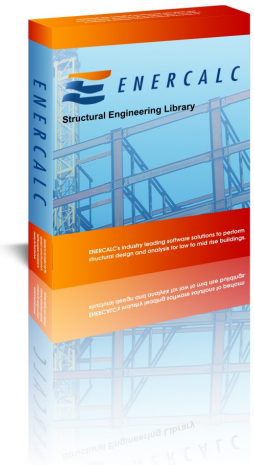
ENERCALC® SEL
Full Mode
User's Manual



ENERCALC, LLC

© 2026 ENERCALC, LLC

ENERCALC SEL

Full Mode

by Michael D. Brooks, S.E., P.E.

A product of

ENERCALC, LLC

ENERCALC SEL
Full Mode

© 2026 ENERCALC, LLC

All rights reserved. No parts of this work may be reproduced in any form or by any means - graphic, electronic, or mechanical, including photocopying, recording, taping, or information storage and retrieval systems - without the written permission of the publisher.

Products that are referred to in this document may be either trademarks and/or registered trademarks of the respective owners. The publisher and the author make no claim to these trademarks.

While every precaution has been taken in the preparation of this document, the publisher and the author assume no responsibility for errors or omissions, or for damages resulting from the use of information contained in this document or from the use of programs and source code that may accompany it. In no event shall the publisher and the author be liable for any loss of profit or any other commercial damage caused or alleged to have been caused directly or indirectly by this document.

Publisher

ENERCALC, LLC

Managing Editor

Chris Conrad, P.E.

ENERCALC, LLC

400 W Broadway Street, STE 101-515
Missoula, MT 59802
(949)-614-0689
(800) 424-2252

Sales: info@enercalc.com
Support : support@enercalc.com
Web : www.enercalc.com

User's Manual
March 2026

Table of Contents

Part I ENERCALC Structural Engineering Library 1

Part II Introduction 3

1 Welcome ... 4
2 Warning & Disclaimer ... 5
3 License Agreement & Copyright ... 5
4 End of Service Policy ... 5
5 RetainPro Mode Limitations ... 6
6 ENERCALC for Revit ... 6
7 License Types ... 7

Part III Access to Cloud Version 9

Part IV Installation (Not required to access cloud version) 12

Part V Activation (Not required to access cloud version) 22

1 Manual Activation ... 29
2 Moving your Activation ... 32

Part VI Software Updates 33

1 Web Update ... 34
2 Update from Website ... 36

Part VII Support & Maintenance 37

1 Subscription vs. Maintenance & Support Plan ... 38
2 Getting Assistance ... 38
3 Troubleshooting with the Browser Console ... 39
4 Viewing Enhancements and Changes to the Software ... 43

Part VIII Getting Started 44

1 System Overview & Design Philosophy ... 45
2 About Our Documentation ... 45
3 Building Codes Supported ... 46
4 Learning Structural Engineering Library ... 46
5 Introductory Videos ... 46

© 2026 ENERCALC, LLC

6 Request for Suggestions ... 47

Part IX Program Overview 48

1 Introduction ... 49
2 Typical Worksession ... 50
3 Program Settings ... 51
4 Files ... 51
5 Projects Tab (Project Assistant) ... 52
6 Project Manager Tab ... 54
7 Current Calculation Tab ... 55
8 General Calculation Screen Usage ... 56
9 Databases ... 57
10 Generating Reports ... 57
11 3D Renderings ... 60
12 Right-Click Context Menu for Tabular Results ... 61

Part X Main Menu 63

1 File ... 64
2 Settings ... 68
Default Values ... 74
3 Databases ... 75
Rolled Steel - AISC Database ... 75
Wood Section Database ... 76
Wood Reference Design Values Database ... 77
Masonry Database ... 78
Load Combination Database ... 78
Block & Geogrid Database ... 84
4 Tools ... 88
5 License ... 91
6 Help ... 91

Part XI Project Manager 95

1 General Division ... 101
Building Code Information ... 101
Project Information ... 104
Client Information ... 104
Designer Notes ... 105
Revision List ... 105
Project Load Group Builder ... 106
2 Loads & Forces Divisions ... 107
3 Calculation List ... 108

© 2026 ENERCALC, LLC

Divisions 111
Adding, Deleting, Copying 113
Sorting by Division, Type & Material 114
Changing Calculation Order 118
Importing Calculations 123
4 Project Printing 125

Part XII Sample Session 128

1 Starting the Program 129
2 Project Assistant 129
3 Creating a Project File 131
4 Entering Project Information 133
5 Setting up your Title Block 134
6 Adding a Calculation 136
7 Viewing the Calculation Screen 138
8 Changing the Default Values/Settings 138
9 Entering Data 139
10 Selecting Sections and Materials from Built-in Databases 155
11 Displaying a Sketch 164
12 Displaying Diagrams 165
13 Displaying a 3D Rendering 168
14 Automatic Member Section Selection 168
15 Printing a Calculation 175
16 Saving a Calculation 179
17 Editing a Division Name and Adding a New Division 179
18 Adding Another Calculation 182
19 Creating a Technical Support Question 187
20 Closing a Project File 190

Part XIII Calculation Modules 191

1 Linked Loads 192
Linked Loads Overview 192
Linked Load Module Support 193
Select a Calculation & Select a Reaction 194
Linking Reactions from Beams 194
Linking Reactions from Columns 195
Edit Link Settings 196
Link Settings for Beams 197
Link Settings for Columns 198

© 2026 ENERCALC, LLC

Link Settings for General Footings ... 199
Linked Reactions Table ... 200
Linked Reactions Table for Beams ... 201
Linked Reactions Table for Columns ... 201
Linked Reactions Table for General Footings ... 202
Status Badges and Linked Load Recalculation ... 203
2 Loads & Forces Divisions ... 207
ASCE Seismic Base Shear ... 210
ASCE 7-22 Seismic Base Shear ... 218
ASCE Seismic Demands on Nonstructural Components ... 224
ASCE 7-22 Seismic Demands on Nonstructural Components ... 224
ASCE Seismic Wall Anchorage ... 225
ASCE 7-22 Seismic Wall Anchorage ... 226
ASCE Wind Enclosure ... 227
ASCE 7-10/16 Wind Forces, Chapter 27, Part 1 ... 228
ASCE 7-10/16 Wind Forces Chapter 28 and 30 ... 240
ASCE 7-22 Wind Forces, Chapter 27 (Directional Procedure) ... 241
ASCE 7-22 Wind Forces, Chapter 28 (Envelope Procedure) ... 241
ASCE 7-22 Wind Forces, Chapter 30, Part 1 (Components & Cladding) 41
IBC Alternate All-Heights Wind Method ... 241
ASCE Snow Loads ... 247
ASCE 7-22 Snow Loads ... 249
ASCE Live and Roof Live Load Reduction ... 252
ASCE 7-22 Live and Roof Live Load Reduction ... 253
Project Load Group Builder ... 254
3 Analysis ... 259
ENERCALC 3D ... 259
License Agreement ... 260
Terms and Conventions ... 260
Related Links ... 261
Introduction ... 261
Graphical User Interface (GUI) ... 263
Spreadsheet Navigation ... 264
Ribbons ... 266
Right-click Menu ... 267
Quick Access Toolbar ... 267
Pan ... 267
Rotate ... 268
Zoom ... 268
Restore Model ... 268
Node Display Options ... 268

© 2026 ENERCALC, LLC

Member Display Options ... 268
Shell Display Options ... 268
Display Options ... 269
Clear All Display Options ... 270
Isometric (and other) View ... 270
Quick Access Tab ... 270
Save ... 270
Save & Close ... 270
Close without Saving ... 270
Undo ... 270
Redo ... 270
Static Analysis ... 271
Node Display Options ... 271
Member Display Options ... 271
Shell Display Options ... 271
Display Options ... 272
Clear All Display Options ... 273
Query ... 273
Hide Selected ... 273
Hide All Except Level ... 273
Hide All Except Plane ... 273
Show All ... 273
Toggle Grid Display ... 273
Pan ... 273
Zoom ... 273
Rotate ... 273
Restore Model ... 273
File ... 273
Save Only ... 274
Save & Close ... 274
Close without Saving ... 274
General Information ... 274
Model Statistics ... 275
DXF Options ... 275
Import from DXF ... 275
Export to DXF ... 276
View Log File ... 276
Create ... 276
Nodes ... 276
Members ... 277
Shells ... 278

© 2026 ENERCALC, LLC

Bricks 279
Boundary Conditions 279
Support 280
Inclined Roller 281
Springs 282
Constraints 283
Diaphragms 283
Coupled Springs 284
Generic Constraints 285
Equal Constraints 286
Model Generators 286
2D Truss/Frame 286
Rectangular Frames 287
Cylindrical Frames 293
Rectangular Shells 295
Circular Shells 297
Arc Members 299
Non-Prismatic Members 300
Rectangular Tank 301
Entities from other Entities 301
Nodes from Grid 301
Members by Nodes 302
Shells by Nodes 304
Bricks by Nodes 304
Grids & Snaps 304
Drawing Grid Setup 305
Toggle Grid Display 306
Snap to Points 306
Perpendicular Point 306
Clear Snap Points 306
Member Properties 306
Member Properties 307
Materials 308
Sections 309
Local Angles 312
Moment Releases 313
Tension/Compression Only 314
Cracking Factors 315
Rigid Offsets 316
Shell Properties 316
Shell Properties 317

© 2026 ENERCALC, LLC

Materials ... 317
Thicknesses ... 318
Element Local Angles ... 319
Story Drift Nodes ... 319
Named Selection ... 320
Levels ... 321
Draw Loads ... 321
Nodal Loads ... 322
Point Loads ... 323
Line Loads ... 324
Area Loads ... 325
Surface Loads ... 326
Thermal Loads ... 326
Self Weights ... 327
Self Weight Exclusion ... 327
Generate Loads ... 327
Fluid Loads ... 328
Pattern Loads ... 329
Moving Loads ... 331
Copy Load Case ... 332
Convert Area Loads to Line Loads ... 332
Convert Local Loads to Global Loads ... 332
Additional Masses ... 333
Response Spectra Library ... 334
Load Cases ... 336
Load Combinations ... 337
Line Select ... 340
Window/Point Select ... 340
Select ... 341
Select Nodes ... 341
Select Members ... 341
Select Shells ... 341
Select Bricks ... 342
Select All ... 342
Unselect All ... 342
Select by Properties ... 342
Select by Materials ... 342
Select by Member Sections ... 343
Select by Member Orientations ... 344
Select Tension/Compression Only Members ... 344
Select by Shell Thickness ... 345

© 2026 ENERCALC, LLC

Select Orphaned Nodes ... 345
Select Elements with Self Weight Excluded ... 345
Select Inactive Elements ... 345
Select Elements by Nodes ... 345
Select by Coordinates ... 346
Select by Selection Names ... 347
Select by RC Beam Design Criteria ... 347
Select by RC Column Design Criteria ... 347
Select by RC Plate Design Criteria ... 347
Select by Steel Design Criteria ... 347
Select by Unity Check Ratios ... 347
Invert Selection ... 347
Advanced Meshing ... 348
Add Region ... 348
Add Hole ... 349
Add Internal Points ... 349
Add Tree ... 350
Edit Region ... 350
Edit Hole ... 351
Edit Internal Points ... 351
Edit Tree ... 352
Delete Region ... 352
Delete Hole ... 353
Delete Internal Points ... 353
Delete Tree ... 354
Clear Mesh Model ... 354
Load Mesh Model from File ... 354
Save Mesh Model to File ... 354
Activate Regions ... 355
Generate Mesh ... 355
Generate Mesh from File ... 356
View Mesh Model ... 356
Annotate Mesh Model ... 356
Modify ... 356
Lock Model ... 356
Copy ... 357
Array ... 358
Mirror ... 359
Move Origin ... 360
Move ... 361
Rotate ... 362

© 2026 ENERCALC, LLC

Scale ... 363
Delete ... 364
Extrude ... 364
Extrude Nodes to Members ... 365
Extrude Members to Shells ... 366
Extrude Shells to Bricks ... 367
Revolve ... 368
Revolve Members to Shells ... 369
Revolve Shells to Bricks ... 371
Split ... 372
Split Members ... 373
Insert Nodes at Intersections of Selected Members ... 374
Split Selected Members at Nodes ... 374
Member Properties ... 375
Member Properties ... 375
Materials ... 375
Sections ... 375
Element Local Angles ... 375
3-Point Member Orientation ... 375
Moment Releases ... 375
Tension/Compression Only ... 375
Convert Selected Members to Rigid Links ... 375
Cracking Factors ... 375
Rigid Offsets ... 375
Shell Properties ... 376
Shell Properties ... 376
Materials ... 376
Thicknesses ... 376
Element Local Angles ... 376
Match Local x-Axes with Source ... 376
Match Local z-Axes with Source ... 376
Align Local z-Axes with Reference Point ... 376
Align Local y-Axes with Reference Point ... 376
Sub-Mesh Shells ... 377
Renumber ... 378
Auto Renumber All Nodes ... 378
Renumber Selected Nodes ... 379
Renumber Selected Members/Shells/Bricks ... 379
Switch Coordinates ... 380
Reverse Node Order for Selected Elements ... 380

© 2026 ENERCALC, LLC

Merge All Nodes & Elements ... 380
Remove All Orphaned Nodes ... 380
Element Activation ... 380
View ... 381
Drawing Grids ... 381
Drawing Grid Setup ... 381
Toggle Grid Display ... 381
Redraw ... 381
Restore Model ... 381
Preset Views ... 381
Named Views ... 381
Zoom Extent ... 381
Zoom Window ... 382
Zoom Object ... 382
Zoom Previous ... 382
Zoom In ... 382
Zoom Out ... 382
Pan ... 382
Pan Left ... 382
Pan Right ... 382
Pan Up ... 382
Pan Down ... 382
Pan Screen ... 382
Rotate ... 382
Rotate +X ... 382
Rotate -X ... 382
Rotate +Y ... 383
Rotate -Y ... 383
Rotate +Z ... 383
Rotate -Z ... 383
Pan ... 383
Zoom ... 383
Rotate ... 383
Load Diagram ... 384
Query ... 385
Measure ... 386
Render Options ... 386
Quick Render ... 386
Diaphragm Render ... 387
Global Axes ... 387
Contour Legend ... 387

© 2026 ENERCALC, LLC

Comment ... 387
Hide Selected ... 387
Hide All Except Level ... 388
Hide All Except Plane ... 388
Show All ... 388
Display Options ... 388
Tables ... 389
The "Fill Down" Command ... 389
Show Data for Selected Entities Only ... 389
Materials ... 389
Sections ... 389
Thicknesses ... 390
Nodes ... 391
Members ... 392
Shells ... 393
Bricks ... 394
Supports ... 395
Springs ... 395
Nodal Springs ... 396
Line Springs ... 396
Surface Springs ... 396
Member Releases ... 397
Diaphragms ... 398
Multi-DOF Constraints ... 399
Load Cases ... 399
Load Combinations ... 399
Nodal Loads ... 400
Point Loads ... 401
Line Loads ... 402
Area Loads ... 403
Surface Loads ... 404
Self Weights ... 405
Thermal Loads ... 405
Member Thermal Loads ... 405
Shell Thermal Loads ... 406
Brick Thermal Loads ... 407
Masses ... 408
Calculated Masses ... 408
Additional Masses ... 409
Response Spectra Library ... 410
Story Drift Nodes ... 412

© 2026 ENERCALC, LLC

Comments ... 413
Analysis ... 413
Analysis Options ... 414
Static Analysis ... 417
Frequency Analysis ... 418
Response Spectrum Analysis ... 419
View Log File ... 420
Analysis Results ... 420
Show Results for Selected Entities Only ... 420
Results Load Combination ... 420
Nodal Displacements ... 421
Story Drifts ... 421
Support Reactions ... 422
Nodal Spring Reactions ... 422
Line Spring Reactions ... 423
Surface Spring Reactions ... 423
Multi-DOF Constraint Forces & Moments ... 424
Member End Forces & Moments ... 424
Member Segmental Results ... 425
Shell Forces, Moments & Stresses ... 426
Shell Forces & Moments ... 426
Shell Principal Forces & Moments ... 427
Shell Stresses [Top & Bottom] ... 428
Shell Principal Stresses ... 429
Shell Nodal Resultants ... 430
Bricks ... 431
Brick Stresses ... 431
Brick Principal Stresses ... 431
Dynamics ... 431
Eigenvalues ... 432
Eigenvectors ... 432
Mode Participation Factors ... 433
Modal Displacements SX, SY and SZ ... 433
Inertial Forces SX, SY and SZ ... 434
Modal Nodal Displacements ... 434
Modal Support Reactions ... 435
Modal Nodal, Line and Surface Spring Reactions ... 435
Modal Multi-DOF Constraint Forces & Moments ... 435
Modal Member End Forces & Moments ... 436

© 2026 ENERCALC, LLC

Modal Member Segmental Results ... 436
Modal Shell Forces & Moments ... 436
Modal Brick Stresses ... 437
Modal Base Shears ... 437
Shear & Moment Diagram ... 438
Deflection Diagram ... 439
Contour Diagram ... 440
Mode Shape ... 443
Response Animation ... 443
Concrete Design ... 444
RC Materials ... 445
RC Model Design Criteria ... 446
RC Design Criteria ... 448
RC Beam Design Criteria ... 448
RC Column Design Criteria ... 449
RC Plate Design Criteria ... 451
Exclude Concrete Elements ... 452
Cracking Factors ... 453
RC Design Properties ... 454
RC Beam Design Properties ... 454
RC Column Design Properties ... 455
RC Plate Design Properties ... 456
RC Member Input ... 457
RC Plate Input ... 458
Perform Concrete Design ... 458
Concrete Design Output ... 459
RC Analysis Envelope ... 459
RC Beam Results ... 459
RC Column Results ... 460
Member Shear Results ... 460
Wood-Armer Moments ... 461
RC Plate Results ... 461
Flexural/Axial Interaction ... 461
Sections ... 462
P-Mx (+) ... 463
P-Mx (-) ... 463
P-My (+) ... 463
P-My (-) ... 463
P-Mx-My ... 463
Print Diagrams ... 464
Concrete Design Diagrams ... 465

© 2026 ENERCALC, LLC

RC Member Envelope Diagram ... 465
RC Plate Envelope Contour ... 466
Concrete Design Tools ... 467
Rebar Database ... 467
K Calculator ... 468
Quick R-Beam Flexural Design ... 469
Quick T-Beam Flexural Design ... 470
Unity Check ... 470
Steel Design ... 470
Steel Materials ... 471
Steel Design Criteria ... 472
Steel Design Criteria ... 473
Steel Member Design Criteria ... 474
Steel Section Pool ... 476
Exclude Steel Elements ... 477
Steel Member Design Properties ... 477
Steel Member Input ... 478
Perform Steel Design ... 479
Steel Design Results ... 480
Steel Design Tools ... 481
K Calculator ... 481
Steel Section Check ... 481
Steel Section Design ... 490
Unity Check ... 490
Reporting ... 490
Print Text Report ... 491
Print Envelope Report ... 491
Print Current View ... 492
Print Options ... 492
Print RC Report ... 493
Print Steel Design Report ... 495
Settings & Tools ... 495
New Window ... 495
Tile Horizontal ... 495
Tile Vertical ... 495
Cascade ... 495
Clear Undo & Redo ... 495
Clear Results ... 495
Clear Everything ... 495
Unit Conversion ... 496
Calculator ... 496

© 2026 ENERCALC, LLC

Text Editor ... 496
Copy Command History ... 496
Clear Command History ... 496
Toolbar ... 497
Units & Precisions ... 498
Data Options ... 499
Graphic Scales ... 501
Colors ... 502
Preferences ... 503
Help ... 504
Online User's Manual ... 504
User's Manual PDF ... 504
Technical Support ... 504
Upload Model to Support ... 504
Online Training Manual ... 504
Training Manual PDF ... 504
Training Videos ... 504
Verification Manual ... 504
Release Notes ... 504
About ENERCALC 3D ... 504
Sign Conventions ... 505
Beam Forces & Moments ... 505
Shell Forces & Moments ... 506
Toolbars ... 507
Input/Output Toolbar ... 507
Status Bar ... 508
Coordinate Systems ... 508
Global Coordinate System ... 508
Local Coordinate Systems - General ... 508
Member Local Coordinate System ... 509
Four-Node Shell Local Coordinate System ... 511
Eight-Node Brick Local Coordinate System ... 512
Nodes ... 512
Nodal Coordinates ... 512
Degrees of Freedom (DOFs) ... 512
Node Numbers ... 513
Loads ... 514
Supports ... 514
Springs ... 515
Members ... 515
Member Sections ... 515

© 2026 ENERCALC, LLC

Local Coordinate System ... 516
Member Numbers ... 516
Beams vs. Trusses ... 516
Elastic Stiffness Matrix ... 516
Geometric Stiffness Matrix ... 517
Moment Releases ... 517
Tension/Compression-Only ... 518
Rigid Links ... 518
Rigid Diaphragms ... 518
Loads ... 518
Line Springs ... 523
Internal Forces and Moments ... 523
Shells ... 524
Shell Thicknesses ... 525
Local Coordinate System ... 526
Shell Numbers ... 526
Element In-Plane Stiffness Matrix ... 526
Element Out-of-Plane Stiffness Matrix ... 526
Combining Element In-Plane and Out-of-Plane Stiffness Matrices ... 526
Loads ... 527
Surface Springs ... 527
Internal Forces or Moments ... 527
Membrane Nodal Resultants ... 530
Bricks ... 531
Local Coordinate System ... 531
Brick Numbers ... 531
Element Stiffness Matrix ... 531
Loads ... 532
Internal Stresses ... 532
Static Analysis ... 532
Load Cases and Load Combinations ... 532
Linear, Non-linear Static Analyses ... 533
P-Delta (P-Δ) vs. P-delta (P-δ) ... 534
Solution Algorithm ... 536
Solution Accuracy and Stability ... 536
Frequency Analysis ... 538
Solution Algorithm ... 539
Mass and Stiffness ... 540
Solution Convergence ... 541
Response Spectrum Analysis ... 542

© 2026 ENERCALC, LLC

Concrete Design – ACI 318-02/05/08/11/14 ... 543
Concrete Column Design ... 545
Concrete Beam Design ... 555
Concrete Slab/Wall Design ... 560
Steel Design ... 561
Section Orientation ... 563
Member Internal Forces and Moments ... 564
Solution Algorithms ... 564
References ... 564
Appendix ... 566
Unit Conversions ... 566
Designations, diameters and areas of standard bars ... 566
2D Frame ... 567
General ... 569
Frame Wizard ... 570
Joints & Joint Loads ... 575
Joint Data ... 580
Joint Loads ... 584
Joint Results ... 586
Members & Member Loads ... 586
Member Data ... 589
Member Loads ... 603
Member Forces ... 609
Sections & Materials ... 610
Section Data ... 610
Material Data ... 612
Load Combinations ... 614
Wood Design ... 617
Results ... 618
Extreme Values ... 619
Joint Displacements & Reactions ... 622
Member End Forces ... 623
Member Details ... 624
Member Check Results ... 625
Sign Convention ... 627
Frame Sketch ... 629
Diagrams ... 629
Reports ... 631
Torsional Analysis of Rigid Diphragm ... 631
Torsional Analysis of Rigid Diaphragm Theory ... 656

© 2026 ENERCALC, LLC

General Section Property Calculator 668
General Beam Analysis 678
4 Beams 684
Multiple Simple Beam 701
Steel Beam 718
Composite Steel Beam 727
Steel Beam with Torsional Loads 742
Concrete Beam 752
Masonry Beam 764
Wood Beam 780
Wood Ledger 794
Flitch Plated Wood Beam 805
5 Columns 810
All Columns 821
Column Slenderness 821
Beam Stability (in Steel Column and Wood Column modules) 825
Steel Column 827
Wood Column 841
Concrete Column 857
Sign Convention for Concrete Column 892
Masonry Column 895
6 Foundations 906
General Footing 907
General Footing by FEM 925
Wall Footing 937
Combined Footing 953
Beam on Elastic Foundation 970
Pole Footing Embedded in Soil 971
Pile Group Analysis 979
Point Load on Slab 986
7 Walls 990
Slender Walls 990
Concrete Slender Wall 990
Masonry Slender Wall 1011
Shear Walls 1031
Concrete Shear Wall 1032
Masonry Shear Wall 1057
Wood Shear Wall 1070
Aspect Ratio Adjustment Options 1087
8 Earth Retention 1087

© 2026 ENERCALC, LLC

All Retaining Walls 1088
General Tab 1089
General Data 1089
Soil Values 1091
Use of Vertical Component (Not in Restrained Retaining Wall module) 1095
Loads Tab 1096
Loads 1096
Seismic Loads 1101
Stem Tab 1105
Stem Tab for Cantilevered Retaining Wall 1106
Summary Section of Stem Tab 1111
Stem Tab for Tapered Stem Retaining Wall 1118
Stem Tab for Gravity Retaining Wall 1121
Stem Tab for Restrained Retaining Wall 1123
Footing Tab 1131
Footing Design Sub-tab 1131
Key Design & Sliding Options 1135
Pier Design 1139
Load Factors 1144
Results Tabs 1145
Summary 1146
Sliding > Resisting Forces 1149
Sliding > Sliding Forces 1150
Overturning > Resisting Moments 1151
Overturning > Overturning Moments 1152
Global Stability 1153
Wall Tilt 1153
Stem Design Values 1154
Stability Tab (Restrained Walls only) 1155
Construction Tab 1156
Wall Loading Tab 1157
Methodology / Analysis & Design Assumptions 1158
Gabion Retaining Wall 1160
Methodology / Analysis & Design Assumptions 1164
Soldier Pile Retaining Wall 1164
General Tab 1165
Lateral Earth Pressure Tab 1167
Lagging Tab 1168
Results Tabs 1169
Summary Tab 1170

© 2026 ENERCALC, LLC

Construction Tab ... 1173
Wall Loading Tab ... 1174
Soil Pressure Reference ... 1175
Solver Results ... 1176
Methodology / Analysis & Design Assumptions ... 1176
Segmental Retaining Walls ... 1179
Segmental Wall Overview ... 1179
Design Assumptions for Geogrid Reinforced Segmental Walls ... 1179
Design Assumptions for Gravity Segmental Retaining Walls ... 1180
Design Parameters Tab ... 1181
Loads Tab ... 1183
Geogrid Reinforced Segmental Retaining Walls ... 1184
Block & Geogrid Data Tab (for Geogrid Reinforced Walls) ... 1184
Results Tab (for Geogrid Reinforced Walls) ... 1187
External Stability Tab (for Geogrid Reinforced Walls) ... 1188
Internal Stability Tab (for Geogrid Reinforced Walls) ... 1189
Construction Tab (for Geogrid Reinforced Walls) ... 1191
Gravity Segmental Retaining Walls ... 1191
Block & Geogrid Data Tab for Gravity Walls ... 1192
Results Tab (for Gravity Segmental Retaining Walls) ... 1194
External Stability Tab (for Gravity Segmental Retaining Walls) ... 1195
Internal Stability Tab (for Gravity Segmental Retaining Walls) ... 1196
Construction Tab (for Gravity Segmental Retaining Walls) ... 1197
Methodology / Analysis & Design Assumptions ... 1198
9 Miscellaneous ... 1198
Steel Base Plate FEM ... 1198
Steel Base Plate ... 1206
Steel Bolt Group Analysis ... 1220
Rebar Development Table ... 1227
10 External Items ... 1228
Relating an External Item to a Project File ... 1230
Limitations of Working with External Items ... 1234

© 2026 ENERCALC, LLC

Scanned Document 1235

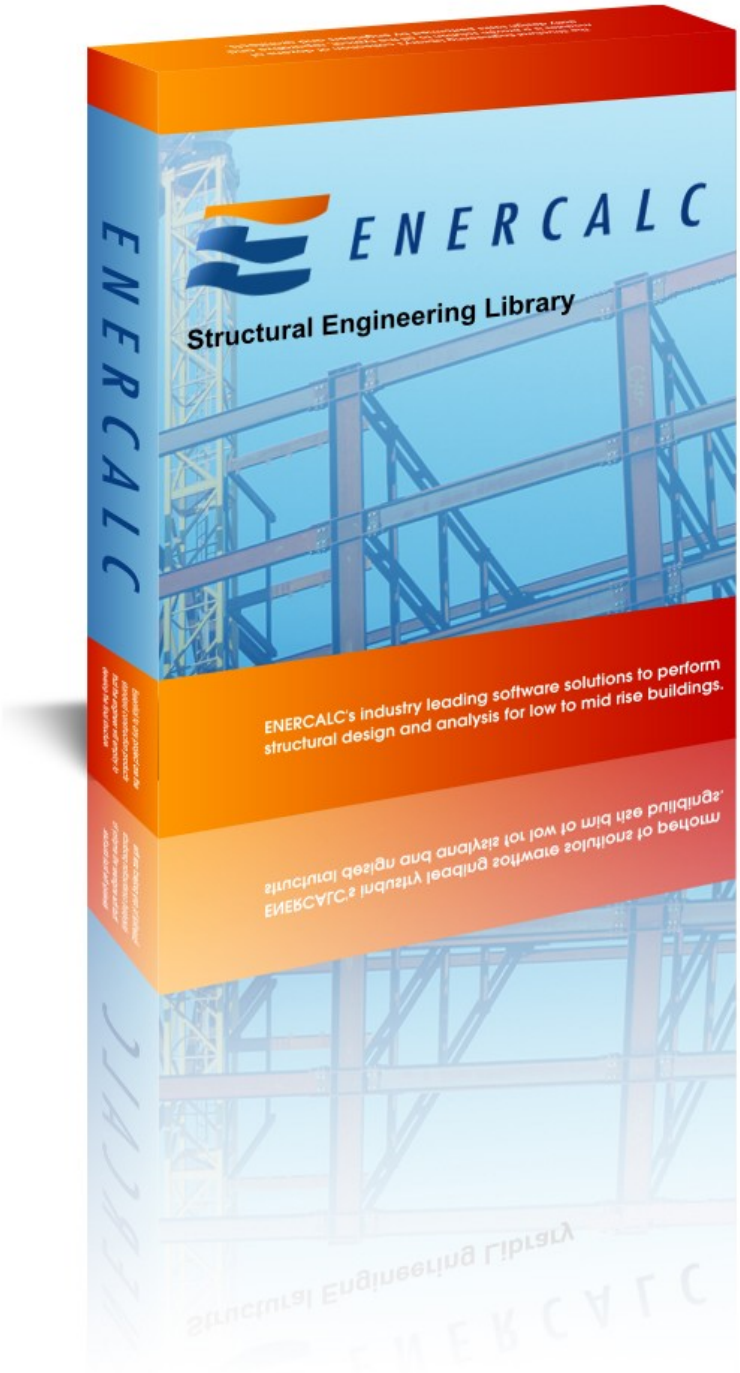
Part XIV Appendices 1239

1 Appendix A - Table of Horizontal Temperature and Shrinkage Reinforcing 1240
2 Appendix B - Development and Lap Lengths 1241
3 Appendix C - Masonry Wall Weights & Section Properties 1242
4 Appendix D - Summary of Concrete & Masonry Design Formulas 1247
5 Appendix E - References Used For The Development Of This Program 1248
6 Appendix F - Rankine and Coulomb Formulas 1248

© 2026 ENERCALC, LLC

Part I

1 ENERCALC Structural Engineering Library



© 2026 ENERCALC, LLC

Part II

2 Introduction

Last Revised: Thursday, March 12, 2026 at 10:04 AM

2.1 Welcome

Welcome to ENERCALC® SEL

You've chosen one of the most respected Structural Engineering software packages available today. In continuous development since 1983, ENERCALC SEL is the culmination of years of development and refinement from suggestions of engineers worldwide.

ENERCALC SEL is developed with the practicing engineer in mind. Although large complex frames are fun projects, structural engineers spend most of their time designing and analyzing the components of structures. Because most of the buildings worldwide rely on simple beams, columns, foundations, walls, and other small items, this software system will quickly become your best friend. **ENERCALC SEL** remembers the mathematics, building code provisions, and standard materials you need to perform a detailed and economical design.

Because we feel that simple, repetitive engineering problems are far more common than extensive 3-D frame analysis, this software package is designed specifically for fast, interactive engineering design of building components. We've combined the typical working methods of engineers, national building code provisions, and construction material databases with the principles of structural mechanics into each "calcsheet" module. You will find that these modules operate very much like an electronic calculation pad.....simply fill in the data entries and the entire calculation will be instantly updated for your review.

To add even more power and utility to the system, we've added detailed design sketches and stress diagrams, automatic design and sizing, an online help system, material databases, and elegant calculation printing to **ENERCALC SEL**.

ENERCALC SEL is designed around a file of calculations called a "**Project**". This single file with the extension "**EC6**" can hold one or thousands of individual calculations. You add, edit, and delete the calculations in your Project File during the in-office design stage. Then, when it's time for submittal to a governing agency, you can print a complete set of calculations.

Because of the ever-expanding number of modules, we invite you to stay in close contact with our website at enercalc.com. Maintenance releases, up-to-the-minute technical advice, revised electronic documentation, and new product information will all be provided there FIRST.

ENERCALC has put years of work into this package in support of the highly technical and dedicated service Structural Engineers provide to the people of the world. We continue to enhance this product weekly and are committed to developing this product well into the future. We extend our thanks for choosing ENERCALC, and look forward to using your suggestions to provide you with ever improving tools for your daily work.

© 2026 ENERCALC, LLC

2.2 Warning & Disclaimer

Although it is our intent that the information contained in this manual and associated software program is accurate and reliable, it is possible that there may be errors, both of omission and commission, that we are not aware of at any time. ENERCALC, Inc. can make no warranties, either express or implied, as to the accuracy of the material in this manual and software nor its suitability for a specific purpose or application for which it is advertised.

ENERCALC, Inc., its owners, directors, and employees, can offer no guarantee and will accept no liability for damages of any kind resulting from the use of the information contained or generated by this document and the accompanying computer software.

If you do not agree to be bound by these conditions and the conditions contained in the License Agreement 5 contained herein, then you may Internet deactivate the program and uninstall the software within the trial period after the date of your order and request a full refund of the License Fee.

2.3 License Agreement & Copyright

The complete License Agreement can be viewed by using the following link: License Agreement

2.4 End of Service Policy

At some point in the future the current version software will undergo a major overhaul of capability increase. At this time the prior version reaches its "**End of Service**" time.

When a version reaches its "End of Service", access to support and maintenance will cease in about 6 months (the time frame will be set by ENERCALC and is at our discretion). The software will continue to operate but technical support, updates, and other support related items will cease to be available.

All software products, ENERCALC SEL included, have a practical commercial lifetime. In order to provide the highest quality products and support to our customers, each product is developed utilizing a product life cycle methodology, which includes an End-of-Service (EOS) phase.

The ENERCALC product EOS policy is to support the current release plus the previous (one back) release for up to six (6) months by default. After this time, ENERCALC's product development ceases active development and support of that software release within the Maintenance and Support Plan. ENERCALC does not create or make available maintenance releases or patches for software that has reached the EOS milestone.

During the EOS phase, ENERCALC will continue to investigate, troubleshoot, and characterize issues in an attempt to provide solutions and workarounds using the production releases. If a solution cannot be found using software that has reached the EOS milestone, ENERCALC will suggest that the system be upgraded to a more recent software release.

© 2026 ENERCALC, LLC

Once the EOS process starts on a product release, a notice will be posted on all relevant pages stating that the product release has entered the EOS process.

Note: This policy is subject to revision.

2.5 RetainPro Mode Limitations

The latest available build of ENERCALC SEL is available as a subscription only. However, it is capable of operating in "RetainPro Mode" to allow usage by RetainPro users with active Maintenance & Support Plans or current subscriptions.

RetainPro mode does have at its core:

- The extensive rewrite of most internal program code
- Newly designed user interface
- Reporting improvements
- Speed improvements in both user interface management and calculations
- Multi-core processor usage
- Activation system improvements including extensive awareness of internet security
- Improvements to 3D image rendering
- Ability to change load combinations in an entire project file with one selection
- New 16+ million data point internal database of USGS Ss & S1 seismic values contained in their latest databases
- New RPN Calculator that "tears off" of our user interface and can remain open while working in SEL
- Extensive internal debug utilities that can be turned on and off by ENERCALC support engineers during TeamViewer & similar screen sharing sessions

The ***full version*** of the latest build has a number of extra components and capabilities not accessible in RetainPro mode:

- Local & Cloud usage with project file synchronization
- QuickCalc capability — Referenced on Projects Tab 52
- ENERCALC 3D FEM — ENERCALC 3D 259
- Steel Base Plate by FEM — Steel Base Plate by FEM 1198
- General Footing by FEM — General Footing by FEM 925
- Flitch Plated Wood Beam — Flitch Plated Wood Beam 805
- AutoDesk Revit Integration – Live 2 way analysis & design of building components
- Future enhancements and maintenance after MSP has expired

2.6 ENERCALC for Revit

ENERCALC for Revit is a separate, powerful program that enables a real-time connection between a Revit model and the member calculations in ENERCALC SEL.

© 2026 ENERCALC, LLC

For more info: enercalc.com/EFR_Help

AUTODESK
Authorized Developer

2.7 License Types

ENERCALC SEL can be licensed in a variety of formats:

Annual Subscription (Installed on your computer)

You will receive a Product Control Code that will allow you to activate the software. After you have entered your Product Control Code and performed an activation, the product will be completely operational. Your user registration number and licensee name will appear on all printouts and you will see a subscription expiration date on the licensing screen. Usage ends when the subscription expires or is canceled, but all Project Files remain intact.

Monthly Subscription (Installed on your computer)

You will receive a Product Control Code that will allow you to activate the software. After you have entered your Product Control Code and performed an activation, the product will be completely operational. Your user registration number and licensee name will appear on all printouts and you will see a subscription expiration date on the licensing screen. Usage ends when the subscription expires or is canceled, but all Project Files remain intact.

Perpetual License (Installed on your computer)

You will receive a Product Control Code that will allow you to activate the software. After you have entered your Product Control Code and performed an activation, the product will be completely operational. Your user registration number and licensee name will appear on all printouts and you will see a Maintenance & Support Plan (MSP) expiration date on the licensing screen. When MSP expires, you can choose to renew MSP to keep up to date on new versions. If you do not renew your MSP, then you can continue to use the software indefinitely at whatever version you have when MSP expired.

Academic License

This is a special version/mode for students. Each time an Academic license is granted an expiration date is set after which time the software stops operating. In addition the printouts have a watermark stating that it is an educational version and the student's name and registration number are printed. Click here for an application for an Academic License.

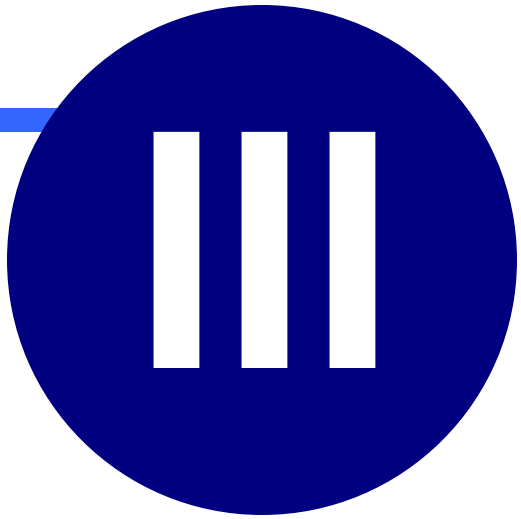
© 2026 ENERCALC, LLC

Plan Check License

This is a special version/mode for public plan review agencies. Each time a Plan Check license is granted an expiration date is set after which time the software stops operating. In addition the printouts have a watermark stating that it is a plan check version and the agency's name and registration number are printed. Click here for an application for a Plan Check License.

© 2026 ENERCALC, LLC

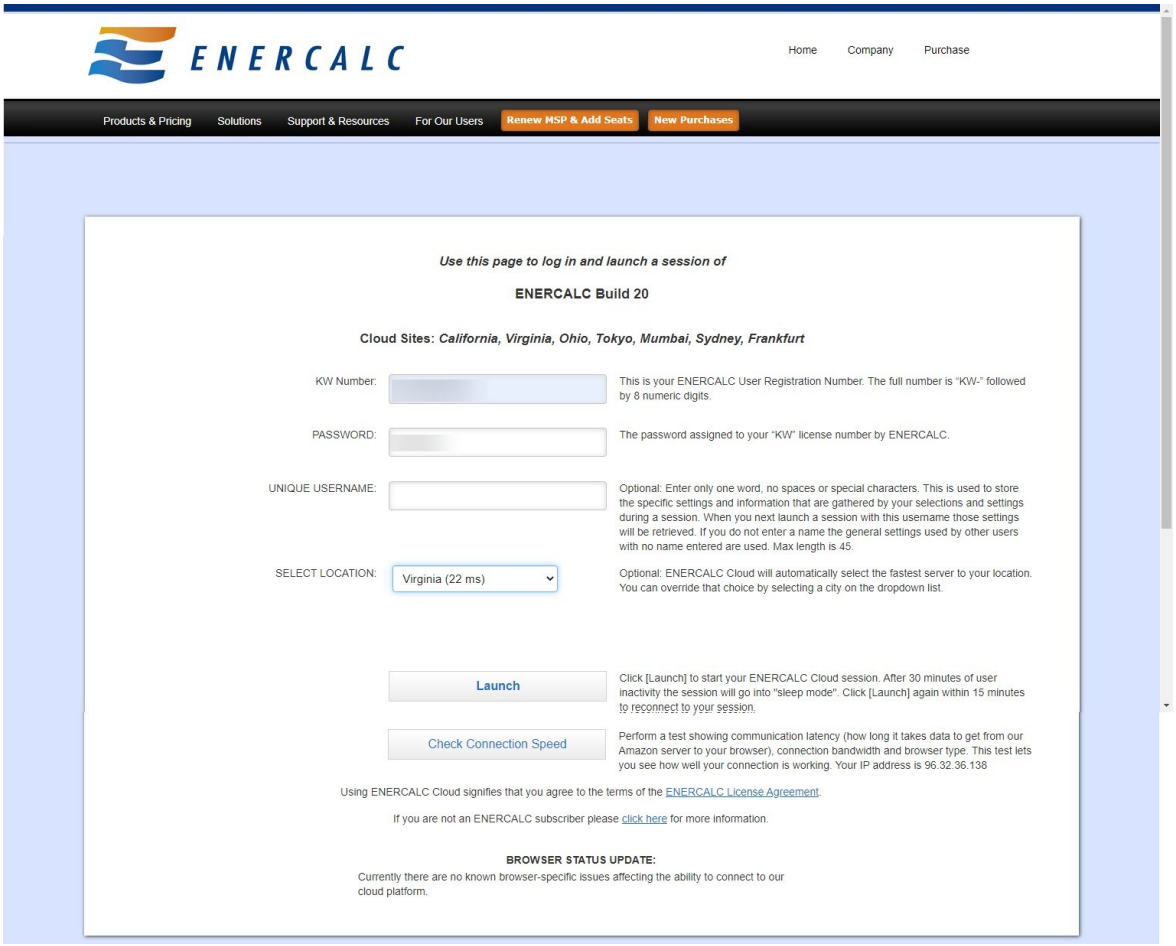
Part



3 Access to Cloud Version

ENERCALC subscribers can run the app on any HTML5 browser.

Simply point your browser to **cloud.enercalc.com**.



Enter your KW User Registration Number in the form of KW-NNNNNNNN.

Enter your password provided by ENERCALC (not your Product Control Code) in the form of three letters or numbers hyphen three letters or numbers.

Optionally, you can enter a unique username. This is used to store the specific settings and information that are gathered by your selections and settings during a session. When you next launch a session with this username those settings will be retrieved. If you do not enter a name the general settings used by other users with no name entered are used. Max length is 45.

© 2026 ENERCALC, LLC

The Select Location dropdown will populate with the available sites, and will default to the site with the lowest latency from your location. You can select any site, but the best performance will always be obtained by using the one with the lowest latency. There is no need to worry about being consistent with your selection of any particular site. Your data is replicated for redundancy, and will be available to the app, regardless of which site you select.

Click the Launch button, and the app will be loaded and displayed.

© 2026 ENERCALC, LLC

Part IV

4 Installation (Not required to access cloud version)

General

ENERCALC® SEL must be installed on each computer where it will be used. DO NOT install on one computer and use a shortcut to the software on a different computer....IT WILL NOT RUN PROPERLY.

You can install on ANY computer where you might want to use it. You will need “activate” it on the computer where it will be used so it runs as an “activated” program.

The activated seat can easily be moved between computers as you desire using our Internet Activation system. An Internet connection is needed only for the brief moment of activating or deactivating.

Download Installation File

If you don't already have the latest installation file, you can download it using this link: https://install.enercalc.com/ECSEL20_SETUP.EXE

This link will ALWAYS install the MOST RECENT build of the software. Don’t save the installation file for later use.....you will be saving the installer for what will soon be an old copy of the software, because we update the builds quite often.

This installation file will ask you to paste in the Product Control Code that ENERCALC has assigned to you. Locate the email you received from ENERCALC that contains that long scrambled looking string of characters. Always use the most recent email because it has your latest PCC.

The PCC has all the information for your license, including your Subscription (or Maintenance & Support Plan) expiration date, name that will appear as the license holder, etc.

Installation Procedure

This section describes installation of the software on any computer where you want to use it. At this time we assume you have downloaded and started the installation program file.

© 2026 ENERCALC, LLC

Immediately after starting the installation program you will view a "Welcome" screen identifying the product to be installed.

Click [Next >]

© 2026 ENERCALC, LLC

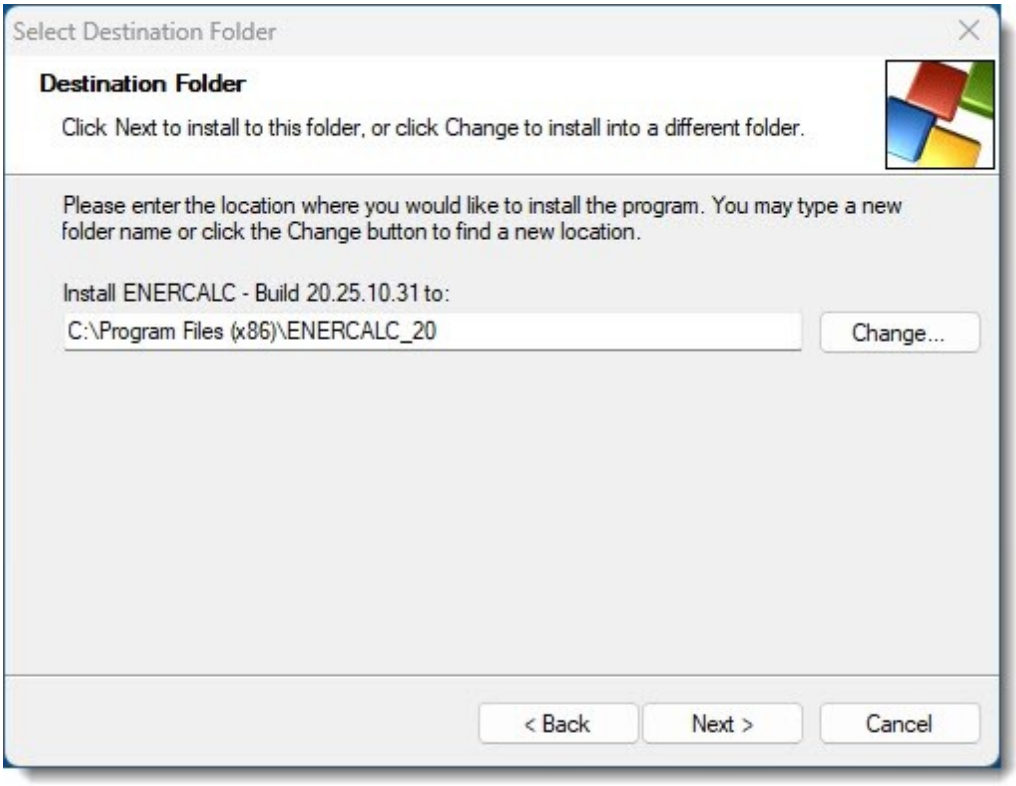
Next you will be asked to review and accept the ENERCALC License Agreement. After installation you can print and read the License Agreement using the License > License Agreement menu selection.

Choose “I Accept....” and click [Next >]

© 2026 ENERCALC, LLC

Next you will be asked to select the drive and folder location where the program files are to be placed. A standard location for Microsoft Windows installations is given and our best advice is to accept it.

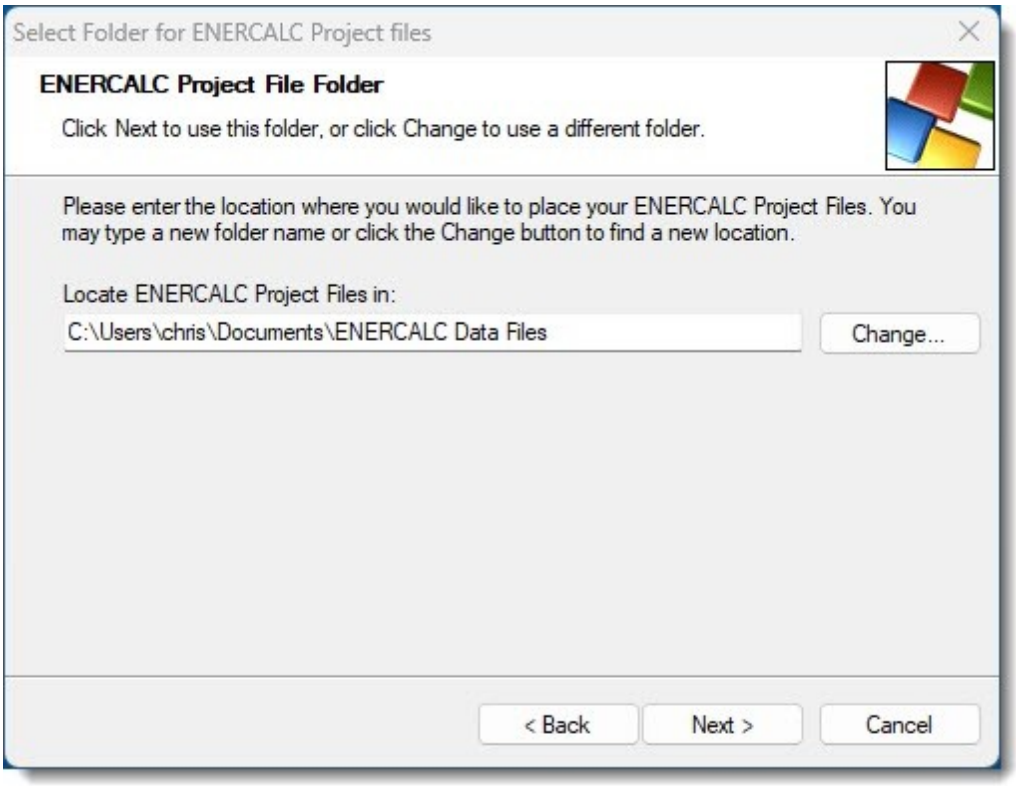
ALWAYS INSTALL THE SOFTWARE ON THE COMPUTER WHERE IT WILL BE USED....NOT A REMOTE DRIVE.



Click [Next >]

© 2026 ENERCALC, LLC

Next you will be asked to select the default drive and folder location where your Project Files will be saved. Feel free to customize this one, but our best recommendation is to avoid Dropbox, OneDrive, and other mirroring systems.

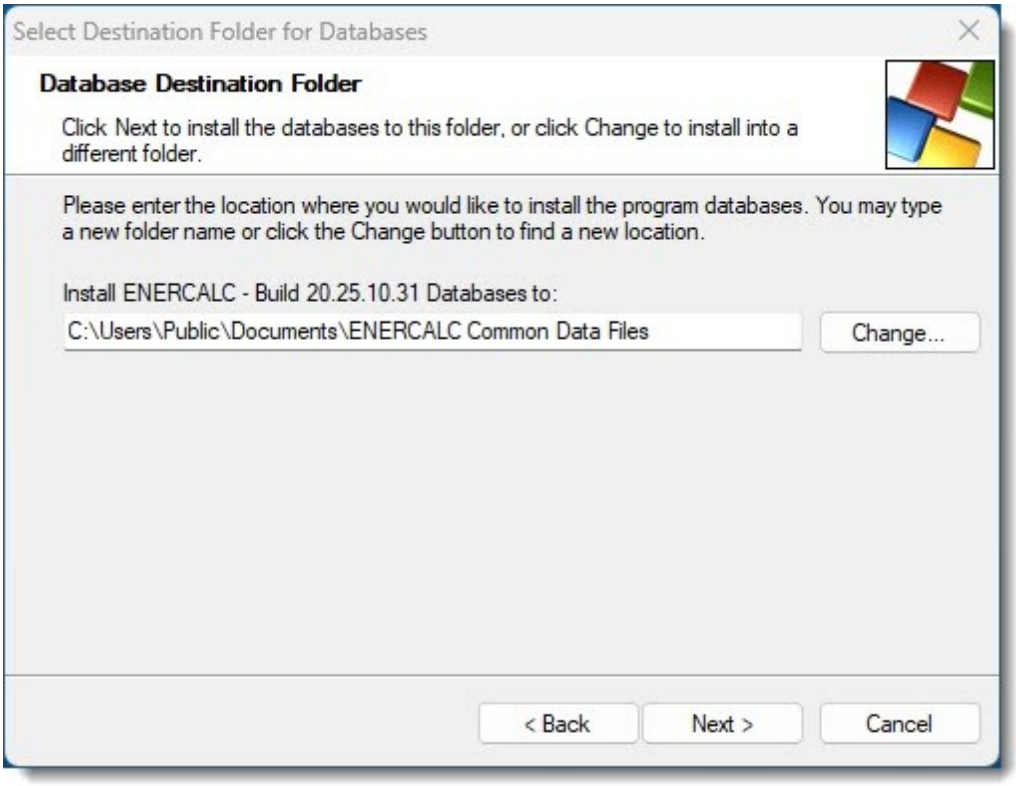


Click [Next >]

© 2026 ENERCALC, LLC

Next you will be asked to select the drive and folder location where shared databases will be saved. These include material and section databases. A standard location for Microsoft Windows installations is given and again, our best advice is to accept it.

ALWAYS INSTALL THE DATABASES ON THE COMPUTER WHERE IT WILL BE USED....NOT A REMOTE DRIVE.

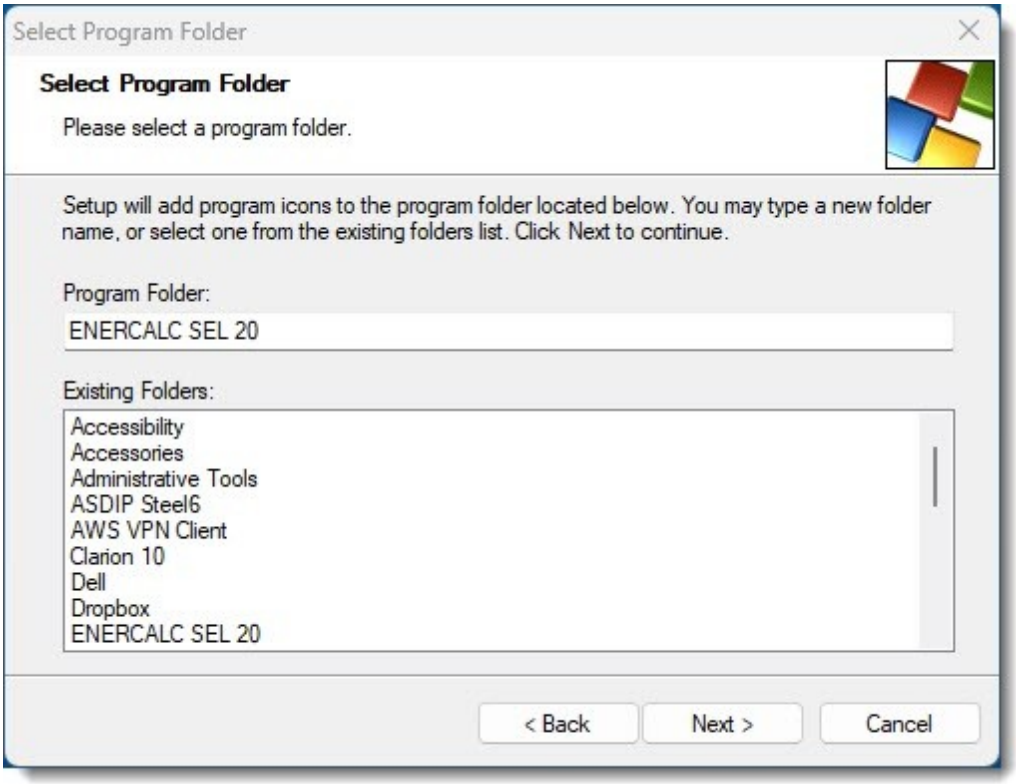


Click [Next >]

© 2026 ENERCALC, LLC

Next you will be asked to name the Start menu program group that will contain the links for starting up various parts of the software. (We recommend using what our installation program suggests.)

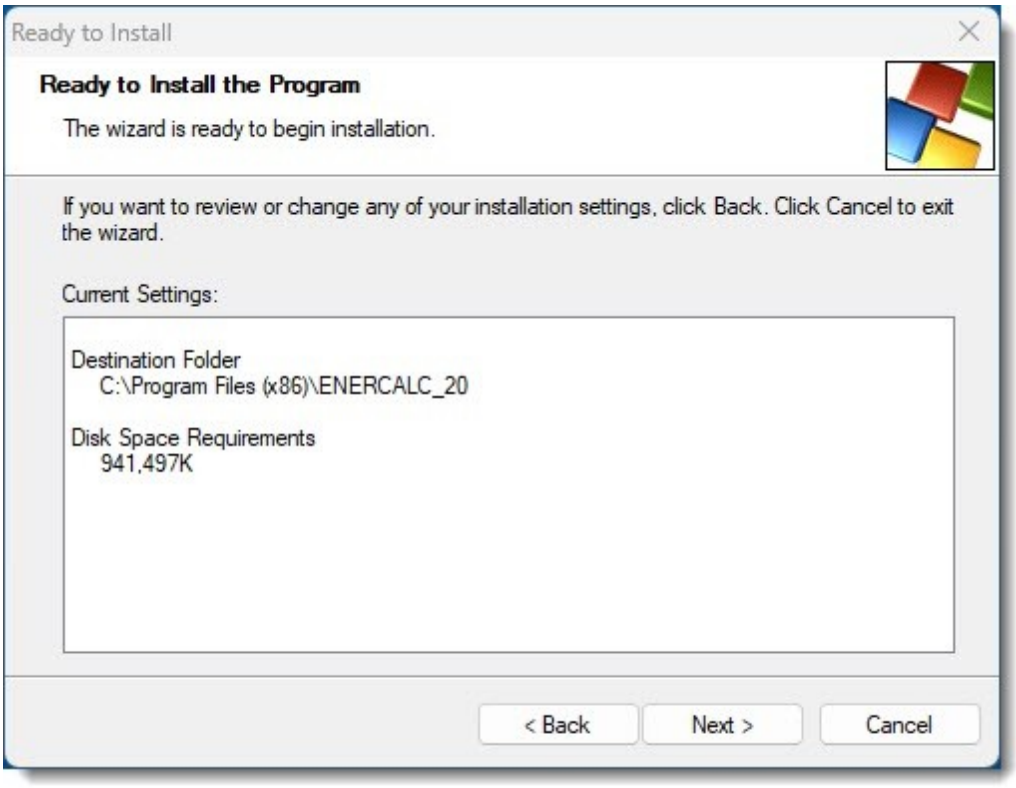
This program group will be placed within the Start > Programs selection within Windows.



Click [Next >]

© 2026 ENERCALC, LLC

Next and immediately prior to the file placement process you will be given a summary of your selected items.

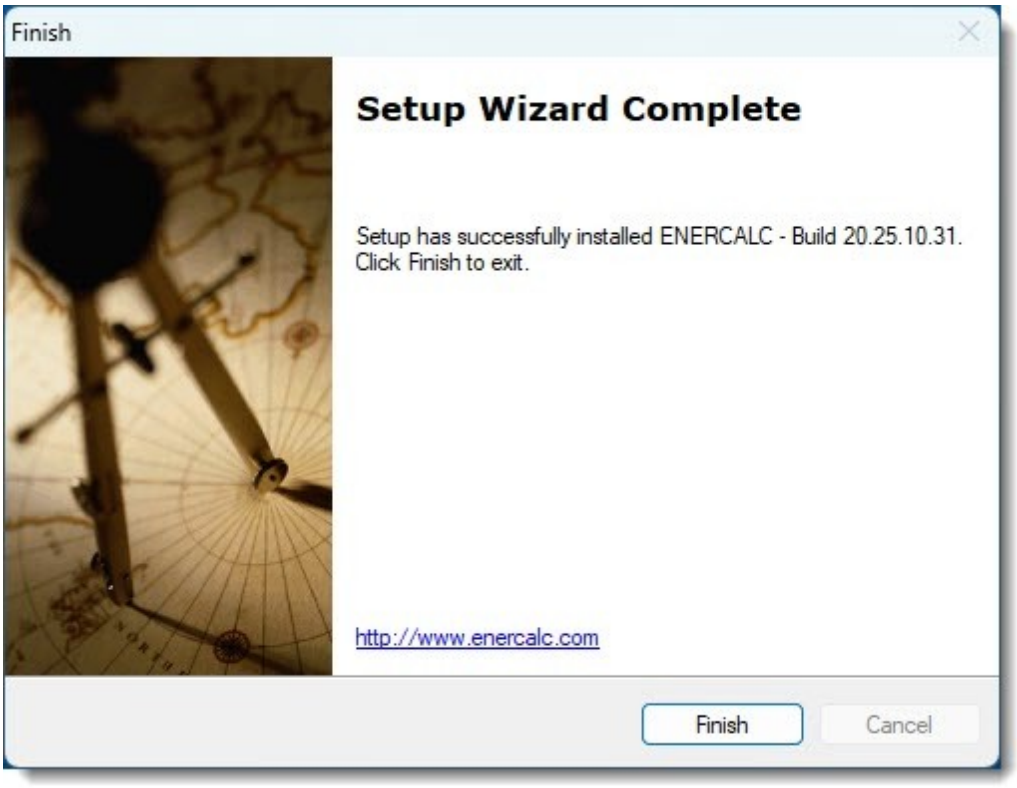


Click [Next >]

The program will then be installed on your computer.

© 2026 ENERCALC, LLC

The final screen announces that the installation process is complete.



Click [Finish >]

SOFTWARE INSTALLATION IS NOW COMPLETE! The next step will be to activate your software.

© 2026 ENERCALC, LLC

Part V

5 Activation (Not required to access cloud version)

Activation

ENERCALC® SEL has a security system that requires you to "Activate" the software. You will copy & paste a **Product Control Code (PCC)** into the software and then use an **[Internet Activate]** button to get permission from our Internet Activation server to run as a registered program.

You can install the software on as many computers you want, and you will use the same Product Control Code in each installation.

You will use “Internet Activation” and “Internet Deactivation” to activate the software on each computer, up to the number of simultaneous seats you have purchased for your “KW” user registration number. Deactivate one or more seats to free them up for use on another computer, or use “Automatic Activation”, so the seats are automatically made available when the application is closed.

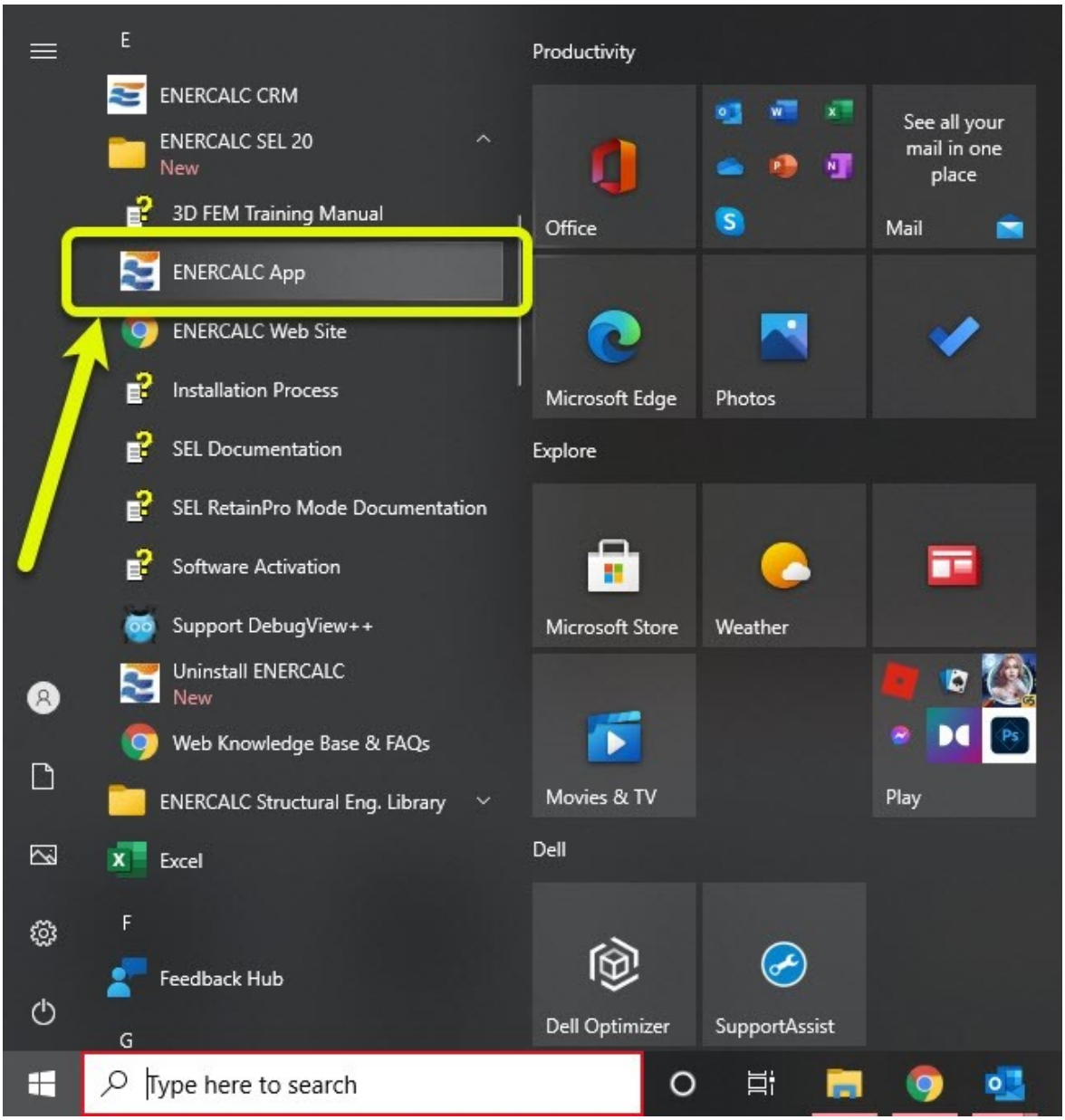
You only need to be Internet connected for the brief moment you activate or deactivate. Otherwise the computer will remember it’s activated (when using Indefinite Activation), or it will reactivate automatically next time you start the program (when using Automatic Activation).

Our Internet Activation System uses HTTPS REST API calls to connect to Enercalc.net for communications. Most of the time it works great without any changes to firewalls or anti-virus software. Sometimes you must pause your anti-virus software briefly during activation or deactivation, or add a firewall rule to allow it to connect. In larger companies, your IT staff will usually manage this for you.

Launch ENERCALC SEL, either by double-clicking the desktop shortcut named "ENERCALC App".

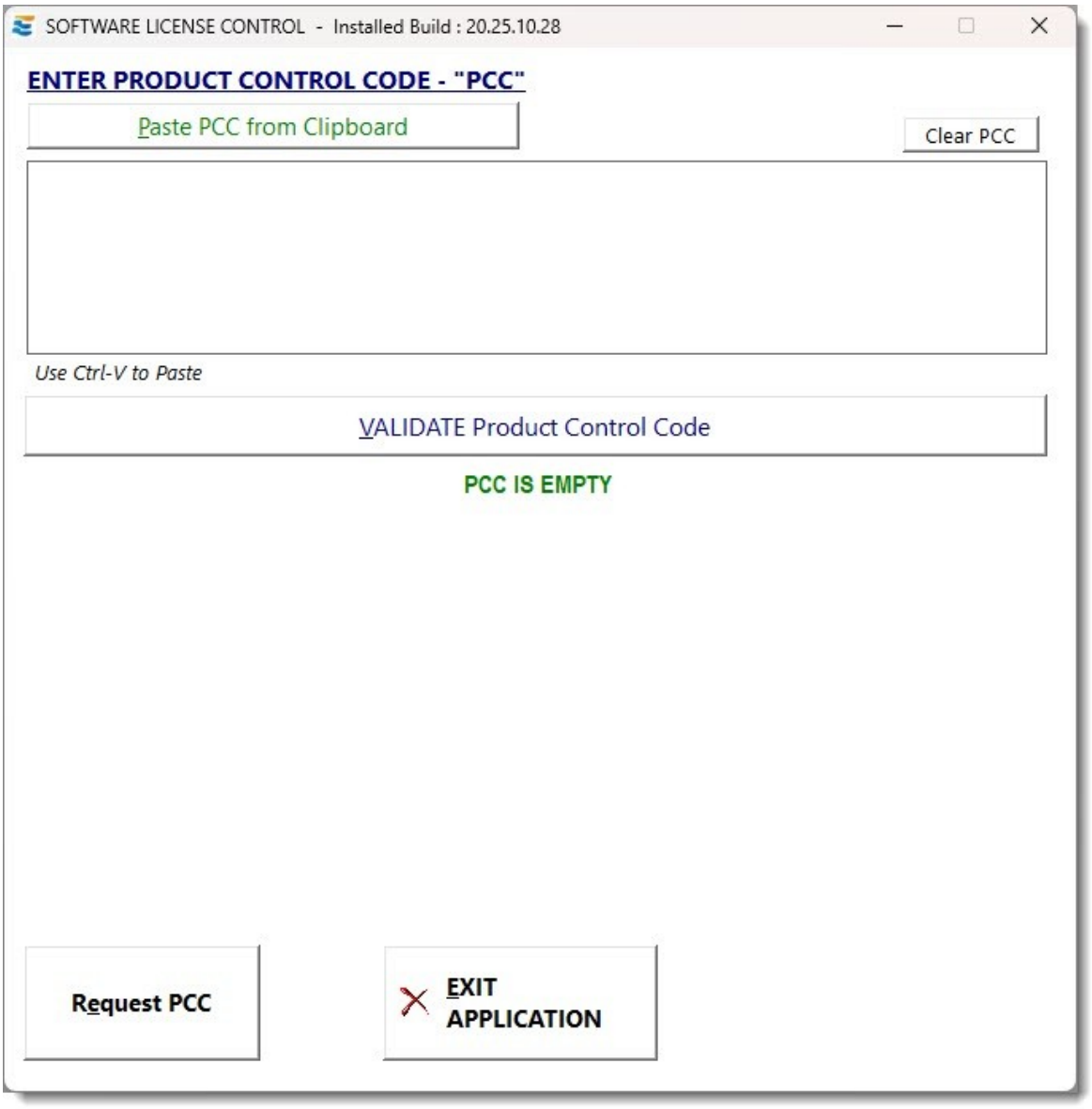
Or by clicking the Windows Start button and navigating to the "ENERCALC SEL 20" program group and clicking "ENERCALC App"...

© 2026 ENERCALC, LLC



© 2026 ENERCALC, LLC

If the software has just been installed for the first time the window below will be displayed.



This screen requests that you enter your Product Control Code (PCC). The PCC is a scrambled looking string about 150 characters long that was supplied to you in one of your product delivery emails.

Note! The same PCC is used in ALL your installations. You only need a new PCC when you renew your subscription.

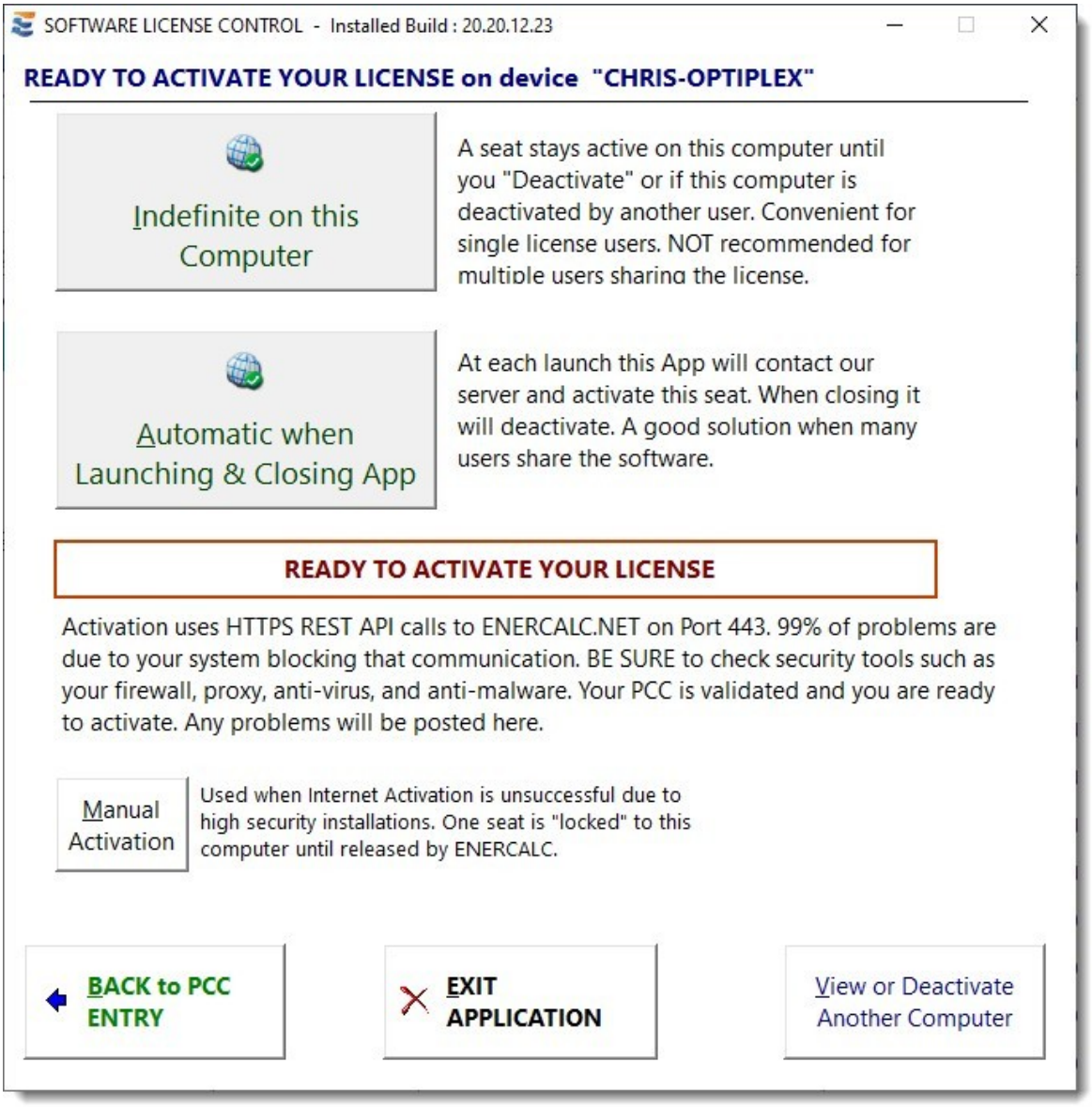
Locate your most recently received PCC, highlight it press Ctrl-C to copy it to the Windows Clipboard. Then click the Paste PCC from Clipboard button.

