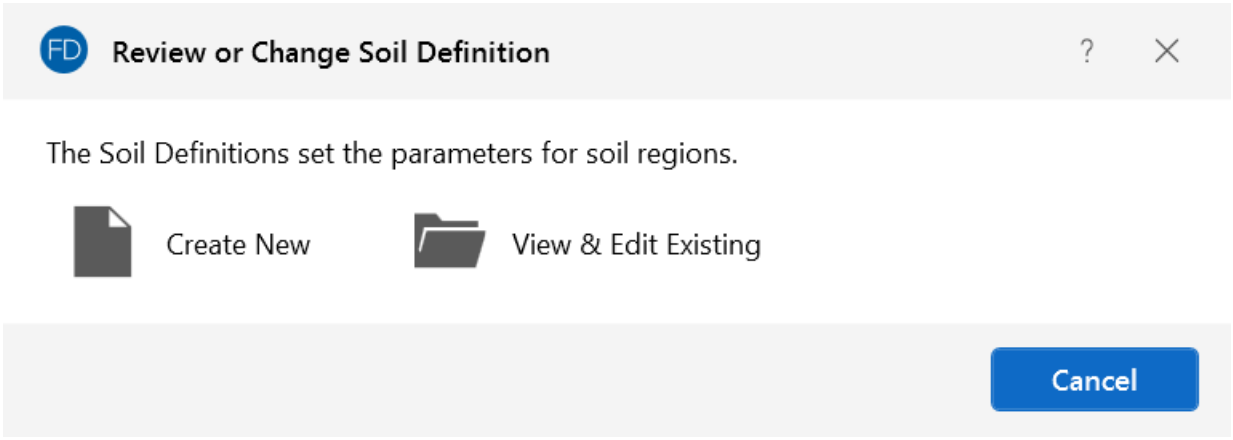
You may apply soil properties by drawing a polygon that defines the area where these properties exist. If two regions overlap, the properties that were drawn most recently will control for the overlapping area.



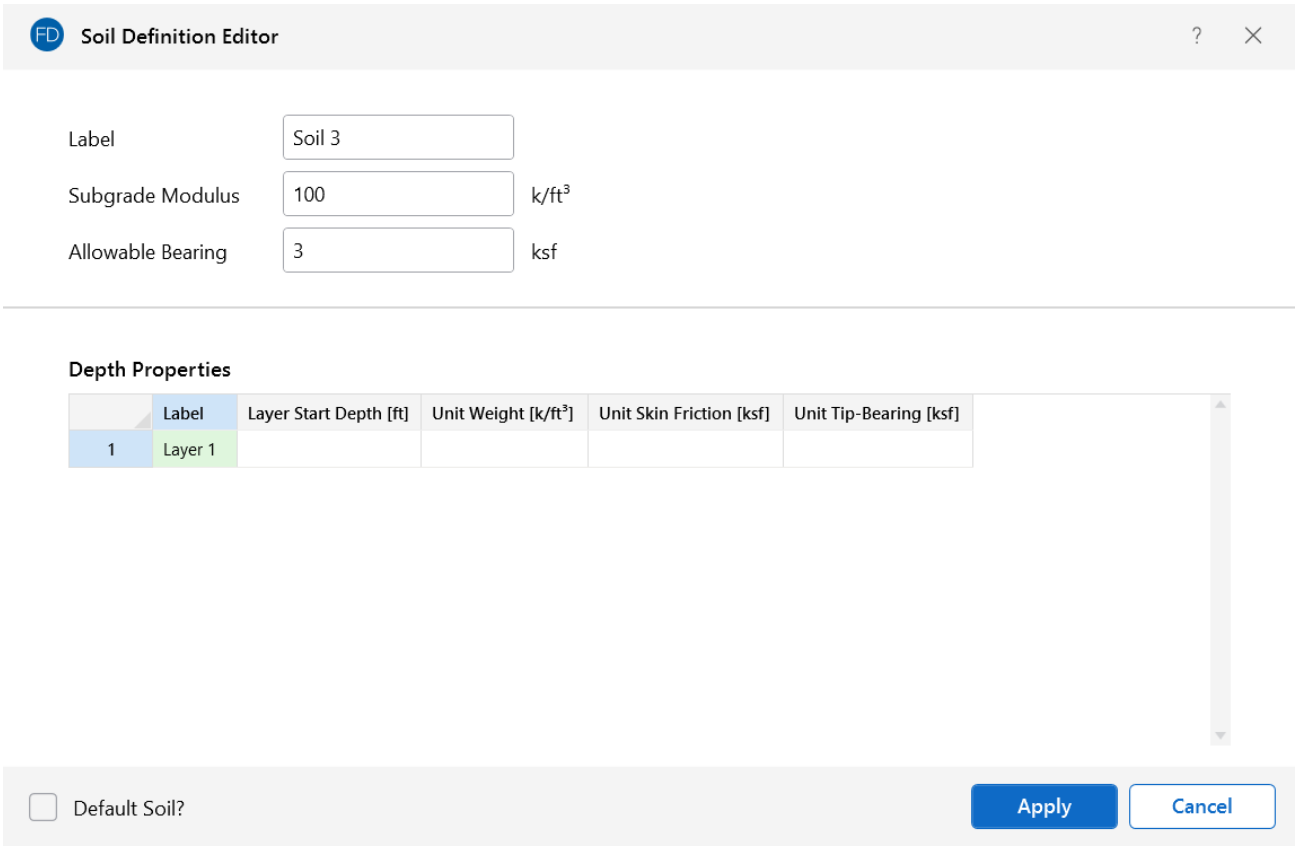
To Assign Soil Properties:

1. If there is not a model view already open then go to the **View** ribbon.
2. Click the **Open 3D Views** icon in the **Window** section to open a new view.
3. Go to the **Home** ribbon.
4. Click the **Soil Regions** icon in the **Draw Elements** section.
5. In the **Properties** panel, set the soil region properties.
6. Start drawing a polygon to define the region that will be controlled by these soil properties. To indicate that you have finished drawing the region, double-click on the last node of the polygon.
7. To cancel and redraw your soil region, click the **right mouse button**. To cancel drawing altogether, right-click the mouse button again, or press the **Esc** key.

To create a new soil definition, click the ··· (ellipsis) button next to the Soil Definition text box in the **Properties** panel. A **Review or Change Soil Definition** pop-up displays asking if you want to create a new soil definition or view and edit an existing soil definition, as shown below.



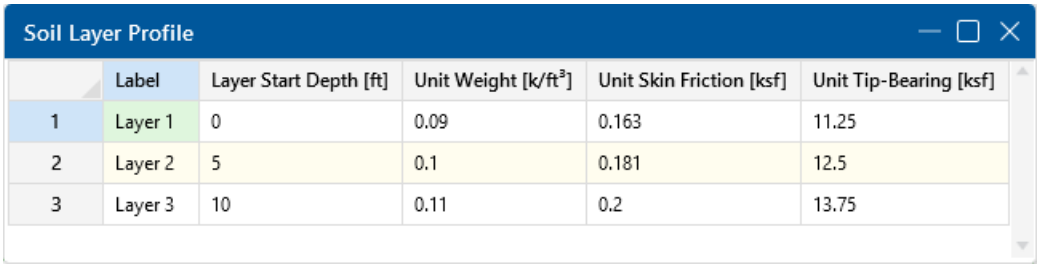
Selecting **Create New** will open up the **Soil Definition Editor** dialog, where you can create a new soil definition. This dialog box can also be accessed to view existing soil definitions from the **Soil Definitions Spreadsheet**. In that spreadsheet, simply click on the ··· (ellipsis) button next to the soil label and the **Soil Definition Editor** will open, showing the inputs for that definition.



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

Soil Layer Profile Spreadsheet

The **Soil Layer Profile Spreadsheet** is used to define soil layers for static pile design. It is only necessary to input soil layer data when using Concrete, Hot Rolled Steel and Wood piles.



	Label	Layer Start Depth [ft]	Unit Weight [k/ft^3]	Unit Skin Friction [ksf]	Unit Tip-Bearing [ksf]
1	Layer 1	0	0.09	0.163	11.25
2	Layer 2	5	0.1	0.181	12.5
3	Layer 3	10	0.11	0.2	13.75

To access the **Soil Layer Profile Spreadsheet**, open the **Soil Definitions Spreadsheet** and click the ··· (ellipses) button in the Layer Cell for the soil definition you wish to edit.

Note: The **Soil Layer Profile Spreadsheet** is associated with an individual soil definition.

The entries in the **Soil Layer Profile Spreadsheet** are explained below.

Label

You may assign a unique label to each layer of soil. Each label has to be unique, so if you try to enter the same label more than once, you will get an error message.

Layer Start Depth

The layer start depth is the elevation at the top of the soil layer. The first layer of soil typically has a start depth of zero. Each layer depth of soil is assumed to exist until it reaches the next layer's start depth.

Unit Weight

The unit weight represents the effective unit weight of the soil layer in lb/ft^3 or kN/m^3. The unit weight, $\gamma$of a soil layer is typically determined from soil tests measuring the weight and volume of a sample. Typical values of the unit weight for various soil types could be(from Lindeberg, *Civil Engineering Reference Manual, 11th ed.*) :

Soil Type	γ(lb/ft^3)	γsat (lb/ft^3)
Sand, loose and uniform	90	118
Sand, dense and uniform	109	130
Sand, loose and well graded	99	124
Sand, dense and well graded	116	135
Glacial Clay, soft	76	110
Glacial Clay, stiff	106	125

Unit Skin Friction

The unit skin friction at each layer is the theoretical unit friction resistance for that layer which can be measured in lb/ft^2 or kN/m^2. There are several methods to calculate this value depending on the engineer's judgment, type of soil and assumptions for the applicable factors. A few of these equations are listed below (Equations reference: Joseph E. Bowles, *Foundation Analysis and Design, 5th Ed.*):

1. The α Method: $q_s = \alpha c$

- α : adhesion factor
- c : average cohesion

2. The λ Method: $q_s = \lambda(\bar{q} + 2s_u)$

- λ : coefficient dependent on pile depth
- $\bar{q}$: effective overburden pressure
- su : undrained shear strength

3. The β Method: $q_s = \beta\bar{q}$

- β : K tan δ
  - K : Earth pressure coefficient
  - δ : undrained friction angle
- $\bar{q}$: effective overburden pressure

Unit Tip-Bearing

The unit tip-bearing is the unit tip or point bearing resistance for each layer of soil which can be measured in lb/ft^2 or kN/m^2. The unit tip-bearing resistance will be used to calculate the ultimate bearing capacity for a pile when the length of the bottom of the pile terminates in that layer. There are several methods to calculate this value depending on the engineer's judgment, type of soil and assumptions for the bearing capacity factors. A few equations are listed below to calculate the unit tip bearing capacity (Equations reference: Joseph E. Bowles, *Foundation Analysis and Design, 5th Ed.*) :

1. Unit Point Resistance: $q_p = cN'_c d_c s_c + \eta \bar{q} N'_q d_q s_q + \frac{1}{2}\gamma' B_p N_\gamma s_\gamma$

- c : Cohesion of soil beneath pile point
- Bp : width of diameter of pile point
- N'c : bearing capacity factor for cohesion
- N'q: bearing capacity factor
- N'γ : bearing capacity factor for base width

2. Meyerhof's Equation: $q_p = (40N)\frac{L_b}{B}$

- N : statistical average standard penetration number near pile point
- Lb : pile penetration depth into point bearing stratum
- B : width of diameter of pile point

Soil Springs Calculation (Automatic)

The program uses what is commonly referred to as a Winkler foundation model. Essentially, the soil effect is concentrated at the node locations and the soil springs are based on a simple multiplication of Tributary Area of the node and the user entered subgrade modulus.

Adjusting the Subgrade Modulus

Typically, the geotechnical engineer will obtain the subgrade modulus for a 1' by 1' or .3m. x .3m sample plate. Typical values for various soil types could be:

Soil Type	k_s (kcf)
Loose Sand	30 ~ 200
Medium Dense Sand	60 ~ 500
Dense Sand	220 ~ 850
Clayey Medium Dense Sand	200 ~ 500
Silty Medium Dense Sand	150 ~ 300
Clayey Soil	80 ~ 300

For settlement calculations, this value would need to be modified to account for our actual foundation size (e.g. 10 ft by 14 ft). For this example we will use equations from Principles of Foundation Engineering, 3rd edition, by Braja Das, pgs. 263 - 264. Note that these equations specifically apply to cohesionless soil. We will assume a 1ft x 1ft sample plate with a base ks value of 500 kcf.

$$K_{10x10} = k_1\left(\frac{B+1}{2B}\right)^2 = 500\left(\frac{10+1}{2*10}\right)^2 = 151 kcf$$

$$K_{10x14} = K_{10x10} * \left(\frac{1+0.5*B/L}{1.5}\right) = 151 * \left(\frac{1+0.5*10/14}{1.5}\right) = 136.6\, kcf = 0.07906\ kip/in^3$$

These are the values that the user must enter for the subgrade modulus. The program does not automatically adjust the modulus according to the size of the foundations.

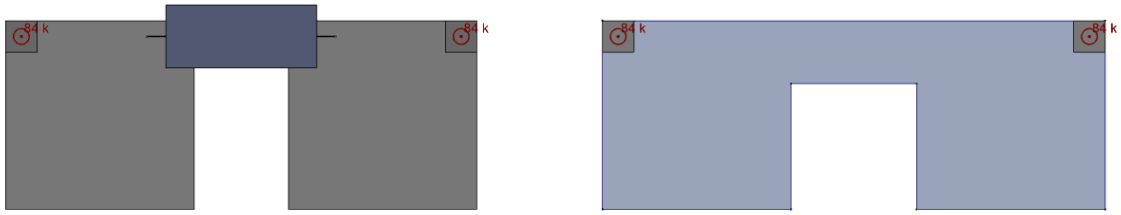
Once the user has entered in these values for the subgrade modulus, the program will automatically calculate the spring stiffness for all the joints in the model based on the tributary area of each node.

Single and Combined Footings

Since RISA optimizes footing dimensions, the final size of the spread footing is not known at the time of the FEM solution. Therefore, the program approximates the size as the average width and length in the user entered range of values.

Unlike large mat foundations, each spread footing is modeled with a single joint. Therefore, that joint needs to have a rotation spring to accurately distribute the loading for a true combined footing. RISAFoundation uses this same assumptions regarding the modulus of subgrade reaction and the dimensions of the footing when approximating these rotational springs.

Note: The nodes that define the footings currently reside at the center of the footing PEDESTAL. This is fine in most cases, but can lead to problems when you have eccentric footing that are part of a combined footing, as shown on the left in the image below. In these cases, it is preferable to model these footings with a small mat foundation, as shown on the right in the image below.



Grade Beams

Grade beams are also looked at as having compression only soil springs along their length. These springs will be equal to the sub-grade modulus times the width of the beam times the tributary length of the beam node.

If it is not desirable to have spring supports along the length of the beam, then a soil region with a very low sub-grade modulus can be locally defined.

Modeling Mat Foundations

Although a little out of date, the ACI publication 336.2R-88, “Suggested Analysis and Design Procedures for Combined Footing and Mats”, is a good reference for the modeling of mat foundations. For more information on these suggestions, refer to the ACI document directly.

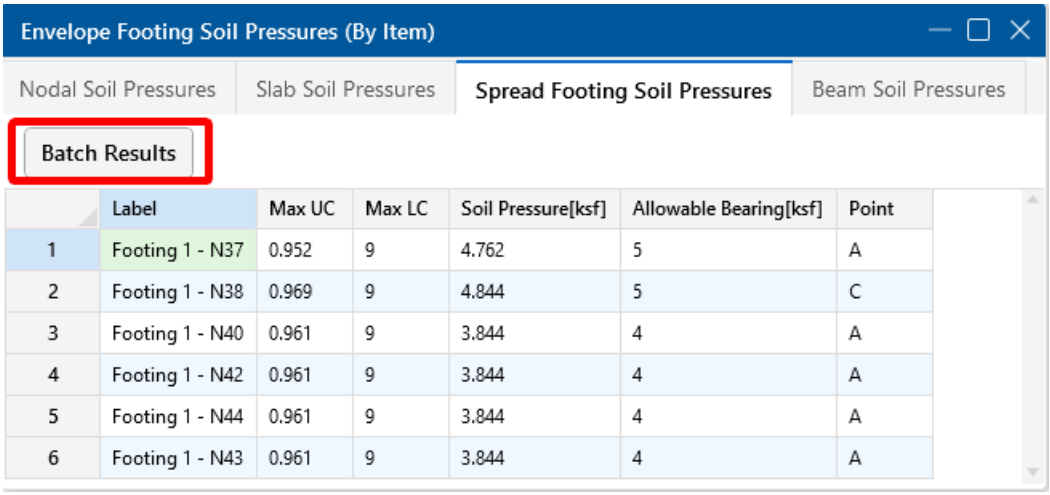
The program uses a "Winkler Foundation" approach to the solution. As such, it does NOT explicitly take into account node coupling effects between soil springs. Therefore, it may be desired to create a "dishing" effect for the mat foundation by adjusting the subgrade modulus for different "zones" of the mat, as shown below. This procedure is described in more detail in the ACI report mentioned above.

Soil Pressure Results

RISAFoundation will automatically calculate the soil pressure profile that is induced by each one of your service level load combinations. For spread footings, these results are discussed in detail in Footing Results.

There is an **Enveloped** and a **Batch** version of the **Soil Pressures Spreadsheet**. Open the **Enveloped** version by selecting **Soil Pressures** from the **Results** section of the **Explorer** panel.

To access the **Batch** version, simply click the **Batch Results** button in the **Enveloped Soil Pressures Spreadsheet**, shown below:



	Label	Max UC	Max LC	Soil Pressure[ksf]	Allowable Bearing[ksf]	Point
1	Footing 1 - N37	0.952	9	4.762	5	A
2	Footing 1 - N38	0.969	9	4.844	5	C
3	Footing 1 - N40	0.961	9	3.844	4	A
4	Footing 1 - N42	0.961	9	3.844	4	A
5	Footing 1 - N44	0.961	9	3.844	4	A
6	Footing 1 - N43	0.961	9	3.844	4	A

Envelope Soil Pressure Results Spreadsheet

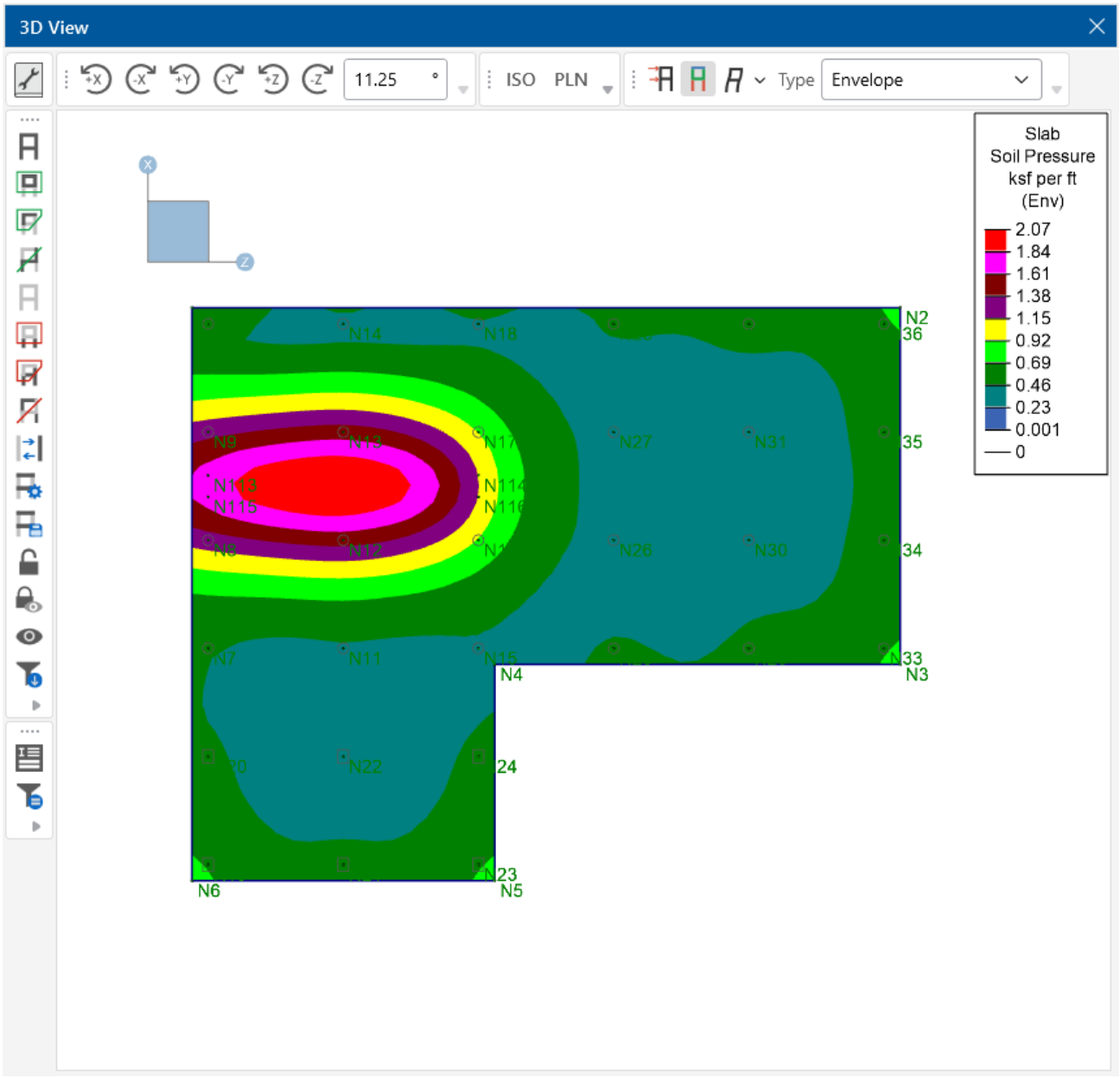
The **Envelope Soil Pressures Results Spreadsheet** displays summaries of the soil pressure results.

The **Nodal Soil Pressures** tab displays the results for every node in the model. It also displays the allowable bearing pressure and the unity check for the load combination that produced the maximum unity check. If the load combination has an ABIF (allowable bearing increase factor) due to the presence of transient loads, then this will be reflected in the Allowable Bearing value listed in this spreadsheet, NOT the U/C value. Therefore, a U/C value greater than 1.0 would always indicate a soil failure. A calculated Soil Pressure of NC indicates that a soil pressure was NOT calculated for that joint. This is most likely because that node was a loading node that was not used as part for the FEM solution of a mat foundation.

The **Slab**, **Spread Footing** and **Beam Soil Pressures** tabs display summaries of each of the foundation elements in the model, along with the point or location that controlled the soil bearing code check for that element.

Plotting the Soil Pressure Results

The **Soil Pressure** results may be plotted using the options under Results View Settings - Slabs. An example of a plot showing the contoured view of the soil pressures as they exist under the slab is shown below.



This view will not directly show if the soil is exceeding the allowable soil pressure. If this is desired, you can specify a custom Range of values to plot. Any values above the specified range will display in red, indicating a failure. These specific display options are discussed in more detail in Graphics - Results View Settings.

Note: The Slab Contour controls that are used to display the soil pressure results will also display the soil pressure contours that exist underneath grade beams and spread footings.

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

Solution



To solve the model, click the button on the **Home** tab. A progress bar displays information as the solution proceeds. In general, loads are attributed to the slabs, beams and footings followed by an optimization of the slab reinforcement, as well as beam and footing sizes and reinforcement. Any warnings are listed in the **Warning Log**, which can be opened from the **Spreadsheets Menu**. For more information on how slabs and beams are optimized, see Design Optimization. For more information on footing optimization, see Footings - Results.

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).

When a solution has been performed and the results are available the icon on the status bar in the lower left corner will change from to . If any changes are made to the model that invalidate the results then the results will be cleared and another solution will be necessary.

The specific type of solution performed by RISAFoundation is **Batch Solution**. This means that a solution may be performed on multiple combinations and the results retained for **each** solution. You may group the results by item or by load combination in the results spreadsheets by choosing the **Sort by Combination** or **Sort by Item** option on the **Results** ribbon. For example, you can have all the combination results for beam M1 grouped together or you can have all the beam results for Load Combination "DL+LL" grouped together. See Load Combinations to learn how to mark combinations to be solved in the batch solution.

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 open as well. You may use the **Application Settings** on the **File Menu** to change the way that you are prompted to save results. For more information, see Output Application Settings .

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 icon on the status bar in the lower left corner will change from to . If any changes are made to the

model that invalidate the results then the results will be cleared and another solution will be necessary.

The solution is based on the widely accepted linear elastic stiffness method for solution of the model. The stiffness of each element of the structure is calculated independently. These element stiffnesses are then combined to produce the model's overall (global) stiffness matrix. This global matrix is then solved versus the applied loads to calculate joint deflections. These joint deflections are then used to calculate the individual element stresses. The primary reference for the procedures used is **Finite Element Procedures**, by K. J. Bathe (Prentice-Hall, 1996).

Skyline Solver

This solution method is also sometimes referred to as an Active Column solution method. In finite element analysis, the nonzero terms of the stiffness matrix are always clustered around the main diagonal of the stiffness matrix. Therefore, the Skyline or Active Column solutions take advantage of this by condensing the stiffness matrix to exclude any zero stiffness terms that exist beyond the last non-zero term in that column of the matrix.

Since the majority of terms in a stiffness matrix are zero stiffness terms, this method greatly reduces the storage requirements needed to store the full stiffness matrix. However, for large models (+10,000 joints) , the memory requirements even for a skyline solution can be problematic.

This solution method has been used successfully in RISA for more than 20 years, and has proven its accuracy continuously during that time.

Sparse Solver

The skyline solver described above is moderately efficient because it only stores and performs operations on the terms within the "skyline" of the stiffness matrix. However, that solution still contains a great number of zero stiffness terms within the skyline of the matrix. A Sparse Solver will reduce the matrix size to an absolute minimum by eliminating the storage of ALL zero stiffness terms.

A sparse solver is the most efficient solution methodology possible because it stores and performs operation only on the non-zero terms of the stiffness matrix. For this reason, the sparse solution is preferred from both a solution speed and memory requirement standpoint.

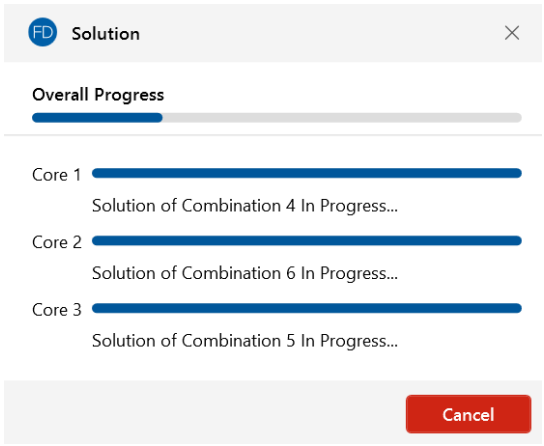


Note: The skyline solver is retained mostly for comparison and verification purposes.

Multi-Threaded Solutions

RISA products now have the ability to use multiple CPU cores if you check the check box in (Global) Model Settings. Set the **Processor Core Utilization** to **Single**, **Balanced** (Optimum) or **Maximum** in the **Solution** tab of **Model Settings**. The program will split up the solving based on load combination and use each available CPU core to solve load combinations in parallel. Once this portion of the solution is complete, the program then combines this data for enveloping and design purposes. This behavior will improve solution time for computers with multiple cores and models with large numbers of load combinations.

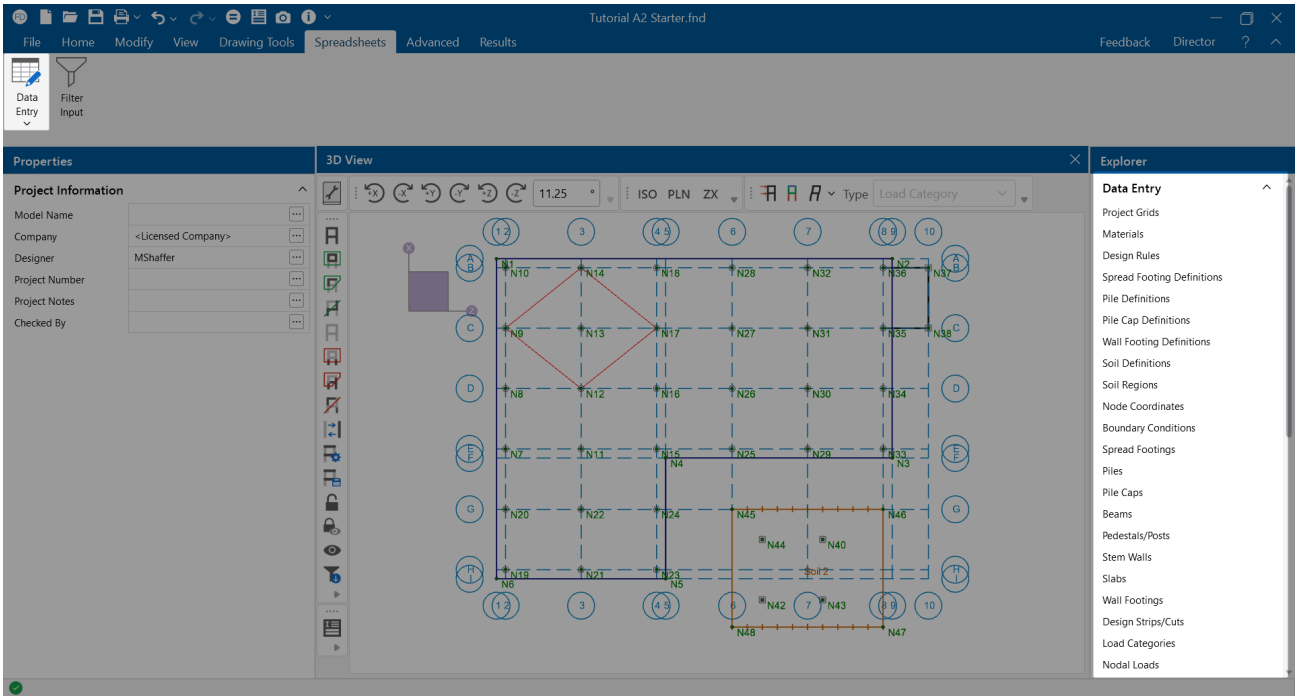
During the load combination portion of the solution, the **Solution** dialog will show progress for each core being used, as shown below:



Spreadsheet Operations

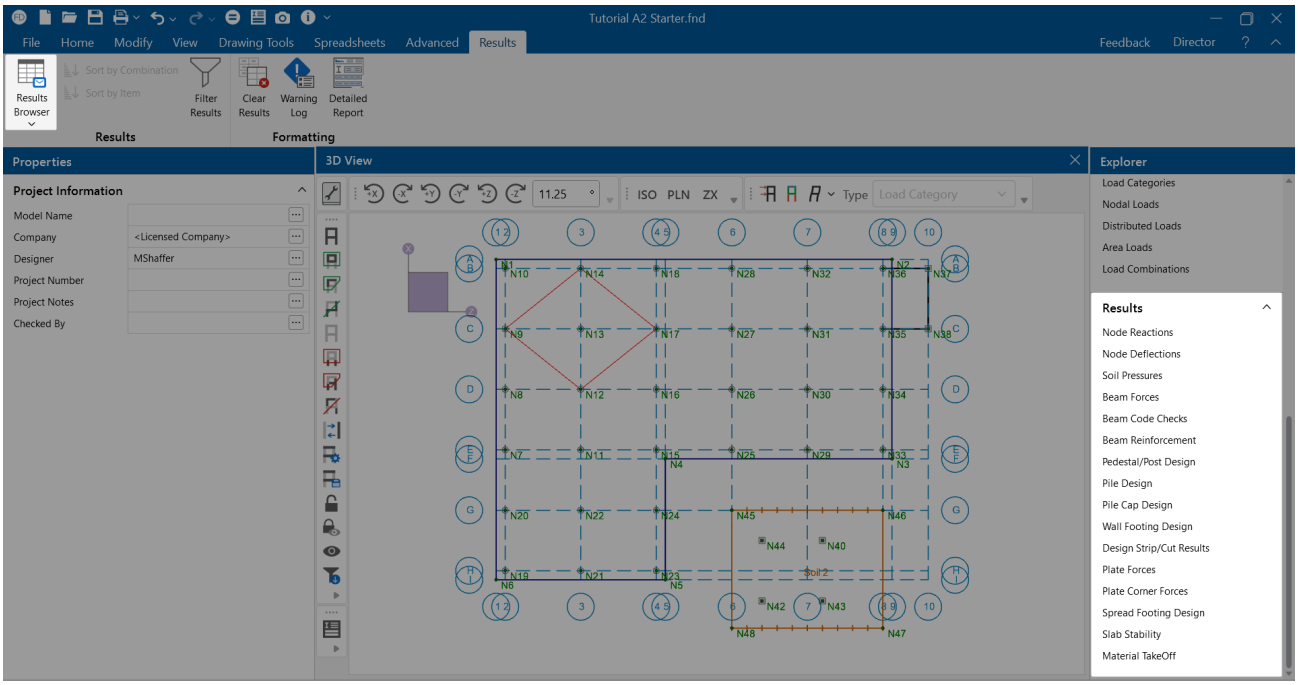
Powerful spreadsheet operations may be used to view, sort and edit the input data and results. The spreadsheets and model views are synchronized. As you edit a model graphically, the spreadsheets are automatically updated and as you make changes in the spreadsheets, the model views reflect these changes immediately.

The input data may be accessed from the **Explorer** panel or the **Data Entry** drop-down on the **Spreadsheets** tab of the ribbon.



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** section of the **Explorer** panel, or the **Results Browser** found on the **Results** tab of the ribbon.



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, 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 overall spreadsheet.
- The mouse wheel is also an excellent tool for scrolling up and down in spreadsheets or graphics. Roll the wheel or click and drag to move around.

Spreadsheet Keyboard Commands

The following keyboard commands are available:

Key	Function
Arrow Keys	Move the active cell one location.
TAB	Move right one cell.
ENTER	Move to the first column of the next line. Adds new line if necessary.
PAGE UP	Move the active cell one full page up.
PAGE DOWN	Move the active cell one full page down.
HOME	Move to the first line of the spreadsheet.
END	Move to the last line of the spreadsheet.
F3	Insert new line below current line and repeat the current values in the new line.
F4	Delete current line.
F8	Add new row to end using values from row above.

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.

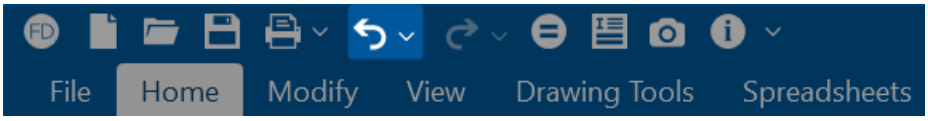
To Select	Do This
A single cell	Click the cell, or press the arrow keys to move to the cell.
A range of cells	Click the first cell of the range, and then drag to the last cell.

To Select	Do This
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.

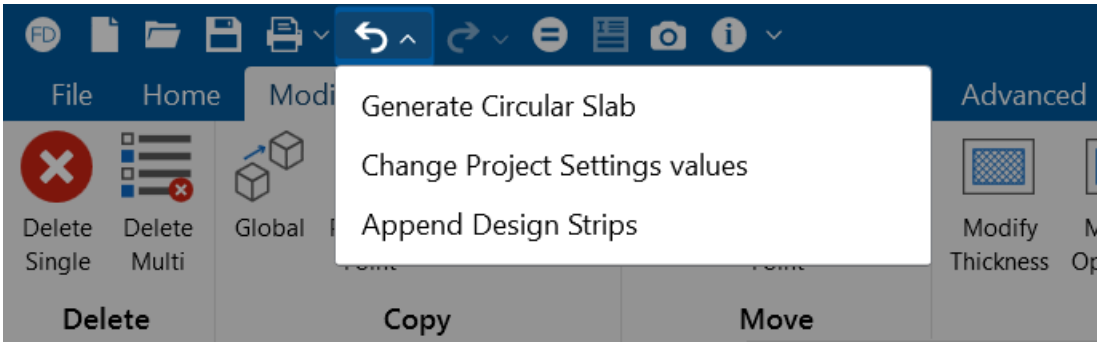
Note: To cancel a selection of cells, click any single cell in the spreadsheet.

Undoing Operations

RISAFoundation 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.



Another option is **Multiple Undo**, accessed by the drop-down arrow next to the **Undo** button on the **Quick Access Toolbar**. This will allow you to choose a specific previous action that you would like to undo.



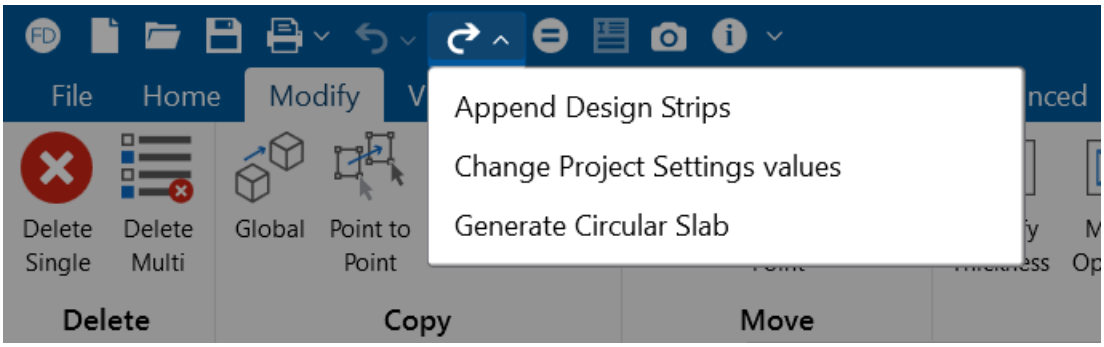
The model view and the spreadsheets will visually display the "undoing". Remember that graphic edits are undone as well.

Redoing Operations

RISAFoundation provides you with virtually unlimited **Redo** capability so that you may reapply actions that were previously undone. Simply click the **Redo** button as many times as the **Undo** button was used just prior.



Another option is **Multiple Redo**, accessed by the drop-down arrow next to the **Redo** button on the **Quick Access Toolbar**. This will allow you to choose a specific previous action that you would like to redo.



The model view and the spreadsheets will visually display the "redoing". Remember that graphic edits are redone as well.

Editing Spreadsheets

The spreadsheets have been specifically developed for the input and editing of structural models. There are many ways to edit the spreadsheets, allowing you to quickly build your model. You may copy and move data from other locations or other files. You may also fill large blocks of cells automatically and perform math on these cells.

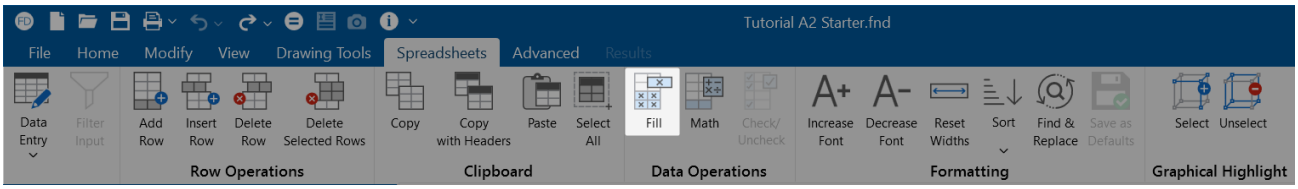
To Change Basic Load Cases in a Load Spreadsheet:

Use the drop-down load list or the adjacent buttons on the **Window** toolbar.

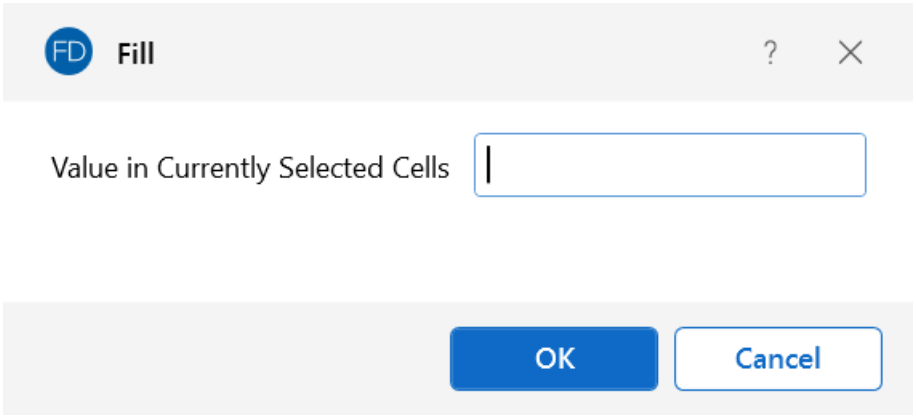


To Fill Selected Cells:

1. Select the cells you wish to fill.
2. Click the **Fill** icon from the **Spreadsheet** ribbon.



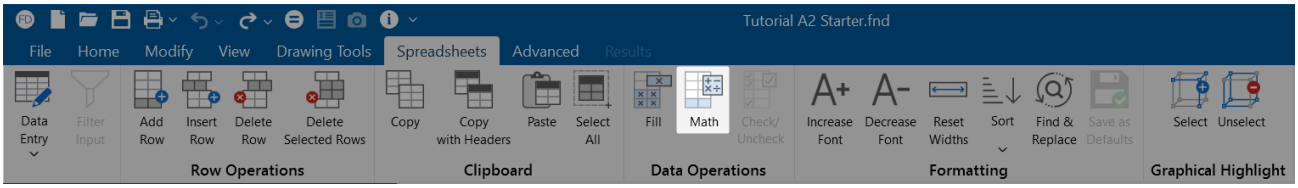
3. Type the value you wish to apply.



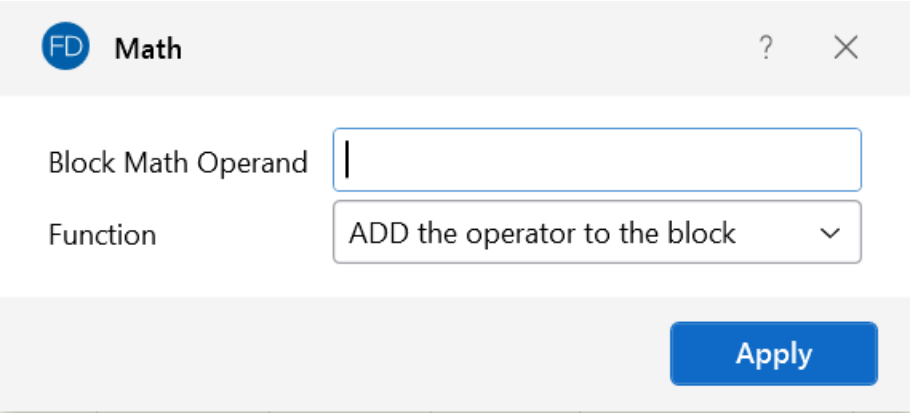
4. Click **OK**.

To Perform Math on Selected Cells:

1. Select the desired cells.
2. Click the **Math** icon from the **Spreadsheet** ribbon.



3. Specify the value and the operation you wish to apply.



4. Select a **Function** from the drop-down list.
5. 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 complete your changes, press **ENTER**.
4. To cancel your changes, press **ESC**.

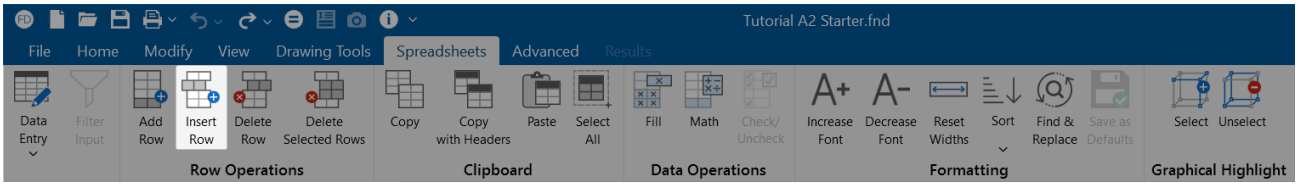
Note:
- To replace the contents of a cell, click on the cell and type the new entry.
- To edit the contents of a cell, double-click on the cell and use the arrow keys to locate the cursor and perform the edit.

Inserting, Deleting and Clearing Cells

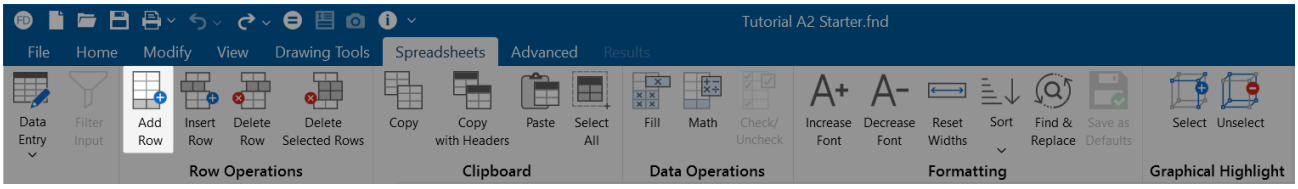
To insert cells, you must insert an entire row. When you delete cells, you must delete an entire row. The row is removed from the worksheet and shifts the lower cells to fill the space. When you clear cells, you remove the cell contents but leave the blank cells in the spreadsheet.

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 Row** button.



3. To have the values of the current line copied in the new line, click the **Add Row** button.

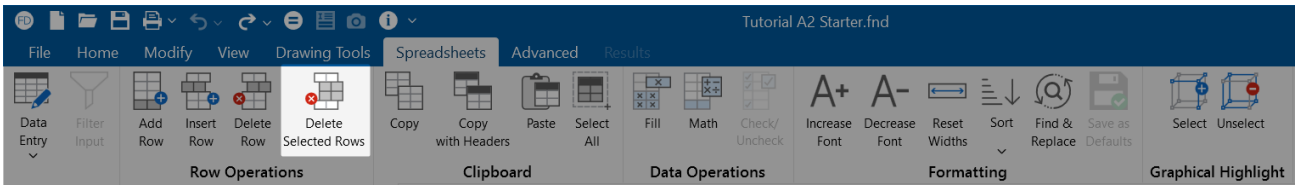


To Clear Cell Contents:

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 Selected Rows** button and the lower cells will shift up to fill the space.

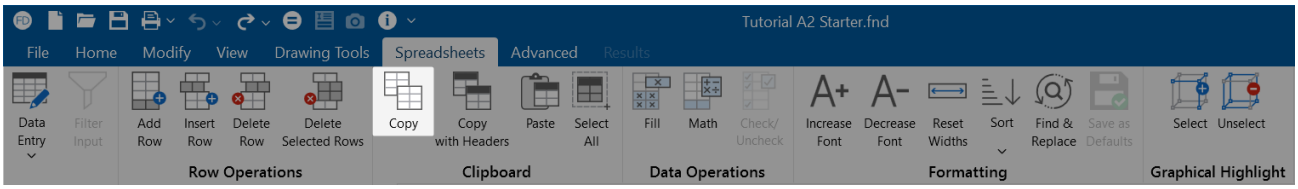


Moving and Copying Cell Contents

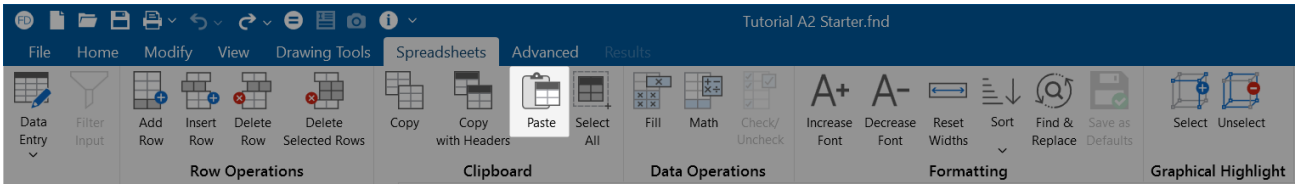
Standard Windows copy and paste functions are fully supported. By using the clipboard, you may move or copy any spreadsheet data to another location in the same spreadsheet, or to another spreadsheet (provided it fits). You may also copy data to or from another model or any other application that supports 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 the **Copy** button from the **Spreadsheet** ribbon.

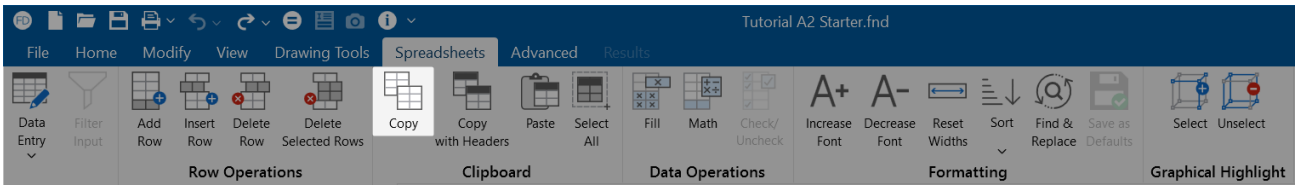


3. Click on the cell you wish to place the data in and click the **Paste** button.

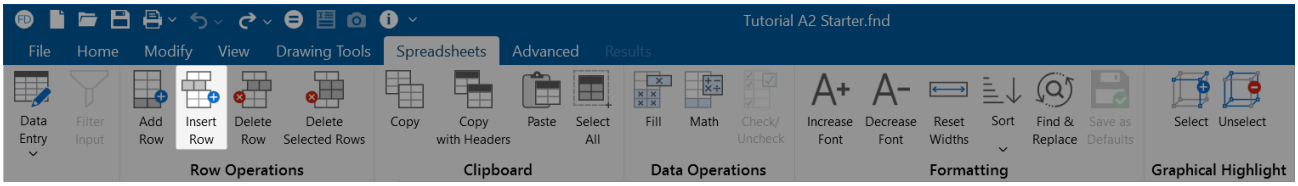


To Move or Copy Cells between Existing Cells:

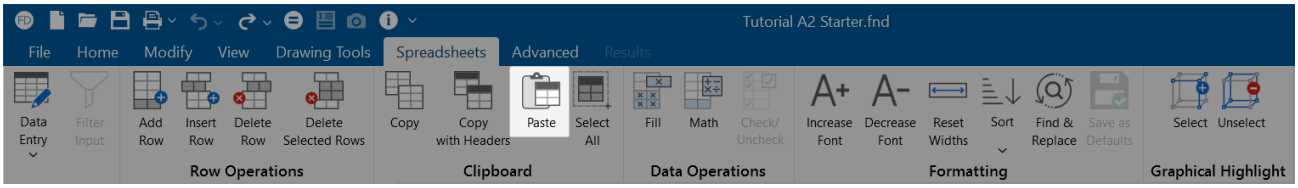
1. Select the cells you want to move or copy.
2. To copy the cells select the **Copy** button from the **Spreadsheet** ribbon.



3. Select a cell on the row above where you want to place the data.
4. Click the **Insert Row** button for each new line that is needed.

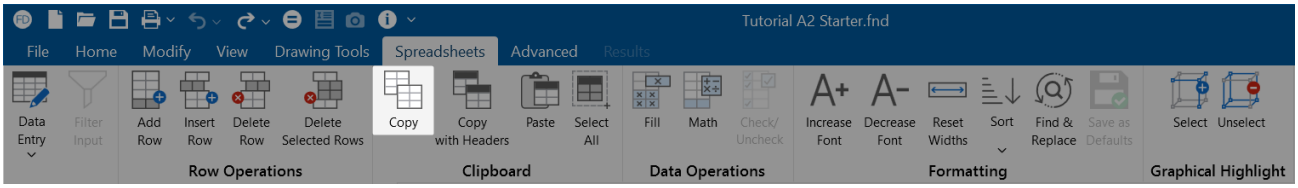


5. Click on the upper-left cell you wish to place the data in and click the **Paste** button.

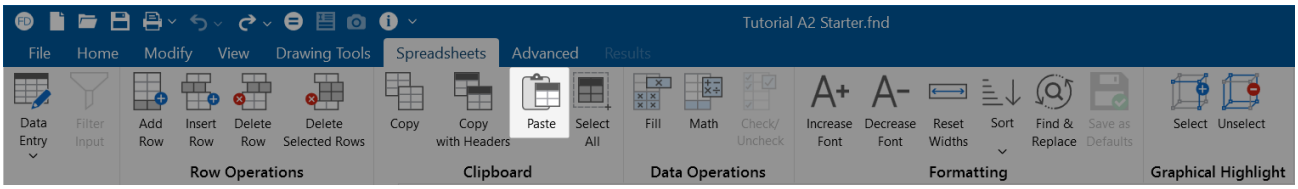


To Move or Copy Cells to Another File:

1. Select the cells you want to move or copy.
2. To copy the cells select the **Copy** button from the **Spreadsheet** ribbon.

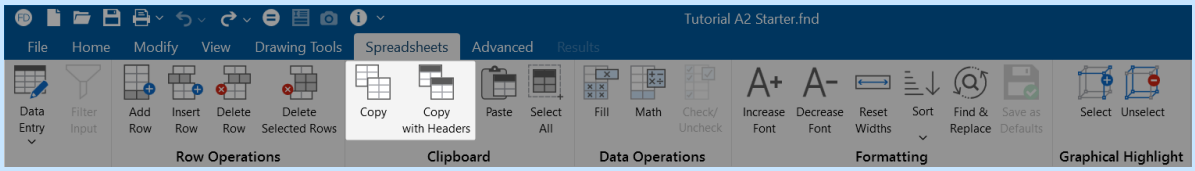


3. Open the file you wish to copy the data to.
4. Click on the cell you wish to place the data in and click the **Paste** button.



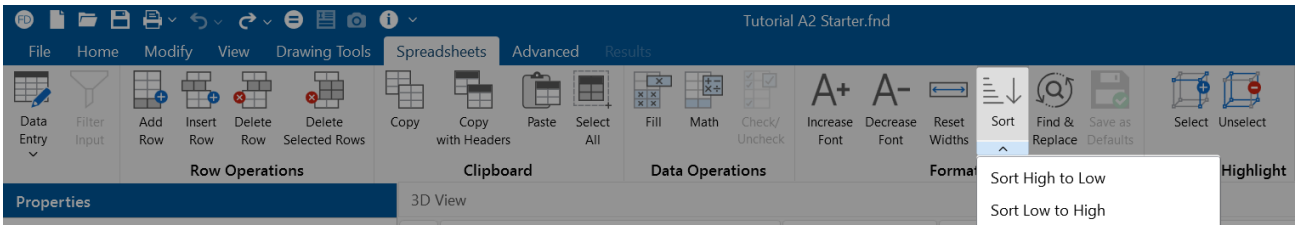
Note:

- Your data stays in the clipboard until you copy new data. You may repeat step 4 to move or copy data to multiple locations.
- When you copy information, you have the option to **Copy** or **Copy with Headers**. This is helpful if you are copying information to a spreadsheet-type program.

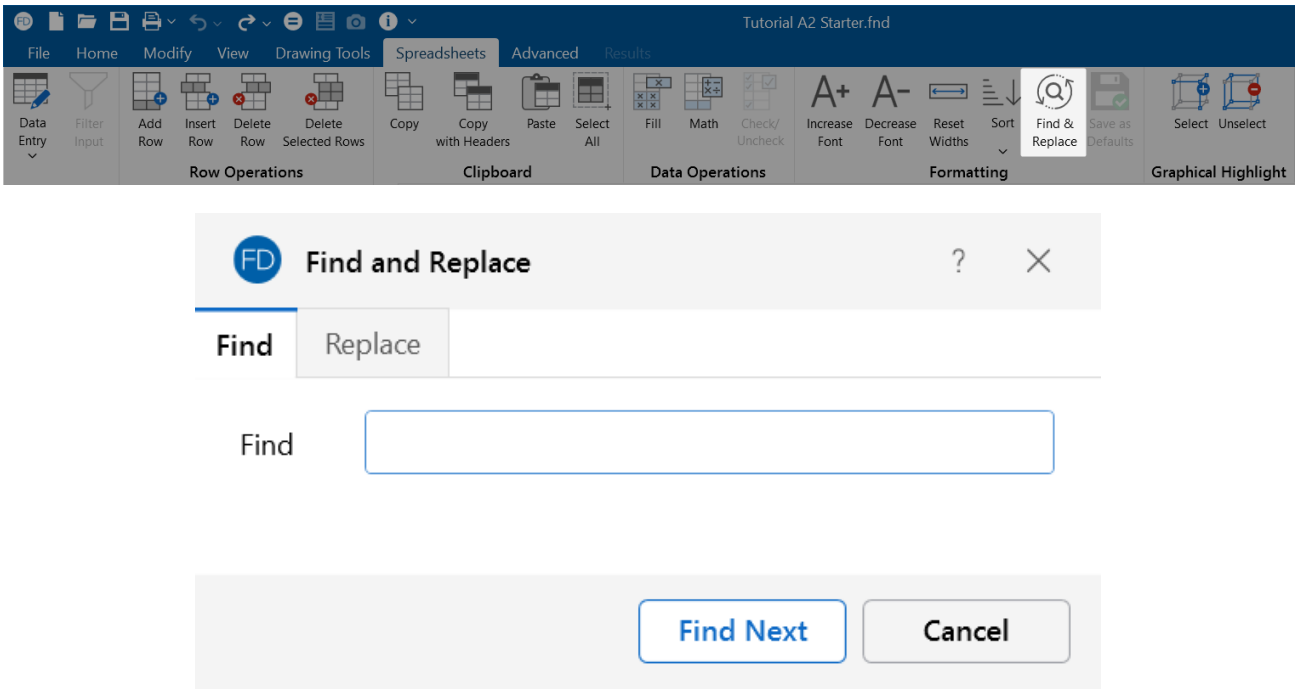


Sorting and Finding in Spreadsheets

You may sort spreadsheets by the values in most columns. Simply click in the column you wish to sort, click on the **Sort** button and choose the sorting method.



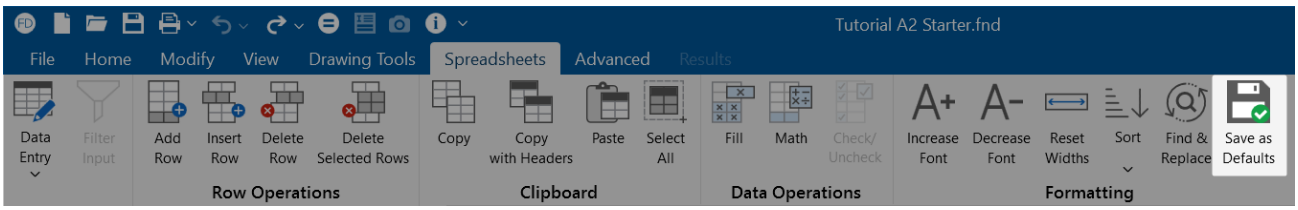
To locate or find a specific point, beam, or slab while in a spreadsheet, click the **Find & Replace** button on the **Spreadsheet** tab of the ribbon.



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.

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.



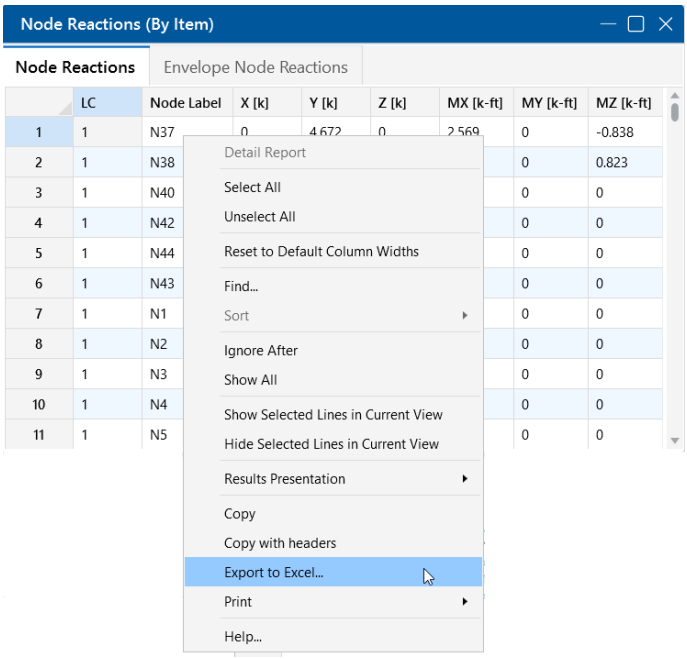
The data is saved for the current, active, spreadsheet only and affects no other open or closed spreadsheets.

Export to Excel Spreadsheet

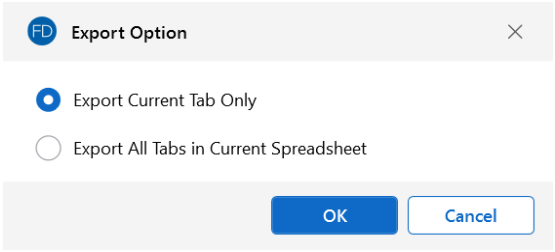
For all results spreadsheets, the program is capable of exporting them into an Excel Spreadsheet.

The process is as follows:

1. Right-click and select the **Export to Excel** option.

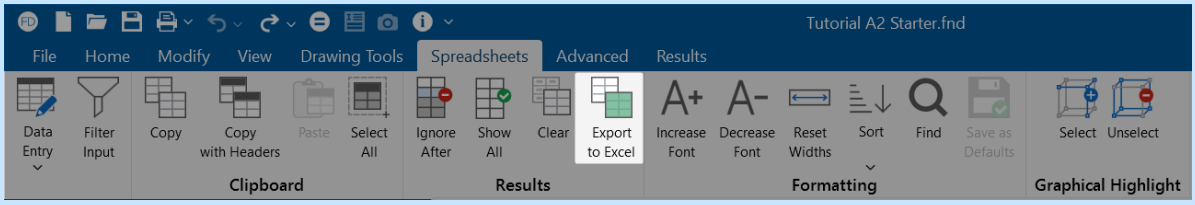


2. Choose if you want to export the current spreadsheet tab or all tabs.



3. Select an Excel file name and file save location and export.

Note: Export to Excel can also be used by clicking the button on the **Spreadsheets** ribbon.

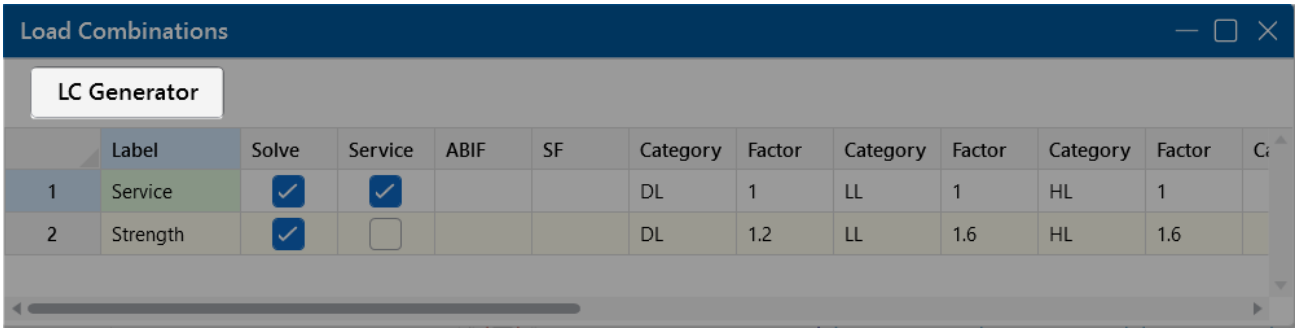


Special Spreadsheet Functions

There are special functions that help you with a particular spreadsheet.

Load Combinations Spreadsheet

You may click the **LC Generator**  button to have the program generate load combinations based on a variety of Building Codes. For more information, see Generating Building Code Combinations.

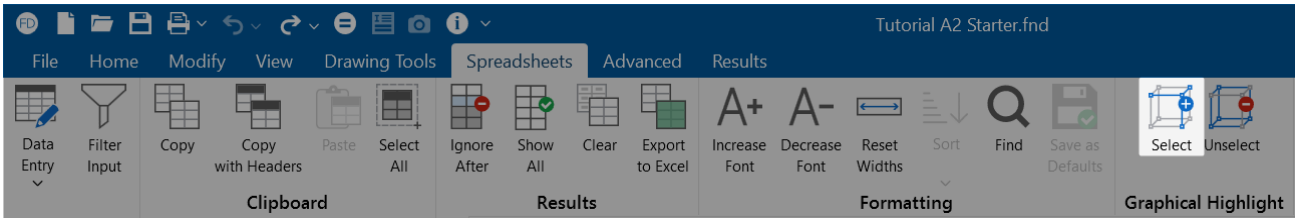


Selecting Items in the Model View

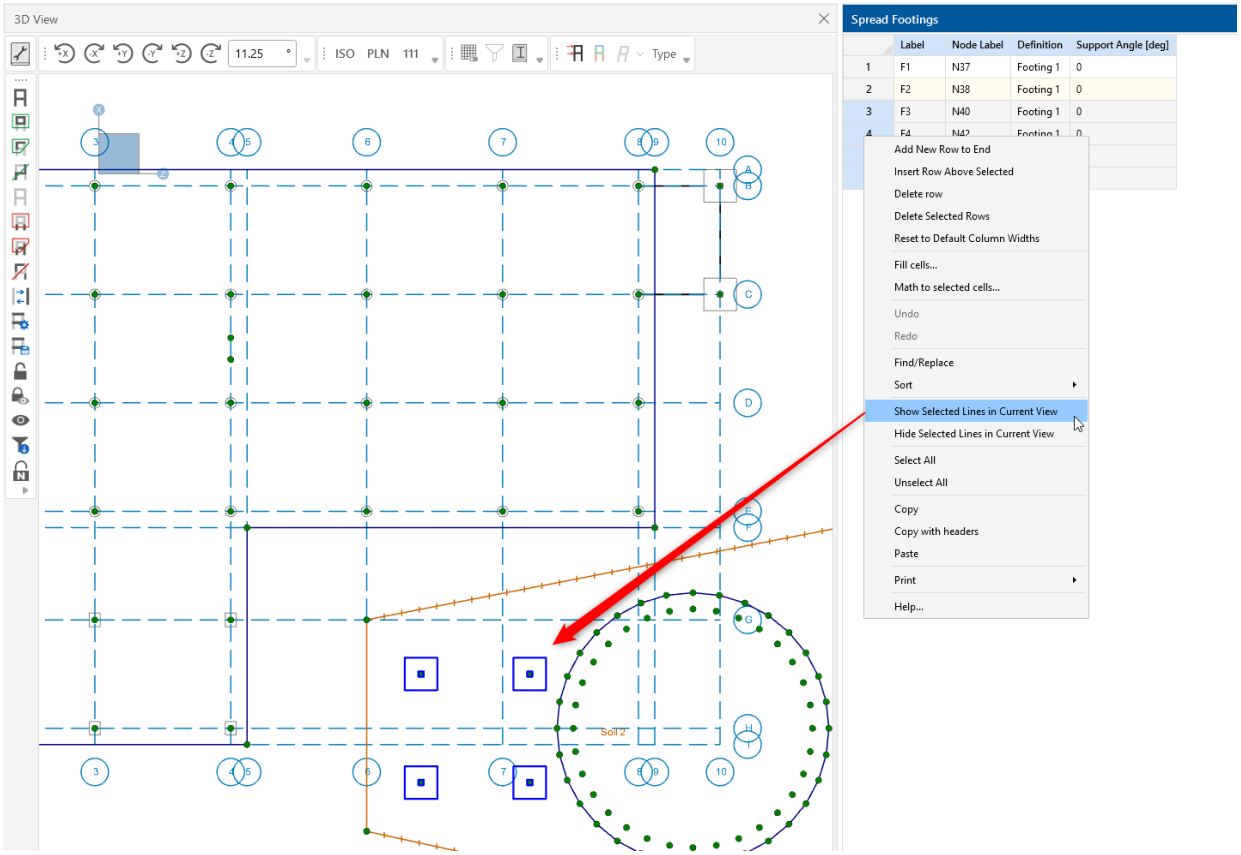
In certain results spreadsheets, there is the capability to have elements selected graphically.