Now click the Validate Product Control Code button, followed by the Activate button.

© 2026 ENERCALC, LLC

At this point the steps will be the same regardless of whether you are activating a new installation for the first time, or if you are reactivating an existing installation.

If the software is not currently activated the window below will be displayed.



This is the typical license activation window. It presents the two primary activation options:

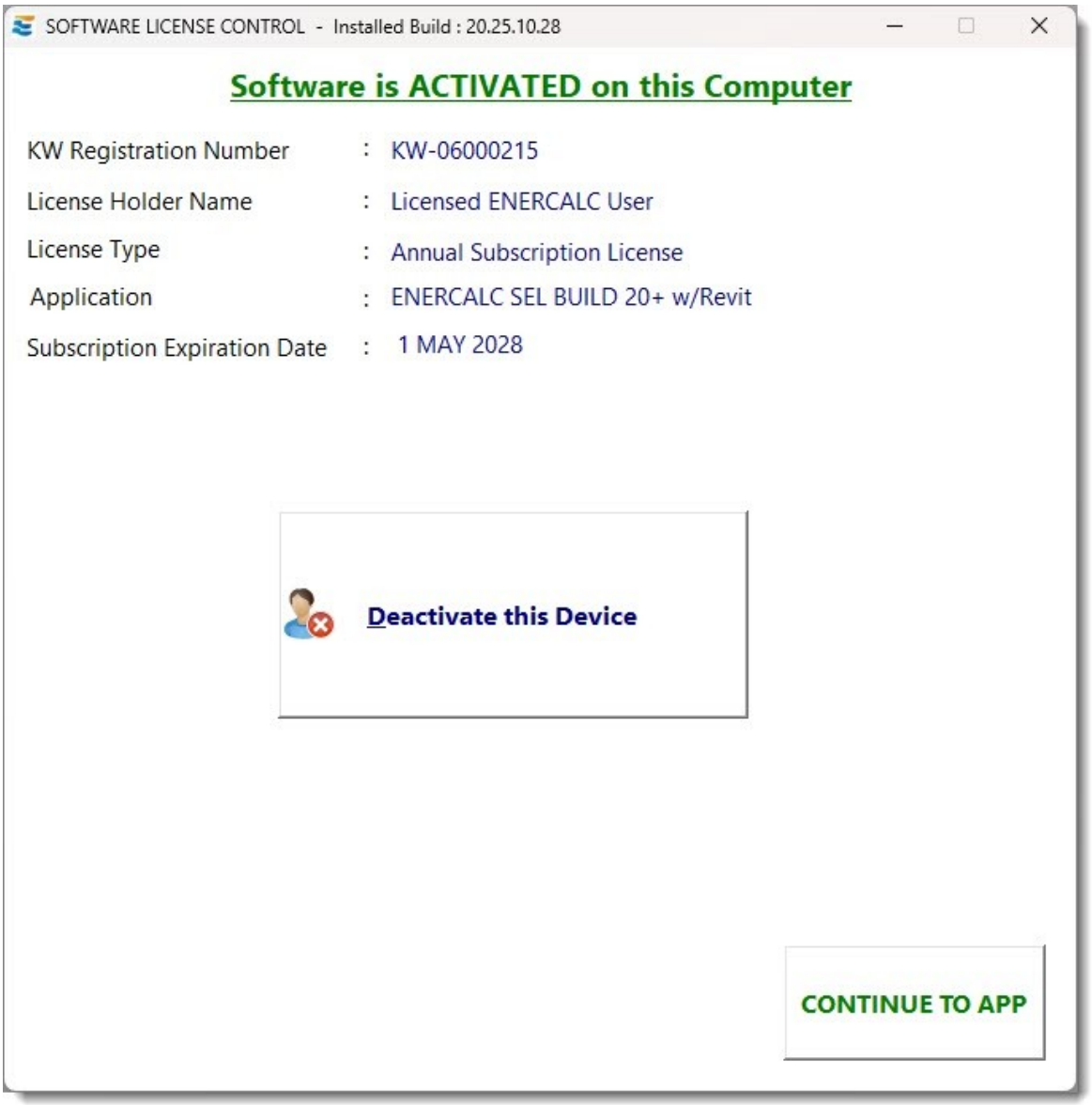
© 2026 ENERCALC, LLC

Indefinite: This selection will activate the program and leave it activated. Each time you start the software thereafter it will launch immediately as “activated”.

Automatic: Deactivates on exit and then automatically activates on launch. This option may be ideal for larger firms with numerous users, sometimes in different locations.

(The [Manual Activation] button is only used in special cases where Internet Activation does not work due to the lack of Internet connectivity or security issues. You will be working directly with our staff at these times.)

When you have made your selection, the program should display a confirmation screen like the one below:



© 2026 ENERCALC, LLC

When you see this confirmation screen, you can dismiss it by clicking the [Continue to App] button.

CONGRATULATIONS!

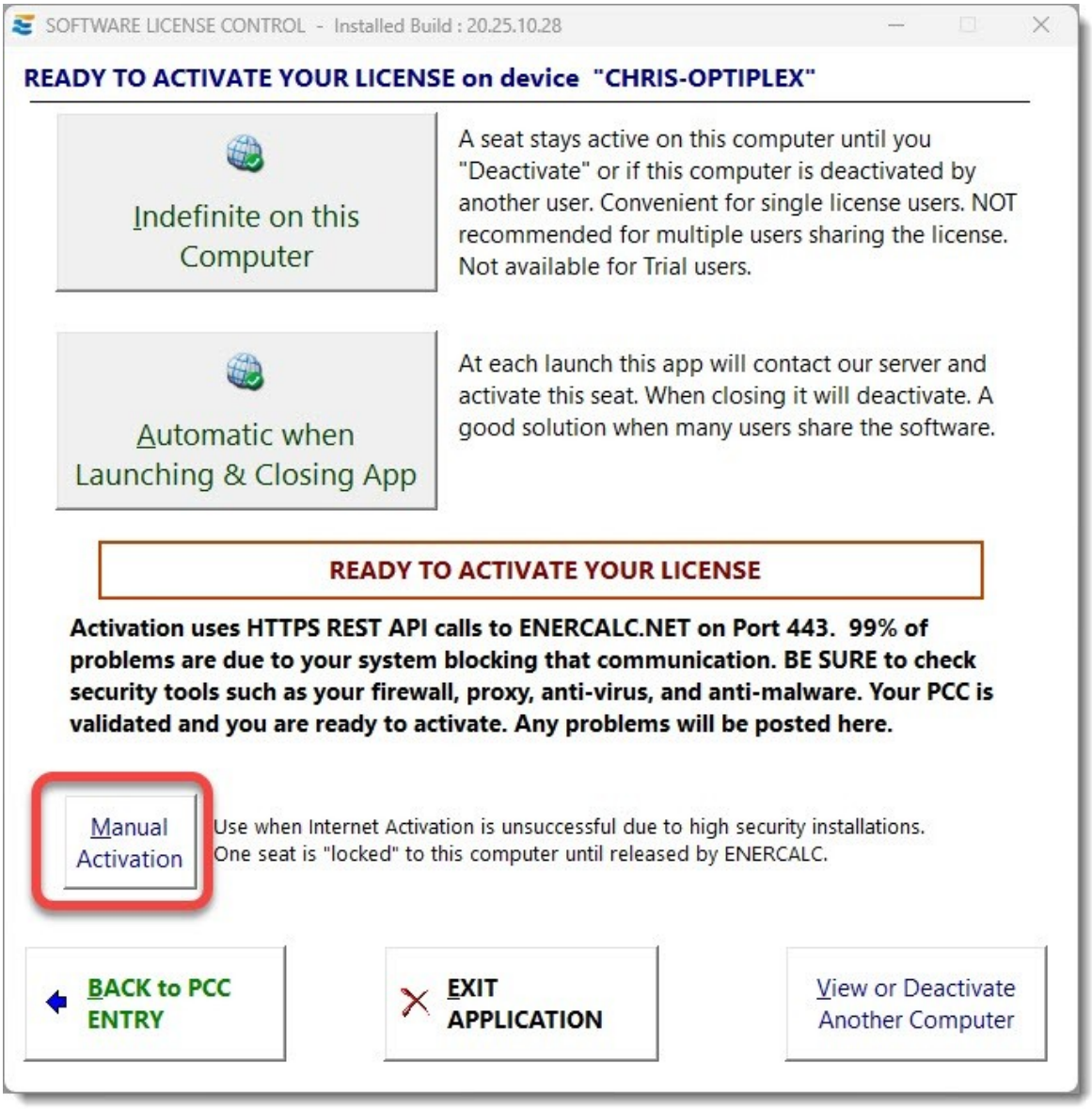
Your software is now fully activated!

NOTE: If you have any problems with activation, please be aware that during the activation process, the application must be able to communicate via Internet with Enercalc.net using HTTPS over port 443 using REST API calls. Anti-Virus software sometimes interferes with this communication. If this happens briefly “pause” your Anti-Virus software until Internet Activation is complete.

© 2026 ENERCALC, LLC

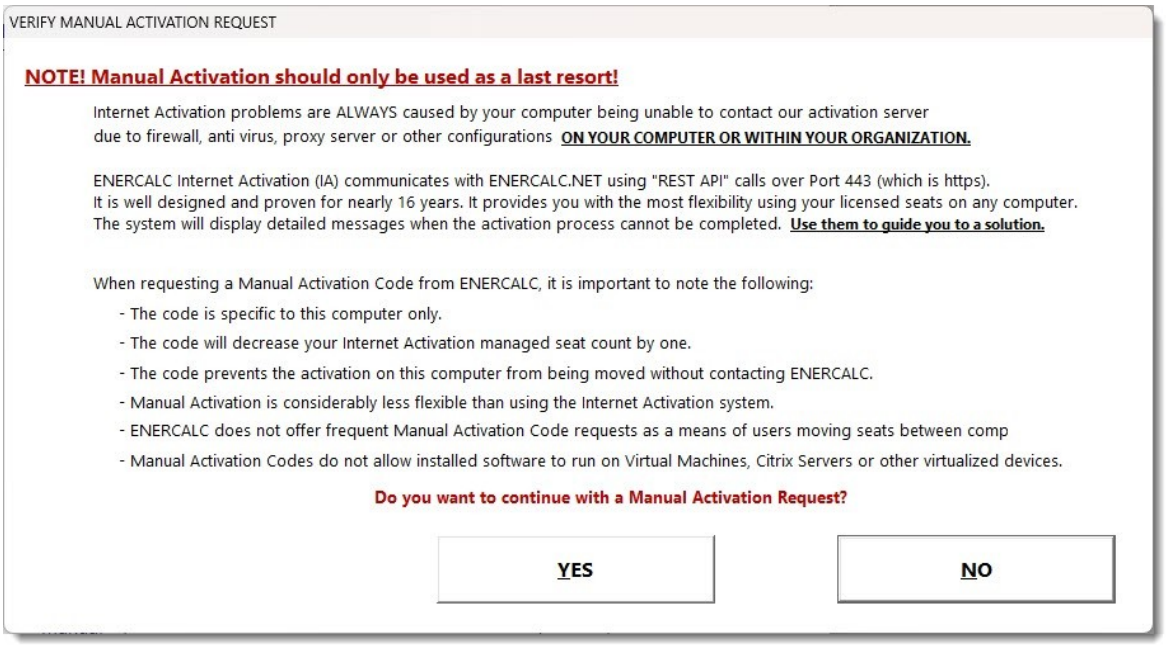
5.1 Manual Activation

If you cannot activate the software using the simple [**Internet Activate**] method (because you have no connection, restricted access to the Internet due to a firewall, or your company prefers not to activate your installation this way), you can activate your software using the [**Manual Activation**] method.



© 2026 ENERCALC, LLC

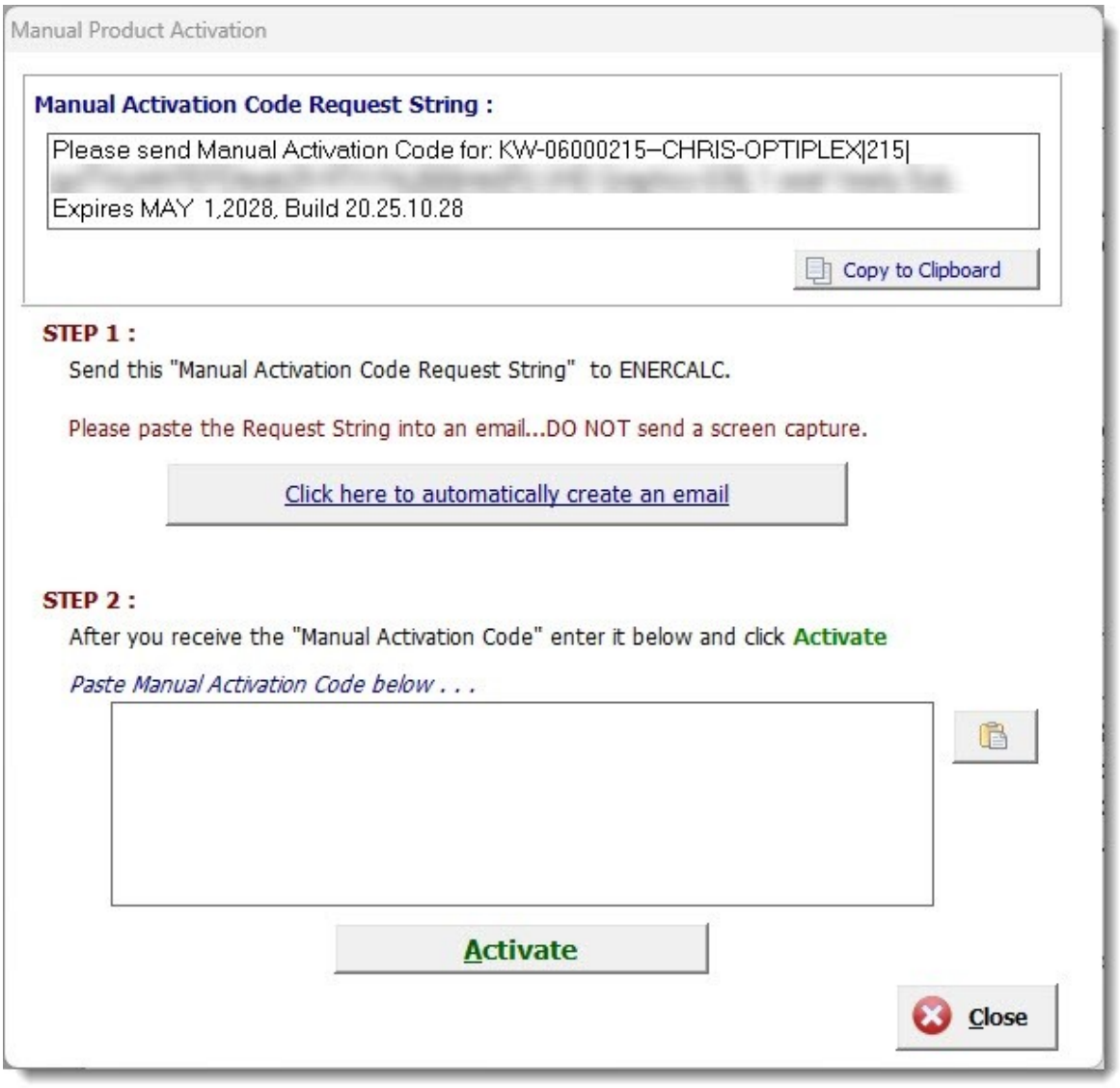
Click the [**Manual Activation**] button. The system will provide some information in a verification screen:



© 2026 ENERCALC, LLC

Click the [**Yes**] button.

Then instructions will be provided to you using the screen shown below:



NOTE: When using the Manual Activation method, the Manual Activation Code that you receive from ENERCALC can only be used to activate the ***specific machine*** that was used to make the Manual Activation Code request, because the request and the code actually contain the Computer Name and other data that is specific to that particular machine. However, you can still use the Internet Deactivate function to return a manually activated seat to your pool if that computer is Internet connected. And then you would be able to use that available seat from your pool to perform a normal Internet Activation on a different computer if desired.

© 2026 ENERCALC, LLC

5.2 Moving your Activation

This section will assist you in moving your activation from one computer to another:
Moving Your Activation

© 2026 ENERCALC, LLC

Part VI

6 Software Updates

6.1 Web Update

At **ENERCALC** we are continually improving and enhancing our software. To deliver the latest builds to all users, we use a built-in **Web Update** system that makes it easy to stay current with the latest fixes and improvements.

How to Check for Updates

Select **Settings > Updates** from the main menu to display the update screen shown below. There are two ways that you can check for updates:

Automatic Updates:

Enable "Automatically check for updates at program startup" to have the software check for updates each time it starts. If an update is available, you will be prompted to install it.

Manual Updates:

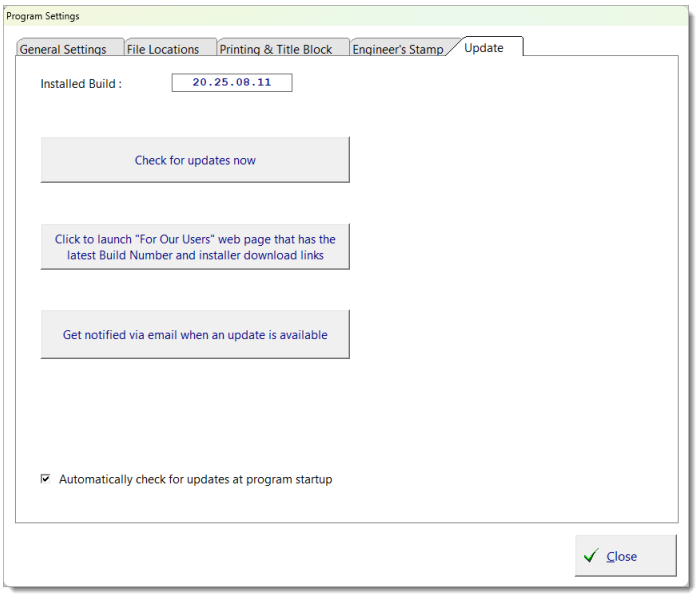
To check manually:

1. Click **Check for updates** now.
2. Follow the on-screen prompts to install the latest version.

TIP: Each ENERCALC build number is based on the date the build was released. For example: **20.25.07.31** → Released on **July 31, 2025** *(YY.YY.MM.DD).* Since ENERCALC releases updates once or twice a month, a build number showing a date more than 1–2 months old usually means you're running an outdated version. In these cases, it's a good idea to run the updater to install the latest fixes and improvements.

Want to be notified by email? Subscribe to our update announcement emails to receive alerts whenever a new build is released. Visit our subscription page and sign up with your preferred email address.

© 2026 ENERCALC, LLC



What Happens During an Update

How It Works

- The program *EC6WebUpdate.exe* securely connects to ENERCALC servers (HTTPS, port 443).
- Your installed build number is compared to the latest build.
- If an update is available, you will be notified. Upon approval, it downloads *EC6_WebUpdate.exe*, which installs the new build.

Both files are digitally signed applications and safe to run.

NOTE: *EC6_WebUpdate.exe* will need write access to the installation folder for updates to work.

Data Shared During Update

The update process may send information to ENERCALC about your installation and use of the software licensed from ENERCALC, including:

- Your User Registration Number
- Installed Build Number
- Internet IP address of the updating computer
- Time usage data for various portions of the software
- Potentially other information only related specifically to the use of the software license

Absolutely no files, configurations, settings, or other information not specifically regarding the usage of the ENERCALC license will be sent. If you are concerned about this, please contact us for information on what is being sent. We have an open policy on providing you with information showing what might be included.

© 2026 ENERCALC, LLC

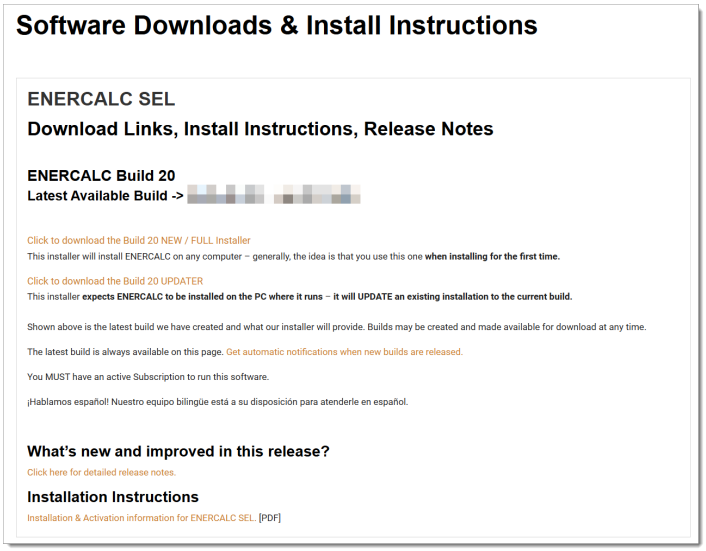
Advanced Options (for IT and Administrators)

If you manage multiple installations, you can control automatic update checks using this registry key:
`REG_CURRENT_USER\Software\ENERCALC\V20\DoCheckForUpdates`

- **1 = Yes** (check automatically)
- **-1 = No** (disable automatic check)

6.2 Update from Website

In addition to the Web Update system that is built into the software, it is also possible to initiate updates by visiting the For Our Users page of www.enercalc.com and using the Build 20 UPDATER link.



© 2026 ENERCALC, LLC

Part VII

7 Support & Maintenance

7.1 Subscription vs. Maintenance & Support Plan

In the past, our **Maintenance & Support Plan** (**MSP**) offering was the way to ensure that you will always had full technical support and access to the latest build of the software during the term of the plan. The industry standard is now to offer software on a **subscription** basis.

While your subscription is active, you will receive every new build, feature enhancement, and improvement for your licensed software, ensuring you're always using the most current technology.

Users covered by an active MSP will receive maintenance updates on those modules and functions that are provided by their plan.

You will have access to voice/fax/email technical assistance from our staff.

You will also receive discounts on new software releases and additional software licenses. This is the easiest and most economical way for you and your company to keep your software investment current.

Please see this web page for details on Subscriptions: Subscription Info

Please see this web page for details on the Maintenance & Support Plan: MSP Info

7.2 Getting Assistance

There are several ways to get assistance with using **ENERCALC** products.

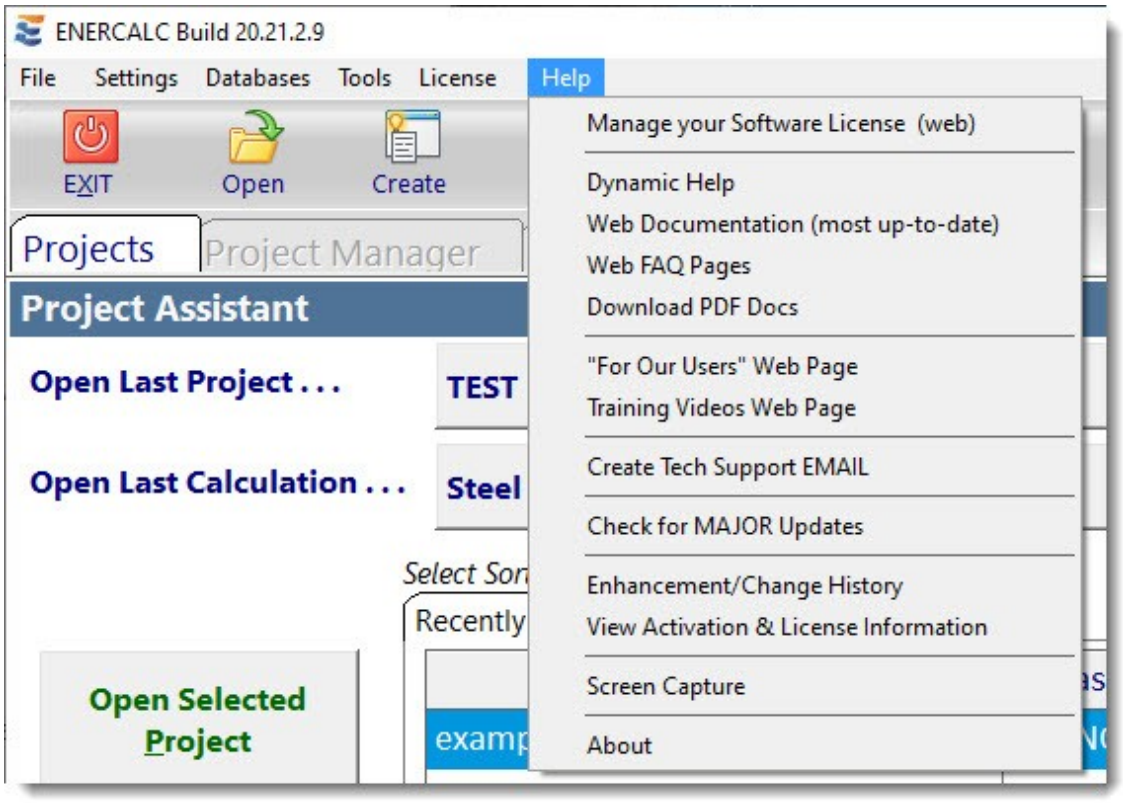
HOWEVER, with the exception of referring to the program documentation and reviewing the FAQ section on the website, all options require that you have an active subscription, or that your Maintenance & Support Plan is current.

- Refer to the program documentation.

© 2026 ENERCALC, LLC

- Contact ENERCALC Technical Services by email:

  From the main menu:



  From your email program: support@enercalc.com

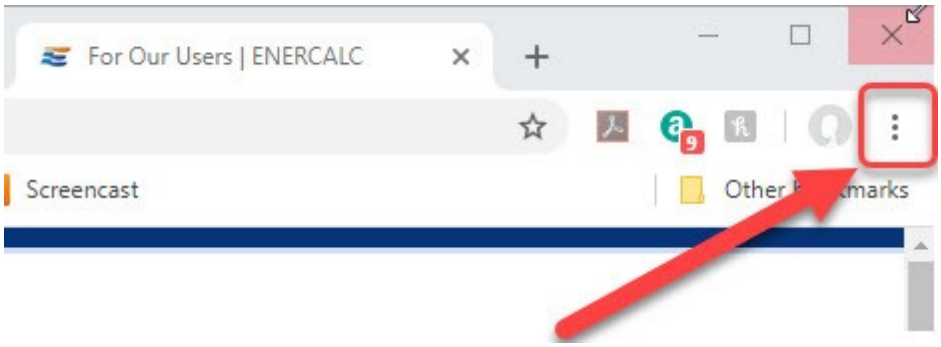
- Contact ENERCALC Technical Services by phone: 800-424-2252 or 949-614-0689
- Review the "Frequently Asked Questions" (FAQ) page on our website: FAQ

7.3 Troubleshooting with the Browser Console

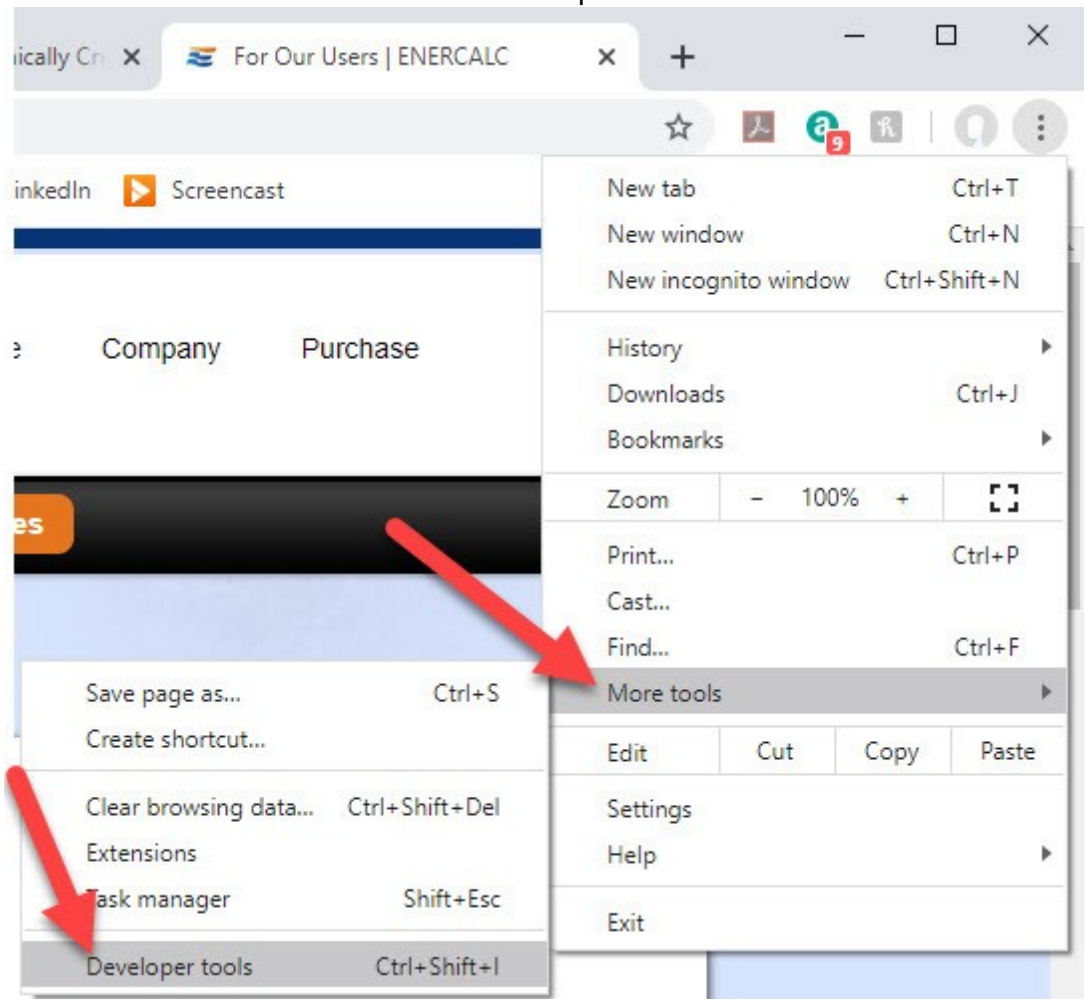
If you are ever in a situation where you are troubleshooting with our staff, they may ask for some info that is reported in the console of your browser. If you need some help locating the console, here are the steps that apply specifically to Chrome:

Click the Customize and control Google Chrome button in the upper right-hand corner of the Chrome window:

© 2026 ENERCALC, LLC

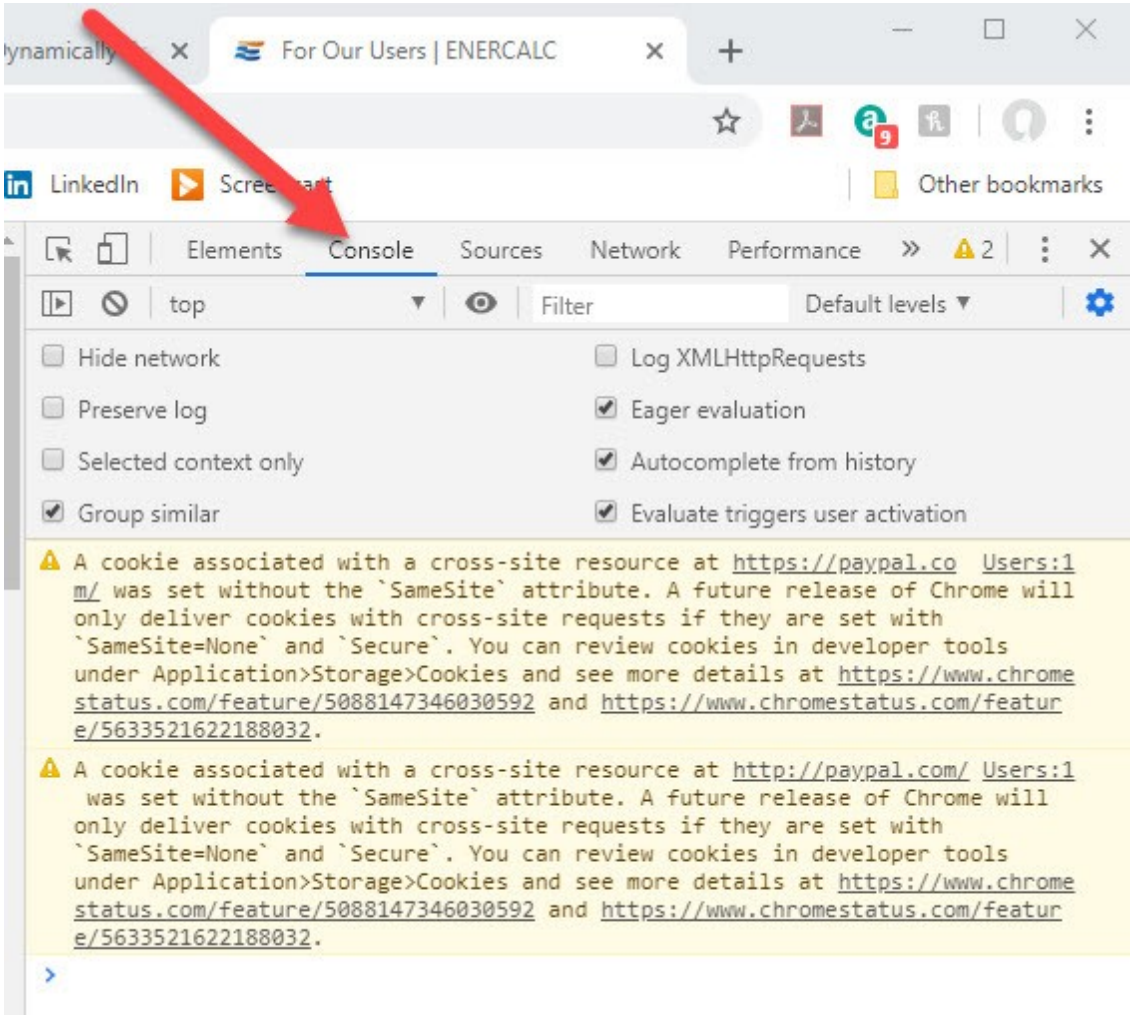


Use the menu to select More Tools > Developer Tools:



Finally, click on the Console tab:

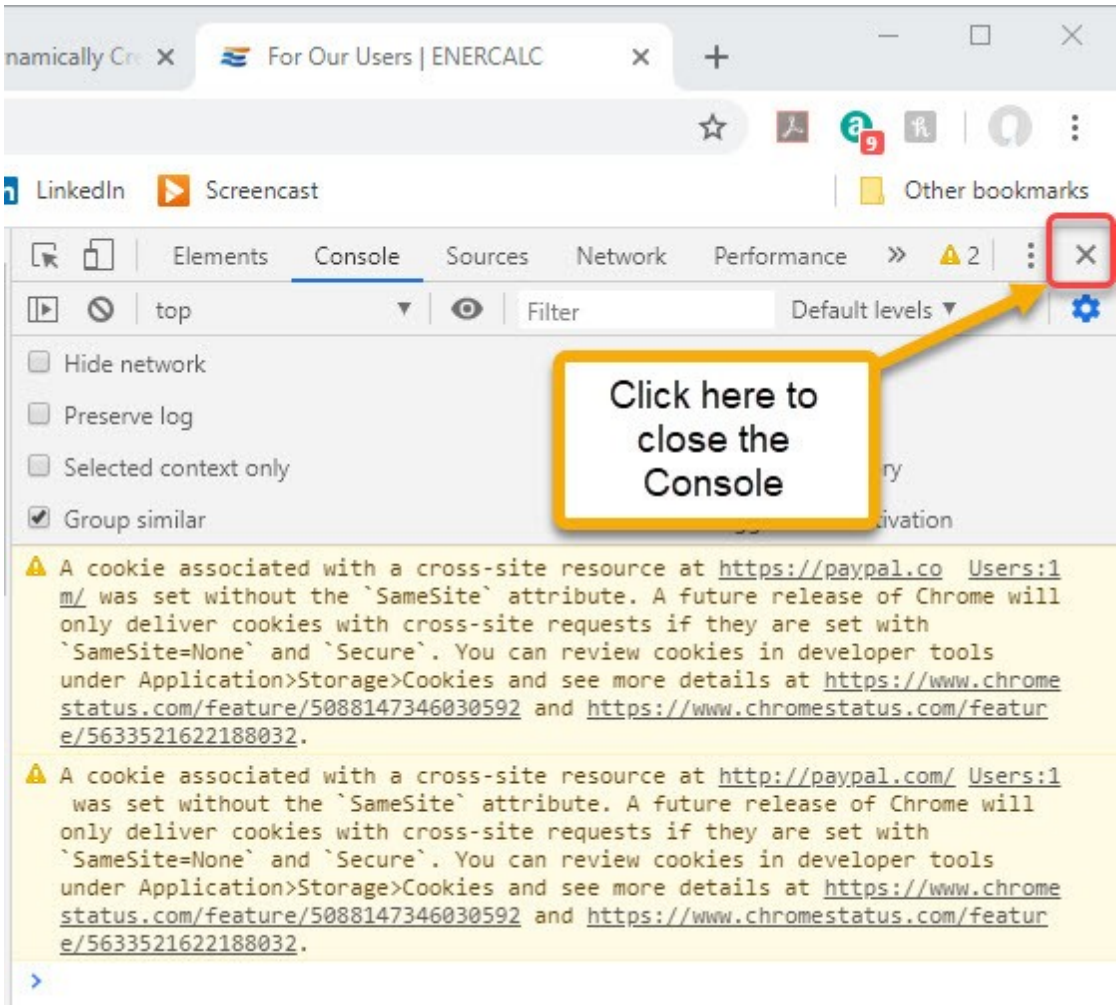
© 2026 ENERCALC, LLC



At this point, you will have access to some information that our developers may request. They may ask you to try to launch the app or perform some action. But sure to let the system work or fail or do whatever it needs to do. This generally means waiting 30 to 60 seconds before taking a screen capture.

When you no longer need to view the Console, you can close it by clicking the X in the upper right hand corner of the Console (not the X that closes Chrome, though):

© 2026 ENERCALC, LLC

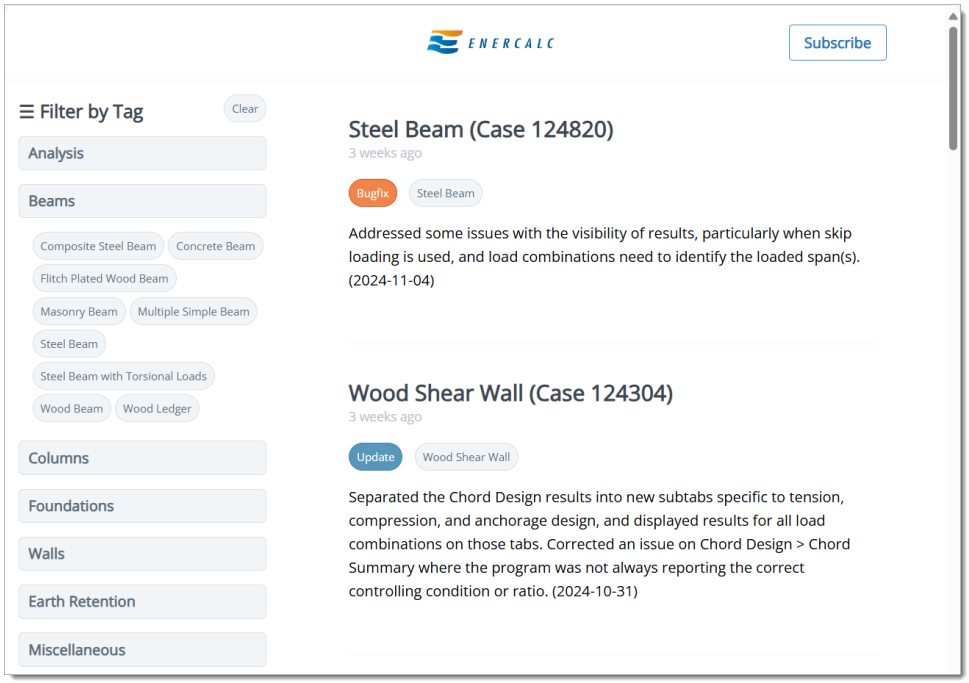


© 2026 ENERCALC, LLC

7.4 Viewing Enhancements and Changes to the Software

A comprehensive list of software revisions is available at enercalc.releasenotes.io.

You can also access this page directly from within the software by navigating to: **Help > Enhancement/Change History**



To view revisions specific to a particular module or feature, use the tags on the left-hand side of the release notes page.

To return to the main release notes page, click the **ENERCALC** logo at the top of the screen.

You can subscribe to stay informed about future updates via email by clicking the "Subscribe" button at the top of the release notes page. Alternatively, you can visit the subscription page directly here: https://enercalc.releasenotes.io/subscribe.

© 2026 ENERCALC, LLC

Part VIII

8 Getting Started

8.1 System Overview & Design Philosophy

ENERCALC SEL is a collection of modules that provides analysis and design functionality for components of buildings.

Walls, columns, beams, footings, diaphragms, frames, and other common elements can be thoroughly engineered through the use of the modules in this package. If you are a typical engineer whose work consists of a regular flow of small and medium-sized projects, ***this package is designed specifically for you***.

As an engineer you will find that each module combines the governing code provisions, mathematical analysis processes, and commonly available construction materials into a simple and effective "calcpad" style fill-in-the-blanks program. You can feel partially relieved that the software will consistently perform all the required checks that may be skipped over when doing repetitive hand calculations....especially when fatigue sets in and a deadline is near! You can enjoy the time to do more exhaustive design studies, come up with safer and more economical designs, and enjoy clearly documented calculations for review and archiving.

This software is not a "black box" program. Each calculation is designed to be a "visible calcpad" where you can work with the data and immediately view the resulting calculations. Automatic design is provided in most modules, and is intended primarily to automate tedious iterative tasks.

You, as an experienced structural engineer or architect, can quickly enter and change member sizes and other design parameters and view the results. In this way, **ENERCALC SEL** maximizes the use of your time and design skills by enabling you to quickly define a concept and then make necessary modifications to refine it into a final design.

8.2 About Our Documentation

Documentation of your software package is essential for your successful and pleasant use of our products. We try hard to supply you, now and on an ongoing basis, with detailed information on all aspects of the software. To support this commitment, we provide documentation of your software in multiple forms:

- A **Windows Help system file** named ENERCALC.CHM is installed with your software. This help system may be accessed by clicking **Help > Dynamic Help** from the main menu.
- A **User's Manual** in Adobe Acrobat PDF file format is available for download at any time from our website: ENERCALC SEL User's Manual.
- An **Online Help system** is available at http://www.enercalc.com/sel_help.

Printed Documentation

© 2026 ENERCALC, LLC

Printed documentation is not provided with ENERCALC products. This is in keeping with the nearly universal industry move away from printed documentation.

Updating your Documentation

The most up-to-date documentation for our software products is always available in electronic form. Whenever the software is updated with the built-in Web Update system, part of the process includes transferring the latest documentation to your computer. This ensures that the content that you view by clicking **Help > Dynamic Help** from the main menu is current and coordinated.

8.3 Building Codes Supported

Click here to view: Supported Building Codes and Design Standards

Databases of all commonly available steel and wood sections and materials properties are also included.

8.4 Learning Structural Engineering Library

There are several sources of information that will assist you with learning to use **ENERCALC SEL**.

- A series of tutorial videos is available at our website: http://www.enercalc.com/training_videos.html
- This documentation. An Internet version of the help system is available at: http://www.enercalc.com/sel_help/
- Responses to "Frequently Asked Questions" are available at our website: http://www.enercalc.com/faq_help/

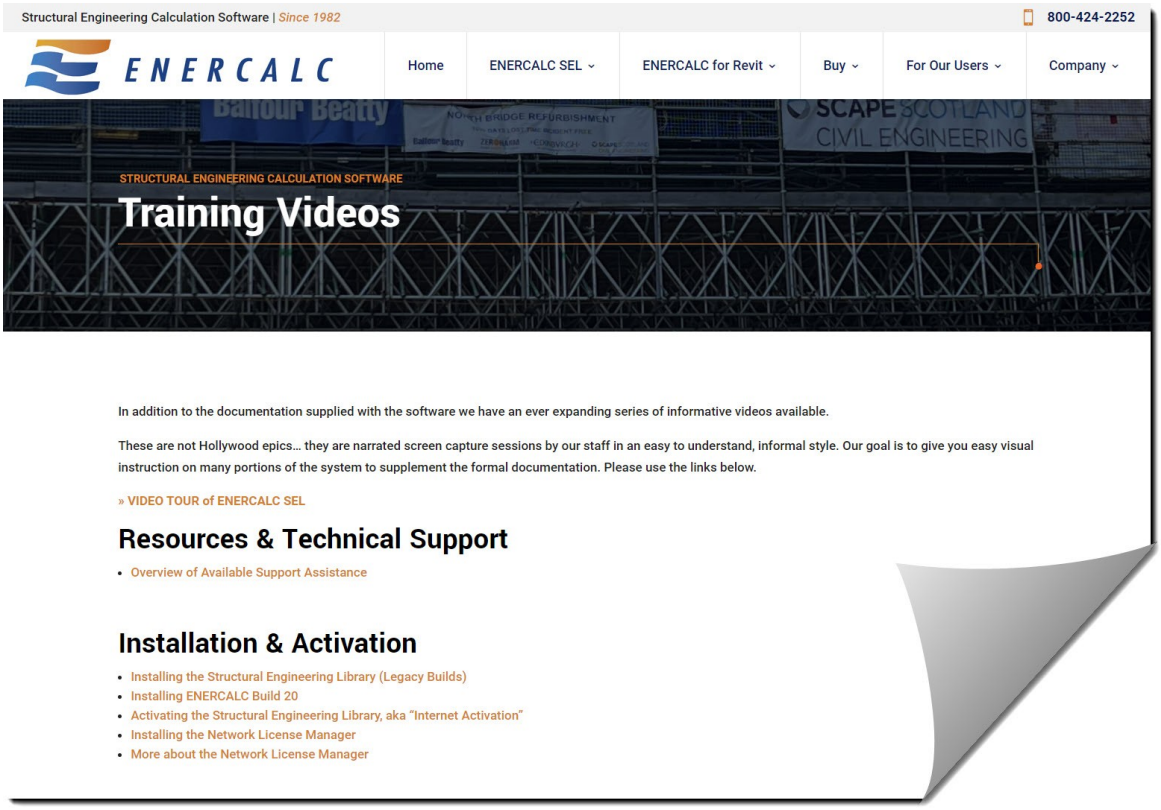
8.5 Introductory Videos

We continue to add topics to a series of videos that discuss all portions of the software.

To access nearly 40 videos please click here: Training Videos

Here is a partial view of what you will see....

© 2026 ENERCALC, LLC



8.6 Request for Suggestions

Although our intentions are to provide you with the best product possible, it is likely that there may be areas of this User's Guide and the software itself that could be improved to better suit your needs, be made simpler to operate, easier to understand, or support engineering technologies that have emerged since this publication.

To call these to our attention and to offer suggestions for improvement, we sincerely request that you send us your thoughts. Please address your comments to support@enercalc.com.

© 2026 ENERCALC, LLC

Part

9 Program Overview

9.1 Introduction

ENERCALC SEL is a collection of modules (also referred to as "calcsheets") that provide functionality for the analysis and design of components of buildings. Walls, columns, beams, footings, diaphragms, frames, and other common elements can be thoroughly engineered through the use of the modules in this package. If you are a typical engineer, whose work consists of a monthly flow of small and medium-sized projects, this package is designed especially for you.

As an engineer, you will find that each module combines the governing code provisions, mathematical analysis processes, and commonly available construction materials into a simple and effective "calcpad" style fill-in-the-blanks program. You can feel partially relieved that the software will consistently perform all the required checks that may be skipped over when doing repetitive hand calculations. You can enjoy the time to do more exhaustive design studies, come up with safer and more economical designs, and enjoy clearly documented calculations for review and archiving.

This software is not a "black box" program. Each calculation is designed to be a "visible calcpad" where you can work with the data and immediately view the resulting calculations. Automatic design is provided in most modules, and is intended primarily to automate tedious iterative tasks.

You, as an experienced structural engineer or architect, can quickly enter and change member sizes and other design parameters and view the results. In this way, **ENERCALC SEL** maximizes the use of your time and design skills by enabling you to quickly define a concept and then make necessary modifications to refine it into a final design.

The "Calcpad" Approach

When **ENERCALC SEL** was designed in 1983, our concept was revolutionary.....design it like an engineer's calculation pad. When an engineer prepares a calculation, the finished product is a neat and organized sheet of paper that follows the design flow from load tabulation, force and stress calculation, to the final adequacy check of the structural component that will satisfy the task.

At the time, all other competing programs were aging versions of mainframe programs that had been modified to run on microcomputers. Many of those programs executed "batch" design, where the user entered all the data, told the computer to run the program, and then opened a crude file to review the results. ENERCALC was unique in that the input and output was mixed on the same screen....easy to see at a glance. But the most revolutionary aspect was the tremendous speed it offered to prepare calculations.

This great speed was due to the fact that you could change a number and instantly see all the updated results on the screen.

© 2026 ENERCALC, LLC

Moving forward, the current version of SEL for Windows maintains that same fill-in-the-blanks approach with instant recalculation of results. When using any of the approximately 30 calculation modules, all input data and output results are presented on the same screen and viewed just by selecting a tab that groups the information. **Whenever you change an input value, the entire module is recalculated and the results are immediately visible**. Thanks to efficient programming and fast modern computers, incredibly complex structural analysis and design is performed in a split second.

This instant updating also happens when you are viewing graphical sketches of designs or stress diagrams.....after any change, the graphics are instantly updated.

9.2 Typical Worksession

A Typical Worksession

In order of occurrence, here are the steps in using the software:

1. Start the program.

2. The Project Assistant is displayed where you can choose to use the last calculation, the last project, a recent project, or create a new project.

3. A project is selected and you view the Project Manager. Here you can add new calculations, edit existing calculations, insert non-calculation items into the project (such as Excel files, Adobe Acrobat PDF files, or scanned images), and initiate project printing capabilities.

4. Editing or adding a calculation takes you to one of the calculation modules.

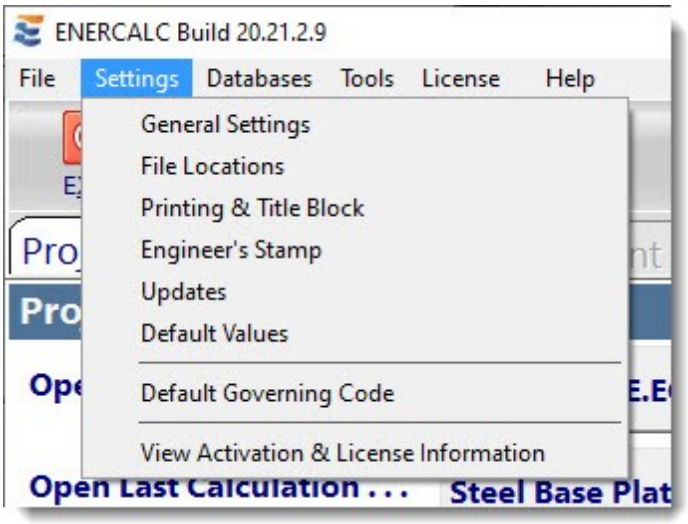
5. Using the calculation sheet, you enter data on the input tabs while reviewing results and graphics on the output tabs.

6. When the structural calculation is complete, you can print it and/or save it.

7. You can always return to the Project Manager where you can add/edit/delete/print other items within the current project, or save the current project and open a new one.

© 2026 ENERCALC, LLC

9.3 Program Settings



Most of these items are covered in another section of this help document. Please click on an item in the list below to jump to that section.

General Settings 69: Defines settings that are typical to the overall installation and to all Project Files such as backup file creation and behavior of the Project Manager.

File Locations 70: Specifies the default file locations for Project Files and databases.

Printing & Title Block 71: Provides the ability for the user to define the look of their title block.

Engineer's Stamp 73: Allows the user to upload a graphic image of their Professional Engineer's stamp for inclusion on printed reports.

Updates 73: Allows control over behavior of web update system and direct access to the update installation program on the web.

Default Values 74: Only accessible when a calculation module is open. Provides control over the default values used for that specific calculation module.

Default Governing Code: Allows the user to select the default governing code for all NEW project files. Note: This is different than changing the governing code for the current Project File or changing load combination sets.

View Activation & License Information: Provides access to user information.

9.4 Files

Project Files

© 2026 ENERCALC, LLC

ENERCALC SEL uses a single file to store the project information and all the calculations and items that are created as a part of that "Project". This file uses an "EC6" extension.

The system automatically creates backup files in the same folder that the ENERCALC Project File is stored.

To control the number of backup files to save, click **Settings > General Settings** and adjust the spinner control.

As **ENERCALC SEL** advances in capabilities, the file formats used will change, but we will always provide conversion programs for previously saved Project Files.

General Comments

- Each ENERCALC Project File contains all information on a Project. There are no other files you will need to keep track of.
- Always remember that the Project Manager is showing you the Divisions and calculations for the current Project. It is NOT showing you a disk directory structure.
- There is no **Save** item on the **File** menu. After you have edited a calculation, you can either click [**SAVE**] or [**SAVE & CLOSE**] within the individual modules. Either one will save the calculation data to the Project File.

Database Files

A number of database files are supplied with **ENERCALC SEL**. These files contain AISC section properties, NDS stress grades, wood section properties, seismic acceleration data, USA cities & Zip Codes, and other files. These files are not to be edited or modified in any way by the user.

User-Created Database Files

The user can create User Defined database files to store their own steel sections, wood sections and wood stress databases. These files are created and stored in the same folder as the other database files. User defined database files are differentiated from the standard database files that are delivered with the software by inserting the word "_USER" in the filename.

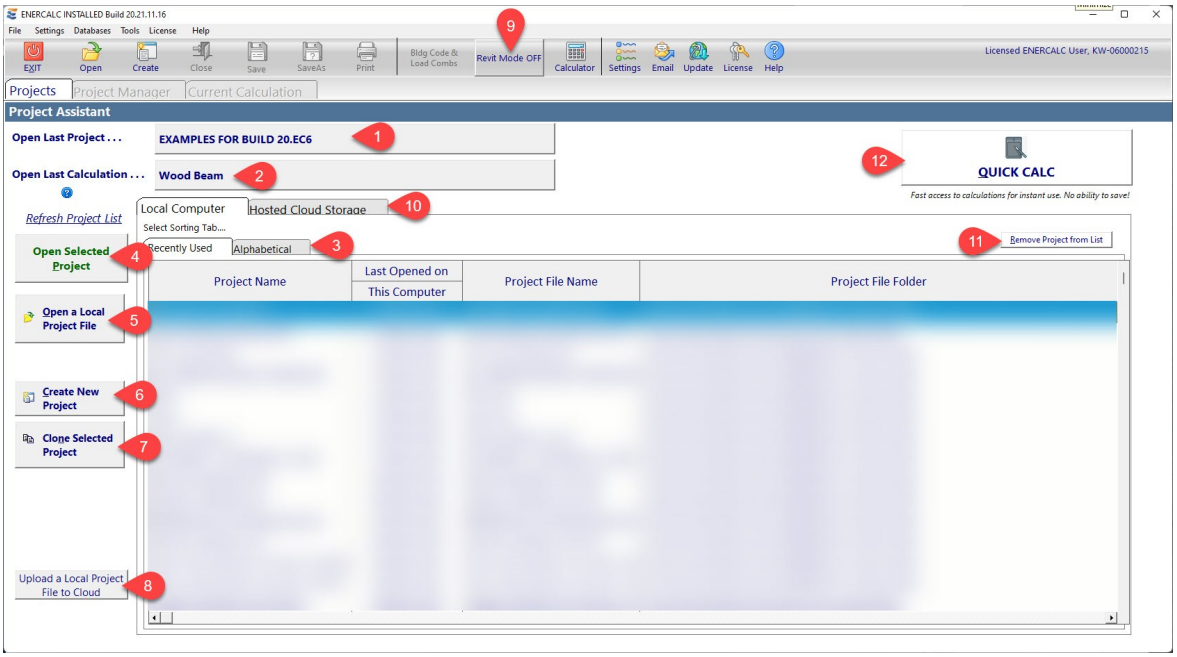
9.5 Projects Tab (Project Assistant)

When you launch **ENERCALC SEL** the Project Assistant is displayed by default, as shown below.

This single dialog allows you instant access to prior Project Files, the last Project File, the last-used calculation, or other functions to create or manage Project Files.

© 2026 ENERCALC, LLC

Please see the numbered references below the graphic for specific descriptions.



(1) Click to immediately open the last Project File that you worked on.

(2) Click to immediately open the last Project File that you worked on and open the last calculation you worked on.

(3) This list shows the most recent Project Files that you opened on this computer. The Recently Used tab displays the most recent Project File at the top of the list. Double-click on any item in the list to open that Project File.

(4) Click to open the highlighted Project File in the list.

(5) Click to open a Windows File Open dialog that allows you to browse through disks and folders to locate and open a Project File.

(6) Click to open the Windows File Create dialog that allows you to browse through disks and folders and create a new ENERCALC Project File.

(7) Click to clone the currently highlighted Project File.

(8) Click to upload the currently highlighted Project File to ENERCALC Hosted Storage in the cloud.

(9) Click to toggle Revit Mode. Controls the ability of SEL to detect if REVIT is sending it data. When Revit Mode is “ON” the program constantly executes background processes to monitor communication. For this reason, it is always best to have Revit Mode off when not needed.

© 2026 ENERCALC, LLC

(10) Click to select the desired Project File storage location.

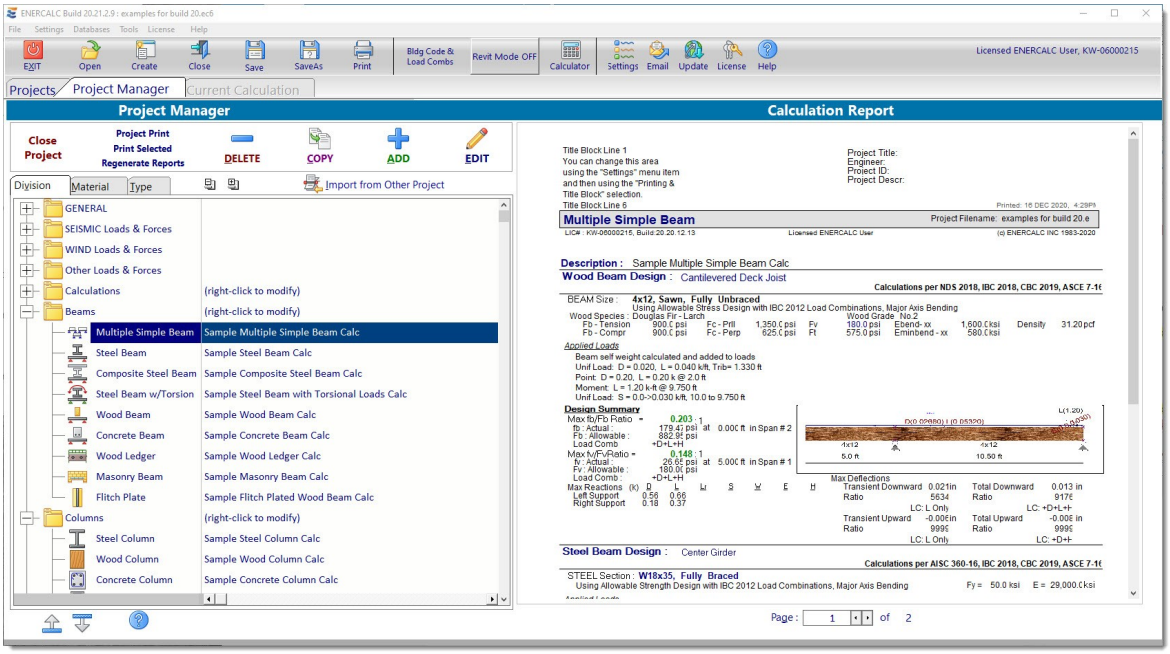
(11) Click to remove the highlighted file from the Recently Used list.

(12) Quick Calc provides fast access to calculation modules without creating a Project File first. ***Be aware, Quick Calc offers no ability to save after a calculation is started. So there will be no way to preserve a Quick Calc other than to print it to paper or PDF.***

9.6 Project Manager Tab

Project Manager

Immediately after a Project File is opened the Project Manager is displayed:

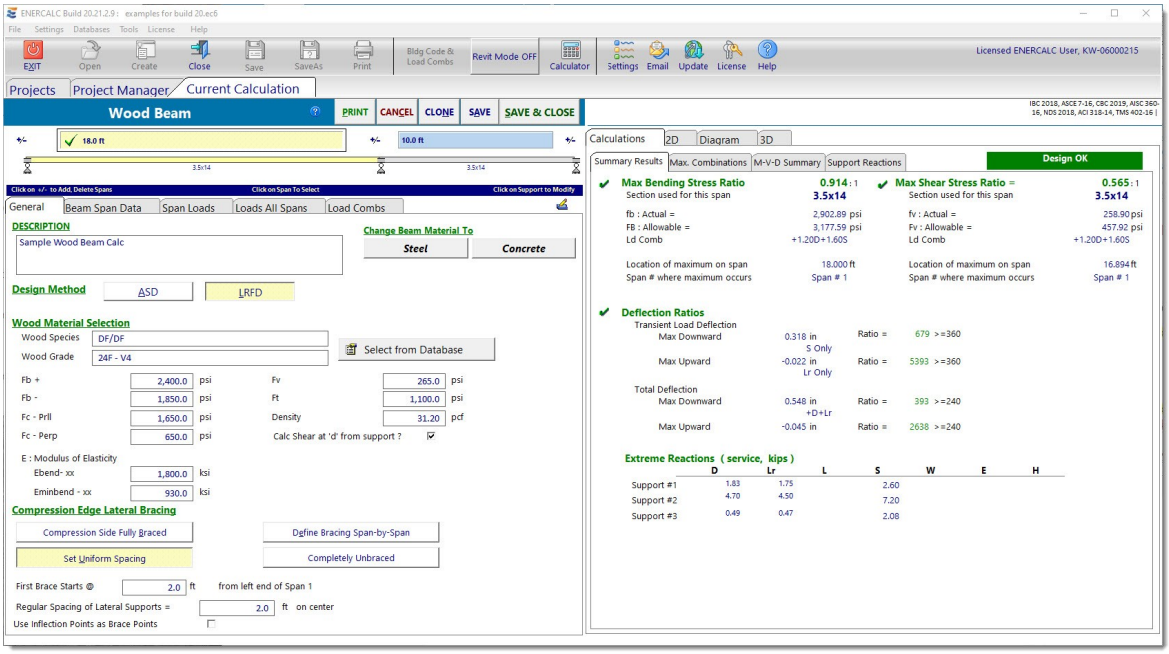


The Project Manager provides you with the ability to control the contents of your Project File. Double-click any calculation to open the associated module and edit the parameters of that calculation. Please click here 96 to jump to detailed information for the Project Manager.

© 2026 ENERCALC, LLC

9.7 Current Calculation Tab

After a calculation is selected for editing, the focus changes from the Project Manager to the Current Calculation tab, and the window fills with the user interface for that particular calculation module as shown below:

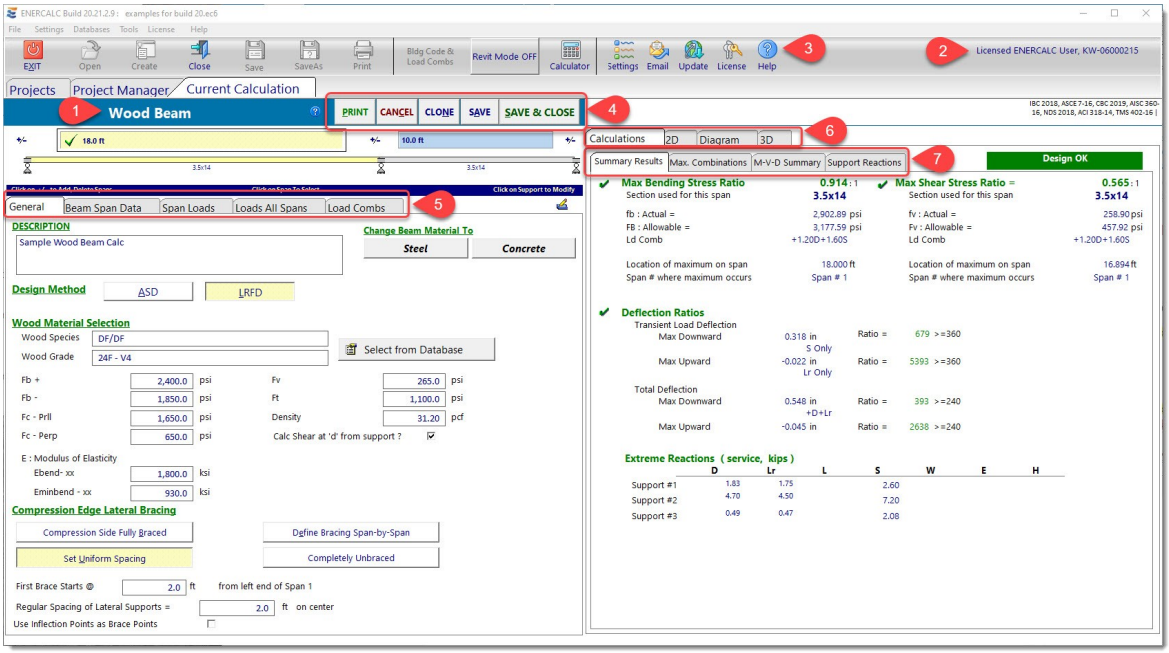


For a review of using a typical calculation screen click here 56.

© 2026 ENERCALC, LLC

9.8 General Calculation Screen Usage

Please refer to the typical screen below and the numbered items that relate screen areas to their descriptions:



(1) Indicates the module that you are working in.

(2) Indicates the name of the licensed owner of this installation & activation of the software.

(3) Access to help system: Opens the help system and displays the section specific to this module.

(4) From left to right:

Print: Opens the Print Previewer where the report can be set up, previewed, printed to a printer, or printed to PDF.

Cancel: Cancels all changes made to this module since the last save, closes the module without saving, and returns to the Project Manager. If this calculation module was just Added to the Project File, and if it was never saved, then this option will cause the calculation to be removed without saving.

Clone: Uses the current calculation data to create an identical new calculation item in the Project Manager.

Save: Saves the current Project File (to capture all the data entered into this module), creates a report (which can be printed at a later time), and keeps the module open for further editing.

Save & Close: Saves the current Project File (to capture all the data entered into this module), creates a report (which can be printed at a later time), closes the module, and returns to the Project Manager.

(5) Data input tabs: Click these tabs to move to various categories of data input.

© 2026 ENERCALC, LLC

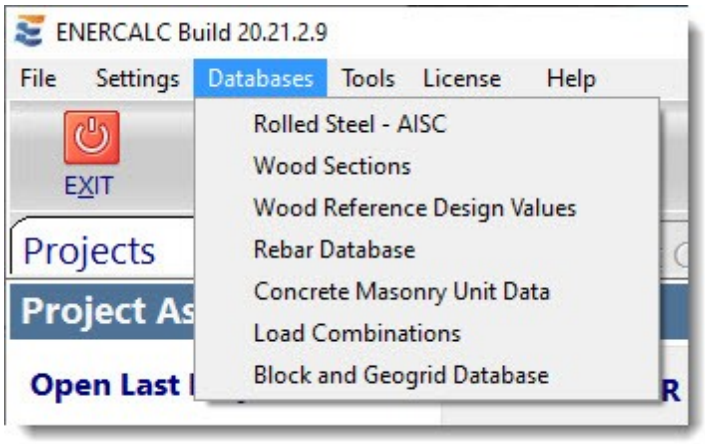
(6) Major Result Category Tabs: Select between major categories of result data to view; numerical values, sketches, diagrams or 3D renderings.

(7) Numeric Result Tabs: Click these tabs to view the various components of calculated results.

9.9 Databases

ENERCALC SEL contains several databases that you can use in the various modules.

To view the databases click **Databases >** (**database**) from the main menu. See below.



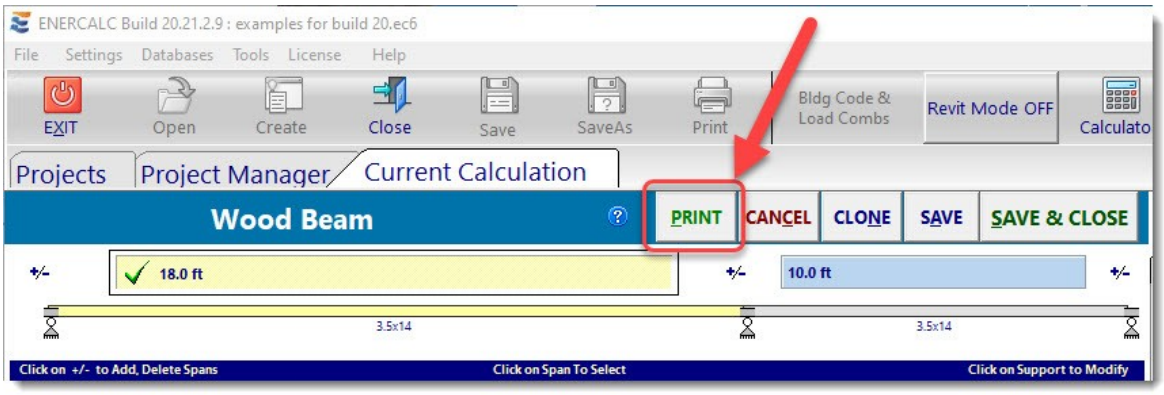
Specific information on each of the databases is provided here: Databases item in Main Menu 75

9.10 Generating Reports

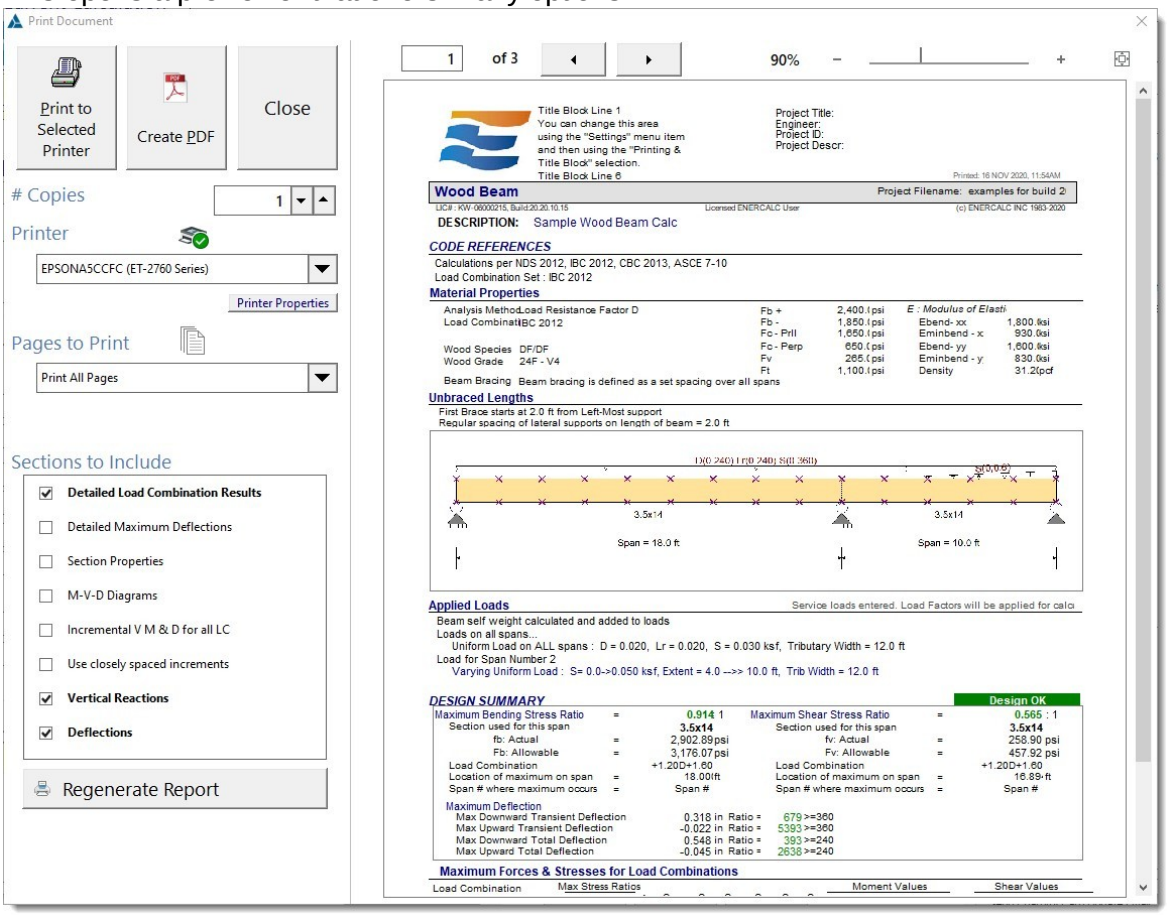
In **ENERCALC SEL**, the term "report" refers to the results documents created by the program in either printed or PDF format.

© 2026 ENERCALC, LLC

For each of the ENERCALC modules, the generation of a report starts by clicking the Print button:



This opens a previewer that offers many options:



Print to Selected Printer: Create a report and send directly to the currently selected printer.

Create PDF: Create a report as an Adobe Acrobat PDF file.

Close: Close the previewer without printing anything.

 # Copies: Set the desired number of copies.

Printer: Select the desired printer.

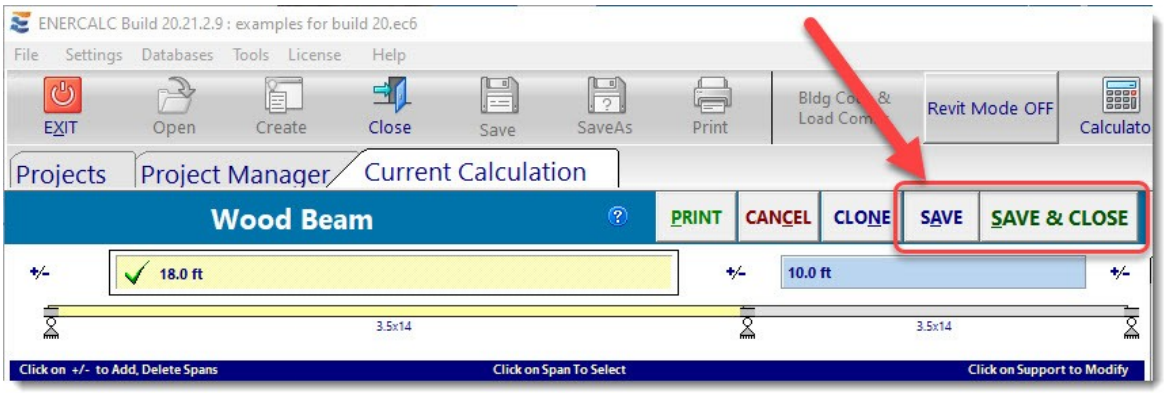
Pages to Print: Select what is to be printed.