The process is as follows:

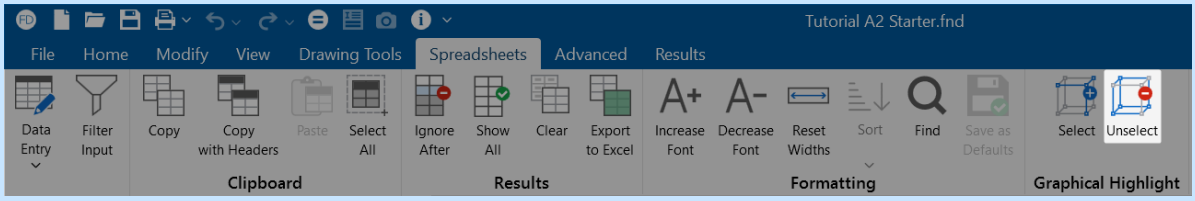
1. Open the desired results spreadsheet.
2. Highlight the lines from your spreadsheet that you wish to view graphically, by clicking on the number of those lines on the far left.
3. Click the **Select** icon in the **Graphical Highlight** section of the **Spreadsheets** ribbon.



4. Only those elements highlighted will be selected graphically.



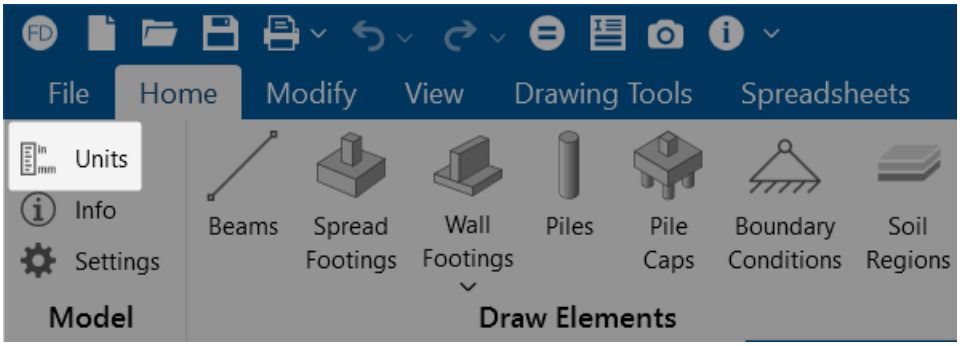
Note:
The inverse process can be used by selecting **Unselect** in the **Graphical Highlight** section of the **Spreadsheets** ribbon.

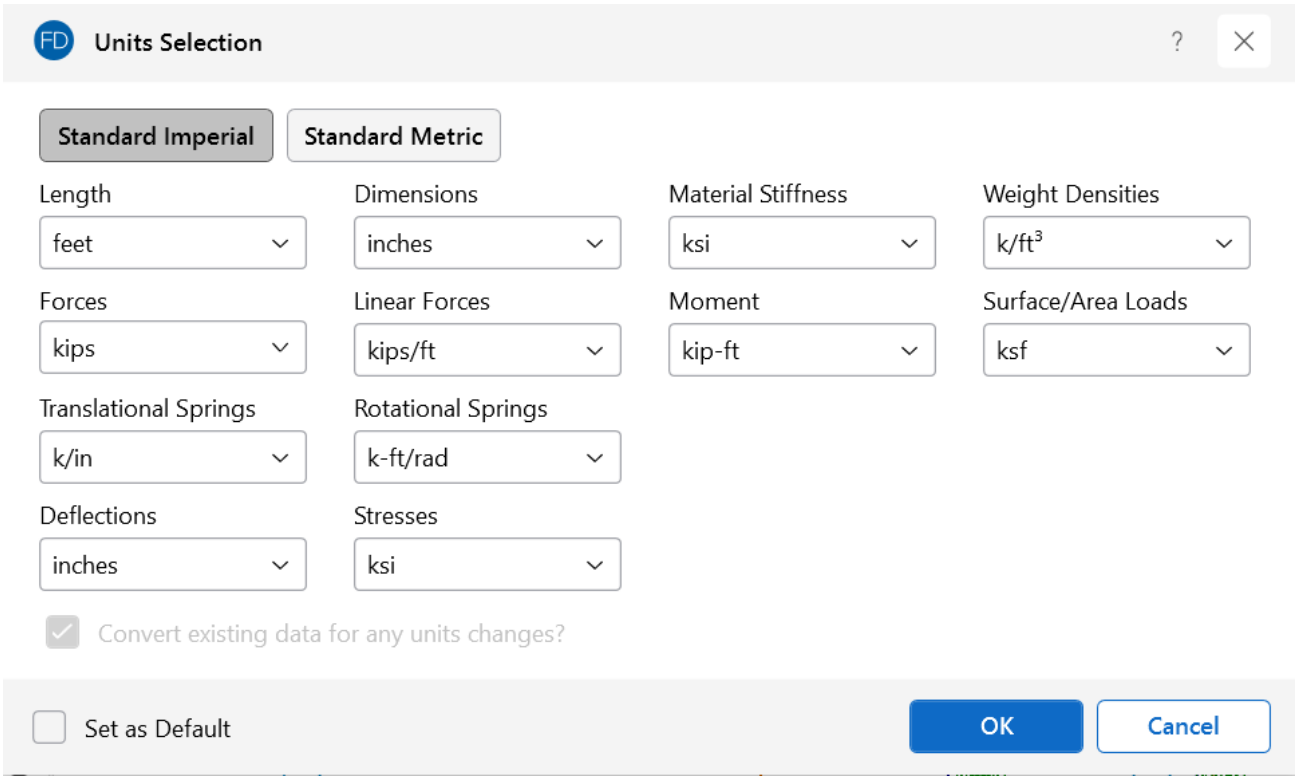


Units

You can work with imperial units (kips, inches, etc.), metric units (KN, meters, etc.), or any combination of the two. The current units appropriate for each data item are shown at the top 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 set for you. To do this, simply enter the information that you want to save and check the **Save as Default** box and click **OK**.

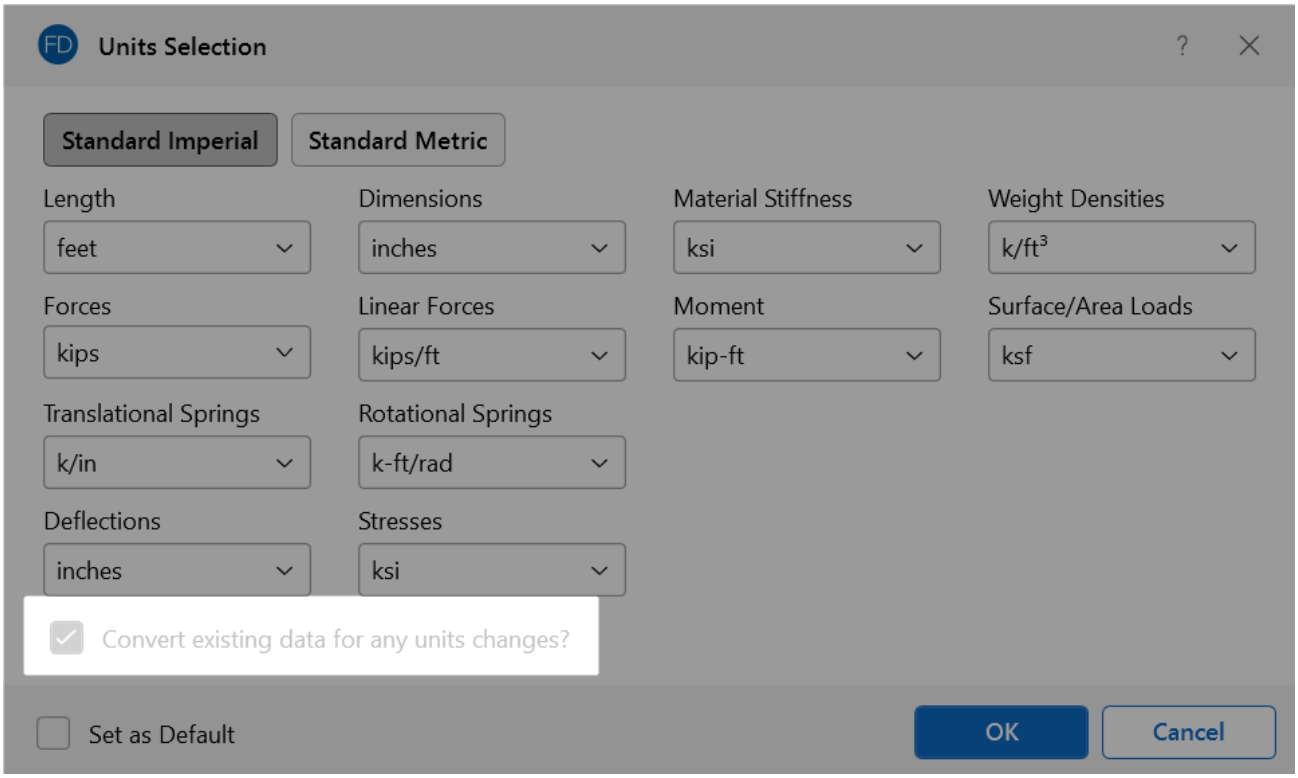




To Change Units:

1. Click the **Units** button in the **Model** section of the **Home** tab.
2. Specify the units you want for each item in the drop-down lists.

   Standard units systems are preset and may be specified by clicking the **Standard Imperial** and **Standard Metric** buttons.
3. If you do not wish to convert values already entered, clear the check box for **Convert existing data for any units changes?**.



Standard Imperial Units

This is the units system currently prevalent in the United States. 'Feet' are used for location entries, such as node coordinates and load locations, and 'Inches' are used for section property entries, such as area and moment of inertia. Force and weight units are in 'kips', where 1 kip = 1000 pounds. Stress units are in 'ksi' (kips per square inch).

Standard Metric Units

This units system uses 'Meters' for location entries and 'Millimeters' 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 and weight units are in 'Kilograms'.

Units Specifications

The following are the unit specifications and their applications:

Measurement	Usage
Lengths	Coordinates, Load Locations
Dimensions	Shape Properties, Slab Thickness
Material Stiffness	E, G
Weight Densities	Material Density
Forces	Loads, Forces
Linear Forces	Distributed Loads
Moments	Loads, Forces
Area Loads	Area Loads
Translational Springs	Supports
Rotational Springs	X Rot, Y Rot, Z Rot Boundary Conditions
Temperatures	Thermal Coefficient, Temperatures
Deflections	Deflections, Displacements
Stresses	Fy, Allowable and Actual Stresses

Wall Footings - Modeling

Here we will talk about the steps in modeling a wall footing and get into advanced wall footing modeling considerations. We will also talk about which type of wall footing would be appropriate for your application.

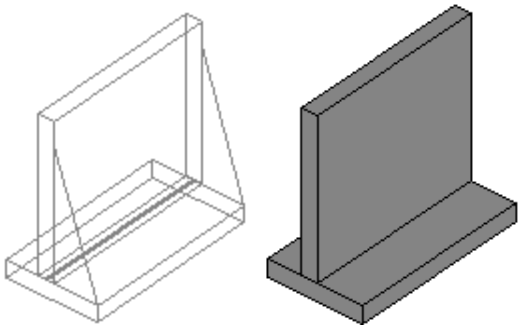
- For more information on defining wall parameters, see Wall Footing Definitions.
- For more information on calculation details, see Wall Footings - Design.
- For more information on the output, see Wall Footing Results.

Wall Footing Types

There are three types of wall footings you can model in RISAFoundation: retaining walls, strip footings and generated slab/stem wall combination. See the Draw Wall Footings section for how you select each option.

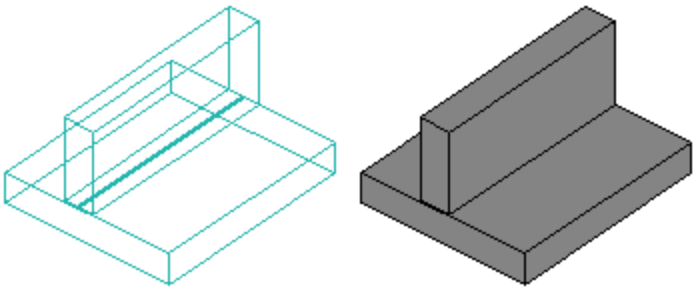
Retaining Wall

This is the traditional retaining wall. This type of wall will perform all stability checks (sliding, overturning and soil bearing) due to hydrostatic pressures and any superimposed load on the wall. This wall type is a single entity in that the wall and footing are considered together and defined in the Wall Footing Definitions Spreadsheet.



Strip Footing (Gravity)

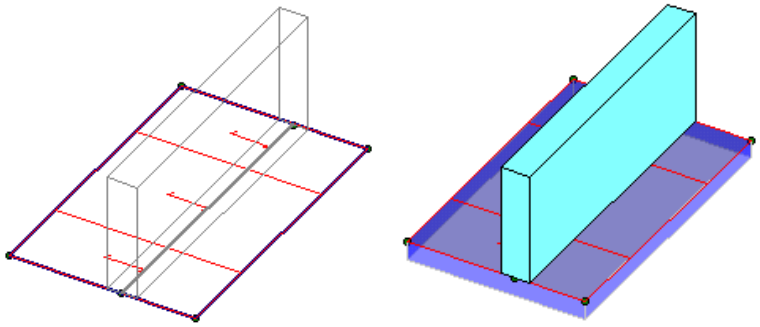
This is a simplified version of the retaining wall and would be used in cases where you have a simple strip footing layout. Hydrostatic pressures on the wall are not a concern here. This type of wall footing pares down the input and output in the program, removing a lot of the complexity necessary for retaining walls, but not required for a simple strip footing. This wall type would generally be used for simple gravity loads. This wall type is also a single entity in that the wall and footing are considered together and defined in the Wall Footing Definitions Spreadsheet.



Strip Footing (Lateral/Custom)

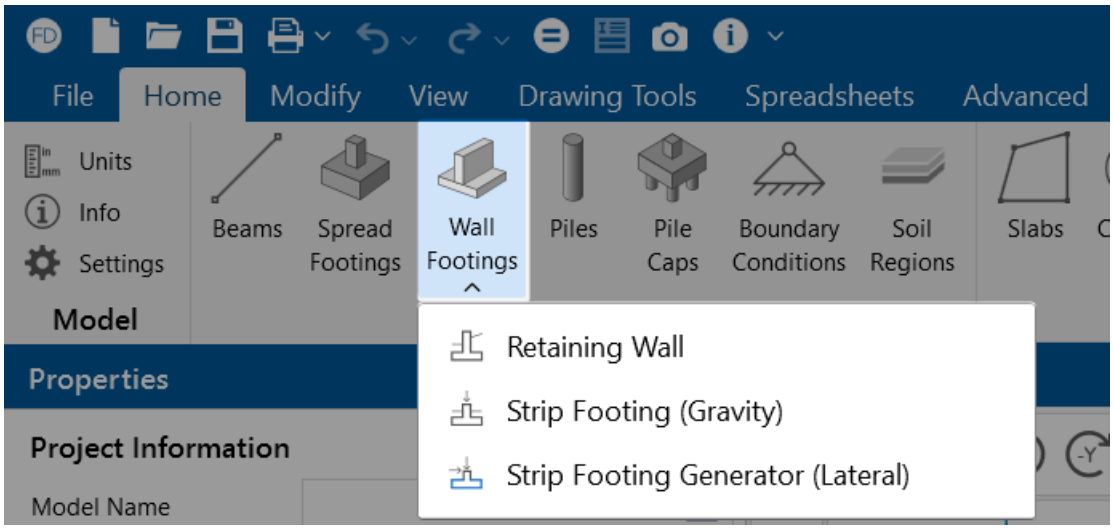
This is a wall type that is really a generation utility that will create a slab, slab design strip(s) and a stem wall to represent your wall footing. This type of footing would not be controlled by the Wall Footing Definitions, but rather by each individual entity (slab, slab design strip and stem wall) separately. This option generates these elements one time and then allows you to edit/modify these elements as necessary.

The main purpose of this type of wall is to allow for any customization and to allow the program to perform a finite element analysis (FEA) of the footing. The other two wall types do not perform an FEA on the footing itself.



Draw Wall Footings

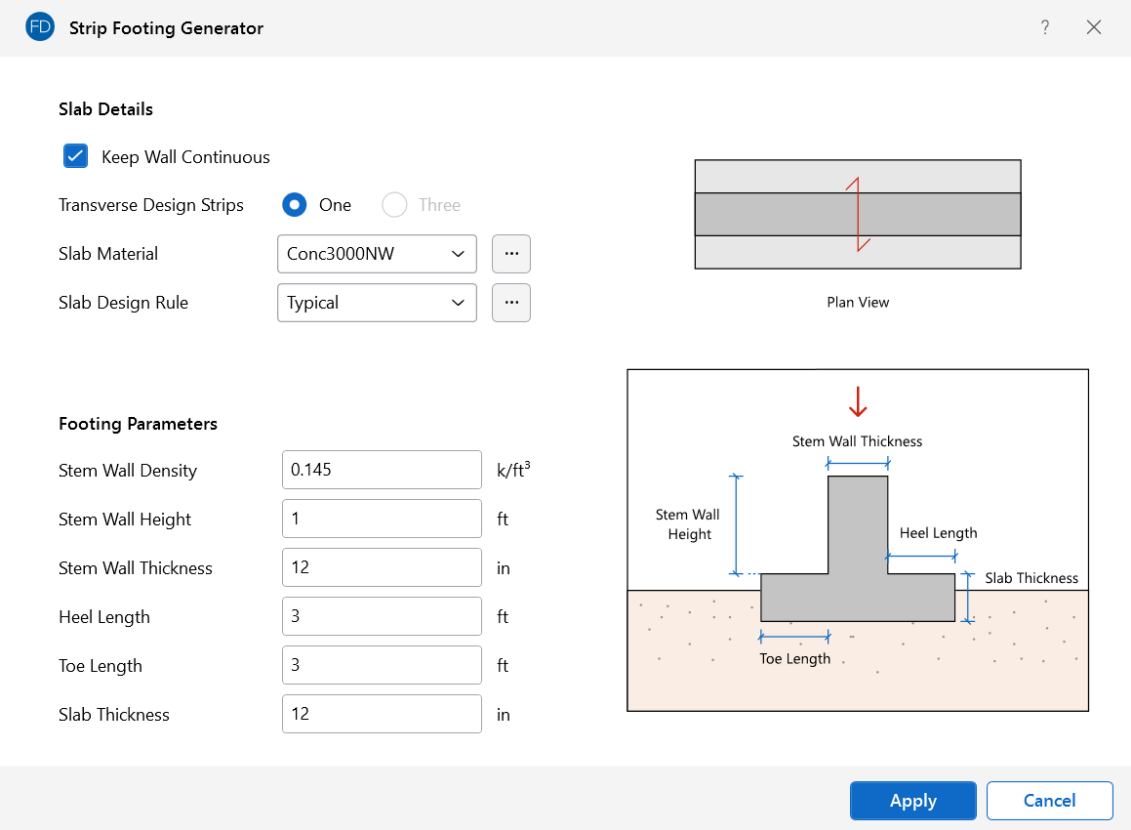
You can create new wall footings using a Drawing Grid, Project Grid or drawing point-to-point clicking existing points. You can set all of the wall footing properties up front, or you can modify these properties after you draw the wall footing. Graphically modifying properties is discussed in the next section.



To draw wall footings:

1. If there is not a model view already open, go to the **View** ribbon.
2. Click the **Open 3D Views** icon in the **Window** section to open a new view.
3. If you are not drawing between existing points, you need to create a drawing grid or define nodes in the **Node Coordinates Spreadsheet**.
4. Go to the **Home** ribbon.
5. Click the **Wall Footing** icon in the **Draw Elements** section.
6. Choose the wall footing type, including the **Definition**, if applicable.
    - If you are modeling a **Strip Footing (Lateral)**, you have the option to make the wall footing continuous by checking **Keep Wall Continuous** in the **Strip Footing Generator** dialog (shown below). Here, you can input the properties of your composite slab, slab design strip and stem wall entity. This dialog is similar to the

Wall Footing Definition Editor.



7. In the **Properties** panel, set the wall footing properties.
8. To start drawing wall footings, click on **points** or **grid intersections** with the left mouse button. The coordinates of the closest point or grid intersection to your cursor are displayed on the screen near your cursor.

   The first click defines the Start Point of the first wall footing. The second click, and each click thereafter, defines the End Point of the first wall footing and also the Start Point of the next wall footing so that you can continue to draw. To stop drawing, click the **right mouse button**. You can then start drawing somewhere else with the left button.

   The new wall footings are shown on screen and recorded in the **Wall Footings Spreadsheet**.
9. To stop drawing altogether, click the **right mouse button** again or press the **Esc** key.

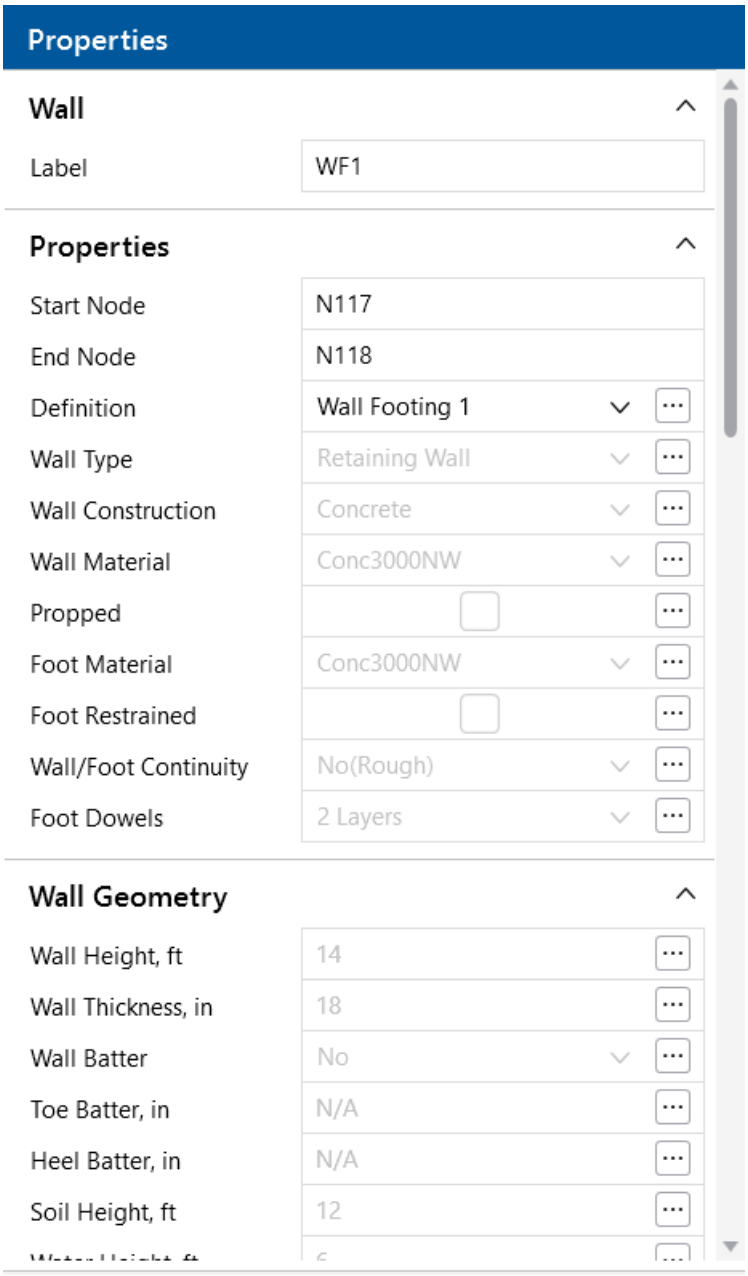
Note:

- If you have not created Wall Footing Definitions ahead of time, you can create a new definition by clicking any of the ... (ellipsis) buttons in the **Properties** panel. This will open up the Wall Footing Definition Editor dialog.
- You may undo any mistakes by clicking the **Undo** icon in the **Quick Access toolbar**.

Modify Wall Footings

The instructions below detail how you modify **Retaining Walls** and **Strip Footings (Gravity)**. Modifying **Strip Footings (Lateral/Custom)** requires you to edit the individual entities (slab, slab design strip(s) and stem wall) that make up that wall footing.

For non-custom wall footings, you may view and edit the wall footing data in the Wall Footings Spreadsheet. You may also click a single wall footing or select multiple footings to view and edit the properties in the member **Properties** panel. You can use the tools in the **Modify** tab to graphically modify the wall footings in the 3D View.



To modify wall footing properties:

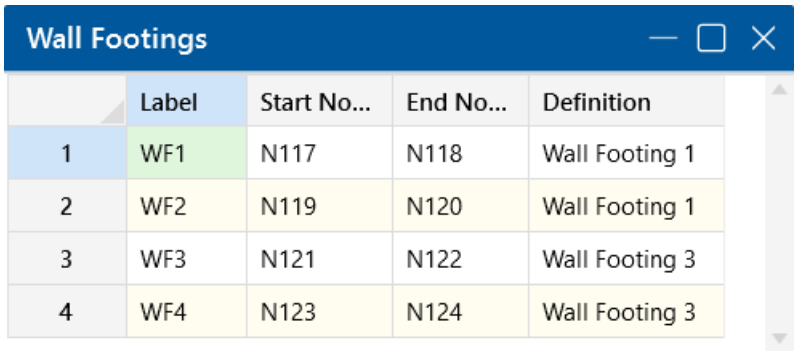
1. If there is not a model view already open, go to the **View** ribbon.
2. Click the **Open 3D Views** icon to open a new view.
3. Click a single wall footing, or select multiple wall footings that you wish to modify.
4. If you have multiple elements selected, specify which element type to view properties for

using the **Selection Properties** drop-down in the **Properties** panel.

5. Set the parameters for the wall footings within the **Properties** panel.

Wall Footings Spreadsheet

The **Wall Footings Spreadsheet** records the properties for the wall footing elements and may be accessed by selecting **Wall Footings** from the **Data Entry** drop-down on the **Spreadsheets** tab of the ribbon, or by clicking **Wall Footings** from the **Data Entry** section of the **Explorer** panel.



	Label	Start No...	End No...	Definition
1	WF1	N117	N118	Wall Footing 1
2	WF2	N119	N120	Wall Footing 1
3	WF3	N121	N122	Wall Footing 3
4	WF4	N123	N124	Wall Footing 3

Label

This gives a unique label to each wall footing in the model.

Start Point/End Node

This defines the start and end location nodes for the wall footing. The nodes can be modified directly in the spreadsheet.

Definition

This is the Wall Footing Definition to be used. This definition gives all of the other information required for design of the wall footing.

Wall Footing Modeling Tips

Here are some tips to think about when modeling wall footings.

Wall Footing Orientation

When drawing wall footings, click on two points to define the length of the wall footing. The wall (with footing) will be drawn so that the centerline of the wall lines up along the two nodes defining the wall. These elements, though drawn in a 3D rendering, are really just line elements.



Figure 1: Plan View of Wall Footing

When drawing a wall footing, the orientation of the local axes for the wall will always have the local x-direction parallel to the line that defines the two nodes drawn.

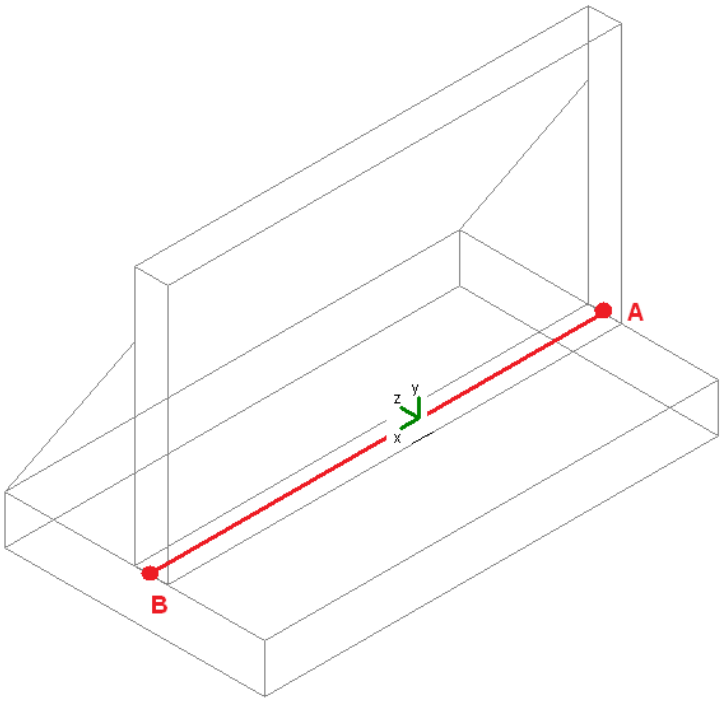


Figure 2: Isometric View of Wall Footing

Here, we can also see the orientation of the positive z-axis and that the soil is on the positive z-axis side of the wall footing. Keep this in mind when drawing your walls.

For example, if you are drawing basement walls for a structure, make sure to draw in a counter-clockwise fashion. This will ensure that the soil is on the exterior side of the walls. Note that this is only applicable for **Retaining Walls**.

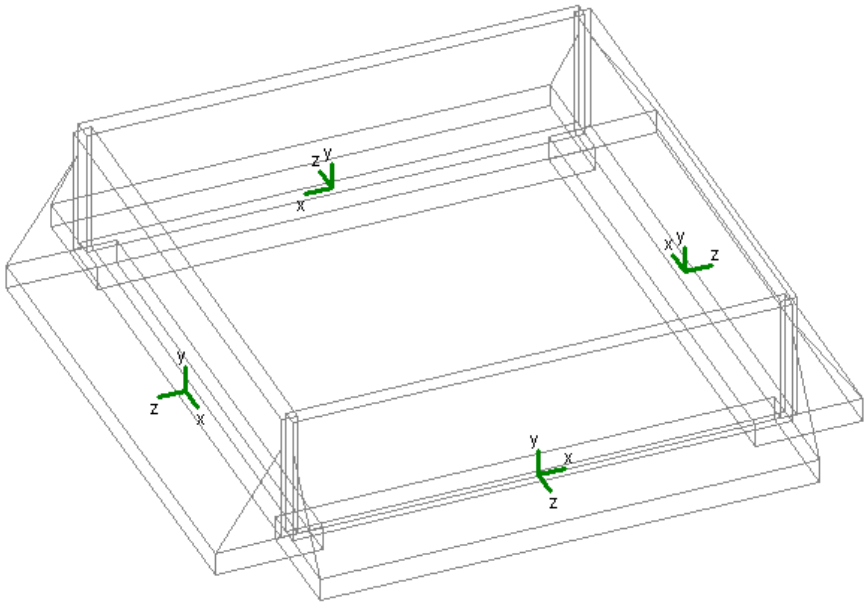


Figure 3: Basement Walls Drawn Counter-Clockwise

Note: If you have drawn your retaining wall footing such that the soil is on the wrong face of the wall, you can switch the Start and End nodes so that the retaining wall is flipped to the correct orientation.

Load Categories

Once you have defined all of the required properties in the Wall Footing Definitions Spreadsheet, the program calculates the weights, hydrostatic loads, soil pressures, etc, and applies them to the wall for analysis and design purposes. The program also places these loads in load categories for you automatically.

- Dead Loads (DL): Weight of concrete and weight of soil.
- Live Loads (LL): Surcharge loads (applicable only to retaining walls).
- Hydrostatic Loads (HL): Any lateral pressure/hydrostatic loads (applicable only to retaining walls).

Note:

- Any loads applied to the top of the wall footing will be considered under the load category they are applied with.
- If you have a sloped backfill on your retaining wall, a portion of the hydrostatic loads will be applied in the vertical direction of your wall. The cosine of these forces will be applied horizontally as a pressure. The sine of these forces will be applied as a vertical force that is conservatively lumped at the top of the wall.
- Water pressure (if there is a water table present) will always be assumed to act horizontally, even in the presence of a sloping backfill.
- It is possible for the horizontal component of hydrostatic force to act ABOVE the height of the wall. All loading that occurs above this height will be lumped as a point load right at the top of the wall.

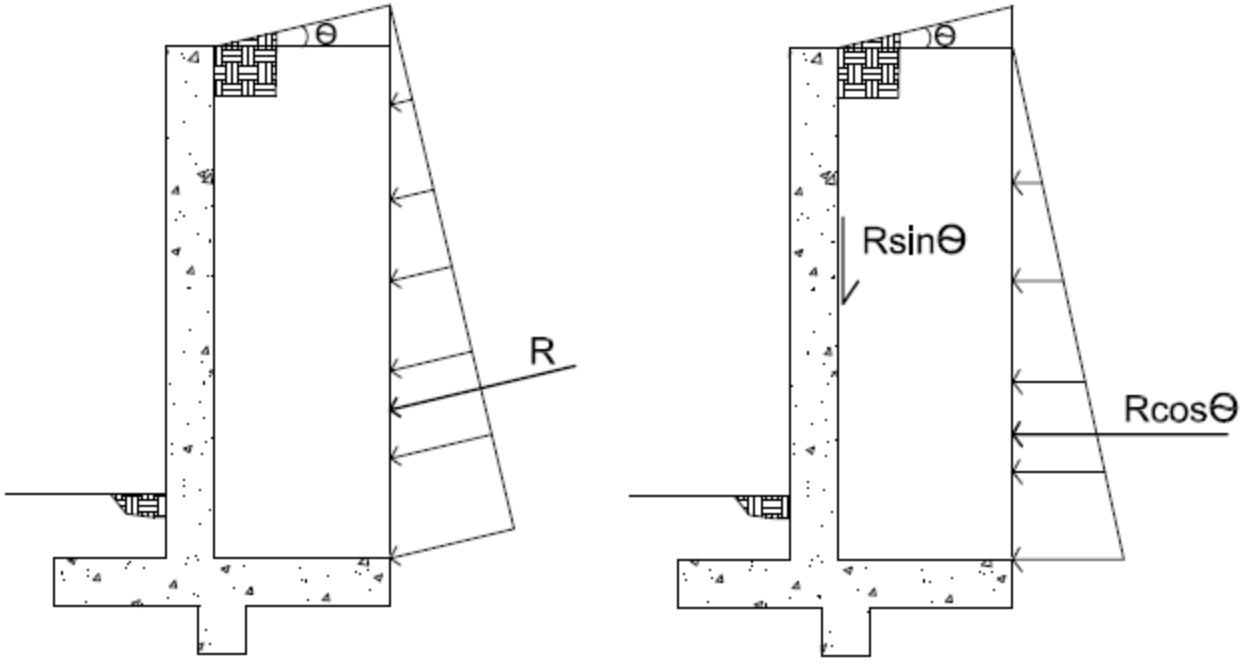
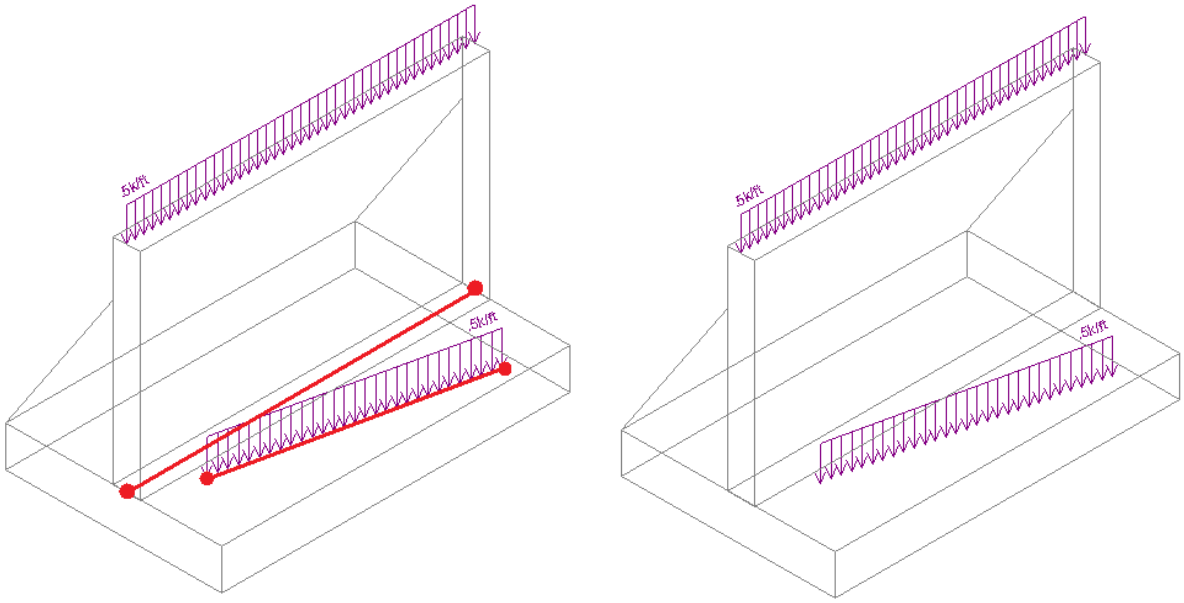


Figure 4: Hydrostatic Loads with Sloped Backfill

Loading of Wall Footings with Externally Applied Loads

Wall footings will accept externally applied loads to them, but only loads applied to the top of the wall. To apply loads to a wall footing, you must apply the loads to the centerline at the base of the wall.



Figures 5A & 5B: Wall Footing with External Loads

In the figures shown above, we can see (from the red lines) how the loads were applied and how they ended up on the wall footing. The distributed load applied along the centerline of the wall ended up being applied to the top of the wall and will be analyzed/designed as such. The other load did not line up with the centerline of the wall, so it will not be considered at all in the wall footing design.

Partial length distributed loads (even trapezoidal) and point loads work in the same fashion.

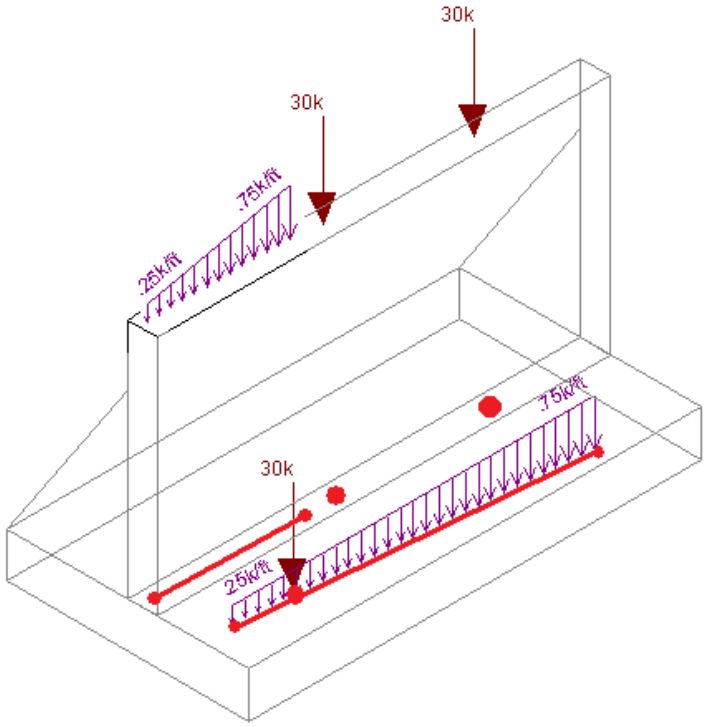


Figure 6: Wall Footings with Trapezoidal Dist. Loads and Point Loads

Looking at the model in a Plan View while drawing can make modeling these loads much easier.

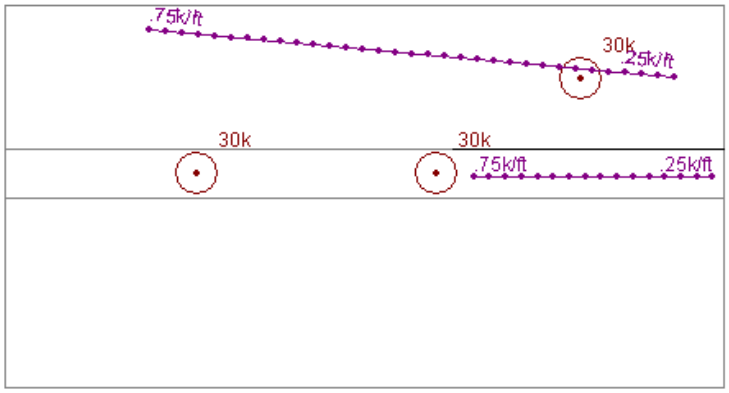


Figure 7: Wall Footing Loading shown in a Plan View

Note: If you wish to apply loads adjacent to the wall to work as surcharge loading, apply the loads in the Wall Footing Definitions - Soil tab in the **Heel Surcharge** column.

If you have two adjacent wall footings (defined by the same node), applying a point load at the common node will always place that load on the wall footing that comes first in the Wall Footings Spreadsheet.

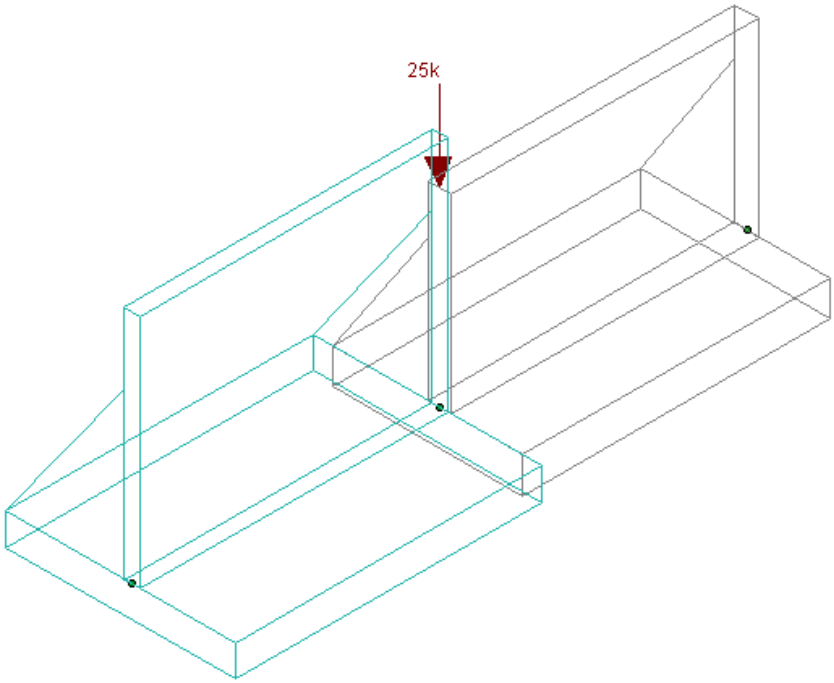


Figure 8: Two Adjacent Wall Footings with a Point Load

Connectivity with Other Elements in the Model

Non-custom **Wall Footings** in RISAFoundation have no connectivity to any other elements in the model, not even adjacent retaining walls. So while it is possible to connect beams and place slabs adjacent to the wall footing, no forces will be transferred between the two. Here are some examples where wall footings and other elements may seem to come into contact with each other.

- **Beams and Wall Footings**: Here it appears that the wall footing and a beam are in the same space. However, if any load was applied along the line where the beam and the wall footing overlap, the load would only be taken by the beam and none of the load would be seen by the wall footing.

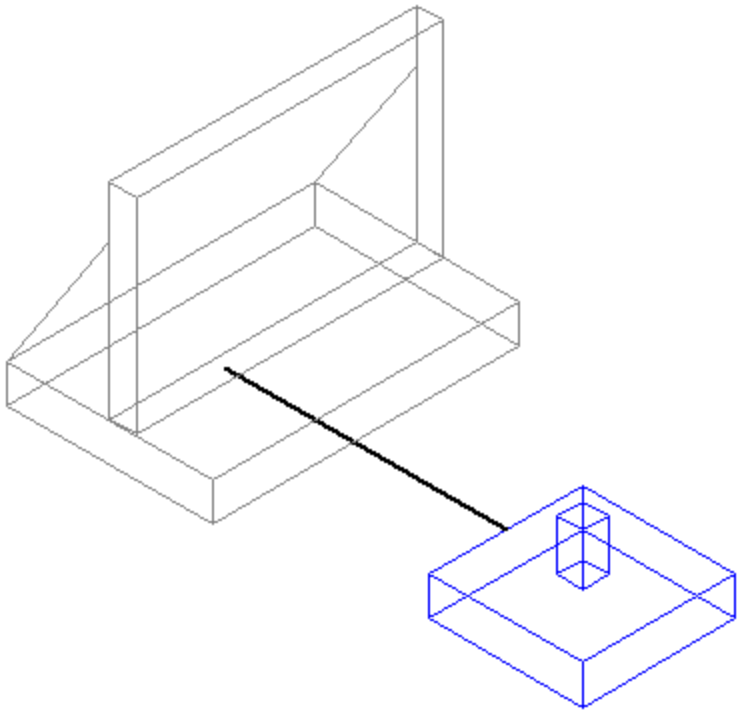
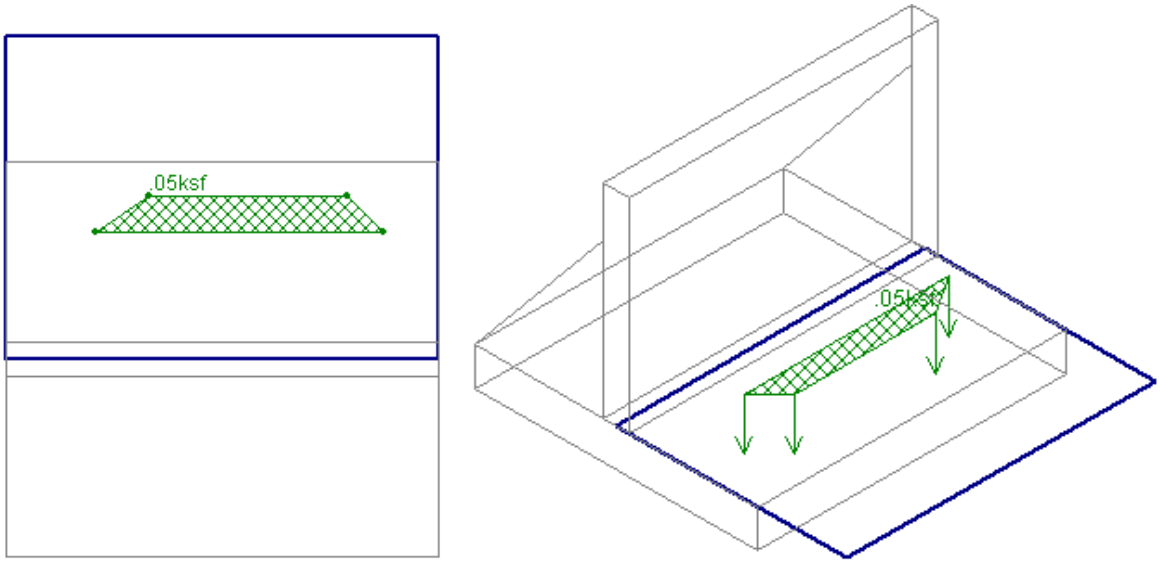


Figure 9: A Beam Overlapping with a Wall Footing

- **Beams and Slabs**: Slabs can be drawn so that they are adjacent with wall footings. Any load that is defined on the area of overlap of a slab and a wall footing will only be taken by the slab. The wall (and footing) will have no interaction with the slab.



Figures 10A & 10B: A Slab Overlapping with a Wall Footing

- **Other Items that Overlap**: Many other overlapping situations in the program are not handled and an error will be given at solution.

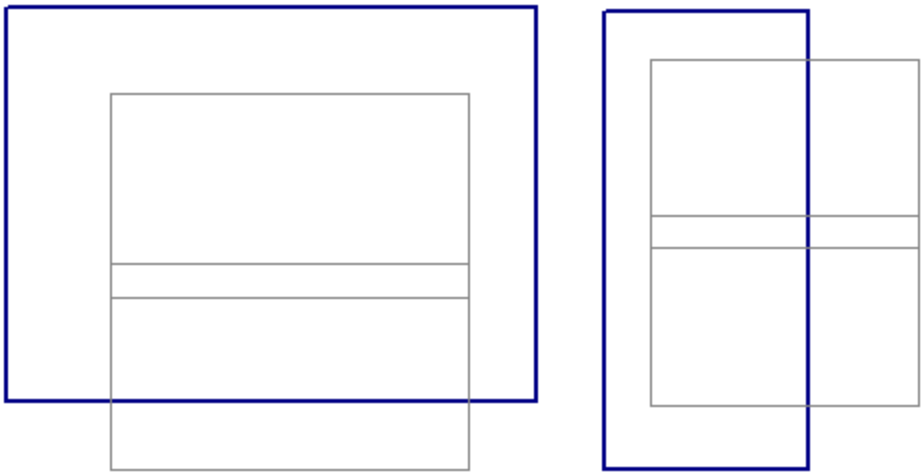


Figure 11: A Slab Drawn Overlapping the Wall Portion

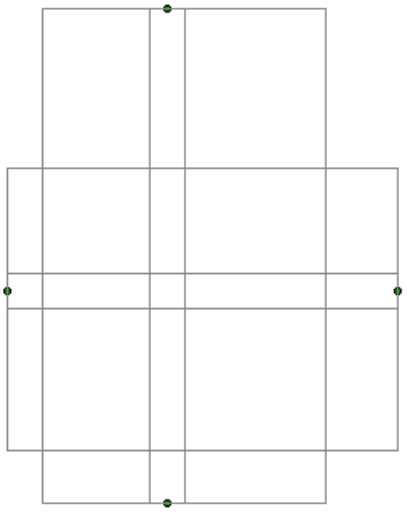


Figure 12: Two Wall Footings Where the Walls Cross

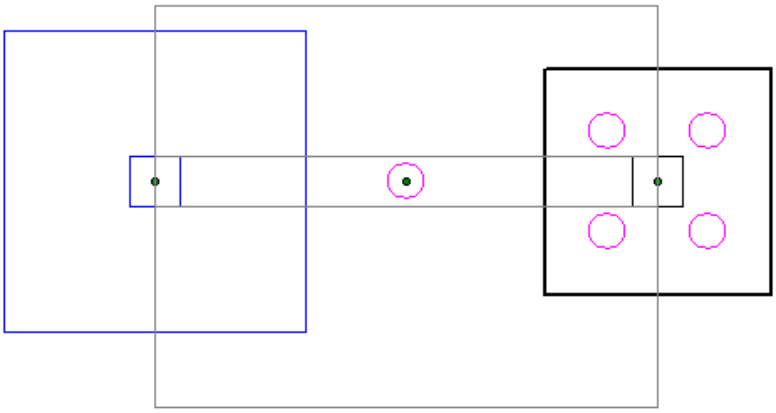


Figure 13: A Pile, Pile Cap, or Footing, that Intersects the Wall Footing Centerline

Wall Footing Definitions

Wall footings consider soil loading and exterior loads for the concrete design of the wall stem and footing. The soil and exterior loads are also considered for overturning and sliding serviceability.

- For more information on modeling requirements, see Wall Footings - Modeling.
- For more information on calculation details, see Wall Footings - Design.
- For more information on the output, see Wall Footing Results.

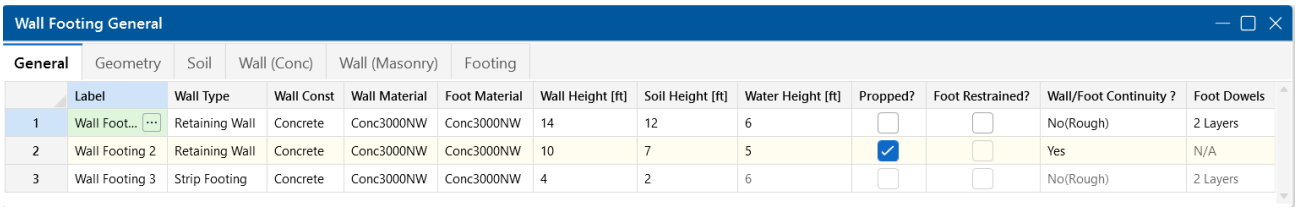
Here we will explain the **Wall Footing Definitions Spreadsheet** and **Wall Footings Spreadsheet**. Both can be found on the **Data Entry** section of the **Explorer** toolbar or from the **Data Entry** drop-down on the **Spreadsheets** tab of the ribbon. Note that you can have multiple wall footings in a given project that may require multiple definitions.

The **Wall Footing Definitions Spreadsheet** and **Wall Footings Spreadsheet** are always synchronized. Making changes from one will update the other.

Note:
- See Wall Footing Definition Editor for more about how this information can be created from a graphic dialog.
- Items with an '*' in the following sections are only applicable for retaining walls, not strip footings.

General Tab

The following inputs can be made on the **General** tab of the **Wall Definitions spreadsheet** for wall footings:



	Label	Wall Type	Wall Const	Wall Material	Foot Material	Wall Height [ft]	Soil Height [ft]	Water Height [ft]	Propped?	Foot Restrained?	Wall/Foot Continuity ?	Foot Dowels
1	Wall Foot...	Retaining Wall	Concrete	Conc3000NW	Conc3000NW	14	12	6	☐	☐	No(Rough)	2 Layers
2	Wall Footing 2	Retaining Wall	Concrete	Conc3000NW	Conc3000NW	10	7	5	☑	☐	Yes	N/A
3	Wall Footing 3	Strip Footing	Concrete	Conc3000NW	Conc3000NW	4	2	6	☐	☐	No(Rough)	2 Layers

Label

This label provides a unique name for each wall footing definition. Clicking the ... (ellipsis) button will open the Wall Footing Definition Editor. See below for more information on this dialog.

Wall Type

This allows you to choose between a retaining wall element and a strip footing element. See the Wall Footings - Modeling topic for more information on this.

Wall Const

This allows you to choose between a concrete or masonry retaining wall.

Wall Material

This is the wall material for the wall. If a concrete wall is selected, the concrete materials will be available. Similarly, masonry materials will be available when a masonry wall has been selected.

Foot Material

This is the concrete material for the footing.

Wall Height

This is the wall height from the top of the footing.

Soil Height

For retaining walls, this is the height of the backfill behind the wall taken from the top of the footing. For strip footings, this is the soil height on both sides of the wall above the footing.

Water Height*

This is the height of water behind the wall taken from the base of the footing. If this is input as zero (fully drained condition), then the water table will be assumed to occur below the bottom of footing or shear key.

Propped?*

This defines whether the top of the wall is propped. If the wall is propped, it is assumed the base is also restrained laterally and sliding and overturning checks for the wall are omitted.

Foot Restrained?*

This defines whether the bottom of the footing is restrained laterally (i.e. a basement slab is poured adjacent to the wall). If so, then the sliding check will be omitted.

Wall/Foot Continuity?*

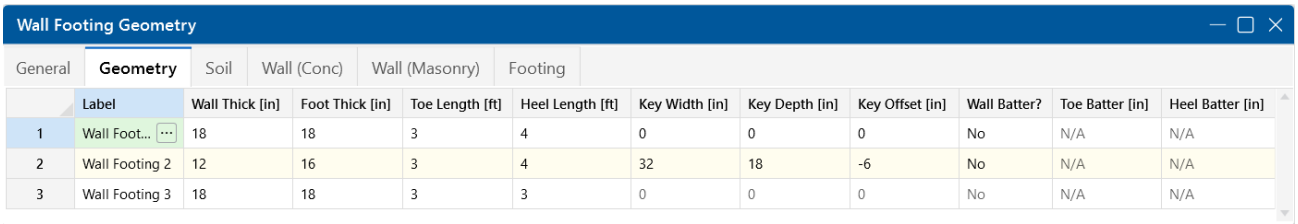
This defines whether the wall and footing are assumed to be monolithic. If the wall and footing are monolithic (the **Yes** option), then we will not do a shear friction check for the dowels from the footing to the wall. If the wall and footing are not monolithic, then we will do a shear friction check. Smooth versus rough defines whether the footing at the location where the wall is poured has been intentionally roughened or not. This affects the shear friction calculation. For more information, see **Shear Friction Check** in the Wall Footings - Design topic.

Foot Dowels*

The number of layers defines whether there are dowels in both faces of the wall, or only the interior face (soil face). This input is only required if the footing and wall are NOT monolithic. If that is the case, then the shear friction check at the base of the wall requires an area of steel for the dowel bars. The program assumes that the dowels from the footing into the wall will match the wall reinforcement and calculates A_{vf} accordingly.

Geometry Tab

The following inputs can be made on the **Geometry** tab of the **Wall Definitions spreadsheet** for wall footings:



	Label	Wall Thick [in]	Foot Thick [in]	Toe Length [ft]	Heel Length [ft]	Key Width [in]	Key Depth [in]	Key Offset [in]	Wall Batter?	Toe Batter [in]	Heel Batter [in]
1	Wall Foot...	18	18	3	4	0	0	0	No	N/A	N/A
2	Wall Footing 2	12	16	3	4	32	18	-6	No	N/A	N/A
3	Wall Footing 3	18	18	3	3	0	0	0	No	N/A	N/A

Wall Thickness

For concrete walls, it is possible to batter the wall. This thickness excludes the battered portion and would be the thickness at the top of the wall.

For masonry, this field is the block nominal thickness and must be set to 4, 6, 8, 10 or 12 inches. 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.

Note: Even if metric units are used for a masonry wall, the nominal inch dimensions are required. The program will use these values in their converted metric form.

Foot Thickness

This is the footing thickness.

Toe/Heel Length

This is the length of the footing from the outer/inner face of the wall.

Note: If the stem wall is battered, these lengths are from the full battered width of the wall.

Key Width/Depth*

These are the dimensions of the shear key, if there is one. If either of these dimensions is zero, then no key will be considered.

Key Offset*

This is the offset of the key from the inside (soil) face of the wall. A positive dimension will move the key in the heel direction. A negative dimension will move the key in the toe direction.

Wall Batter?*

This allows you to slope the toe face of the wall, the heel face or both.

Toe/Heel Batter*

If you've chosen to batter the wall from the **Wall Batter?** section, this is where you can define the batter dimension. This value is an additional thickness from the **Wall Thick** entry, at the base of the wall.

Soil Tab

The following inputs can be made on the **Soil** tab of the **Wall Definitions spreadsheet** for wall footings:



	Label	Method	Gamma Heel [k/ft³]	Phi Heel(Deg)	K Lat Heel	Gamma Toe [k/ft³]	Phi Toe(Deg)	K Lat Toe	Delta Wall (Deg)	Backfill Angle(Deg)	Heel Surcharge [ksf]	Toe Depth [ft]	EQ Max [ksf]	Kh	Gamma Sat Heel [k/ft³]	Phi Sat Heel(Deg)	K Lat Sat Heel
1	Wall Foot...	Rankine	0.111	30		0.111	30		N/A	0	0.5	2	N/A	N/A	0.156	30	
2	Wall Footing 2	Rankine	0.111	25	0.33	0.111	25	3	N/A	15	0	0	N/A	N/A	0.164	25	0.33
3	Wall Footing 3	Rankine	0.111	30		0.111	30		N/A	0	0.5	2	N/A	N/A	0.156	30	

Method (Rankine or Coulomb)*

Here you have the choice of how you want lateral earth pressures calculated. Note that if you want seismic forces applied to your retaining wall, **Coulomb** must be selected.

Note:
- Calculations for soil pressures for both methods are defined in Wall Footings - Design.
- Seismic forces are described in Wall Footings - Design.

Gamma Heel/Gamma Toe

These are the soil densities on both the heel and toe sides of the wall for a retaining wall. For a strip footing, the Gamma Heel value is used for both the heel and toe.

Phi Heel/Phi Toe*

This is the internal angle of friction of the soil on both the heel and the toe sides of the wall. This is used, along with the backfill angle, in determining the lateral force coefficient (K). Note that there is a K input field. If this field is input, then we will override the calculation and simply use the input K value. For more information, see **Soil Considerations** in the Wall Footings - Design topic.

K Lat Heel/K Lat Toe*

This allows you to directly input the soil lateral force coefficient, K, for the soil on the heel and toe sides. You can leave this blank and the program will calculate K for you, using the **Phi** and **Backfill Angle** values. For more information, see **Soil Considerations** in the Wall Footings - Design topic.

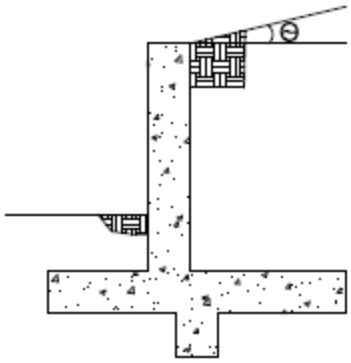
Delta Wall*

This is the wall friction angle. It is used only if **Coulomb** is set for the Lateral Pressure Method. For more information, see **Soil Considerations** in the Wall Footings - Design topic.

Note: If **Rankine** is selected, this input is ignored.

Backfill Angle*

This is the slope angle of the backfill on the heel side from horizontal. A negative slope is not allowed.



Heel Surcharge*

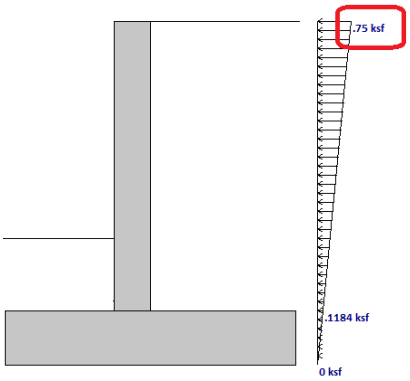
This is a vertical surcharge pressure on the soil adjacent to the heel side of the wall and is always considered in the **LL load category**.

Toe Depth*

This is the soil depth above the top of footing on the toe side of the wall. If this is set to zero, then the soil level will be assumed at the bottom of footing elevation. Thus, if there is a shear key, you can consider the passive pressure only on the key.

EQ Max*

This is used for seismic loading and, if input, would define the maximum earthquake pressure at the top of an inverted earthquake pressure diagram. If this value is blank, the program will use **Kh** and calculate the seismic forces per the Mononobe-Okabe formulation. Seismic forces are described in Wall Footings - Design.



Note: If **Rankine** is selected, this input is ignored.

Kh*

This is the design horizontal ground acceleration (in units of g). This is an input into the Mononobe-Okabe seismic force calculation method. For more information, see Wall Footings - Design.

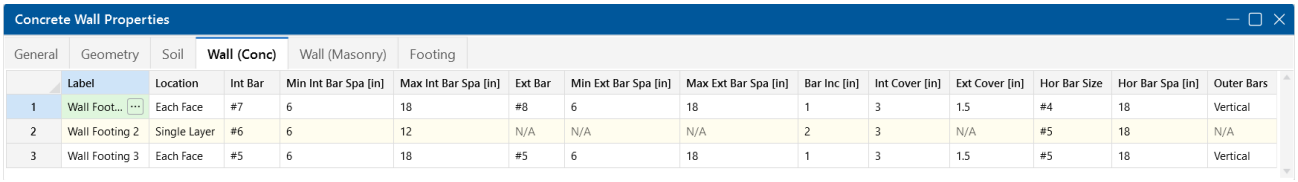
Note: If **Rankine** is selected, this input is ignored.

Gamma Sat/Phi Sat Heel/K Lat Sat Heel*

These are the soil parameters in the saturated portion of soil and are only considered if the water height is greater than zero. If **K Lat Sat Heel** is left blank, the program will use the **Phi Sat Heel** and the **Backfill Angle** to calculate K. For more information, see **Soil Considerations** in the Wall Footings - Design topic.

Wall Tab (Conc)

This tab is only used if your wall is made of concrete to define reinforcement parameters.



	Label	Location	Int Bar	Min Int Bar Spa [in]	Max Int Bar Spa [in]	Ext Bar	Min Ext Bar Spa [in]	Max Ext Bar Spa [in]	Bar Inc [in]	Int Cover [in]	Ext Cover [in]	Hor Bar Size	Hor Bar Spa [in]	Outer Bars
1	Wall Foot...	Each Face	#7	6	18	#8	6	18	1	3	1.5	#4	18	Vertical
2	Wall Footing 2	Single Layer	#6	6	12	N/A	N/A	N/A	2	3	N/A	#5	18	N/A
3	Wall Footing 3	Each Face	#5	6	18	#5	6	18	1	3	1.5	#5	18	Vertical

Location

This defines whether you have a single layer of reinforcement or bars at each face.

Int Bar and Ext Bar

This defines the vertical bar on the interior (soil) and exterior faces of the wall. If you have defined a single layer of reinforcement, then the **Ext Bar** field will not be used.

Min/Max Int and Ext Bar Space

The program will design the vertical reinforcement spacing based on these guidelines. If you want the reinforcement to be at an exact spacing, enter that spacing as both the min and max in order to force the desired spacing.

Note:
- If you have a cantilevered wall, the exterior bar spacing design will be designed solely to meet minimum requirements.
- If you have defined a single layer of reinforcement, then the exterior min and max spacing will not be used.

Bar Inc

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.

Int and Ext Cover

These are cover dimensions for the interior (soil) and exterior faces of the wall. If you have defined a single layer of reinforcement, then the exterior cover will not be used.

Hor Bar Size and Hor Bar Spacing

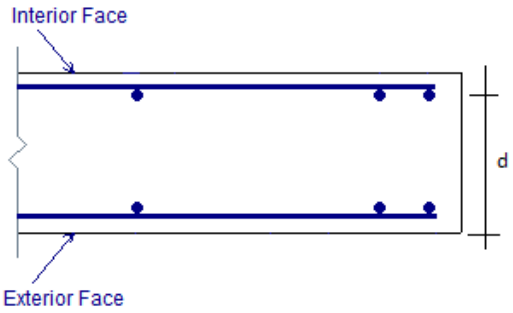
This defines the bar size and spacing of the horizontal bars in the wall. The program will check that these bars meet minimum requirements.

Note: The program does not consider two way bending, so this reinforcement is only for non-strength requirements.

Outer Bars

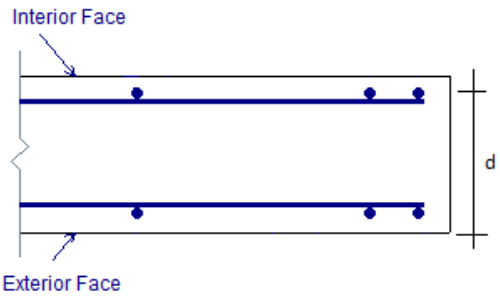
This defines whether the vertical or horizontal bars in the wall are nearest the faces of the wall. This affects the "d" calculation for the wall. If you have defined a single layer of reinforcement, then this field will not be used and it will be assumed that the vertical bar is nearest the inside (soil) face.

The images below are a section cutting horizontally through the wall. In this first image, **Horizontal** outer bars have been specified.



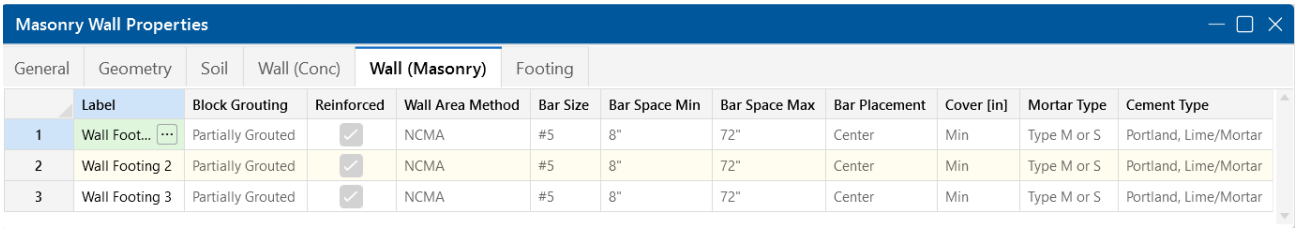
In this second image, **Vertical** outer bars have been specified.

Vertical



Wall Tab (Masonry)

This tab is only used if your wall is made of masonry to define reinforcement and design parameters.



	Label	Block Grouting	Reinforced	Wall Area Method	Bar Size	Bar Space Min	Bar Space Max	Bar Placement	Cover [in]	Mortar Type	Cement Type
1	Wall Foot...	Partially Grouted	✓	NCMA	#5	8"	72"	Center	Min	Type M or S	Portland, Lime/Mortar
2	Wall Footing 2	Partially Grouted	✓	NCMA	#5	8"	72"	Center	Min	Type M or S	Portland, Lime/Mortar
3	Wall Footing 3	Partially Grouted	✓	NCMA	#5	8"	72"	Center	Min	Type M or S	Portland, Lime/Mortar

Block Grouting

This option defines how the wall is grouted. If **Partially Grouted** is chosen, the spacing of grout will be based on the Bar Spacing Min/Max.

Reinforced

This defines whether the wall is reinforced or not. Note that if the wall is unreinforced, the program will also consider the wall ungrouted.

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.