Sections to Include: Select individual report topics to include in the report.

Regenerate Report: Recreate the report after changing the options in the Sections to Include area.

The right side of the screen displays a preview of the current calculation report and offers zooming and page scrolling controls.

In addition to the Regenerate Report button, reports are created and saved when you click the [**Save**] or the [**Save & Close**] buttons in the upper-right corner of all modules:

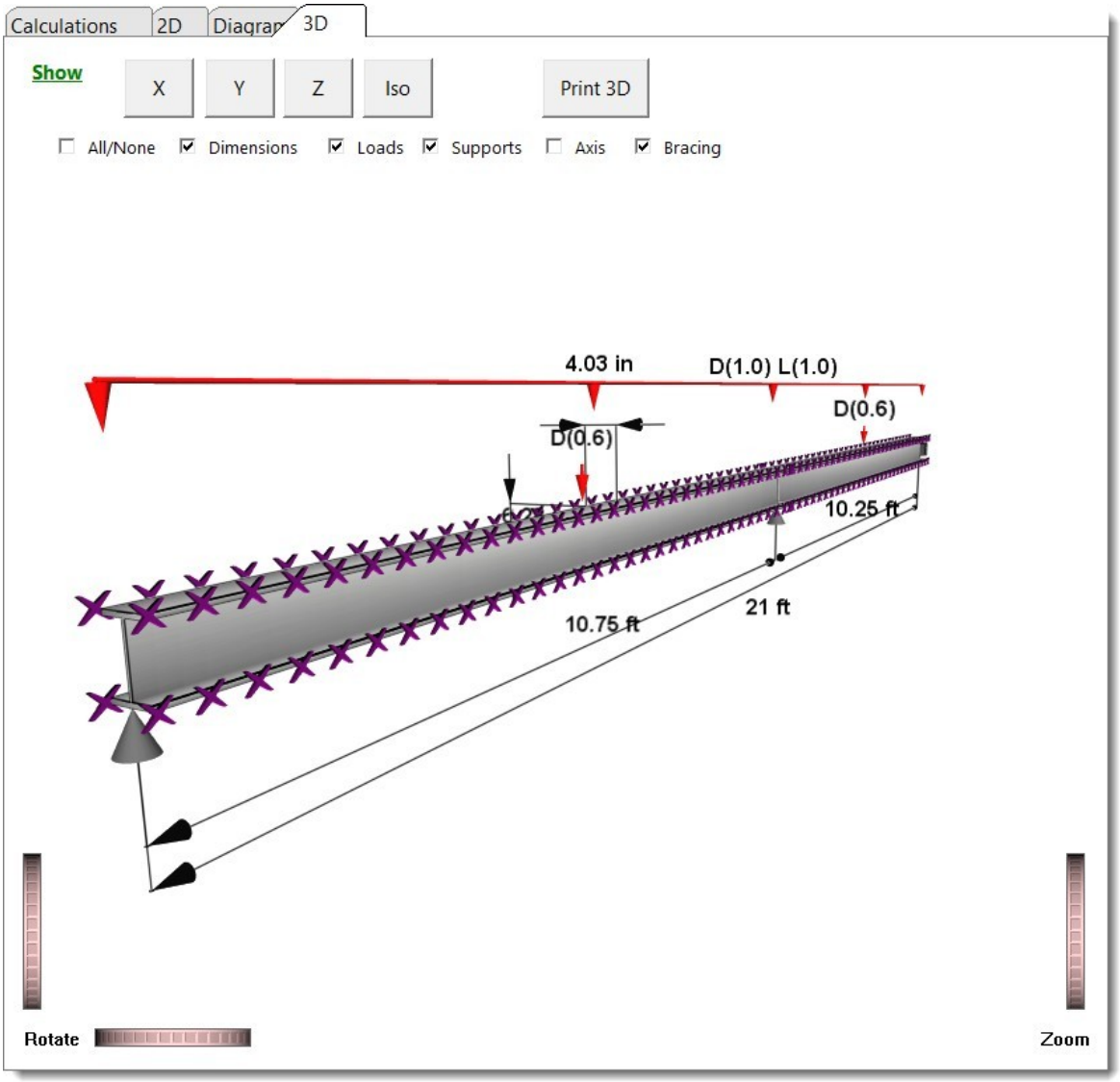


Clicking either of these buttons causes a full report for the current module to be created and saved into the Project File along with all of the input data for that module. This is how the printouts are prepared for instant viewing and batch printing in the Project Printing system.

© 2026 ENERCALC, LLC

9.11 3D Renderings

The 3D tab in most modules offer a 3D rendering of the element in the current calculation:



Each calculation offers "Show" options that are specific to the type of element being displayed. Note that some of the show options will cause some renderings to switch into a semi-transparent mode for clarity. For example, when viewing the rendering for a General Footing, the rendering will automatically switch to semi-transparent mode when the display of rebar is turned on.

The X/Y/Z/Iso buttons allow the rendering to quickly be snapped to specific view angles. Rotate and Zoom controls make it easy to adjust the view, but note that the rendering can also be manipulated by clicking and dragging with the left mouse button to rotate or with the middle mouse button to pan.

© 2026 ENERCALC, LLC

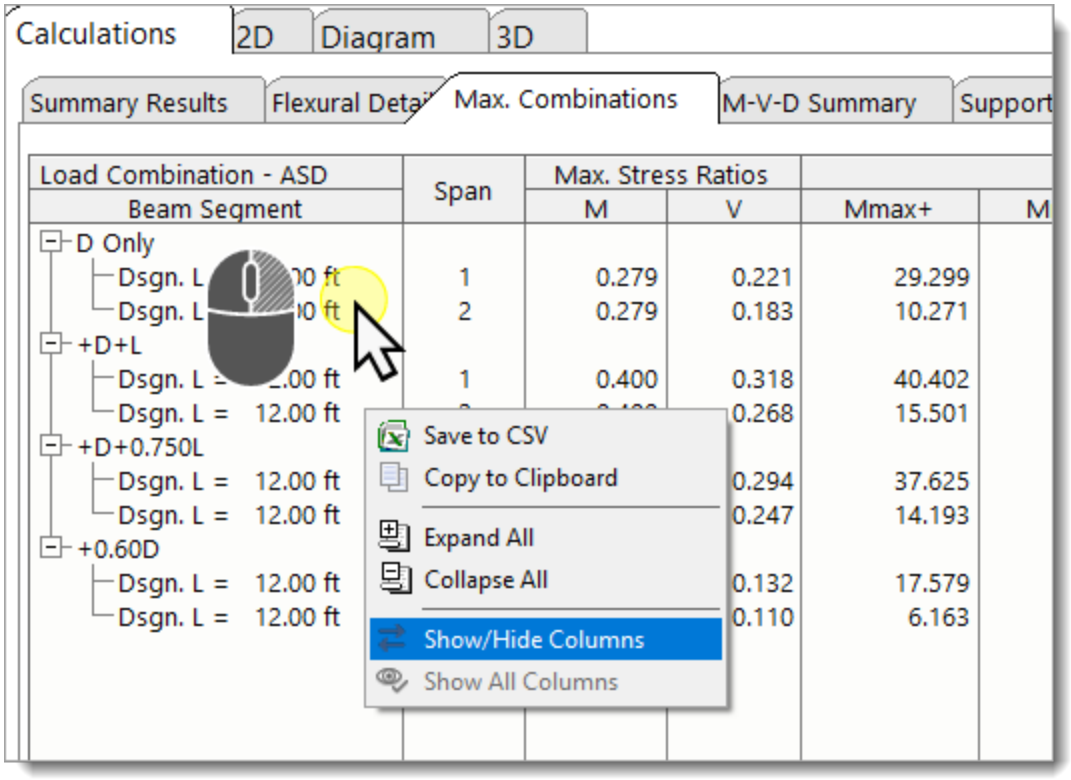
The Enlarge/Reduce button toggles between a large and small display window for the rendering.

The Print button allows the 3D rendering to be printed to PDF or paper.

If you experience any issues with viewing the 3D renderings, type Run in the Windows search box and then enter sysdm.cpl and press enter. Select the Advanced Tab. In the Performance category click Settings. Select "Adjust for best performance" and make sure Enable Peek is checked. Do not allow Windows to choose for you.

9.12 Right-Click Context Menu for Tabular Results

Many tabular result windows in the software provide a convenient right-click context menu that allows for quick data handling and view customization.



Note: These options are context-sensitive and may not appear in flat or non-tabular views.

When you right-click on a tabular result, the following options will appear:

Save to CSV:

- Saves the entire table content to a .csv (Comma-Separated Values) file.
- You will be prompted to choose a file location and name.
- Once saved, you will also be prompted if you want to automatically open the file.

© 2026 ENERCALC, LLC

- CSV files can be opened by all major spreadsheet applications and are useful for permanent records or advanced analysis.

Copy to Clipboard:

- Copies the right-clicked table content to your system clipboard.
- The copied content is formatted using standard tab delimiters, allowing for easy pasting into Microsoft Excel, Google Sheets, or any other spreadsheet software.

NOTE: For helpful tips to ensure proper formatting when opening or pasting the results in Excel, check out the following FAQ topic: ENERCALC SEL > General Operation > Exporting Tabular Results

Expand All:

- Expands all rows and subrows in the table, showing the full hierarchy of grouped or nested data. (Tree-structured tables only)

Collapse All:

- Collapses all nested sections in the table, reducing the view to top-level items only. (Tree-structured tables only)

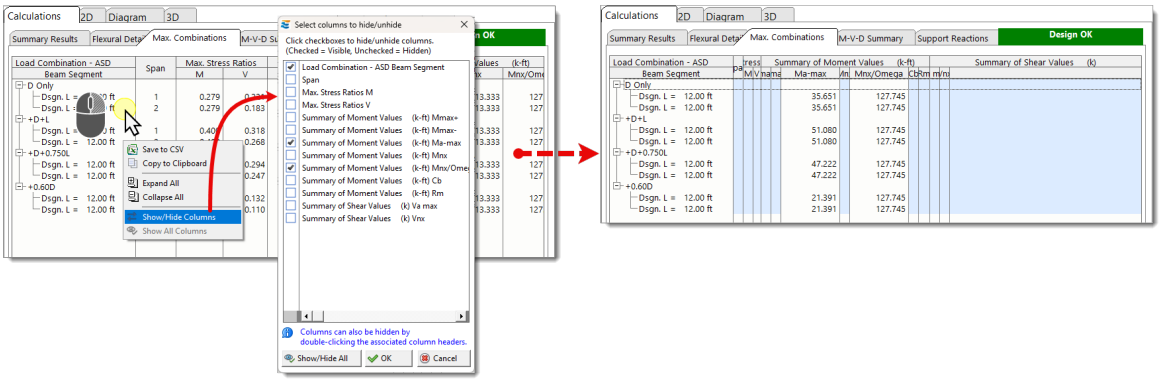
Show/Hide Columns:

- Opens a dialog listing all available columns for that table.
- From this dialog, columns can be turned on or off individually. This makes it easy to focus on specific results without resizing columns.

TIP: Columns can also be quickly hidden or made visible directly by double-clicking their column headers.

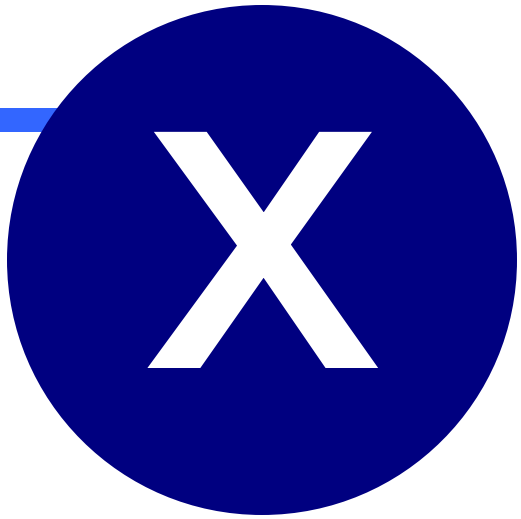
Show All Columns:

- Restores every column in the table to a visible state. Use this command to quickly reset the table after columns have been hidden.



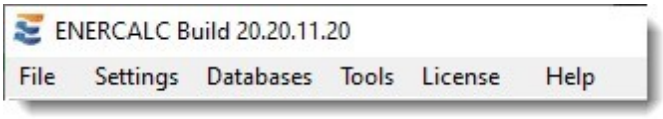
© 2026 ENERCALC, LLC

Part



10 Main Menu

The main menu of **ENERCALC SEL** is always displayed.

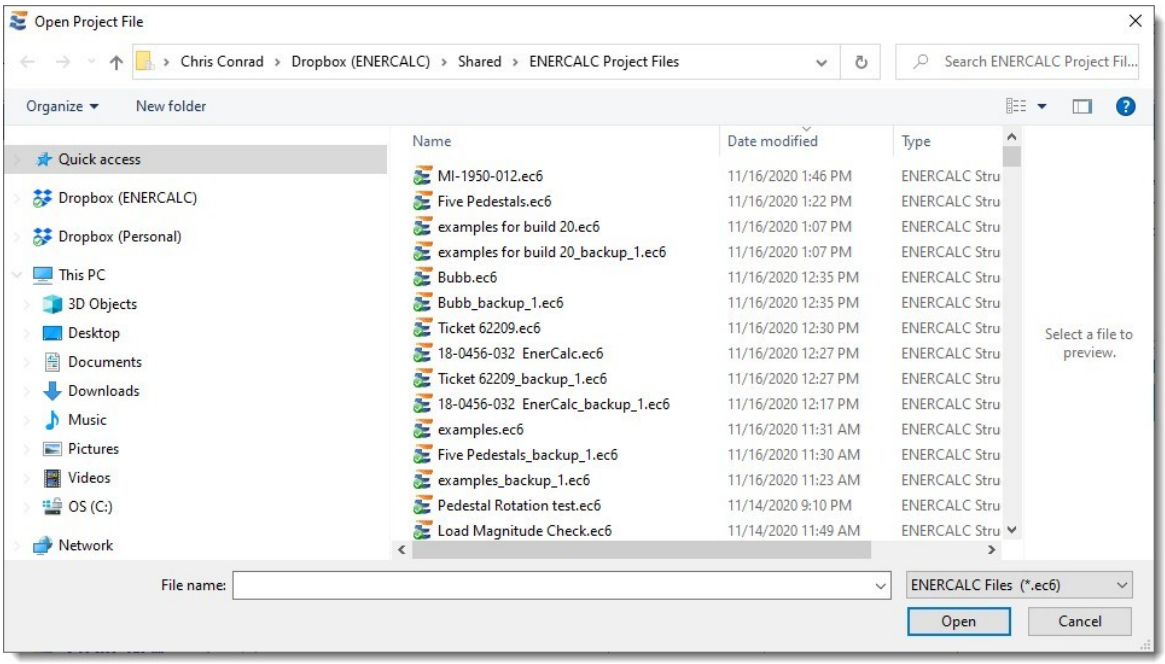


The menu offers the following selections. Click an item below to jump to that help section page:

File 64: To create, load, close and print Project Files.
Settings 68: To establish settings, user information and default values
Databases 75: To view the various databases supplied with the software
Tools 88: To access utilities provided with the software
License 91: To directly access the licensing control window
Help 91: To access the help system, technical support options, Knowledge Base with FAQs, Enhancement/Change History and the ENERCALC website.

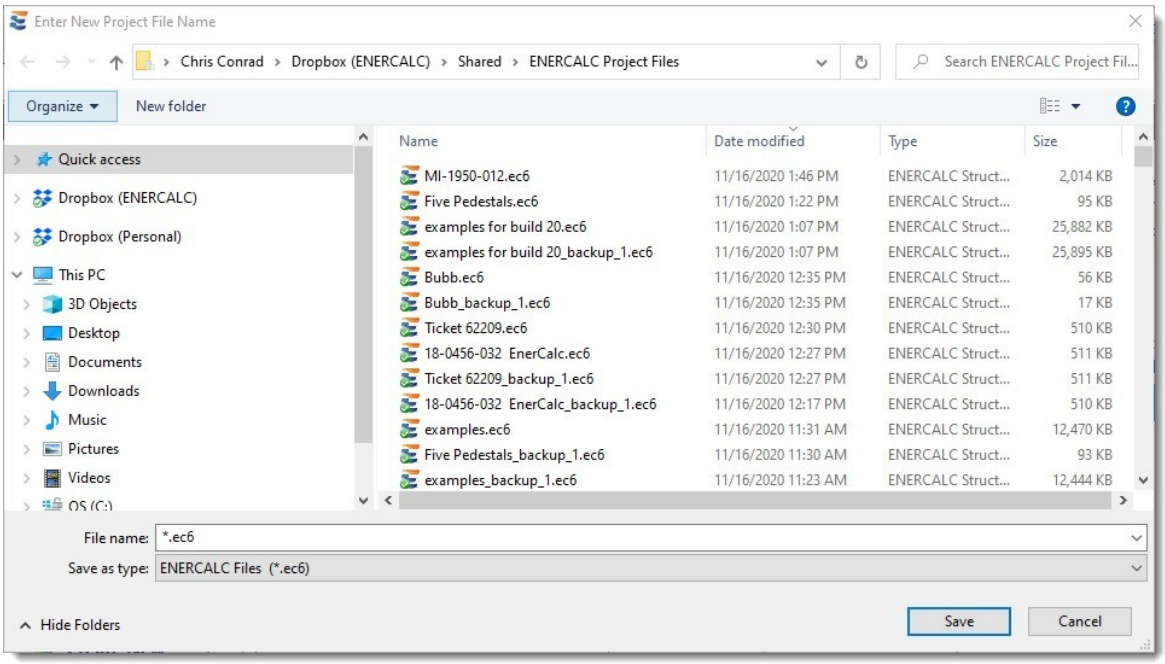
10.1 File

Open Project: Opens a standard Windows File Open dialog allowing you to navigate your disk drives and open a Project File.

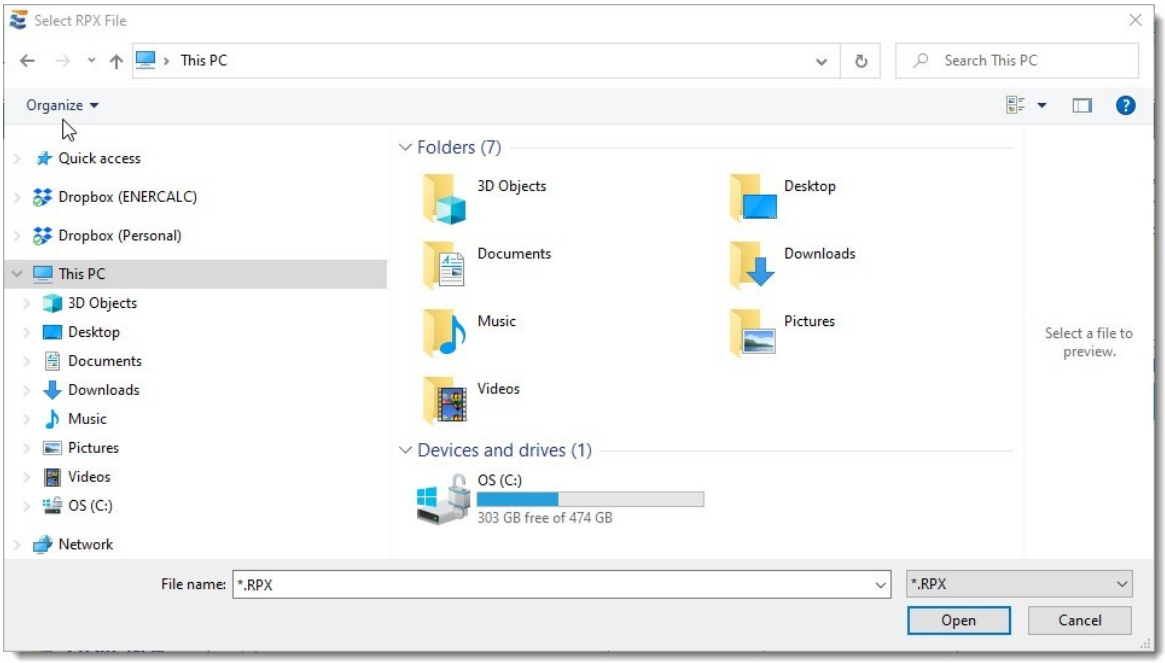


© 2026 ENERCALC, LLC

New Project: Opens a standard Windows File Create dialog allowing you to navigate your disk drives and specify the name of a Project File to create.

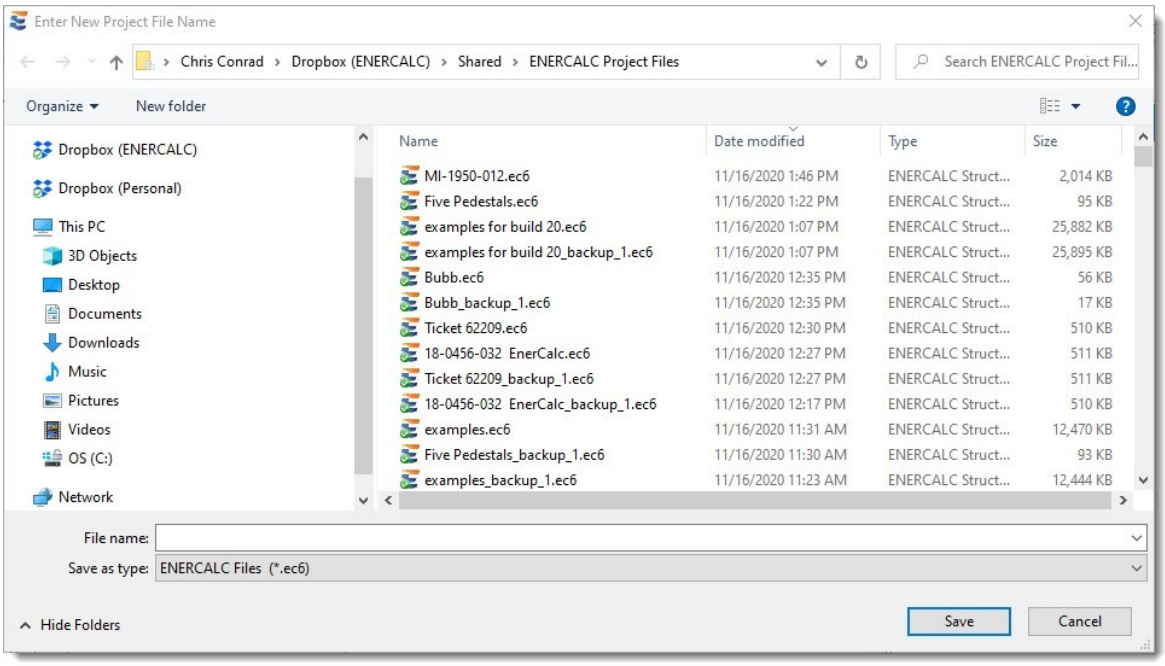


Convert RetainPro RPX file to EC6 Project File: Allows you to select an existing RetainPro RPX file and convert it into EC6 format, so it can be opened in ENERCALC SEL.



© 2026 ENERCALC, LLC

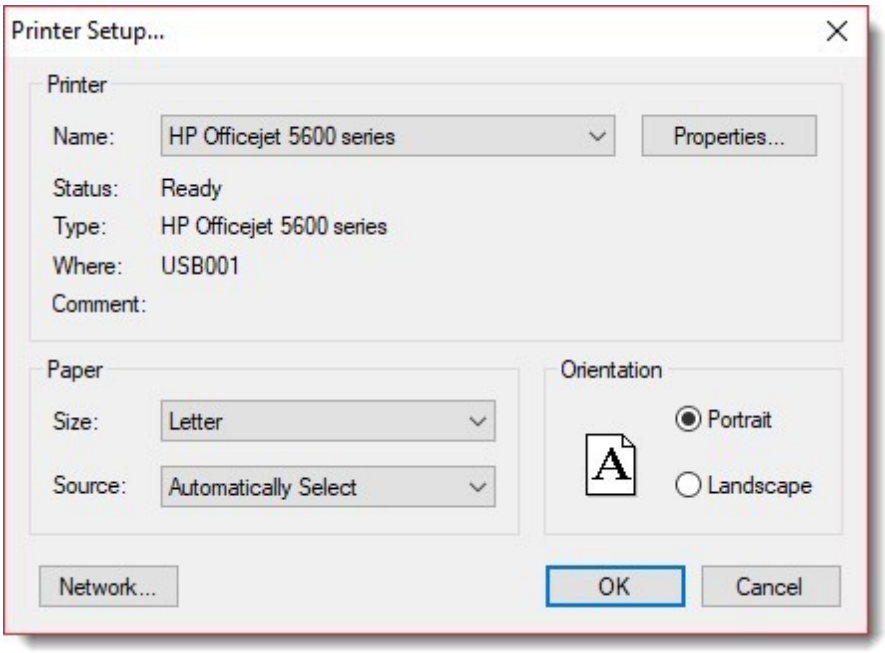
Save Project as New: Allows you to save the currently open Project File as a new file using a different name. Opens a standard Windows File Create dialog allowing you to navigate your disk drives and specify the name of the Project File to create.



Close Project: Closes the currently open Project File (making sure any unsaved data is purged).

© 2026 ENERCALC, LLC

Print Setup: Displays a typical Windows Printer Selection dialog to specify the printer to be used for this work session.



Print Project: Opens the Project Printing Manager.

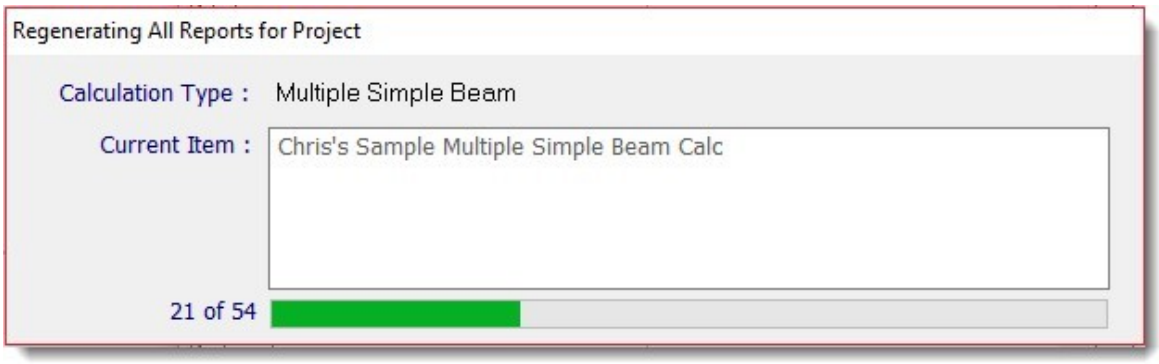
Regenerate Reports: This selection is used to regenerate printouts so they will be updated and displayed correctly in the Project Printing Manager. It can be used any time the following information has changed:

- Project Information in the GENERAL Division of the Project Manager for the currently open Project File
- Title block information filled in using **Settings > Printing & Title Block** from the main menu

There is also a third situation where this command can come in handy. When running in Evaluation mode, the software adds a watermark on the background of all printouts. This watermark says "Evaluation Version" and "Unlicensed Usage", so the report is inconvenient to use for submittal purposes. When the user later runs the software in Licensed mode, those reports with the "Evaluation Version" watermark remain in the Project File until they are reopened and saved. This can be a laborious process if the Project File contains many calculations. So the easier way to remove the watermarks in one pass is to use the Regenerate Reports command.

© 2026 ENERCALC, LLC

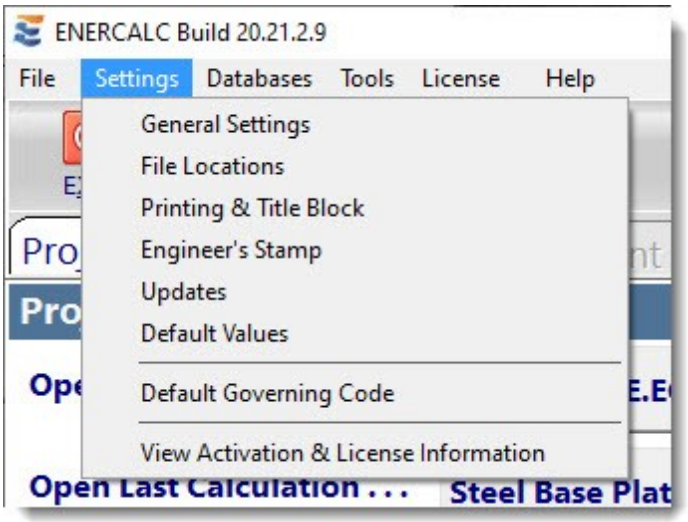
When you click **File > Regenerate Reports > All**, the program will automatically save and close any open calculations. Then the program will show a progress window as it regenerates all the reports in your Project File:



Exit: Offers the option to save any pending work, then closes the currently open Project File, and exits the software.

10.2 Settings

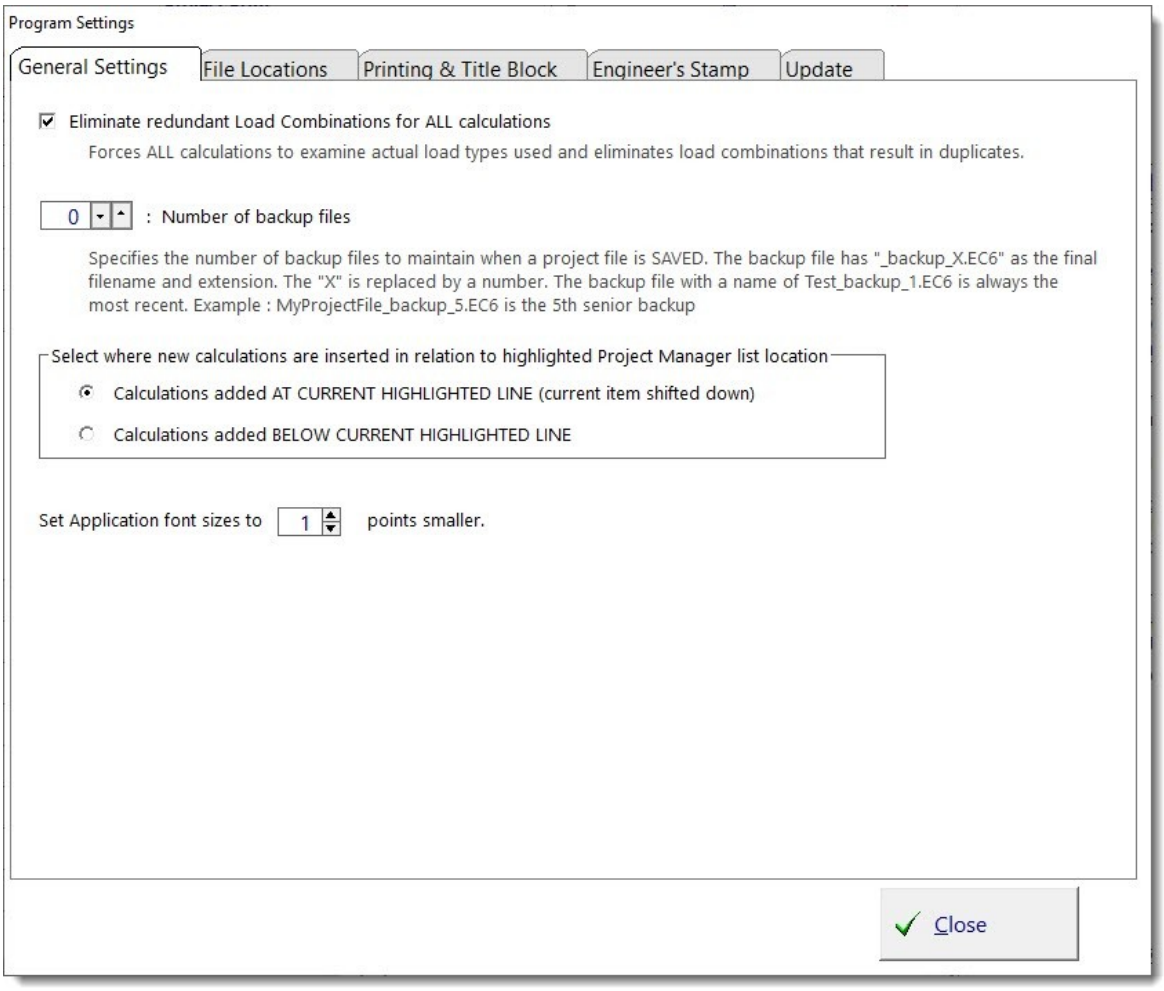
The **Settings** item in the main menu provides access to several selections that control how various aspects of **ENERCALC SEL** operates.



A selection of one of the first five items will open the Program Settings window and preselect the appropriate tab for the chosen item.

© 2026 ENERCALC, LLC

General Settings



Eliminate redundant Load Combinations for ALL calculations: With this box checked the software will only run load combinations that are unique. It will automatically eliminate any load combinations that result in a combination that is redundant.

Number of backup files: Specifies the number of backup files to maintain for each Project File. When the number is set to anything greater than zero, the software will automatically create a backup file with `_backup_[n].ec6` appended to the file name, in the same folder as the original Project File. The zero setting is provided under the presumption that the user has another backup system in place that they trust to protect their work, therefore automatic backups are set to 0 to avoid unnecessary redundancy. The backup files are created when a Project File is CLOSED.

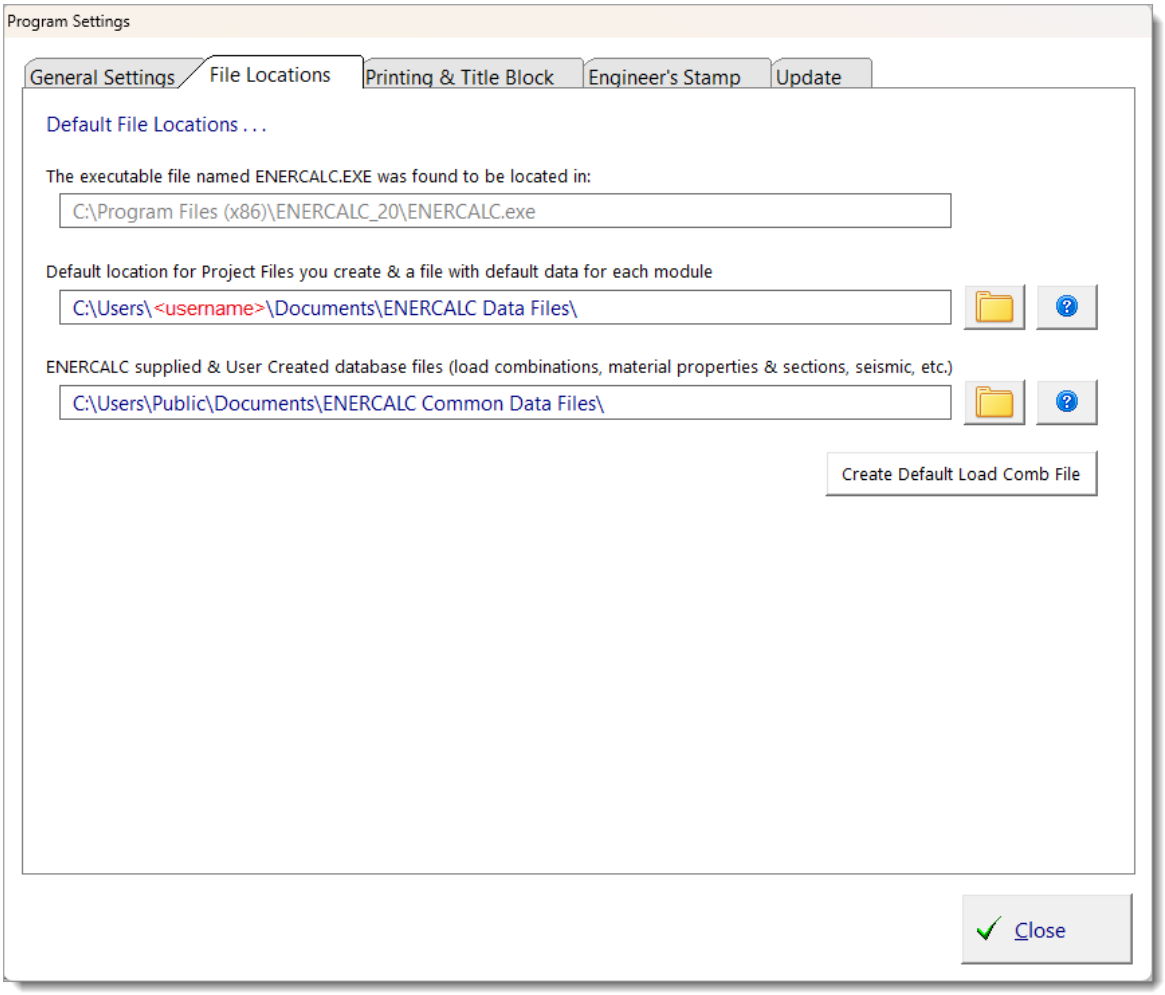
Select where new calculations are inserted...: This applies to the way calculations are inserted when using the Project Manager.

© 2026 ENERCALC, LLC

- When the first option is selected and you use the [**+Add**] button, it adds a new calculation to the project, and the calculation is inserted at the currently highlighted location (currently highlighted item is shifted down).
- When the second option is selected and you use the [**+Add**] button, it adds a new calculation immediately below the currently highlighted location.

Set Application font sizes to X points smaller: Provides a way to scale fonts down, for high resolution displays.

File Locations



Because modern operating systems allow multiple users per computer, Microsoft suggests that software manufacturers create their own folder under Documents and set that as the default location for user-created files. The program conforms to this recommendation by suggesting an appropriate directory during the installation process.

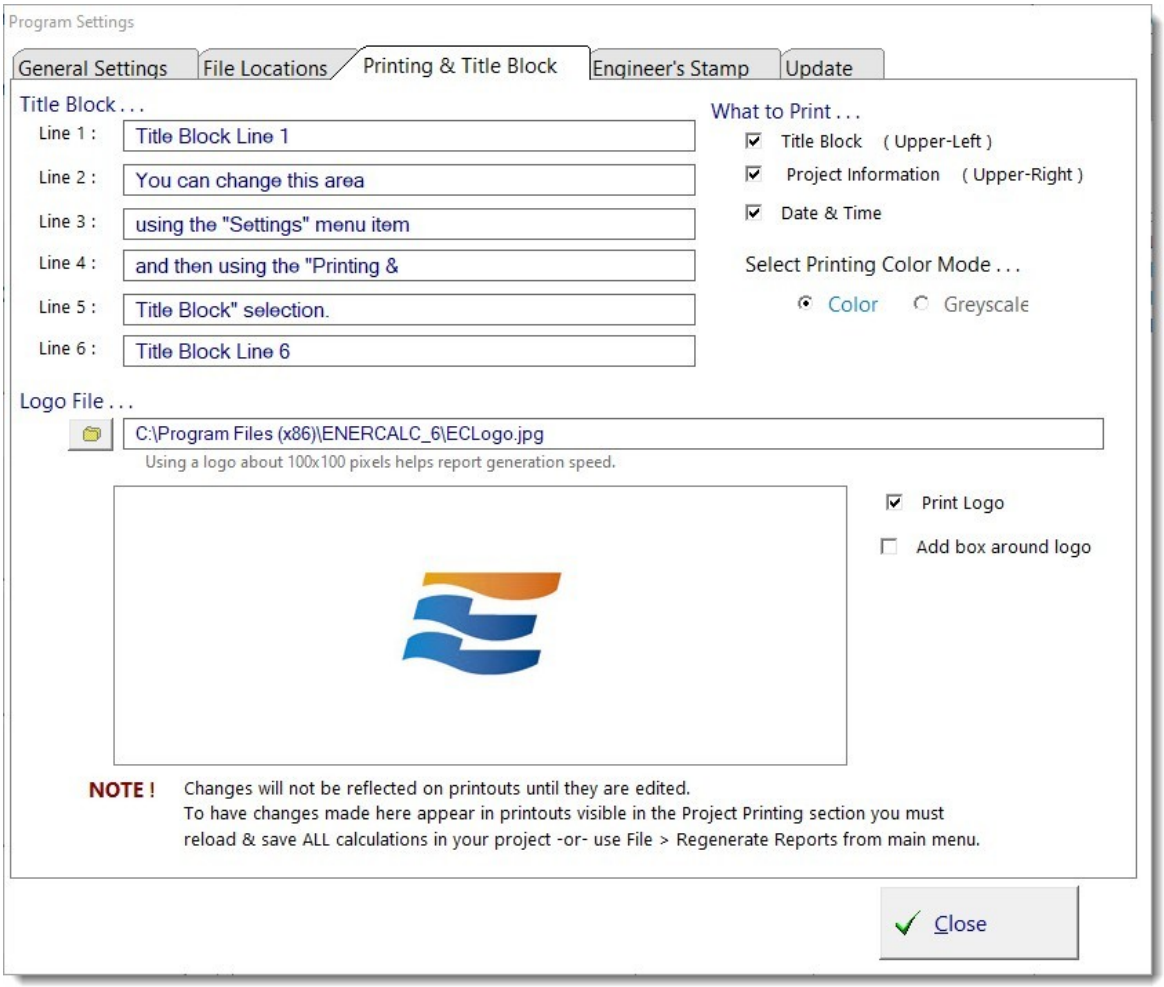
Executable File Location: Reports the location where *ENERCALC.exe* is located.

© 2026 ENERCALC, LLC

Default Project File Location: Specifies the default location where the program will point when using **File > Open** or **File > New** from the main menu.

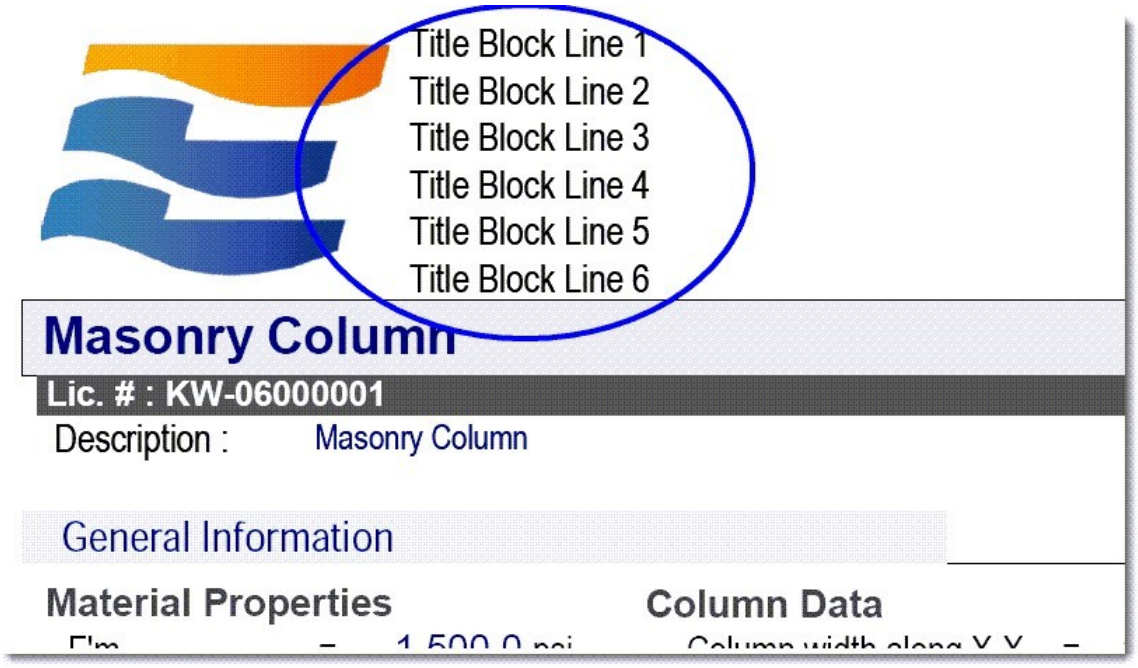
Database Database File Location: Specifies where the program should look for the load combination database, steel database, wood database, and other files that contain the databases of standard values. This location needs to be accessible to all users on a specific computer. Microsoft suggests that software manufacturers set the following as the default location for data files that are referenced by the software: `C:\Users\Public\Documents`.

Printing & Title Block



© 2026 ENERCALC, LLC

Title Block: These six entries correspond to the six lines in the upper-left corner of the printout.



Logo File: Provides an option to specify a graphics file to be printed at the LEFT edge of the title block. If used, the six lines of Title Block information will be printed immediately to the right of the logo. Logos can be JPG or JPEG formats only.

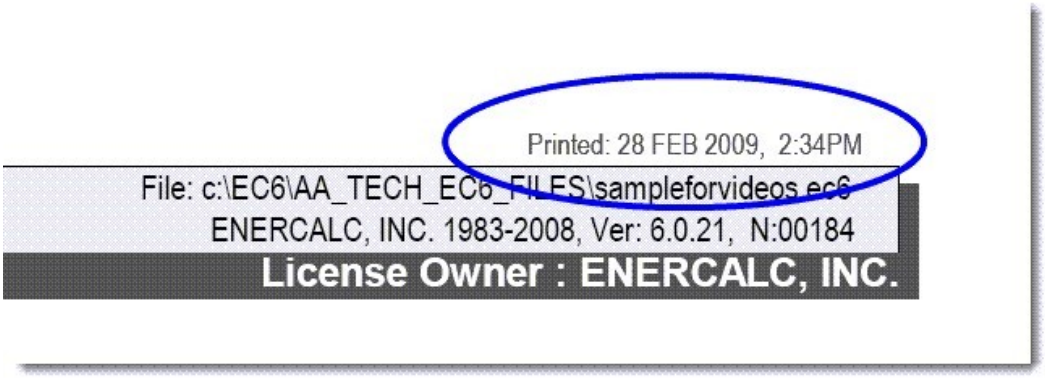
When a logo is specified, the height is adjusted to fit into the title block area and the right-side "floats" according to the width of the image.

What to Print - Title Block: Check this box to print the title block at the top left of the report. With this item unchecked the printout begins with the calculation title bar (shown as "Masonry Column" in the image above)

What to Print - Project Information: Check this box to print the Project Information at the top right of the report.

© 2026 ENERCALC, LLC

What to Print - Date & Time: Check this box to print the computer date and time on the report as shown below:



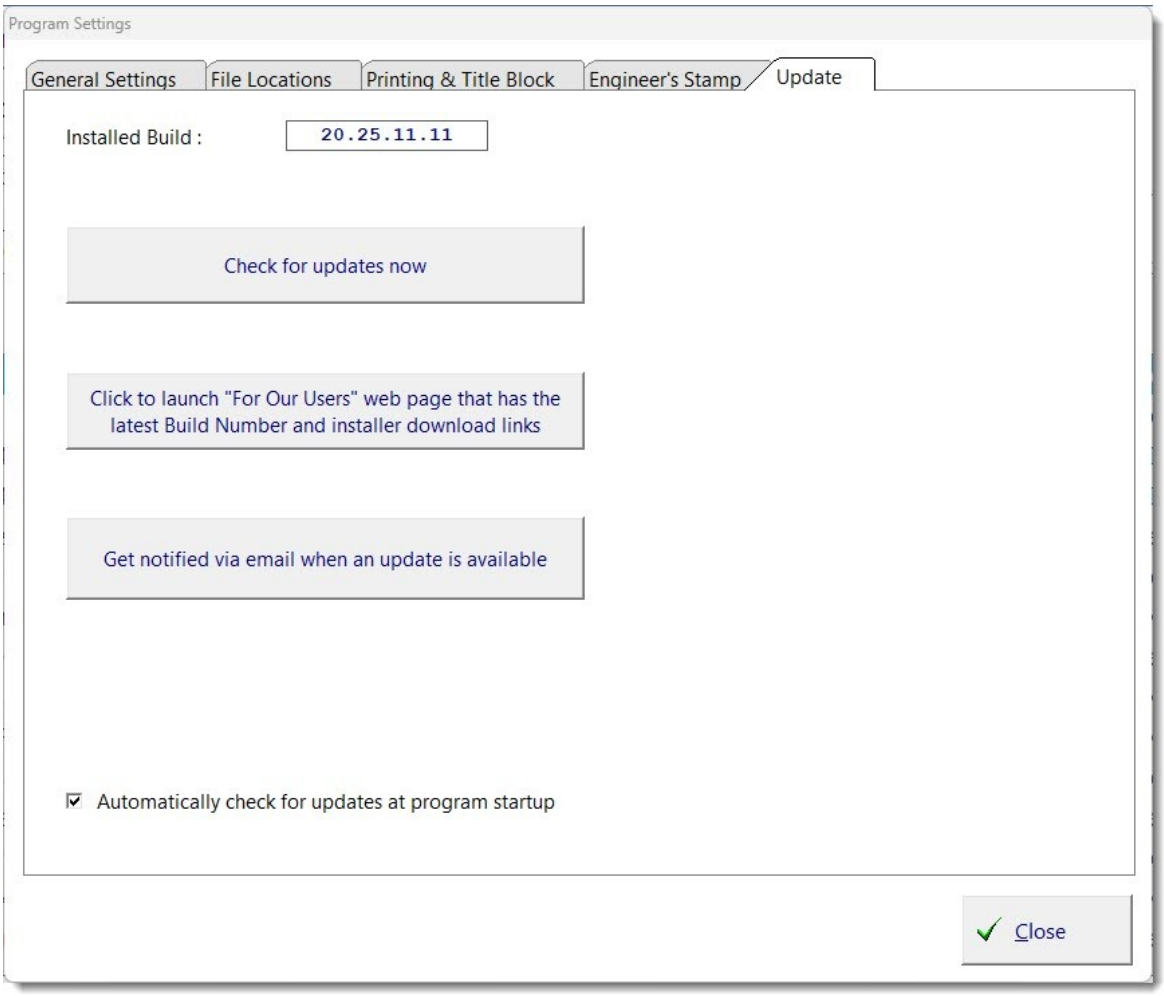
Select Printing Color Mode: This sets the default mode for printed reports created from **ENERCALC SEL**.

Professional Engineer's Stamp

This tab allows a graphic image of the user's PE stamp to be uploaded. Then the stamp can be placed on printout by selecting the checkbox.

Update

© 2026 ENERCALC, LLC



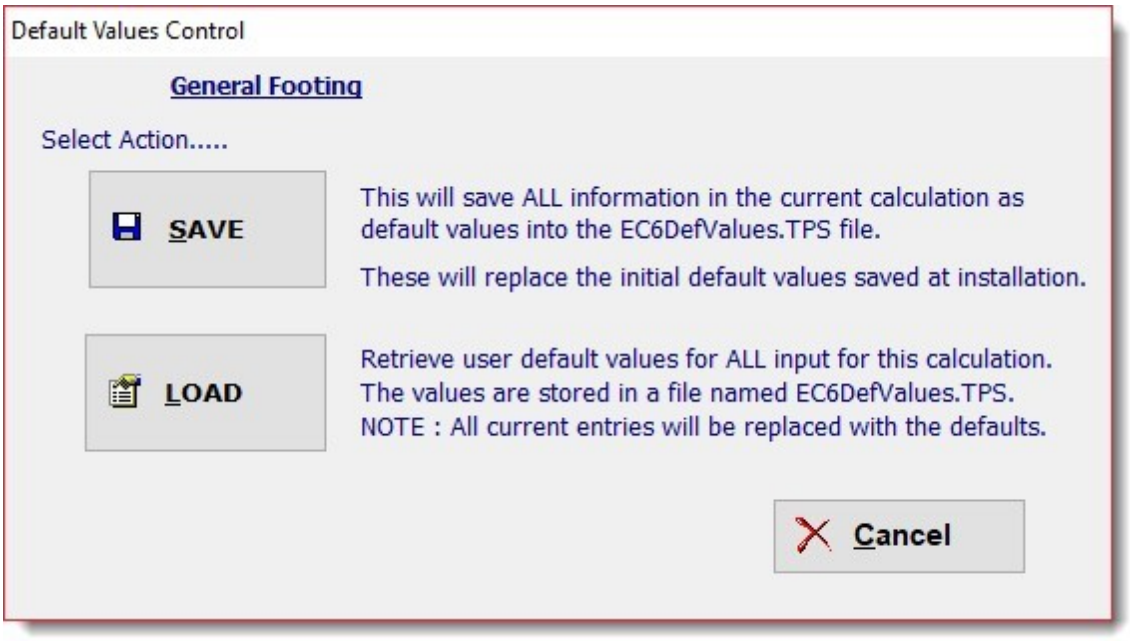
Click here to check for updates: This button will download the updating program from our server. This program will check and update the software to the latest version.

10.2.1 Default Values

Most of the user input values in the individual calculation modules have a default value or setting that is stored within the program. Many of these default values can easily be changed by the user. The procedure is as follows:

1. Open a new calculation of the particular module of interest.
2. Revise all of the initial values and settings as desired so that they represent the desired default values or settings.
3. Click Settings > Default Values > Save.

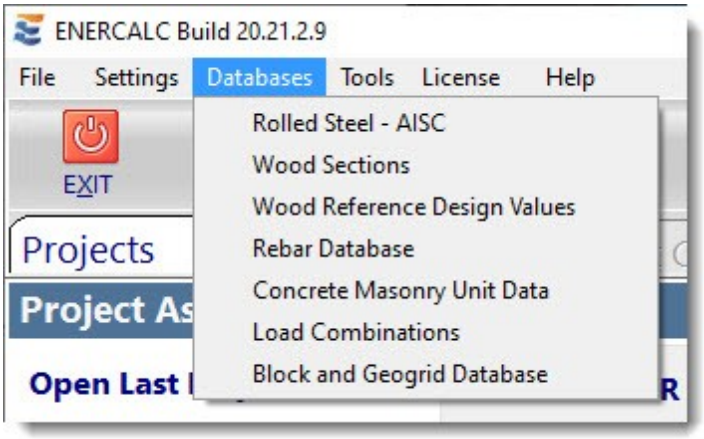
© 2026 ENERCALC, LLC



The current values will be saved as the new default values for that module, and these initial values and settings will be used when a new instance of this calculation module is added to any Project File.

10.3 Databases

This menu allows viewing of various databases supplied with the software.

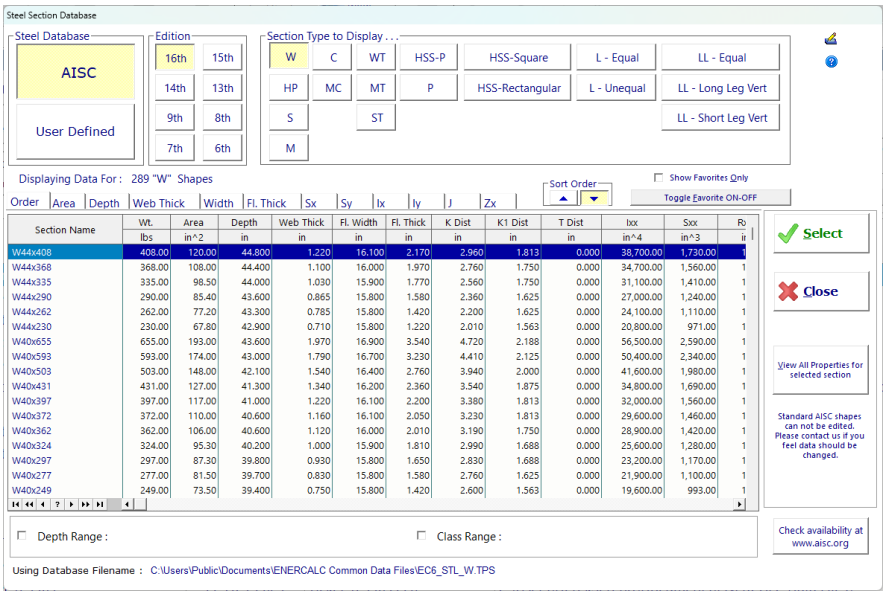


10.3.1 Rolled Steel - AISC Database

The steel section database is available in all applicable steel modules. It contains typical AISC rolled sections from many Editions of the AISC Steel Construction Manual.

© 2026 ENERCALC, LLC

Note that it is also possible to create User Defined entries that can be referenced in calculations.

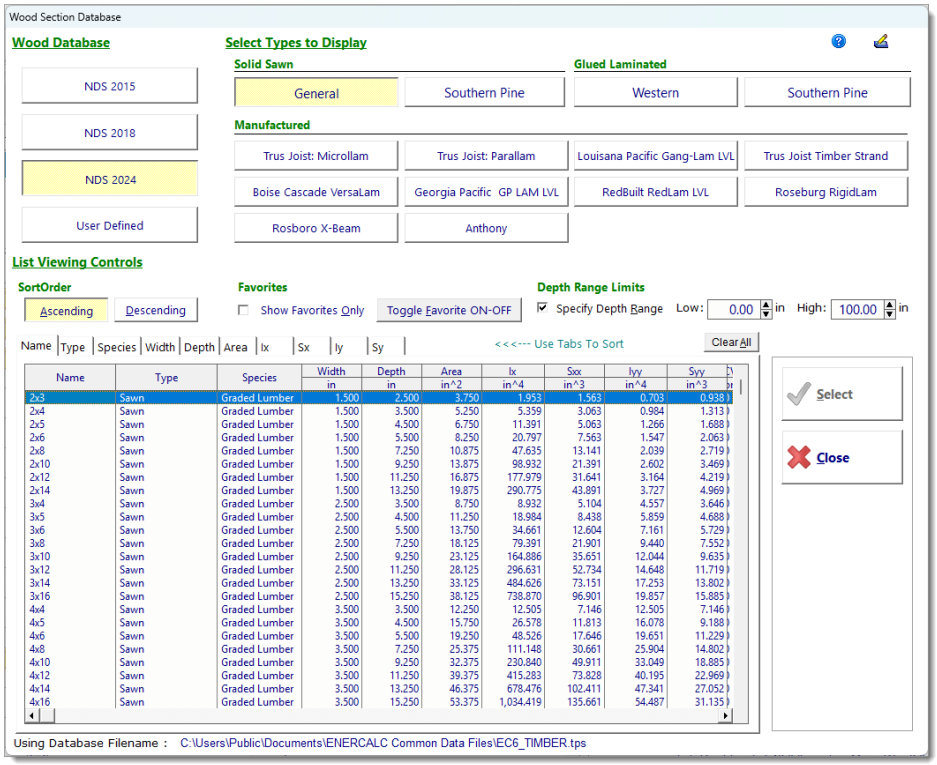


10.3.2 Wood Section Database

The wood section database contains the section properties for most of the sections listed in the NDS and for many Engineered Wood Products.

© 2026 ENERCALC, LLC

The wood section database is available in all applicable wood modules. It contains typical wood sections available in the United States. These sections are also shown in the NDS. Also included are manufactured sections, however the list of those sections is only updated occasionally. Note that it is also possible to create User Defined entries that can be referenced in calculations.



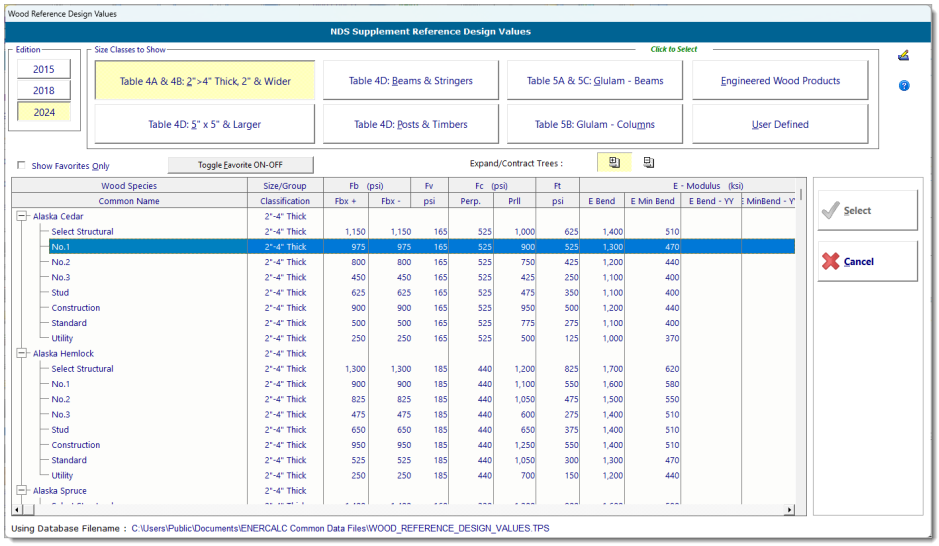
Name	Type	Species	Width in	Depth in	Area in^2	Ix in^4	Sxx in^3	Iyy in^4	Syy in^3
2x3	Sawn	Graded Lumber	1.500	2.500	3.750	1.953	1.563	0.703	0.938
2x4	Sawn	Graded Lumber	1.500	3.500	5.250	5.359	3.063	0.984	1.313
2x5	Sawn	Graded Lumber	1.500	4.500	6.750	11.391	5.063	1.266	1.688
2x6	Sawn	Graded Lumber	1.500	5.500	8.250	20.797	7.563	1.547	2.063
2x8	Sawn	Graded Lumber	1.500	7.250	10.875	47.635	13.141	2.039	2.719
2x10	Sawn	Graded Lumber	1.500	9.250	13.875	98.932	21.391	2.602	3.469
2x12	Sawn	Graded Lumber	1.500	11.250	16.875	177.979	31.641	3.164	4.219
2x14	Sawn	Graded Lumber	1.500	13.250	19.875	290.775	43.891	3.727	4.969
3x4	Sawn	Graded Lumber	2.500	3.500	8.750	8.932	5.104	4.557	3.646
3x5	Sawn	Graded Lumber	2.500	4.500	11.250	18.984	8.438	5.859	4.688
3x6	Sawn	Graded Lumber	2.500	5.500	13.750	34.661	12.604	7.161	5.729
3x8	Sawn	Graded Lumber	2.500	7.250	18.125	79.391	21.901	9.440	7.552
3x10	Sawn	Graded Lumber	2.500	9.250	23.125	164.886	35.651	12.044	9.635
3x12	Sawn	Graded Lumber	2.500	11.250	28.125	296.631	52.734	14.648	11.719
3x14	Sawn	Graded Lumber	2.500	13.250	33.125	484.626	73.151	17.253	13.802
3x16	Sawn	Graded Lumber	2.500	15.250	38.125	738.870	96.901	19.857	15.885
4x4	Sawn	Graded Lumber	3.500	3.500	12.250	12.505	7.146	12.505	7.146
4x5	Sawn	Graded Lumber	3.500	4.500	15.750	26.578	11.813	16.078	9.188
4x6	Sawn	Graded Lumber	3.500	5.500	19.250	48.526	17.646	19.651	11.229
4x8	Sawn	Graded Lumber	3.500	7.250	25.375	111.148	30.661	25.904	14.802
4x10	Sawn	Graded Lumber	3.500	9.250	32.375	230.840	49.911	33.049	18.885
4x12	Sawn	Graded Lumber	3.500	11.250	39.375	415.283	73.828	40.195	22.969
4x14	Sawn	Graded Lumber	3.500	13.250	46.375	678.476	102.411	47.341	27.052
4x16	Sawn	Graded Lumber	3.500	15.250	53.375	1,034.419	135.661	54.487	31.135

10.3.3 Wood Reference Design Values Database

The Wood Reference Design Values database is available in all applicable wood modules.

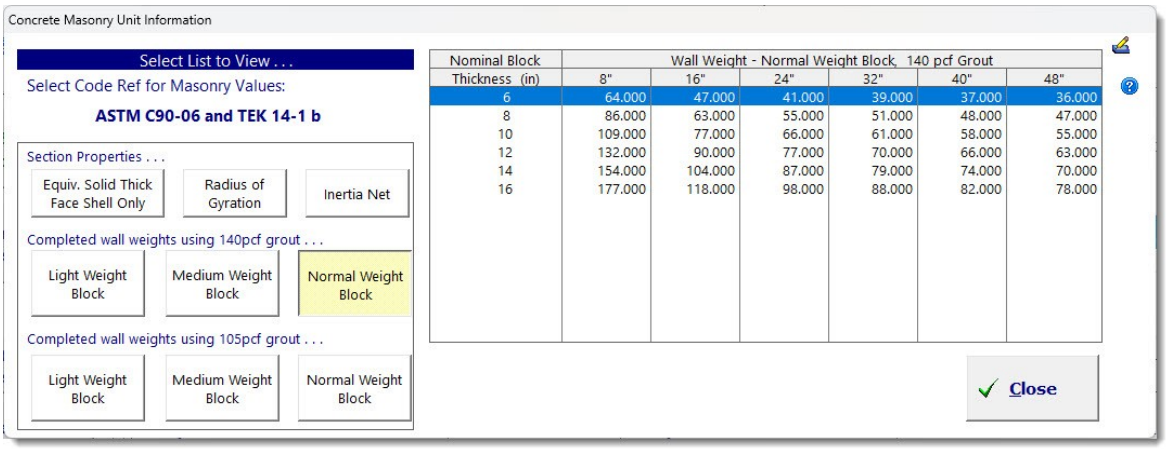
© 2026 ENERCALC, LLC

The Wood Reference Design Values database contains typical wood stress grades as defined in the NDS. Also included are manufactured sections, however the list of those sections is only updated occasionally. Note that it is also possible to create User Defined entries that can be referenced in calculations.



10.3.4 Masonry Database

This is a reference table only. It consists of data for common hollow concrete masonry units:

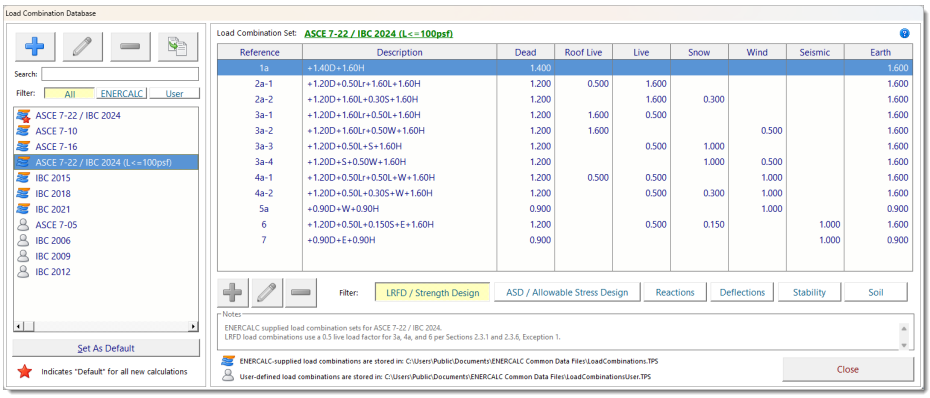


10.3.5 Load Combination Database

This database stores named load combination sets that can be automatically retrieved and applied within calculations.

You can assign a named load combination set as the default, which will be used whenever a new calculation is created in a Project File. You may also create new sets or edit existing ones as needed.

© 2026 ENERCALC, LLC

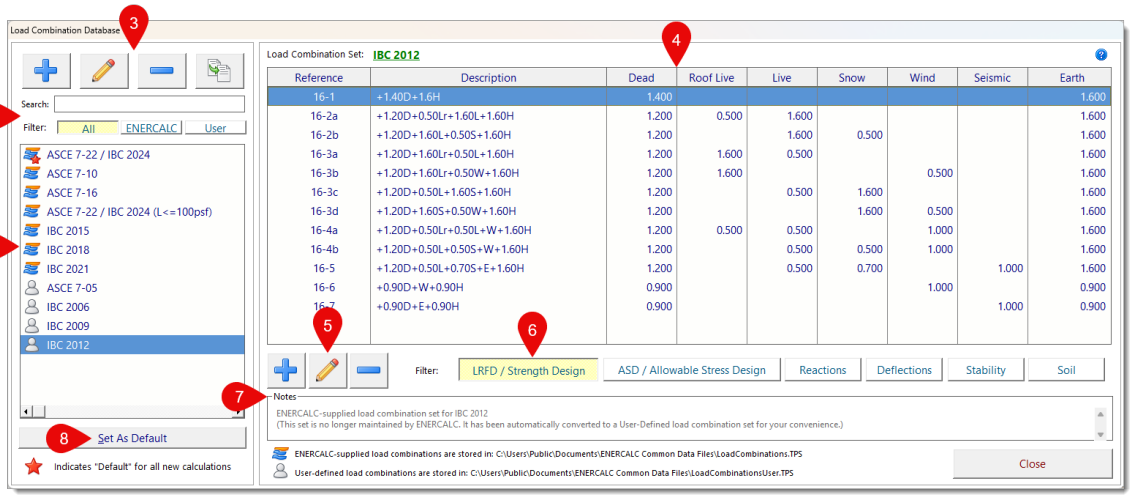


The Load Combination Database is shared across all Project Files, and the sets defined here are not specific to any individual Project File.

> **NOTE:** ENERCALC-supplied load combination sets are read-only and stored in the file `LoadCombinations.tps`, while custom user-defined sets are stored in `LoadCombinationsUser.tps`. Both files are typically located in `C:\Users\Public\Documents\ENERCALC Common Data Files\`.

Within each set, you can define both **LRFD/Strength Design** (factored) and **ASD/Allowable Stress Design** (service) load cases, as well as load combinations for **Reactions**, **Deflections**, **Stability**, and **Soil** checks.

Please see the descriptions for the numbered keynotes on the screen capture below:



(1) Load Combination Set List: This area displays all available load combination sets. ENERCALC-supplied sets are marked with the symbol, and user-defined sets are marked with the symbol.

© 2026 ENERCALC, LLC

(2) Search and Filter Controls: Use these controls to search for a specific load combination set or filter the list by All, ENERCALC-supplied, or User-defined sets.

(3) Add, Edit, Delete, and Copy Load Combination Sets: These buttons allow you to manage load combination sets:

Add creates a new user-defined set. The *Add* function always creates a user-defined set.

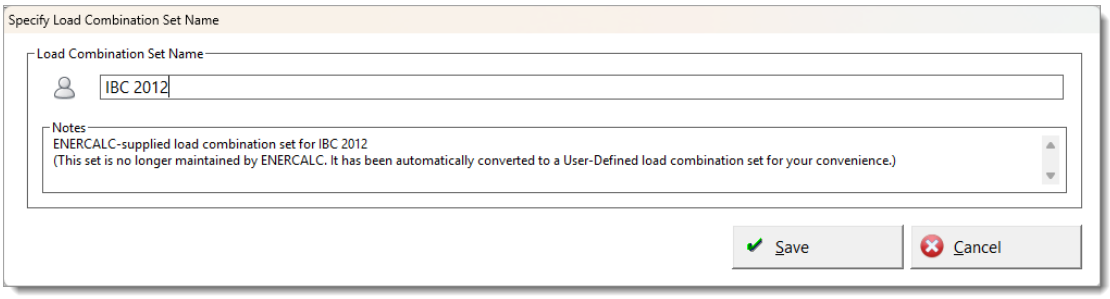
Edit allows you to modify the name and notes of the selected set.

 Delete removes a user-defined set.

Copy duplicates any set, whether ENERCALC-supplied or user-defined, into a new user-defined set. The *Copy* function always creates a user-defined set.

NOTE: *Edit* and *Delete* are only available for user-defined sets. These actions are disabled when an ENERCALC-supplied set is selected.

When you click **Add** or **Edit**, the following dialog appears. Use this dialog to define the load combination set name and notes.



(4) Load Combination Table: This area displays the individual load combinations and their corresponding load factors for each load type.

(5) Add, Edit, and Delete Load Combinations: These buttons allow you to manage the individual load combinations contained within the load combination set:

Add creates a new load combination.

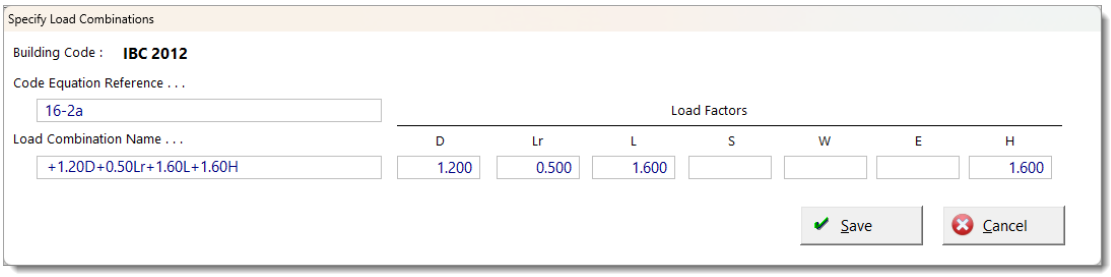
Edit opens a dialog to modify the code reference and numeric load factors for the selected combination.

 Delete removes the selected combination.

Note: *Edit* and *Delete* are only available for user-defined load combinations. ENERCALC-supplied combinations cannot be modified. These buttons are disabled when an ENERCALC load combination set is active.

© 2026 ENERCALC, LLC

When you click **Add** or **Edit**, the following dialog appears. Use this dialog to enter a code reference and assign values for each load factor in the combination.



(6) Load Combination Type Selector: Click these buttons to choose which category of load combinations to display, such as **LRFD**, **ASD**, **Reactions**, **Deflections**, **Stability**, and **Soil**.

(7) Notes Field: This area displays any notes associated with the currently selected load combination set. To edit these notes, use the load combination set **Edit** button in the top-left corner of the window (see keynote (3) 80).

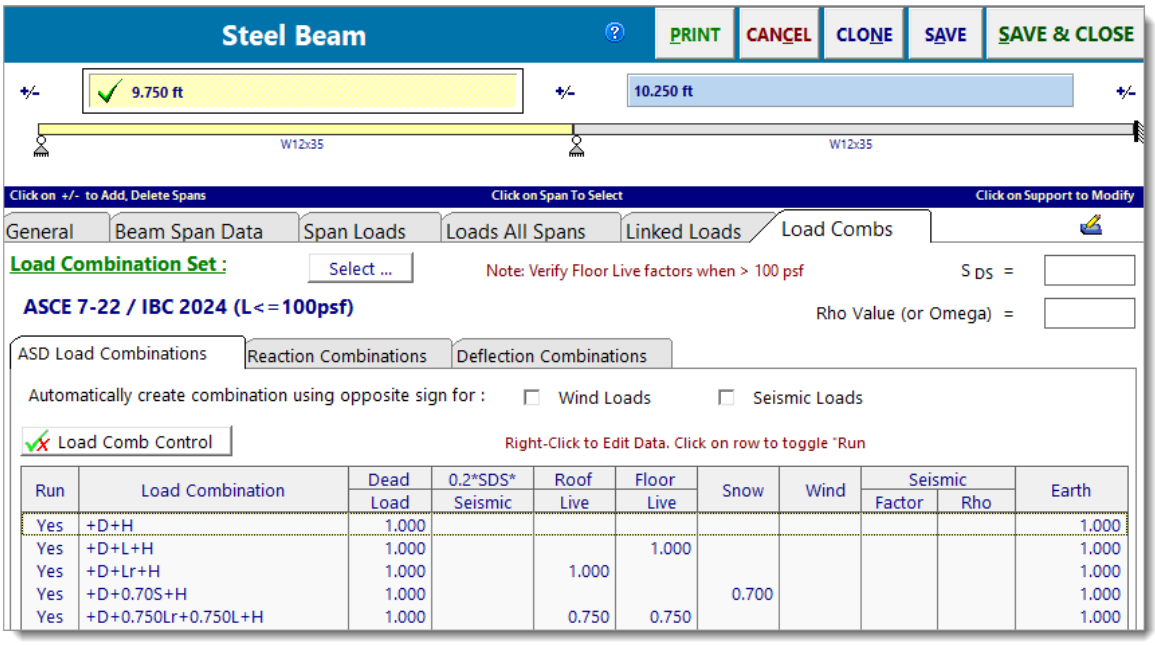
(8) Set As Default: Click this button to designate the selected load combination set as the default for all new ENERCALC calculations. The current default is indicated with the symbol.

Using Load Combinations

When you create a new calculation in the Project Manager, the default load combination set is automatically loaded into the Load Combinations tab of the selected module.

In the example below, the Load Combinations tab in the Steel Beam module displays the combinations that were loaded.

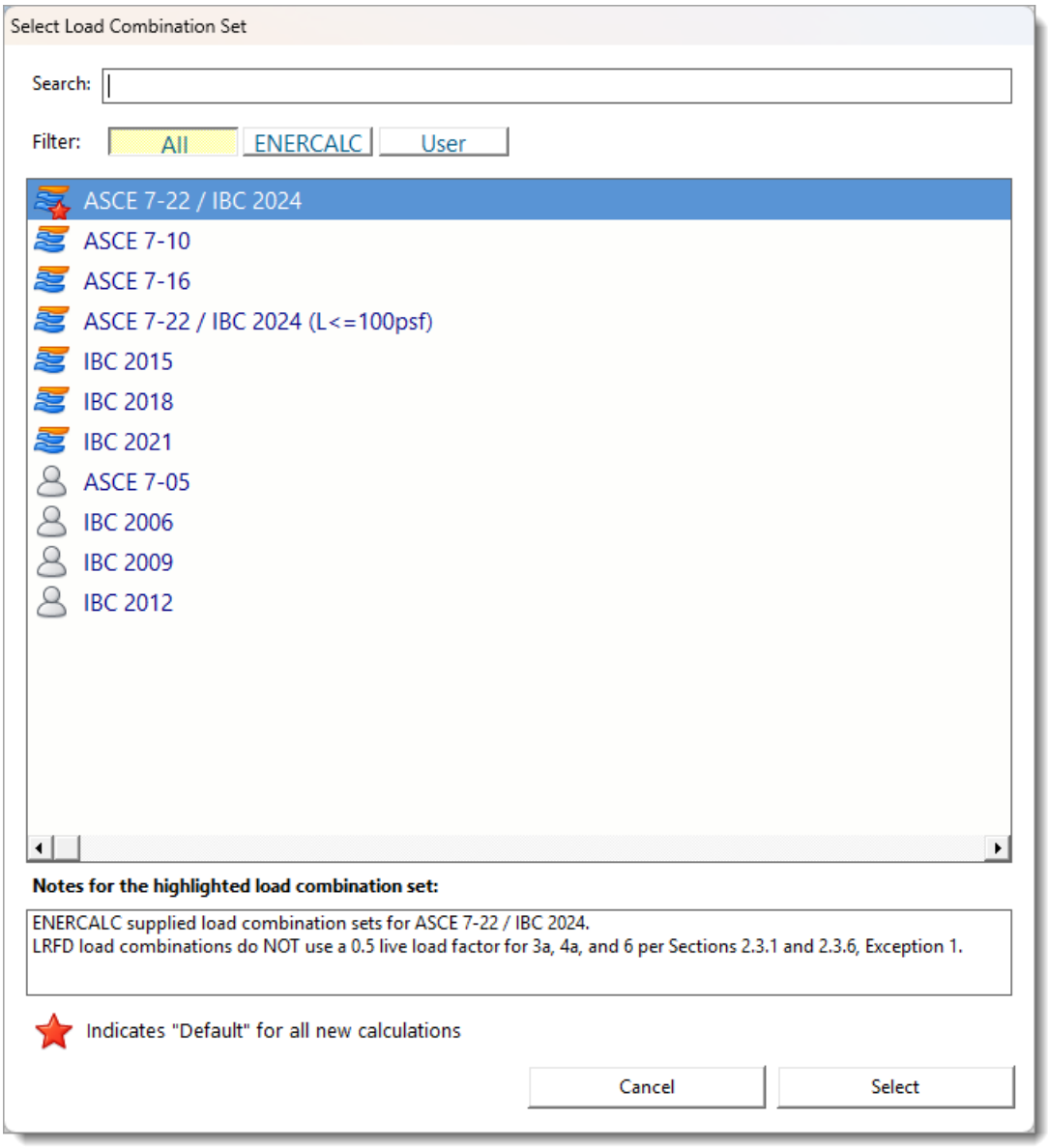
© 2026 ENERCALC, LLC



Because the design method is set to **ASD**, the load combinations tab is showing **allowable stress design** combinations.

Clicking the **Select** button opens a pop-up window showing all available load combination sets from the Load Combination Database.

© 2026 ENERCALC, LLC

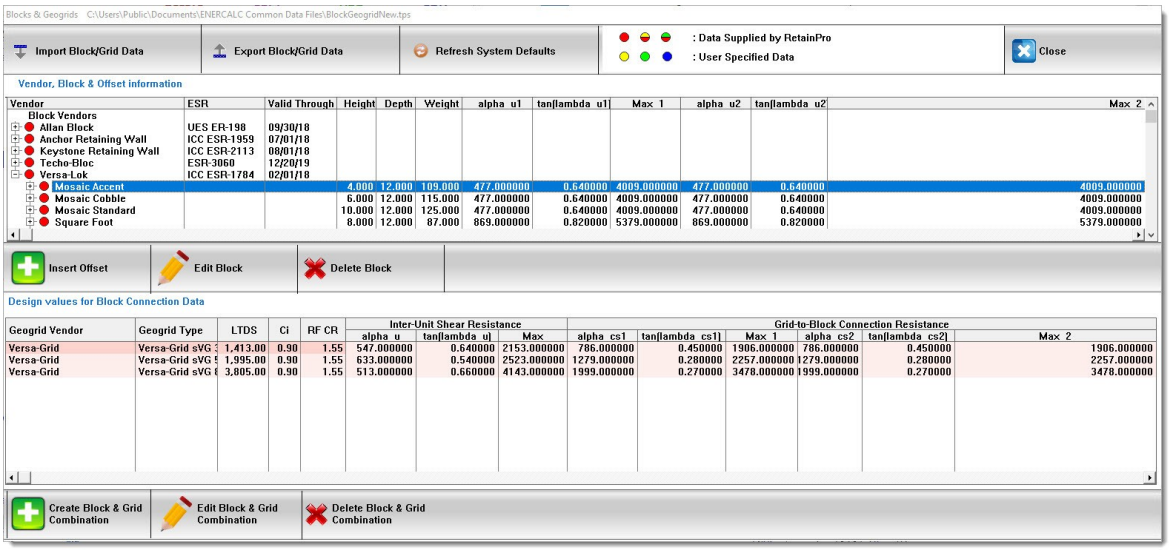


Choosing a different set from this list will load that set into the current calculation only. It does not change the global default.

© 2026 ENERCALC, LLC

10.3.6 Block & Geogrid Database

The Block & Geogrid Database is used to pair block data with geogrid data, to create viable design combinations in the geogrid reinforced Segmental Retaining Wall module. The editor allows the user to add, edit, or delete block vendors, blocks, block offset values, and then add, edit, or delete geogrid design data for use with each type of block.



Overview: The editor shows a list of block vendors on the top of the screen. Expanding any of the block vendors will reveal the individual block types offered by that vendor. Expanding any of the individual block types will reveal the list of applicable offset values that can be used for a wall assembled with that block type. Clicking on any individual block type will display the list of geogrids that have published design values for use with the selected block. Any combination of block and geogrid that is visible in the editor will be available for selection in the geogrid reinforced Segmental Retaining Wall module.

The functions of the various tools and buttons on the Block & Geogrid Editor are described below:

Import Block/Grid Data: Allows the user to import a file (such as from a colleague) that contains all of the block and grid data from another installation.

Export Block/Grid Data: Allows the user to export a file that contains all of the block and grid data from the current installation (such as to share with a colleague).

Refresh System Defaults: Allows the user to reset the design values for block and grid to the initial values as they were when the software was originally installed. This function is safe to use even if the user has created any custom block/grid definitions after installation, because the system automatically differentiates between system-installed data and custom user data. This function will ***not*** alter or remove your custom user data if any has been entered.

© 2026 ENERCALC, LLC

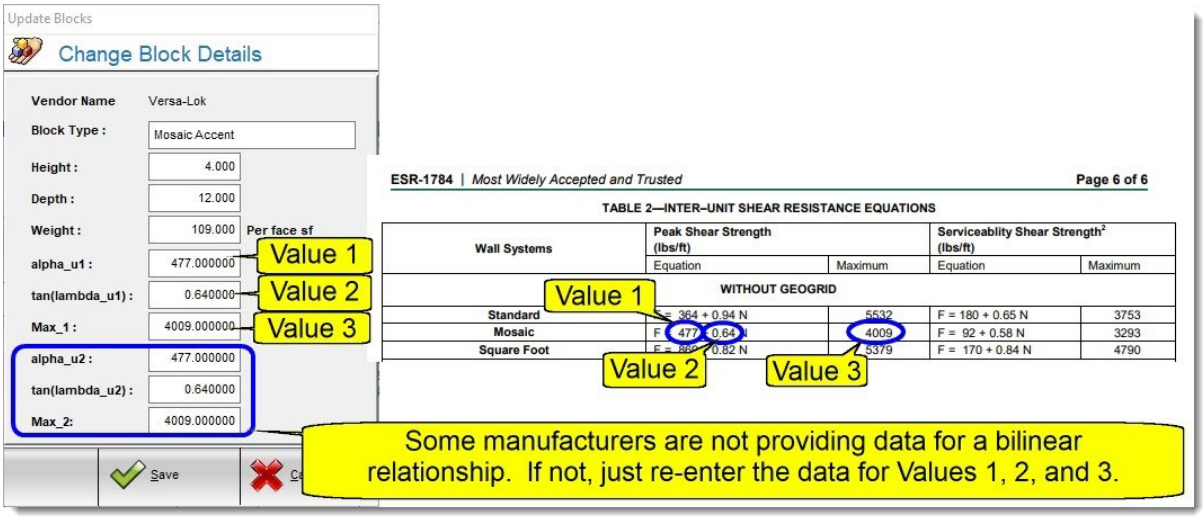
Insert Vendor/Insert Block/Insert Offset: The function of this button changes depending upon what is selected in the top pane.

- When the focus is on the label "Block Vendors" at the top of the tree, this button takes the form of Insert Vendor, and it allows a new block vendor to be added to the tree structure.
- When the focus is on any one of the block vendors in the tree, this button takes the form of Insert Block, and it allows a new block type to be added to the selected vendor.
- When the focus is on any one of the blocks in the tree, this button takes the form of Insert Offset, and it allows a new offset value to be added to the selected block.

Edit Vendor/Edit Block/Edit Offset: The function of this button changes depending upon what is selected in the top pane.

- When the focus is on any one of the block vendors in the tree, this button takes the form of Edit Vendor, and it allows the selected vendor name and website to be edited.
- When the focus is on any one of the blocks in the tree, this button takes the form of Edit Block, and it allows the selected block properties to be edited.
- When the focus is on one of the offset values, this button takes the form of Edit Offset, and it allows the selected offset value to be edited.

When editing block definitions, the program presents the following dialog:



Delete Vendor/Delete Block/Delete Offset: The function of this button changes depending upon what is selected in the top pane.

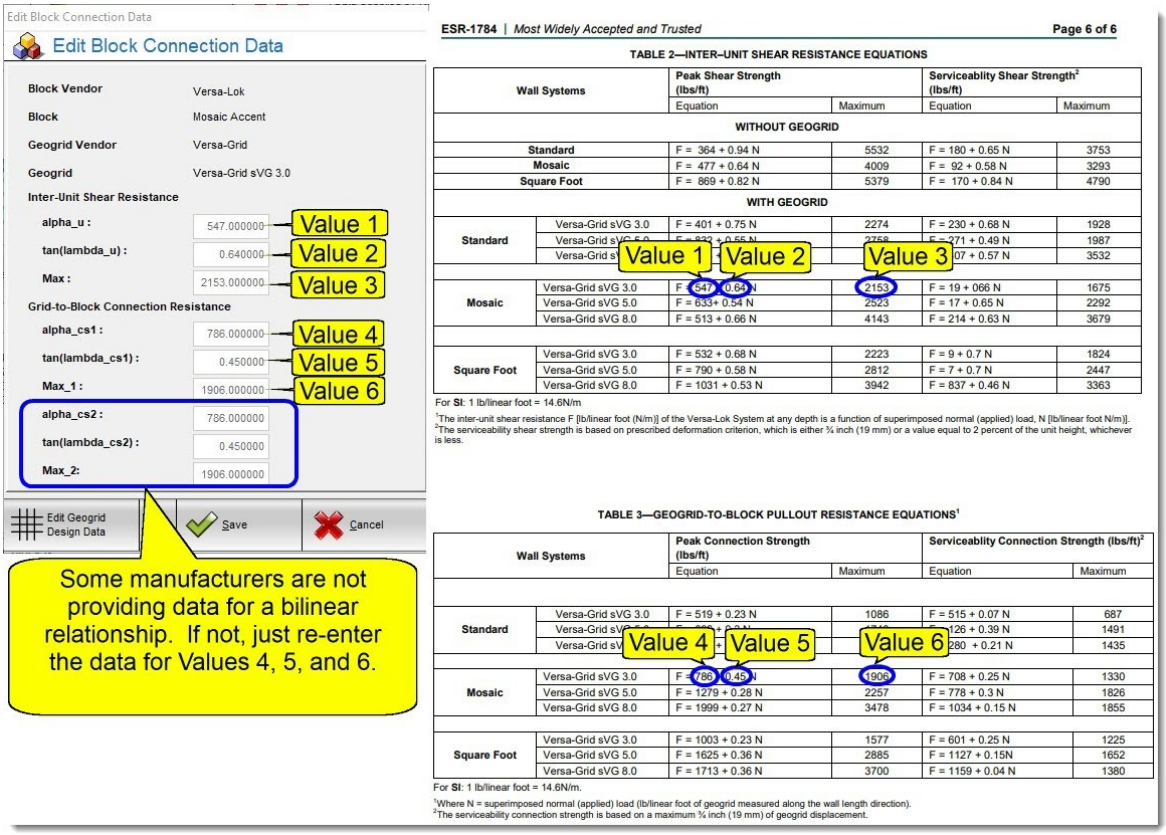
- When the focus is on any one of the block vendors in the tree, this button takes the form of Delete Vendor, and it allows the selected vendor to be deleted.
- When the focus is on any one of the blocks in the tree, this button takes the form of Delete Block, and it allows the selected block to be deleted.

© 2026 ENERCALC, LLC

- When the focus is on one of the offset values, this button takes the form of Delete Offset, and it allows the selected offset value to be deleted.

Create Block & Grid Combination: When the focus on the top of the screen is on any one of the blocks in the tree, this button allows the user to define a new geogrid/block combination by specifying geogrid vendor, type, and design properties for use with the selected block. It also provides access to the Edit Geogrid Design Data button that allows the user to edit LTDS, Ci and RF_{CR} values for the selected geogrid (see next item).

Edit Block & Grid Combination: When the focus on the top of the screen is on any one of the blocks in the tree, this button allows the user to edit the connection design properties for use with the selected block/geogrid combination. When editing Block Connection Data for the selected block/geogrid combination, the program presents the following dialog:



Wall Systems		Peak Shear Strength (lbs/ft) Equation	Maximum	Serviceablity Shear Strength[2] (lbs/ft) Equation	Maximum
WITHOUT GEOGRID					
Standard		F = 364 + 0.94 N	5532	F = 180 + 0.65 N	3753
Mosaic		F = 477 + 0.64 N	4009	F = 92 + 0.58 N	3293
Square Foot		F = 869 + 0.82 N	5379	F = 170 + 0.84 N	4790
WITH GEOGRID					
Standard	Versa-Grid sVG 3.0	F = 401 + 0.75 N	2274	F = 230 + 0.68 N	1928
	Versa-Grid sVG 5.0	F = 832 + 0.55 N	2758	F = 271 + 0.49 N	1987
	Versa-Grid sVG 8.0	F = 1067 + 0.57 N	3532	F = 307 + 0.57 N	3532
Mosaic	Versa-Grid sVG 3.0	F = 547 + 0.64 N	2153	F = 19 + 066 N	1675
	Versa-Grid sVG 5.0	F = 633+ 0.54 N	2523	F = 17 + 0.65 N	2292
	Versa-Grid sVG 8.0	F = 513 + 0.66 N	4143	F = 214 + 0.63 N	3679
Square Foot	Versa-Grid sVG 3.0	F = 532 + 0.68 N	2223	F = 9 + 0.7 N	1824
	Versa-Grid sVG 5.0	F = 790 + 0.58 N	2812	F = 7 + 0.7 N	2447
	Versa-Grid sVG 8.0	F = 1031 + 0.53 N	3942	F = 837 + 0.46 N	3363

For SI: 1 lb/linear foot = 14.6N/m

[1]The inter-unit shear resistance F [lb/linear foot (N/m)] of the Versa-Lok System at any depth is a function of superimposed normal (applied) load, N [lb/linear foot N/m)].
[2]The serviceability shear strength is based on prescribed deformation criterion, which is either ¾ inch (19 mm) or a value equal to 2 percent of the unit height, whichever is less.

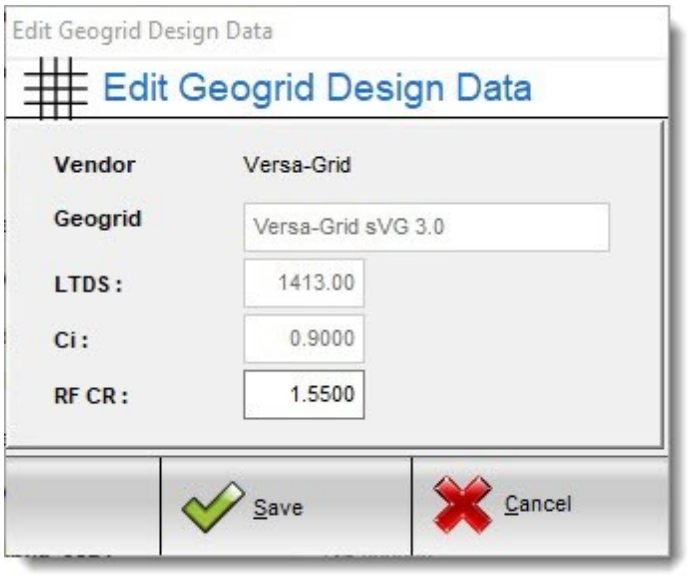
Wall Systems		Peak Connection Strength (lbs/ft) Equation	Maximum	Serviceablity Connection Strength (lbs/ft)[2] Equation	Maximum
Standard	Versa-Grid sVG 3.0	F = 519 + 0.23 N	1086	F = 515 + 0.07 N	687
	Versa-Grid sVG 5.0	F = 1126 + 0.39 N	1491	F = 1126 + 0.39 N	1491
	Versa-Grid sVG 8.0	F = 1280 + 0.21 N	1435	F = 1280 + 0.21 N	1435
Mosaic	Versa-Grid sVG 3.0	F = 786 + 0.45 N	1906	F = 708 + 0.25 N	1330
	Versa-Grid sVG 5.0	F = 1279 + 0.28 N	2257	F = 778 + 0.3 N	1826
	Versa-Grid sVG 8.0	F = 1999 + 0.27 N	3478	F = 1034 + 0.15 N	1855
Square Foot	Versa-Grid sVG 3.0	F = 1003 + 0.23 N	1577	F = 601 + 0.25 N	1225
	Versa-Grid sVG 5.0	F = 1625 + 0.36 N	2885	F = 1127 + 0.15N	1652
	Versa-Grid sVG 8.0	F = 1713 + 0.36 N	3700	F = 1159 + 0.04 N	1380

For SI: 1 lb/linear foot = 14.6N/m.

[1]Where N = superimposed normal (applied) load (lb/linear foot of geogrid measured along the wall length direction).
[2]The serviceability connection strength is based on a maximum ¾ inch (19 mm) of geogrid displacement.

© 2026 ENERCALC, LLC

This dialog also provides access to the Edit Geogrid Design Data button that allows the user to edit the LTDS, Ci and RF_{CR} values for the selected geogrid. When editing Geogrid Design Data for the selected geogrid, the program presents the following dialog:



Delete Block & Grid Combination: When the focus on the top of the screen is on any one of the blocks in the tree, this button allows the user to delete the selected block/geogrid design combination. (Note: This does not delete the selected block from the block database, nor does it delete the selected geogrid from the geogrid database. It merely removes the *combination* of the selected block and grid from the list of available assemblies to choose from.)

Disclaimer: The design values for block/grid combinations have a direct and significant influence on the results produced in the Segmental Retaining Wall module. The licensed practicing professional user must therefore exercise due caution when editing values in the data table and when incorporating this data into subsequent calculations. We strongly recommend that all block & geogrid design values be obtained from test reports that are acceptable to the code enforcement agency where the wall will be constructed. The design values in this program are supplied as a convenience to the user and must be verified with the manufacturers, test reports and code agencies.

Inserting Custom Block and Geogrid:

The procedure to add a new block and geogrid is as follows:

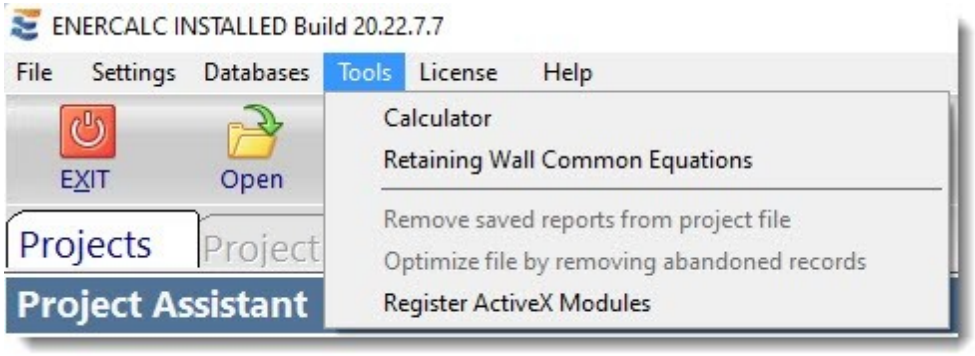
1. Click Databases > Block & Geogrid Database.
2. Click Insert Vendor and add a vendor for the new block.

© 2026 ENERCALC, LLC

3. Click on the new block vendor and click Insert Block.
4. Enter the new block data and click the Save button.
5. Select the new block and click Insert Offset.
6. Enter an offset value and click the Save button.
7. Click on the block again and click Create Block & Grid Combination.
8. Click the ... button next to Geogrid Vendor Name and either select an existing geogrid vendor or click Insert button and enter a new geogrid vendor name.
9. Click the ... button next to Geogrid Type and either select an existing geogrid or click the Add button, and enter the name and data for the new geogrid.
10. Click the Select button.
11. Enter the necessary values.
12. Click the Save button.
13. The new combination of block and grid with connection data appears in the list on the bottom, and is available for selection in the Segmental Retaining Wall module.

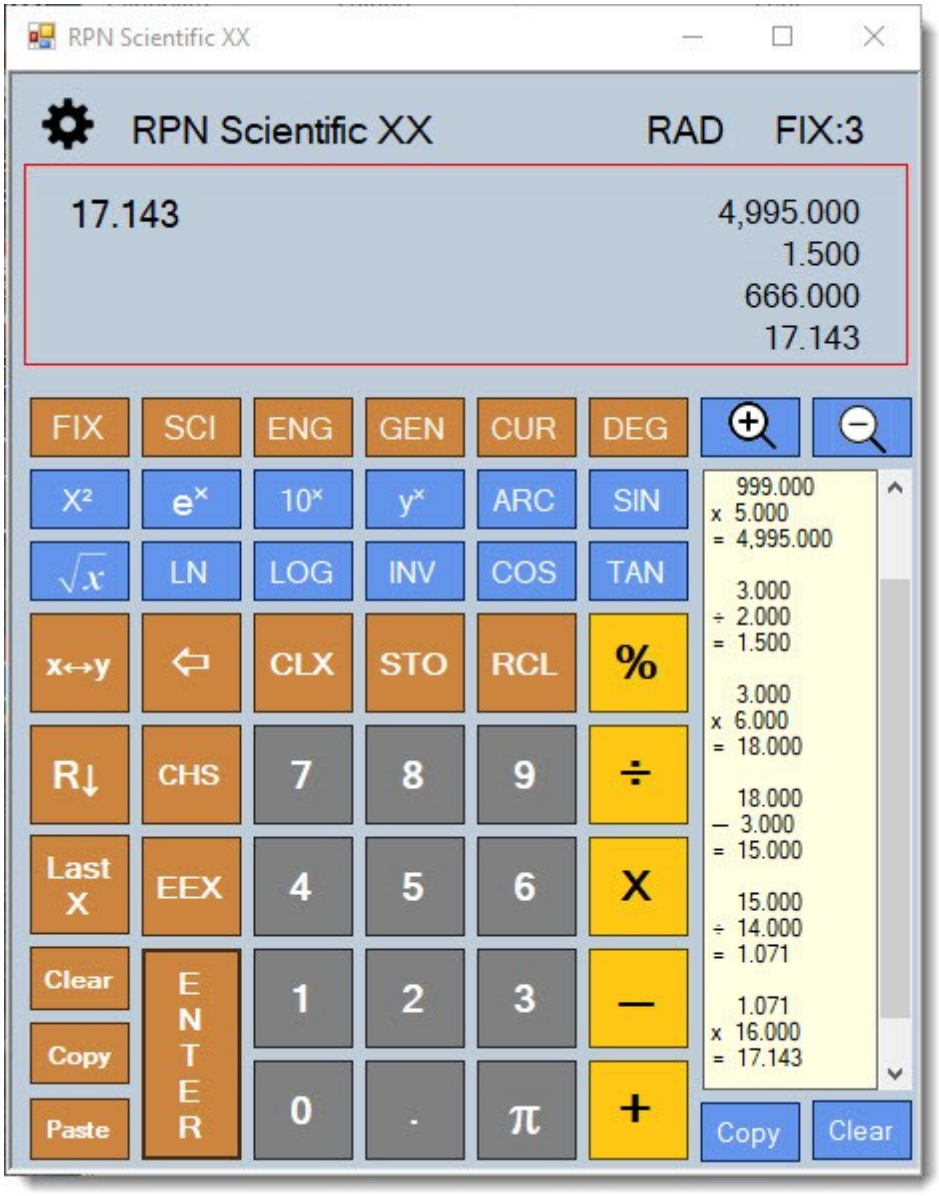
10.4 Tools

This menu item provides access to useful tools that are supplied with **ENERCALC SEL**.



© 2026 ENERCALC, LLC

Calculator: We supply a **Reverse Polish Notation Calculator** (RPN) for engineers to use for intermediate calculations when entering data. RPN calculators are the most commonly used among engineers because they allow faster mathematics solutions when nested (parenthetical) calculations are used. It also includes a Copy button, which allows a result value to be copied to the clipboard and then pasted into an input field in ENERCALC SEL.



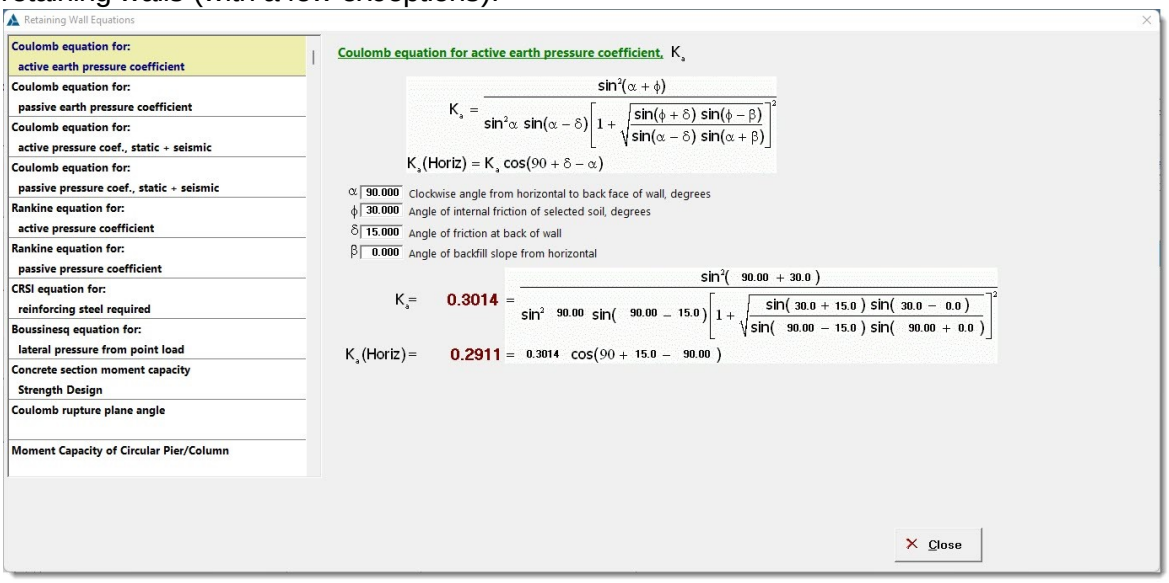
For more information on Reverse Polish Notation try these links:

http://en.wikipedia.org/wiki/Reverse_Polish_notation

http://www.hpmuseum.org/rpn.htm

© 2026 ENERCALC, LLC

Retaining Wall Common Equations: The Retaining Wall Equations Editor provides an easy way to plug in parameters and get quick results for equations that generally relate to retaining walls (with a few exceptions).



Remove Saved Reports from Project File: This command removes any reports that have been saved in the current Project File. It can be helpful in terms of reducing file size if a Project File is to be emailed, but it will require that the reports be regenerated before they can be printed.

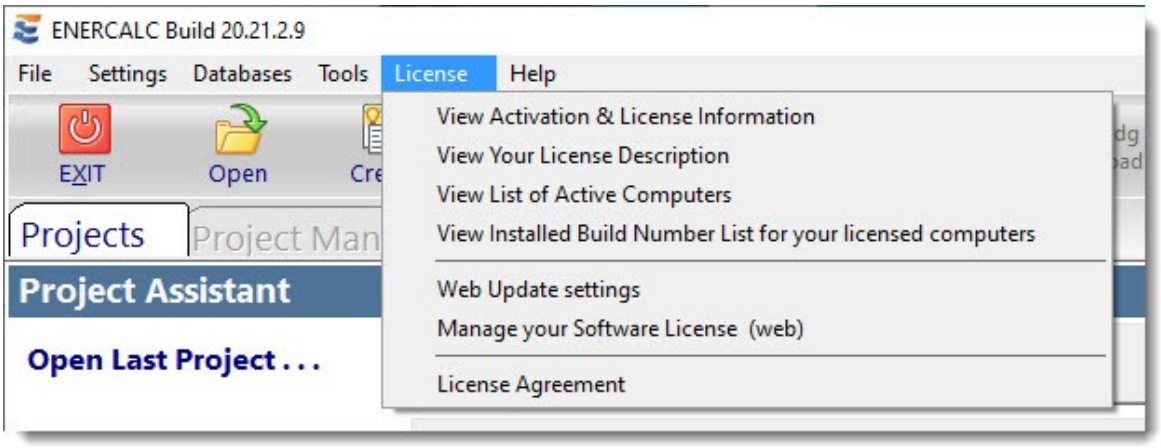
Optimize file by removing abandoned records: This command removes lost records in a Project File which could occur if the program is terminated abnormally (perhaps from a power outage, lockup, network failure, lockup of another program, etc).

Register ActiveX Modules: This selection provides a process of registering components used by SEL with the operating system.

© 2026 ENERCALC, LLC

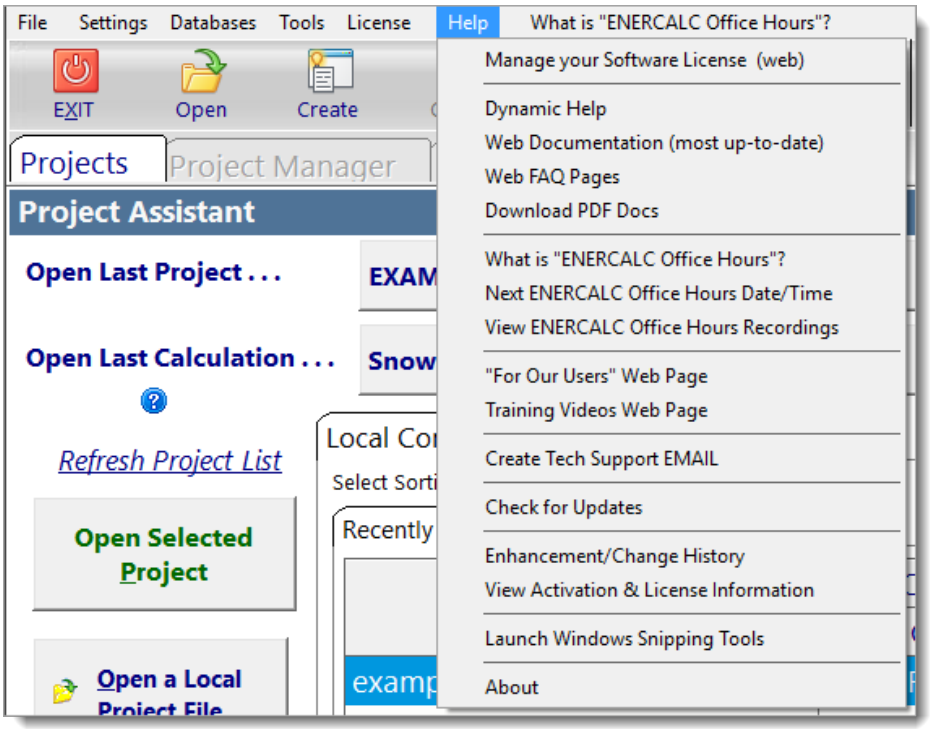
10.5 License

The License item displays the licensing and activation menu.



10.6 Help

The Help item displays the Help menu.

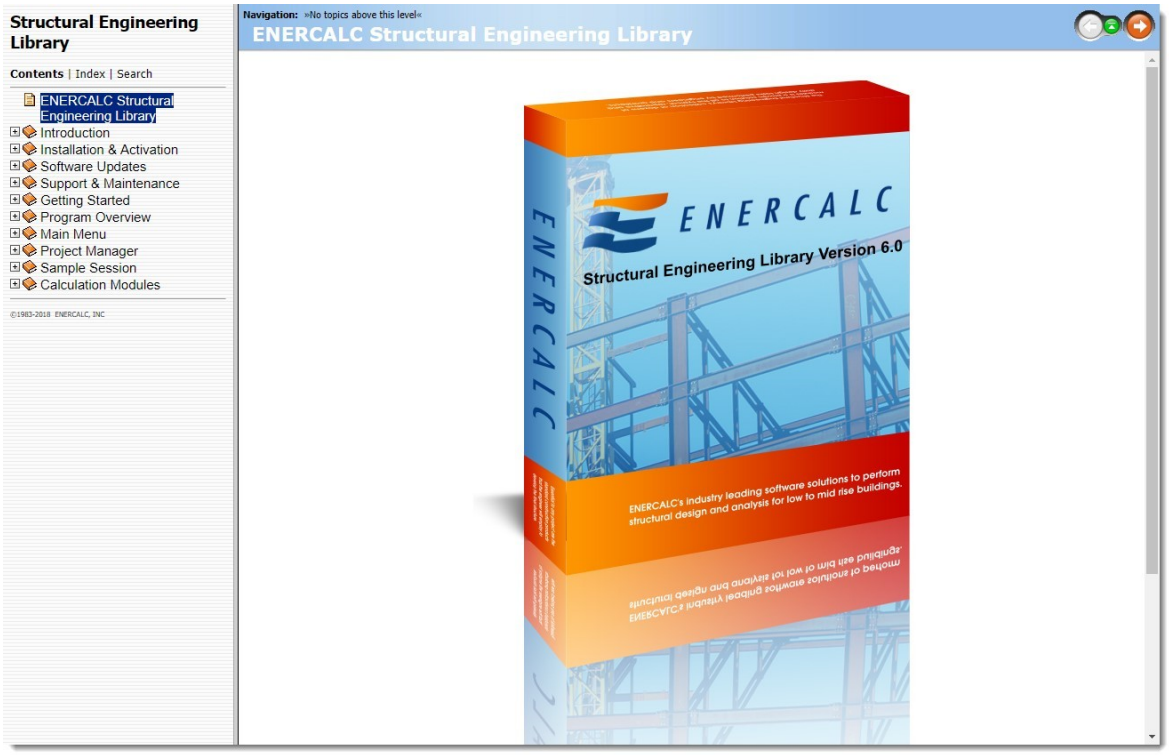


Manage your Software License (web): Automatically opens the Manage License page of our website.

© 2026 ENERCALC, LLC

Dynamic Help: Displays the help system for the software that is installed on your computer.

Web Documentation (most up-to-date): Displays the most up-to-date help information located on our website.



Web FAQ Pages: Provides a direct link to the **Frequently Asked Questions** section at www.enercalc.com.

Download PDF Docs: Downloads the PDF version of the documentation for the software currently available at our website. The documentation on our website will always be for the latest available version.

What is "ENERCALC Office Hours"?: Provides a direct link to the **ENERCALC Office Hours** section at www.enercalc.com.

Next ENERCALC Office Hours Date/Time: Provides a direct link to the next Office Hours countdown timer on the ENERCALC Office Hours page.

View ENERCALC Office Hours Recordings: Provides a direct link to the Office Hours recordings archive on the ENERCALC Office Hours page.

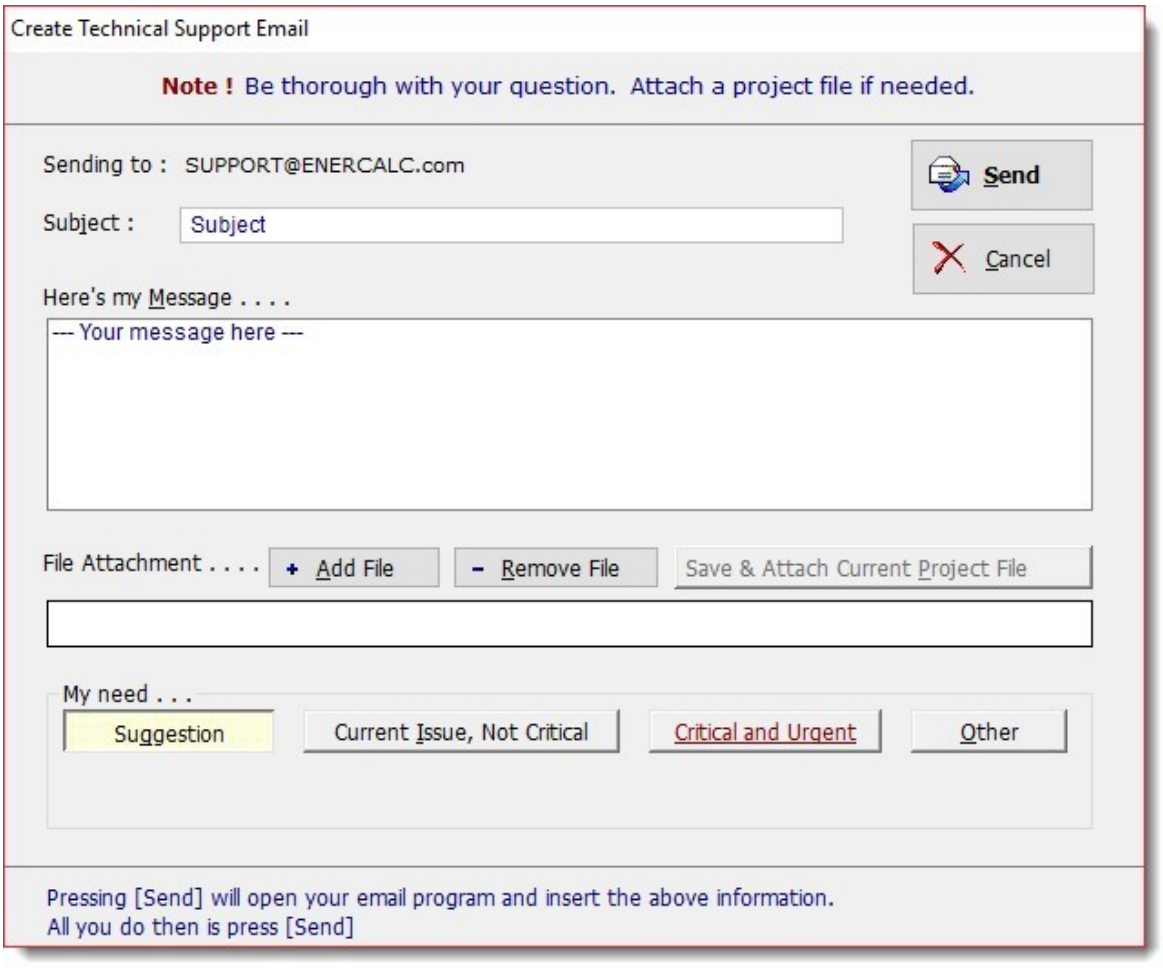
"For Our Users" Web Page: Provides a link directly to the "For Our Users" page of our website, which offers information on current builds, Customer Information, Software Installation & Update Links, Installation Info & FAQ links, info on Maintenance Plans &

© 2026 ENERCALC, LLC

Upgrades, Documentation Links, Links to PDF Files of Relevant Information and Useful Forms, and links for Prior Version Software Reinstallation.

Training Videos Web Page: Provides a link directly to the "Training Videos" page of our website, which offers videos on topics such as Installation, General Operation, Using Project the Manager, Designing Beams, Columns, Foundations, Slender Walls, working with some of the Miscellaneous modules, and working with External Files.

Create Tech Support Email: Provides you with a Tech Support form to fill out. When finished, the data is transferred to an email form to send to our Tech Support Group. (Note: Some computers and email software will not be able to paste the information into your email program. This is not an ENERCALC issue. In those situations, feel free to compose an email to support@enercalc.com directly from your email program, and please remember to indicate your "KW" User Registration number on all correspondence.)



Check for Updates: Immediately runs the update check program named EC6WebUpdate.exe, which resides in the ENERCALC program installation folder. If a newer build is found, a window will be displayed prompting you to decide if you want to continue to update your software. If there is no newer build of the software available, then you will be informed that your installation is currently up-to-date.

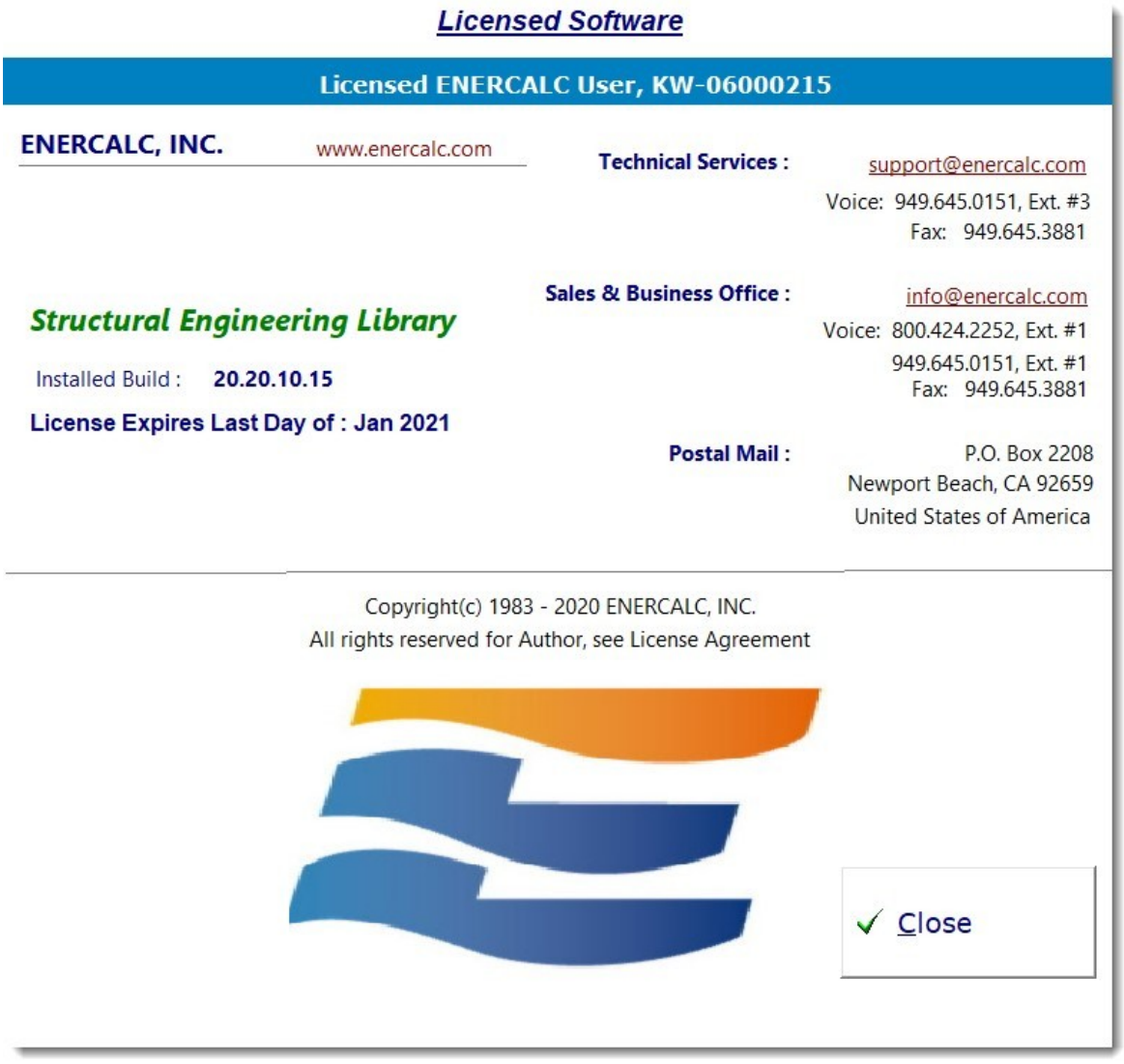
© 2026 ENERCALC, LLC

Enhancement/Change History: Provides a direct link to the full change log and release notes at https://updates.enercalc.com/.

View Activation & License Information: Another way to access user information.

Launch Windows Snipping Tools: Opens the Windows Snipping Tool, allowing you to capture and share program behavior when submitting a technical support inquiry.

About: Displays a general window giving information about software version, copyright, licensed user, activation status, and contact information for ENERCALC.



Part

11 Project Manager

The Project Manager is displayed when a Project File has been opened.

The Project Manger provides the ability to create and modify a single Project File that can contain a set of calculations and external items for a specific project. The layout, which will be ever-improving, is designed to allow a Project File to be a single collection point for documents relating to the load development and structural design of a building or other structure.

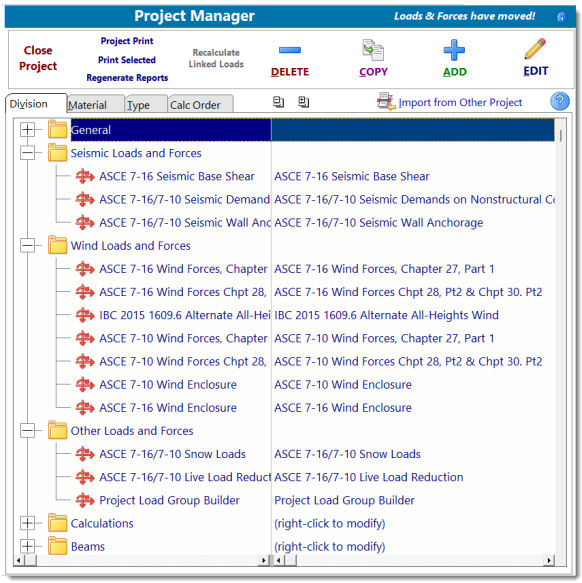
Components that all Project Files AUTOMATICALLY contain are:

GENERAL Division (Only one can exist in a Project File)
A storage location for pieces of information pertaining to areas of the entire project, such as:

- Project Info, Building Department Contact Info, Designer Notes, Revisions, Client, Designer.
- Future additions

Calculations Division
A storage location for calcs that the user adds to the Project File.

Seismic, Wind, Other Loads & Forces Division (Not available in RetainPro mode)
Historical Note: When Loads & Forces calculations were first introduced, they were automatically included in the Project Manager when starting a new project file, appearing as a set of preloaded Loads & Forces divisions. These divisions and modules will continue to appear in ENERCALC SEL project files when the governing code is set to ASCE 7-10 or ASCE 7-16.



© 2026 ENERCALC, LLC

Components that any Project File CAN contain are:

Custom Divisions (An unlimited number can exist in a Project File)
Storage locations (folders) for organizing user-generated calculations and external items.

- Can be created, named, copied, organized, expanded, collapsed, and deleted.
- Are typically named in ways that are meaningful to the designer and that suit the specific project.
- Are generally populated with calculations and external items.
- Provide a convenient way to selectively control project printing operations for logical sets of calculations.
- Every new Project File is automatically populated with one Division named "Calculations".

Calculations (An unlimited number can exist in a Project File)
User-generated calculations that are created using the built-in modules and are stored in Divisions.

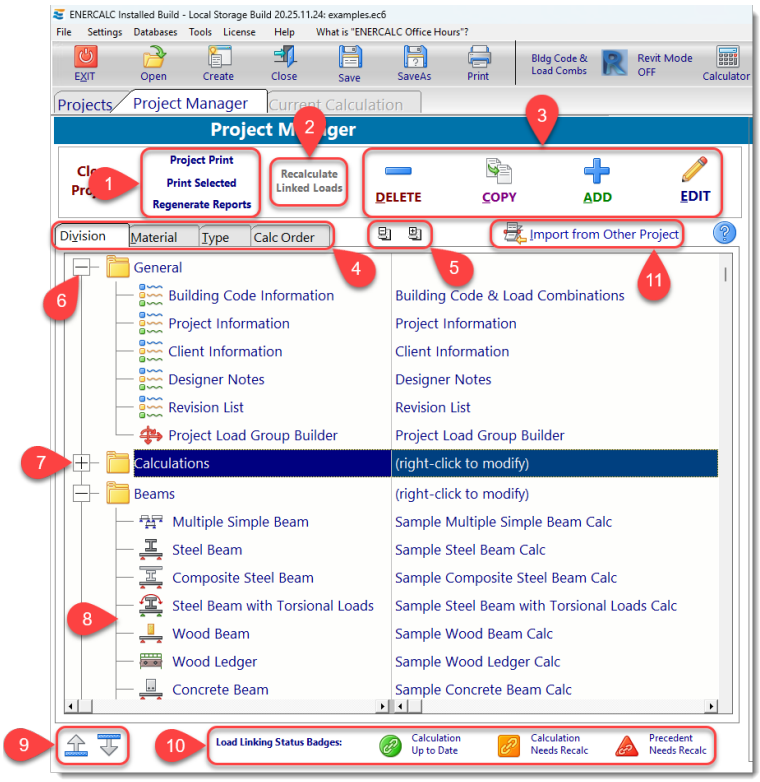
- Can be created, moved between Divisions, and deleted.
- Can be copied to serve as the basis for a new calculation with similar input data.
- Can be imported from one Project File to another.
- Can be printed singly or in batch mode using the Project Printing Manager.

External Items (An unlimited number can exist in a Project File) (Not available in RetainPro mode)
User-generated external files that are created using external programs and then stored in the Custom Divisions.

- Can consist of MS Excel spreadsheets, Adobe Acrobat PDFs, and scanned documents/images.
- Can be embedded within the Project File for maximum portability, or can be linked to the Project File to minimize file size.
- Can be created, moved between Divisions, and deleted.

© 2026 ENERCALC, LLC

Please see the descriptive keynotes on the following screen capture for information on various areas of the Project Manager.



(1) **Project Printing:** These buttons allow you to print the report for the selected calculation, or open the Project Printing system, which allows you to review all report pages for all calculation items in the entire project. Click here 125 to review the specific section on Project Printing.

(2) **Recalculate Linked Loads:** This button becomes available when linked load calculations are out of date. Clicking it automatically traverses the load path and recalculates all affected calculations in a single action. Click here 203 to review the specific section on Linked Loads.

(3) **Buttons to Add, Copy, Edit & Delete Individual Calculations:** These buttons allow you to add, copy, edit and delete ENERCALC calculations or external source items (such as Excel sheets) in the selected Division. Click here 113 to review specific section.

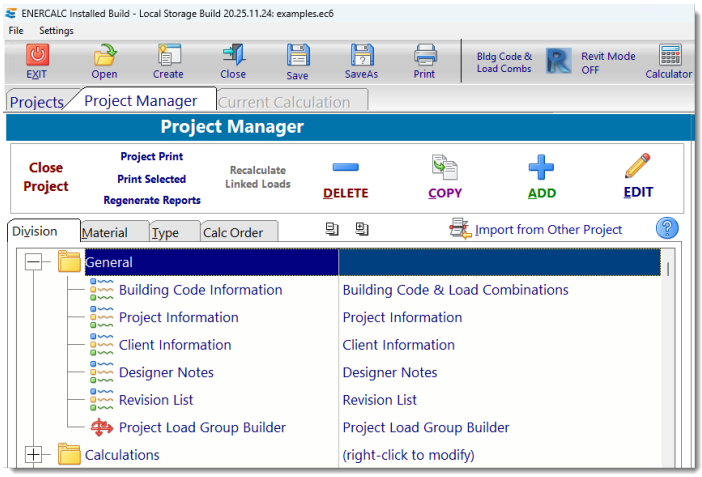
NOTE: The Delete, Add, and Copy buttons are only accessible when the Sort Options "Division" tab is selected.

(4) **Sort Options:** These four tabs sort your Project File content by Division grouping, Type of calculation (Steel Beam, Concrete Column, Excel sheet, etc.), Material (Concrete, Steel, Wood, etc.) or Calc Order (for Linked Load calculations). Click here 114 to review specific section. Note that the [**Add**], [**Copy**], and [**Delete**] options are not available in the Type, Material or Calc Order views.

© 2026 ENERCALC, LLC

(5) **Expand & Contract Tree:** These two buttons fully expand or fully contract the tree structure to display or hide the full contents of the Project File.

(6) **GENERAL Division:** The GENERAL Division is automatically created in each new Project File. There can only be one GENERAL Division, and its purpose is to contain and organize certain hard-coded pieces of general information that apply to the Project as a whole. Click here 101 to review the section describing these items. The screen capture below shows a view of the Project Manager with the GENERAL Division expanded to display its contents:

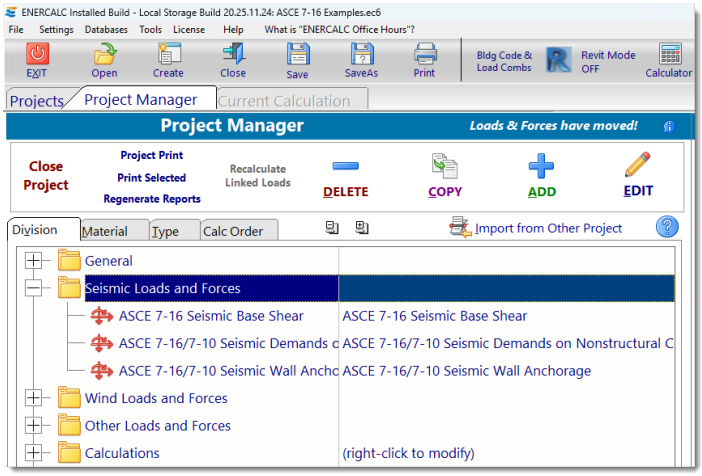


(7) **Calculations Division:** The Calculations Division is also automatically created in each new Project File, but it is created merely as a user-convenience. It can be thought of as a "custom" Division, because its name can be changed and it can be moved or deleted. The user is free to create as many custom Divisions as the Project warrants, and to name them as best suits the Project.

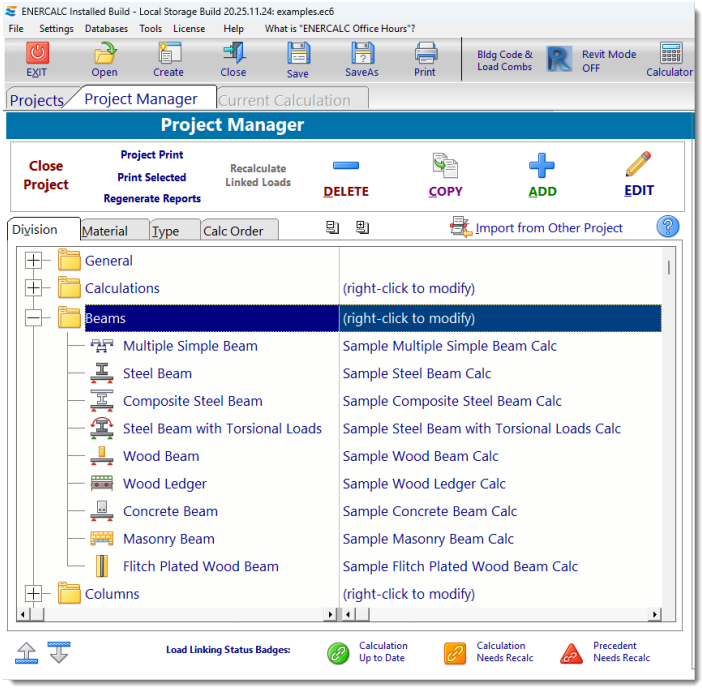
LOADS & FORCES Divisions (Not available in RetainPro mode):

Historical Note: When Loads & Forces calculations were first introduced, they were automatically included in the Project Manager when starting a new project file, appearing as a set of preloaded Loads & Forces divisions. These divisions and modules will continue to appear in ENERCALC SEL project files when the governing code is set to ASCE 7-10 or ASCE 7-16. The screen capture below shows the Project Manager view for a SEL project with the governing code set to ASCE 7-16. The Seismic Loads & Forces division is expanded to display its contents.

© 2026 ENERCALC, LLC



(8) **Calculation List:** This is the main list that displays all the calculation items that you have added into your project. It allows you to organize your calculations into Divisions. See image below for another look at two Divisions labeled "Beams" and "Columns". Columns shows a [**+**] button to its left, indicating that the tree containing the calculations in that Division is compressed. Click here 108 to review the section describing these items.



(9) **Move Item in List:** These buttons move the highlighted calculation up and down within the list. Click here 118 to review specific section.

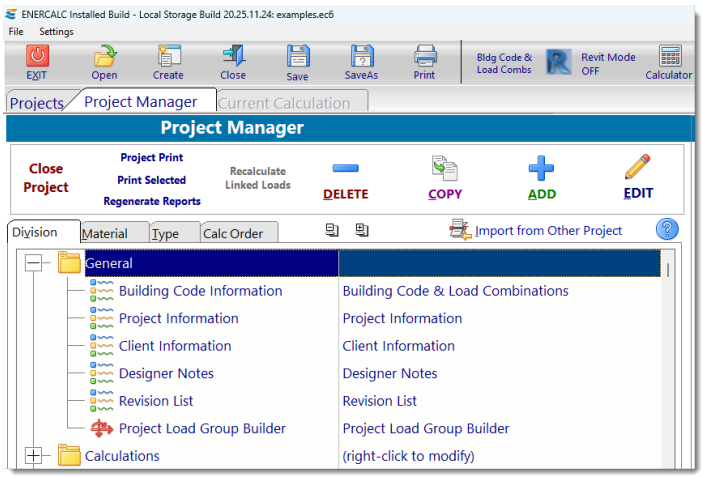
(10) **Load Linking Status Badges:** This area displays the color-coded status badge legend and brief descriptions for each badge. Click here 203 to review the specific section on Linked Loads.

© 2026 ENERCALC, LLC

(11) **Import Calculations from other Project Files:** Click here 123 to review specific section.

11.1 General Division

This Division contains a list of items that describe the entire project.



Building Code Information 101: Allows entry of information relating to the building code, jurisdiction and building official's contact information.

Project Information 104: Allows entry of general information about the project.

Client Information 104: Allows entry of information specific to the engineer's client.

Designer Notes 105: Allows entry of up to 18 specific notes assigned by a specific person on a certain date.

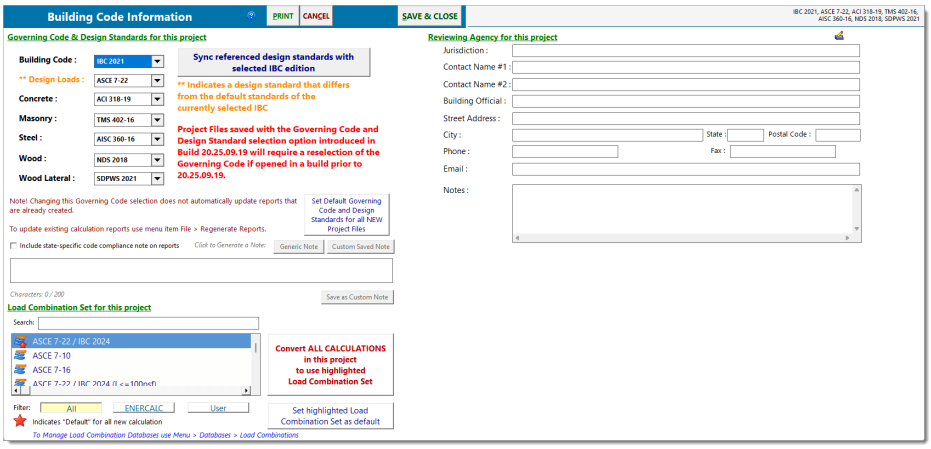
Revisions 105: Allows entry of up to 18 specific revisions assigned by a specific person on a certain date.

Project Load Group Builder 254: Allows storage of commonly used Load Items and the ability to combine them into Groups for calculating total loads.

11.1.1 Building Code Information

The **Building Code Information** form defines the governing codes, design standards, load combination set, and reviewing agency information for the current project. All entries apply only to this specific project file and do not set defaults for other projects.

© 2026 ENERCALC, LLC



Governing Code & Design Standards

These selections establish the reference standards used in the calculations for the current project file. Users may select the governing building code and each referenced design standard through dropdown menus. These controls allow full flexibility to mix and match editions as required by the state or local authority having jurisdiction (AHJ).

NOTE: Project files saved with the Governing Code and Design Standard selection (introduced in Build 20.25.09.19) will need the Governing Code re-selected if they're opened in an earlier build. If you're collaborating on files across a team, we strongly recommend everyone upgrade to the latest build to keep things smooth and hassle-free.

Changing code selections does not automatically update previously generated reports. To apply changes, regenerate reports using *File > Regenerate Reports*.

Sync referenced design standards with selected IBC edition: When clicked, the program sets all referenced design standards to align with the default referenced design standards of the currently selected IBC.

Set Default Governing Code and Design Standards for all NEW Project Files: Updates the program-wide default standards for future projects. Existing projects are not affected.

Include state-specific code compliance note on reports: When enabled, a state-specific code compliance note will be included on each page of the printed report. This can be useful when it might be helpful to provide plan reviewers with confirmation that the output is based on a specific version of the IBC and conforms to state-specific adoptions and amendments. Generic text is provided to assist in structuring such a note, and users may also customize and save their own note for reuse.

Load Combination Set

Defines the default load combinations used for all new calculations created in the current project file.

© 2026 ENERCALC, LLC

Search: Filters available load combination sets by name.

Selection List: Displays available load combination sets. Sets are categorized as:

- ENERCALC – predefined load combination sets by the software
- User – custom load combination sets created by the user

Filter Options: Filter the selection list to show specific load combination sets only.

- All – show all sets
- ENERCALC – show only program-supplied sets
- User – show only user-defined sets

Convert ALL CALCULATIONS in this project to use highlighted Load Combination Set: Applies the selected load combination set to all existing calculations within the current project.

Set highlighted Load Combination Set as default: Sets the highlighted Load Combination Set as the default for new calculations in this project and in all future projects.

NOTE: Load combination sets cannot directly be modified from this screen. To edit load combination sets, use *Menu* > *Databases* > *Load Combinations*.

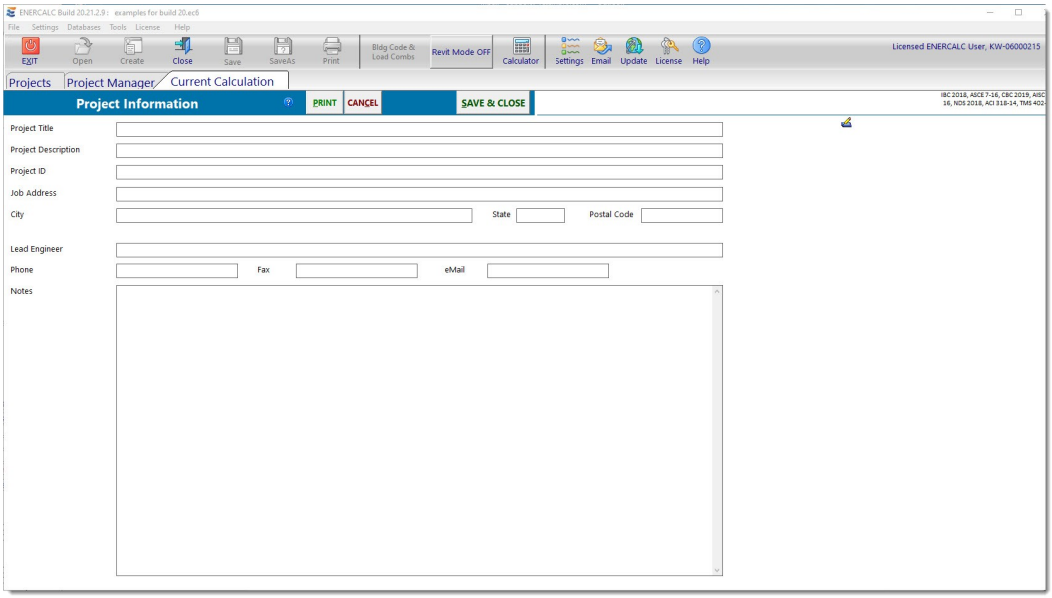
Reviewing Agency Information

This section is optional and allows users to record jurisdictional and contact details for the reviewing authority. This information may appear in printed reports if the user chooses to include it.

© 2026 ENERCALC, LLC

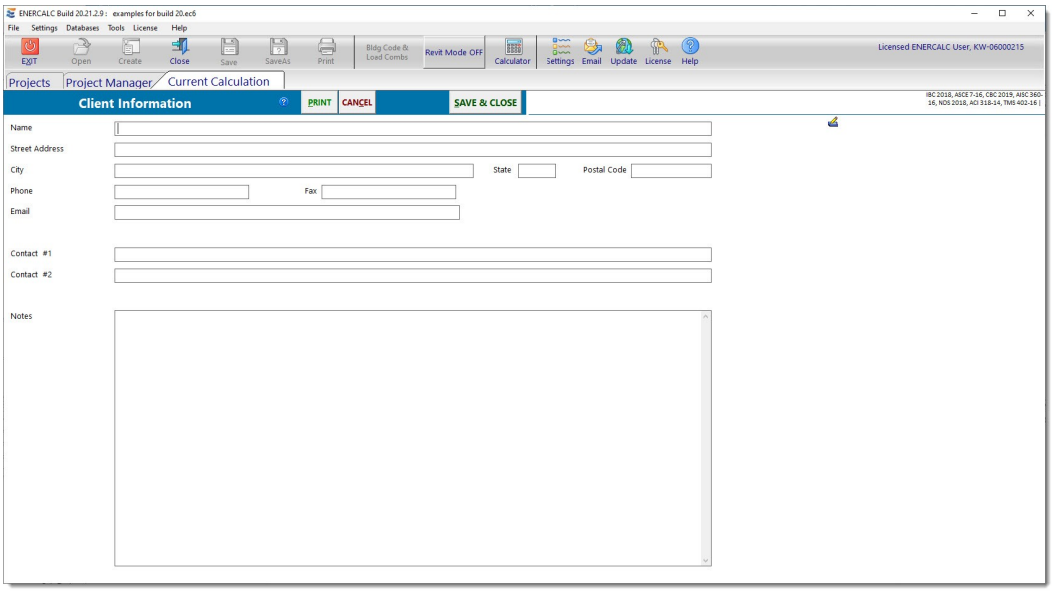
11.1.2 Project Information

This form allows you to enter information specific to your project. This data is specific to your Project File.



11.1.3 Client Information

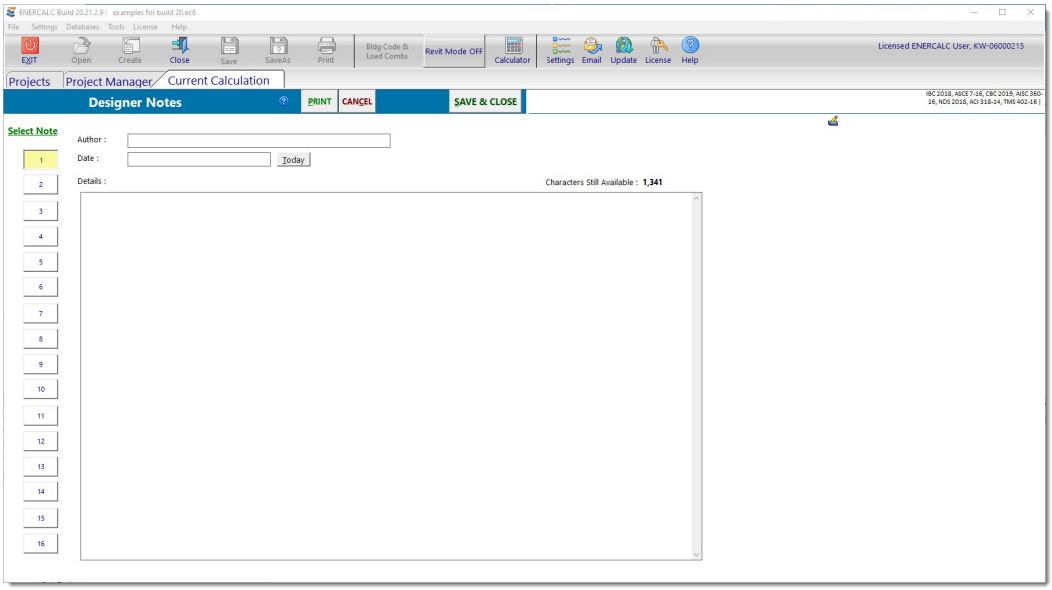
This form allows you to enter information on your client for a specific project. This data is specific to your Project File.



© 2026 ENERCALC, LLC

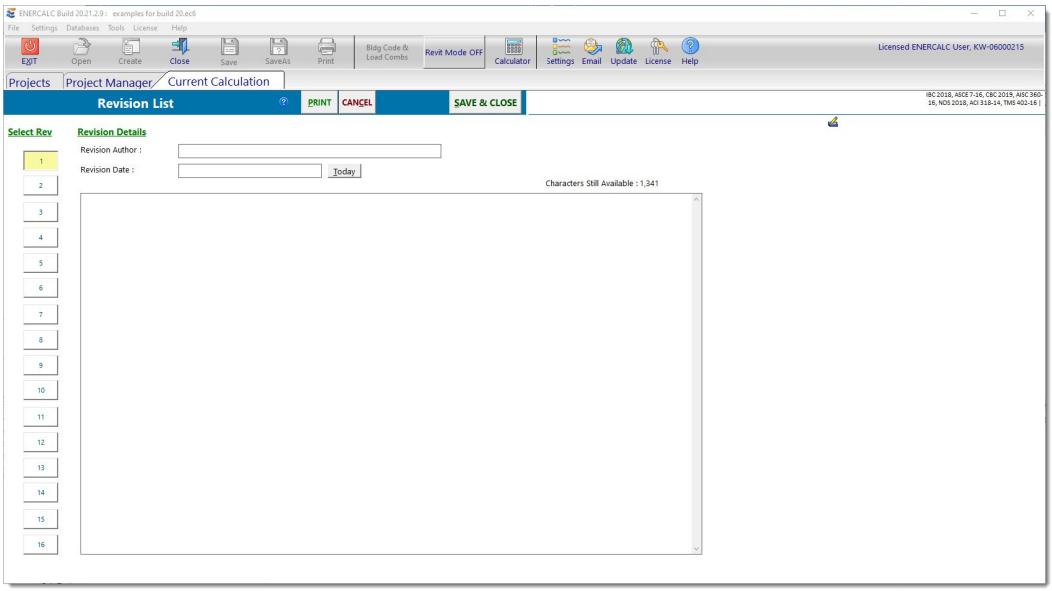
11.1.4 Designer Notes

This form allows you to enter up to 18 distinct notes on the project, each with a specific author and creation date. This data is specific to your Project File.



11.1.5 Revision List

This form allows you to enter up to 18 distinct revision explanations on the project, each with a specific author and creation date. This data is specific to your Project File.

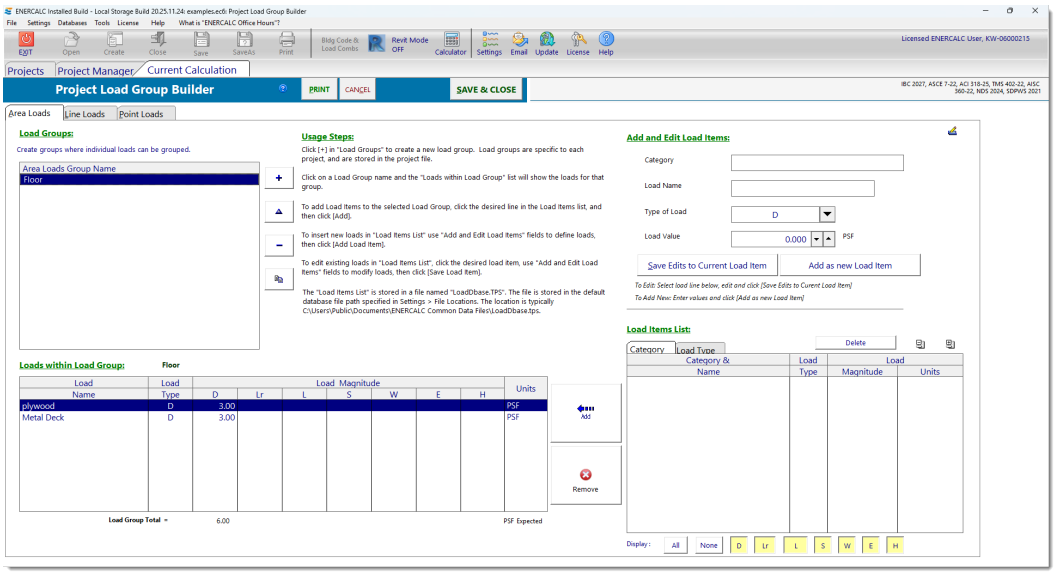


© 2026 ENERCALC, LLC

11.1.6 Project Load Group Builder

The Project Load Group Builder defines commonly used Load Items and allows them to be combined into Groups for calculating total loads. All entries apply only to the current project file and do not set defaults for other projects.