Research on these two methods of calculating the area for a masonry wall found that 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, the program will default to using the **NCMA** option.

For more information on this, see Masonry Wall Considerations in Wall Footings - Design.

Note: 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.

Bar Size

This is the main vertical bar size that will be used for 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, the program will optimize the reinforcement spacing according to code check requirements.

Bar Placement

This defines how the reinforcement is placed in the wall.

- **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 the governing provision between the following two sections of the specification:

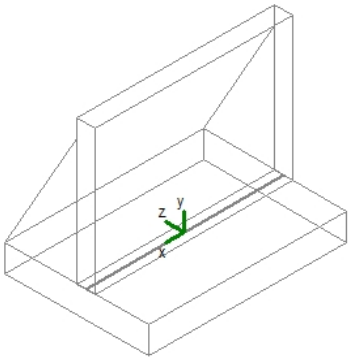
- Section 6.1.3.5 states a minimum distance between the inside of the face shell and the reinforcement. RISA is conservative and always assumes this to be 1/2". Thus, the cover calculated in this fashion = $t_{faceshell}$ + 1/2"

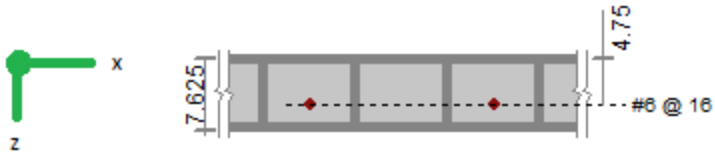
- Section 6.1.4 requires a cover dimension = 1-1/2" or 2" depending on bar size and whether the masonry is exposed to weather. In the program, this is always assumed to be 1-1/2".

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, the cover will always be from the ***+z local axis face of the wall***.

For example, let's assume the wall shown in the first image below is using the "Non-Centered" reinforcement and the cover is set to 2.5". The second image shows exactly where this bar is 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.

Footing Tab

The footing tab contains the reinforcement parameters for your wall footing.



	Label	Location	Bot Bar	Min Bot Bar Spa [in]	Max Bot Bar Spa [in]	Top Bar	Min Top Bar Spa [in]	Max Top Bar Spa [in]	Bar Inc [in]	Top Cover [in]	Bot Cover [in]	Long Bar Size	Long Bar Spa [in]
1	Wall Foot...	Each Face	#6	4	18	#6	4	18	1	3	3	#4	18
2	Wall Footing 2	Single Layer	#6	6	18	N/A	N/A	N/A	2	N/A	3	#5	18
3	Wall Footing 3	Each Face	#5	4	18	#5	4	18	1	3	3	#5	18

Location

This defines whether you have a single layer of reinforcement or bars both top and bottom.

Bot Bar and Top Bar

This defines the bottom and top bar sizes in the footing. If you have defined a single layer of reinforcement, then the **Top Bar** field will not be used.

Min/Max Bot and Top Bar Space

The program will design the main footing 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 the desired spacing. If you have defined a single layer of reinforcement, the top min and max spacing will not be used.

Bar Inc

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.

Top Cover and Bot Cover

These are cover dimensions for the footing. If you have defined a single layer of reinforcement, the top cover dimension will not be used.

Long Bar Size and Long Bar Spacing

This defines the bar size and spacing of the longitudinal bars in the footing. The program will check that these bars meet minimum requirements.

Note:

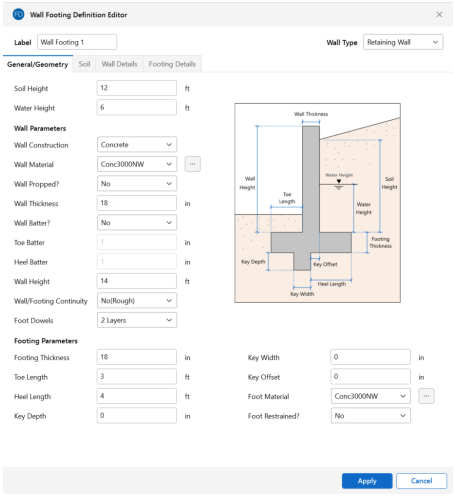
- The program does not consider two way bending. This reinforcement is only for non-strength requirements.

Wall Footing Definition Editor

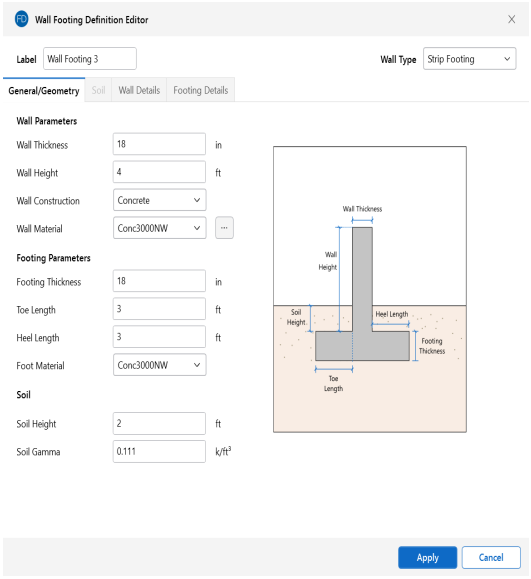
The **Wall Footing Definition Editor** enables you to create or modify wall footing definitions in a graphical way. This editor automatically updates any information input into the **Wall Footing Definitions Spreadsheet**.

Note:

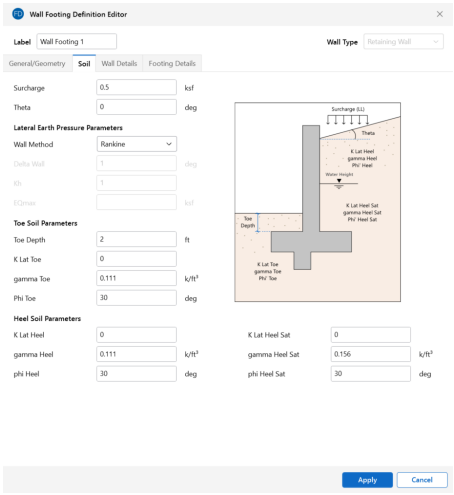
- Keep in mind that this dialog has multiple tabs. Make sure to fill out the information on all relevant tabs before clicking **OK**.

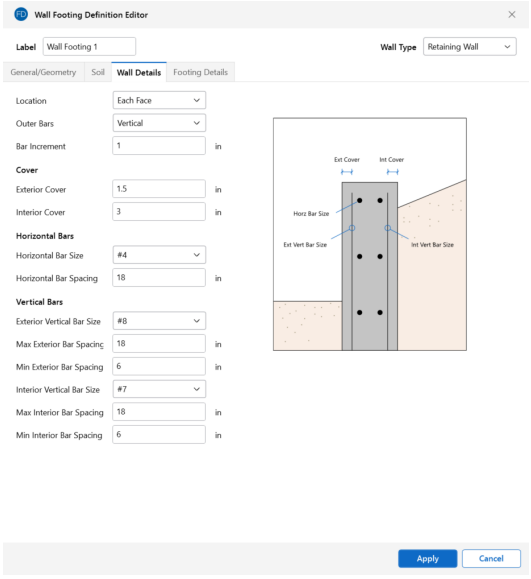


General/Geometry tab for Retaining Walls



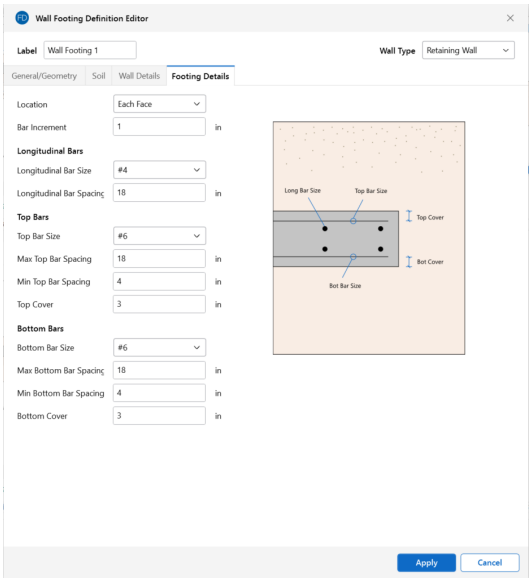
General/Geometry tab for Strip Footings





Soil tab (not applicable for Strip Footings)

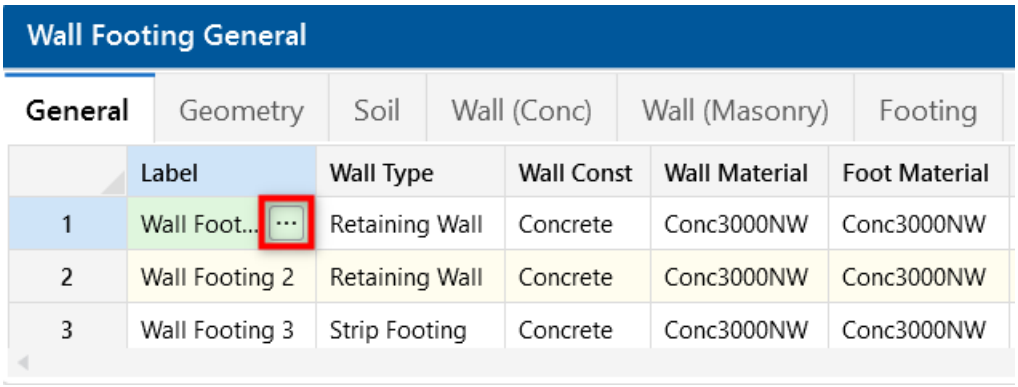
Wall Details tab (applicable for Concrete Retaining Walls and Strip Footings)



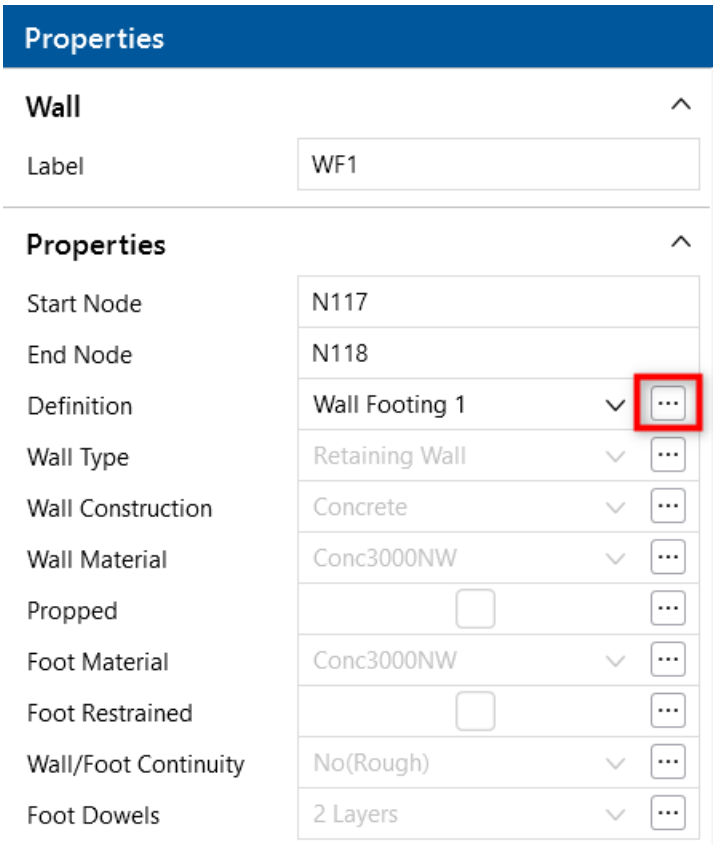
Footing Details tab

Accessing the Wall Footing Editor

To access this editor, you can click the ... (ellipsis) button on any tab in the **Wall Footing Definitions Spreadsheet**.



This dialog can also be accessed from the **Properties** panel, as well.



Wall Footings - Design

Here we will talk specifically about the calculations for wall footings.

- For more information on defining wall parameters, see Wall Footing Definitions.
- For more information on modeling requirements, see Wall Footings - Modeling.
- For more information on the output, see Wall Footing Results

RISAFoundation currently designs wall footings for all of the **ACI** codes, as well as the **CSA A23.3-04/14 Canadian** code. In this topic we will make specific reference to provisions for some of these codes. Note that the only differences in behavior with the differing codes is with the reinforcement design. All soil and serviceability checks are identical across codes.

Soil Considerations

The following details discuss some of the considerations for the computation of soil pressures.

Note: This section is only applicable for retaining walls. Strip footings use soil solely for vertical load considerations.

Lateral Force Coefficient (K) Calculations

From the **Wall Footing Definition Editor** (or the **Wall Footing Definitions Spreadsheet**), you can define either **Rankine** or **Coulomb** for the Lateral Earth Pressure Methodology.

Rankine

You can define the coefficient of internal friction (φ) and backfill angle (θ) in the **Wall Footing Definition Editor** (or the **Wall Footing Definitions Spreadsheet**) and allow the program to calculate K. Alternatively, you can directly input the K factors. If the program is used to calculate K, these are the equations used:

$$K_o := 1 - \sin(\phi)$$

$$K_a := \cos(\theta) \cdot \left[\frac{\cos(\theta) - \sqrt{(\cos(\theta))^2 - (\cos(\phi))^2}}{\cos(\theta) + \sqrt{(\cos(\theta))^2 - (\cos(\phi))^2}} \right]$$

$$K_p := \cos(\theta) \cdot \left[\frac{\cos(\theta) + \sqrt{(\cos(\theta))^2 - (\cos(\phi))^2}}{\cos(\theta) - \sqrt{(\cos(\theta))^2 - (\cos(\phi))^2}} \right]$$

Note:

- The calculation of K_o assumes normally consolidated granular soil.
- K_a and K_p are calculated considering Rankine theory.
- Because of the use of Rankine theory, battered retaining walls behave identically to walls with uniform thickness for hydrostatic loading.

Coulomb

You can also directly input K with the Coulomb method and the program will ignore the other values. If the program is used to calculate K, these are the equations used.

$$K_o := 1 - \sin(\phi)$$

$$K_a := \frac{(\sin(\alpha + \phi))^2}{(\sin(\alpha))^2 \cdot \sin(\alpha - \delta) \cdot \left(1 + \sqrt{\frac{\sin(\delta + \phi) \cdot \sin(\phi - \theta)}{\sin(\alpha - \delta) \cdot \sin(\alpha + \theta)}} \right)^2}$$

$$K_p := \frac{(\sin(\alpha - \phi))^2}{(\sin(\alpha))^2 \cdot \sin(\alpha + \delta) \cdot \left(1 - \sqrt{\frac{\sin(\delta + \phi) \cdot \sin(\phi + \theta)}{\sin(\alpha + \delta) \cdot \sin(\alpha + \theta)}}\right)^2}$$

where:

- θ = Backfill angle
- φ = Soil friction angle
- δ = Wall friction angle
- α = Wall slope angle measured from horizontal (90 degrees for a vertical wall face)

Both Methods

There is a potential to have three K factors for a single retaining wall: K Heel, K Sat Heel and K Toe.

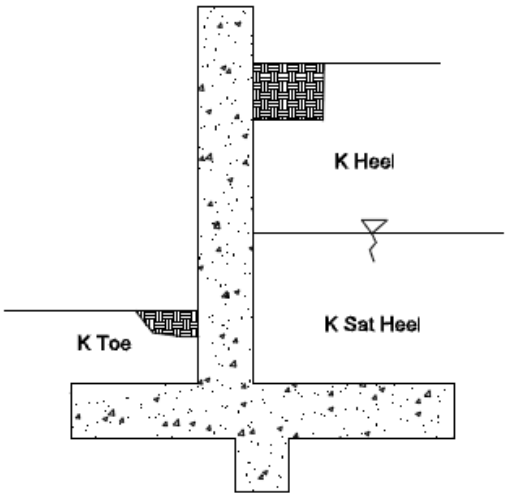


Figure 1: Three Possible K factors

Note:
- If the wall is a **cantilever**, the program will use the K_a equation for the heel side K factors and K_p equation for the toe side K factors.
- If the wall is a **propped cantilever**, the program will use the K_o equation for both the heel and toe K factors.
- RISAFoundation performs a calculation to compare the total passive forces to the active forces. If the passive forces exceed the active force, the wall is assumed to be in an at rest condition and the program will use the K_o equation for both the heel and toe K factors. When this occurs, the wall footing detail report will show (At Rest) after both the heel and toe K factors.

Lateral Soil Summation

The program will perform a calculation comparing the total active force from the heel side and the total passive force from the toe side. If the passive force exceeds the active force, you then have an irrational solution. In this case, the program will give a message in the Detail Report and not design your retaining wall. The message states "The hydrostatic passive forces (toe side) exceed the hydrostatic active forces (heel side). Consider manually entering K factors for this wall." In this case, the Rankine active vs passive pressures will not work, so instead you may need to manually input your K factors in the Wall Footing Definitions Spreadsheet.

Loading Considerations

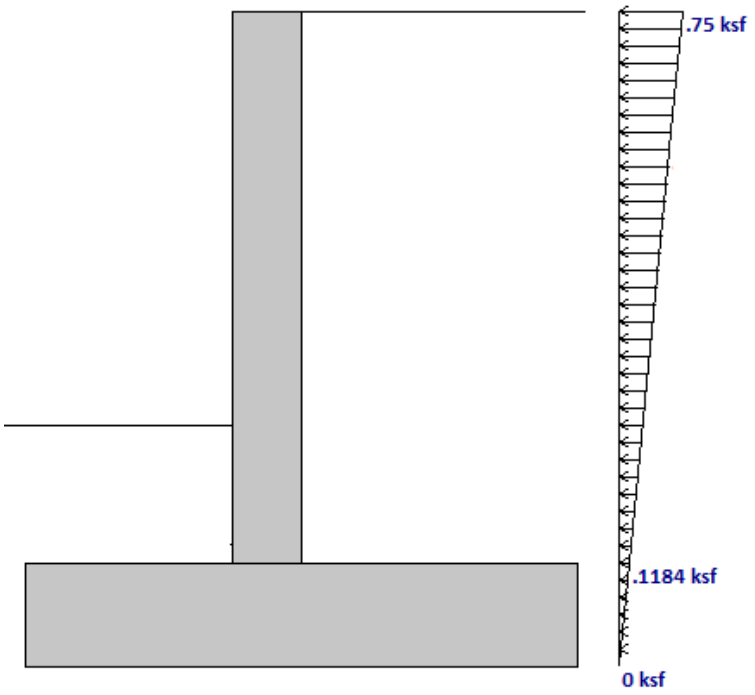
Regarding the loading of retaining walls, all hydrostatic, soil and surcharge loads are taken into account on a per foot basis. For more information, see Wall Footings - Modeling. Externally applied loads to retaining walls or strip footings are summed over the entire length of the wall and then divided out on a per foot basis. Because of this, concentrated stresses and areas of the wall that require tighter reinforcement are not taken into account. For more information on applying external loads to the model, see Wall Footings - Modeling.

Note: For highly loaded areas of wall footings where you want the reinforcement to be modified, break the wall footing into multiple pieces. Each piece will have its reinforcement optimized separately.

Seismic Loading Considerations

RISAFoundation will consider an inverted triangular seismic loading diagram if:

- The "EL" category specifically is used in the load combination.
- The Coulomb method is used for the Lateral Earth Pressure Methodology



There are two paths to applying your seismic loads:

User Define the Magnitude of the Maximum Value of the Diagram

In the Wall Footing Definitions - Soil tab, there is an input for **EQ Max**. If there is a value in here, this is used as the maximum.

Allowing the Program to Calculate this Magnitude Using the Mononobe-Okabe/Seed-Whitman Formulation

In the Wall Footing Definitions - Soil tab, there is an input for **EQ Max** and **Kh**. For the program to calculate the magnitude of the maximum value of the diagram, EQ Max should be left blank and a value for Kh should be input.

The Mononobe-Okabe/Seed-Whitman equation is an expression for the total lateral earth force, including seismic plus active forces. Thus, to calculate the portion due solely to seismic, we must remove the active pressure portion. The M-O pressure coefficient is calculated as follows:

$$K_{ae} := \frac{\left(\sin\left(\alpha+\phi-\kappa\right)\right)^2}{\cos\left(\kappa\right)\cdot\left(\sin\left(\alpha\right)\right)^2\cdot\sin\left(\alpha-\delta-\kappa\right)\cdot\left(1+\sqrt{\frac{\sin\left(\delta+\phi\right)\cdot\sin\left(\phi-\theta-\kappa\right)}{\sin\left(\alpha-\delta-\kappa\right)\cdot\sin\left(\alpha+\theta\right)}}\right)^2}$$

where:

- θ = Backfill angle
- φ =Soil friction angle
- δ = Wall friction angle
- κ = Seismic Factor = tan-1(Kh)
  - Kh = Input on Soils tab
- α = Wall slope angle measured from horizontal (90 degrees for a vertical wall face)

The new term here is κ. This value uses the input **Kh** value for the Design Horizontal Ground Acceleration (in g). **Kh** is generally defined as **Sds/2.5**, but the actual value is up to the user. The vertical component of earthquake acceleration is not considered in the program.

The total seismic lateral force (per foot) is then defined as:

$$P_{Seismic} := \frac{1}{2} \cdot \gamma_{soil} \cdot H^2 \cdot (K_{AE} - K_a)$$

Per the Seed-Whitman variation, this seismic force is assumed to act at 0.6*Hwall. Thus, to calculate the magnitude at the top of the wall this equation is used:

$$EQ_{max} := \frac{P_{Seismic}}{0.3 \cdot H^2}$$

Note:
- Seismic calculations only apply to the Coulomb Lateral Earth Pressure Methodology.
- Seismic is ignored if K factors are hard-coded and EQ Max is left blank in the Wall Footing Definitions - Soil tab.
- Seismic is ignored if the wall is propped at the top.

Serviceability Considerations

Overturning and sliding checks are only provided for **Service** load combinations (LC's with the **Service** check box checked).

Overturning

Overturning is checked at the base of the footing at the extreme edge of the toe. Each portion of soil weight, concrete weight, surcharge loading, lateral soil pressure and lateral water pressure is taken into account. Each piece is either part of the resisting moment or the overturning moment.

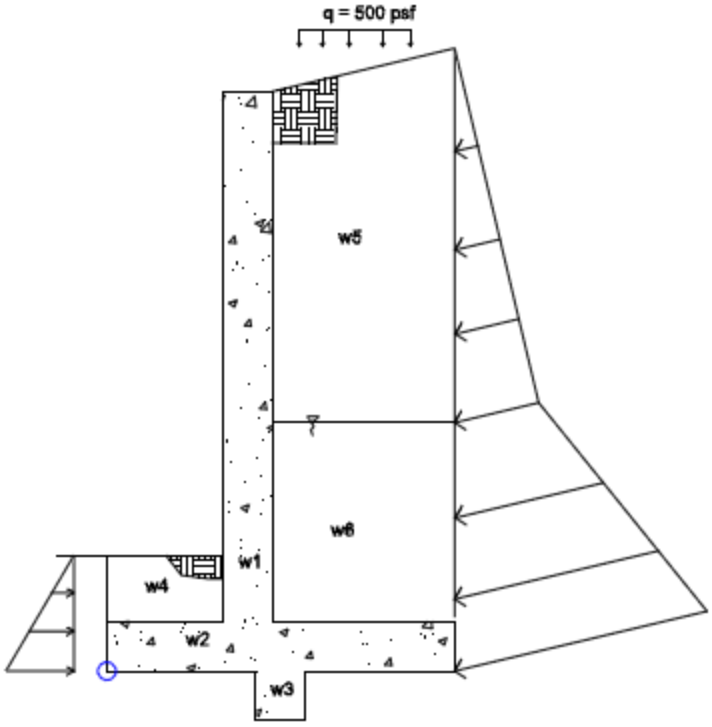


Figure 2: Overturning Diagram

From here, all of the resisting moments and overturning moments are summed. The ratio of resisting moments to overturning moments is checked against the **SF** given in that individual load combination.

$$OSF := \frac{M_{Resisting}}{M_{Overturning}}$$

$$OT_{UC} := \frac{SF}{OSF}$$

Note:

- If the wall is a strip footing there is no consideration for lateral hydrostatic forces. Only vertical soil loads, self-weight and external forces are considered.
- If the wall is a propped cantilever the overturning checks will be omitted.
- The program considers the passive soil resistance from the toe in overturning considerations.
- For sloped backfills, the vertical component of the hydrostatic load will be included in the resisting moment.
- The presence of a key has no bearing on overturning except that the weight of the key will be accounted for.

Sliding

Sliding is checked at the extreme bottom face of the footing. This occurs at the bottom of footing if there is no shear key. If there is a shear key, this occurs at the bottom of the key.

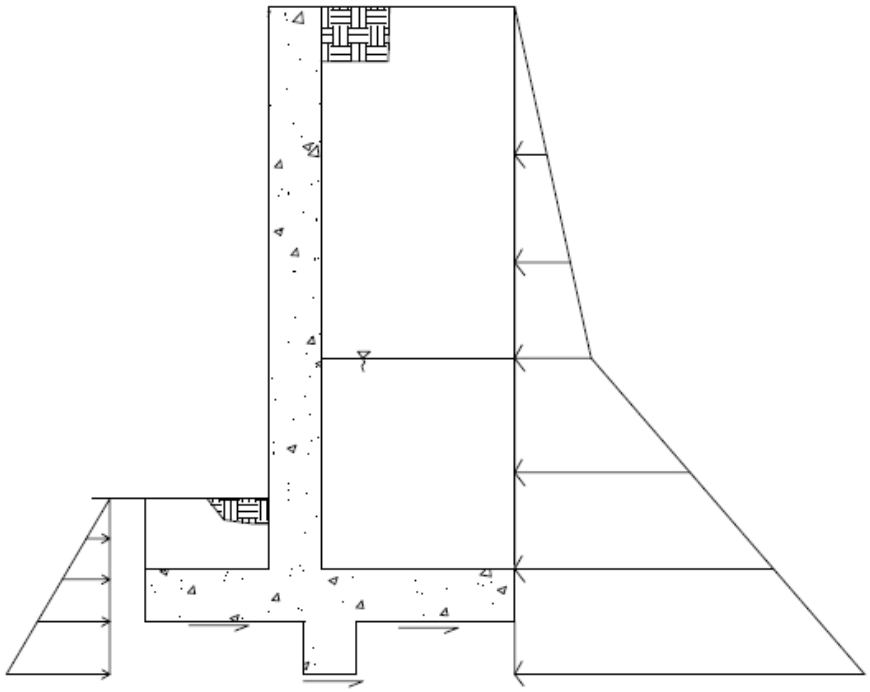


Figure 3: Sliding Forces Diagram

Both the passive soil pressure and friction resistance are accounted for.

$$Friction_{Resistance} := Total_{Weight} \cdot \mu \qquad PassiveSoil_{Resistance} := \frac{1}{2} K_p \cdot \gamma_{toe} \cdot D_{toesoil}^{2}$$

$$Sliding_{Resistance} := Friction_{Resistance} + PassiveSoil_{Resistance}$$

where:

- Total Weight = Total downward force acting on the footing due to soil, concrete and surcharge forces.
- μ = Soil coefficient of friction defined in the Model Settings - Solution tab.

The sliding resistance is then compared against the sliding force caused by the hydrostatic pressure. The ratio of resisting forces to sliding forces is checked against the **SF** given in that individual load combination.

$$SSF := \frac{Sliding_{Resistance}}{Sliding_{Force}}$$

$$Sliding_{UC} := \frac{SF}{SSF}$$

Note:

- Deflection calculations for the top of the wall are not given.
- Sliding is not checked for retaining walls where the footing is restrained or if the top of the wall is propped.
- For strip footings, the hydrostatic lateral forces from either the heel or the toe are not considered. Sliding is only resisted by frictional forces.

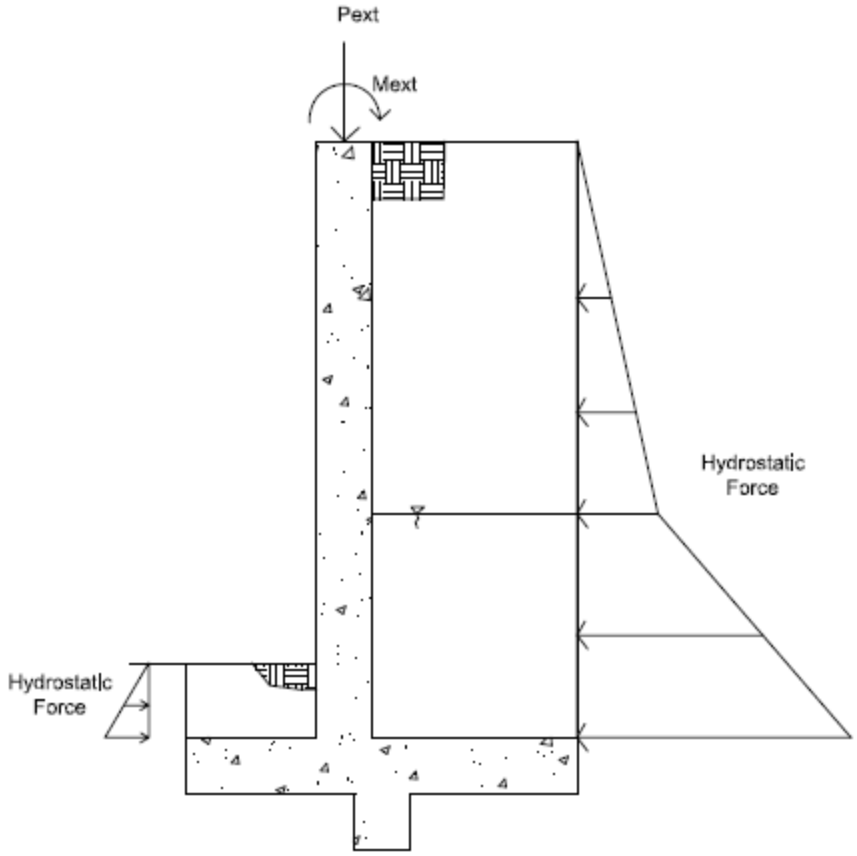
Stem Wall Design Considerations

Stem walls can be made of concrete or masonry materials. Design details for walls made with each of these materials are discussed below.

General Considerations

The stem wall design is analyzed along its height to come up with axial, shear and moment diagrams. The wall is designed for the worst case forces at the critical section. The program checks 20 locations over the height of the wall to determine the critical section. For a cantilevered uniform stem wall, the critical section is always at the wall intersections with the top of footing. For a propped retaining wall and/or one with a batter, this will be somewhere along the length of the wall. The program will take any externally applied loads in combination with hydrostatic pressures, surcharges, etc. The program

will then factor the loads according to load combinations to come up with the axial, shear and moment diagrams.



Note:

- Because the combined effects of the flexural deflection of the stem and the rotational deflection of the overall wall/footing is generally small and hard to quantify, wall deflections are not considered in the analysis, .
- Strip footing stem walls require the soil height to be equal on both sides. Thus, hydrostatic forces are ignored. Only applied forces will induce any bending into a strip footing stem wall.
- The vertical portion of that load will be assumed to act at the top of the wall and be included in the axial force in the wall if there is a sloping backfill.
- **P-Delta** (big and little) is not considered for stem walls. Typically, axial forces are small in retaining walls and P-Delta will have little effect.

Concrete Wall Considerations

For concrete walls, the stem wall design is only considered for non-service load combinations.

Axial/Bending Design

The program, given the wall thickness, reinforcement sizes, and max and min spacing and spacing increment, will design the wall reinforcement spacing to meet strength and minimum steel requirements.

The program will start with the maximum spacing. If that does not work for design, the spacing will drop by the spacing increment and be checked again. The program will work its way down until it reaches a spacing that meets all reinforcement requirements. The wall is checked considering the entire wall as a column. Thus, interaction and compression reinforcement is considered in the code check. Results are presented on a per foot basis, so the entire wall demand and capacity is calculated and then divided out on a per foot basis.

Note:

- The exact location and number of bars in the wall is used in capacity calculations. Use the **As Provided** in the Detail Report when doing any hand calculation verification.
- Reinforcement checks include the minimum reinforcement requirements of ACI 318-14 Section 11.7, ACI 318 Section 14.3 and CSA A23.3-04 Clause 14.
- For battered retaining walls, the controlling section will be reported, though this may not be the location of maximum moment.
- For battered retaining walls, the reinforcement on the battered face of the wall will be parallel to the face.

Moment and Axial Force Thresholds

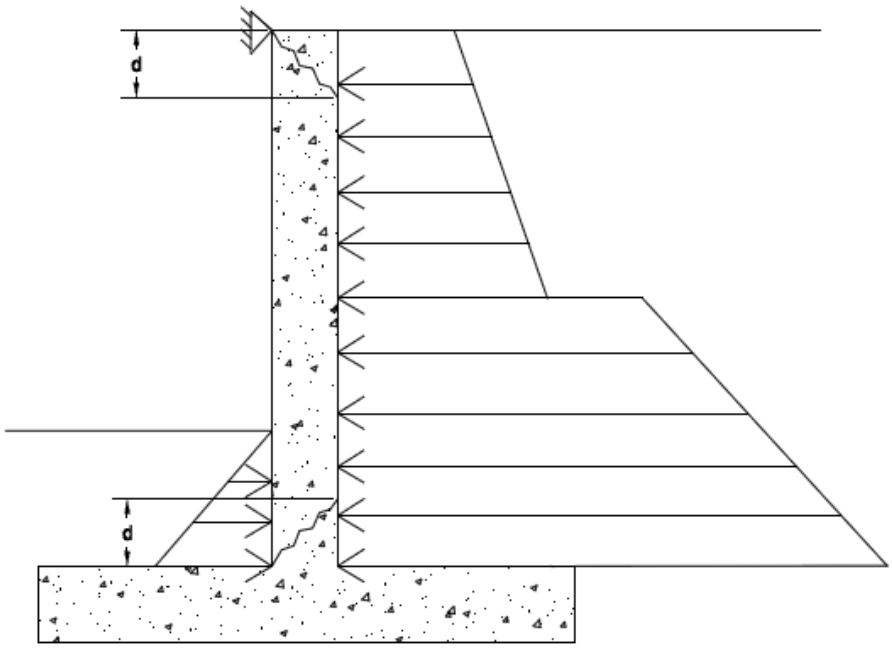
The program will ignore axial forces and moments that are below a certain threshold. If the moment or axial force is deemed to be inconsequential to the code check, the program will not include the interaction of that force. There are two thresholds that are considered:

- Axial Force Threshold: If **Pu < 0.01*f'c*Ag** for a LC, the axial force in the wall will be ignored for code checks for that LC.
- Moment Threshold: If **Mu < 0.01*d*Pu** for a LC, the moment in the wall will be ignored for code checks for that LC.

These two thresholds allow the concrete solver to work much more efficiently while having little to no effect on code check values.

Shear Check (Concrete)

The program will perform a concrete shear check. This check is taken at a distance "d" from the top of footing for a **cantilevered** wall. It is taken at a distance "d" from both the top of footing and from the top of the wall for a **propped cantilevered** wall.



Note:

- This check will occur at 20 sections along the height of the wall and the worst case code check will be given in the results for a battered retaining wall.
- The shear check does not consider axial force. The program check is using the ACI 318 equation (and a similar equation for the **CSA A23.3-04**):

$$V_c := 2 \cdot \lambda \cdot \sqrt{f'_c} \cdot b_w \cdot d$$

Masonry Wall Considerations

For masonry walls, per the Model Settings, you can use either ASD or Strength code checks. Stem wall design is only considered for non-service load combinations. Here, we will break down both ASD and Strength aspects. Codes currently supported for masonry include:

- The 2022 and 2016 TMS 402 ASD and Strength codes.
- The 2013 ACI 530 ASD and Strength codes.

Note: All code references in this section refer to the **TMS 402-22** specification.

ASD Design Considerations

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

$$A_{nRMEH} := Eq_{solidthickness} \cdot L_{wall}$$

$$A_{nvNCMA} := A_{nvNCMAperfoot} \cdot L_{wall}$$

Note: The program does not design masonry for net tension forces. fa will equal 0 ksi in these cases.

Axial Capacity, Fa

The calculation of Pa for reinforced masonry is Equation 8-16 or 8-17 and Fa for unreinforced masonry is per either Equation 8-11 or 8-12. depending on the h/r ratio. The equations are as follows:

For reinforced masonry:

if h/r <= 99

$$P_a := \left(0.30\ f'_m \cdot A_n + 0.65 \cdot A_{st} \cdot F_s\right) \cdot \left(1 - \left(\frac{h}{140\ r}\right)^2\right)$$

if h/r > 99

$$P_a := \left(0.30\ f'_m \cdot A_n + 0.65 \cdot A_{st} \cdot F_s\right) \cdot \left(\frac{70\ r}{h}\right)^2$$

For unreinforced masonry:

if h/r <= 99

$$F_a := \frac{1}{4} \cdot f'_m \cdot \left(1 - \left(\frac{h}{140\ r}\right)^2\right)$$

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 = Total height of wall

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.

Stresses fb and fs

The masonry bending stresses are referenced in UBC Section 2107.2.15 and are as follows:

$$f_b := \frac{2 \cdot M}{j \cdot k \cdot b \cdot d^2} \qquad f_s := \frac{M}{A_s \cdot j \cdot d}$$

where:

$$\rho := \frac{A_s}{A_n} \qquad n := \frac{E_s}{E_m}$$

$$k := \sqrt{2 \cdot \rho \cdot n + (\rho \cdot n)^2} - \rho \cdot n$$

$$\rho := \frac{A_s}{A_n} \qquad n := \frac{E_s}{E_m}$$

$$j := 1 - \frac{k}{3}$$

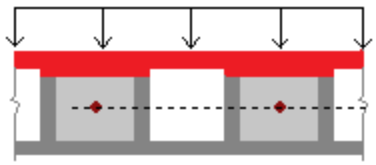
b = spacing of reinforcement

M = moment over a width b

However, if you are using a partially grouted wall where the neutral axis passes through the webs of your masonry, RISA will do a T-section analysis to define the section properties. We use a similar analysis as if you were doing a t-beam analysis for a concrete tee section. For more information on this, see "Design of Reinforced Masonry Structures" by Narendra Taly, copyright 2001, example 6.3, P6.61.

T-SECTION ANALYSIS

kd	:	1.638	in
Flange thickness	:	1.375	in
Web width	:	8.5	in



The area in red represents the compression block in the image above.

Bending Capacity, Fb

For unreinforced masonry, Equation 8-13 is:

$$F_b := \frac{1}{3} \cdot f_m$$

For reinforced masonry, Section 8.3.4.2.2 states:

fa + fb <= 0.45*fm

Because of this provision, RISA defines:

$$F_b := 0.45 \cdot f_m$$

Steel Capacity, Fs

Section 8.3.3.1 defines the allowable steel stress, Fs.

Shear Stress, fv

This stress is calculated from Equation 8-19:

$$f_V := \frac{V}{A_{nv}}$$

V = Shear out of plane, per foot

$$A_{nvRMEH} := Eq_{solidthickness} \cdot 1 \cdot ft$$

$$A_{nvNCMA} := A_{nvNCMAperfoot}$$

Shear Capacity, Fv

The program calculates the capacity, Fv, from Equation 8-20, except that only the Fvm term is considered. There is no way to add shear reinforcing steel. The program also checks to verify we do not exceed the **Fv max** value from Equations 8-21 and 8-22 (or interpolation between them) that is reported in the detail report.

Strength Design Considerations

Axial Strength, Pn at max Mom

Here there are two different possible calculations. If there is out-of-plane moment on the walls (from a retaining wall, for example), this is calculated from Section 9.3.4.4.2 as follows:

- If h/t < 30, **Pn at max Mom** = 0.20*f'm*An
- If h/t > 30, **Pn at max Mom** = 0.05*f'm*An

If there is no out-of-plane moment (from a strip footing, for example), this is calculated from Equations 9-11 and 9-12:

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:
- t from the above equations is taken as the Block Nominal Thickness.
- h = Height of wall region.

Moment Strength, Mn

This calculation comes from the Commentary of Section 9.3.4.2 as follows:

$$M_n := \left(\frac{P_u}{\phi} + A_s \cdot f_y\right) \cdot \left(\frac{t_{sp} - a}{2}\right) + A_s \cdot f_y \cdot \left(d - \frac{t_{sp}}{2}\right)$$

where:

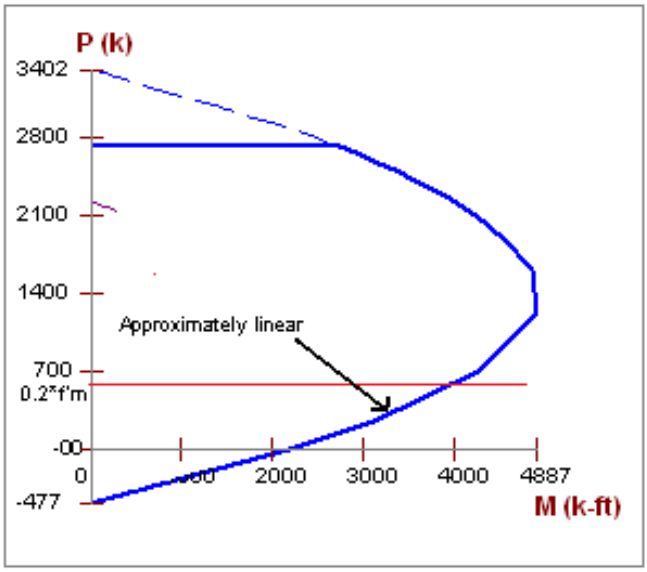
$$a := \frac{A_s \cdot f_y + \frac{P_u}{\phi}}{0.80 \cdot f_m \cdot b}$$

b = reinforcement spacing

tsp = actual block thickness

The program is actually using an interaction diagram for out of plane bending as well. However, the code places a limit on axial force from Section 9.3.4.4.2. This limit essentially means that only the

lower portion of the interaction diagram will be used. In this lower portion of the interaction diagram, the bending capacity changes in a linear fashion with respect to axial force.



Thus, the equation above is nearly identical to the value that the program's interaction diagram will produce.

Note:

- In the case of high axial tension, it is possible that moment capacity will near zero.
- In partially grouted walls, an effective width is calculated for b from Section 5.1.2.

ASD Masonry Design For Unreinforced Walls

Compression Check (per Equation 8-9)

$$Unity := \frac{f_a}{F_a} + \frac{f_b}{F_b}$$

where:

- f_a and F_a are computed the same way as for reinforced masonry.
- f_b = M/S. S = S_x value from NCMA TEK 141B (regardless of Wall Area Method designation).
- F_b = 1/3*f'm per Equation 8-15

Tension Check (per Section 8.2.4.2)

$$Unity := \frac{(f_b - f_a)}{F_{bt}}$$

where:

- Fbt = Allowable flexural tensile stress per Table 8.2.4.2

Slenderness Check (per Equation 8-10)

$$Unity := \frac{P}{0.25 \cdot P_e}$$

where:

$$P_e := \frac{\pi^2 \cdot E_m \cdot I}{h^2} \cdot \left(1 - 0.577 \frac{e}{r}\right)^3$$

- Per Equation 8-14.
- I comes directly from NCMA TEK 141B for both Wall Area Methods, NCMA and RMEH.
- e is always assumed equal to 0.
- h = Height of wall

Note: If using an **Unreinforced** wall, the program will ALWAYS use an ungrouted wall for design, regardless of what is set in the masonry wall design rules.

Shear Design (per Equation 8-19)

$$f_V := \frac{V}{A_{nv}}$$

V = Shear out of plane, per foot

$$A_{nvRMEH} := Eq_{solidthickness} \cdot 1 \cdot ft$$

$$A_{nvNCMA} := A_{nvNCMAperfoot}$$

Note: We do not use Equation 8-15 to calculate shear stress.

The program calculates the capacity, Fv, from Equation 8-20 shown above. The program also checks to verify we do not exceed the **Fv max** value from Equations 8-21 and 8-22 (or interpolation between them) that is reported in the Detail Report.

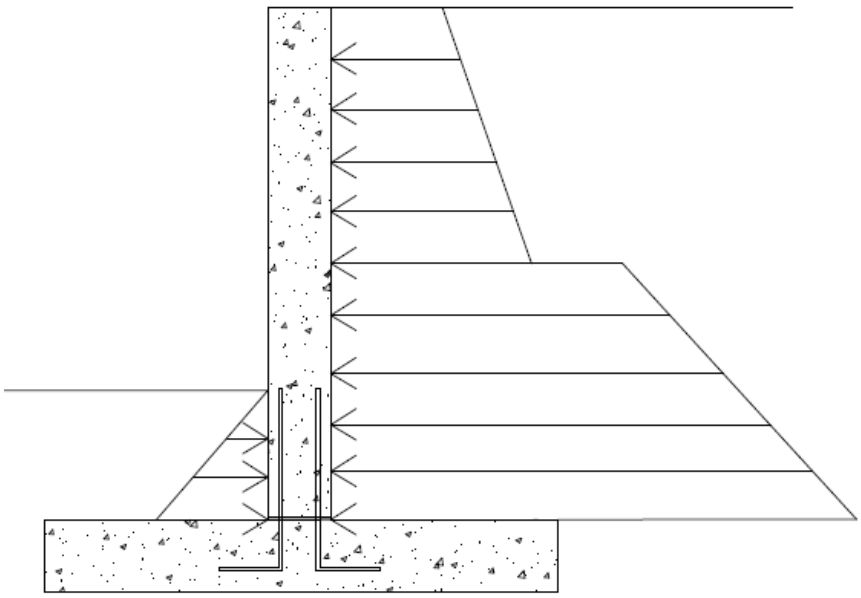
Note: Unreinforced strength design is not available.

Footing Design Considerations

Here we will discuss some of the details of footing and dowel design (shear friction). Note that footing design is only considered for non-service load combinations. Shear friction checks are strength level checks.

Shear Friction Check (for dowels between the wall and footing)

If the stem wall is not poured monolithically with the footing, a shear friction check is required at the base of the wall. This check considers the dowels from the footing to the wall and the program assumes that these dowels match the vertical reinforcement (either on the inside face or both faces; see Wall Footing Definitions for more information).



ACI 318 Codes

This check is done using ACI 318-14 Equation 22.9.4.2 (ACI 318-11 Equation 11-25) for shear friction:

$$V_n := A_{vf} \cdot f_y \cdot \mu$$

where:

- A_{vf} = the reinforcement that crosses the shear plane
- μ = 1.0*λ for intentionally roughened surface; 0.6*λ for a smooth surface

CSA A23.3-04 Code

This check is done using the CSA A23.3-04 shear friction capacity equation (11-24):

$$\sigma := \rho_v \cdot f_y \cdot \sin\left(\alpha_f\right) + \frac{N}{A_{wall}}$$

$$v_r := min\left(\lambda \cdot \phi_c \cdot \left(c + \mu \cdot \sigma\right), 0.25 \cdot \phi_c \cdot f_c\right) + \phi_s \cdot \rho_v \cdot f_y \cdot \cos\left(\alpha_f\right)$$

$$Vn := A_{wall} \cdot v_r$$

where:

- N = unfactored dead load acting on the wall (1.0*DL)
- α_f = angle between shear plane and reinforcement (always taken as 90 degrees)
- ρ_v = ratio of shear friction r/f to total wall area (from Eq 11-27)
- λ = factor for lightweight concrete
- c and μ = taken from Clause 11.5.2
- A_{vf} = the reinforcement that crosses the shear plane

ACI 530-13 Masonry Codes (ASD & Strength)

This code uses the ACI 318 procedure discussed above.

TMS 402-16 and 402-22 Masonry ASD

This check is done using Equations 8-28 and 8-29 for **TMS 402-16**, depending on the M/Vd ratio:

$$F_{f0.5} := \frac{\mu \cdot \left(A_{sp} \cdot F_s + P\right)}{A_{nv}} \geq 0$$

$$F_{f1.0} := \frac{0.65 \cdot \left(0.6 \cdot A_{sp} \cdot F_s + P\right)}{A_{nv}} \geq 0$$

The check uses Equations 8-25 and 8-26 for **TMS 402-22**, depending on the M/Vd ratio:

$$F_{f0.5} := \frac{\mu \cdot \left(A_{sp} \cdot F_s + P\right)}{A_{nv}} \geq 0$$

$$F_{f1.0} := \frac{0.65 \cdot \left(0.75 \cdot A_{sp} \cdot F_s + P\right)}{A_{nv}} \geq 0$$

where:

- μ = coefficient of friction, 1.0 rough, 0.7 smooth
- A_{sp} = area of reinforcement within net shear area
- A_{nv} = net shear area
- F_s = Allow reinforcement stress in tension/compression
- P = axial load

Note: The program will interpolate M/Vd between 0.5 and 1.0.

TMS 402-16 and 402-22 Masonry Strength

This check is done using Equations 9-33 and 9-34 for **TMS 402-16**, depending on the M/Vd ratio:

$$V_{nf0.5} := 0.42 \cdot f_m \cdot A_{nc}$$

$$V_{nf1.0} := \mu \cdot \left(A_{sp} \cdot f_y + P\right)$$

This check is done using Equations 9-33 and 9-34 for **TMS 402-22**, depending on the M/Vd ratio:

$$V_{nf0.5} := \mu \cdot \left(A_{sp} \cdot f_y + P_u\right)$$

$$V_{nf1.0} := 0.65 \cdot \left(0.75 \cdot A_{sp} \cdot f_y + P_u\right)$$

where:

- μ = coefficient of friction, 1.0 rough, 0.7 smooth
- A_{sp} = area of reinforcement within net shear area
- f_y = yield stress in steel
- A_{nc} = net cross-sectional area between neutral axis of bending and fiber of max compressive strain, = b_{eff}*c.
- F_s = Allow reinforcement stress in tension/compression
- P = axial load

Note: The program will interpolate M/Vd between 0.5 and 1.0.

Note:

- The coefficient in the μ factor is defined by the **Wall/Foot Continuity?** entry in the Wall Footing Definitions Spreadsheet.
- The shear friction check is omitted for strip footings, as loads are generally vertical as opposed to lateral.

Soil Pressure Considerations

Calculating the pressure under a footing requires both the moment (M) and the axial force (P) on the centerline of the footing at the base. An alternative hand calculation approach is to take the sum of moments (factored) from the overturning check and divide that out by the total axial force (factored). This gives you the resultant force location from the toe end.

$$x := \frac{M}{P}$$

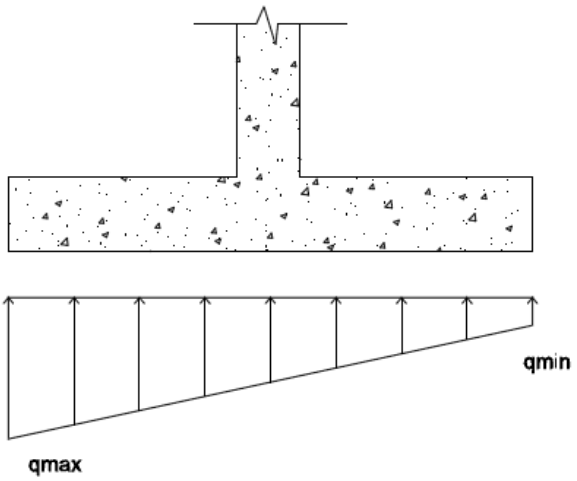
Note: For propped cantilevers, the moment, M, used in this calculation is equal to the moment at the base of the footing. Hence, we assume a fixed (bottom of footing) - pinned (top of wall) condition and calculate the moment at the base using the hydrostatic forces.

From the x-value you can calculate the eccentricity from the footing centerline as:

$$e := \frac{L_{foot}}{2} - x$$

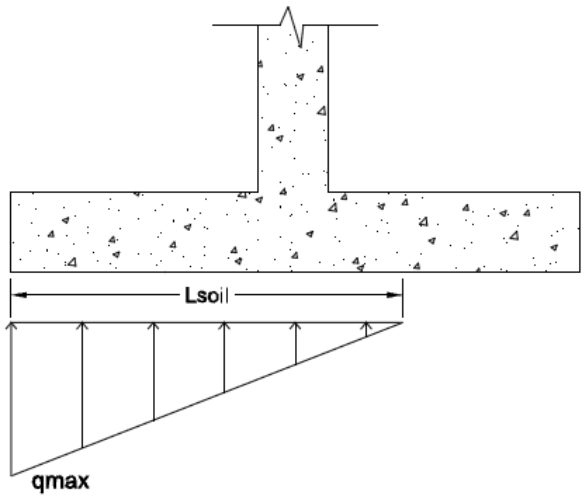
If the resultant is in the middle third of the footing the force distribution is:

$$q_{max} := \frac{P}{L_{foot}} + \frac{6 \cdot (P \cdot e)}{L_{foot}^{2}} \qquad q_{min} := \frac{P}{L_{foot}} - \frac{6 \cdot (P \cdot e)}{L_{foot}^{2}}$$

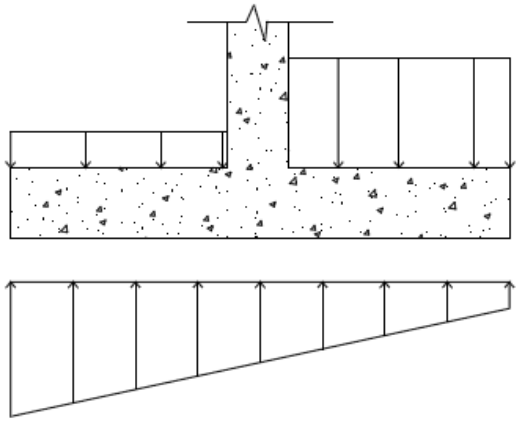


If the resultant is outside the middle of the footing, the force distribution is:

$$q_{max} := \frac{4 \cdot P}{3 \cdot (L_{foot} - 2 \cdot e)} \qquad L_{soil} := 3 \cdot x$$



The soil (and surcharge) bearing down on the soil is also considered:

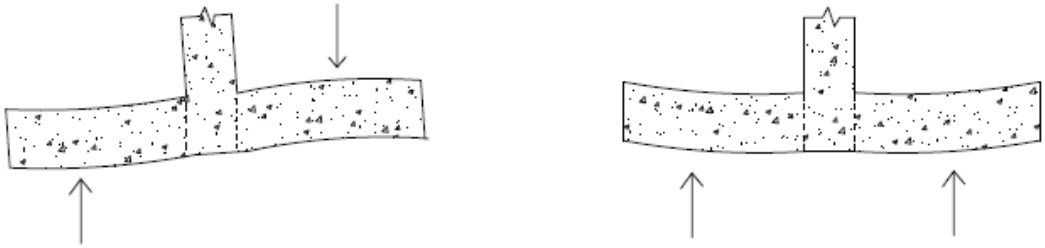


Note:

- Many design examples may disregard certain portions of load for conservatism or for ease of calculation, however, the program uses the exact loading conditions defined.
- Per ACI 318-14 Section 13.3.1.2 (ACI 318-11 Section 15.7) the depth of footing above bottom reinforcement shall not be less than 6".

Flexural Design

Flexural design for wall footings assumes that the heel and toe portions of the footing are designed as cantilevered beams from the face of the retaining wall. There are generally two deflected shapes that will occur for footings depending on the loading.

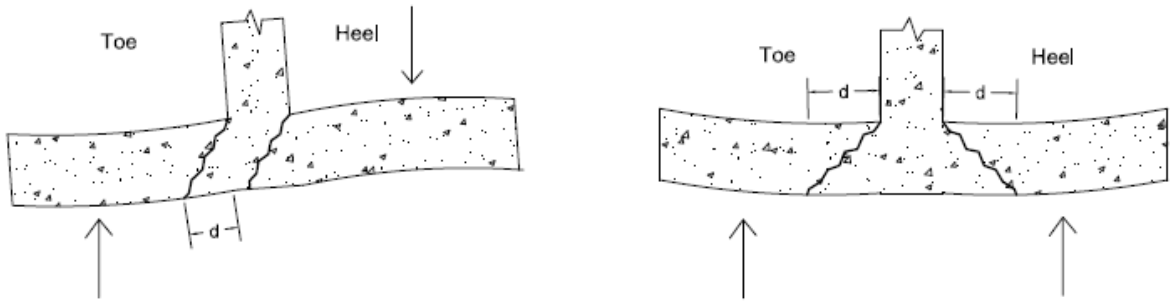


The left image is likely the most common for retaining walls. The right image will generally occur for strip footings. In either case, the program considers the moment at the face of both the heel and toe and designs reinforcement for the worst case load combination. Both the flexural reinforcement (parallel to the footing) and the shrinkage and temperature reinforcement (perpendicular to the footing) are designed for minimum steel requirements as well.

Note: If the "d" distance for the footing is very small, the program may not be able to properly calculate an As Req'd. In these cases, a "-1" will be displayed and an area of 0 will be used. The program will, however, still meet minimum reinforcement requirements and base the capacity on those values. So in many cases, your footing may still work.

Shear Checks

Shear design of footings also considers both the toe and heel sections as cantilevered beams. The difference here is where the shear failure will occur. The left image below shows the standard shear failure planes for typically loaded retaining walls. The right image below shows the standard shear failure planes for typically loaded strip footings. The program will check the loading of the footing and check the shear at the appropriate location.



From the left image, you see that the shear check would occur at a distance "d" from the face of the toe and right at the face of the heel. From the right image, you can see that both the heel and the toe would have their shear check occur at a distance "d" from the face of wall.

Note: The loading (top and bottom of footing) will always be taken from the location where the code check is taken from.

Wall Footing - Results

The locations for output of wall footings are the **Wall Footing Results** browsers and the wall footing Detail Report.

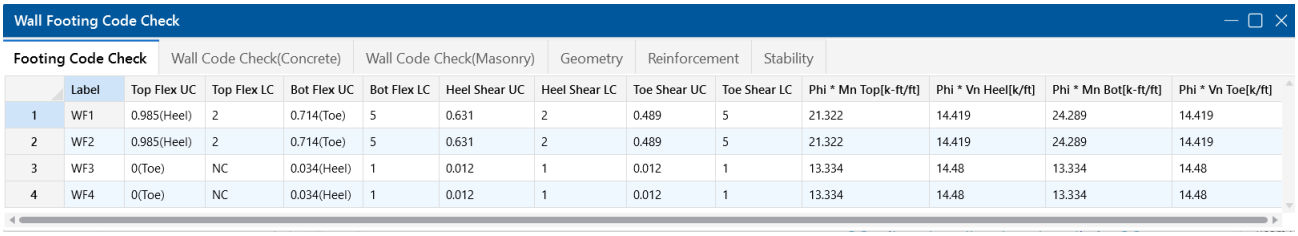
- For more information on defining wall parameters, see Wall Footing Definitions.
- For more information on modeling requirements, see Wall Footings - Modeling.
- For more information on calculation details, see Wall Footings - Design.

Wall Footing Results Browser

To view the wall footing results browser, click **Wall Footing Design** from **Results Browser** drop-down in the **Results** tab on the ribbon, or select **Wall Footing Design** from the **Results** section in

the **Explorer** panel. These output browsers are meant to summarize the results for wall footings. To see more detailed information, view the Detail Report by selecting **Detailed Report** in the **Results** tab on the ribbon.

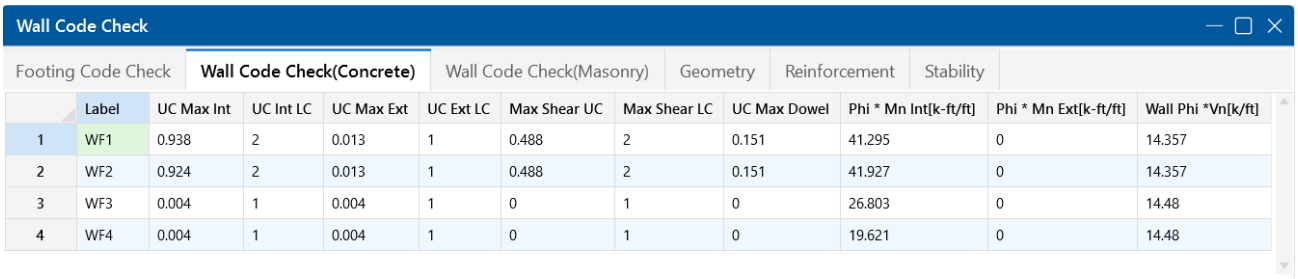
Footing Code Checks



	Label	Top Flex UC	Top Flex LC	Bot Flex UC	Bot Flex LC	Heel Shear UC	Heel Shear LC	Toe Shear UC	Toe Shear LC	Phi * Mn Top[k-ft/ft]	Phi * Vn Heel[k/ft]	Phi * Mn Bot[k-ft/ft]	Phi * Vn Toe[k/ft]
1	WF1	0.985(Heel)	2	0.714(Toe)	5	0.631	2	0.489	5	21.322	14.419	24.289	14.419
2	WF2	0.985(Heel)	2	0.714(Toe)	5	0.631	2	0.489	5	21.322	14.419	24.289	14.419
3	WF3	0(Toe)	NC	0.034(Heel)	1	0.012	1	0.012	1	13.334	14.48	13.334	14.48
4	WF4	0(Toe)	NC	0.034(Heel)	1	0.012	1	0.012	1	13.334	14.48	13.334	14.48

The **Wall Footing Code Check Spreadsheet** gives the flexural and shear checks and capacities for both the footing toe and heel design. For more information on the design of these portions, see the Footing Design Considerations section of the Wall Footings - Design topic and the Detail Report section.

Wall Code Check (Concrete)

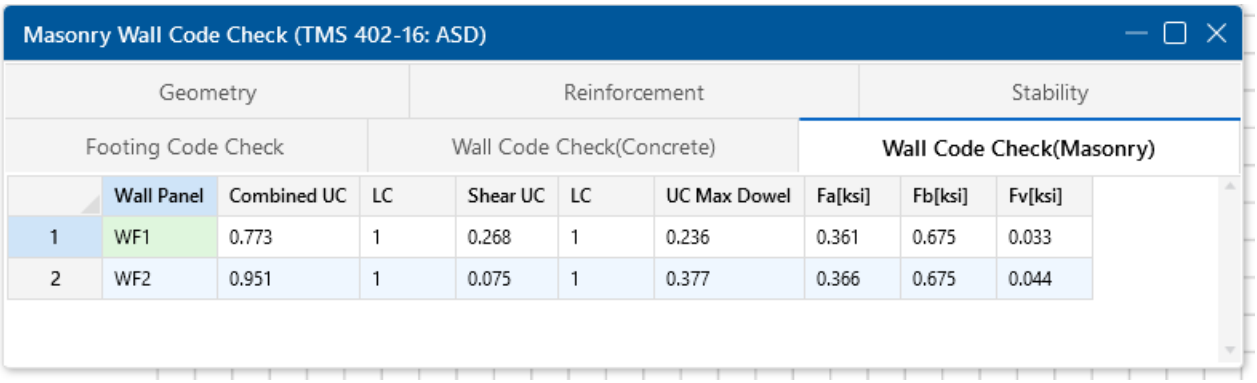


	Label	UC Max Int	UC Int LC	UC Max Ext	UC Ext LC	Max Shear UC	Max Shear LC	UC Max Dowel	Phi * Mn Int[k-ft/ft]	Phi * Mn Ext[k-ft/ft]	Wall Phi *Vn[k/ft]
1	WF1	0.938	2	0.013	1	0.488	2	0.151	41.295	0	14.357
2	WF2	0.924	2	0.013	1	0.488	2	0.151	41.927	0	14.357
3	WF3	0.004	1	0.004	1	0	1	0	26.803	0	14.48
4	WF4	0.004	1	0.004	1	0	1	0	19.621	0	14.48

The **Wall Code Check (Concrete) Spreadsheet** gives bending and shear code checks, as well as capacity values for the wall stem. To get a detailed sketch of this information, see the Detail Report.

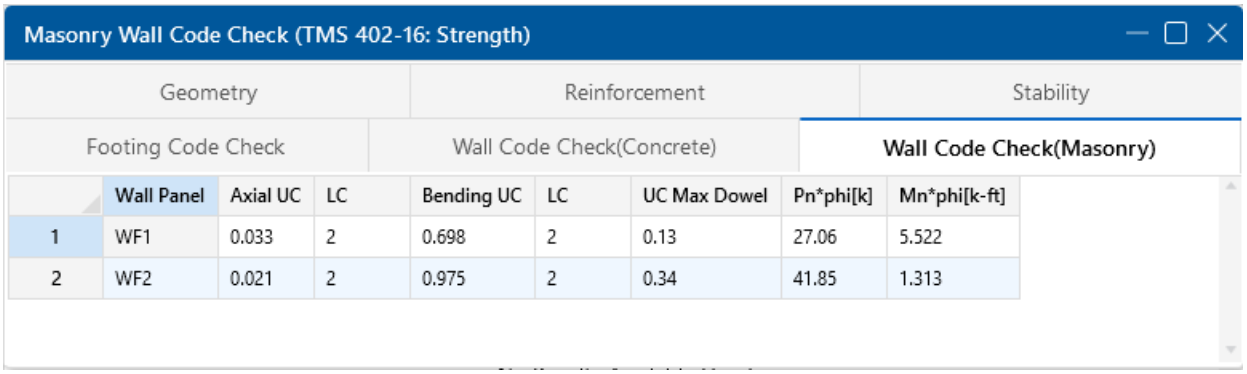
Masonry Wall Results

Service



	Wall Panel	Combined UC	LC	Shear UC	LC	UC Max Dowel	Fa[ksi]	Fb[ksi]	Fv[ksi]
1	WF1	0.773	1	0.268	1	0.236	0.361	0.675	0.033
2	WF2	0.951	1	0.075	1	0.377	0.366	0.675	0.044

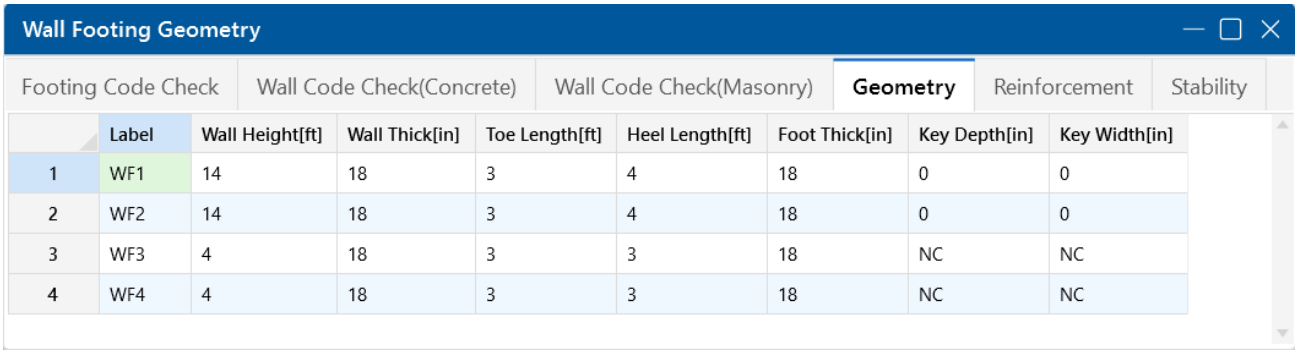
Strength



	Wall Panel	Axial UC	LC	Bending UC	LC	UC Max Dowel	Pn*phi[k]	Mn*phi[k-ft]
1	WF1	0.033	2	0.698	2	0.13	27.06	5.522
2	WF2	0.021	2	0.975	2	0.34	41.85	1.313

The **Wall Code Check (Masonry) Spreadsheet** gives bending and shear code checks for the wall stem. To get a detailed sketch of this information, see the Detail Report.