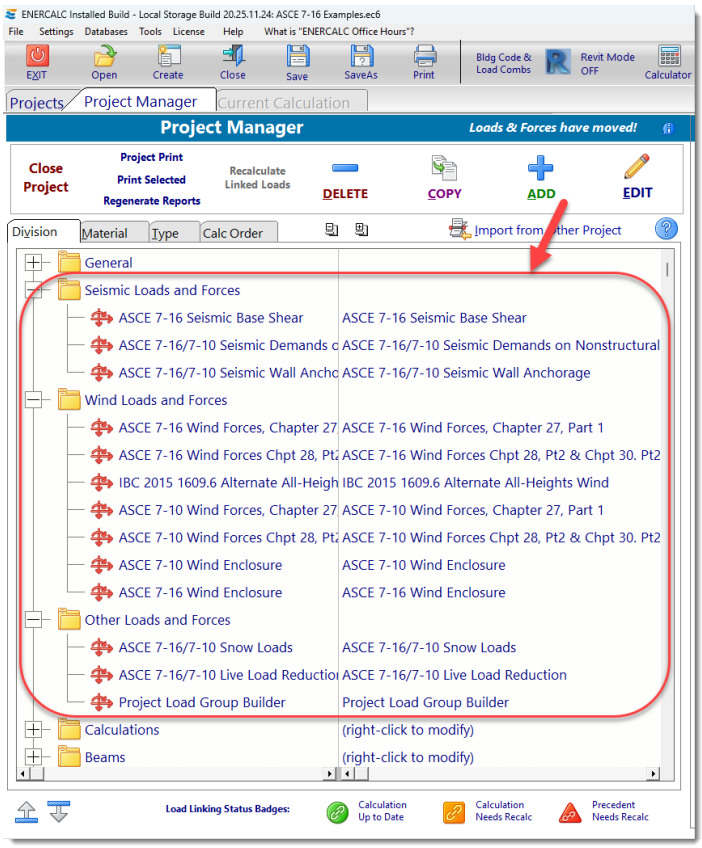
Click here 254 to review specific section.



11.2 Loads & Forces Divisions

Historical Note: When Loads & Forces calculations were first introduced, they were automatically included in the Project Manager when starting a new project file, appearing as a set of preloaded Loads & Forces divisions. These divisions and modules will continue to appear in ENERCALC SEL project files when the governing code is set to ASCE 7-10 or ASCE 7-16.

The screen capture below shows the Project Manager view for a SEL project with the governing code set to ASCE 7-16.

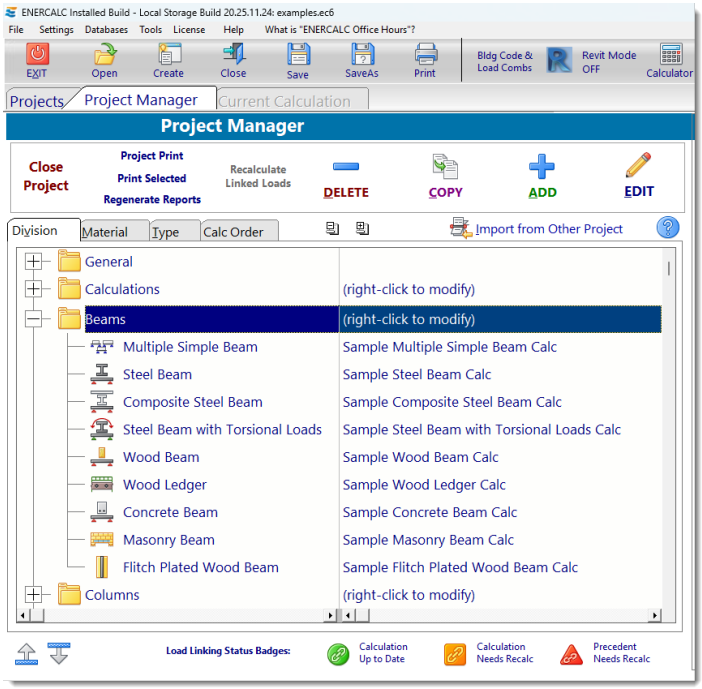


See specific descriptions under Calculation Modules > Loads & Forces Division 107.

© 2026 ENERCALC, LLC

11.3 Calculation List

The Calculation List is the main Project Manager view that you will use when building your calculation sets for projects.

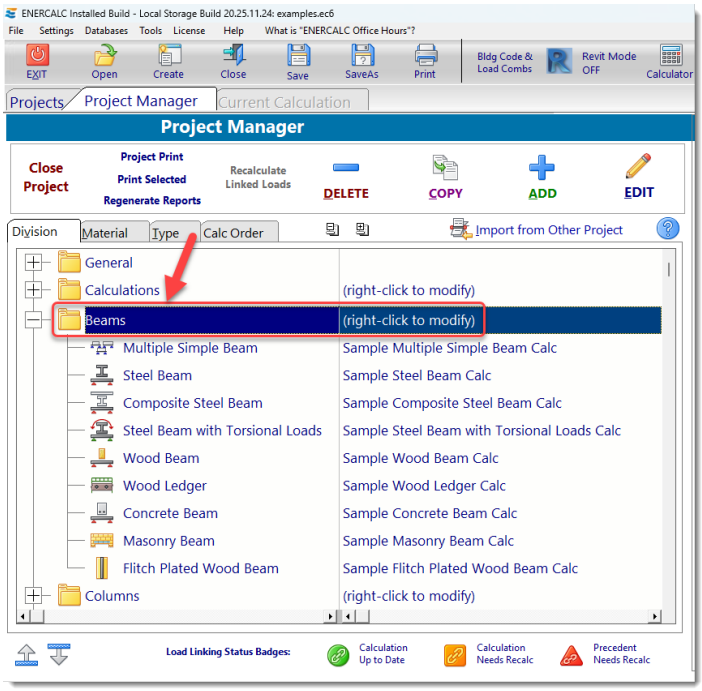


Most of a normal work session consists of adding and editing calculations in the list.

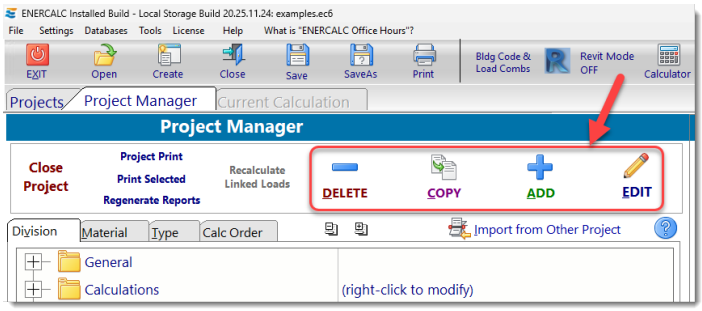
Please see the notes below as well as the following sections to learn what each button provides and how to manage the list of calculations.

© 2026 ENERCALC, LLC

Divisions: Divisions are the main organizational category for calculations. These are the left-most text item in the calculation list and will display either a [**+**] or [**-**] to the left of their name. Divisions can be created, copied, renamed, moved, and deleted. Calculations must always exist within a Division, but they can be moved freely between Divisions. Click Here 111 for the specific information section. The screen capture below illustrates an example of how a Division can be renamed with a title that is meaningful to your project:

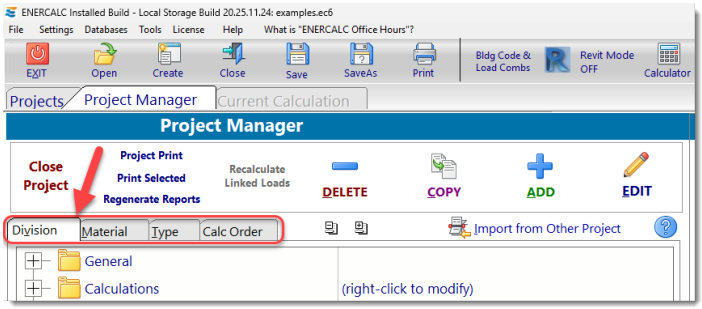


Adding, Deleting, Copying: The four buttons shown below allow you to add, copy, edit, or delete calculations and external items. Click here 113 for the specific information section.

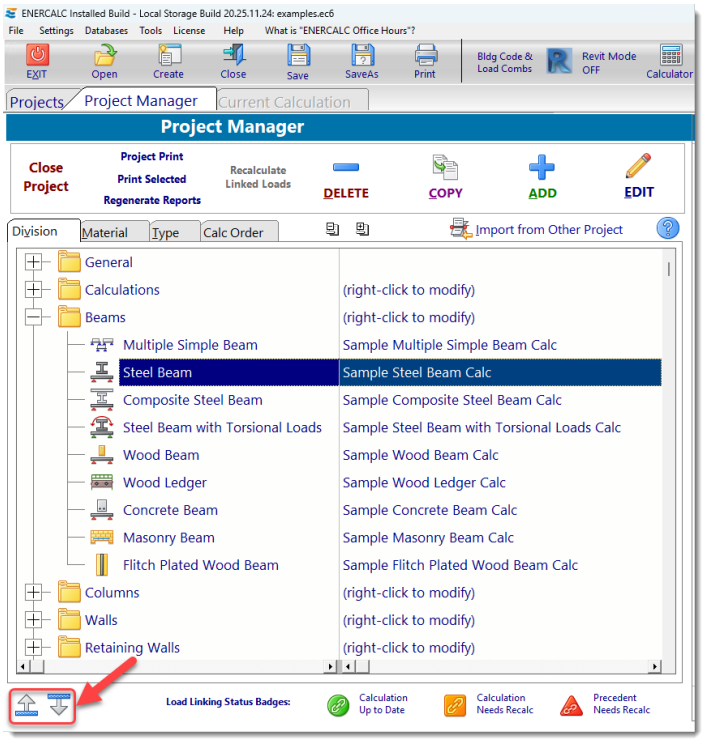


© 2026 ENERCALC, LLC

Sorting by Division, Type, Material and Calc Order: Click one of these tabs to change the sorting view of the calculations in your project. Click here 114 for the specific information section.

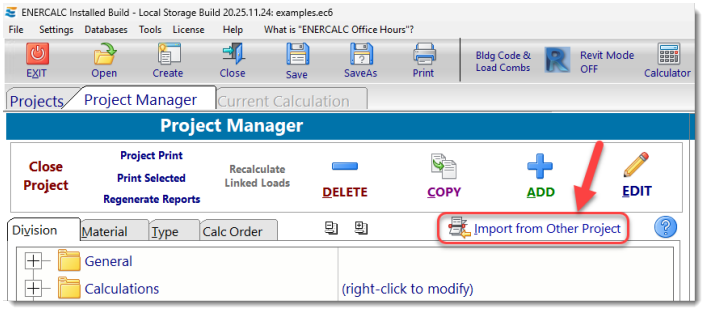


Changing Calculation and Division Order: The buttons you see below are used to move the **highlighted** calculation or Division up or down in the list. When moving a calculation downward, if moving a calculation would tend to replace a Division name, then the calculation item is moved into that Division. When moving a calculation upward, if moving a calculation would tend to replace a Division name, then the calculation item is moved into the Division above. Click here 118 for the specific information section.



© 2026 ENERCALC, LLC

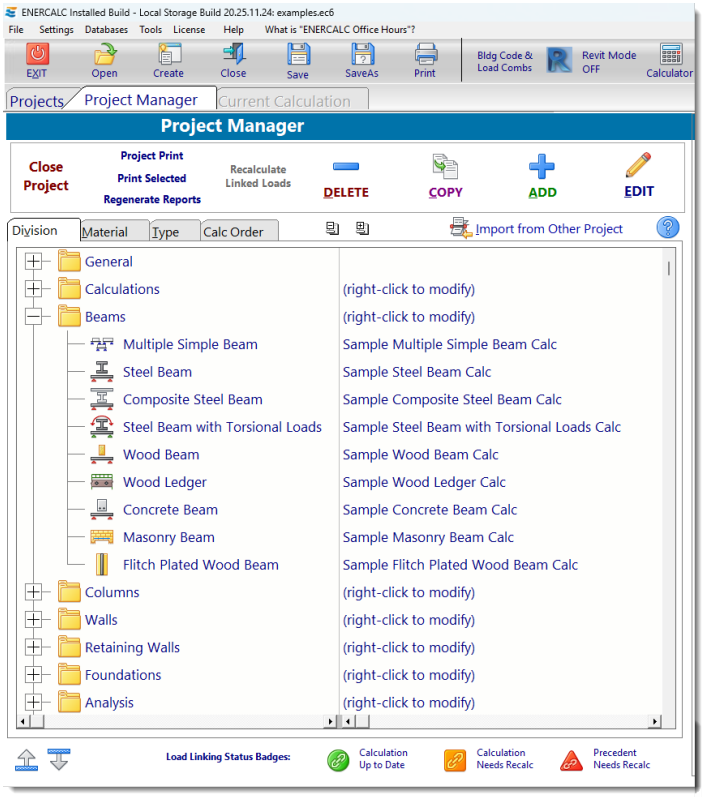
Importing Calculations: Clicking the [**Import**] button displays the calculation import system.



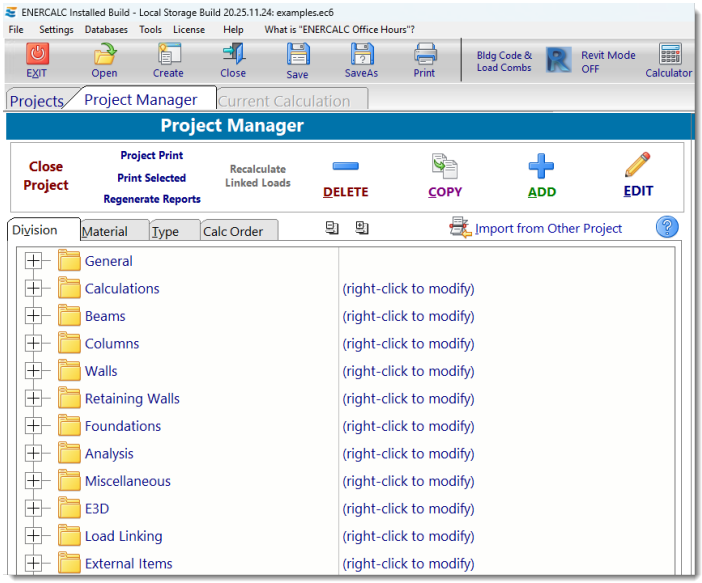
Click here 123 for the specific information section.

11.3.1 Divisions

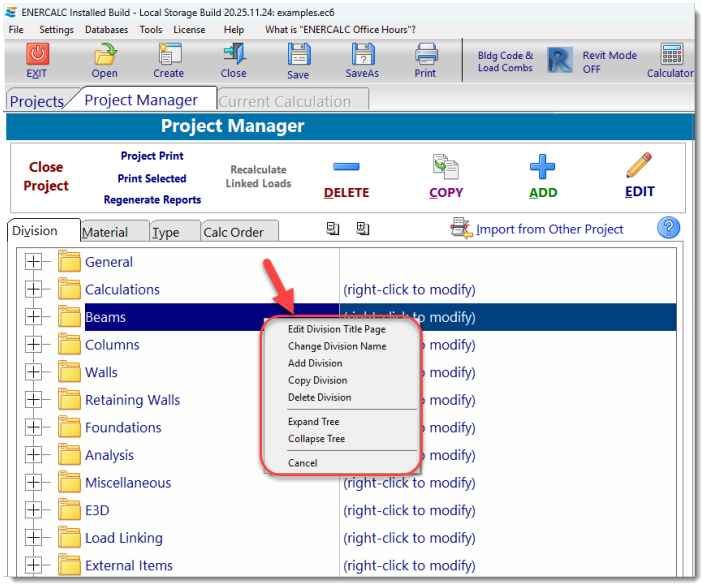
Divisions are used to organize calculations. The image below has custom Divisions named Beams, Columns, Analysis, etc.



- Clicking the [-] icon to the left of the Division name will compress the Division tree. See image below...note how only the Division names are displayed, but their contents are currently not visible.

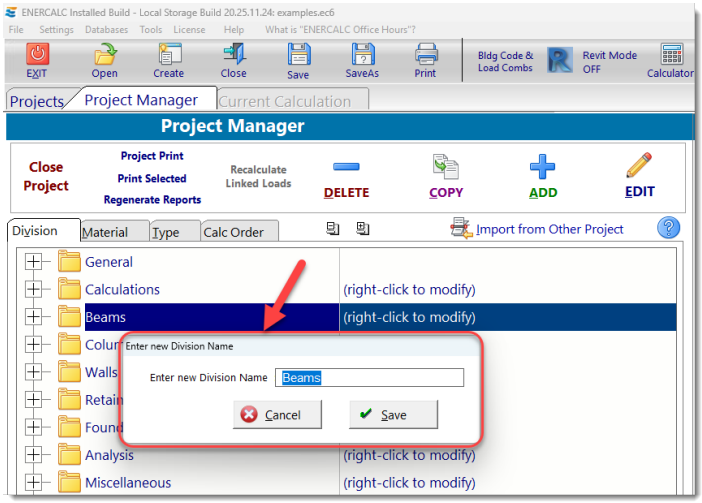


Right-clicking on a Division name will display the following pop-up menu:



© 2026 ENERCALC, LLC

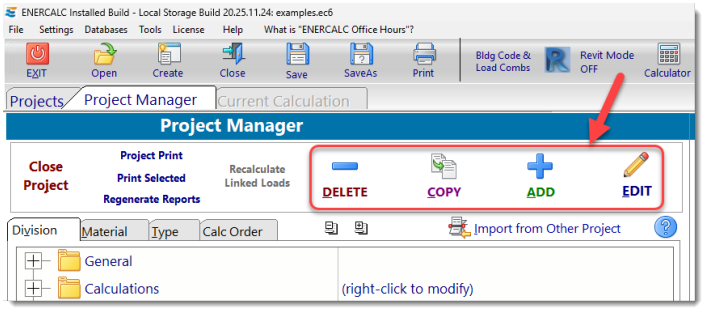
Selecting [**Change Division Name**] will display the following dialog:



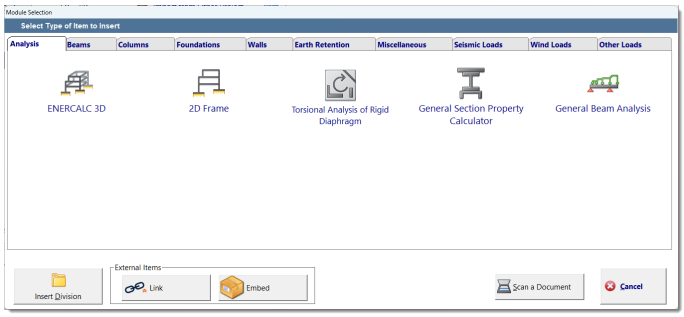
11.3.2 Adding, Deleting, Copying

The four buttons shown below are used to manipulate the calculations and Divisions in your current Project File:

NOTE: The Delete, Add, and Copy buttons are only accessible when the Sort Options "Division" tab is selected.



Add: Clicking [**Add**] displays the dialog below, where you can select the type of item to add to your Project File. The window contains two categories of items: Calculations and External Items.



The available Calculations are listed in the top portion of the dialog (and there will be more as the product matures). These are ENERCALC-created structural engineering calculations that you can use. The lower portion of the dialog lists the available External Items that can be created externally from the ENERCALC software package and then inserted into an ENERCALC Project File. Click here 1228 to go to the specific section.

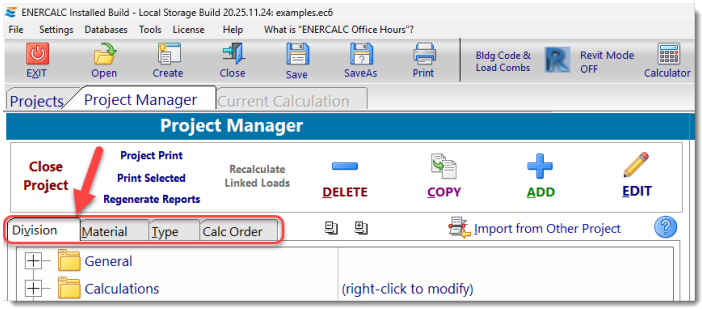
Copy: For the currently highlighted item in the calculation list, clicking [**Copy**] creates a new calculation of the same type using all of the information in the current calculation to make the new calculation.

Edit: For the currently highlighted item in the calculation list, clicking [**Edit**] opens the calculation for editing.

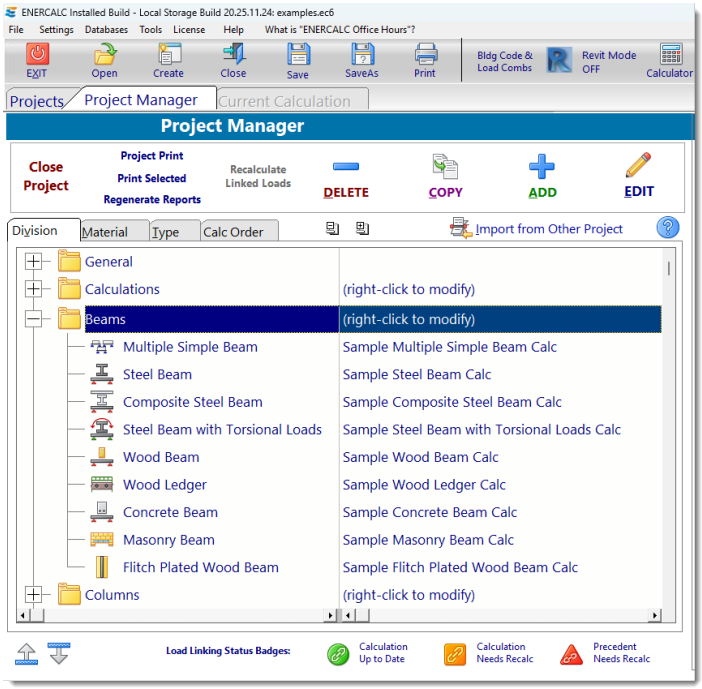
Delete: Deletes the item that is currently highlighted in the calculation list. You are prompted to confirm before the deletion is made.

11.3.3 Sorting by Division, Type & Material

Select the "Division", "Material", "Type" or "Calc Order" tab to sort the calculation list based on these three primary organizational hierarchies.

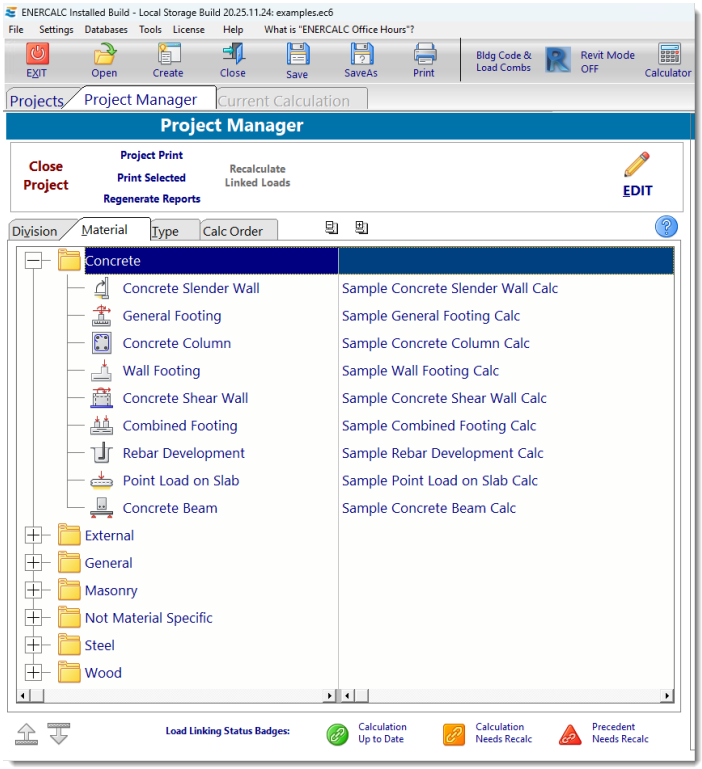


Division: Organizes the calculation list by the Divisions you created, and sorts the calculations within each Division in the order that you established. See below.



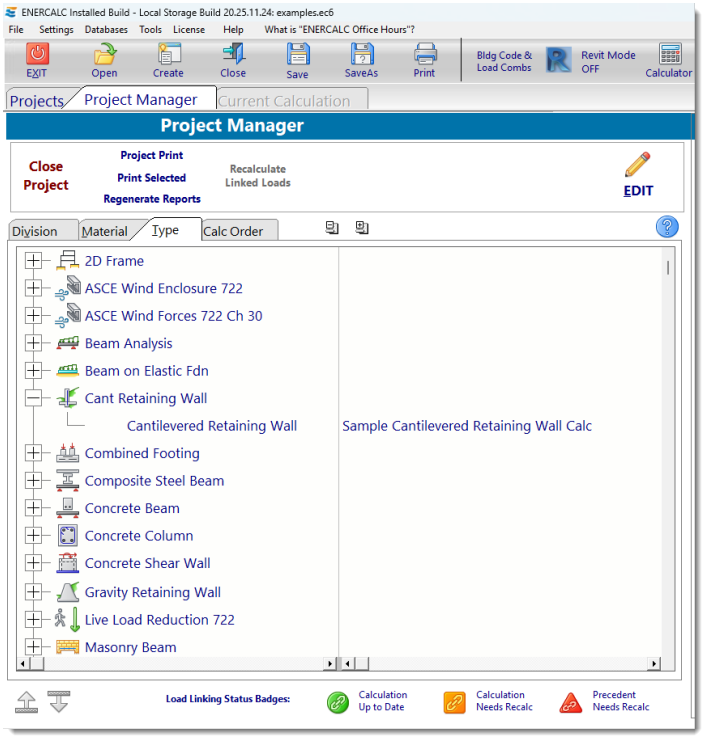
© 2026 ENERCALC, LLC

Material: Organizes the calculation list by the materials to which they refer, and sorts the calculations within each material in the order that you established. See below.



© 2026 ENERCALC, LLC

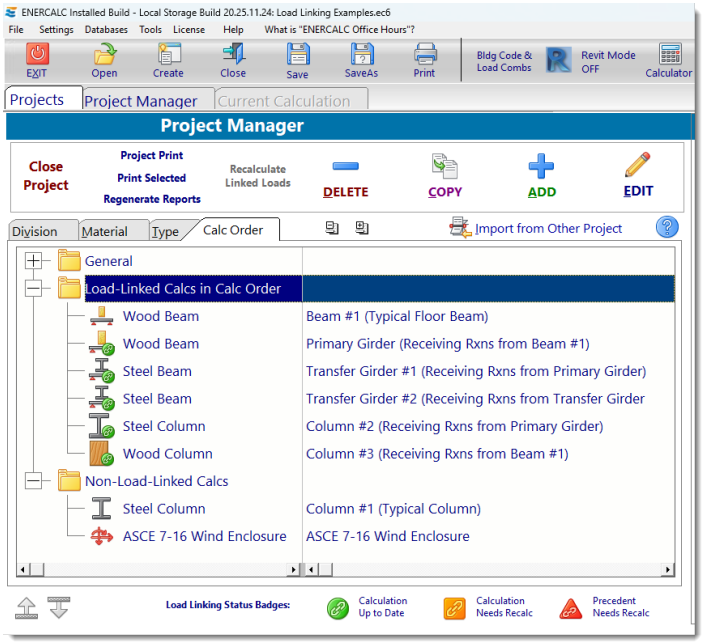
Type: Organizes the calculation list by calculation type, and sorts the calculations within each type in the order that you established. See below.



Calc Order: Organizes the calculation list in the sequence they appear within the load path, with precedent calculations listed at the top and dependent calculations appearing towards the bottom. Non–load-linked calculations are grouped separately from load-linked calculations and are not sorted. See below.

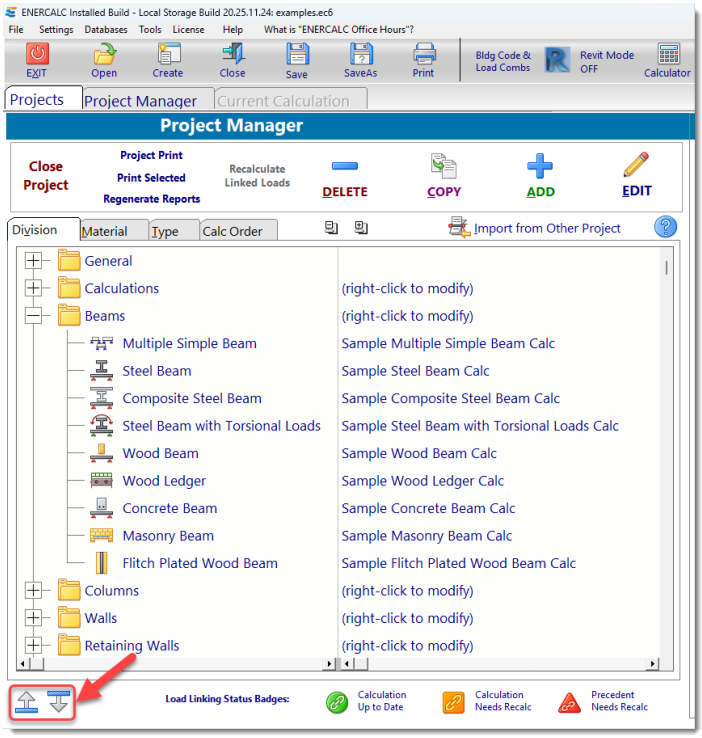
Click here 203 to review the specific section on Linked Loads.

© 2026 ENERCALC, LLC



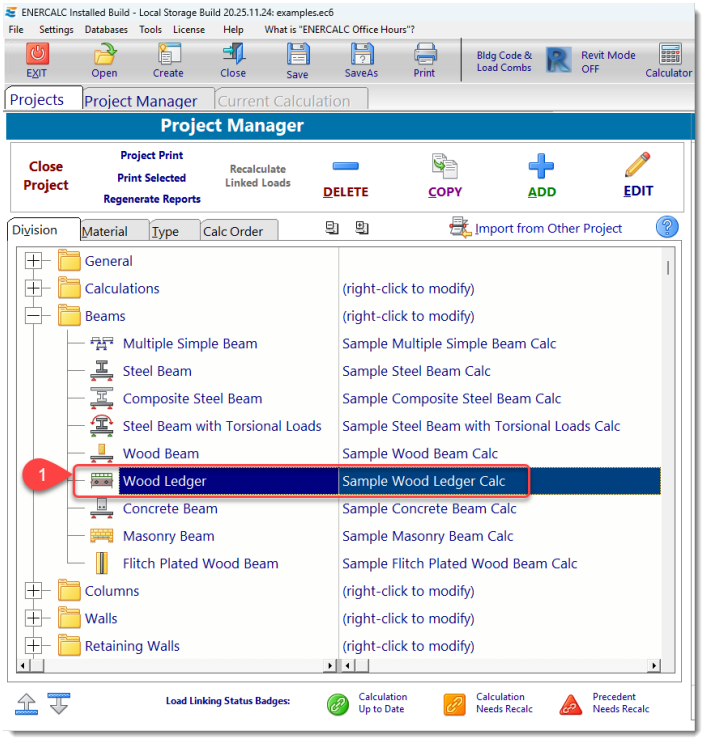
11.3.4 Changing Calculation Order

The two buttons shown at the bottom of the calculation list (see image below) are used to move the currently highlighted calculation or Division up or down in the list.



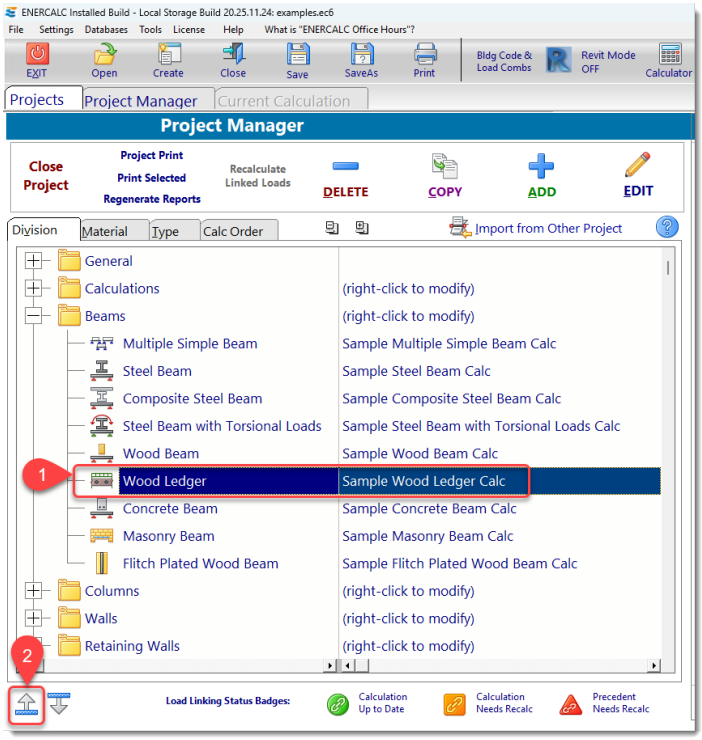
© 2026 ENERCALC, LLC

In the image below, we have clicked on a Wood Ledger calculation:

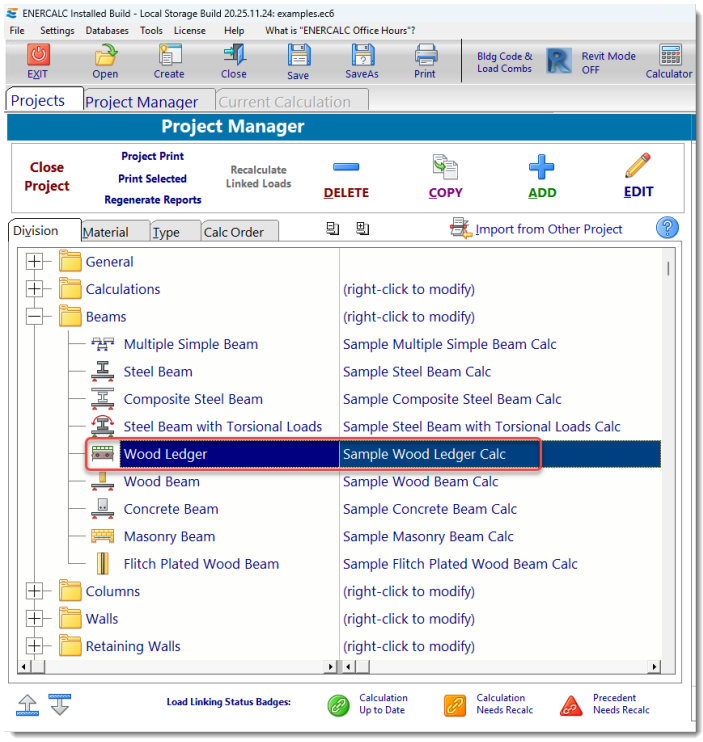


The next step is to click the Shift Calculation Up button .

© 2026 ENERCALC, LLC

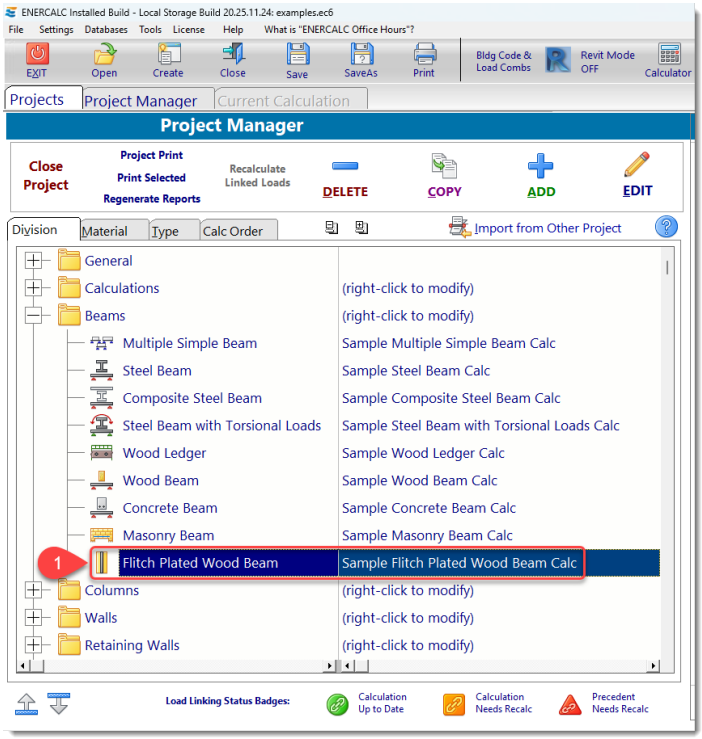


Now notice that the Wood Ledger calc has shifted up one position:

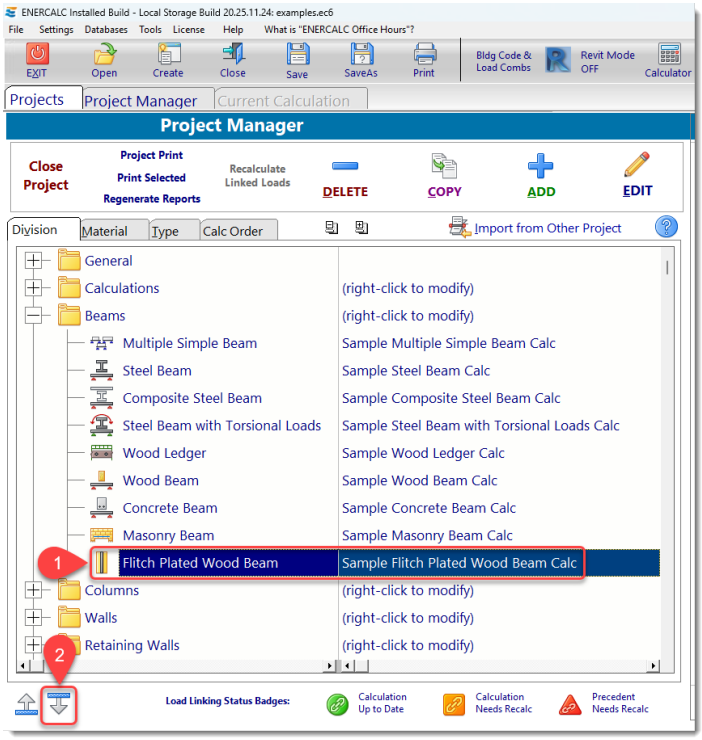


© 2026 ENERCALC, LLC

Now we will show that moving the last item in a Division downward will move the item to the top of the next Division. Notice that we have highlighted the Flitch Plate calculation in the screen capture below:

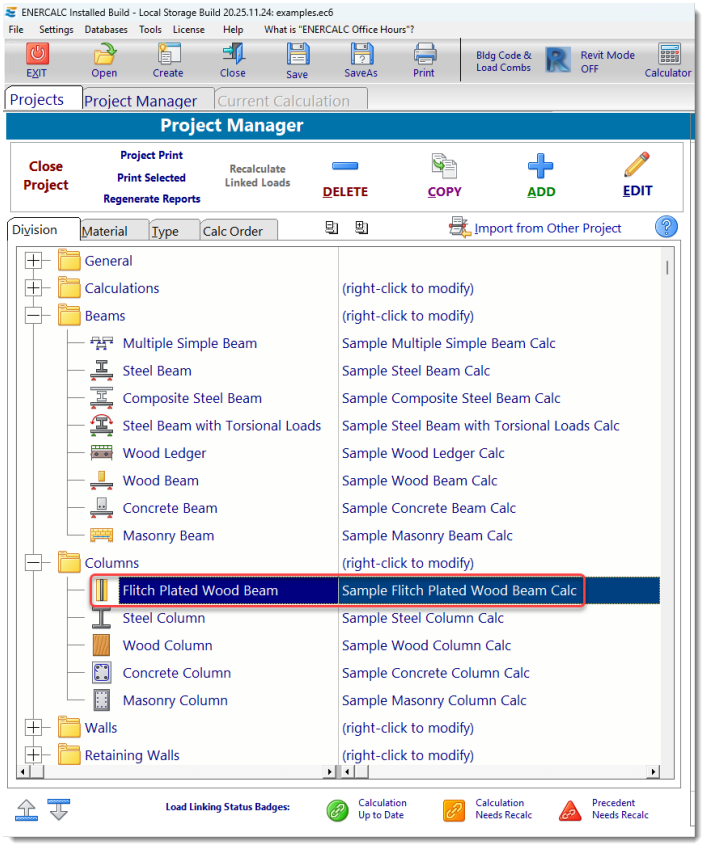


The next step is to click the Shift Calculation Down button .



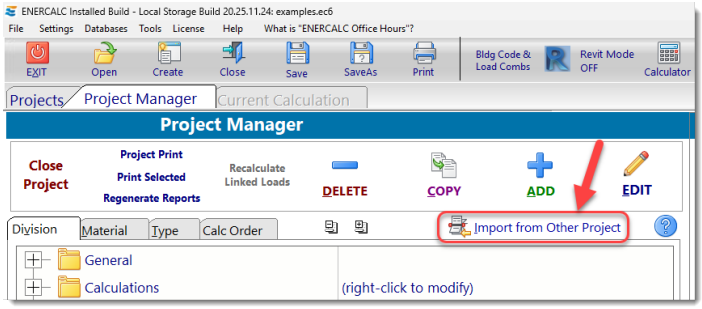
© 2026 ENERCALC, LLC

Now notice that the Flitch Plate calc has shifted down one position, which puts it at the top of the calcs in the Columns Division:



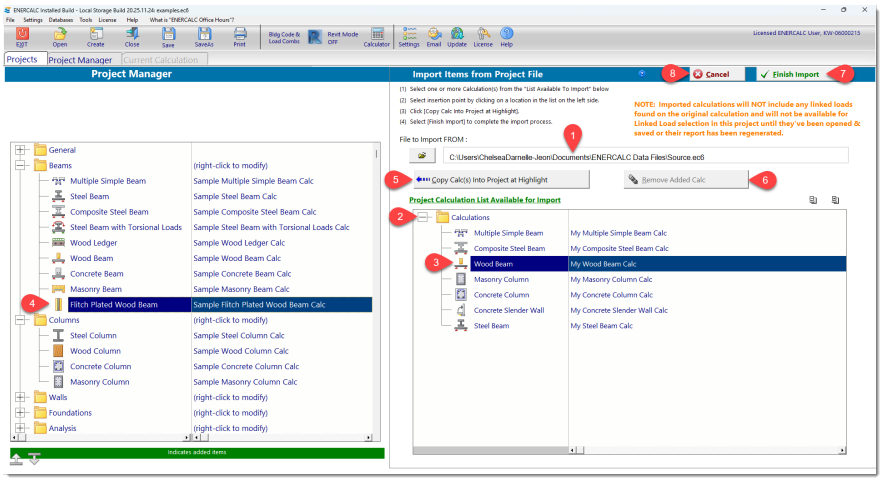
11.3.5 Importing Calculations

The Import button is located as shown below:



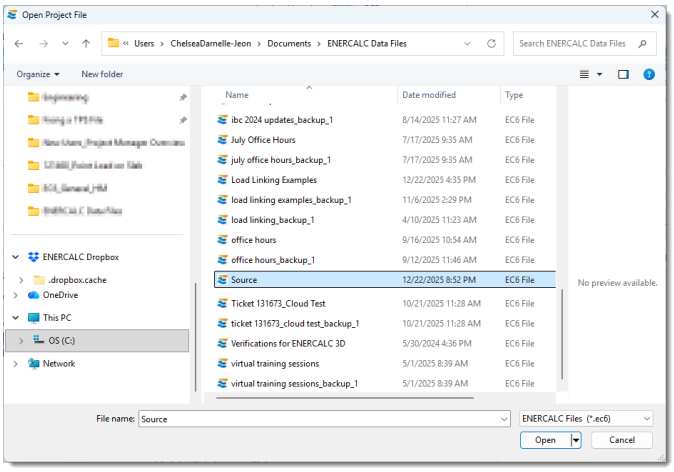
Clicking the Import button will change the Project Manager screen to appear as shown in the following image.

Please see the descriptions following the numbered keynotes. The order of the notes follows the order of usage.



(1) Specify the source ENERCALC Project File from which you will import calculations into the current (destination) Project File.

Clicking the [**Select Source File**] icon will display a standard Windows File Open dialog:



(2) After identifying the source Project File, area **(2)** will show the calculations in that source file in a tree-structure list.

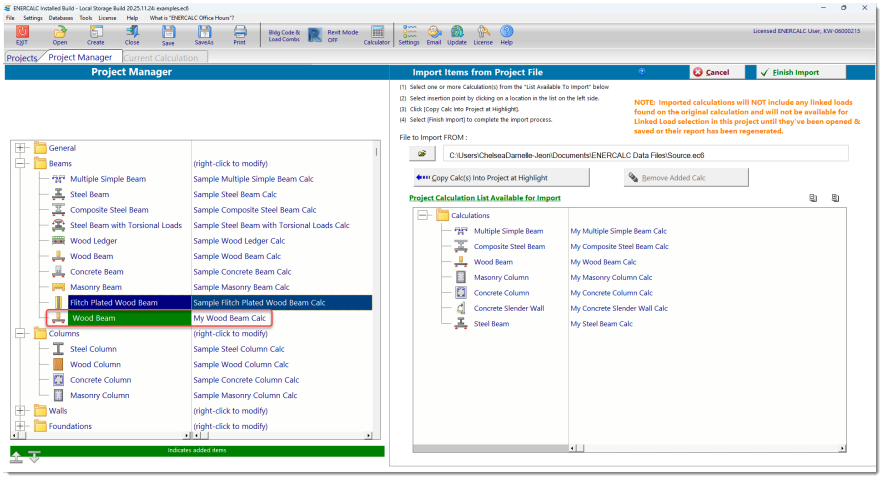
(3) Click on the desired calculation in the source file to select it. It will become highlighted.

(4) Click in the current (destination) Project File to highlight a line where the calculation should be imported. (The only restriction is that calculations cannot be imported into the GENERAL Division or the LOADS & FORCES Divisions, so be sure to select a location other than those two Divisions or any of their contents.)

© 2026 ENERCALC, LLC

(5) Click the [**Copy Calc(s) Into Project at Highlight**] button.

Notice in the image below the [Wood Beam] calculation is now highlighted in green in the current (destination) file to indicate the location where it will be inserted.



Note: If you made a mistake and want to cancel the selection, just highlight the calculation in green on the left and click the [**Remove Added Calc**] button **(6)**.

(7) To finish the import process click the

button.

(8) To completely cancel the import click the

button.

Note: Imported calculations will NOT include any linked loads found on the original calculations and will not be available for Linked Load selection until they've been opened & saved or their report has been regenerated.

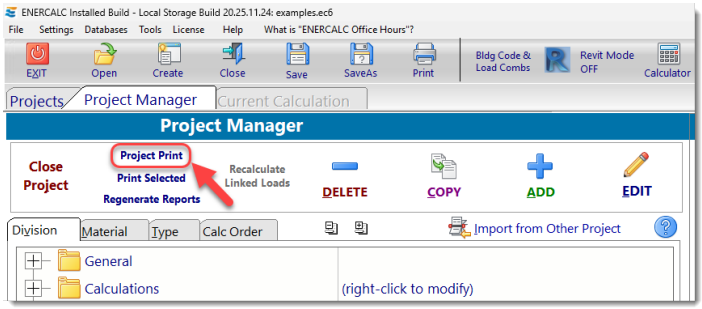
11.4 Project Printing

The **Project Printing** system allows you to print a complete project full of reports in one simple process.

You can review all the reports in your project and then select which reports to print.

© 2026 ENERCALC, LLC

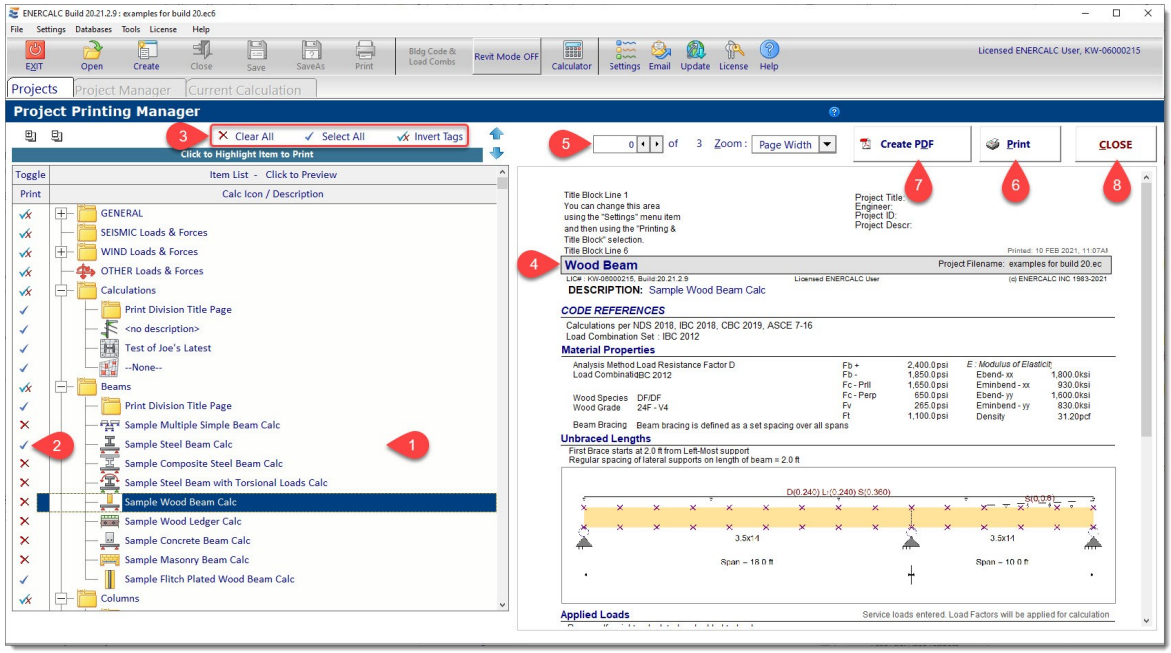
To open the Project Printing system, click the button indicated in the screen capture below:



The screen will change to display the Project Printing Manager window.

On the left will be listed all of the calculations and external items in the Project File. The items in the GENERAL Division and Loads & Forces Divisions are placed at the top, and the items in the other Divisions are placed on the bottom. When you highlight an item the full report appears on the right side of the screen.

See the numbered keynotes on the image below as we describe how to use the various controls on the screen. The information is given in order of typical usage.



(1) List of project items for which reports are available : This area is similar to the item list in the Project Manager. All available report items are listed here. Simply click an item and the report for that item will be displayed in area **(4)**. Please REMEMBER that the reports shown were created when you clicked [**Save**] or [**Save & Close**] in the specific calculation.

© 2026 ENERCALC, LLC

(2) Print / No-print check mark: This column contains a check mark or an X to denote whether that particular report item is scheduled for printing.

(3) Clear All / Select All / Invert Selection buttons: These three buttons perform bulk changes to the print/no-print status of all items in the project.

Clears the print tags on all reports, so none will be printed. A is displayed to the left of all reports after clicking this button.

Selects the print tags on all reports, so all will be printed. A is displayed to the left of all reports after clicking this button.

Inverts the print/no-print tags on all reports.

(4) Report Preview area: The complete preview of the report is shown in this area for the report item highlighted in the list.

(5) Report page selection: When the highlighted item is a multi-page report, this box selects the page to view.

(6) Print selected reports to printer: Displays the Windows standard Print dialog box and begins printing all selected project items.

(7) Create single PDF file for selected reports: Displays a Windows Open File dialog box to request a filename for a PDF file and then creates a PDF of all selected project items.

(8) Click [**Close**] to dismiss the Project Printing Manager.

© 2026 ENERCALC, LLC

Part XII

12 Sample Session

This section will guide you through a session of preparing and printing a calculation. We will use the Wood Beam calculation because it contains all the features available in ENERCALC SEL.

12.1 Starting the Program

First....let's start the program

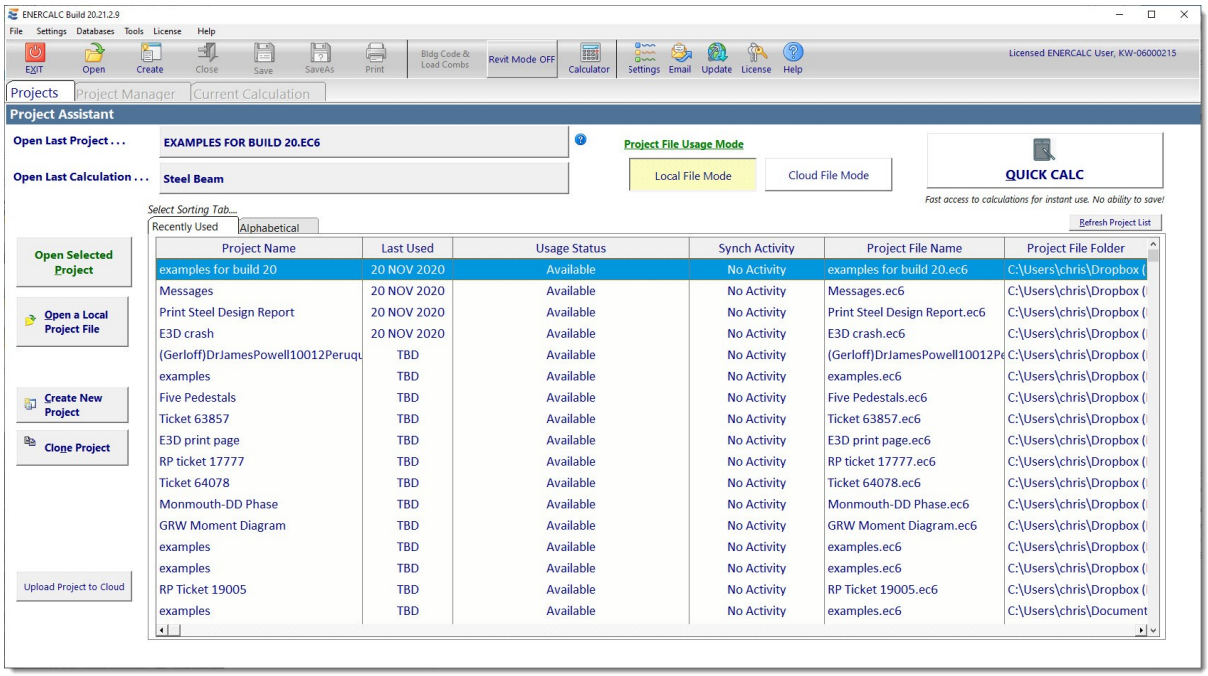
To start **ENERCALC SEL**, navigate to the ENERCALC program group from the Start menu.

Click the Windows [**Start**] button and move upward to locate the ENERCALC program group and highlight it. You will see several selections. Move the cursor to highlight Structural Library and release the mouse button.

ENERCALC SEL will now open, and you will be viewing the Project Assistant as described below.

12.2 Project Assistant

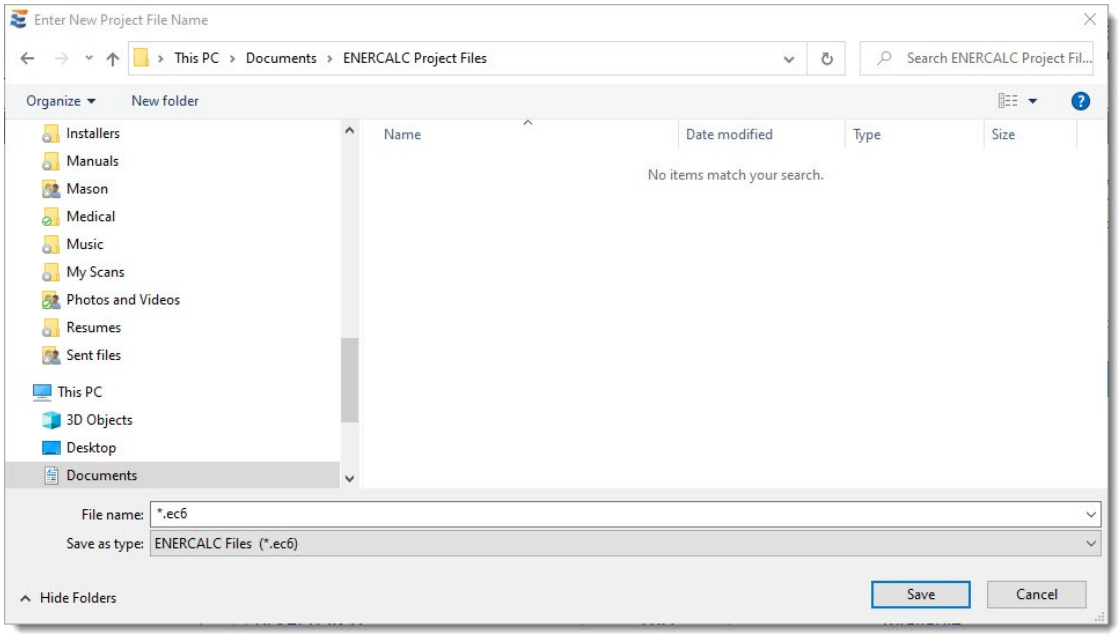
This screen gives you the ability to instantly choose how you wish to begin your work session. All of these options can also be easily chosen from the main menu.



For this Sample Session we want to create a new Project File, so the next step will be to click the [**Create New Project**] button.

© 2026 ENERCALC, LLC

The next window to be displayed (see below) will allow you to define the file name and select the drive & directory where the file should be placed.



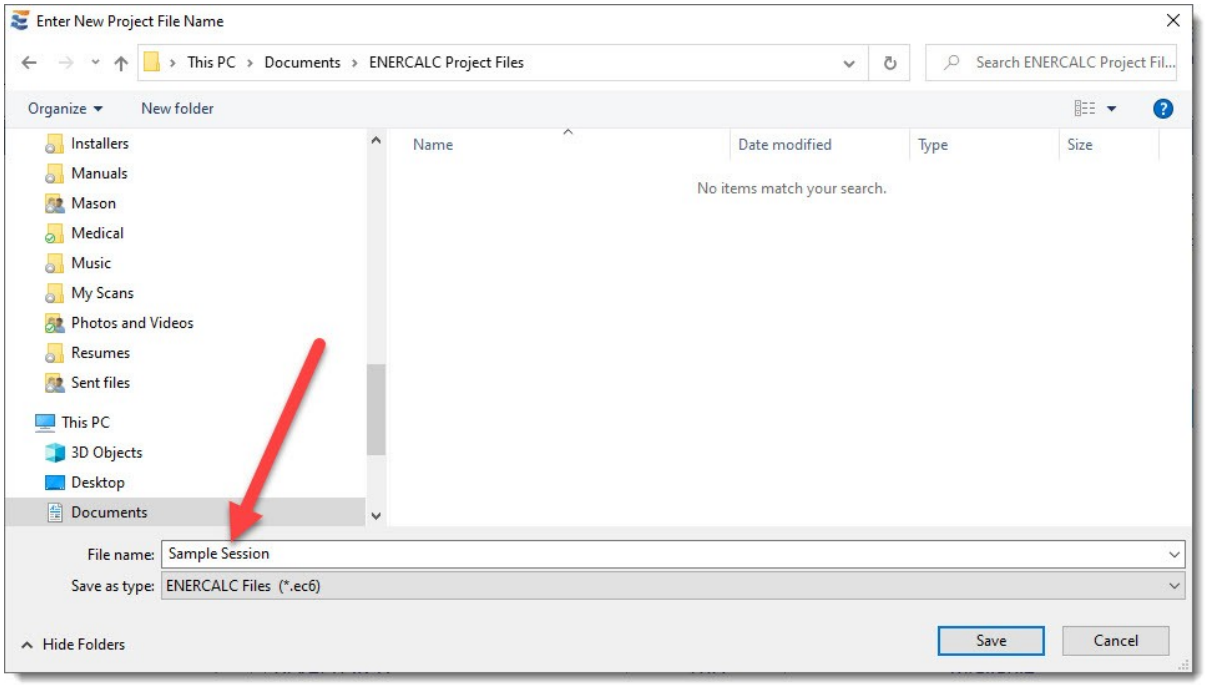
We will use the filename Sample Session and place it in a folder named ENERCALC Project Files that we created specifically to hold Project Files.

Note: You could place the file in other directories, but keep in mind that Microsoft is now recommending specific locations for saving user-generated files with the newer operating systems, and the default ENERCALC data folder conforms to these new best-practice recommendations. The important thing to recognize is that an ENERCALC Project File can hold thousands of calculations, so it is likely that there will only need to be a single ENERCALC Project File for each design project in the office.

© 2026 ENERCALC, LLC

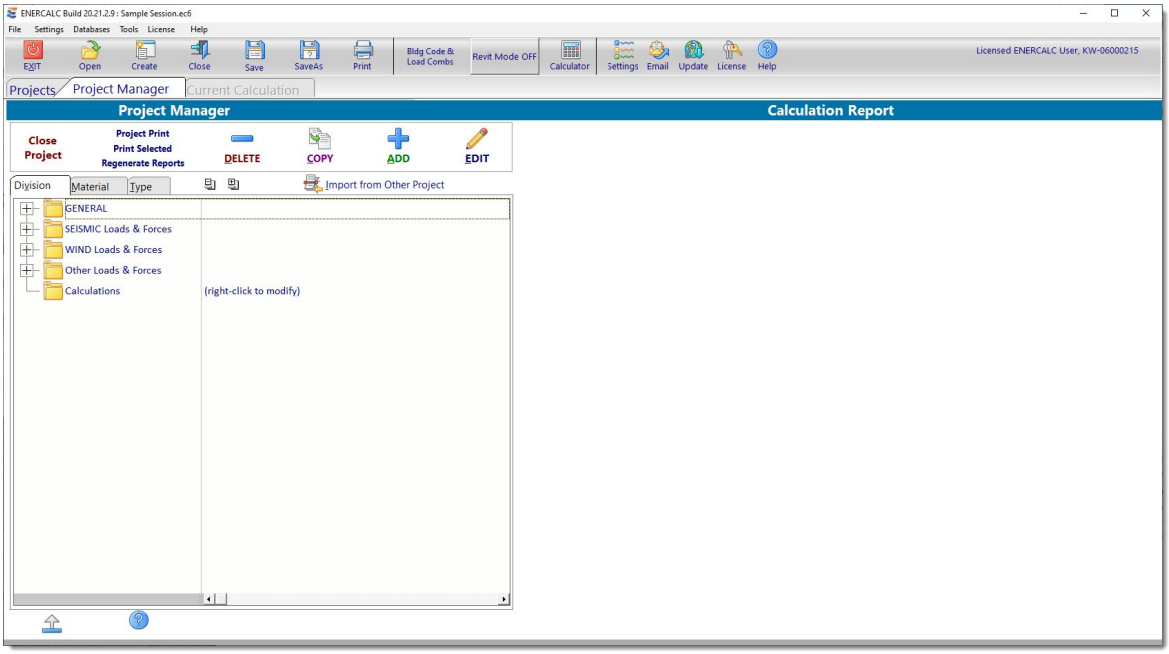
12.3 Creating a Project File

Here's the Create NEW Project File dialog box with the new filename entered. NO FILE EXTENSION is necessary....the program will append it. When the dialog appears as it does below, just click [**Save**] and the file will be created.



© 2026 ENERCALC, LLC

When a new Project File is created, the new Project File is immediately opened and displayed in the Project Manager window. You can see in the screen image below that the new Project File looks rather plain and simple.



Notice that the ONLY items appearing in the calculation list are the GENERAL Division, the LOADS & FORCES Divisions, and a "starter" Calculations Division.

You can create as many Divisions as you like. They serve as a way to organize your calculations into logical groups for your convenience .

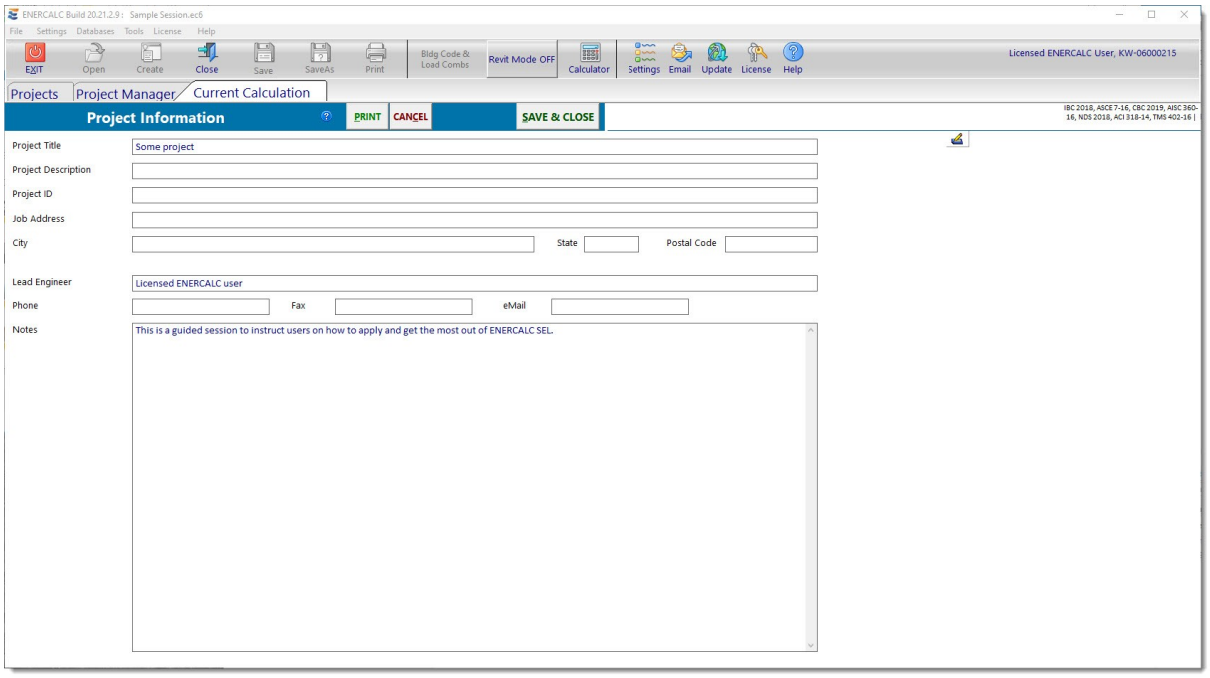
All newly created Project Files will contain these Divisions by default. The "GENERAL" Division and the "LOADS & FORCES" Divisions cannot be renamed or deleted, and they serve specific purposes, which we will see shortly. But the "Calculations" Division is just created for convenience. It can be renamed, deleted, copied, etc.

Before we add any calculations to this Project File, the next step will be to enter some general information about the project itself.

© 2026 ENERCALC, LLC

12.4 Entering Project Information

Click the [**+**] icon to the left of the **GENERAL** Division in the Project Manager to expand its contents, and then double-click the item named **Project Info**.

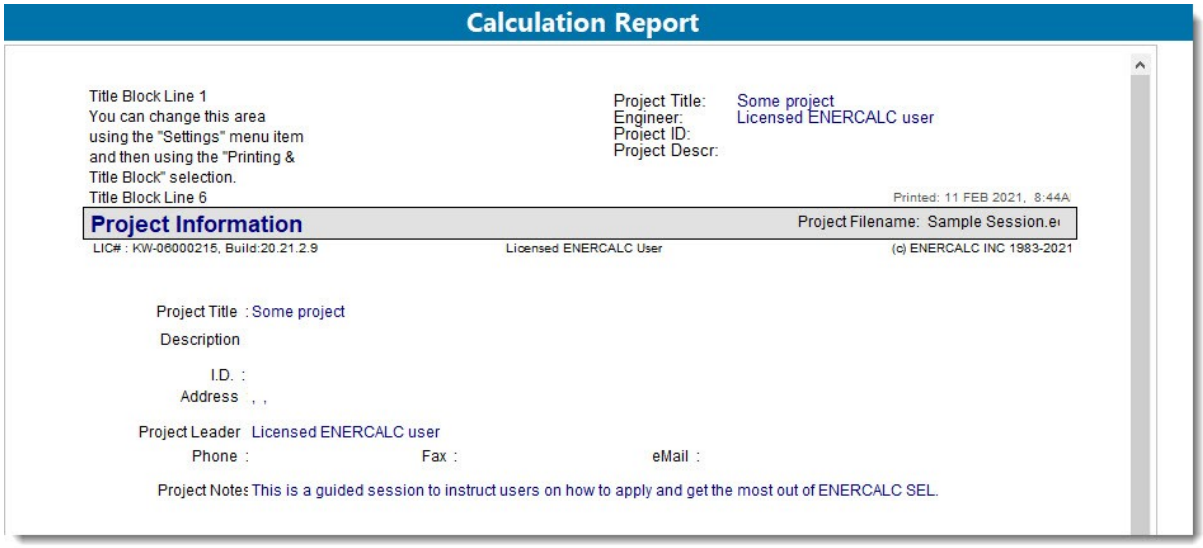


For this sample session you can fill in anything you like, just to see how the Project Information fields can be used. Then click the [**Save & Close**] button. If you see a reminder relating to report regeneration, just click OK.

The purpose of the Reminder dialog is to alert us to the fact that some data just changed in the title block. Under normal conditions, **ENERCALC SEL** generates the report for a single calculation at a time, and it does that when the Save command is issued while a calculation is open. This makes for a very efficient use of time by generating reports only on an as-needed basis. However, because we just made a change to title block data, it is likely that we want all reports regenerated and brought up to date at this time. This Reminder dialog guides us to the command in the main menu that manually forces all reports to be immediately regenerated.

© 2026 ENERCALC, LLC

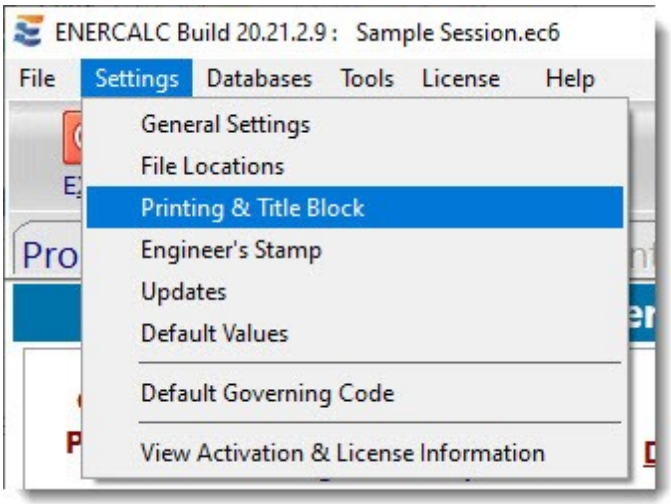
NOTE: This Project Information is printed in the upper-right corner of your printouts. All **ENERCALC SEL** printouts have a Title Block area that contains your company's information in the upper-left corner and the Project Information in the upper-right corner. The above project information (and the title block information described in the next section) will look like this on the printouts:



12.5 Setting up your Title Block

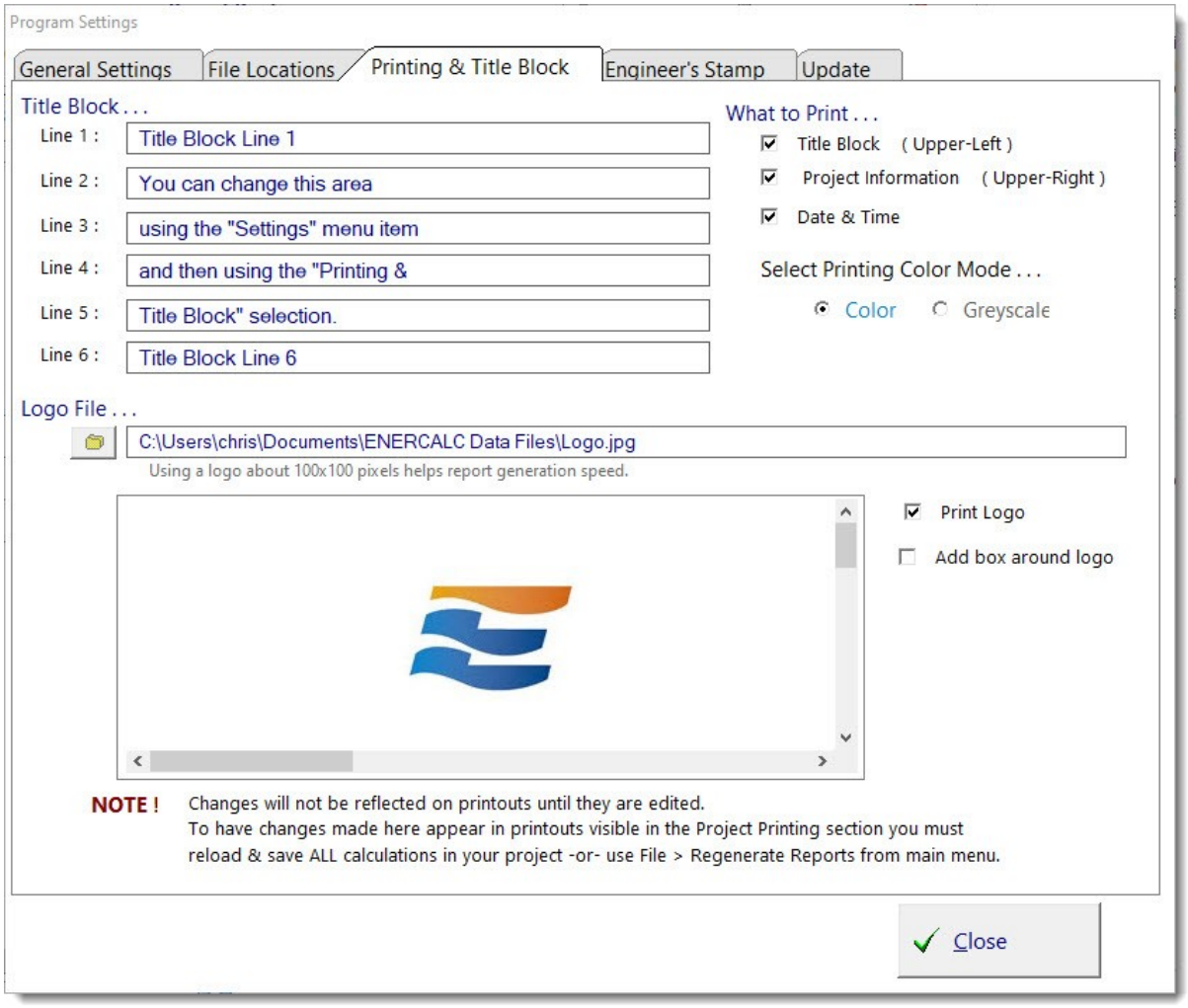
Before continuing on to add some calculations to the project, we'll demonstrate where to enter company information to be included on ALL calculations produced within **ENERCALC SEL**.

Click **Settings > Printing & Title Block** from the main menu.



© 2026 ENERCALC, LLC

The **Printing & Title Block** tab of the Program Settings dialog will be displayed. Here you can use six lines to customize the printout with your own company information. This information will be printed in the upper-left corner of ALL printouts (see sample previously).



The Logo File area of this dialog also offers the option to identify a graphic file of your company's logo and specify that it be printed as part of the Title Block. Clicking the button lets you use a file selection dialog to select a Windows Bitmap (BMP), GIF, JPG, or WMF file.

At this time, please enter some information in all six lines so you can see it printed later in this sample session. Click the [**Close**] button when finished, and then click the [**OK**] button in the Reminder dialog.