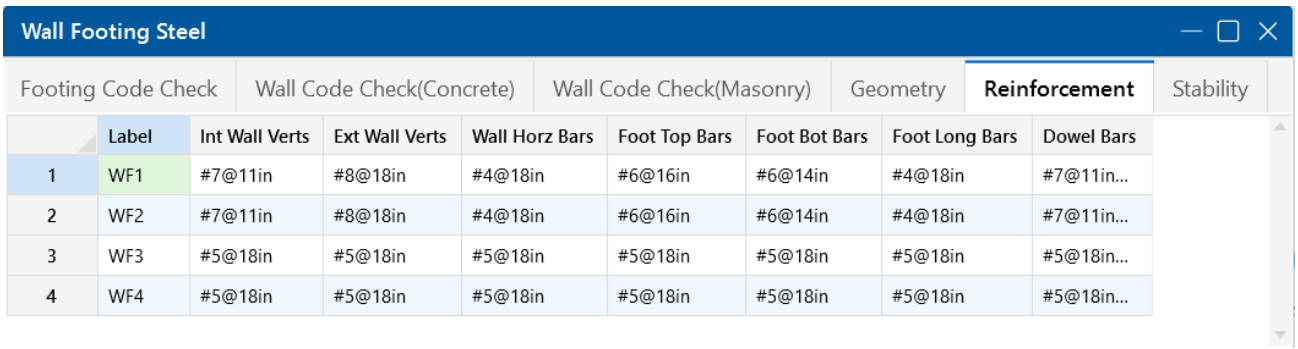
Geometry



	Label	Wall Height[ft]	Wall Thick[in]	Toe Length[ft]	Heel Length[ft]	Foot Thick[in]	Key Depth[in]	Key Width[in]
1	WF1	14	18	3	4	18	0	0
2	WF2	14	18	3	4	18	0	0
3	WF3	4	18	3	3	18	NC	NC
4	WF4	4	18	3	3	18	NC	NC

The **Wall Footing Geometry Spreadsheet** gives a summary of the overall geometry of the wall and footing. This is a reiteration of input parameters.

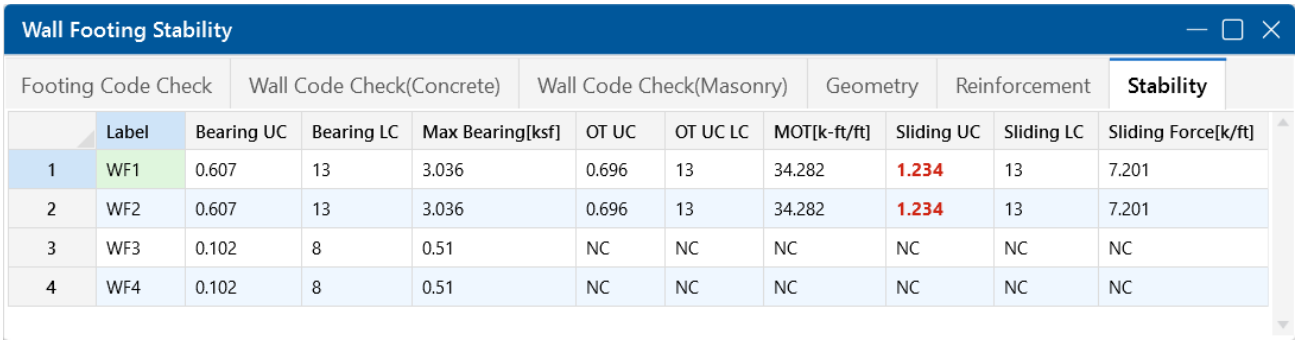
Reinforcement



	Label	Int Wall Verts	Ext Wall Verts	Wall Horz Bars	Foot Top Bars	Foot Bot Bars	Foot Long Bars	Dowel Bars
1	WF1	#7@11in	#8@18in	#4@18in	#6@16in	#6@14in	#4@18in	#7@11in...
2	WF2	#7@11in	#8@18in	#4@18in	#6@16in	#6@14in	#4@18in	#7@11in...
3	WF3	#5@18in	#5@18in	#5@18in	#5@18in	#5@18in	#5@18in	#5@18in...
4	WF4	#5@18in	#5@18in	#5@18in	#5@18in	#5@18in	#5@18in	#5@18in...

The (**Reinforcement** tab gives all of the steel requirements for the wall footing, including wall, footing and dowel reinforcement.

Stability



	Label	Bearing UC	Bearing LC	Max Bearing[ksf]	OT UC	OT UC LC	MOT[k-ft/ft]	Sliding UC	Sliding LC	Sliding Force[k/ft]
1	WF1	0.607	13	3.036	0.696	13	34.282	1.234	13	7.201
2	WF2	0.607	13	3.036	0.696	13	34.282	1.234	13	7.201
3	WF3	0.102	8	0.51	NC	NC	NC	NC	NC	NC
4	WF4	0.102	8	0.51	NC	NC	NC	NC	NC	NC

The **Wall Footing Stability Spreadsheet** gives information for soil bearing, overturning and sliding checks for wall footings.

For the **Soil Bearing Checks**, the **Bearing UC** column is meant to act as the code check value. Anything exceeding 1.0 here means you are failing this check.

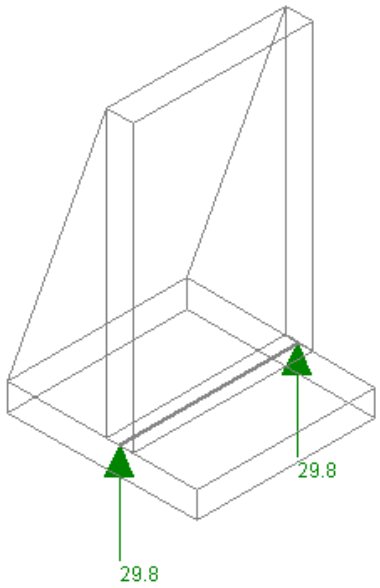
Note:

- The **Allowable Bearing Pressure** value is defined for each soil region in the model. If your wall footing lands on multiple soil regions, the smallest allowable bearing pressure will be used.
- If you fail the soil bearing checks we will not give a full solution to your model. We will, however, give you the soil pressure diagrams so you can decide how you want to reconcile this failure.

For more information on the specifics of the overturning and sliding checks, see the Footing Design Considerations section of the Wall Footings - Design topic, as well as the Detail Report.

Graphic Results

The reactions for the wall footing can be viewed in the Model Display Options dialog. For wall footings, only the vertical reactions are given. The total vertical load is calculated and the display divides that value by two and places reactions at the ends of the walls.

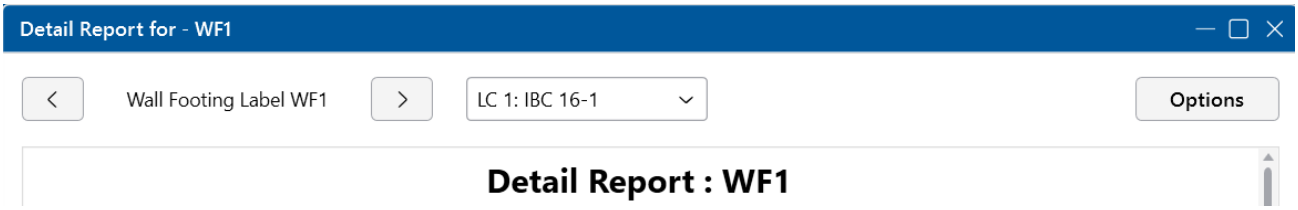


Note: Because there may be the presence of a key and because the program doesn't know which reactions you are looking for laterally, the program does not give these values. To find these values look at the detail report for that wall footing.

Wall Footing Detail Report Information

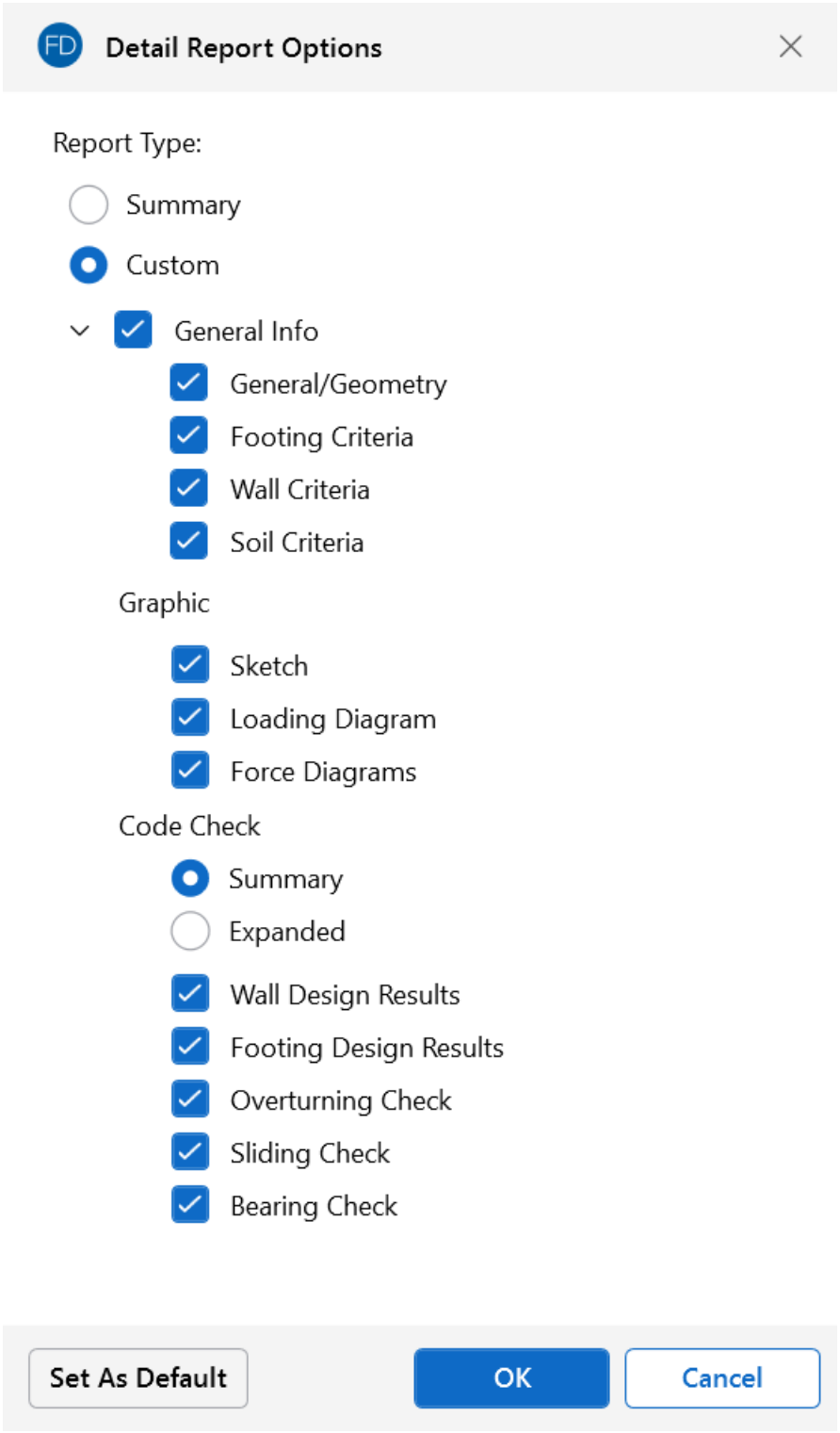
The Retaining Wall Detail Report provides detailed information about geometry, materials, criteria, design and stability checks and sketches associated with the retaining wall. Here we will walk through each section of the report.

Header



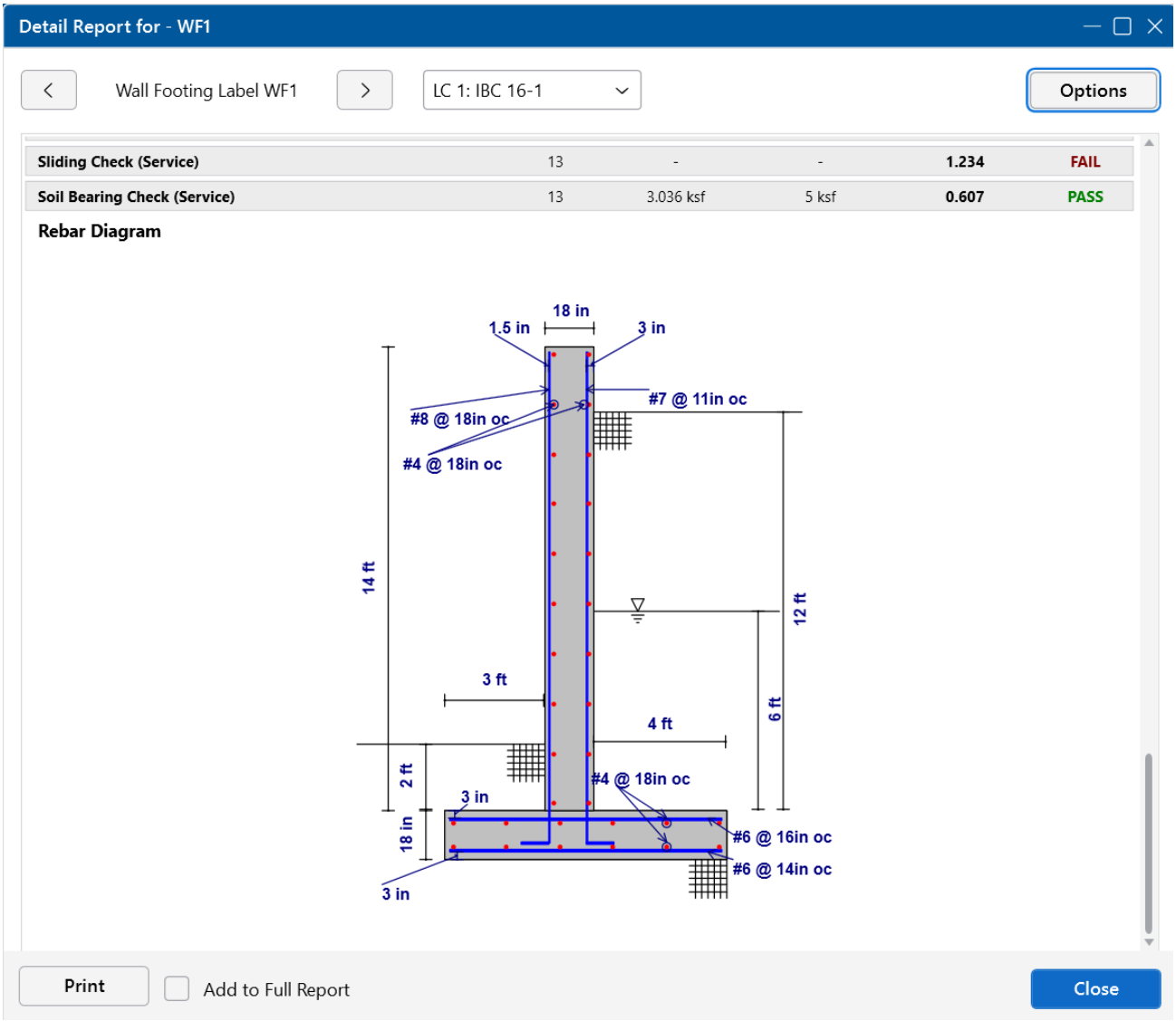
The header of the Detail Report is useful for navigating and viewing information in the wall footing reports.

- The **LC** drop-down allows you to view the overall loading diagram and the axial, shear and moment diagrams for the stem by individual load combination. Every other item in the Detail Report is enveloped, or results are given for each LC individually.
- The **arrows** allow you to click through Detail Reports for the different wall footings in the model.
- The **Options** button opens up the **Detail Report Options** dialog, shown below. This allows you to customize exactly what you see in the Detail Report.


Detail Report Options
Report Type:
Summary
Custom
General Info
General/Geometry
Footing Criteria
Wall Criteria
Soil Criteria
Graphic
Sketch
Loading Diagram
Force Diagrams
Code Check
Summary
Expanded
Wall Design Results
Footing Design Results
Overturning Check
Sliding Check
Bearing Check
Set As Default
OK
Cancel

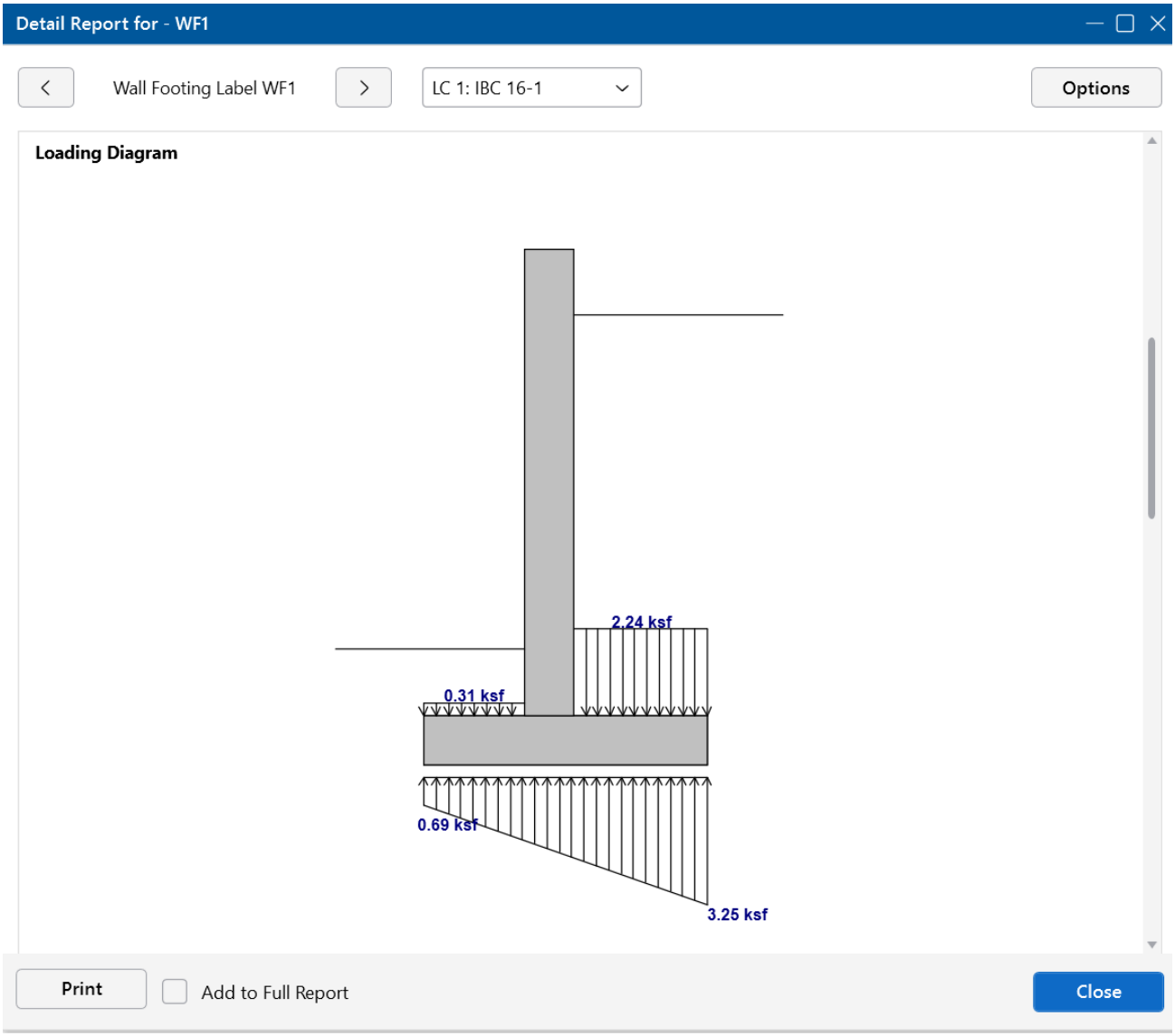
- The **Print** button opens the print dialog to print the report.
- The **Add to Full Report** check box will include the Detail Report in the available output when printing your overall project report.

Rebar Diagram



The **Rebar Diagram** section gives a full sketch of wall footing dimensions, reinforcement size and spacing and cover dimensions.

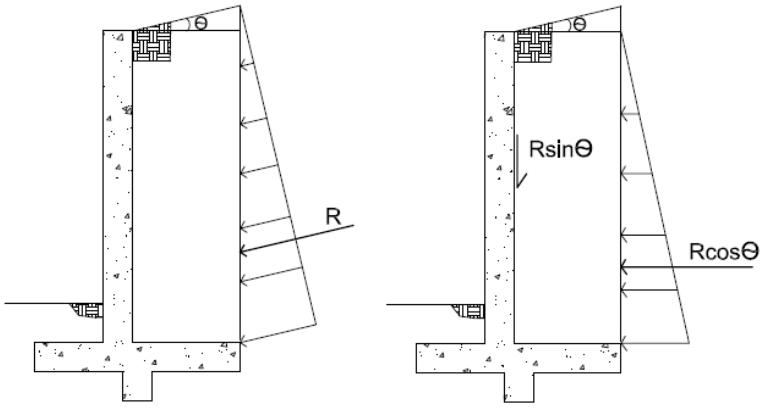
Loading Diagram



The **Loading Diagram** section gives a visual of all the calculated pressures and loading on the wall. This loading information is what is used to provide the design checks for the wall footing. Loads shown on the top of the wall footing are any externally applied loads to the wall, divided out on a per foot basis. For more information on the calculation of these pressures, see the Footing Soil Pressure Considerations and Wall Pressure Considerations sections in the Wall Footings - Design topic.

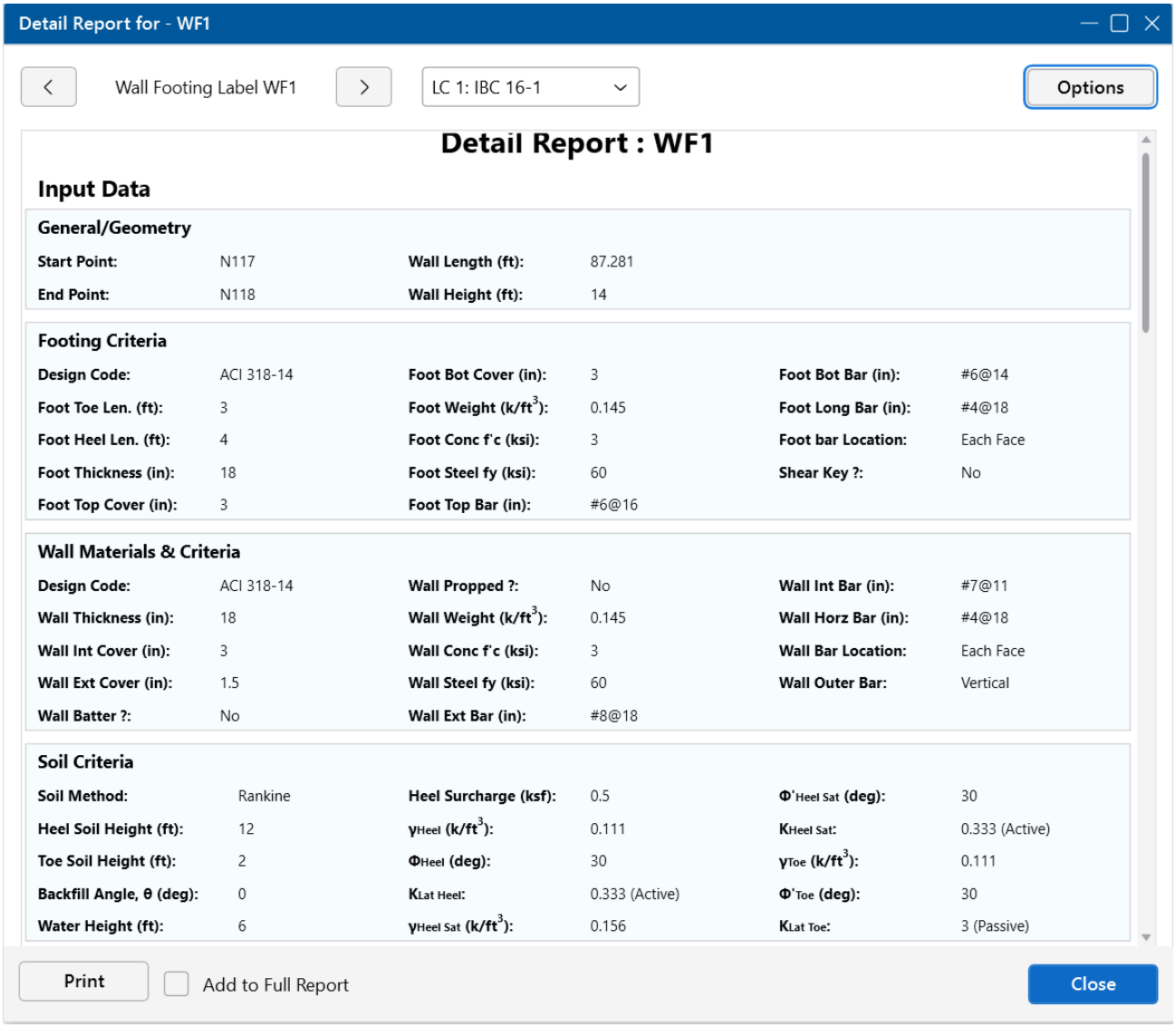
Note:

- This diagram is based on the load combination shown in the dropdown at the top of the report.
- If your retaining wall has a sloped, a portion of the hydrostatic loads will be applied in the vertical direction of your wall. The cosine of these forces will be applied horizontally as a pressure. The sine of these forces will be applied as a vertical force that is shown along the inside face of the wall and conservatively applied at the top of the wall.

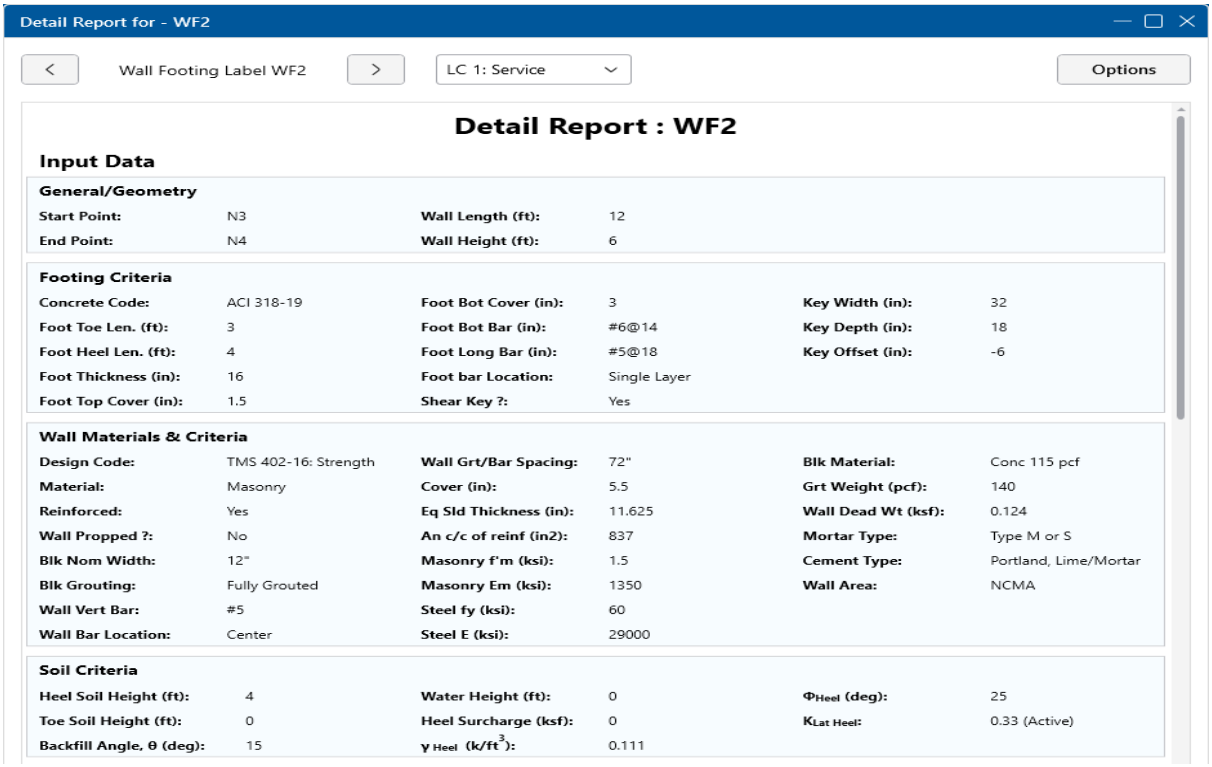


Input Data

Concrete Wall



Masonry Wall



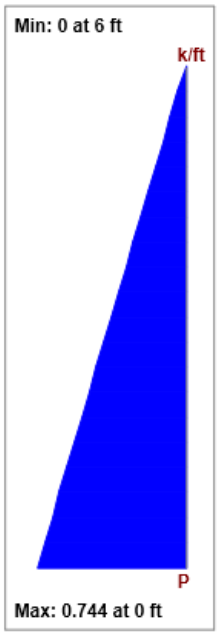
The **Input Data** section provides input information from the Wall Footing Definitions and design information, such as the reinforcement spacing and K calculations (see Soil Considerations).

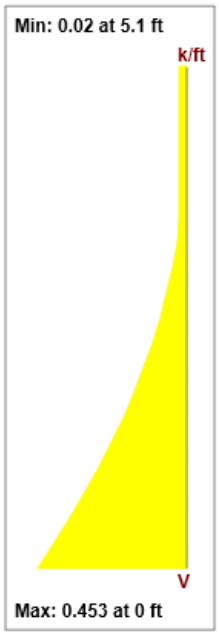
Wall Design

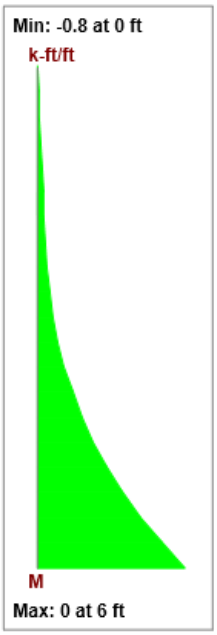
Concrete Wall

Force Diagrams

LC 1 DIAGRAMS







Axial/Bending Details Int(+z)			2	-	-	**0.938**	**PASS**
UC Max Int(+z):	0.938	**Φ *Pn Int(+z) (k/ft):**	0		**Φ eff. Int(+z):**	0.9	
Location (ft):	0	**Gov Mu Int(+z) (k-ft/ft):**	-38.737		**Gov LC Int(+z):**	2	
Gov Pu Int(+z) (k/ft):	0	**Φ *Mn Int(+z) (k-ft/ft):**	41.295				
Axial/Bending Details Ext(-z)			1	-	-	**0.013**	**PASS**
UC Max Ext(-z):	0.013	**Φ *Pn Ext(-z) (k/ft):**	321.829		**Φ eff. Ext(-z):**	0.65	
Location (ft):	0	**Gov Mu Ext(-z) (k-ft/ft):**	0		**Gov LC Ext(-z):**	1	
Gov Pu Ext(-z) (k/ft):	4.263	**Φ *Mn Ext(-z) (k-ft/ft):**	0				
Wall Reinforcement			-	-	-	-	-
As Provided(H) (in^2):	3.927 (20 #4)	**Int As Provided(V) (in^2):**	57.125 (95 #7)		**As min(V) (in^2):**	28.279	
ρ Provided(H):	0.0013	**Int ρ Provided(V):**	0.003		**ρ min(V):**	0.0015	
As min(H) (in^2):	6.048	**Ext As Provided(V) (in^2):**	46.338 (59 #8)				
ρ min(H):	0.002	**Ext ρ Provided(V):**	0.0025				
As provided(H) does not meet code required minimum.							
Shear Details			2	7.003 k/ft	14.357 k/ft	**0.488**	**PASS**
UC Max:	0.488	**Φ*Vn (k/ft):**	14.357		**μ:**	1	
Location (ft):	1.4	**Gov LC:**	2		**A_{vf} (in^2/ft):**	1.18	
Gov Vu (k/ft):	7.003	**UC Max Dowel:**	0.151		**Dowel Bars (in):**	#7@11 Int #8@18Ext	
Enveloped Wall Forces				-	-	-	-
M in Plane (k-ft):	0 (LC : 1)	**V in Plane (k):**	0 (LC : 1)				
Enveloped Wall Forces Not Used For Design.							

The **Axial, Shear** and **Moment Diagrams** are based on each load combination from the drop-down list.

The **Axial/Bending Checks** are given for both the interior and exterior faces of the wall. Note that only a propped cantilevered wall will have any bending in the "Gov Mu Ext", unless there is a large externally applied load in the direction opposite the soil loading. The code checks here are based on a concrete wall interaction diagram and bending capacity checks also consider compression reinforcement.

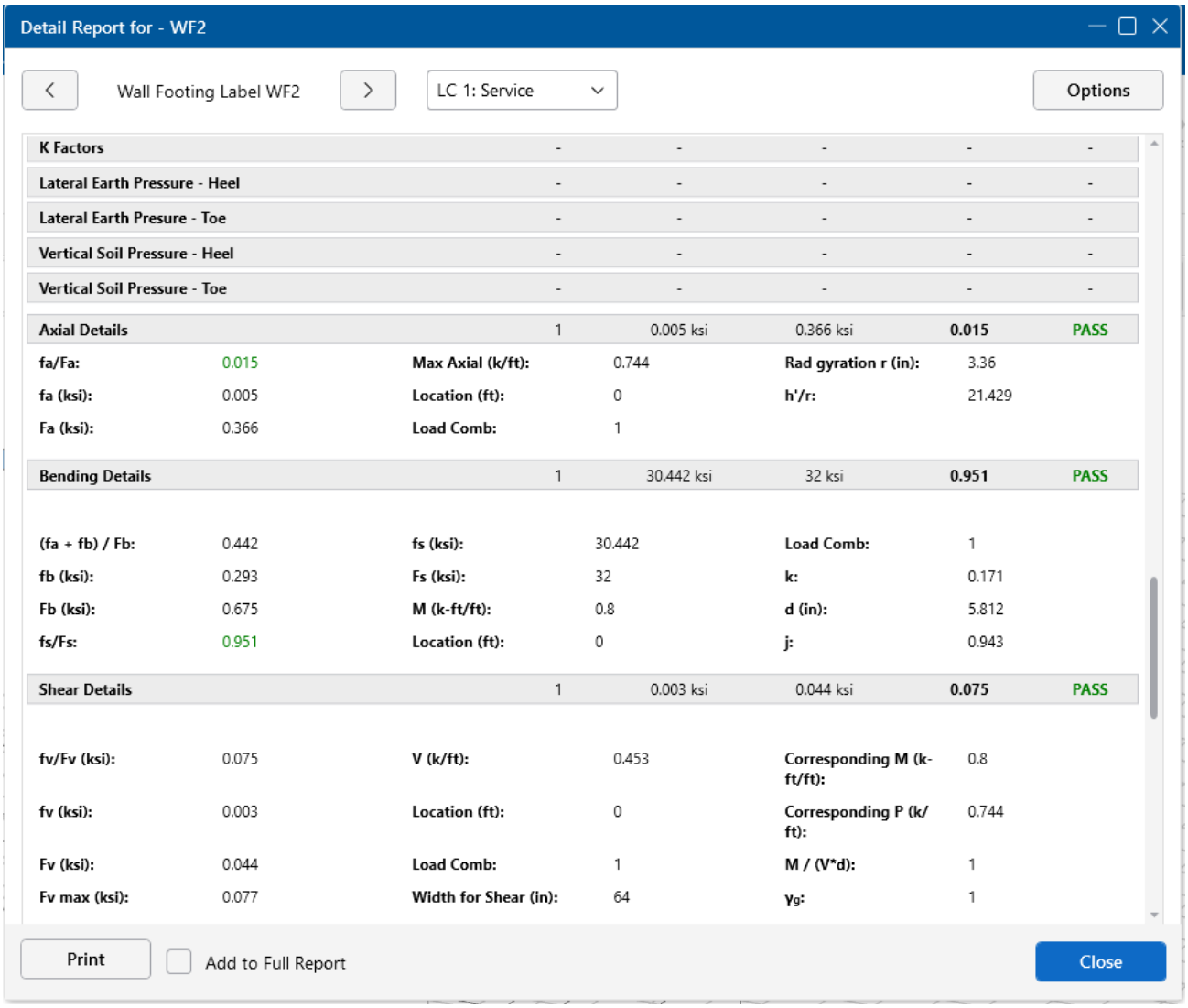
The **Wall Reinfrocement** section provides the minimum reinforcement requirements and the reinforcement provided for the wall.