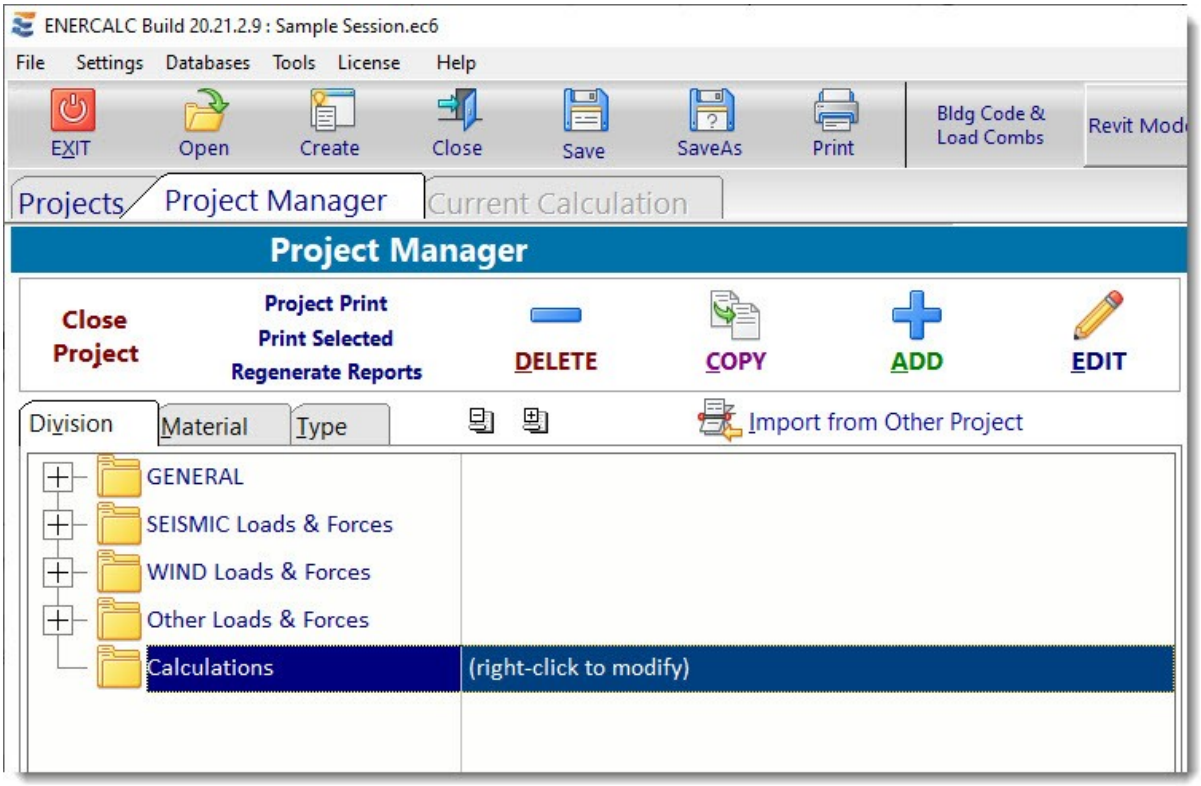
The purpose of the Reminder dialog is to alert us to the fact that some data just changed in the title block. Under normal conditions, **ENERCALC SEL** generates the report for a single calculation at a time, and it does that when the Save command is issued while a

© 2026 ENERCALC, LLC

calculation is open. This makes for a very efficient use of time by generating reports only on an as-needed basis. However, because we just made a change to title block data, it is likely that we want all reports regenerated and brought up to date at this time. This Reminder dialog guides us to the command in the main menu that manually forces all reports to be immediately regenerated.

12.6 Adding a Calculation

To add a calculation in the Project Manager, first highlight the location where you would like the new calculation to be inserted. At present we only have one Division that can receive user-created calculations, and it is named "Calculations". So we will click on "Calculations" to highlight it as shown below:

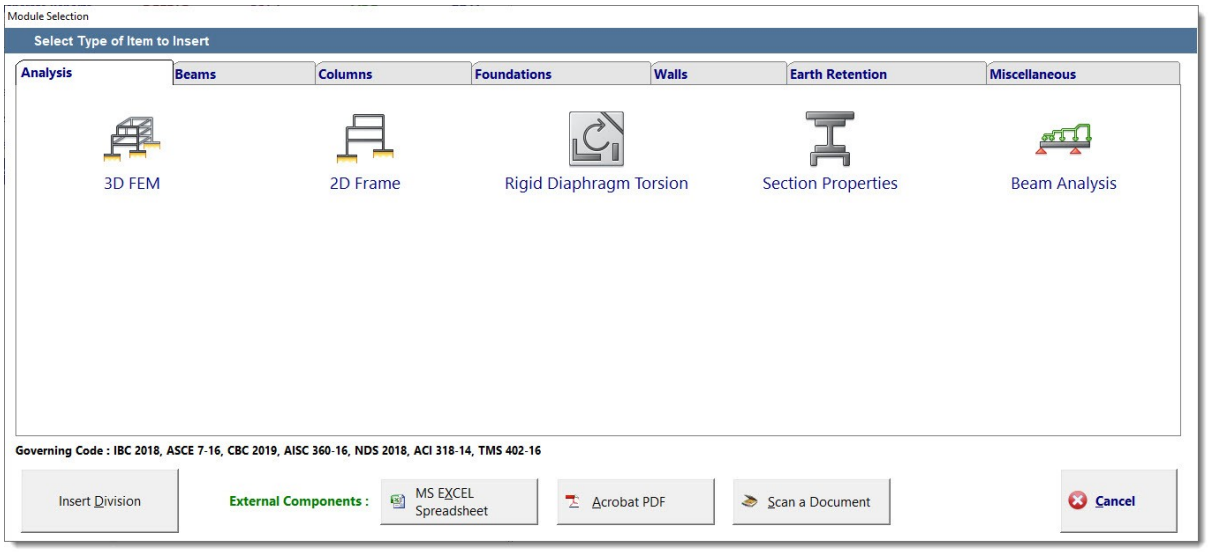


Next, click the [**Add**]

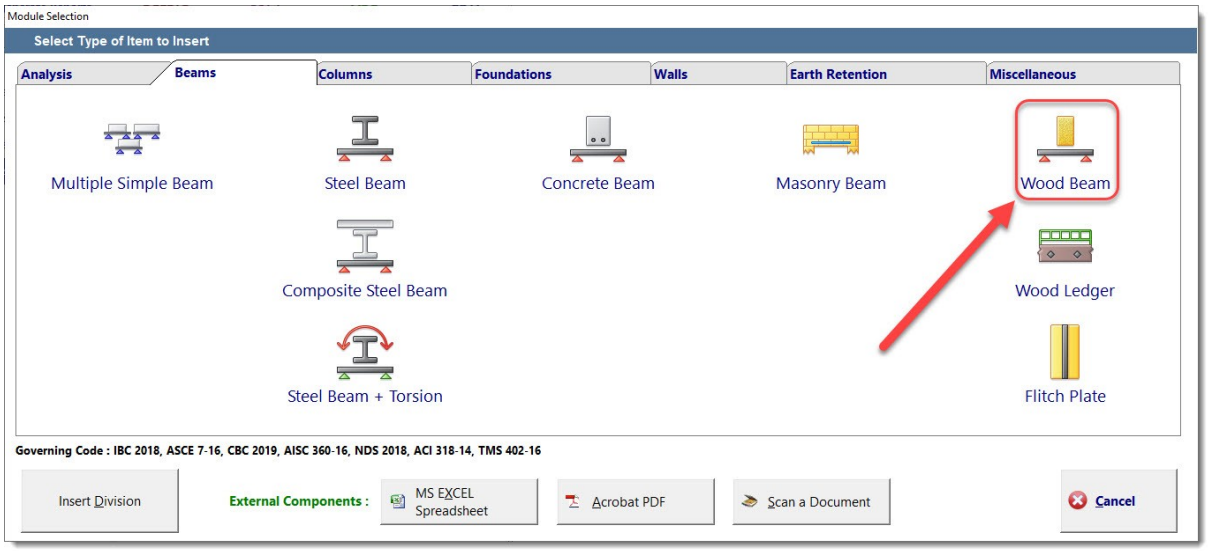
button.

© 2026 ENERCALC, LLC

The next dialog will allow you to select exactly which calculation type you wish to add:



For this sample session we will use the **Wood Beam** module, so click the Beams tab, and then click the [**Wood Beam**] button as shown below.

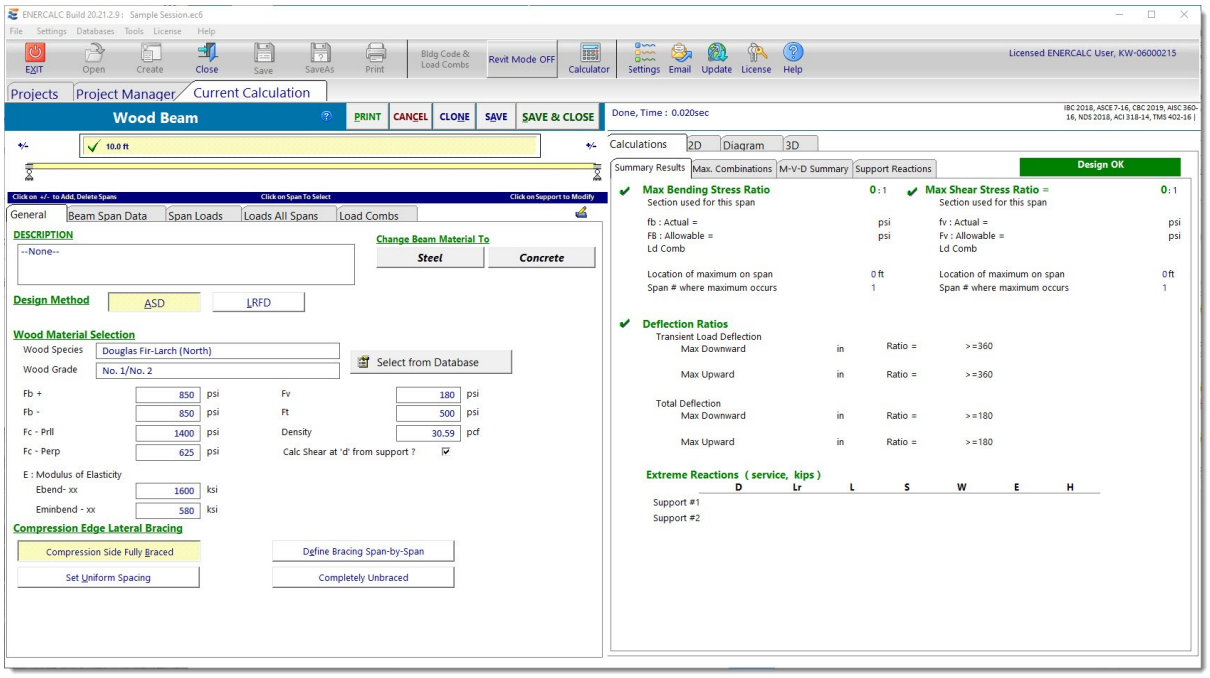


A Wood Beam calculation will be added in the Project Manager, and it will automatically be opened for editing. The screen will reconfigure itself to display the graphical user interface that is specific to the Wood Beam module.

© 2026 ENERCALC, LLC

12.7 Viewing the Calculation Screen

The screen should now appear as shown below. Please observe the following areas of the screen....they are typical of ALL calculations in **ENERCALC SEL**.



The data entry tabs on the left side of the screen are where you enter your input values.

The results and graphics tabs on the right side of the screen are where you review the calculated values, sketches, and graphs that result from calculations based on your input values.

This screen layout is consistent throughout the majority of **ENERCALC SEL** calculation modules. This enables you to display the tab of interest on the right and see how changes to input data on the left affect it.

12.8 Changing the Default Values/Settings

When you first add any calculation to a Project File, that calculation will come in with default values/settings for all user input items. If you find that you are almost always having to change certain items, it may make sense for you to revise those defaults to better suit your work. Revising the default values for any module is easy. Just follow these steps:

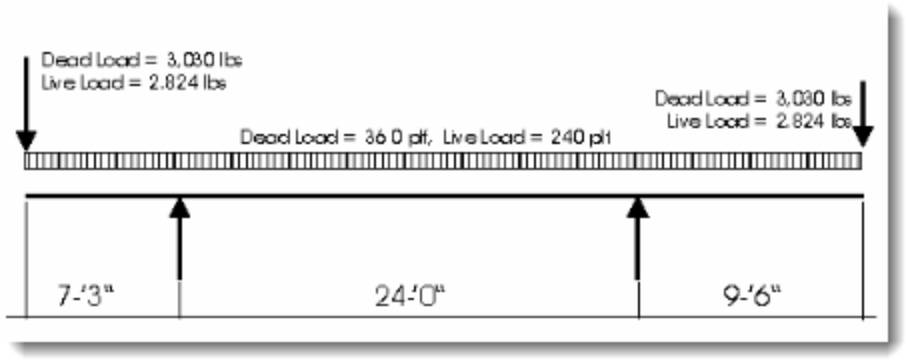
1. Add a new calculation of the module of interest to a Project File. It will come in with the existing default values/settings.
2. Revise the values/settings to suit.
3. Click Settings > Default Values > Save.

© 2026 ENERCALC, LLC

This will save all of your current settings/values as the new defaults for that particular module. From that point onward, any time you add a new calculation of that module type, it will automatically be populated with the default values/settings that you just established.

12.9 Entering Data

Below is a diagram for a wood roof beam we wish to design.



The following steps will guide you through the process of setting the span lengths and entering the loads as shown. (In a subsequent step we will select the member section size and reference design values from the internal databases.)

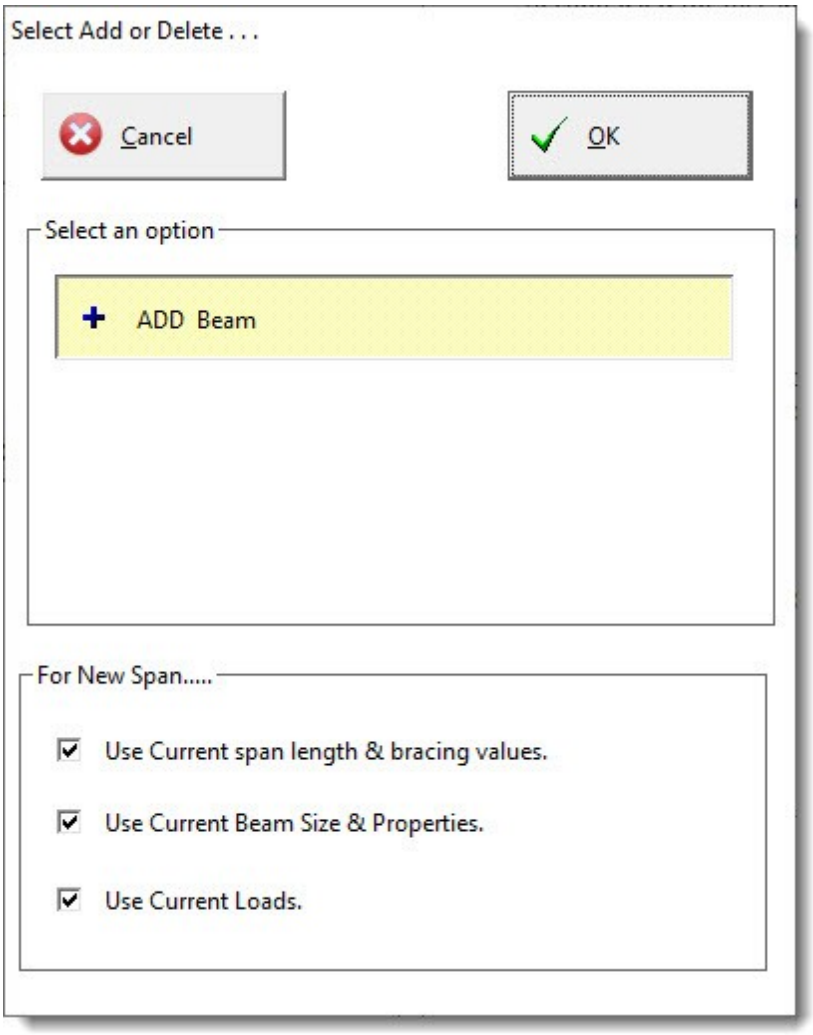
The top portion of the screen should currently look like this:

The first thing we will do is add the left cantilever. To do this, click the [**+/-**] icon at the left end of the beam as shown bubbled below:

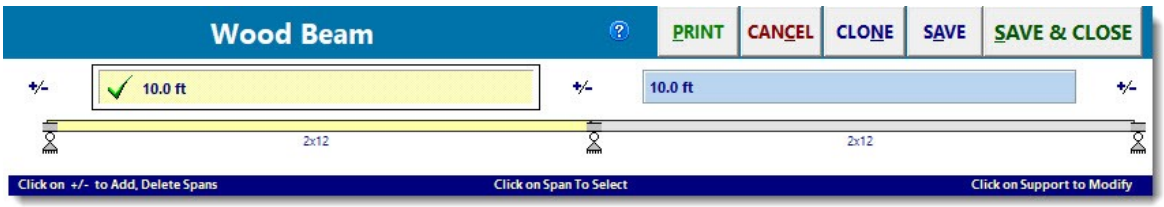
This opens the dialog named Select Add or Delete. It is currently set up to allow us to add a new beam span, which is exactly what we want to do. There are some additional settings offered at the bottom of the dialog in the form of checkboxes. We don't need to worry about these at this point, but it's good to know that they are available, because they can offer some time-saving conveniences in certain modeling situations.

© 2026 ENERCALC, LLC

See the image of the Select Add or Delete dialog shown below:

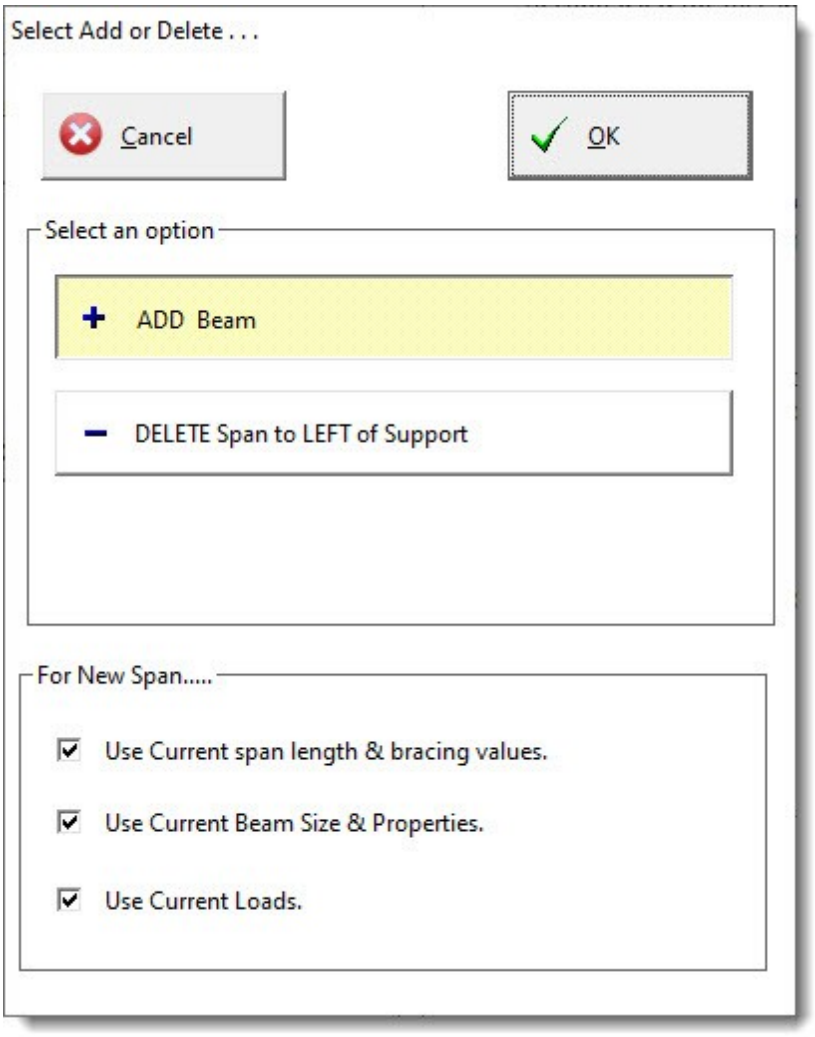


For our purposes, all we need to do is click the [**OK**] button. This inserts a new beam span to the left of the original beam as shown below:



© 2026 ENERCALC, LLC

At this point, don't be concerned about the fact that the new span is not a cantilever and that it is not the intended length. We'll correct both of those items in just a moment. For now, let's repeat a similar process to add a new beam span on the right. First click the [**+/-**] icon that appears above the right-most support. The following dialog appears:



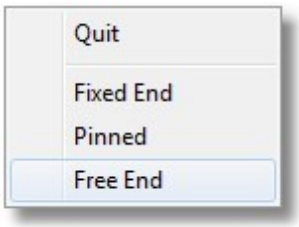
This is very similar to the Select Add or Delete dialog we saw before, except that it now has an extra option to delete the span that is to the left of the support. Although we don't want to do this, it's good to know that this is the way that you would delete a span if necessary. What we want to do is add a new beam span, and that is the option that is currently selected (highlighted in yellow), so we can just click the [**OK**] button.

© 2026 ENERCALC, LLC

The screen should now display a 3-span continuous beam as shown below:

Now that we have three spans to work with, let's modify the support conditions to create the cantilevered spans at the left and the right. To do this, click on the left-most support icon shown bubbled below:

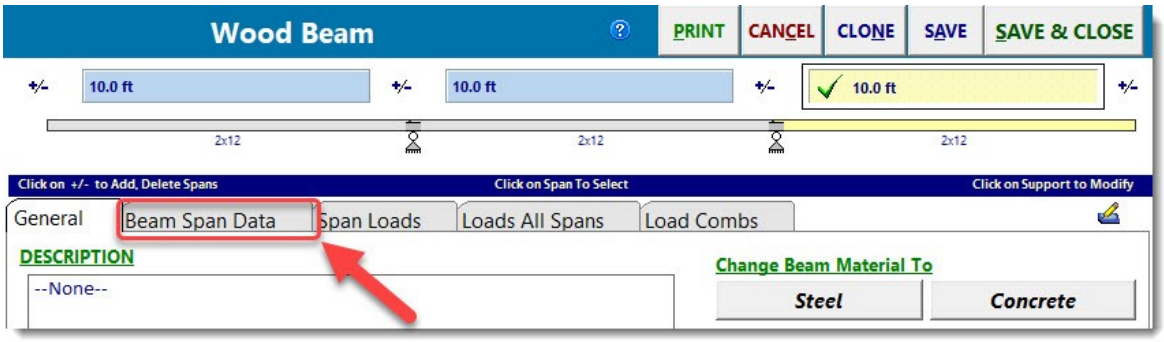
This will open the following pop-up menu that offers many options for defining the support condition at that location:



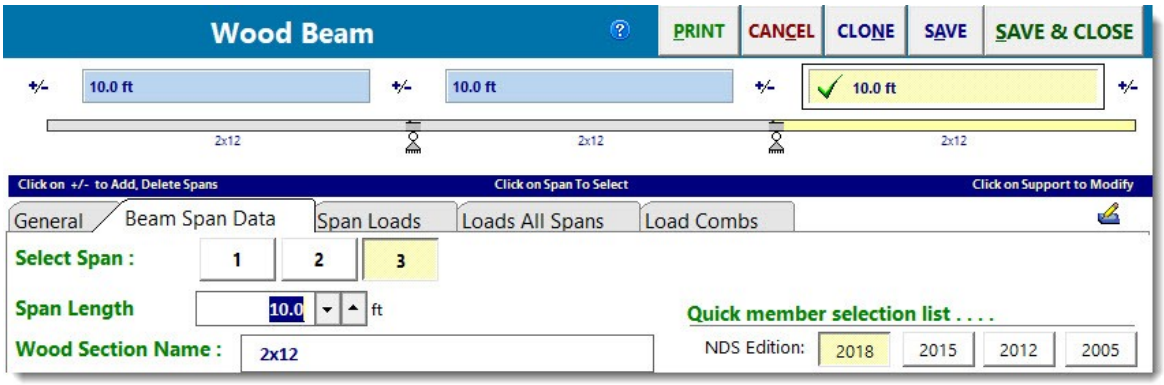
Click the **Free End** option to remove all restraint from the left end, creating a cantilever. Repeat the same process at the right-most support, but leave the remaining two supports as they are currently configured. (They represent pinned conditions that prevent translation in all directions but offer no moment restraint.) The result will be the double-cantilevered beam as shown below:

© 2026 ENERCALC, LLC

Now that the support conditions have been established correctly, we can wrap up this exercise by modifying the span lengths to match the problem definition. Start by clicking the Beam Span Data tab shown bubbled below:



The screen layout will change to display the Beam Span Data tab as shown below:

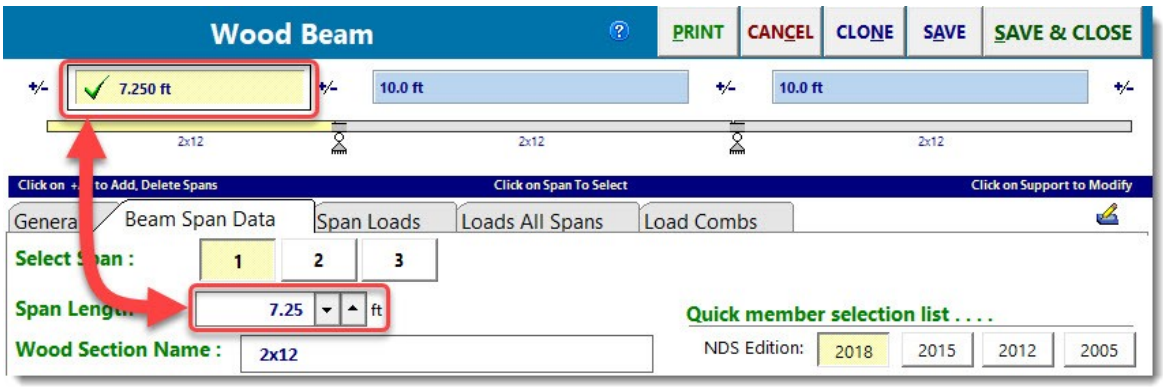


Now, we just need to adjust the span lengths for each of the three spans.

First, note that the graphical depiction of the beam helps us to make the association between the three individual spans and the span numbers. Click any of the three span number buttons in the Select Span area and notice that a corresponding span becomes highlighted in the large graphic at the top of the view. Using this method, we can confirm that the spans are logically ordered from number one on the left to number three on the right. (Note that you can also click the highlighted rectangles shown above each beam span, and they act in exactly the same way that the numbered buttons do.)

© 2026 ENERCALC, LLC

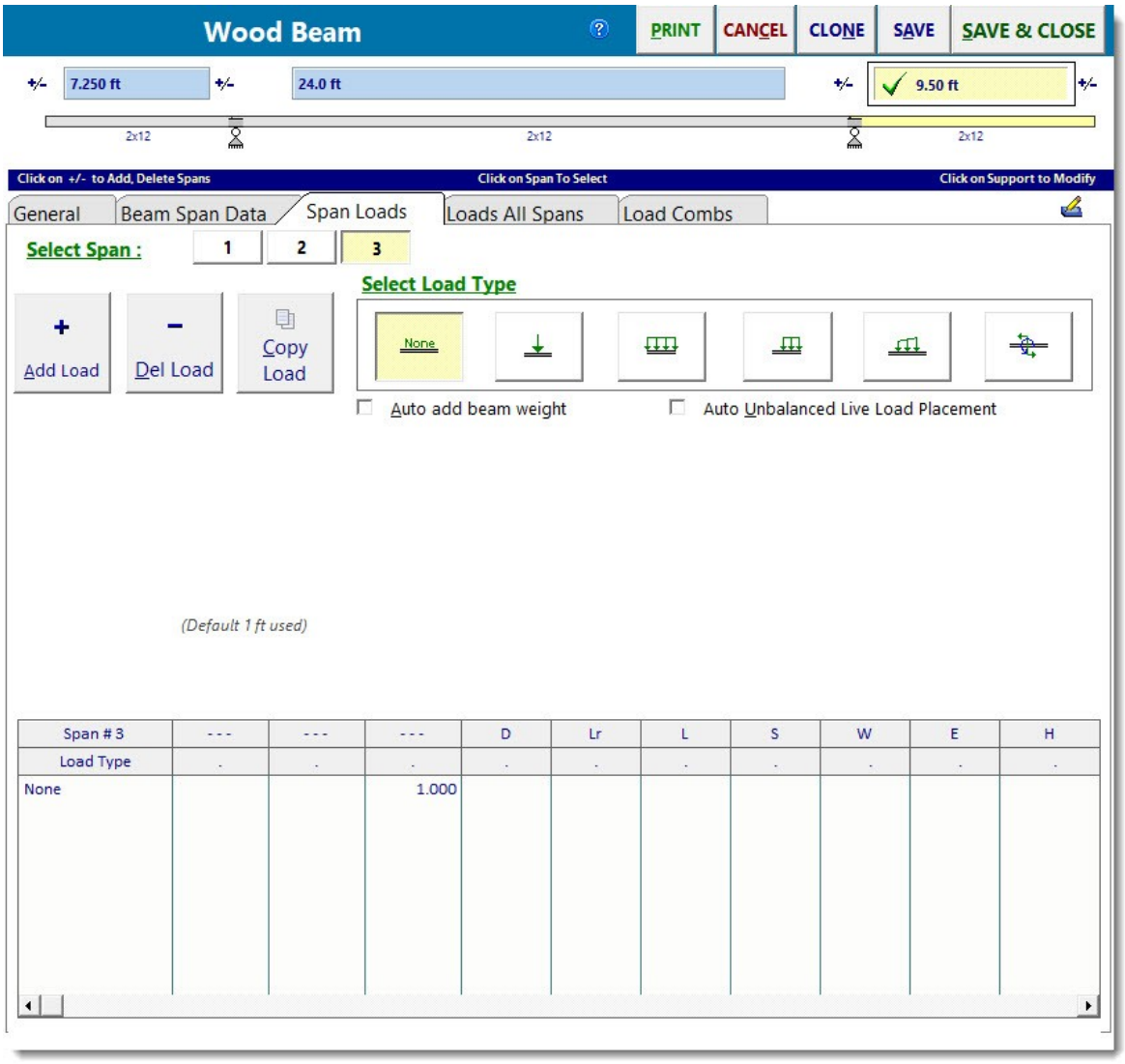
So to finish things off, click the button for span [**1**] and verify that the left span becomes highlighted. According to the problem statement, this span is supposed to be 7'-3", so you can highlight the value that is presently displayed in the Span Length field, manually edit it to say 7.25, and then press the [**Tab**] key to enter that value. The display will update to show the proper span length for the left beam span as shown below:



Repeat the same process to set the Span Length for the middle span to 24'-0" and to set the Span Length for the right span to 9'-6", at which time the display should appear as shown below:

© 2026 ENERCALC, LLC

The next step will be to enter the loads on the beam. Let's begin by entering the concentrated load at the free end of the left beam span. Click the Span Loads tab to display the following screen:

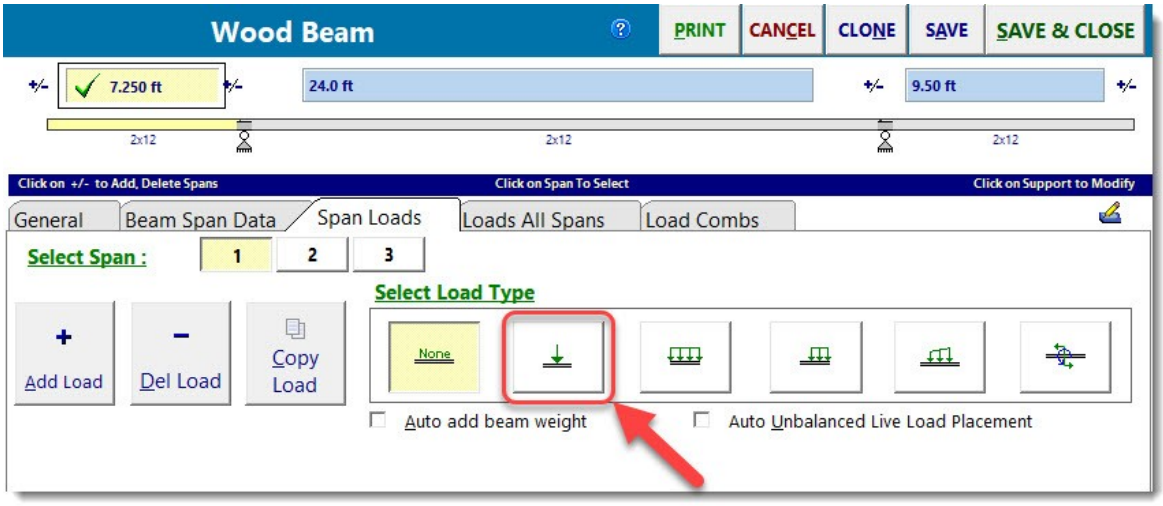


This screen contains tools to apply loads to a selected span at a time. (Shortly, we will see a convenience tool that will allow us to apply loads to ALL spans of a beam at one time.)

Since we want to start by entering the concentrated load at the free end of the left beam span, click the [**1**] button in the Select Span area to put the focus on Span 1.

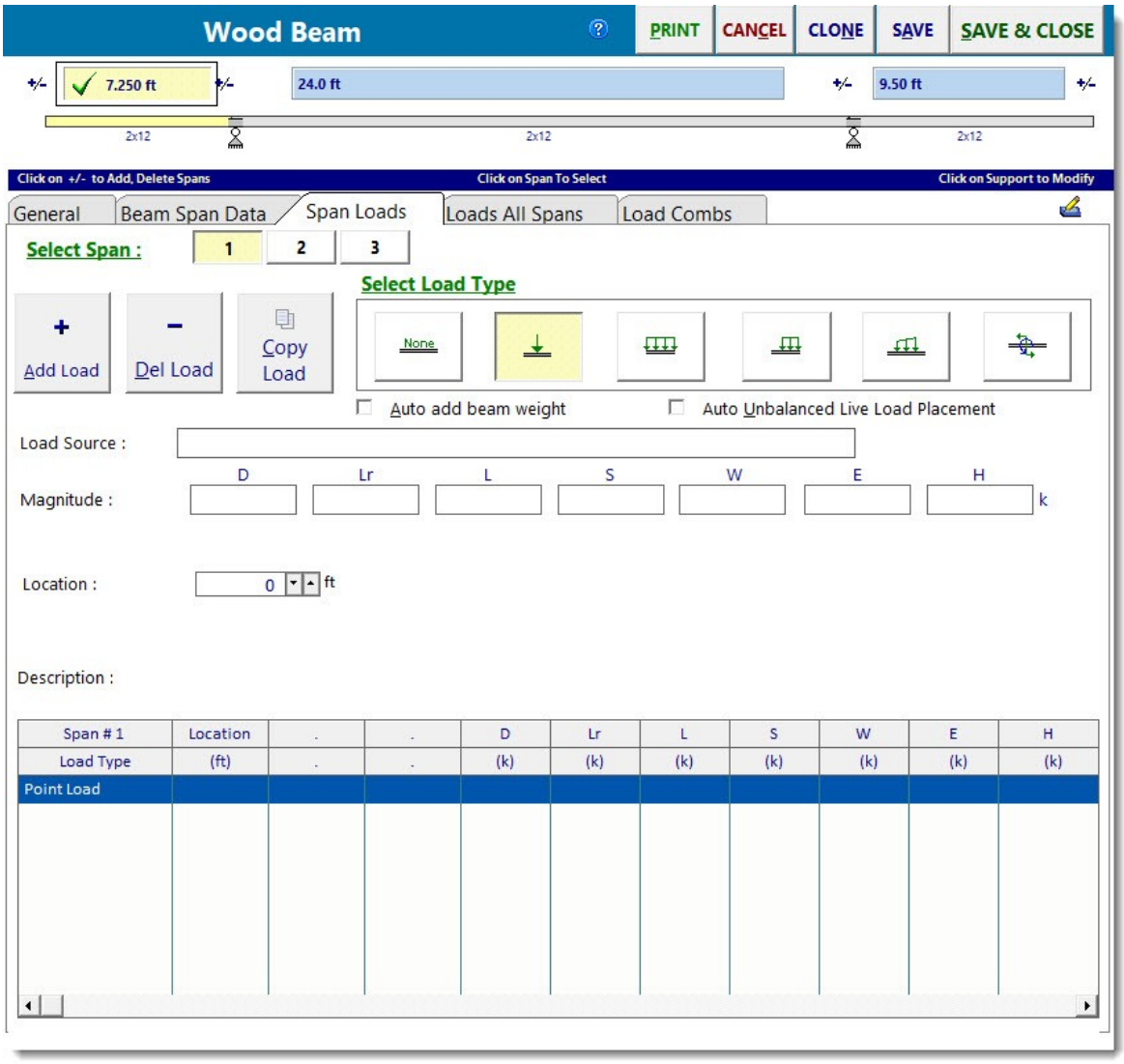
© 2026 ENERCALC, LLC

The blue highlighted band in the table now represents the first load on Span 1. But it currently indicates Load Type = None, because we have not defined the load yet. Click the button indicated in the screen capture below to identify the load as being a concentrated load:



© 2026 ENERCALC, LLC

This will reconfigure the screen with the appropriate input fields to fully define the magnitude, direction, and location of the concentrated load as shown below:



Select the checkbox for the Auto add beam weight option. This will trigger the program to determine the self weight of the beam that we eventually choose, and apply it as dead load when the analysis is performed.

Select the checkbox for the Auto Unbalanced Live load Placement option. This will initiate the generation of additional loading conditions to ensure that all possible permutations of live load placement are properly examined. Note that selecting this option can lead to longer analysis times because of the number of different conditions that must be analyzed. However, with the double-cantilevered beam configuration, it would be important for us to have the module examine all of these potential loading permutations.

© 2026 ENERCALC, LLC

The Load Source input field provides a location where you can add some descriptive text for your own use to identify this particular load item. The use of this field is optional, so for this sample session we will leave it blank.

When defining the magnitude of the concentrated load, note that the module is set up to receive this kind of load in units of kips, and notice that it has input fields for the common load cases, such as Dead, Live, Snow, etc.

Click in the input field for Dead, enter a magnitude of 3.03, and then press the [**Tab**] key. This enters the Dead load magnitude and moves the cursor to the next input field.

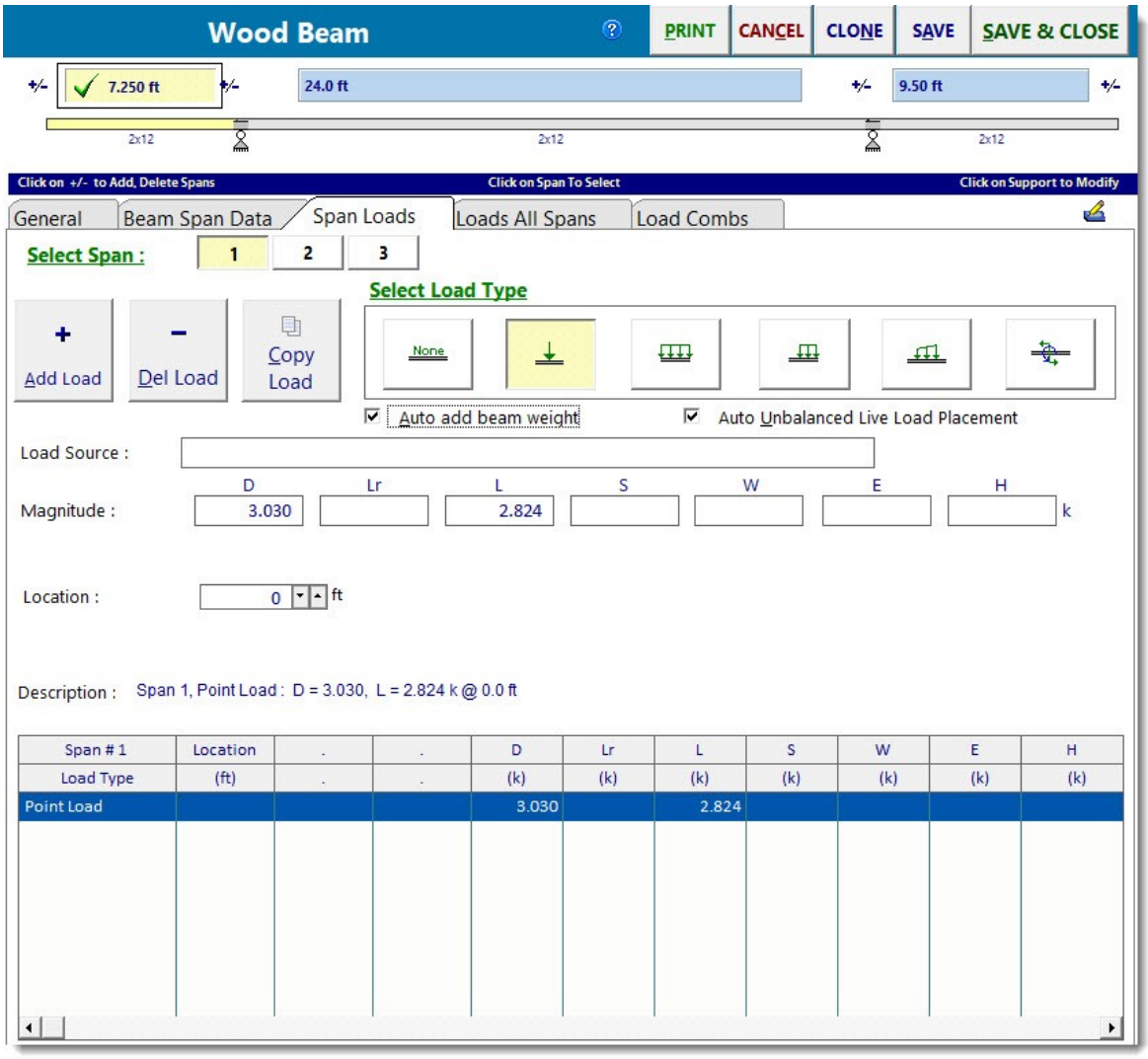
Note: *For convenience this module is configured such that loads with positive magnitudes are assumed to act downward, since the majority of the load items are typically gravity loads of some sort.*

The cursor is currently in the Lr input field, which represents Roof Live load. We don't need to enter any magnitude for Roof Live, so press the [**Tab**] key once again to place the cursor on the Live load input field. Enter a magnitude of 2.824, and then press the [**Tab**] key.

The final step in specifying this concentrated load is to identify its position using the Location input field. Always specify the distance from the left end of this span to the position of the concentrated load. In this case, the correct value is zero, which also happens to be the default value for the Location field, so the Location field can be left as-is.

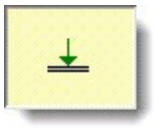
© 2026 ENERCALC, LLC

The screen should now appear as shown below:



The next step will be to define the other concentrated load, which occurs at the free end of the cantilever on the right end of the beam, by following a very similar process.

Click the [**3**] button in the Select Span area to put the focus on Span 3.

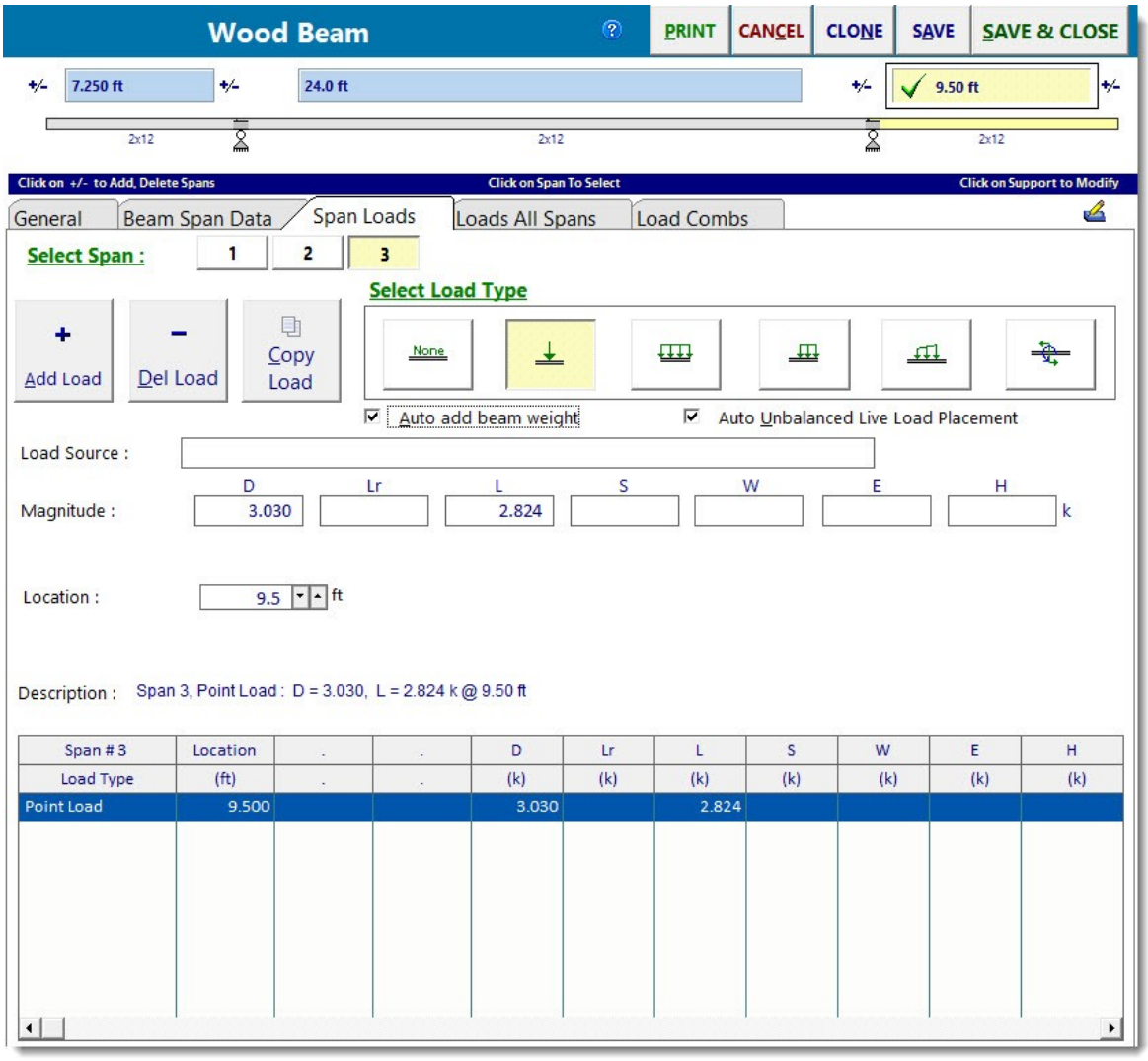
Click the  button to identify the load as being a concentrated load.

The magnitudes of this load are the same as those used at the opposite end of the beam, so click in the input field for Dead, enter a magnitude of 3.03, and then press the [**Tab**] key twice.

In the Live load input field enter a magnitude of 2.824, and then press the [**Tab**] key.

© 2026 ENERCALC, LLC

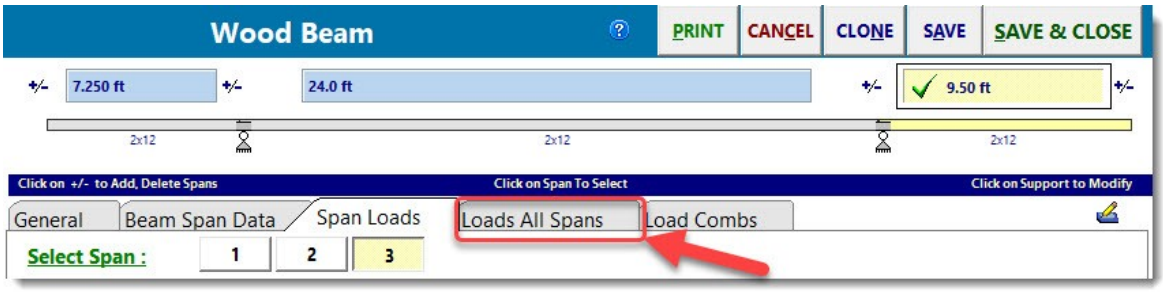
This concentrated load is to be positioned at the extreme right end of this span, so click in the Location input field, enter a distance of 9.5 (9'-6" from the support at the left end of Span 3), and then press the [**Tab**] key. The screen should now appear as shown below:



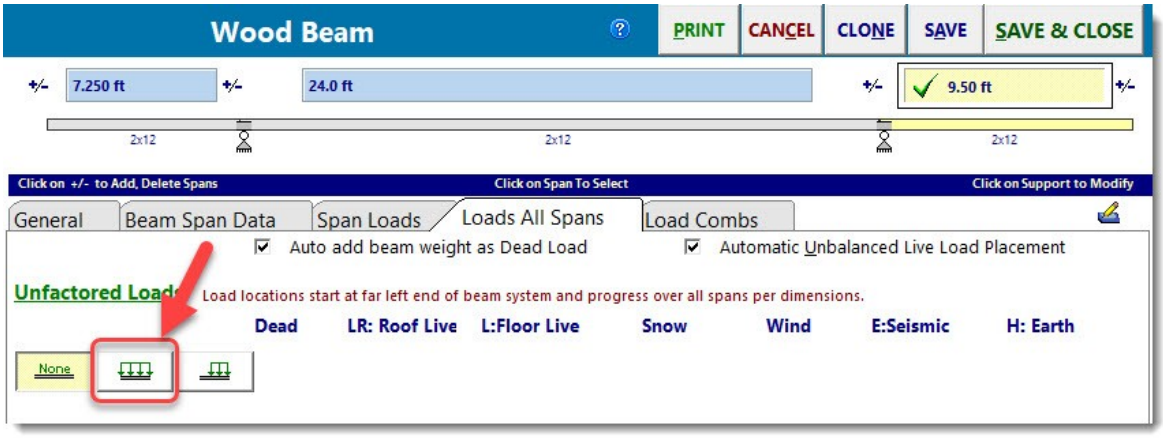
The next step in defining loads is to apply the uniformly distributed load across all three spans. This *could* be done by using the Uniformly Distributed Loading Type here on the Span Loads tab. But this would require some repetition, because the Span Loads tab is specifically set up to facilitate the application of loads to one span at a time.

© 2026 ENERCALC, LLC

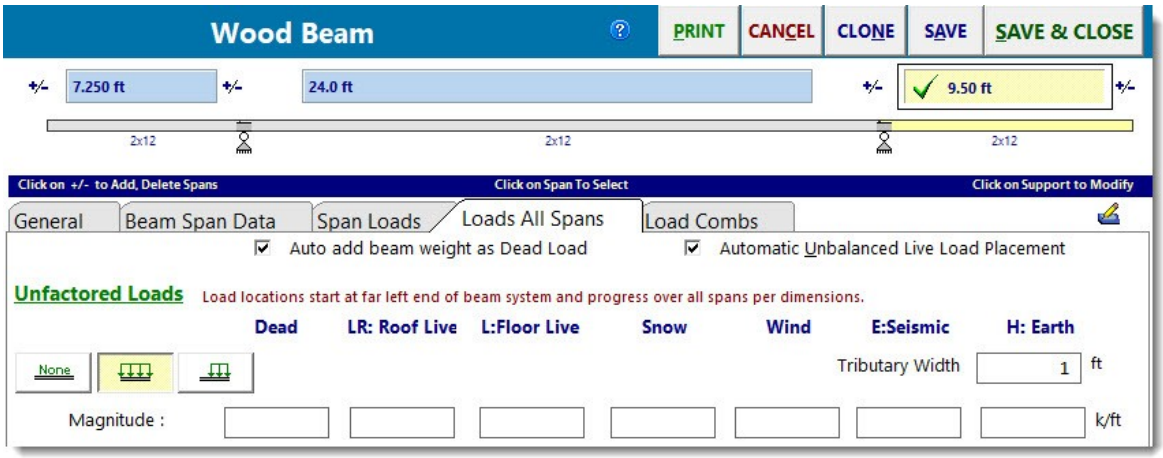
To apply this load in a more efficient manner, click the Loads All Spans tab shown bubbled below:



The screen will reconfigure itself to display different tools for applying various types of loads to all spans of the beam. The one that is most appropriate for our application is the one shown bubbled below:



Click that tool and note that the screen displays input fields for Tributary Width and load magnitudes for the various Load Cases as shown below:



© 2026 ENERCALC, LLC

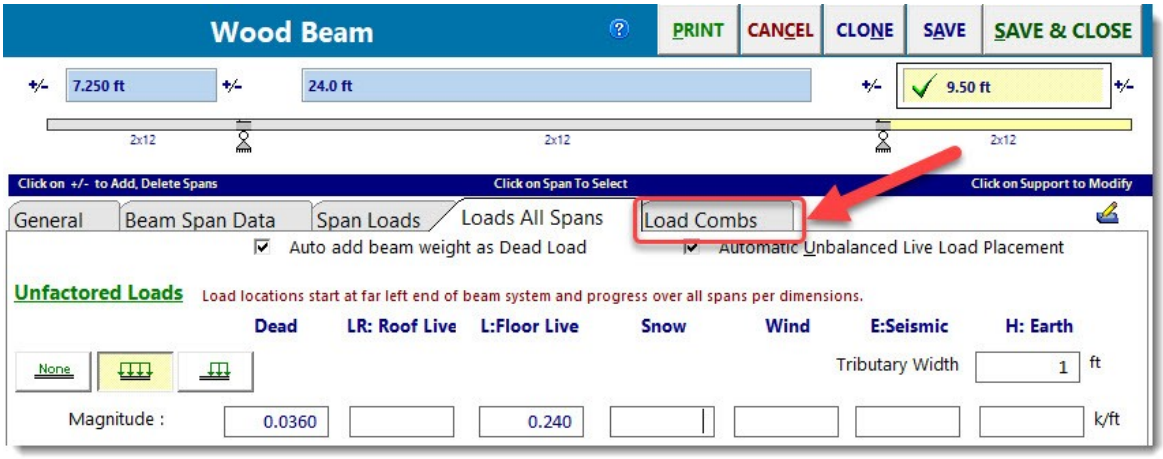
Note: When using this tool, you can either leave the Tributary Width input field at its default value of 1.0, and then enter the magnitudes for the various load cases in units of kips per foot, or you can specify a Tributary Width magnitude *other* than 1.0 foot, and then enter the magnitudes for the various load cases in units of kips per square foot, which will be multiplied by the Tributary Width to determine the magnitude of the effective line load in units of kips per foot.

For the purposes of our exercise, we will leave the Tributary Width input field at its default value of 1.0.

Click in the Dead load input field, enter a magnitude of 0.036, and press the [**Tab**] key twice.

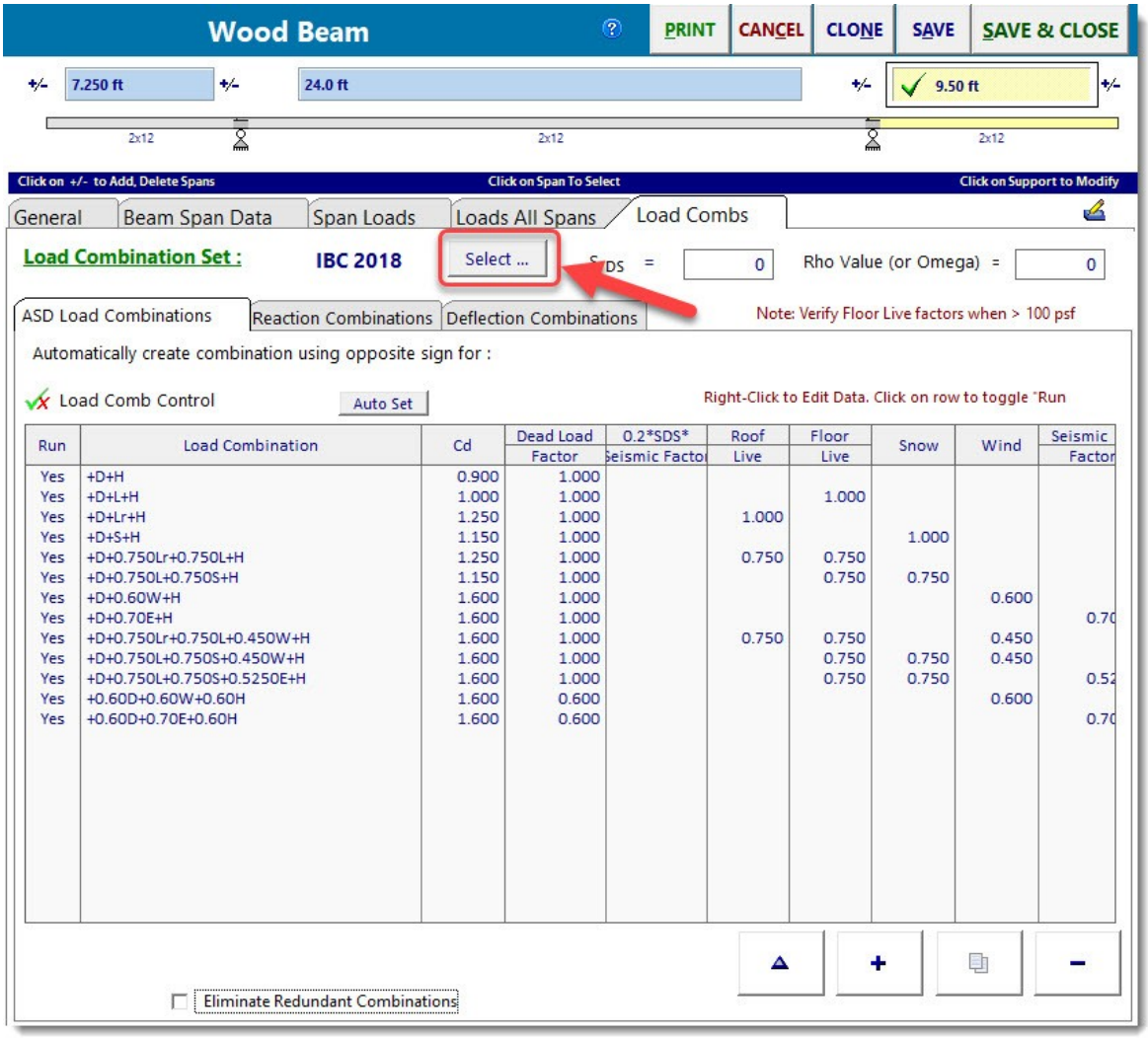
In the Live load input field, enter a magnitude of 0.24, and press the [**Tab**] key once more.

The last item to take care of is to observe the load combinations that will be used for design. Click the Load Combinations tab indicated below:



© 2026 ENERCALC, LLC

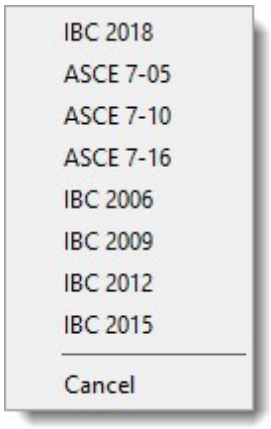
This will display the Load Combinations screen. On this screen, you can select the desired Load Combination set. In this case, we are already referencing the IBC 2018 load combinations, which is our goal. But if that selection did need to be changed, click the "Select" button indicated in the screen capture below:



Run	Load Combination	Cd	Dead Load Factor	0.2*SDS* Seismic Factor	Roof Live	Floor Live	Snow	Wind	Seismic Factor
Yes	+D+H	0.900	1.000						
Yes	+D+L+H	1.000	1.000			1.000			
Yes	+D+Lr+H	1.250	1.000		1.000				
Yes	+D+S+H	1.150	1.000				1.000		
Yes	+D+0.750Lr+0.750L+H	1.250	1.000		0.750	0.750			
Yes	+D+0.750L+0.750S+H	1.150	1.000			0.750	0.750		
Yes	+D+0.60W+H	1.600	1.000					0.600	
Yes	+D+0.70E+H	1.600	1.000						0.70
Yes	+D+0.750Lr+0.750L+0.450W+H	1.600	1.000		0.750	0.750		0.450	
Yes	+D+0.750L+0.750S+0.450W+H	1.600	1.000			0.750	0.750	0.450	
Yes	+D+0.750L+0.750S+0.5250E+H	1.600	1.000			0.750	0.750		0.52
Yes	+0.60D+0.60W+0.60H	1.600	0.600					0.600	
Yes	+0.60D+0.70E+0.60H	1.600	0.600						0.70

© 2026 ENERCALC, LLC

This will open a pop-up menu offering choices of load combination sets that exist on the current machine, similar to the one shown below:



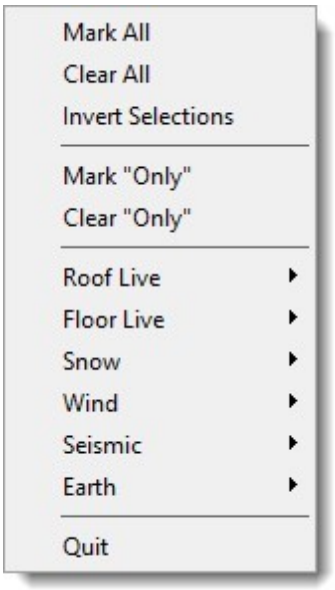
Choose the option named IBC 2018. This will set up the calculation to use load combinations based on IBC 2018.

Next, click the

button, and confirm by clicking the [Yes] button. This is a very convenient option for wood designs, because it automatically sets the C_D value based on the shortest duration load case within each load combination. Note the various values of C_D that result.

© 2026 ENERCALC, LLC

The final step in this section is to ensure that all of the listed load combinations will be used in the analysis and design. To do this, click the Load Combination Control button button. This will open a pop-up menu offering options for marking and clearing the Run status of all of the load combinations in the list, as shown below:



Simply click the [**Mark All**] option, and all of the load combinations in the list will be marked for use in the analysis and design.

12.10 Selecting Sections and Materials from Built-in Databases

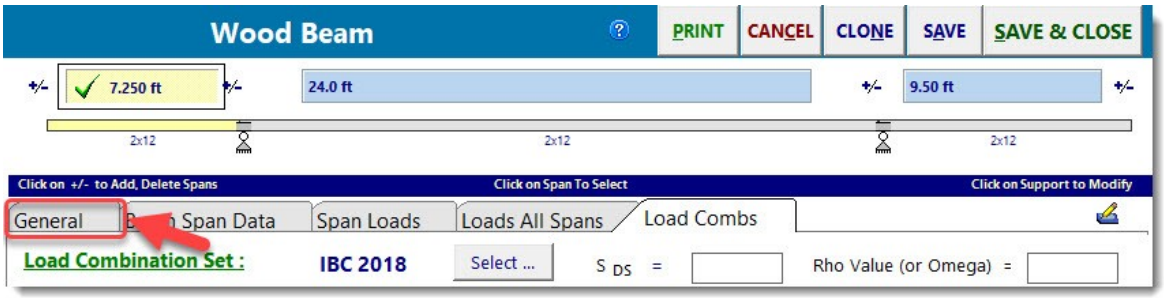
In the previous topic you experienced part of the advantage of using **ENERCALC SEL**...simplified data entry. The input screens naturally guide you through all of the necessary items, and you are simply entering data into a form.

We still have two critical items to specify.....the design properties of the wood member to use and the physical size of the beam.

First, let's retrieve the reference design values. For this sample we want to use a Douglas Fir glued-laminated beam of type 24F-V8.

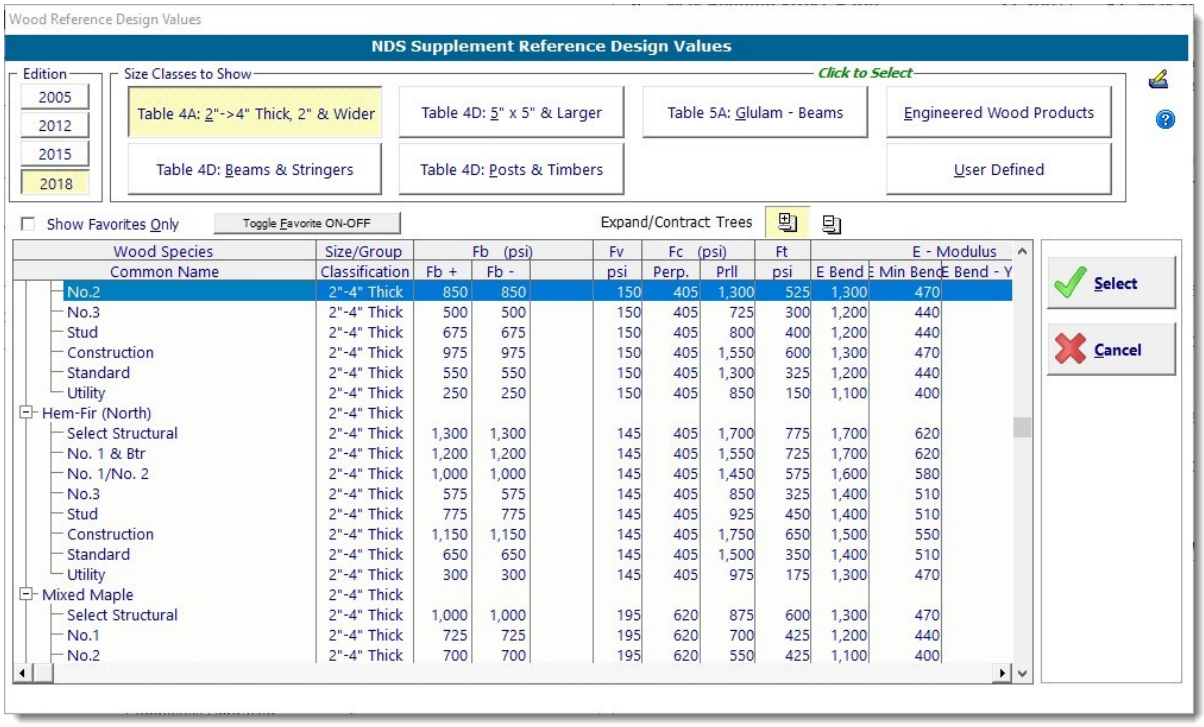
© 2026 ENERCALC, LLC

Click the General tab shown below:



In the Wood Material Selection category, click the [**Select from Database**] button

. The Wood Reference Design Values database will be displayed as shown below:

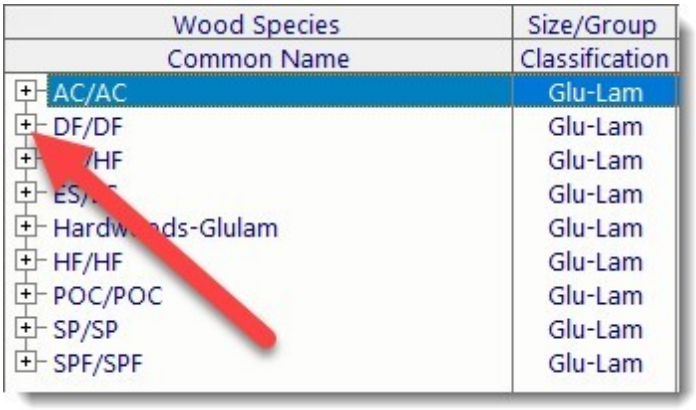


We want to display ONLY the glued-laminated sections for Douglas Fir. In the Size Classes to Show category, click the [**Table 5A Glulam - Beams**] button:

© 2026 ENERCALC, LLC

Click the [**Contract Trees**] button to collapse the display of Wood Species Common Names.

Click the [**+**] in front of the DF/DF item to expand the Douglas Fir list:

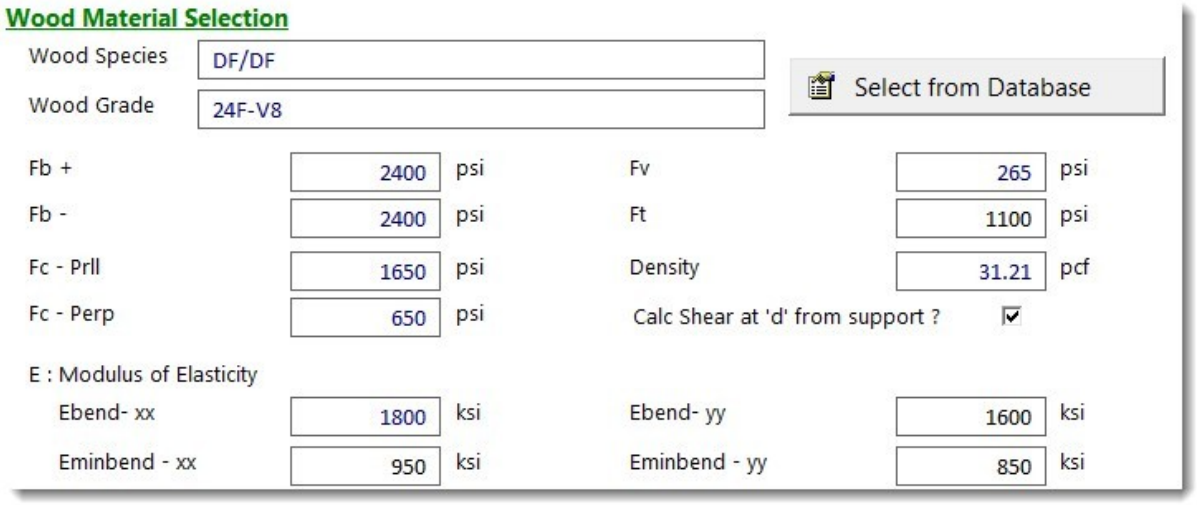


The result of these choices will be a list of stress grades as shown....

Wood Species / Common Name	Size/Group Classification	Fb (psi) Fb +	Fb -	Fy	Fv psi	Fc (psi) Perp.	Prll	Ft psi	E - Modulus E Bend	E Min Bend	E Bend - YY
AC/AC	Glu-Lam										
DF/DF	Glu-Lam										
16F-V3	Glu-Lam	1,600	1,250	1,450	265	560	1,500	975	1,500	790	1,500
16F-V6	Glu-Lam	1,600	1,600	1,450	265	560	1,600	1,000	1,600	850	1,500
16F-E3	Glu-Lam	1,600	1,200	1,400	265	560	1,600	975	1,600	850	1,500
16F-E6	Glu-Lam	1,600	1,600	1,550	265	560	1,600	1,000	1,600	850	1,500
20F-V3	Glu-Lam	2,000	1,450	1,450	265	560	1,550	1,000	1,600	850	1,500
20F-V7	Glu-Lam	2,000	2,000	1,450	265	650	1,600	1,050	1,600	850	1,600
20F-E3	Glu-Lam	2,000	1,200	1,400	265	560	1,600	1,050	1,700	900	1,600
20F-E6	Glu-Lam	2,000	2,000	1,550	265	560	1,650	1,150	1,700	900	1,600
24F-V4	Glu-Lam	2,400	1,850	1,450	265	650	1,650	1,100	1,800	950	1,600
24F-V8	Glu-Lam	2,400	2,400	1,550	265	650	1,650	1,100	1,800	950	1,600
24F-E4	Glu-Lam	2,400	1,450	1,400	265	650	1,700	1,100	1,800	950	1,700
24F-E13	Glu-Lam	2,400	2,400	1,750	265	650	1,700	1,250	1,800	950	1,700
24F-E18	Glu-Lam	2,400	2,400	1,550	265	650	1,700	975	1,800	950	1,700
26F-V1	Glu-Lam	2,600	1,950	1,850	265	650	1,850	1,350	2,000	1,060	1,800
26F-V2	Glu-Lam	2,600	2,600	1,850	265	650	1,850	1,350	2,000	1,060	1,800
DF/HF	Glu-Lam										
ES/ES	Glu-Lam										

© 2026 ENERCALC, LLC

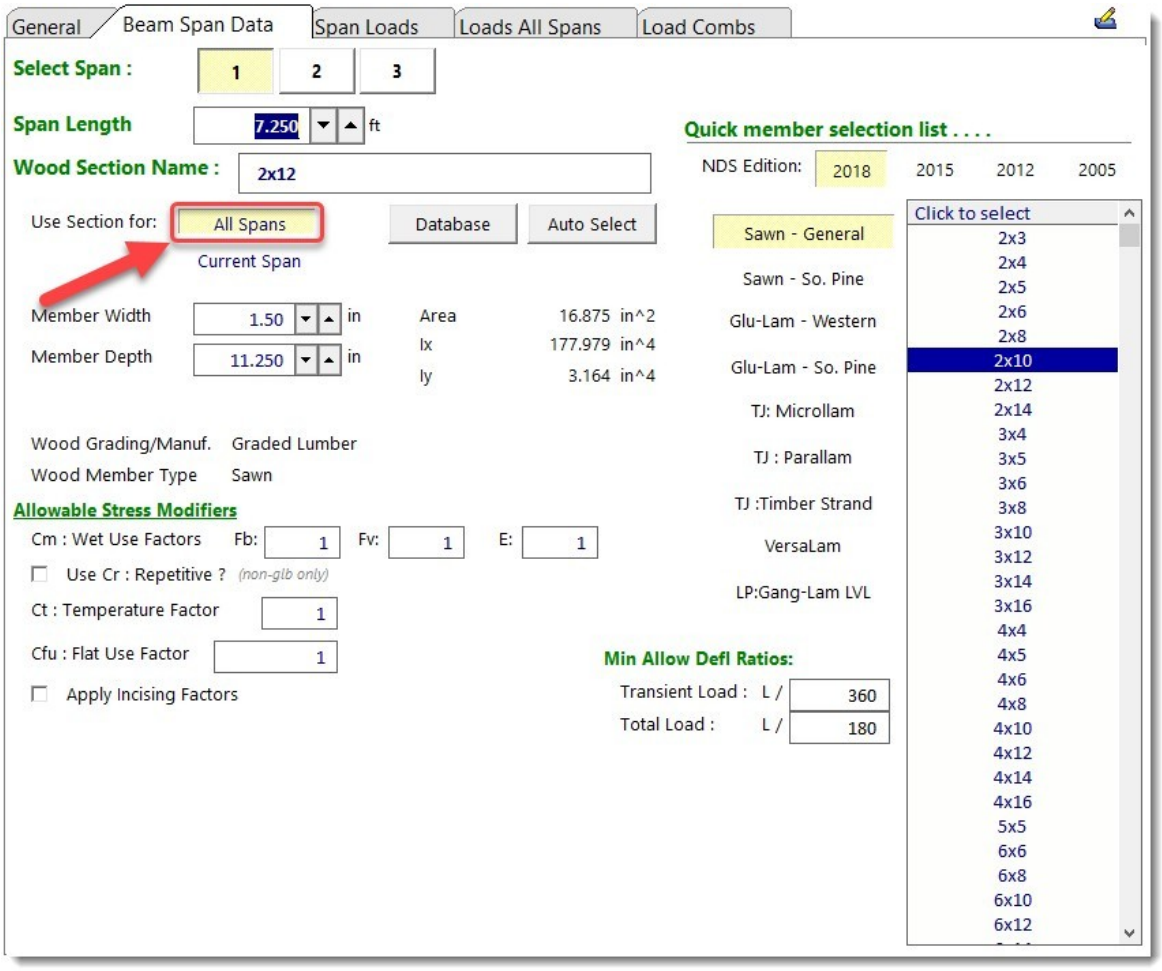
Click the **24F-V8** item as shown, and then click the [**Select**] button. The stress information area on the General tab will immediately be updated to reflect your selection:



Next, let's retrieve the section property data for a 6.75" x 30" glulam beam

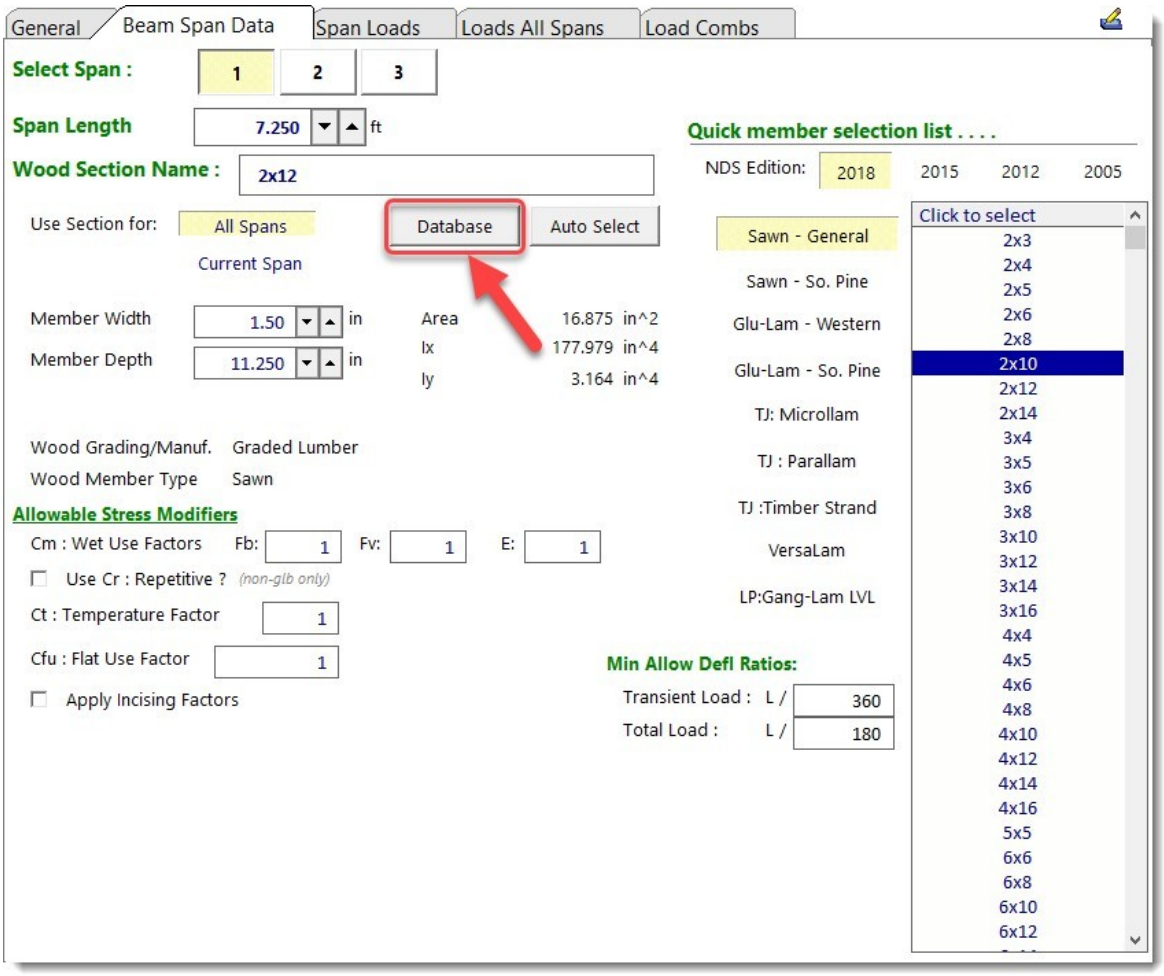
© 2026 ENERCALC, LLC

Click the Beam Span Data tab and make sure that the [**All Spans**] option is selected as shown below:



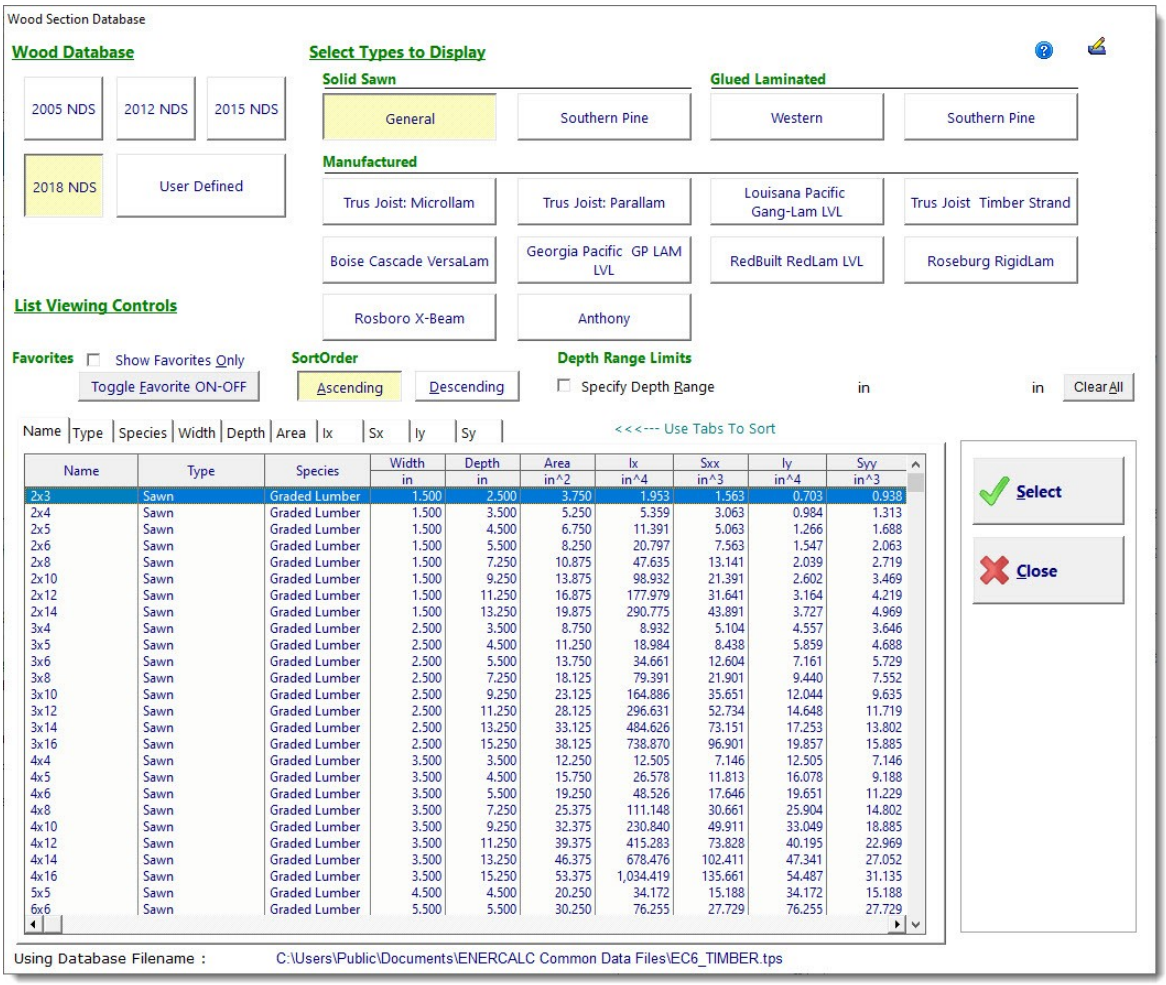
© 2026 ENERCALC, LLC

Then click the [**Database**] button shown bubbled below:



© 2026 ENERCALC, LLC

The Wood Section Database will be displayed as shown below:

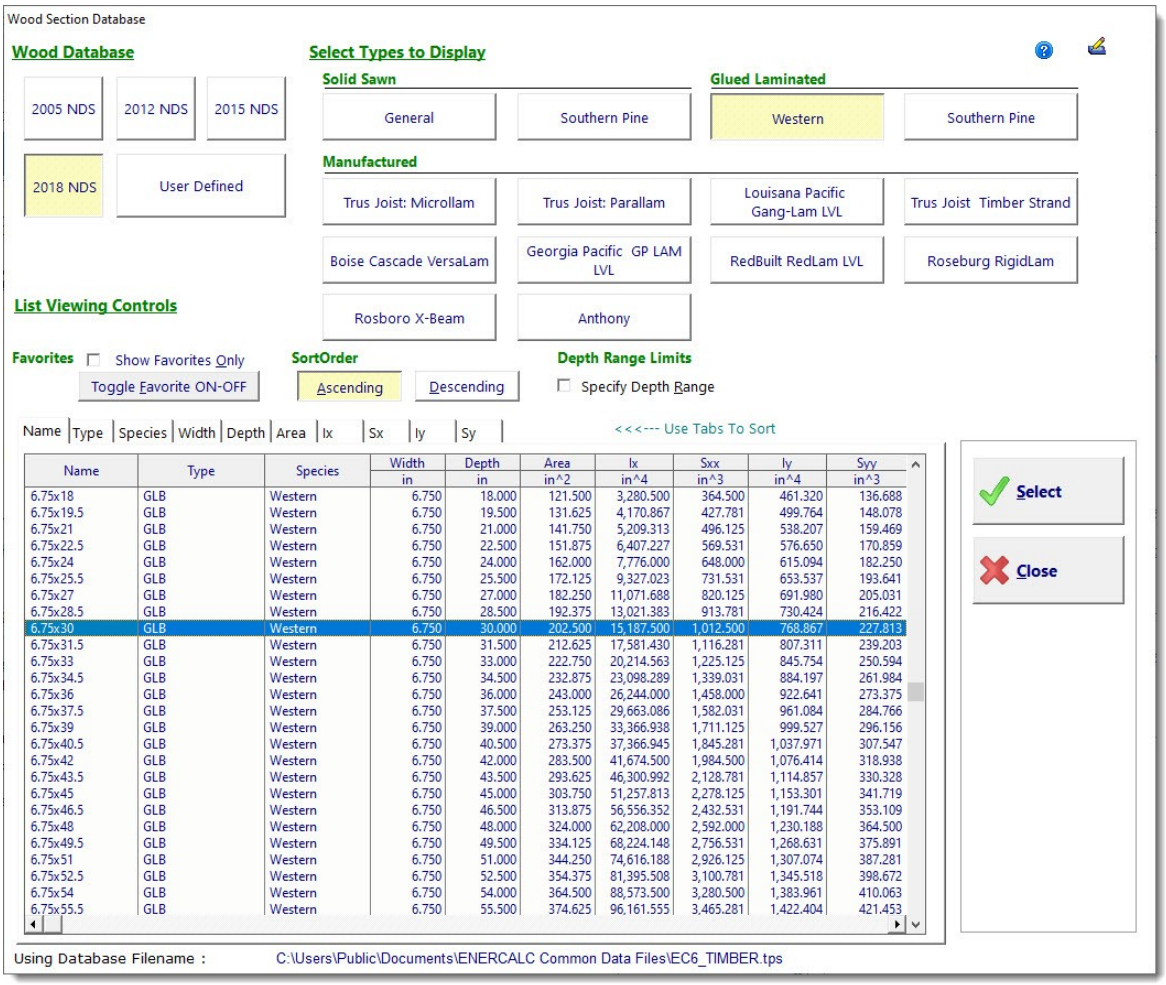


Now click the

button in the Glued Laminated category so that only those members are displayed. The result of this choice will be a list of glulam beams.

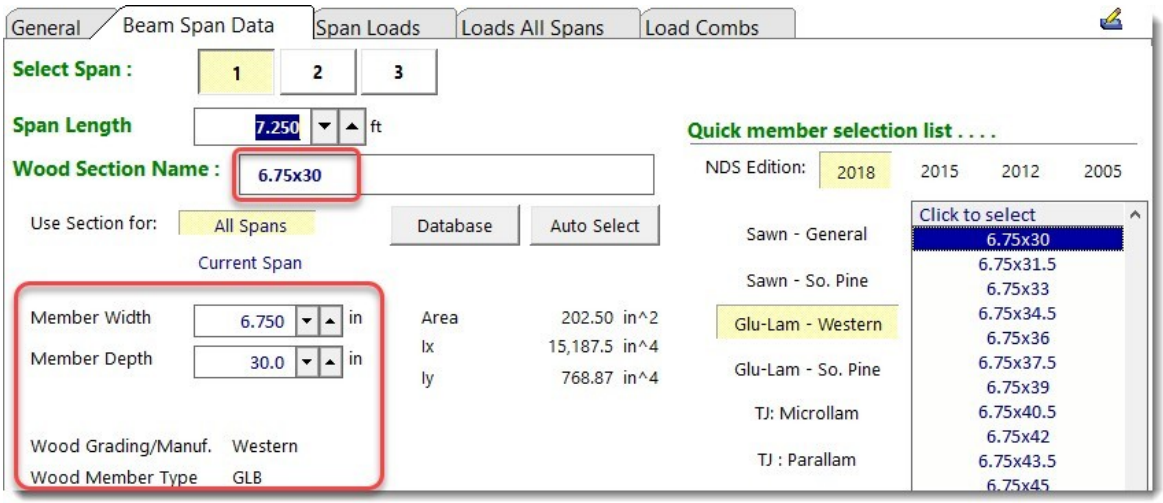
© 2026 ENERCALC, LLC

Scroll down through the database and highlight the 6.75" x 30" beam as shown below:



Name	Type	Species	Width in	Depth in	Area in^2	Ix in^4	Sxx in^3	Iy in^4	Syy in^3
6.75x18	GLB	Western	6.750	18.000	121.500	3,280.500	364.500	461.320	136.688
6.75x19.5	GLB	Western	6.750	19.500	131.625	4,170.867	427.781	499.764	148.078
6.75x21	GLB	Western	6.750	21.000	141.750	5,209.313	496.125	538.207	159.469
6.75x22.5	GLB	Western	6.750	22.500	151.875	6,407.227	569.531	576.650	170.859
6.75x24	GLB	Western	6.750	24.000	162.000	7,776.000	648.000	615.094	182.250
6.75x25.5	GLB	Western	6.750	25.500	172.125	9,327.023	731.531	653.537	193.641
6.75x27	GLB	Western	6.750	27.000	182.250	11,071.688	820.125	691.980	205.031
6.75x28.5	GLB	Western	6.750	28.500	192.375	13,021.383	913.781	730.424	216.422
6.75x30	GLB	Western	6.750	30.000	202.500	15,187.500	1,012.500	768.867	227.813
6.75x31.5	GLB	Western	6.750	31.500	212.625	17,581.430	1,116.281	807.311	239.203
6.75x33	GLB	Western	6.750	33.000	222.750	20,214.563	1,225.125	845.754	250.594
6.75x34.5	GLB	Western	6.750	34.500	232.875	23,098.289	1,339.031	884.197	261.984
6.75x36	GLB	Western	6.750	36.000	243.000	26,244.000	1,458.000	922.641	273.375
6.75x37.5	GLB	Western	6.750	37.500	253.125	29,663.086	1,582.031	961.084	284.766
6.75x39	GLB	Western	6.750	39.000	263.250	33,366.938	1,711.125	999.527	296.156
6.75x40.5	GLB	Western	6.750	40.500	273.375	37,366.945	1,845.281	1,037.971	307.547
6.75x42	GLB	Western	6.750	42.000	283.500	41,674.500	1,984.500	1,076.414	318.938
6.75x43.5	GLB	Western	6.750	43.500	293.625	46,300.992	2,128.781	1,114.857	330.328
6.75x45	GLB	Western	6.750	45.000	303.750	51,257.813	2,278.125	1,153.301	341.719
6.75x46.5	GLB	Western	6.750	46.500	313.875	56,556.352	2,432.531	1,191.744	353.109
6.75x48	GLB	Western	6.750	48.000	324.000	62,208.000	2,592.000	1,230.188	364.500
6.75x49.5	GLB	Western	6.750	49.500	334.125	68,224.148	2,756.531	1,268.631	375.891
6.75x51	GLB	Western	6.750	51.000	344.250	74,616.188	2,926.125	1,307.074	387.281
6.75x52.5	GLB	Western	6.750	52.500	354.375	81,395.508	3,100.781	1,345.518	398.672
6.75x54	GLB	Western	6.750	54.000	364.500	88,573.500	3,280.500	1,383.961	410.063
6.75x55.5	GLB	Western	6.750	55.500	374.625	96,161.555	3,465.281	1,422.404	421.453

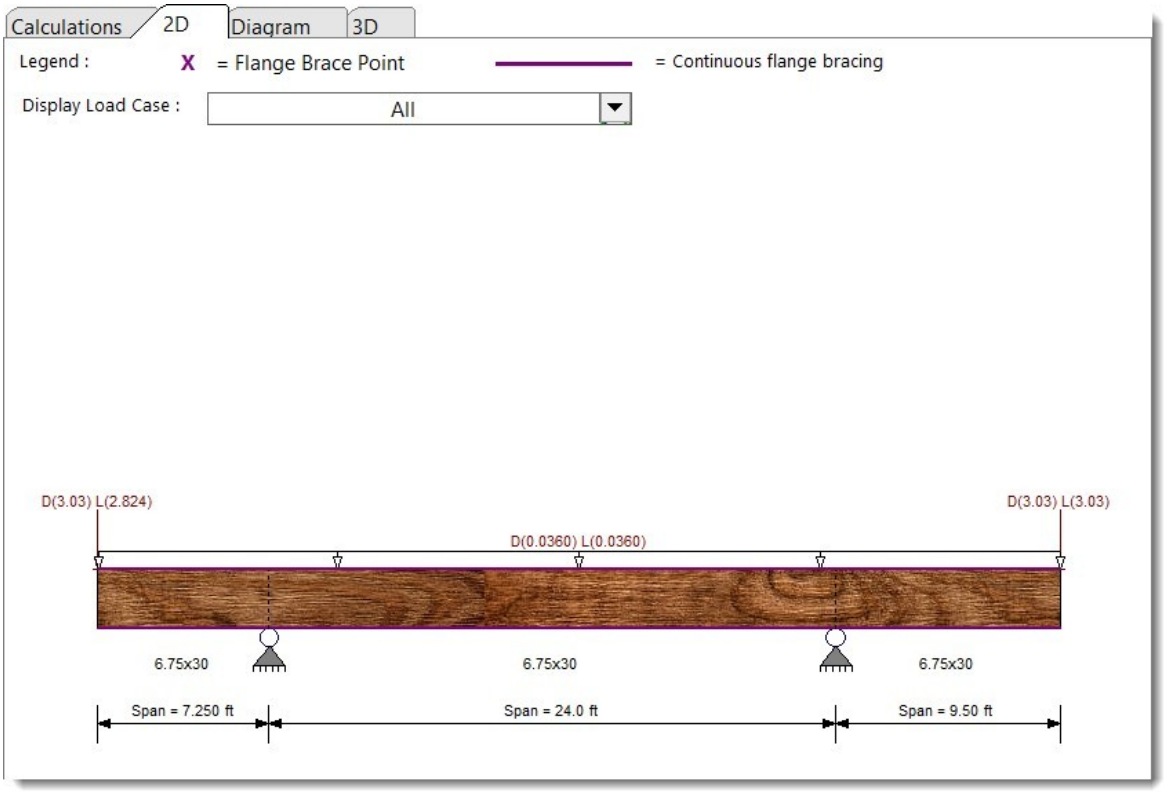
Click the button. The beam size information area on the Beam Span Data tab will immediately be updated to reflect your selection:



© 2026 ENERCALC, LLC

12.11 Displaying a Sketch

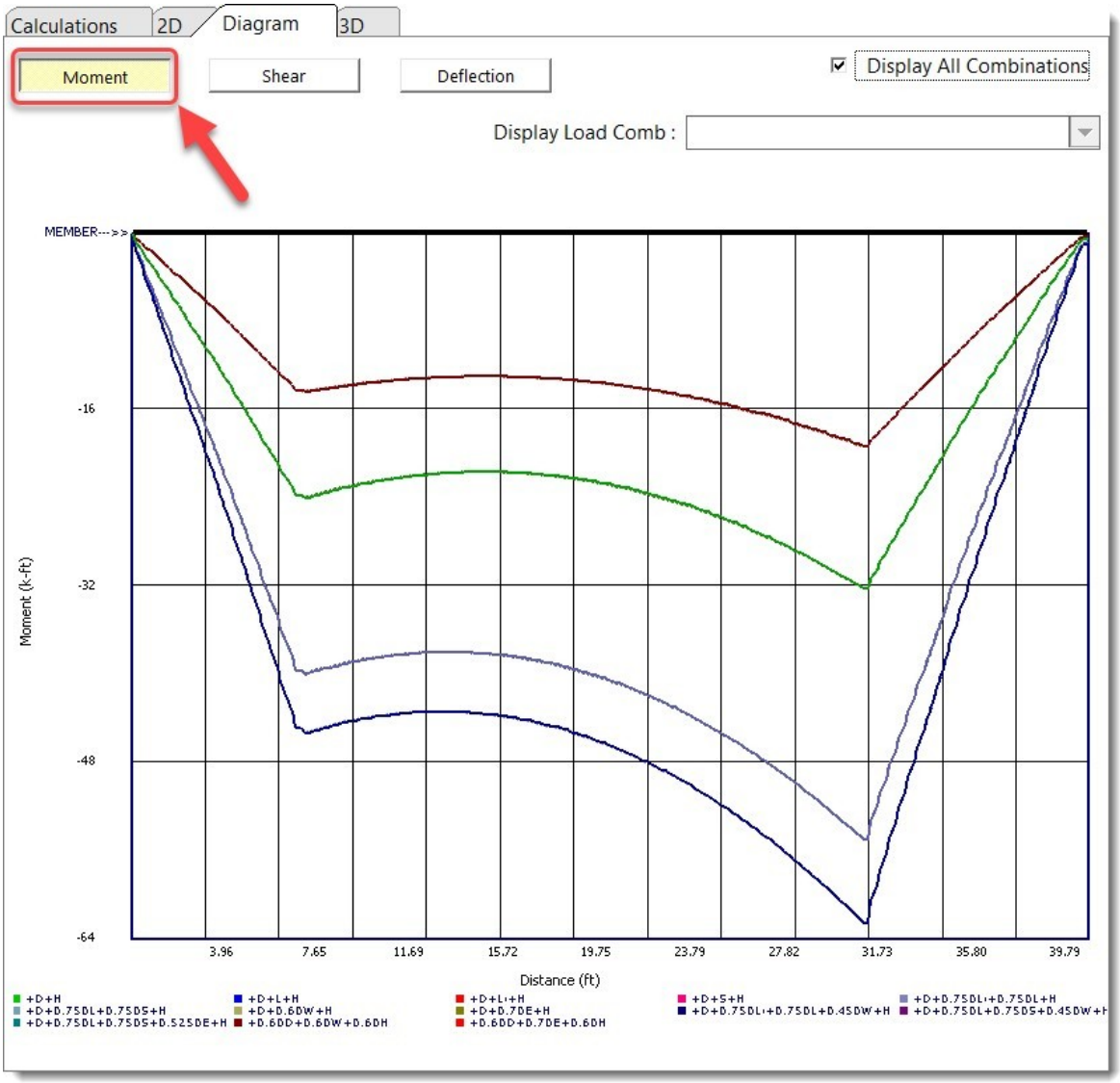
We now have a calculation with span lengths, loads, allowable stresses, and a beam size. It's time to review the sketch to confirm that we have entered the critical span and load information correctly. Click the 2D tab, and the working area of your screen will be replaced with a simple diagrammatic representation of the beam.

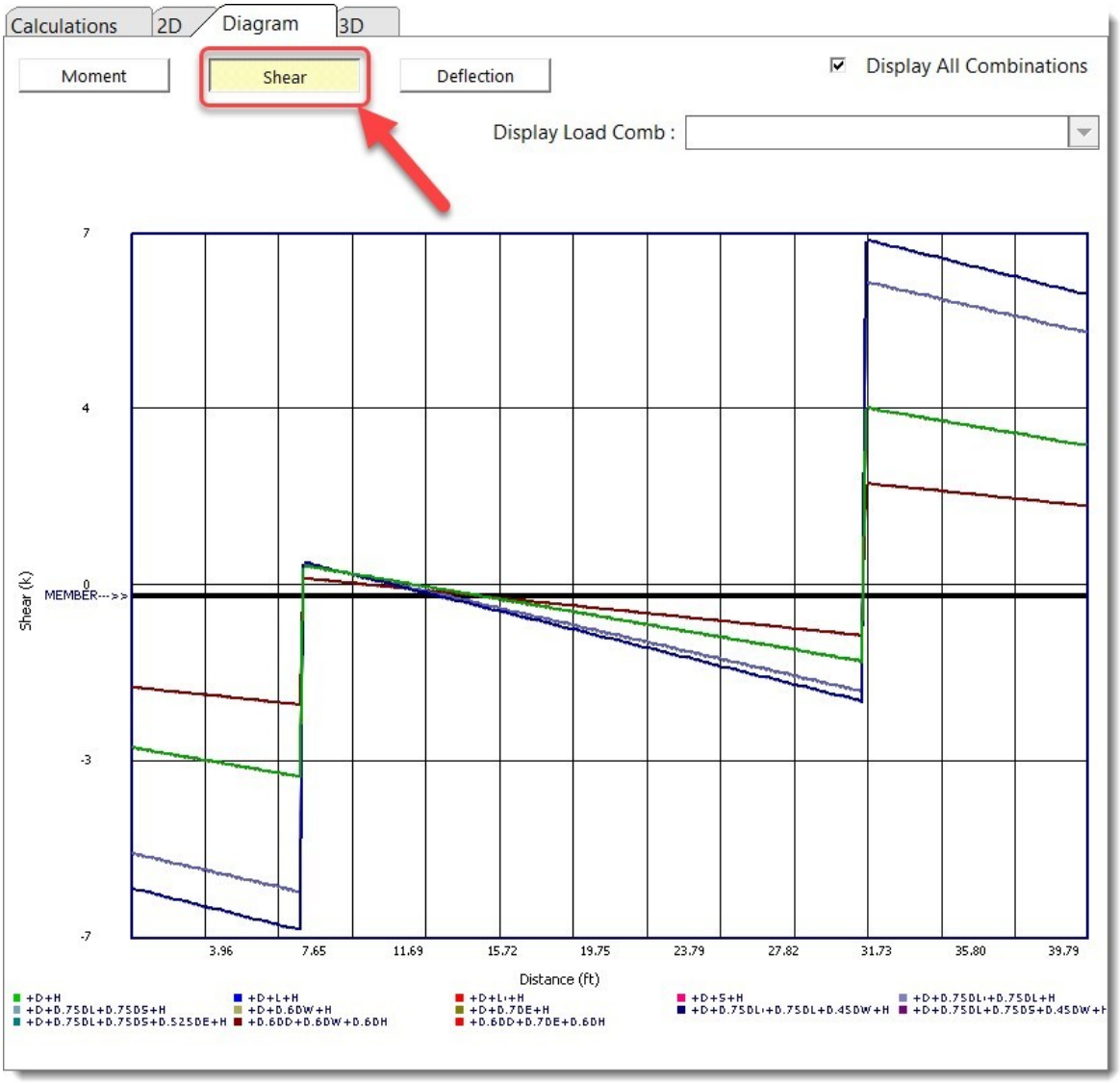


© 2026 ENERCALC, LLC

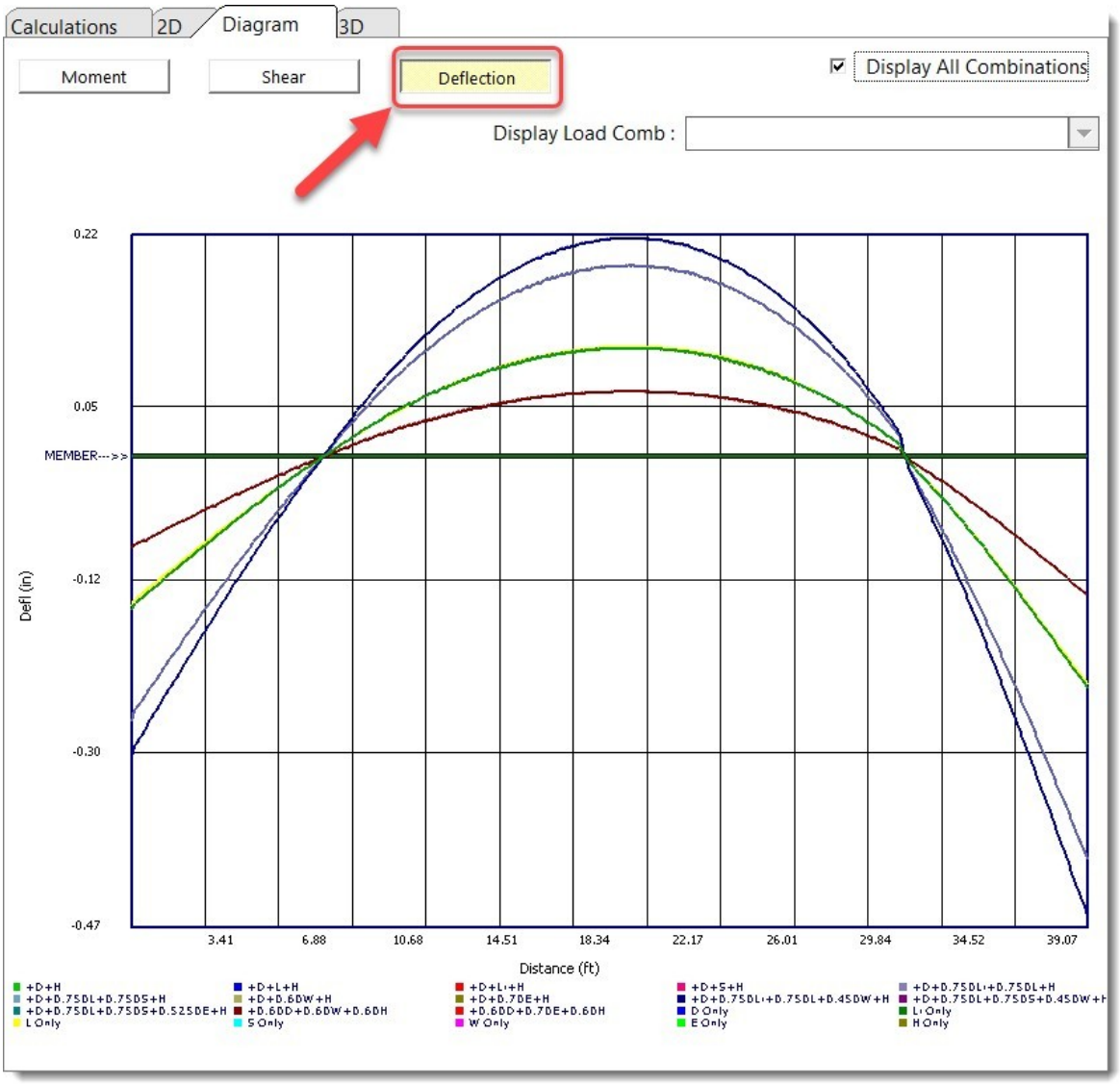
12.12 Displaying Diagrams

In addition to the Sketch of the beam, we can also display several types of diagrams. Selecting the Diagram tab will display the Shear/Moment/Deflection diagram as shown below:





© 2026 ENERCALC, LLC



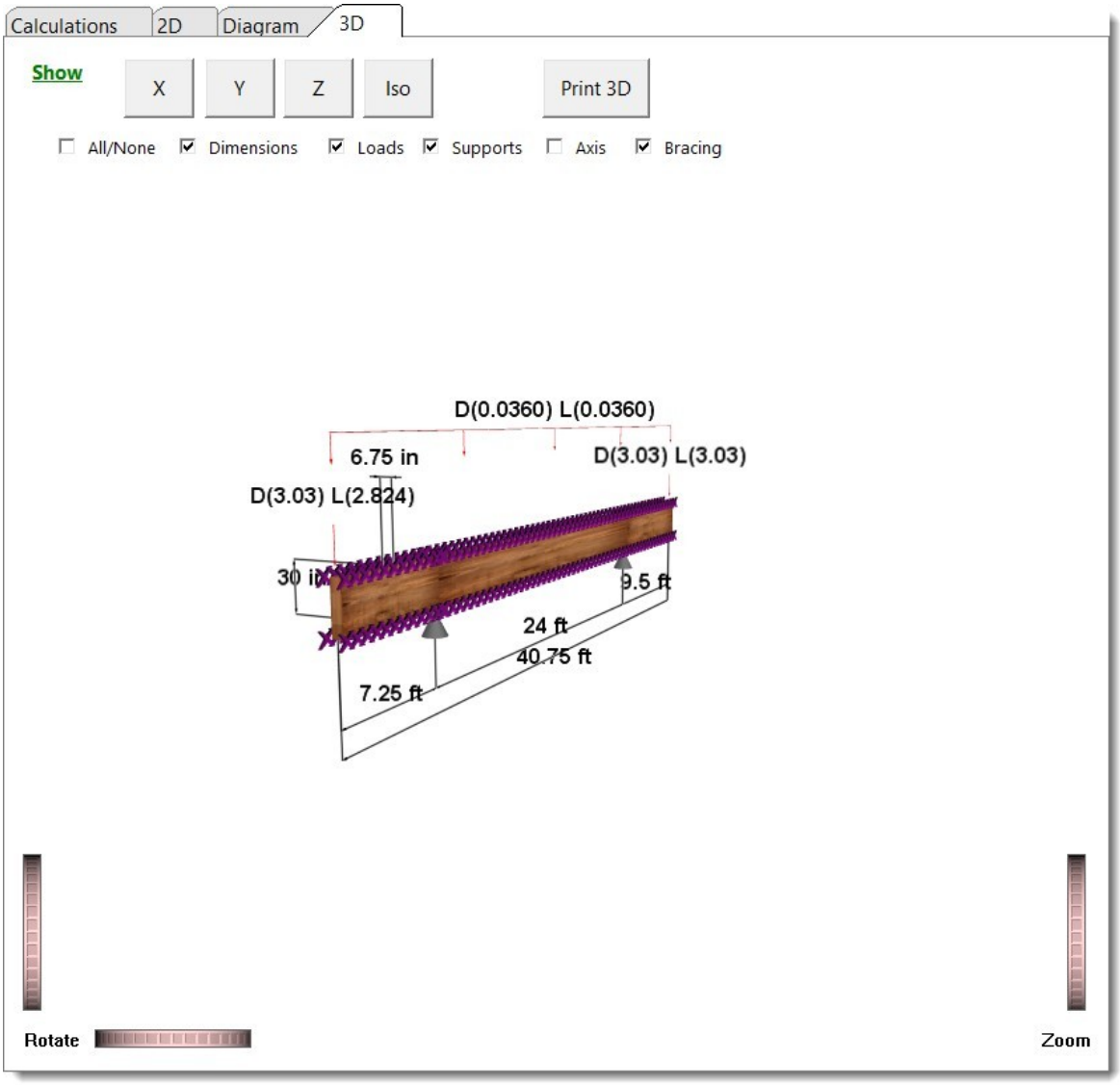
Of special interest are the various load combinations that can be viewed. By selecting the desired load combination here you can view the detailed shear/moment/deflection variations produced by each combination.

Note: In the previous steps, we turned on the option to have the program automatically handle unbalanced Live load placement. While this is a helpful feature for design, it tends to produce an abundance of results. If you'd like to simplify the view of shear, moment, and deflection diagrams, click the Span Loads tab and temporarily turn off the option to have the program automatically perform unbalanced Live load placement. To view even more specific diagrams, deselect the option to Display All Combinations, and then select the load combination of interest in the dropdown list box.

© 2026 ENERCALC, LLC

12.13 Displaying a 3D Rendering

To view a 3D rendering of the beam, click on the 3D tab. Controls are provided to manipulate the view angle, to turn various items on and off, and to print the view.



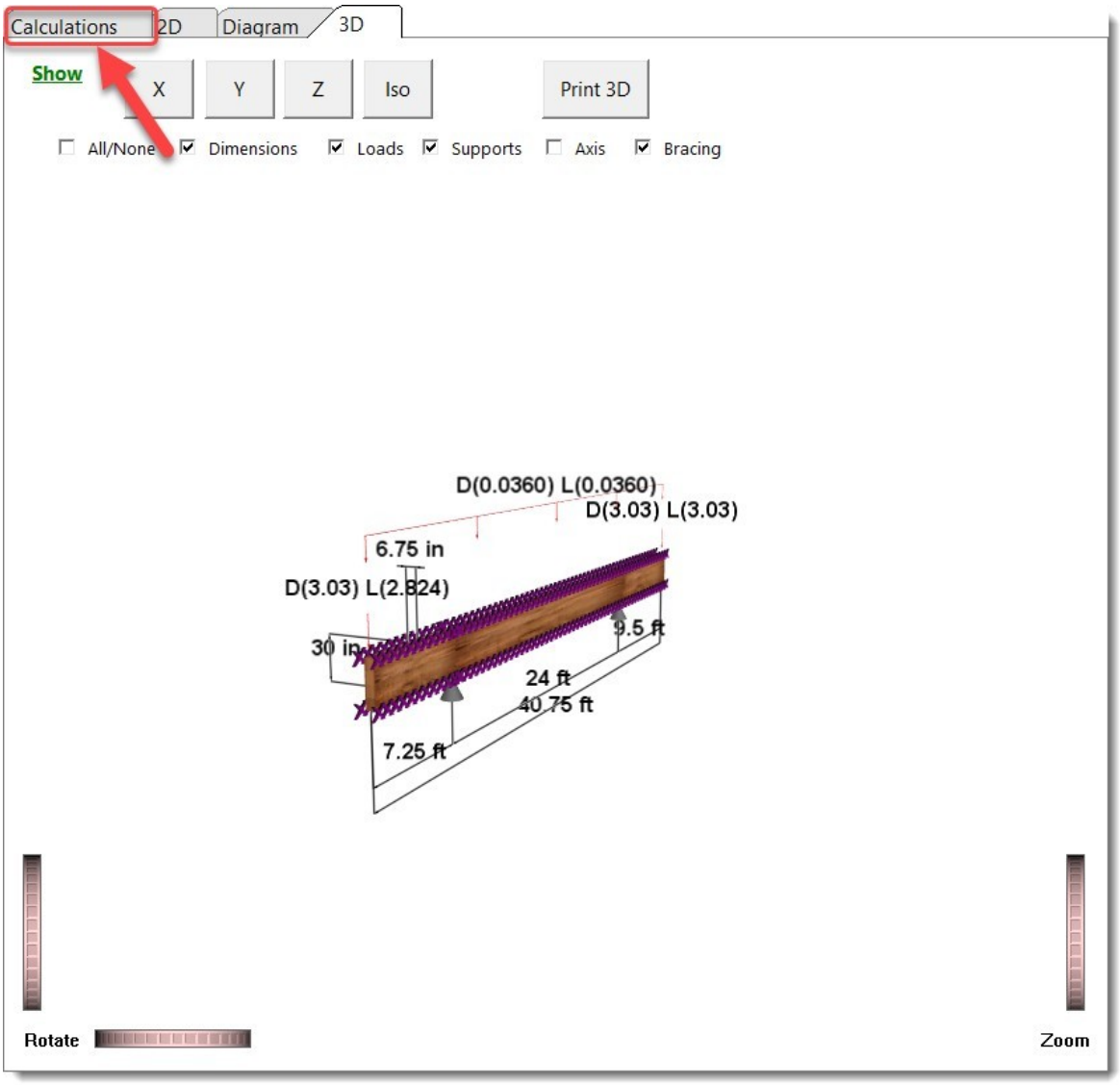
12.14 Automatic Member Section Selection

An additional feature of many of **ENERCALC SEL** calculation modules is automatic selection of sizes that satisfy your specified design criteria.

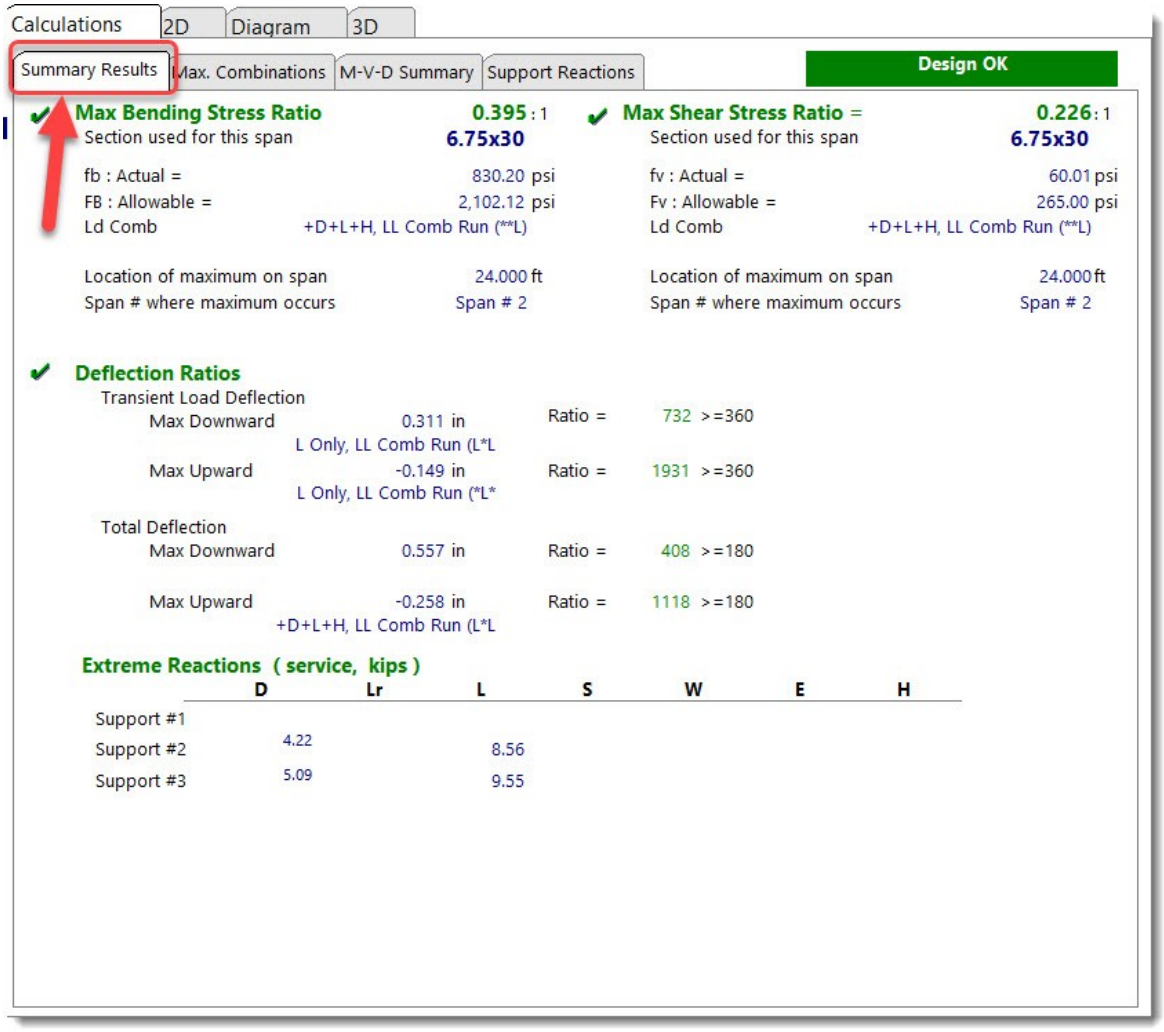
Note: If you turned off the option to have the program automatically perform unbalanced Live load placement in the previous topic, please click back to the Span Loads tab now and make sure the option for Auto Unbalanced Live Load Placement is selected once again.

© 2026 ENERCALC, LLC

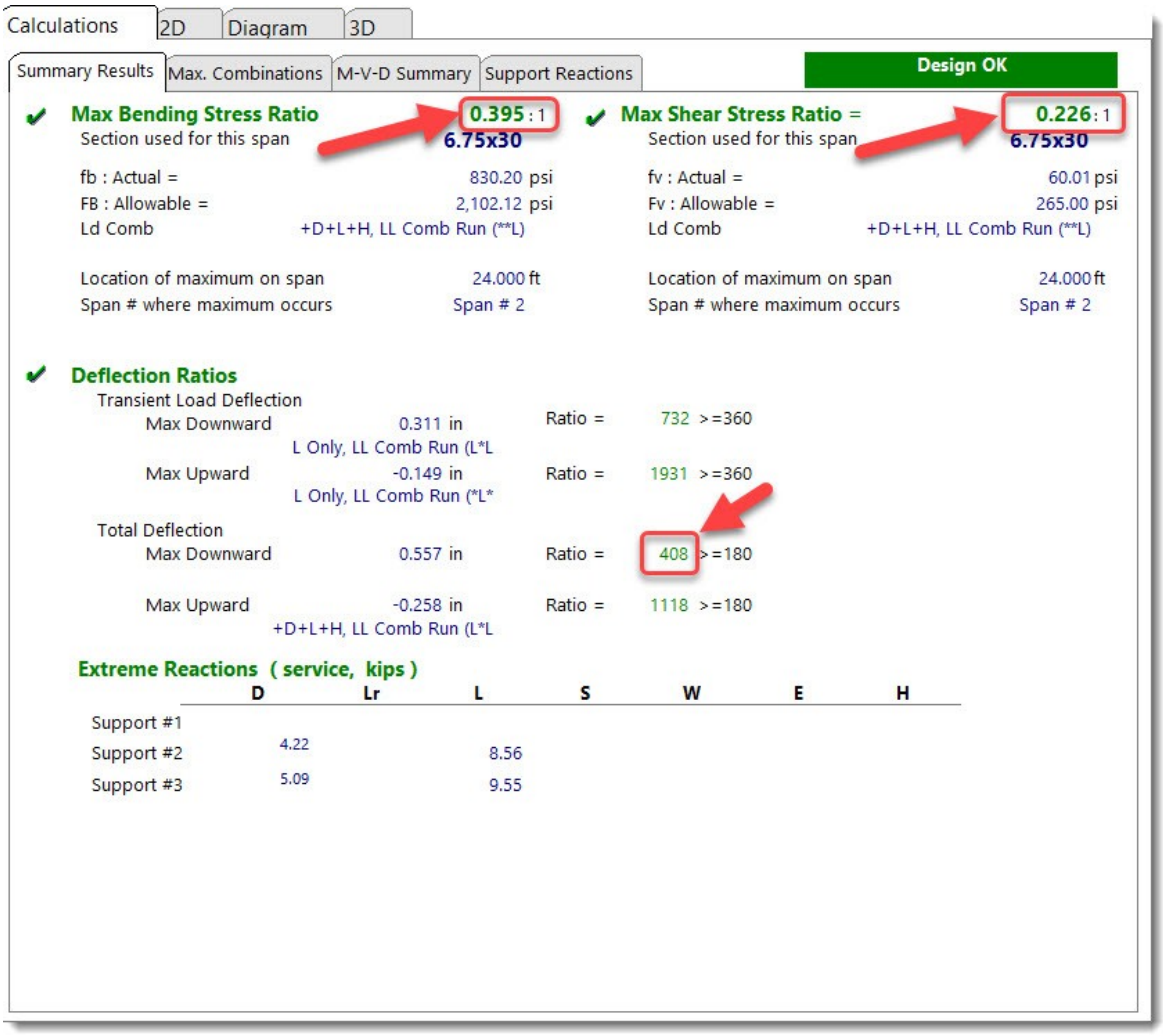
Click the Calculations tab as shown below:



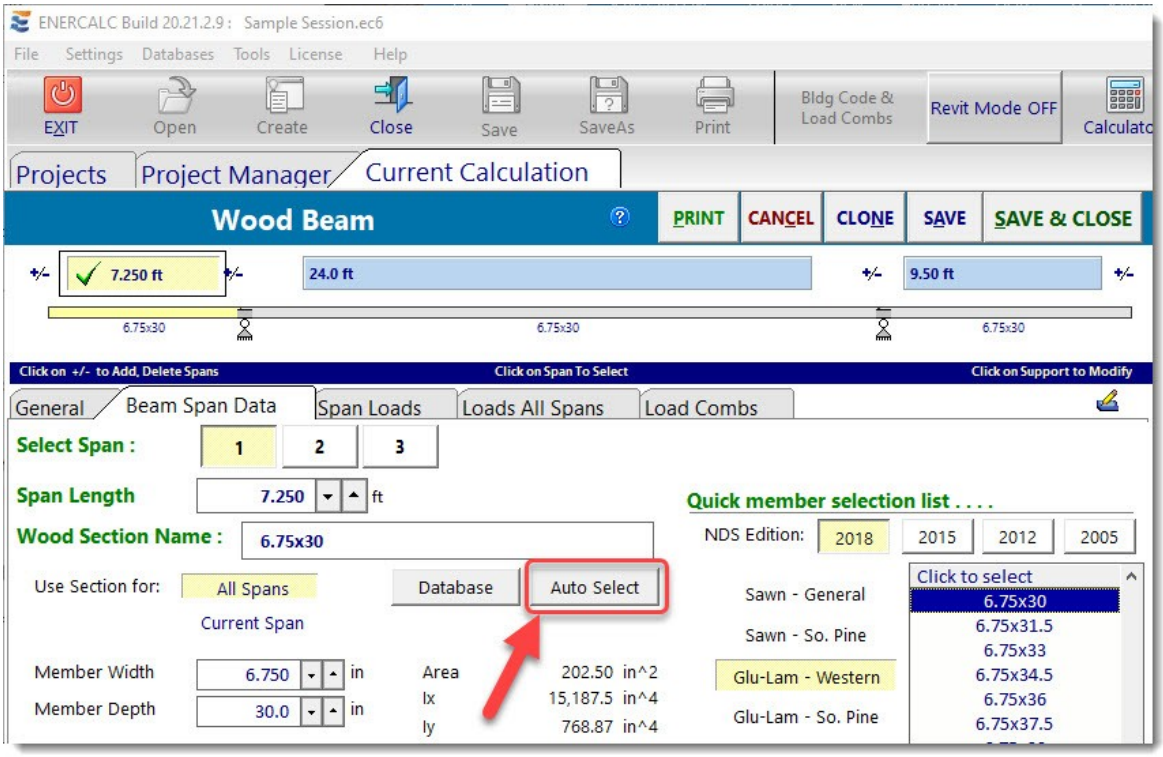
When the screen displays the Calculations environment, the Summary Results tab will be displayed by default as shown below:



Take a look at the stress and deflection ratios indicted in the screen capture below:

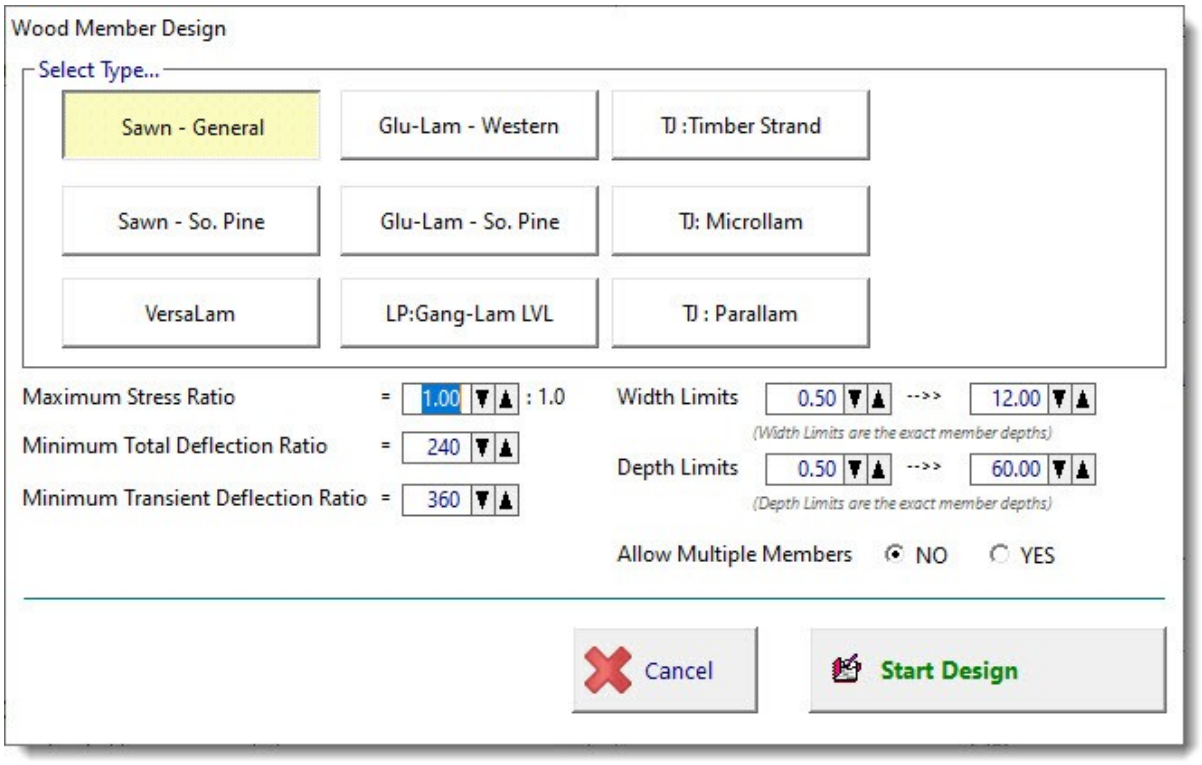


We can see that our 6.75" x 30" size guess works, but may not be an optimum use of the section. It would be interesting to see if we could find a more efficient section. Rather than manually trying other sizes from the database, we will use the Automatic Design function to select an optimum member size for us. Click the [**Auto Select**] button on the Beam Span Data tab shown below:



© 2026 ENERCALC, LLC

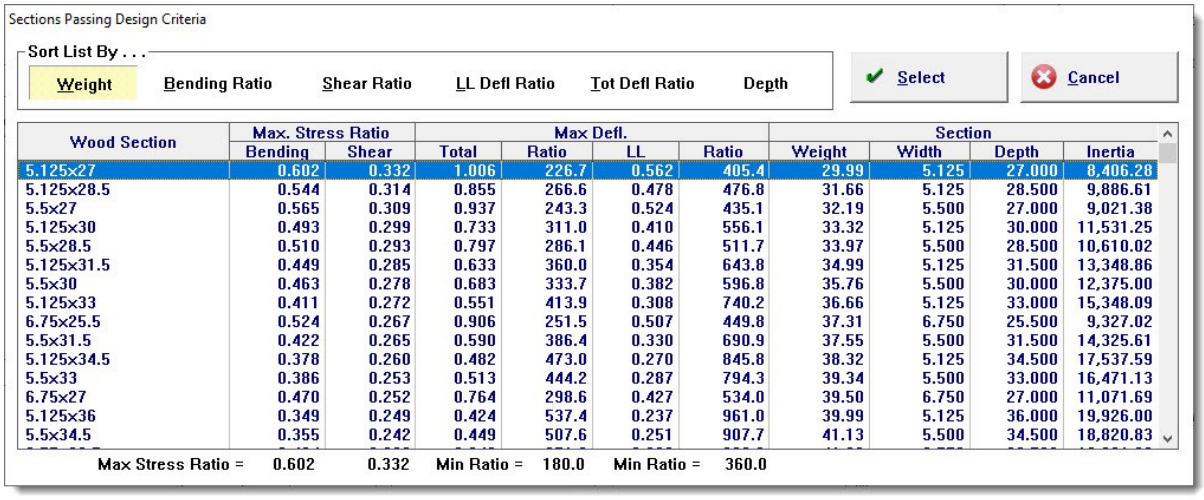
The Wood Member Design dialog is displayed, and it allows you to set the design criteria:



On the Wood Member Design dialog, click the [**Glu-Lam - Western**] section type, set the Maximum Stress Ratio to 1.0, and the Minimum Total Deflection Ratio to 180, then click the [**Start Design**] button.

© 2026 ENERCALC, LLC

The software will automatically search the built-in Wood Section Database for all sections that satisfy this criteria using the beam span, stress grade, and loading you have specified. In a few moments the following dialog will be displayed:

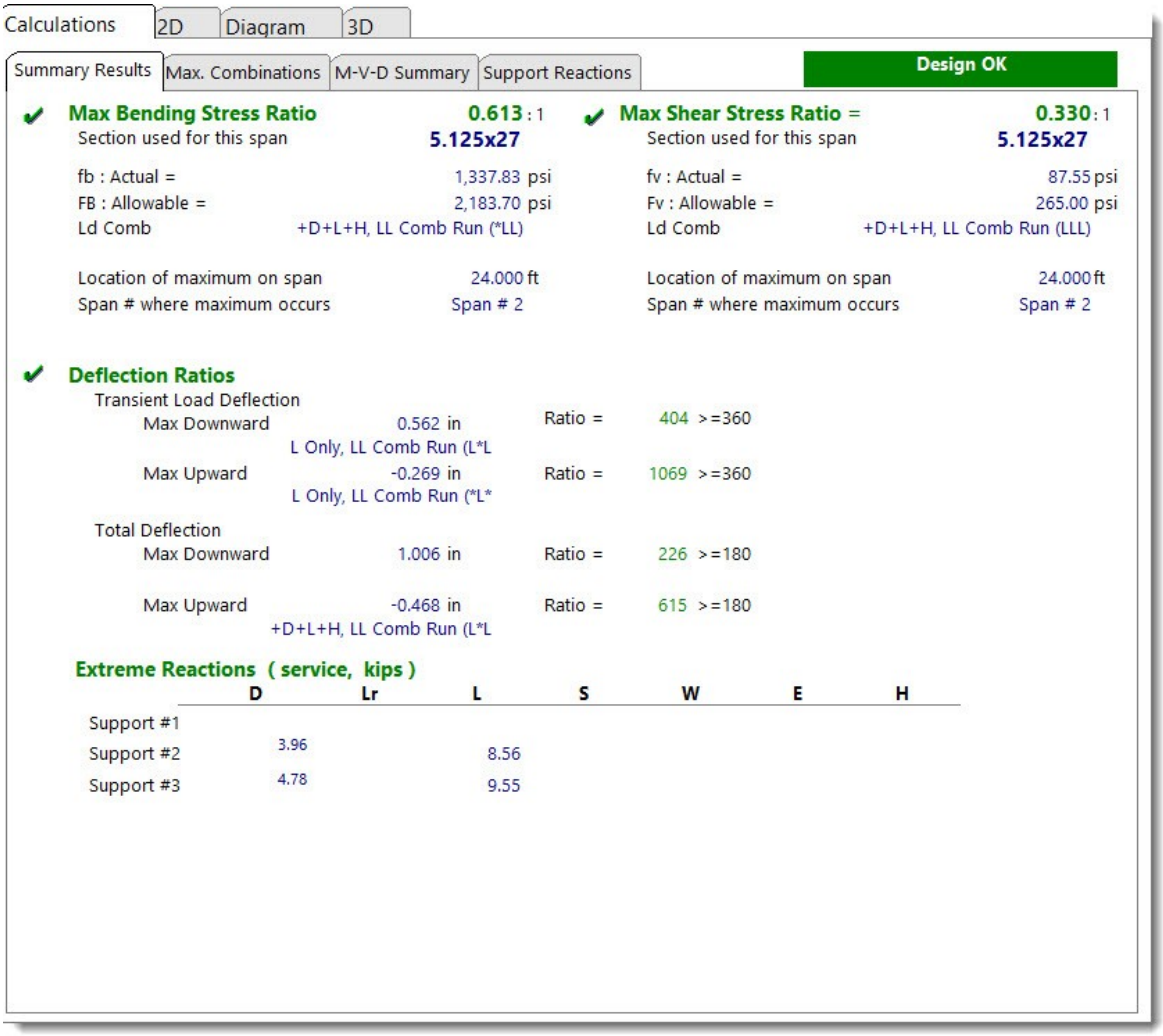


Wood Section	Max. Stress Ratio		Max Defl.				Section			
	Bending	Shear	Total	Ratio	LL	Ratio	Weight	Width	Depth	Inertia
5.125x27	0.602	0.332	1.006	226.7	0.562	405.4	29.99	5.125	27.000	8,406.28
5.125x28.5	0.544	0.314	0.855	266.6	0.478	476.8	31.66	5.125	28.500	9,886.61
5.5x27	0.565	0.309	0.937	243.3	0.524	435.1	32.19	5.500	27.000	9,021.38
5.125x30	0.493	0.299	0.733	311.0	0.410	556.1	33.32	5.125	30.000	11,531.25
5.5x28.5	0.510	0.293	0.797	286.1	0.446	511.7	33.97	5.500	28.500	10,610.02
5.125x31.5	0.449	0.285	0.633	360.0	0.354	643.8	34.99	5.125	31.500	13,348.86
5.5x30	0.463	0.278	0.683	333.7	0.382	596.8	35.76	5.500	30.000	12,375.00
5.125x33	0.411	0.272	0.551	413.9	0.308	740.2	36.66	5.125	33.000	15,348.09
6.75x25.5	0.524	0.267	0.906	251.5	0.507	449.8	37.31	6.750	25.500	9,327.02
5.5x31.5	0.422	0.265	0.590	386.4	0.330	690.9	37.55	5.500	31.500	14,325.61
5.125x34.5	0.378	0.260	0.482	473.0	0.270	845.8	38.32	5.125	34.500	17,537.59
5.5x33	0.386	0.253	0.513	444.2	0.287	794.3	39.34	5.500	33.000	16,471.13
6.75x27	0.470	0.252	0.764	298.6	0.427	534.0	39.50	6.750	27.000	11,071.69
5.125x36	0.349	0.249	0.424	537.4	0.237	961.0	39.99	5.125	36.000	19,926.00
5.5x34.5	0.355	0.242	0.449	507.6	0.251	907.7	41.13	5.500	34.500	18,820.83
Max Stress Ratio =	0.602	0.332	Min Ratio =	180.0	Min Ratio =	360.0				

Notice how the listed members can be sorted in different orders based on: Weight, Bending Ratio, Shear Ratio, LL Deflection Ratio,Total Deflection Ratio, Depth, etc. This allows you to review the beam sizes that pass your criteria in different ways.

© 2026 ENERCALC, LLC

Click on the **5.125 x 27** beam and then click the [**Select**] button. The Summary Results tab now appears like this:



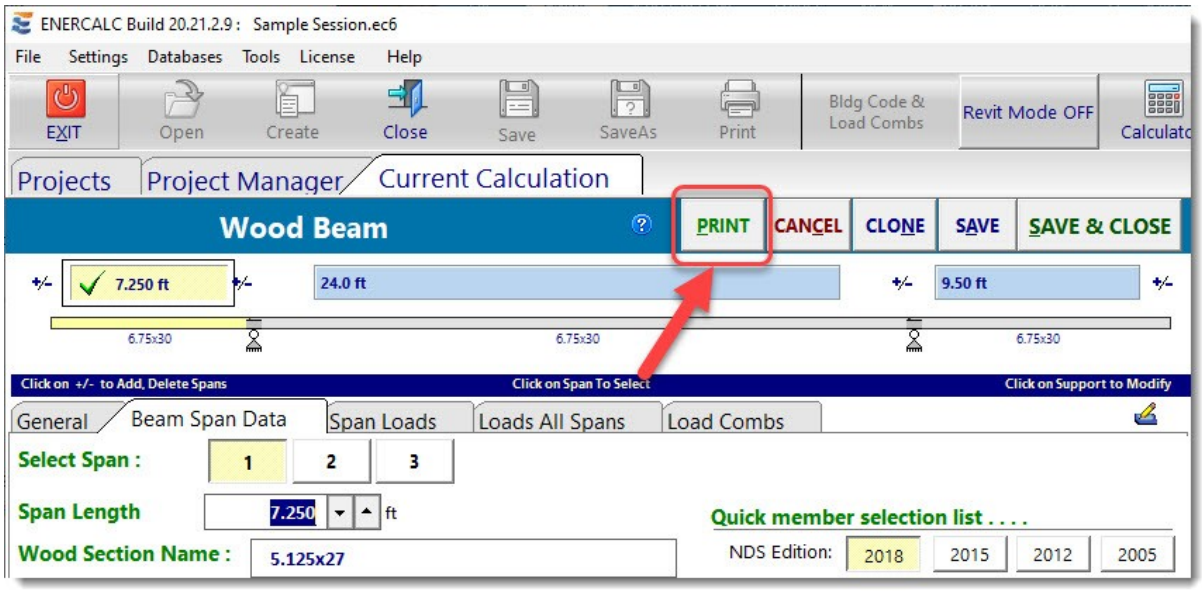
12.15 Printing a Calculation

Okay...congratulations! You've created a Project File, added a calculation, entered data, retrieved database information, performed an automatic design, and viewed some graphics. It's time to print this calculation and take a look at what the software provides for documentation.

Please remember that you have previously entered project information, title block information, and of course now you have a valid engineering calculation to print.

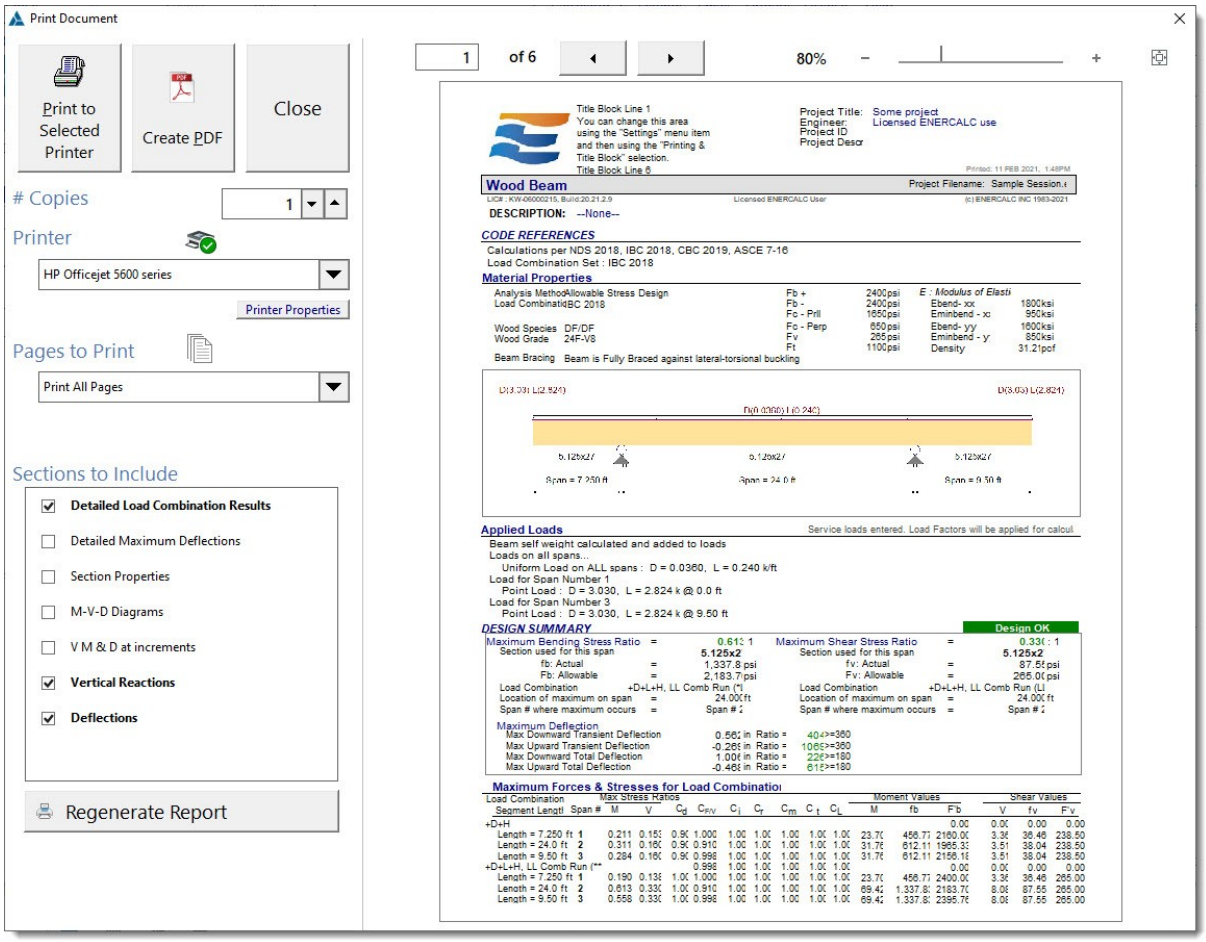
© 2026 ENERCALC, LLC

Printing is easy in **ENERCALC SEL**. To print an individual calculation report while the module is still open, click the [PRINT] button:



© 2026 ENERCALC, LLC

The Print Document window opens as shown below:



This window provides controls to decide which sections to include in your report. Be sure to click [Regenerate Report] after making any changes in the Sections to Include area.

© 2026 ENERCALC, LLC

Once selections have been made and the report has been regenerated, the report can be previewed, printed, and/or printed to PDF.

Title Block Line 1
You can change this area using the "Settings" menu item and then using the "Printing & Title Block" selection.
Title Block Line 6

Project Title: Some project
Engineer: Licensed ENERCALC user
Project ID:
Project Descr:

Printed: 11 FEB 2021, 1:48PM

Wood Beam

Project Filename: Sample Session.ec6

LIC# : KW-06000215, Build:20.21.2.9 Licensed ENERCALC User (c) ENERCALC INC 1983-2021

DESCRIPTION: --None--

CODE REFERENCES

Calculations per NDS 2018, IBC 2018, CBC 2019, ASCE 7-16
Load Combination Set : IBC 2018

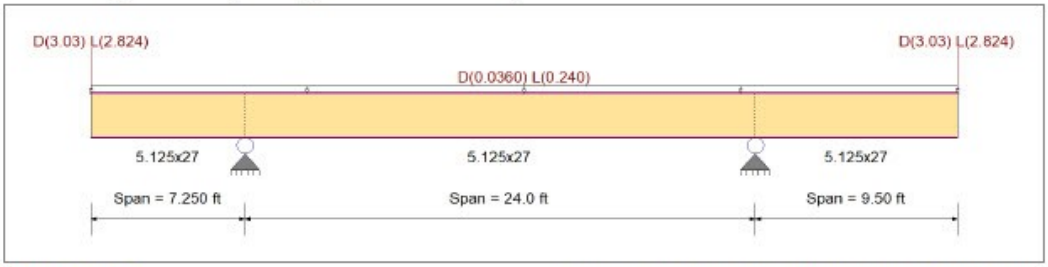
Material Properties

Analysis Method :Allowable Stress Design
Load Combination IBC 2018

Wood Species DF/DF
Wood Grade 24F-V8

Beam Bracing Beam is Fully Braced against lateral-torsional buckling

		E : Modulus of Elasticity	
Fb +	2400 psi	Ebend- xx	1800 ksi
Fb -	2400 psi	Eminbend - xx	950 ksi
Fc - Prll	1650 psi	Ebend- yy	1600 ksi
Fc - Perp	650 psi	Eminbend - yy	850 ksi
Fv	265 psi	Density	31.21 pcf
Ft	1100 psi		



Applied Loads

Service loads entered. Load Factors will be applied for calculations.

Beam self weight calculated and added to loads
Loads on all spans...
Uniform Load on ALL spans : D = 0.0360, L = 0.240 k/ft
Load for Span Number 1
Point Load : D = 3.030, L = 2.824 k @ 0.0 ft
Load for Span Number 3
Point Load : D = 3.030, L = 2.824 k @ 9.50 ft

DESIGN SUMMARY

Design OK

Maximum Bending Stress Ratio	=	0.613 : 1	Maximum Shear Stress Ratio	=	0.330 : 1
Section used for this span		5.125x27	Section used for this span		5.125x27
fb: Actual	=	1,337.83 psi	fv: Actual	=	87.55 psi
Fb: Allowable	=	2,183.70 psi	Fv: Allowable	=	265.00 psi
Load Combination		+D+L+H, LL Comb Run (*LL)	Load Combination		+D+L+H, LL Comb Run (LLL)
Location of maximum on span	=	24.000 ft	Location of maximum on span	=	24.000 ft
Span # where maximum occurs	=	Span # 2	Span # where maximum occurs	=	Span # 2

Maximum Deflection

Max Downward Transient Deflection	0.562 in	Ratio =	404 >=360
Max Upward Transient Deflection	-0.269 in	Ratio =	1069 >=360
Max Downward Total Deflection	1.006 in	Ratio =	226 >=180
Max Upward Total Deflection	-0.468 in	Ratio =	615 >=180

Maximum Forces & Stresses for Load Combinations

Load Combination		Max Stress Ratios									Moment Values			Shear Values		
Segment Length	Span #	M	V	C_d	$C_{F/V}$	C_i	C_r	C_m	C_t	C_L	M	fb	Fb	V	fv	Fv
+D+H													0.00	0.00	0.00	0.00
Length = 7.250 ft	1	0.211	0.153	0.90	1.000	1.00	1.00	1.00	1.00	1.00	23.70	456.77	2160.00	3.36	36.46	238.50
Length = 24.0 ft	2	0.311	0.160	0.90	0.910	1.00	1.00	1.00	1.00	1.00	31.76	612.11	1965.33	3.51	38.04	238.50
Length = 9.50 ft	3	0.284	0.160	0.90	0.998	1.00	1.00	1.00	1.00	1.00	31.76	612.11	2156.18	3.51	38.04	238.50
+D+L+H, LL Comb Run (**L)					0.998	1.00	1.00	1.00	1.00	1.00			0.00	0.00	0.00	0.00
Length = 7.250 ft	1	0.190	0.138	1.00	1.000	1.00	1.00	1.00	1.00	1.00	23.70	456.77	2400.00	3.36	36.46	265.00
Length = 24.0 ft	2	0.613	0.330	1.00	0.910	1.00	1.00	1.00	1.00	1.00	69.42	1,337.83	2183.70	8.08	87.55	265.00
Length = 9.50 ft	3	0.558	0.330	1.00	0.998	1.00	1.00	1.00	1.00	1.00	69.42	1,337.83	2395.76	8.08	87.55	265.00

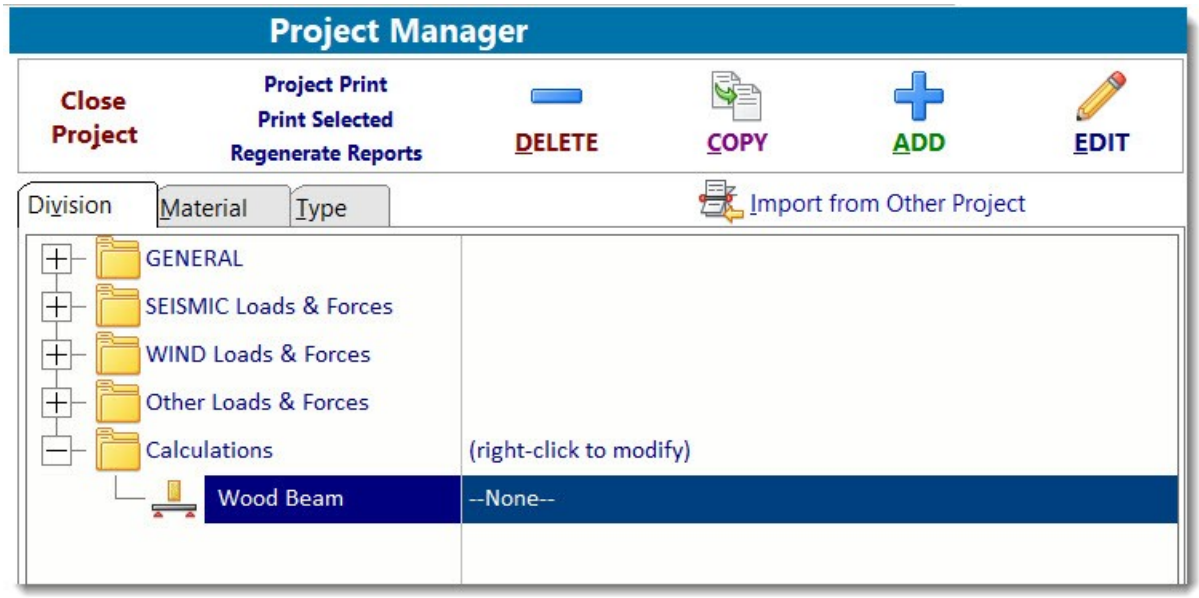
© 2026 ENERCALC, LLC

12.16 Saving a Calculation

With as much as we have accomplished in this short period of time, it is very important to remember that we have not yet SAVED this calculation. Everything associated with this calculation is still stored in the computer's RAM. A power outage or severe system lockup would cause a loss of your current calculation.