The **Shear Details** provide both the concrete shear check (at a distance "d" from the base of the wall) and the dowel shear checks (at the base of the wall). Note that the dowel checks are only done for retaining walls with a footing that is not defined as monolithic with the wall. For more information, see Wall Footing Definitions - General.

The **Enveloped Wall Forces** section gives other forces found in the wall that the program did not explicitly use. For example, the wall footing will not be designed for in plane shears, but those forces will be reported here.

Note: If the minimum spacing requirements defined in the **Wall Footing Definition Editor** does not meet minimum code requirements, you will receive warning messages in the Detail Report.

Masonry Wall (ASD)



The **Axial and Bending Details** give the values used in the axial and flexural code checks at the location which controls the combined check (fa/Fa)+(fb/Fb).

For ASD design, the maximum bending stress in the flexural reinforcement is reported as **fs**, and the allowable steel bending stress as **Fs**. These values are given for the Load Combination and section that produce the maximum code check (fs/Fs).

The **Shear Details** give the maximum shear demand as **fv**. The allowable shear stresses are then reported as **Fv**.

Design Details

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.

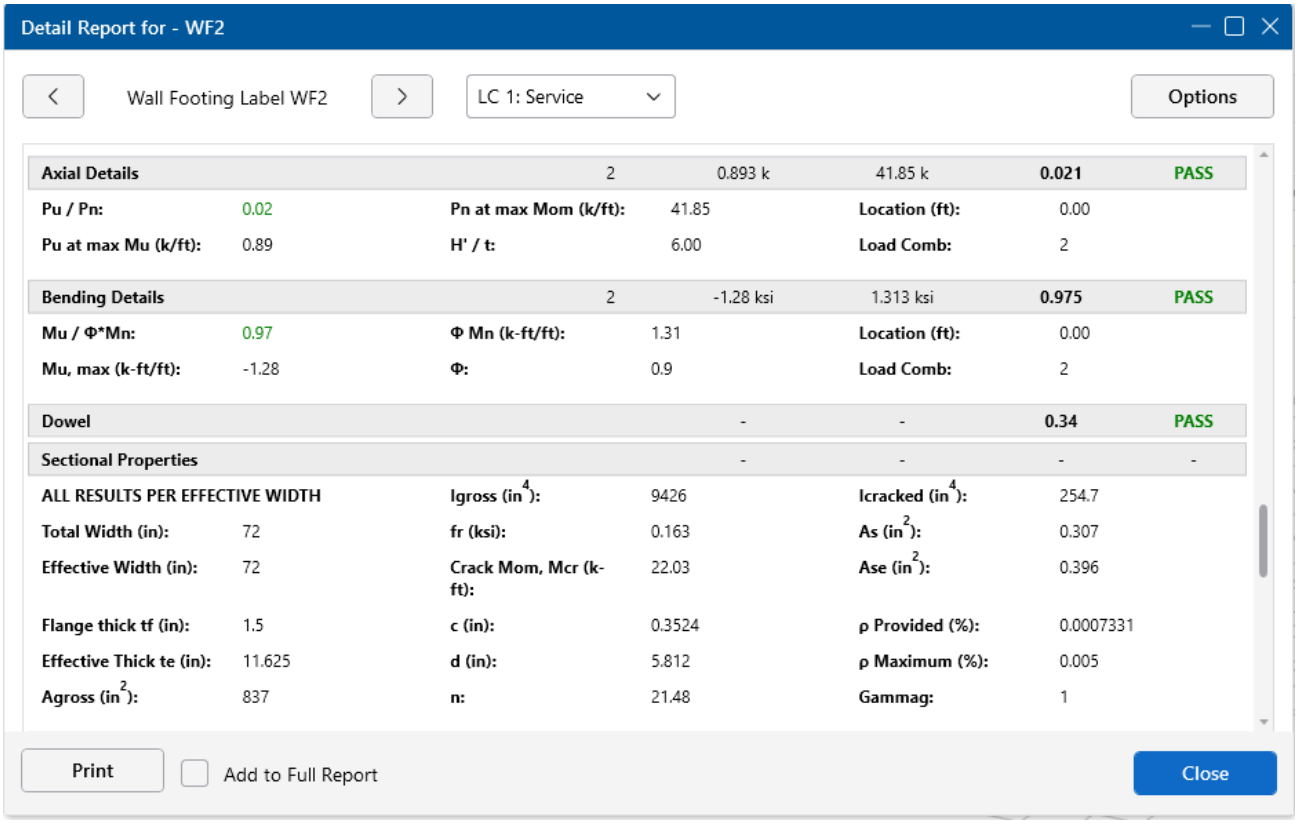
j: This value multiplied by d gives the moment arm between the compression and tension resultants.

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 wall, 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.

Masonry Wall (Strength)



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 either Section 9.3.5.4.2 or Equations 9-11 and 9-12.

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) or Pu/Pn 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.

For more information on a T-section analysis, see Wall Footings - Design.

Note: An over-reinforced section 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.

The **Sectional Properties** provides design values based on the center to center spacing of reinforcement and are explained below:

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.

Footing Design

Footing Design			2	20.992 (k-ft/ft)	21.322 (k-ft/ft)	0.985	PASS
As-min z-dir (Top Flexure) (in^2/ft):	0.389			z-dir (T & S) (in^2/ft):	0.389		
As-min z-dir (Bot Flexure) (in^2/ft):	0.389			x-dir Long. (T & S) (in^2):	3.305		

Bottom Bar Design (Toe) : Shear (Toe)

Mu UC Max	Gov Mu LC	Max Mu (k-ft/ft)	Φ*Mn (k-ft/ft)	As Req'd (in^2/ft)	As Prov'd (in^2/ft)	Vu UC Max	Gov Vu LC	Vu (k/ft)	Φ*Vn (k/ft)
0.714	5	17.353	24.289	0.269	0.379	0.489	5	7.047	14.419

Top Bar Design (Heel) : Shear (Heel)

Mu UC Max	Gov Mu LC	Max Mu (k-ft/ft)	Φ*Mn (k-ft/ft)	As Req'd (in^2/ft)	As Prov'd (in^2/ft)	Vu UC Max	Gov Vu LC	Vu (k/ft)	Φ*Vn (k/ft)
0.985	2	20.992	21.322	0.326	0.331	0.631	2	9.093	14.419

Footing longitudinal rebar spacing does not meet T & S minimum.

This section provides an enveloped report of the footing design for both the top and bottom bars. The top of this section provides the flexural and temperature & shrinkage minimums. For the toe and heel, we provide shear and moment checks for the worst case load combination. For more information on footing design considerations, see Wall Footings - Design.

Note:
- If the bottom bar is in tension in any solved combination for both the toe and the heel, the program will envelope the worst case reinforcement for both the toe and the heel.
- If the minimum spacing requirements defined in the Retaining Wall Definition do not meet minimum code requirements, you will receive a warning message in the Detail Report.

Overturning Check (Service)

Overturning Check (Service)			13	-	-	0.696	PASS
Description	**Categories and Factors**	**Moverturn (k-ft/ft)**	**Mresist (k-ft/ft)**	**SF**	**SF Min**	**SF Min/SF**	
IBC 16-8	1DL	0	61.785	0	1.5	0	
IBC 16-9	1DL+1HL+1LL +1LLS	34.282	77.155	2.251	1.5	0.666	
IBC 16-10 (a)	1DL+1HL+1RLL	19.095	64.156	3.36	1.5	0.446	
IBC 16-10 (b)	1DL+1HL+1SL +0.75..	19.095	64.156	3.36	1.5	0.446	
IBC 16-10 (c)	1DL+1HL+1RL	19.095	64.156	3.36	1.5	0.446	
IBC 16-11 (a)	1DL+1HL +0.75LL+0..	34.282	73.905	2.156	1.5	0.696	
IBC 16-11 (b)	1DL+1HL +0.75LL+0..	34.282	73.905	2.156	1.5	0.696	
IBC 16-11 (c)	1DL+1HL +0.75LL+0..	34.282	73.905	2.156	1.5	0.696	

This section gives overturning results for all service load combinations that are solved. The **SF Min/SF** column is meant to act as the code check value. Anything exceeding 1.0 here means you are failing this check. See the **Serviceability Considerations** section of the Wall Footings - Design topic for more information.

Note: The **SF Min** column is the *minimum safety factor*. This value is defined for each load combination separately in the **Load Combinations Spreadsheet**.

Sliding Check (Service)

Sliding Check (Service)				13	-	-	1.234	FAIL
Description	Categories and Factors	Vsliding (k/ft)	Vresist (k/ft)	SF	SF Min	SF Min/SF		
IBC 16-8	1DL	0	5.972	0	1.5	0		
IBC 16-9	1DL+1HL+1LL +1LLS	7.201	9.004	1.25	1.5	1.2		
IBC 16-10 (a)	1DL+1HL+1RLL	4.951	8.004	1.617	1.5	0.928		
IBC 16-10 (b)	1DL+1HL+1SL +0.75..	4.951	8.004	1.617	1.5	0.928		
IBC 16-10 (c)	1DL+1HL+1RL	4.951	8.004	1.617	1.5	0.928		
IBC 16-11 (a)	1DL+1HL +0.75LL+0..	7.201	8.754	1.216	1.5	1.234		
IBC 16-11 (b)	1DL+1HL +0.75LL+0..	7.201	8.754	1.216	1.5	1.234		
IBC 16-11 (c)	1DL+1HL +0.75LL+0..	7.201	8.754	1.216	1.5	1.234		

This section gives sliding results for all service load combinations that are solved. The **SF Min/SF** column is meant to act as the code check value. Anything exceeding 1.0 here means you are failing this check. For more information, see the **Serviceability Considerations** section in Wall Footings - Design.

Note: The **SF Min** column is the *minimum safety factor*. This value is defined for each load combination separately in the **Load Combinations Spreadsheet**.

Soil Bearing Check (Service)

Soil Bearing Check (Service)			13	3.036 ksf	5 ksf	0.607	PASS
Description	Categories and Factors	Max Bearing (ksf)	Bearing Allowed (ksf)	Bearing UC			
IBC 16-8	1DL	2.321	5	0.464			
IBC 16-9	1DL+1HL+1LL+1LLS	3.001	5	0.6			
IBC 16-10 (a)	1DL+1HL+1RLL	1.879	5	0.376			
IBC 16-10 (b)	1DL+1HL+1SL+0.75..	1.879	5	0.376			
IBC 16-10 (c)	1DL+1HL+1RL	1.879	5	0.376			
IBC 16-11 (a)	1DL+1HL+0.75LL+0..	3.036	5	0.607			
IBC 16-11 (b)	1DL+1HL+0.75LL+0..	3.036	5	0.607			
IBC 16-11 (c)	1DL+1HL+0.75LL+0..	3.036	5	0.607			

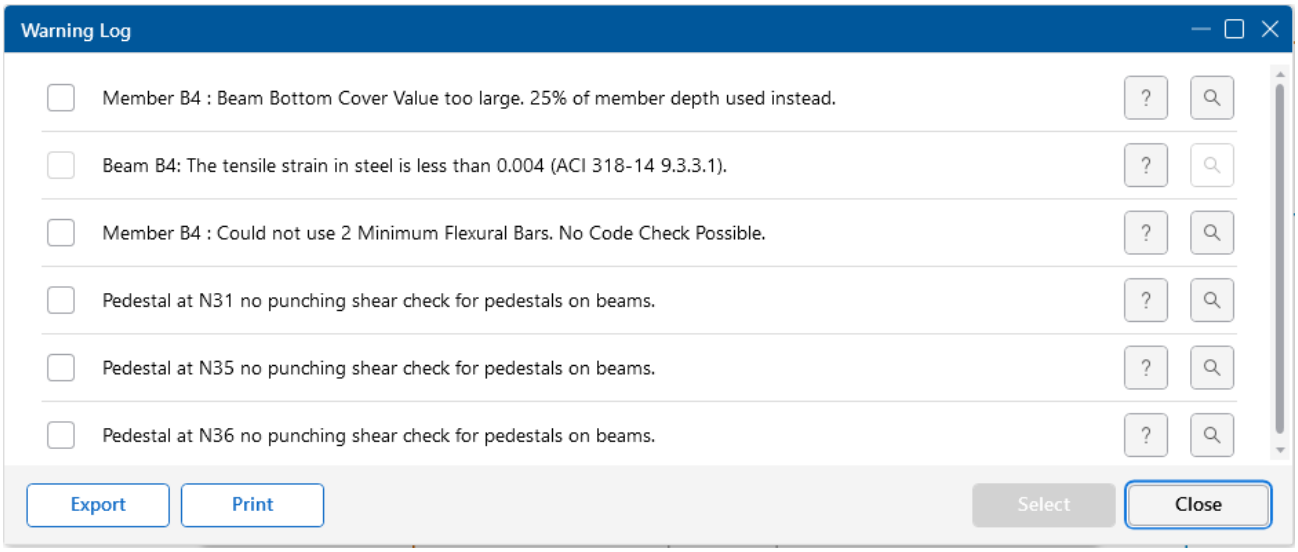
Rebar Diagram

This section gives soil bearing check results for all service load combinations that are solved. The **Bearing UC** column is meant to act as the code check value. Anything exceeding 1.0 here means you are failing this check.

Note:

- The **Bearing Allowed** column is the *allowable bearing pressure*. This value is defined for each soil region in the model.
- If your wall footing lands on multiple soil regions, the smallest allowable bearing pressure will be used.
- If you fail the soil bearing checks, we will not give a full solution to your model. We will, however, give you the soil pressure diagrams so you can decide how you want to reconcile this failure.

Warning Log



The **Warning Log Spreadsheet** provides you with a record of any warnings or errors that occurred during the solution of your model. The log should be reviewed for warnings or errors that would affect the design of your structure. The error log reports back the item label for which the error occurred.

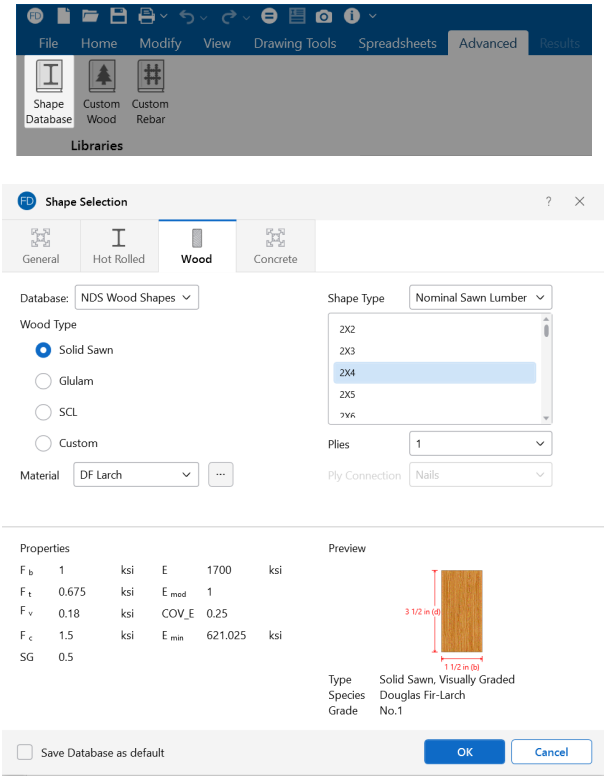
You can view the warning log by clicking on the **Spreadsheets** menu item and then clicking on the **Warning Log** selection.

Pressing the **?** will open the help file to give further explanation on this item.

*Please refer to the **Help File** in RISAFoundation for detailed information about each error message. The Help File may be accessed by selecting in the upper right-hand corner. Choose the **Warning Log** topic*

Wood - Database

The Wood Database may be accessed by clicking the **Shape Database** button on the **Advanced** tab of the ribbon and then clicking the **Wood** tab of the **Shape Database** dialog.



Wood Material/Shape Selection

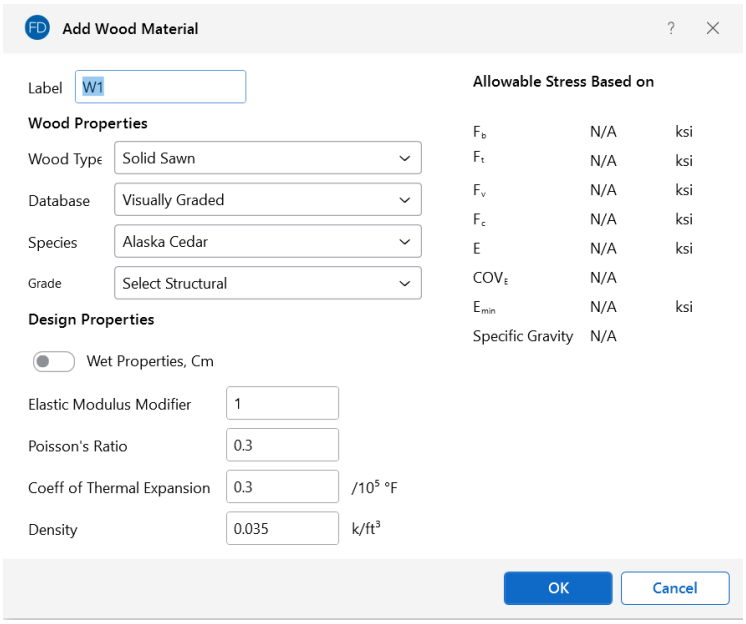
Unlike any other material, the design properties for wood members depends both on the material selection, as well as the shape selection. Therefore, the **Wood** tab of the **Shape Selection** database contains shape and material selections.

Material Selection

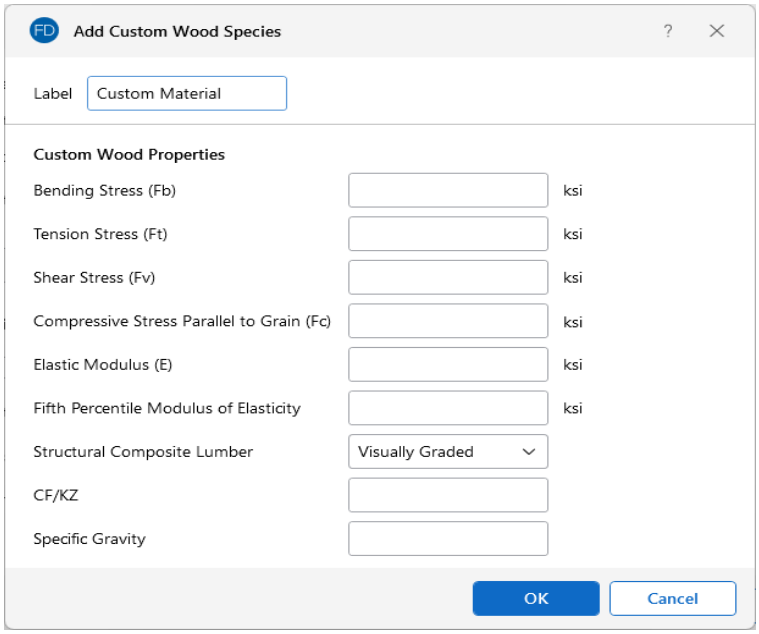
The **Material Selection** menu reads in material options that are currently listed on the **Wood** tab of your **Materials Spreadsheet**. You may select one of these existing materials, or click on the ... (ellipsis) button to open the **Review or Change Material** dialog, which will allow you to add in a new material, or edit an existing material without having to go to the **Materials Spreadsheet**.

Create/Edit Wood Material dialog

The **Create/Edit 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** Property panel without having to first manually enter the information in the **Materials Spreadsheet**.



To create a new custom wood material, first choose **Custom** from the **Wood Type** drop-down. Then, click on the **Add** button to manually enter the design values for a new custom wood material. This information will then be automatically saved to the Custom Wood Species Spreadsheet.



Size Selection

Currently there are two ways to define a member size. In the **Shape Type** drop-down, the options available are from either the defined list of **Nominal Sawn Lumber** sizes, or by using the **Net Rectangular/Round option** (typically referred to as Full Sawn).

- The **Nominal Sawn Lumber** option will populate a list of nominal size 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 **Net Rectangular** or **Net Round** option will 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 **Ply Connection** drop-down menu. If the bolted option is selected 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, 2018 and 2024. If you define the multi-ply section as bolted a "B" (or "MB" for metric sizes) will appear after the shape name.

View Design Properties

Once you have selected the appropriate member size and material, you can view the un-adjusted wood design values (Fb, Fv, Ft, etc.) for the selected options in the **Properties** table at the bottom of the dialog. The final adjusted values (Fb', Fv',Ft', etc.) will then be displayed after solution in the 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.

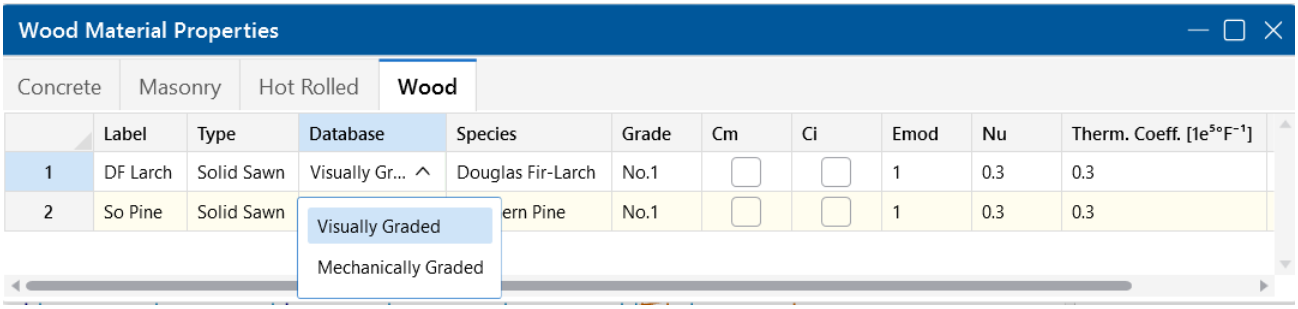
Glulam Databases

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, 2018 and 2024* 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 2024 Table 4C* which says that all species from *NDS 2024 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 2024 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 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:

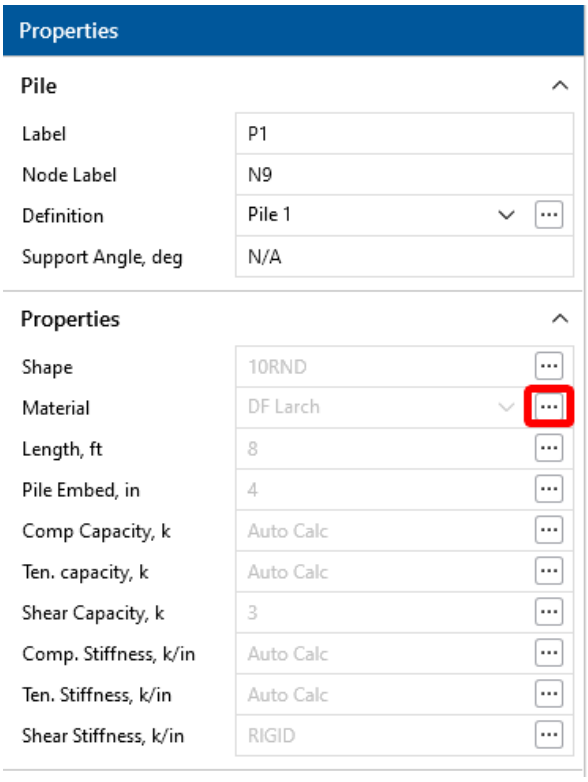
Each of the above reports includes tables with design values which were used to create the RISA SCL design property material databases. However, due to the difficulty of design variation, the following limitations are present:

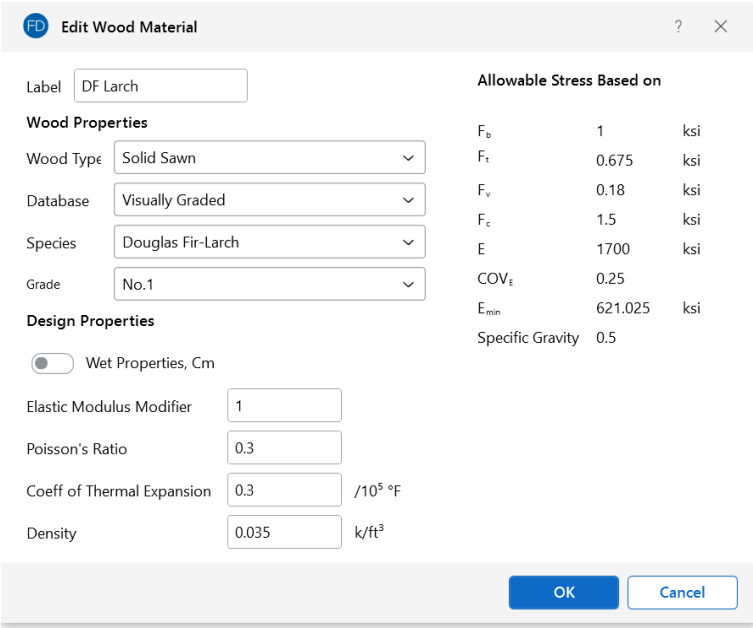
- The Fb design values are not factored in accordance with the depth of the member. Currently, RISA always uses the single tabular Fb value regardless of member size.
- The Ft design values are not factored in accordance with the length or thickness of the member. Currently, RISA always uses the single 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 the  (ellipsis) button next to the **Material** section of the specific member's information in the **Property** panel.





The values are as follows:

- **Fb** - Bending Stress Limit
- **Ft** - Tension Stress Limit

- **Fv** - Shear Stress Limit
- **Fc** - Compression (Parallel to Grain) Stress Limit
- **E** - Modulus of Elasticity
- **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.

NDS Emin Calculation

RISA will calculate the Emin value for NDS wood materials rather than read it in from the design tables. In the 2024 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:

$$Emin = E * (1 - 1.645 * COV_E) * \left(\frac{1.05}{1.66}\right)$$

Note

Currently, the program only designs using the *Joist* properties of the SCL materials. This assumes that the loading is always on the small dimension side of the member. In a future version we will be taking the Plank values into consideration as well. In the meantime you can always enter the appropriate design values from these tables as a Custom Wood Material for use in your model.

Note: Canadian wood member design only includes SCL design for the CSA O86-14 code.

Custom Wood Sizes

If you would like to enter explicit dimensions of a member or if the member is "Not Dressed", the member must be designated as a "Full Sawn" member by choosing **Net Rectangular** as the **Shape Type** and entering the exact dimensions. This applies to regular wood species, custom wood species, Structural Composite Lumber (SCL) and Glu-Lam members.

Wood - Design

Full code checking can be performed on Dimension Lumber and Post and Timber size wood shapes based on the following codes:

- The 2024 edition of the NDS (with 2012 SDPWS and 2018 or 2024 NDS Supplement)
- The 2018 edition of the NDS (with 2015 and 2021 SDPWS)
- The 2015 edition of the NDS
- The 2012 edition of the NDS
- The 2005/08 edition of the NDS
- The 2001 edition of the NDS
- The 1991 / 1997 editions of the NDS
- The 2014 edition of the CSA O86 Canadian wood design code
- The 2009 edition of the CSA O86 Canadian wood design code

Note: When the 1991 / 1997 NDS is selected, the 1991 NDS specification will be used with the 1997 stress tables. This is consistent with the requirements of the 1997 UBC.

Glu-Lams

Glu-Lams are treated as any other wood species and may be selected from the list of species on the **Wood** tab of the **Materials** spreadsheet.

Glulam Material Properties

Available Glulam Materials are per Tables 5A and 5C of the NDS Supplement and Table 6.3 of the CSA O86. When a Glu-Lam is selected, the grade will be listed as "na" or not applicable.

If you prefer to use a material that is not listed in the design code glulam tables, please enter the material type as a Custom Wood Species.

Note:

- Glu-Lams from Table 5A are always assumed to have the special tension laminations. Therefore, the 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, the size will be assumed to be regular dimensional lumber.

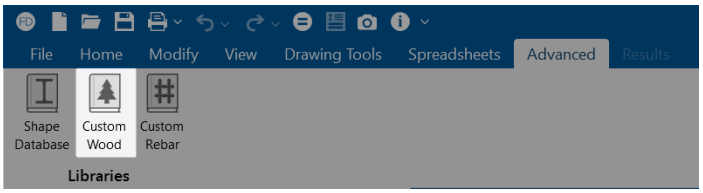
Glulam Limitations

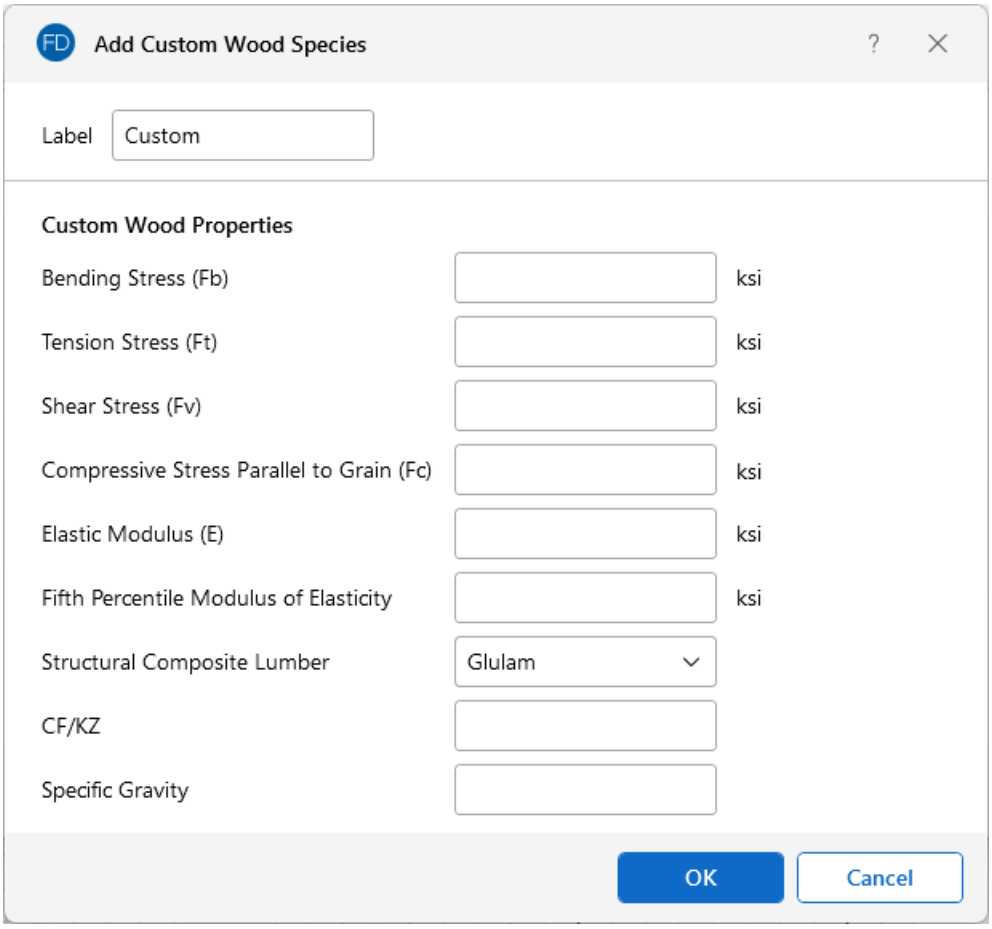
Please note that glulam design is not supported for the 91/97 NDS design code.

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 **Materials Spreadsheet** or by clicking the **Custom Wood** button on the **Advanced** tab.

To access the **Custom Wood Species** input dialogue, click the **Custom Wood** button on the **Advanced** tab.





- **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.

Note:
- 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 completing the following:

1. First, open the **Materials Spreadsheet** and click on the **Wood** tab.
2. Select the last row in the spreadsheet and press **Enter** to create a new entry.
3. Select **Custom** as the material **Type**.
4. Select a custom Wood Species that has been defined in the Custom Wood Species Database from the drop down menu.

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 affects the design factors and the applicable provisions of the selected design code. Please see the **Wood** tab of the **Materials Spreadsheet** 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: **Custom Wood Species**.

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 RISAFoundation 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

RISAFoundation 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 RISAFoundation for detailed information about the file format. The Help File may be accessed by selecting **Help Topics** from the **Help Menu**.*

Technical Support

Technical support is an important part of the RISAFoundation package. There is no charge for technical support for all licensed owners of, or subscribers to, the **current** version of RISAFoundation. 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 RISAFoundation. 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. Click the **Help/Info** button on the Quick Access Toolbar.
3. Visit our website at **risa.com**.
4. Make sure you understand the problem, and make sure your question is related to the program or structural modeling.
5. Take a few minutes to experiment with the problem to try to understand and 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 .FND extension. Make sure you tell us your name, company name, serial number, Key ID, or Cloud ID, phone number and provide a thorough problem description. If you have multiple beams, slabs 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.