So to save this new calculation to the Project File, click the [**Save & Close**] button

. The current calculation will be saved and the display will return to the Project Manager. In the image below, you can see that a Wood Beam calculation has been added to the Division named "Calculations":



Congratulations! Your calculation is now saved and you have a real Project File with an actual calculation.

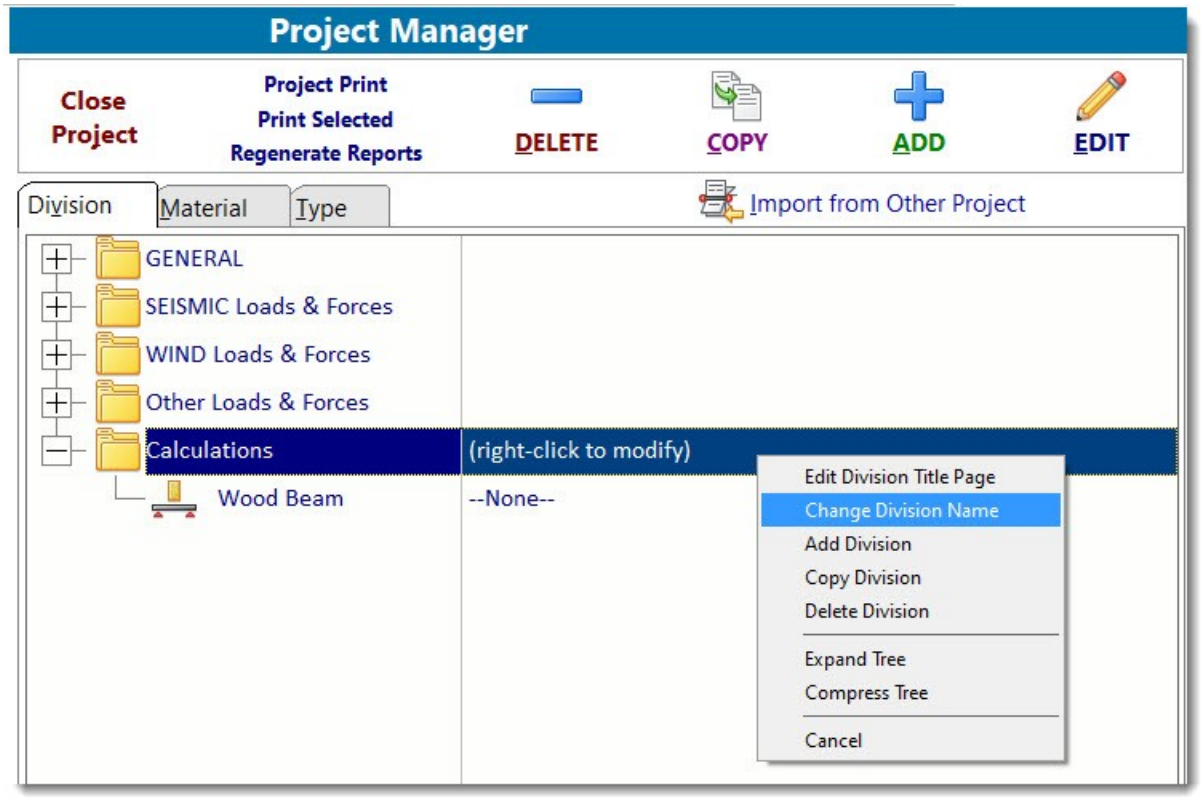
12.17 Editing a Division Name and Adding a New Division

Let's do a little more work here in the current Project File. We will edit the Division named "Calculations", and we will add a new Division.

© 2026 ENERCALC, LLC

Edit a Division Name

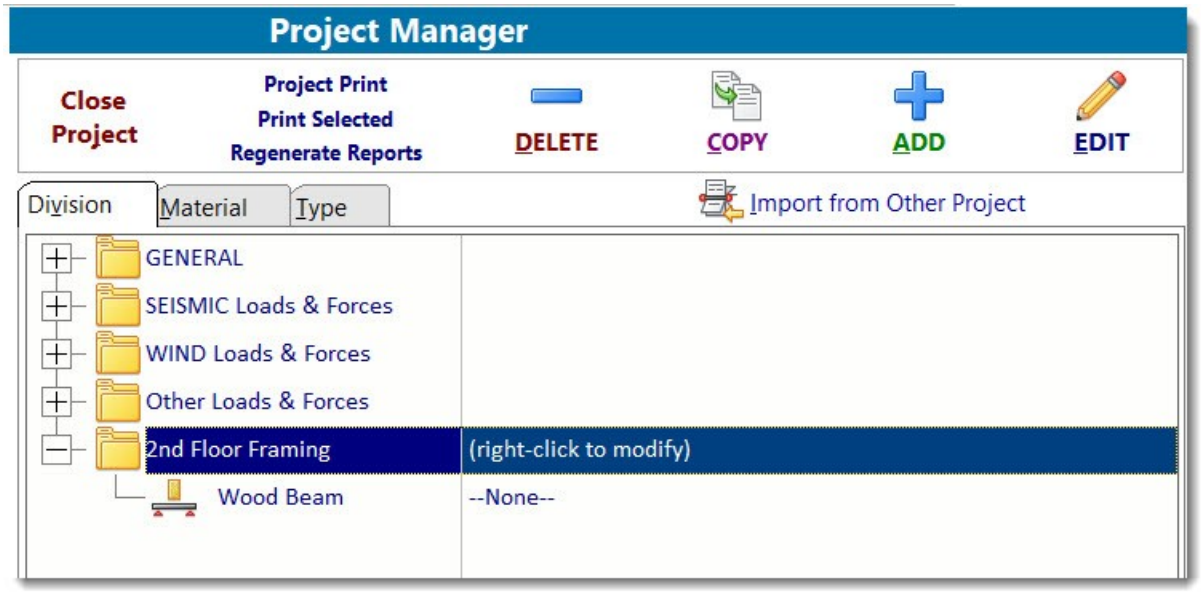
To edit a Division name, simply right-click the name and click **Change Division Name** in the dropdown menu.



In the Enter New Division Name dialog, revise the Division name to "2nd Floor Framing" as shown below:

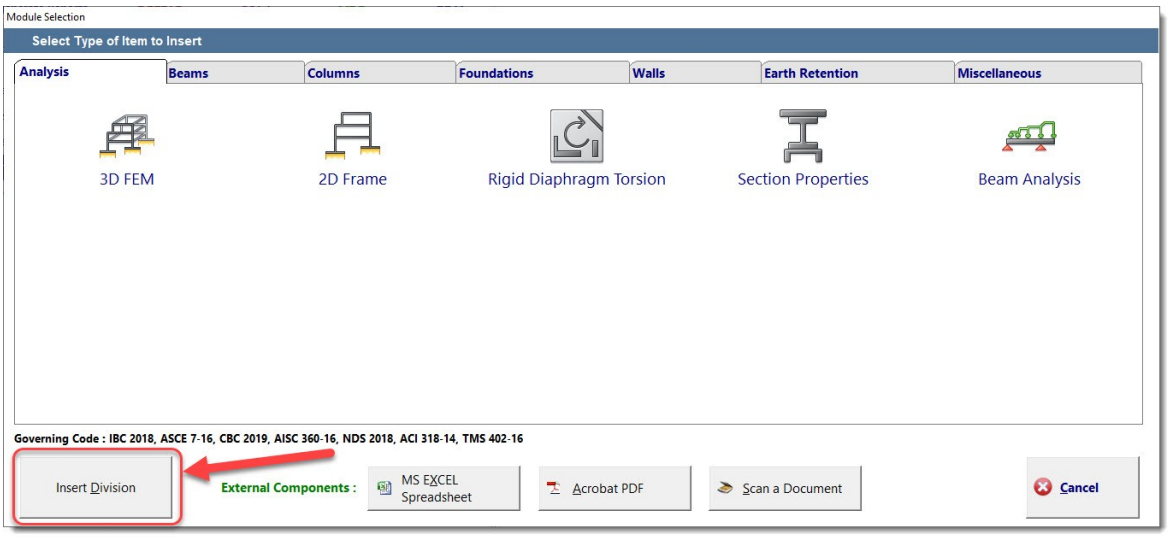
© 2026 ENERCALC, LLC

Click [**Save**]. You will notice that the Division name has been revised from "Calculations" to "2nd Floor Framing", but it still contains the Wood Beam calculation.



Now we will add a NEW Division named "3rd Floor Framing". To do this, click on the Division named "2nd Floor Framing" and then click the [**Add**] button

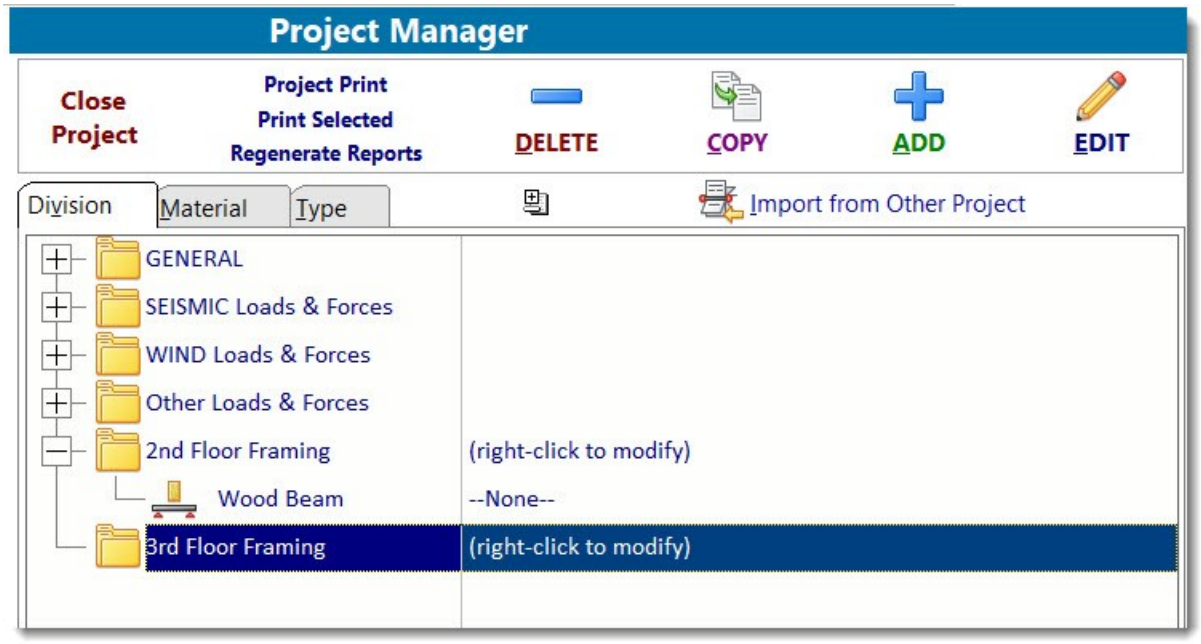
. In the Module Selection dialog, click the [**Insert Division**] button shown below:



Note: There is also an option in the right-click dropdown menu named Add Division.

© 2026 ENERCALC, LLC

Then enter the name "3rd Floor Framing" and click [**Save**]. Your Project calculation list should look like this:

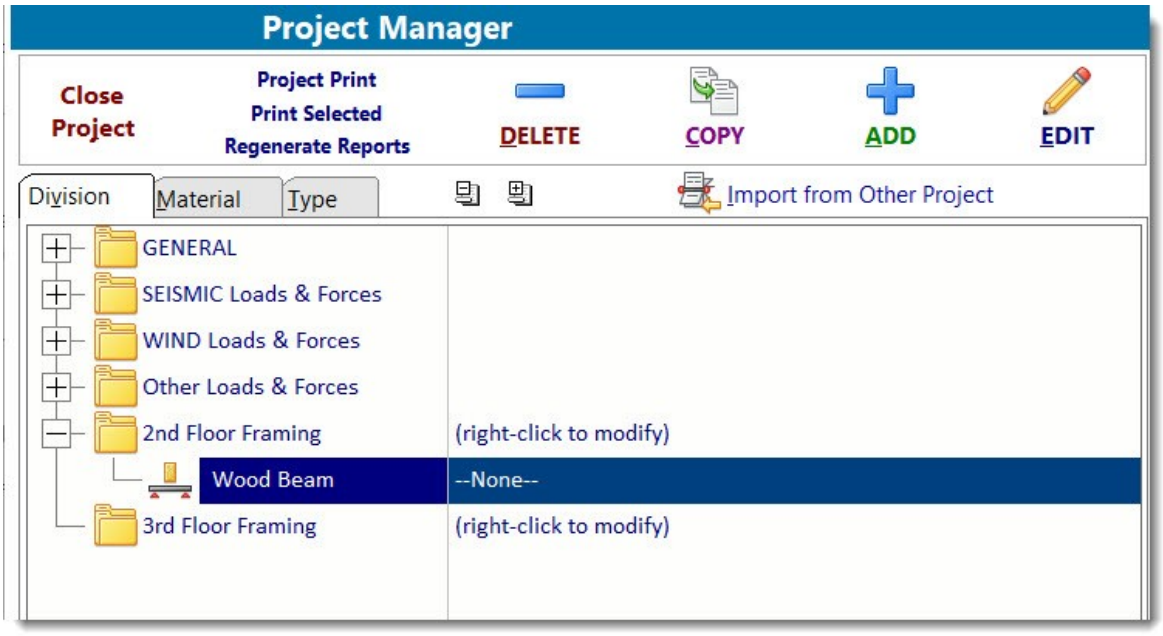


12.18 Adding Another Calculation

Similar to the way we added our first calculation, we will now add another calculation in the 2nd Floor Framing Division by making a copy of the current Wood Beam calculation and then move it into the 3rd Floor Framing Division.

© 2026 ENERCALC, LLC

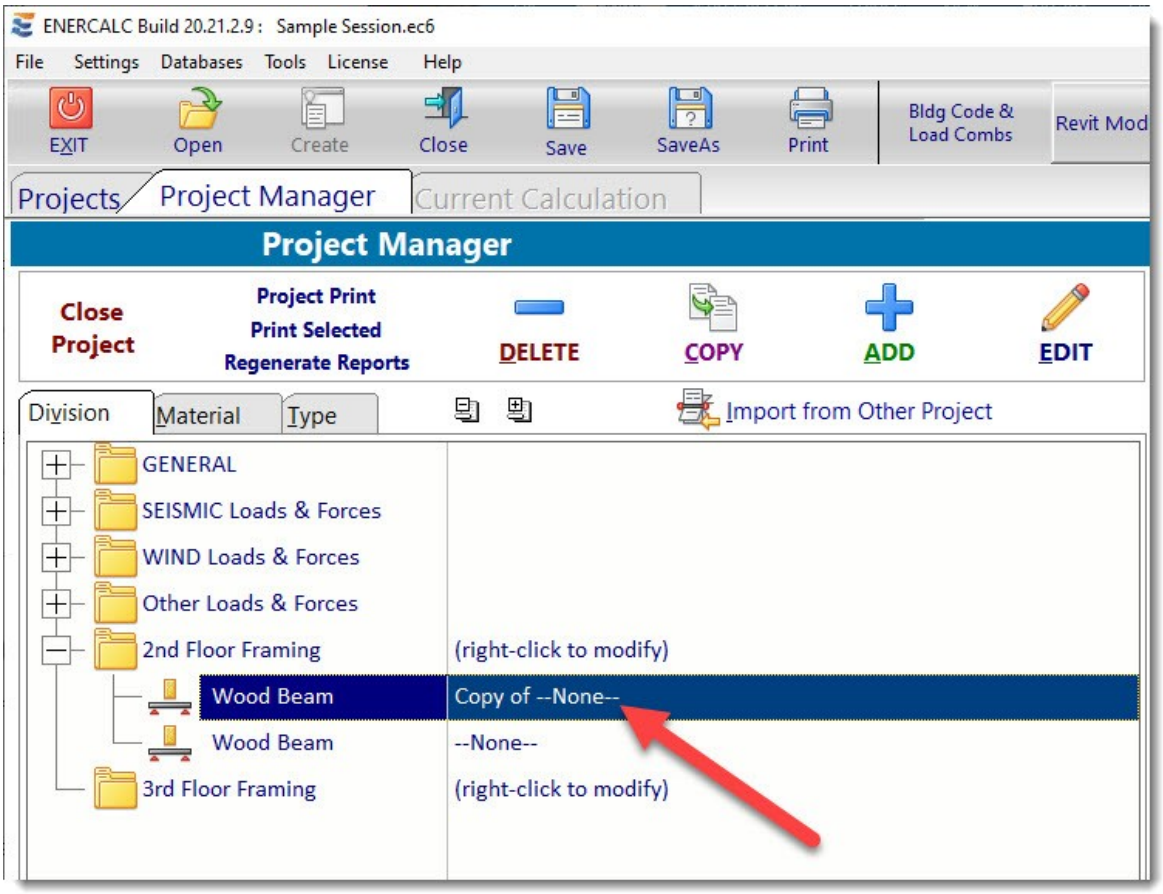
First, click on the Wood Beam calculation as shown in the screen capture below:



Then, click the button. This will read all the data from the highlighted calculation and create an entirely new calculation that is an exact copy of the highlighted one.

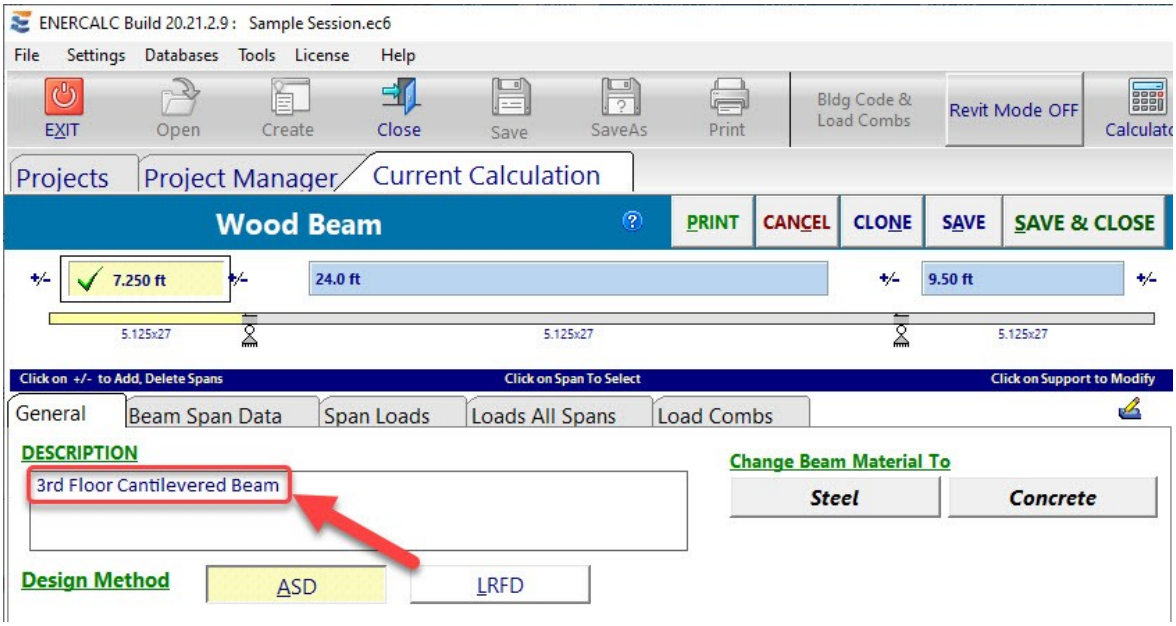
© 2026 ENERCALC, LLC

Here is a view of the newly created calculation:



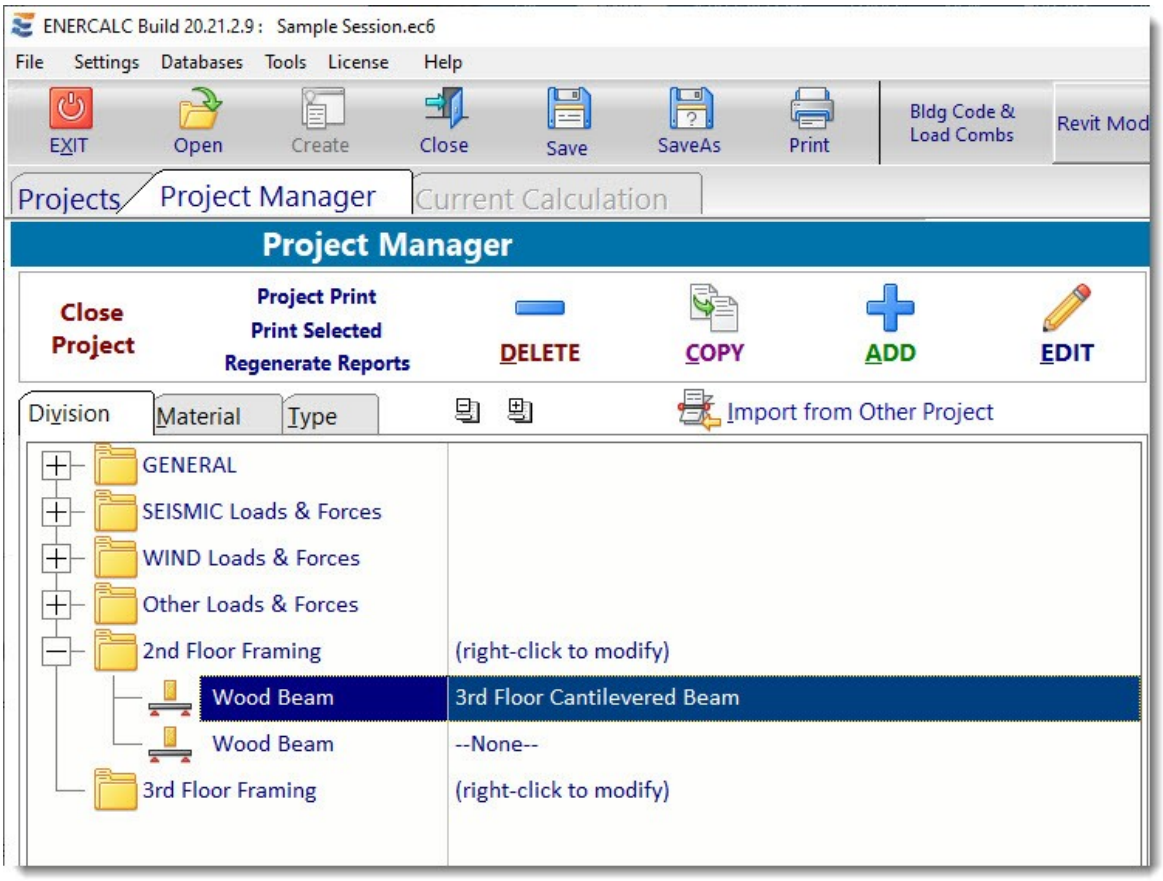
© 2026 ENERCALC, LLC

Double-click the new calculation to open it. When the Wood Beam calculation module opens, change the Description item to read "3rd Floor Cantilevered Beam" as shown below:



© 2026 ENERCALC, LLC

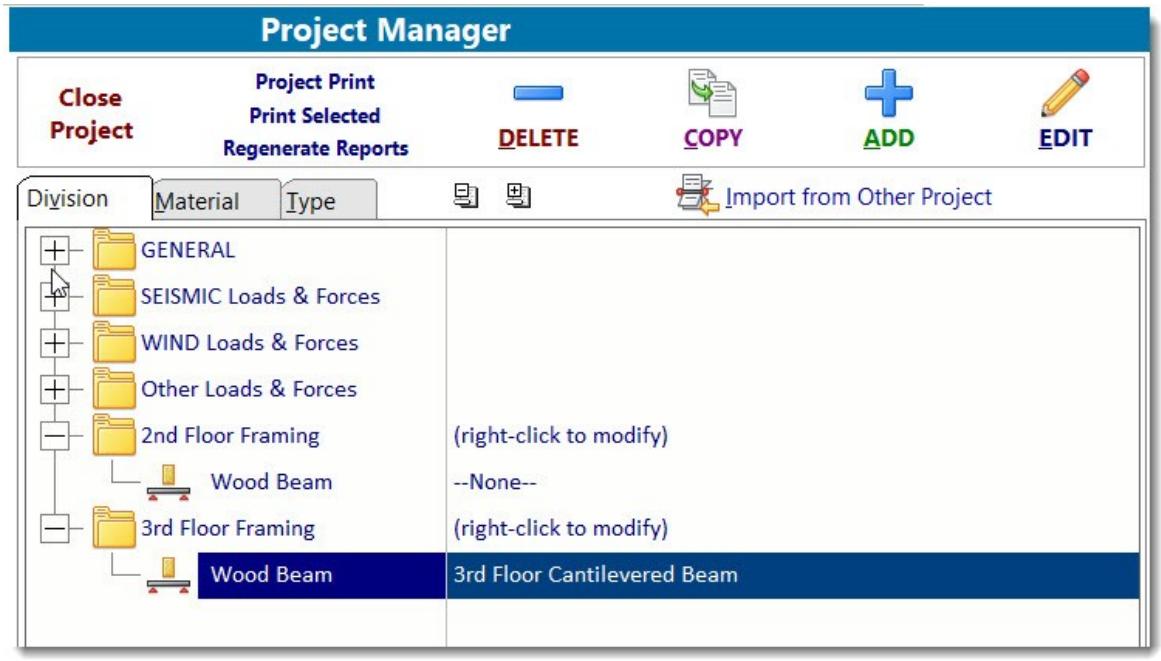
Next click the [**Save & Close**] button to save this new calculation to the Project File and close the calculation. The Project Manager will appear as follows:



The final step is to move this new calculation into the 3rd Floor Framing Division. This is easily done by using these buttons:

© 2026 ENERCALC, LLC

While the new calculation is highlighted, click the Move Highlighted Item Down button twice to move it down to the 3rd Floor Framing Division. The Project Calculation list will appear like this:



12.19 Creating a Technical Support Question

The occasion may arise that you will have a question about the software system. To best serve you, there are certain pieces of information that are essential. Among these are your name, the name of your company, your "KW" user registration number, and the build number of the software you are using.

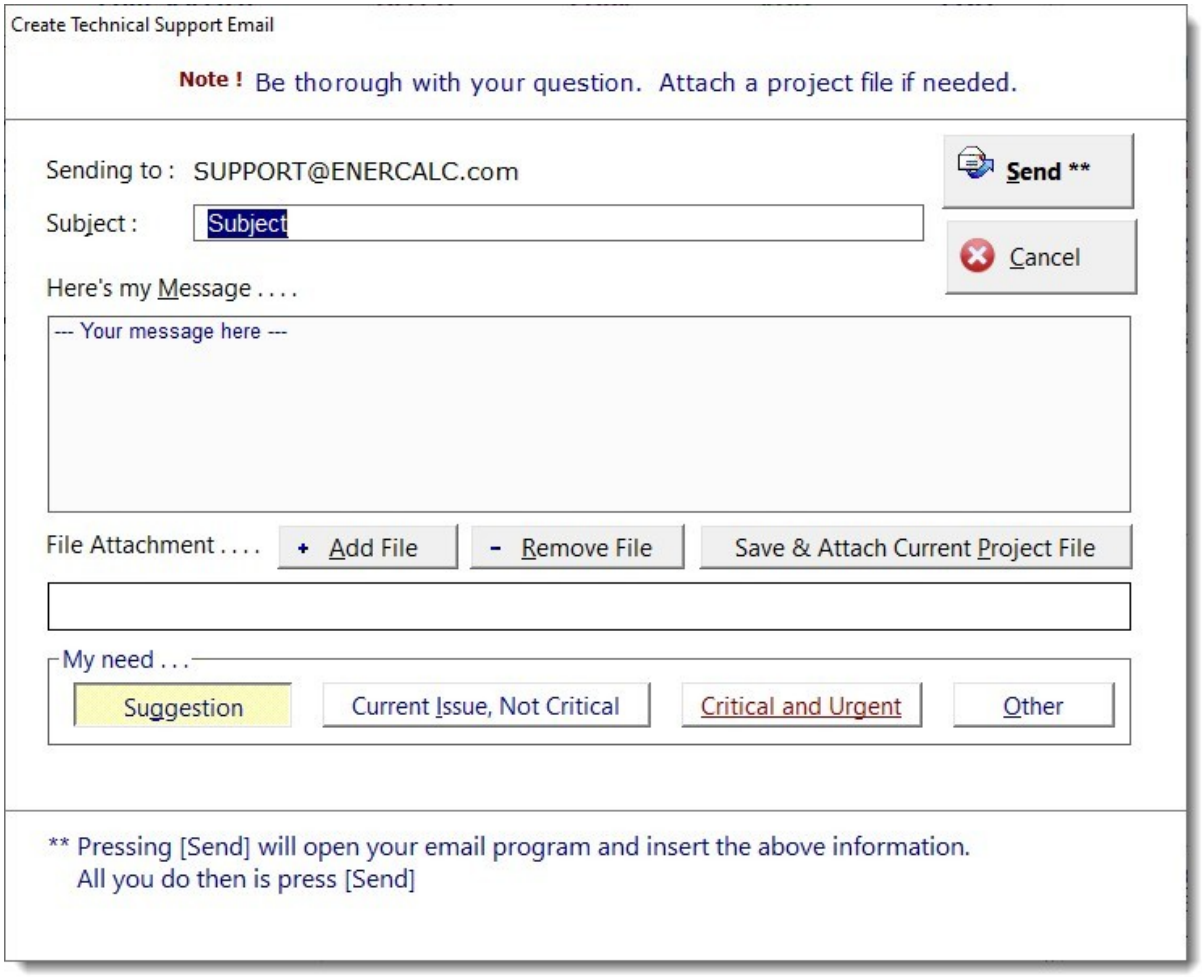
To make it easy for you to give us all this AND state your question, **ENERCALC SEL** has a built-in technical support form. It allows you to simply type in your question and email it directly to our Technical Support Group.

By emailing us your Technical Support questions, we can easily identify you, determine if you are in need of a maintenance release, and easily read your questions. In addition, email offers the option of attaching the subject ENERCALC file.

© 2026 ENERCALC, LLC

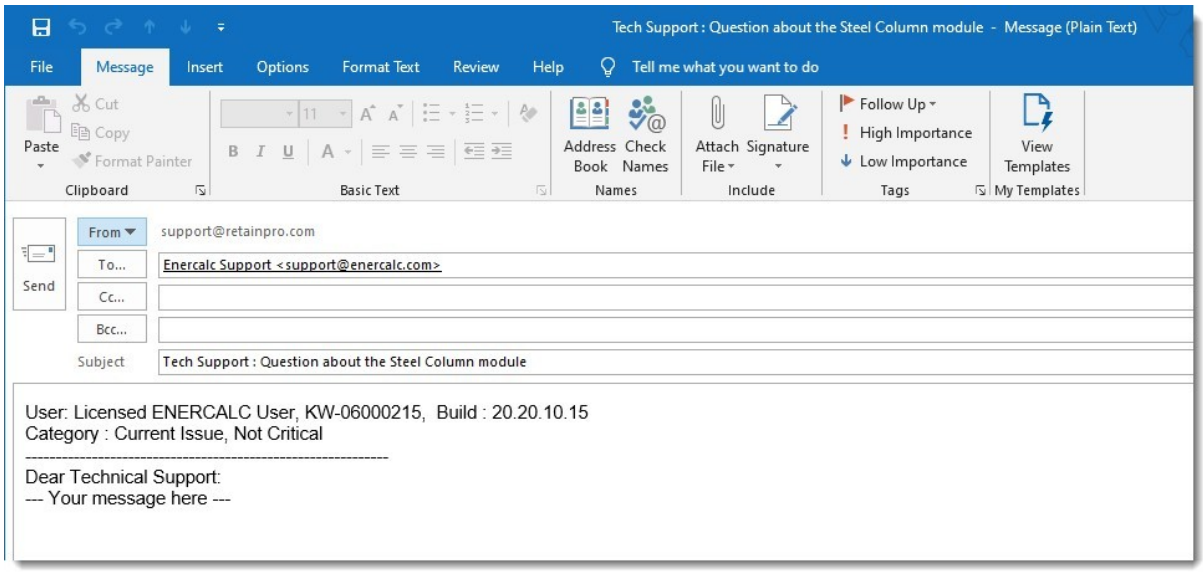
To create an email:

Click **Help > Create Tech Support EMAIL** from the main menu. This opens the form shown below, in which you can enter your question and attach your Project File if desired.



© 2026 ENERCALC, LLC

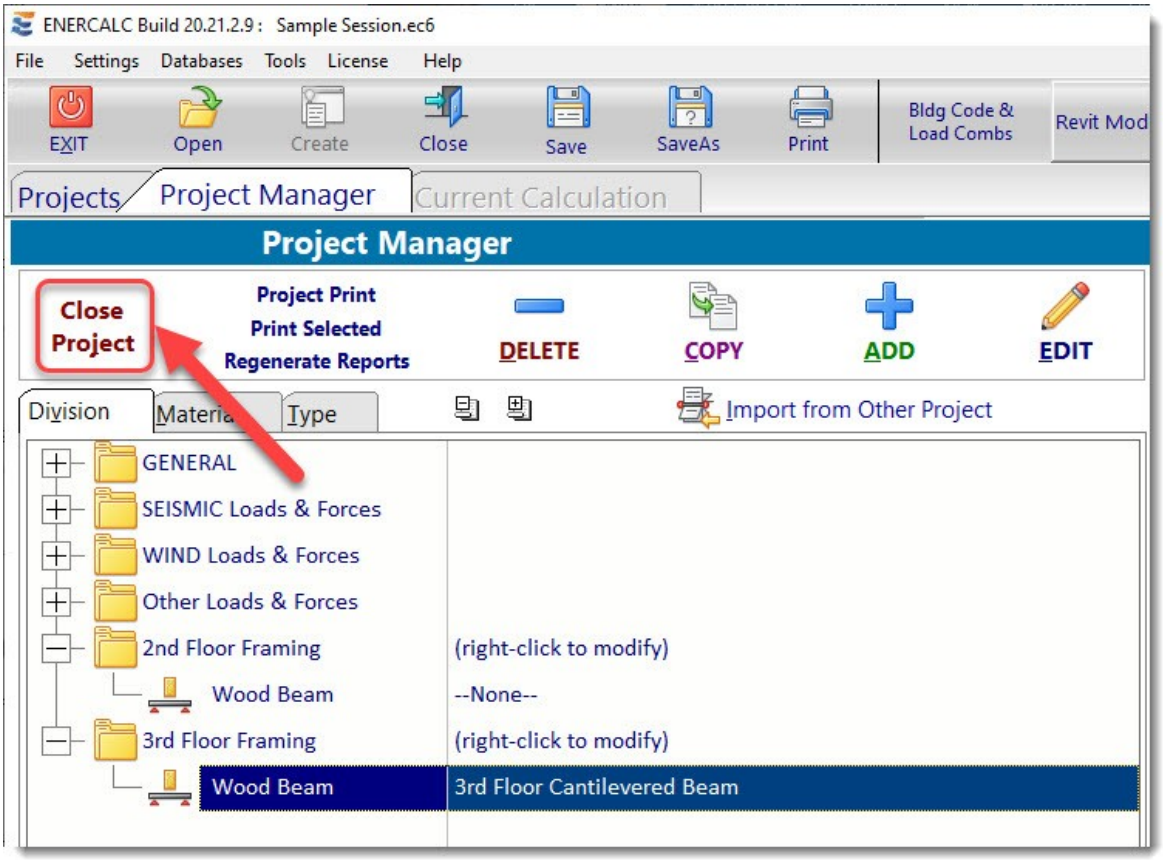
When finished, click the [**Send**] button to transfer the completed message (with attachments) to your email program:



© 2026 ENERCALC, LLC

12.20 Closing a Project File

It is now time to close out our work session. The data is saved in the Project File already, so at this point we can close the current Project File by clicking **File > Close Project** from the main menu or by clicking the Close Project button:



© 2026 ENERCALC, LLC

Part XIII

13 Calculation Modules

13.1 Linked Loads

Please select a subtopic.

- Linked Loads Overview 192
- Linked Load Module Support 193
- Select a Calculation & Select a Reaction 194
- Edit Link Settings 196
- Linked Reactions Table 200
- Status Badges and Linked Load Recalculation 203

13.1.1 Linked Loads Overview

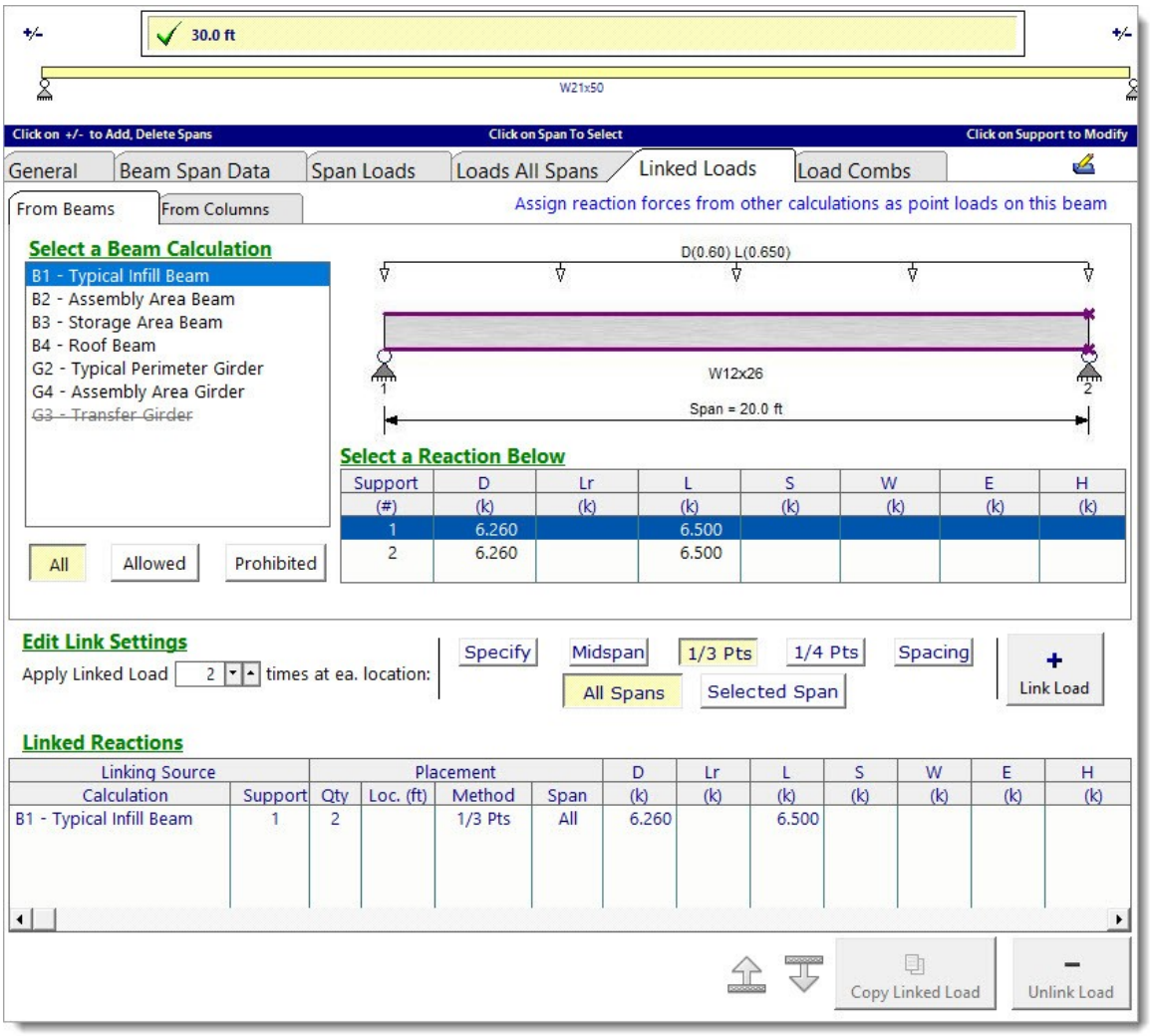
Linked Loads streamline calculations by intuitively linking reactions from one calculation to another while allowing the program to maintain the necessary calculation order for proper recalculation.

With dynamic tracking of support reactions for all members in the load path, manual transcription of reactions between modules is no longer necessary. Reactions can be easily linked from one calculation to the next, creating a seamless and robust load path. The ability to dynamically update applied forces in response to changes within the load path ensures that all calculations remain up-to-date throughout the design process, simplifying the maintenance of your calculations, and enhancing the efficiency of your workflow.

The Project Manager has also been updated to include linked load status badges which enable you to monitor and manage the entire load path within the project by identifying which calculations are up-to-date and which need recalculation. A "Recalculate Linked Loads" button has been added. Clicking this button will automatically traverse the load path of impacted calculations, initiating a recalculation for each and updating the output with a single click. Additionally, a "Calc Order" tab has been introduced in the Project Manager. This tab displays calculations in the sequence they appear within the load path, with precedent calculations listed at the top and dependent calculations appearing towards the bottom.

For more information regarding support for lined loads, see Linked Load Module Support 193.

© 2026 ENERCALC, LLC



The "Linked Loads" tab is typically divided into three main sections:

1. **Select a Beam or Column Calculation:** Viewing and selection of calculations and associated support reactions within the current project file. See Select a Beam Calculation & Select a Reaction 194.
2. **Edit Link Settings:** Load quantity and location definition. See Edit Link Settings 196.
3. **Linked Reactions:** Linked load relationships defined within the existing calculation. See Linked Reactions Table 200.

When defining new linked loads, it is recommended to begin at the top of the tab and proceed through each section sequentially down the screen.

13.1.2 Linked Load Module Support

The following modules currently support load linking:

© 2026 ENERCALC, LLC

TO / FROM	Wood Beam	Steel Beam	Wood Column	Steel Column	General Footing
Wood Beam	✓	✓	✓	✓	--
Steel Beam	✓	✓	✓	✓	--
Wood Column	--	--	--	--	✓
Steel Column	--	--	--	--	✓
General Footing	--	--	--	--	--

Note: "From" refers to the upstream elements whose reactions are being linked "to" the downstream supporting element.

The ENERCALC team is actively working on expanding this feature to other modules within SEL.

13.1.3 Select a Calculation & Select a Reaction

The **Select a Calculation** and **Select a Reaction** sections go hand-in-hand and allow for the selection and viewing of reactions from other calculations within the active project file.

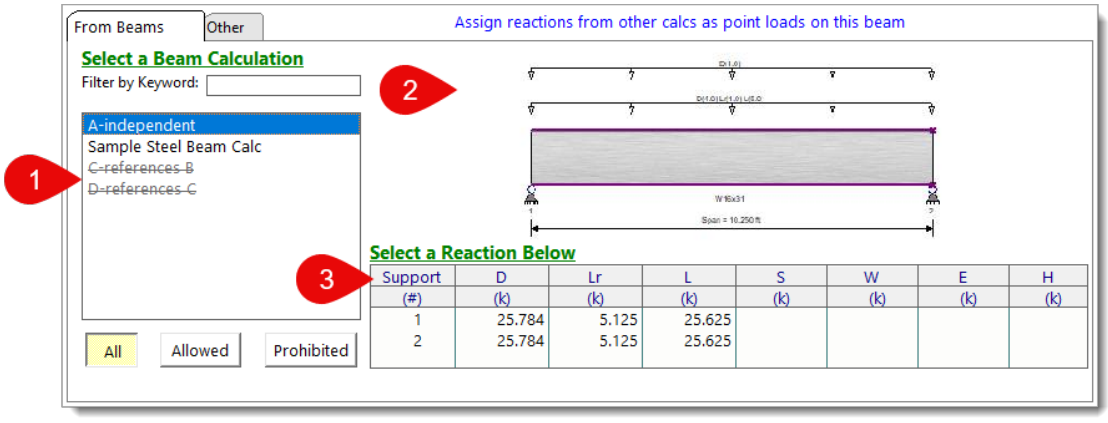
The types of calculations you can link reactions from will differ slightly based on the type of active calculation. See Linked Load Module Support 193 for more information.

The process of selecting a calculation and associated reaction vary slightly depending on the type of element:

- To link reactions from beam calculations, see Linking Reactions from Beams 194.
- To link reactions from column calculations, see Linking Reactions from Columns 195.

13.1.3.1 Linking Reactions from Beams

When linking reactions from beam calculations, there are three main components to this portion of the window:



1) Beam Calculation Table: Displays a list of beam calculations within the active project file. Creating links to calculations that are dependent on the active beam calculation is prohibited to avoid circular references. Prohibited calculations are included in the table for reference but are marked with strikethrough font. Buttons at the bottom of the table allow for the inclusion or exclusion of prohibited calculations from view.

> **TIP:** For projects with many calculations, use the **Filter by Keyword** text box to narrow down the list. Only calculations with names that contain your keyword will be shown. To make filtering even more effective, consider giving your calculations unique names or prepending them with a key identifier (such as "B1", "B2", etc.). This can help you quickly recognize and differentiate calculations at a glance.

2) Reference Sketch: A visual reference of the currently selected beam.

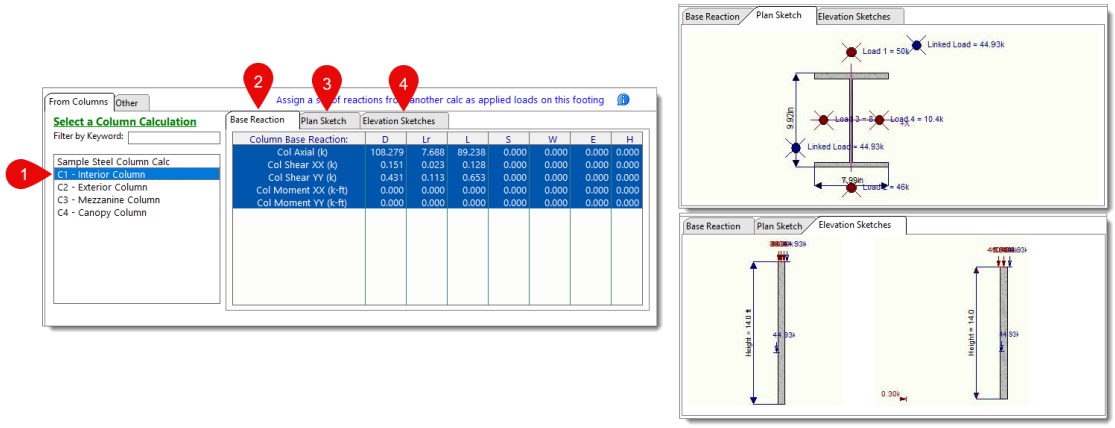
3) Reaction Table: Displays the reactions by load case at each support of the selected beam. Reactions for D, Lr, L, S, and H are shown at the service level, while W and E are presented at the strength level. It is important to note that these reactions are already formatted for direct incorporation into further calculations, ensuring there is no double-factoring of the referenced reaction forces.

> **NOTE:** Since loads are linked on a load case basis, any live load reductions which have been manually accounted for on an upstream live load will carry through to downstream calculations. For example, if a 20 psf roof live load (unreduced) was manually input on a roof joist at 16 psf (reduced), the resulting "L Only" reaction would be based on 16 psf. The associated linked reaction on the supporting girder would also be based on the manually inputted 16 psf.

Once you've selected the reaction you'd like to link to the current calculation, move on to Edit Link Settings 196.

13.1.3.2 Linking Reactions from Columns

When linking reactions from column calculations, there are four main components to this portion of the window:



1) Column Calculation Table: Displays a list of column calculations within the active project file.

> **TIP:** For projects with many calculations, use the **Filter by Keyword** text box to narrow down the list. Only calculations with names that contain your keyword will be shown. To make filtering even more effective, consider giving your calculations unique names or prepending them with a key identifier (such as "B1", "B2", etc.). This can help you quickly recognize and differentiate calculations at a glance.

2) Base Reaction Table: Displays all base reactions (axial, shear, and moment) by load case for the selected column. Reactions for D, Lr, L, S, and H are shown at the service level, while W and E are presented at the strength level. It is important to note that these reactions are already formatted for direct incorporation into further calculations, ensuring there is no double-factoring of the referenced reaction forces.

> **NOTE:**
> - The sign conventions displayed in this table are relative to the column's local coordinate system. For example, a column with (+) loading in the X direction will tend to have a (-) shear base reaction, and a column with (-) loading in the X direction will tend to have a (+) shear base reaction. Similar is true for the Y direction.
> - Column base reactions are always linked in full for General Footings. This means that when you link a column reaction to a footing, all components (axial, shear, and moment) are included. When you select the base reaction, all associated components are selected together.

3) Reference Plan Sketch: A visual reference of the plan view for the currently selected column.

4) Reference Elevation Sketches: A visual reference of the elevation views for the currently selected column.

Once you've selected the reaction you'd like to link to the current calculation, move on to Edit Link Settings 196.

13.1.4 Edit Link Settings

After selecting a support reaction, you must define how the support reaction will be applied to the active calculation.

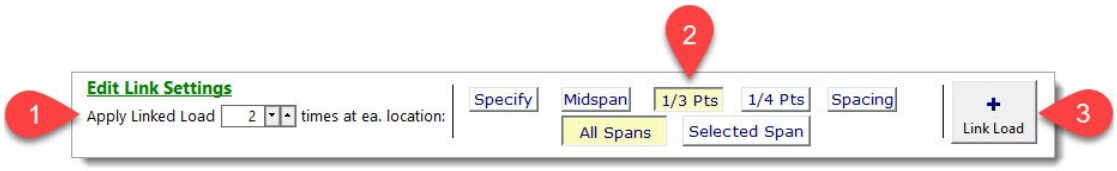
The **Link Settings** will differ slightly based on the type of active calculation.

- To edit link settings when the active calculation is a beam, see Link Settings for Beams 197.
- To edit link settings when the active calculation is a column, see Link Settings for Columns 198.
- To edit link settings when the active calculation is a general footing, see Link Settings for General Footings 199.

© 2026 ENERCALC, LLC

13.1.4.1 Link Settings for Beams

When linking reactions to the active beam calculation, there are three main components to this portion of the window:



1) Linked Load Quantity: Start by specifying how many times the support reaction should occur at each location by updating the **"Apply Linked Load ___ times at ea. location"** input. This can be useful when a given support reaction occurs multiple times at a single location on the active beam (for example, when an infill beam frames into each side of a girder).

2) Linked Load Location: Next, specify where the support reaction occurs on the current beam. The available options are:

Specify: Specify an exact location for the load.

- Selecting "Beam" measures the dimension from the far left end of the beam regardless of support conditions. The resulting linked load will always be at this fixed dimension even if the underlying span lengths and quantities change. This dimensional value is restricted to the maximum length of the entire beam. If span length adjustments cause the load to extend beyond the beam's end, it will automatically reposition to the far right end of the beam.
- Selecting "Selected Span" measures the dimension from the left end of the currently selected span. The resulting linked load will always correspond to the span index of the selected span. For example, if there are two spans and the second span is selected, the linked load will apply to the second span (span index = 2). As long as there are at least two spans, the linked load will consistently be associated with span index 2, regardless of any spans that are added or deleted in the future.

Midspan, 1/3 Pts, & 1/4 Pts: Automatically place the load at the midspan, one-third points, or one-quarter points. These locations are relative to the span length and will adjust automatically if the span lengths change.

- Selecting "All Spans" will apply the linked load to all spans within the currently active beam calculation, including any new spans which may be created in the future.
- Selecting "Selected Span" will apply the linked load to the span index of the currently selected span. As described above, the resulting linked load will always correspond to the span index of the selected span regardless of any spans that are added or deleted in the future.

© 2026 ENERCALC, LLC

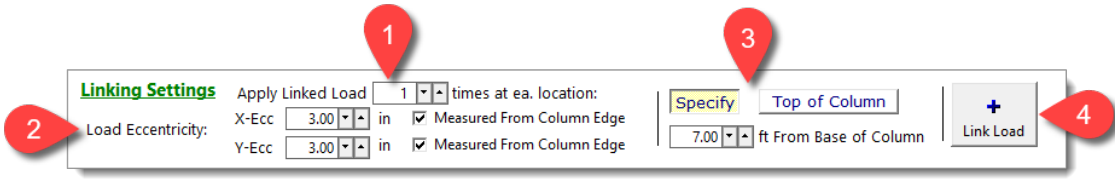
Spacing: Specify repetitive loads by defining a start point and a spacing. The first load will be placed at the "From" dimension, and additional loads will be repeated at the defined spacing throughout the entire length of the beam.

3) Add the Linked Load: Once you have defined the settings for the linked load, click the **Link Load** button.

Once you've click the **Link Load** button, you will see that the new linked load has been added to the Linked Reactions Table 200.

13.1.4.2 Link Settings for Columns

When linking reactions to the active column calculation, there are four main components to this portion of the window:



1) Linked Load Quantity: Start by specifying how many times the support reaction should occur at each location by updating the **"Apply Linked Load ___ times at ea. location"** input. This can be useful when a given support reaction occurs multiple times at a single location on the active column (for example, when multiple beams frame into each side of a column).

2) Load Eccentricity: Specify the "X" and "Y" eccentricities to generate moments about the Y-Y and X-X axes, respectively. You can also choose whether the eccentricity is measured from the column centroid or from the column edge, similar to how typical applied loads are handled.

3) Linked Load Location: Define the vertical position of the axial load relative to the bottom of the column. Options include:

Specify: Manually enter an exact location from the bottom of the column.

Top of Column: The load is applied at the top of column. This location will automatically adjust if the column height changes.

4) Add the Linked Load: Once you have defined the settings for the linked load, click the **Link Load** button.

Once you've click the **Link Load** button, you will see that the new linked load has been added to the Linked Reactions Table 200.

© 2026 ENERCALC, LLC

13.1.4.3 Link Settings for General Footings

When linking reactions to the active general footing calculation, there are three main components to this portion of the window:

1) Align Column +X to: Start by selecting how the column should be oriented relative to the General Footing. Since columns and footings use different local axis conventions, it's important to define the column's orientation accurately.

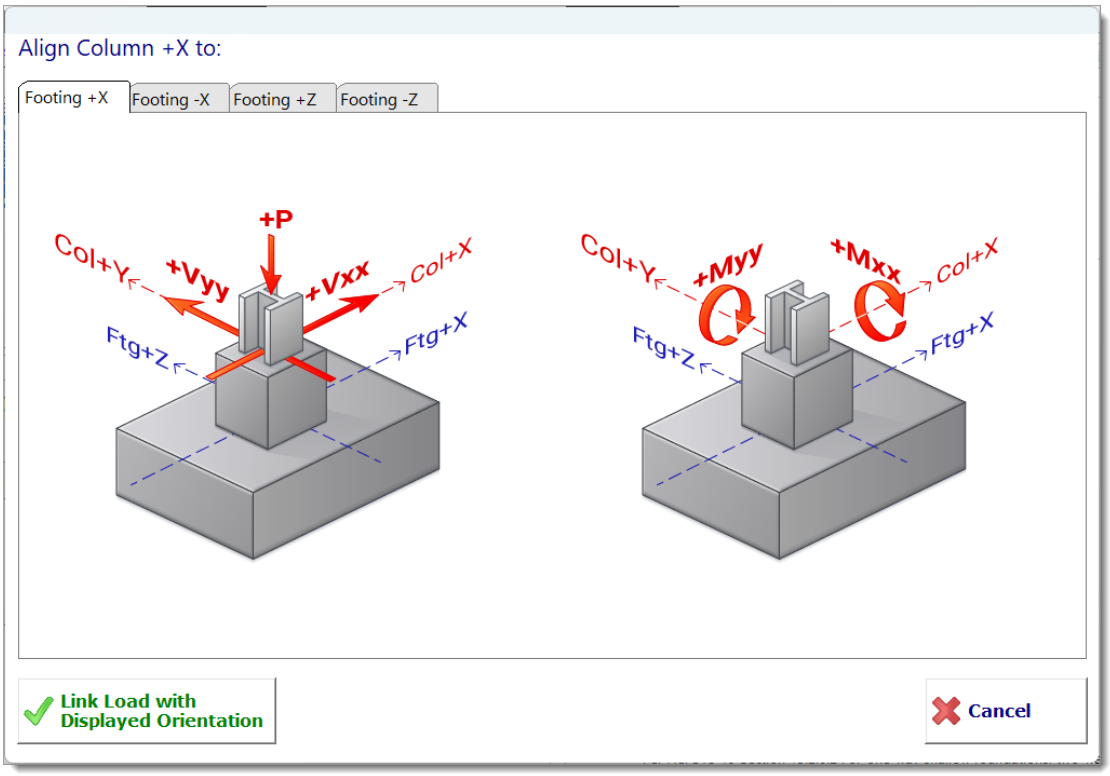
NOTE:

- Reactions are always applied equal and opposite from the upstream member. For example, if the upstream column calculation reports a +Vxx reaction, this will impose a -Vxx reaction on the footing.
- Additionally, by specifying how the column's local +X axis aligns with the footing's local +X axis, the program can correctly interpret the direction of all shears and moments as they relate to the footing.
- All required sign conversions are handled automatically and will account for differences in local axis conventions.

2) Info button: Click this button to view a graphic that illustrates how different column orientations and their reactions apply to the supporting footing.

By changing tabs at the top of the window, you can view the various options for column orientation. For convenience, you can also click **Link Load with Displayed Orientation** at the bottom of the window to apply the reactions using the current view.

© 2026 ENERCALC, LLC



3) Add the Linked Load: Once you've defined the orientation settings, click **Link Load** to add the linked reactions.

> **NOTE:** The General Footing module supports only one linked load at a time. If the Link Load button is disabled, it usually means a linked load already exists. In that case, select the existing load from the Linked Reactions Table 202 and remove it before adding a new one.

Once you've click the **Link Load** button, you will see that the new linked load has been added to the Linked Reactions Table 200.

13.1.5 Linked Reactions Table

This table displays all support reactions and their associated link settings that are linked to the current calculation.

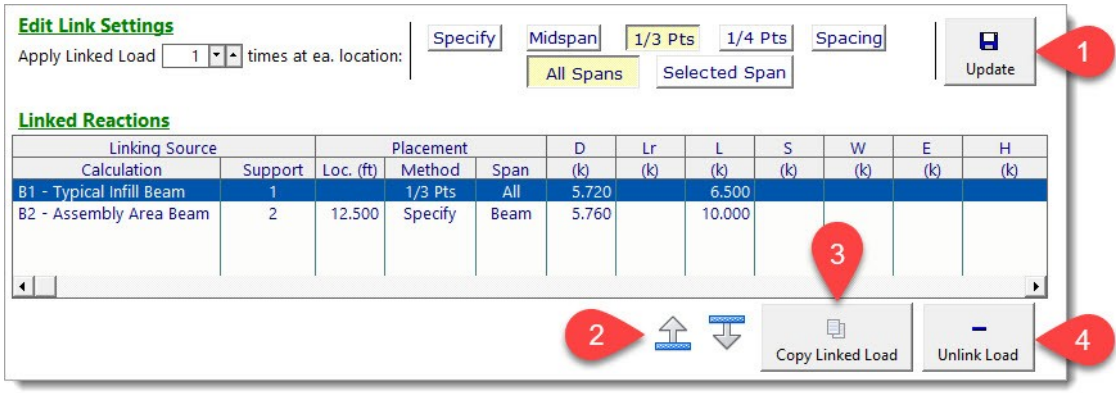
The contents of the **Linked Reactions** table will vary slightly depending on the type of active calculation:

- When the active calculation is a beam, see Linked Reactions Table for Beams 201.
- When the active calculation is a column, see Linked Reactions Table for Columns 201.
- When the active calculation is a general footing, see Linked Reactions Table for General Footings 202.

© 2026 ENERCALC, LLC

13.1.5.1 Linked Reactions Table for Beams

The **Linked Reactions** table displays all support reactions and their associated link settings that are linked to the active beam calculation.



1) To Edit a Linked Load: Select a linked load from the table. The Edit Link Settings[197] section will populate with the existing link settings. Modify the settings as desired and click the **Update** button.

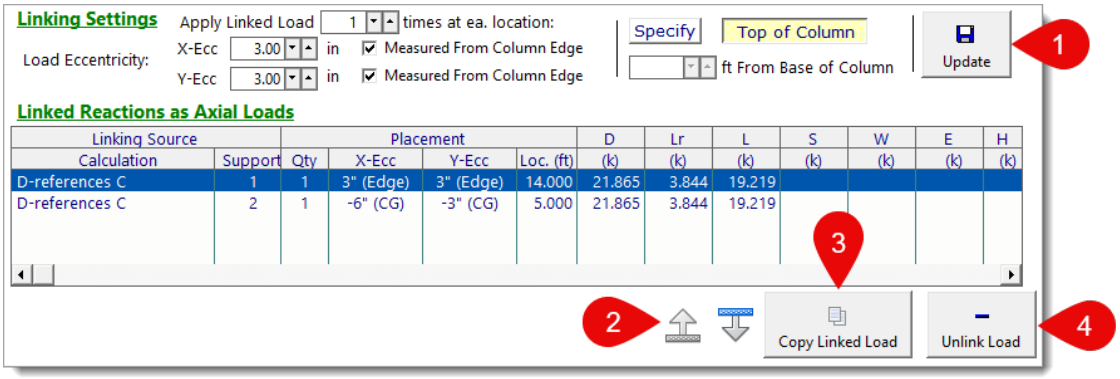
2) To Reorganize the Display Order of Linked Loads: Select a linked load from the table and click the **Shift Up** or **Shift Down** buttons.

3) To Duplicate a Linked Load: Select a linked load from the table and click the **Copy Linked Load** button.

4) To Delete a Linked Load: Select a linked load from the table and click the **Unlink Load** button.

13.1.5.2 Linked Reactions Table for Columns

The **Linked Reactions** table displays all support reactions and their associated link settings that are linked to the active column calculation.



1) To Edit a Linked Load: Select a linked load from the table. The Edit Link Settings 198 section will populate with the existing link settings. Modify the settings as desired and click the **Update** button.

2) To Reorganize the Display Order of Linked Loads: Select a linked load from the table and click the **Shift Up** or **Shift Down** buttons.

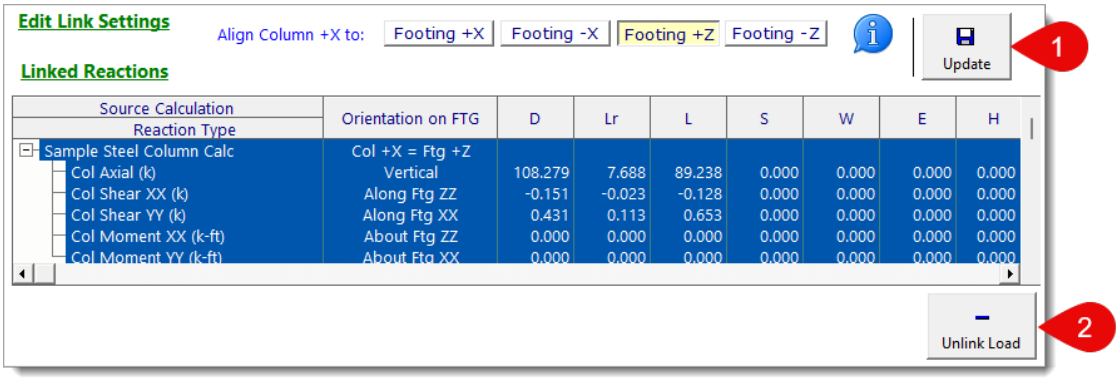
3) To Duplicate a Linked Load: Select a linked load from the table and click the **Copy Linked Load** button.

4) To Delete a Linked Load: Select a linked load from the table and click the **Unlink Load** button.

13.1.5.3 Linked Reactions Table for General Footings

The **Linked Reactions** table displays all support reactions and their associated link settings that are linked to the active general footing calculation.

NOTE: The sign conventions shown in this table are relative to the footing's local coordinate system. All required sign conversions are handled automatically based on the specified Link Settings 199 and will account for differences in local axis conventions. It can be helpful to compare the values in this table with those in the Base Reaction Table 195 to see how orientation settings affect how the reactions are applied to the footing.



1) To Edit a Linked Load: Select the existing linked load from the table. The Edit Link Settings 199 section will populate with the existing link settings. Modify the settings as desired and click the **Update** button. You can also use the **Info button** to edit the existing linked load as well.

NOTE: Column base reactions are always linked in full. This means that when you link a column reaction to a General Footing, all components (axial, shear, and moment) are included. When you select an existing linked reaction, all associated components are selected together.

2) To Delete a Linked Load: Select a linked load from the table and click the **Unlink Load** button.

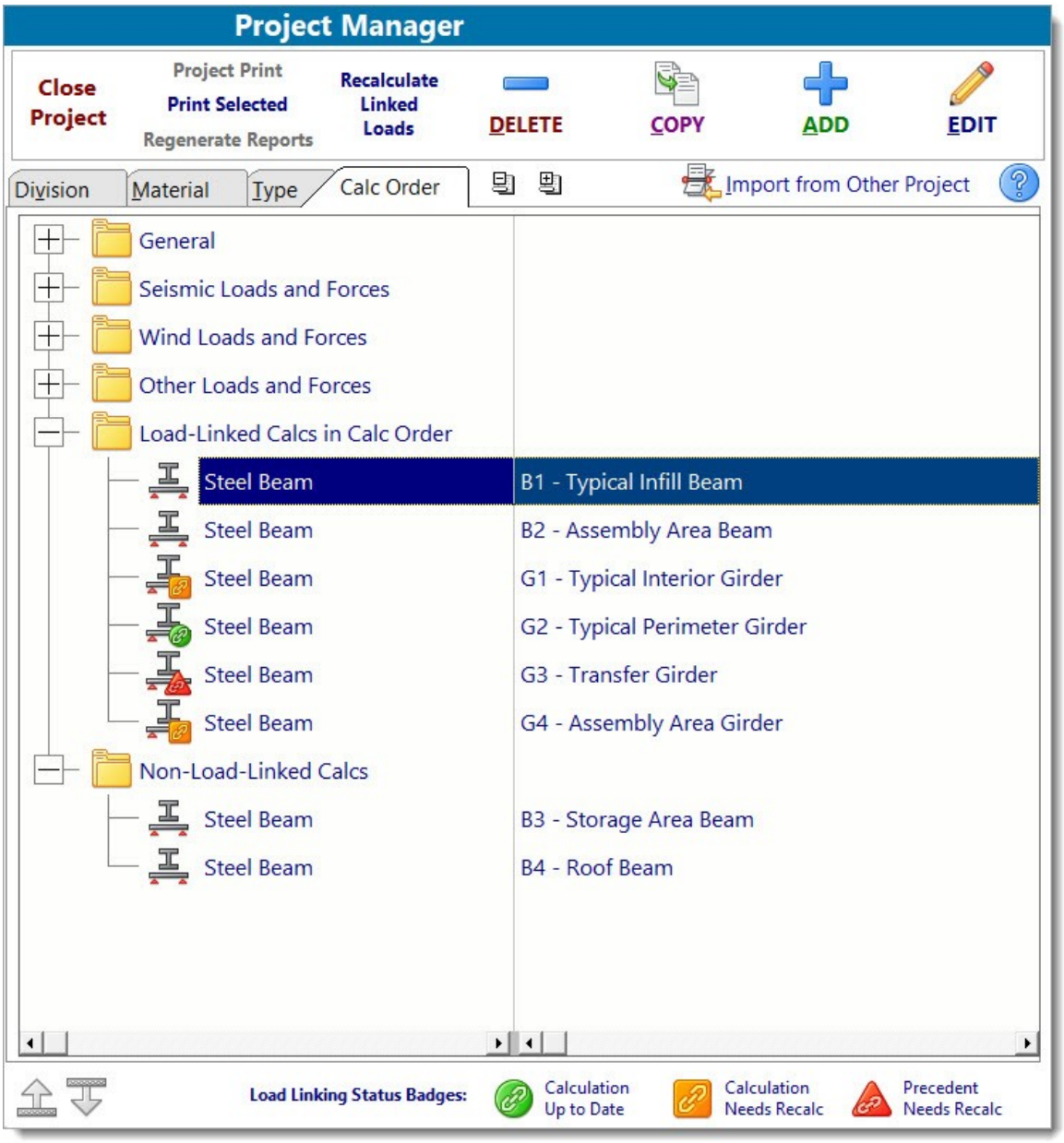
© 2026 ENERCALC, LLC

NOTE: The General Footing module supports only one linked load at a time. If the Add Link Load button is disabled, this usually means a linked load is already present. In that case, you'll need to select and remove the existing linked load before adding a new one.

13.1.6 Status Badges and Linked Load Recalculation

The Project Manager includes linked load status badges for calculations that contain linked loads. Calculations which do not contain linked loads will not display a badge. Additionally, a "Calc Order" tab has been introduced in the Project Manager. This tab displays calculations in the sequence they appear within the load path, with precedent calculations listed at the top and dependent calculations appearing towards the bottom. Both of these features help you monitor and manage the load path within a project by quickly identifying which calculations are up-to-date and which need recalculation.

© 2026 ENERCALC, LLC



Calculations with linked loads will display one of the following statuses:

- **Calculation Up to Date:** Indicates that the current calculation and all its precedent calculations are up to date.
- **Calculation Needs Recalc:** Indicates that the current calculation is referencing out-of-date reactions, but all precedent calculations are up-to-date. Triggering a recalc on these calculations (for example, by opening, saving, and closing the calculation) will resolve the status.
- **Precedent Needs Recalc:** Indicates that a precedent calculation is still using out-of-date linked loads. Therefore, its associated reactions on the current calculation are

© 2026 ENERCALC, LLC

out of date as well. Triggering a recalc on these calculations will not resolve the status because the current calculation will remain out of date as long as any precedent calculation is out of date.

NOTE: It is important to recognize that the linked load status is entirely independent from the "OK"/"NG" design status of the associated calculation. For example, a calculation showing as **Calculation Up to Date** does not mean the calculation is passing unity checks or prescriptive code requirements. You will still need to ensure the accuracy and integrity of your project calculations throughout the design process.

Resolving Out-of-Date Calculations

To resolve out-of-date calculations, you can recalculate Linked Loads automatically by clicking the **Recalculate Linked Loads** button. This feature will automatically traverse the load path, initiating a recalculation for each calculation and updating all calculations with a single click.

Alternatively, you may wish to resolve the out-of-date calculations manually so that you can monitor how changes to precedent calculations affect the dependent calculations further down the load path. You can manually resolve out-of-date calculations by following these general steps:

1. Open any calculation marked as "Calculation Needs Recalc." This action will trigger a recalculation of that calculation.
2. Make any necessary changes to ensure the design is satisfactory given the new linked load reactions.
3. Save & Close the calculation.
4. The calculation should now report as "Calculation Up to Date" within the Project Manager, and any dependent calculations should change from "Precedent Needs Recalc" to "Calculation Needs Recalc."
   a. **NOTE:** If a dependent calculation still reports "Precedent Needs Recalc," it indicates that it has multiple precedents, some of which are still out of date. Repeat the process outlined above for other precedent calculations that need recalculation.
5. Repeat this process until all calculations are reporting as "Calculation Up to Date".

TIP: Utilize the Calc Order tab view within the Project Manager to display calculations in the sequence they appear within the load path. This simplifies the process of manually resolving out-of-date calculations, as you can address them sequentially, working your way down the list one by one.

Example: Manually Resolving Out-of-Date Calculations

The following represents a work flow example of what it looks like to manually resolve out-of-date calculations. Imagine the following load scenario:

- B1 is a typical infill beam.
- B2 is a typical girder which supports B1.

© 2026 ENERCALC, LLC

- B3 is a transfer girder which support B2.

Assuming all reactions are up-to-date at the start of this example, the linked load status badges would be as follows:

- B1 = No linked load status badge. It is at the beginning of the load path and is therefore not referencing any linked loads.
- B2 = **Calculation Up to Date**
- B3 = **Calculation Up to Date**

Now, we open B1 and modify the calculation in such a way to change the reactions (updated the span loads, span length, section size, etc.). The new linked load status badges would be as follows:

- B1 = No linked load status badge.
- B2 = **Calculation Needs Recalc**. B2 is still referencing B1's old reactions and needs to be recalculated to process B1's new reactions.
- B3 = **Precedent Needs Recalc**. B3 is still referencing B2's old reactions. However, B2's reactions are also out-of-date. We must recalculate B2 before we recalculate B3.

Next, we open B2 to trigger a recalculation, make any necessary updates to ensure a good design, then click Save & Close. The new linked load status badges would be as follows:

- B1 = No linked load status badge.
- B2 = **Calculation Up to Date**. B2 has been recalculated using B1's new reactions.
- B3 = **Calculation Needs Recalc**. B3 is still referencing B2's old reactions. However, since B2 is now up-to-date, we can now safely recalculate B3.

Finally, we open B3 to trigger a recalculation, make any necessary updates to ensure a good design, then click Save & Close. The new linked load status badges would be as follows:

- B1 = No linked load status badge.
- B2 = **Calculation Up to Date**.
- B3 = **Calculation Up to Date**. B3 has been recalculated using B2's new reactions.

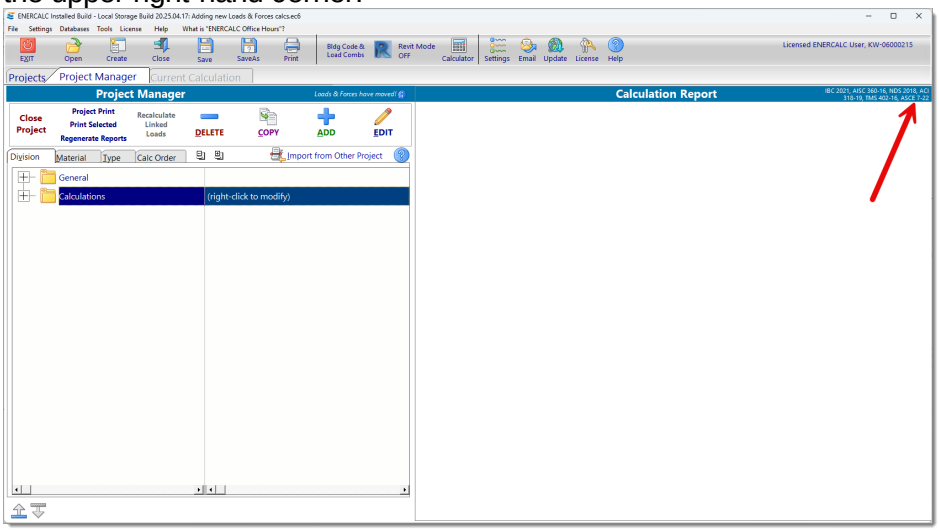
© 2026 ENERCALC, LLC

13.2 Loads & Forces Divisions

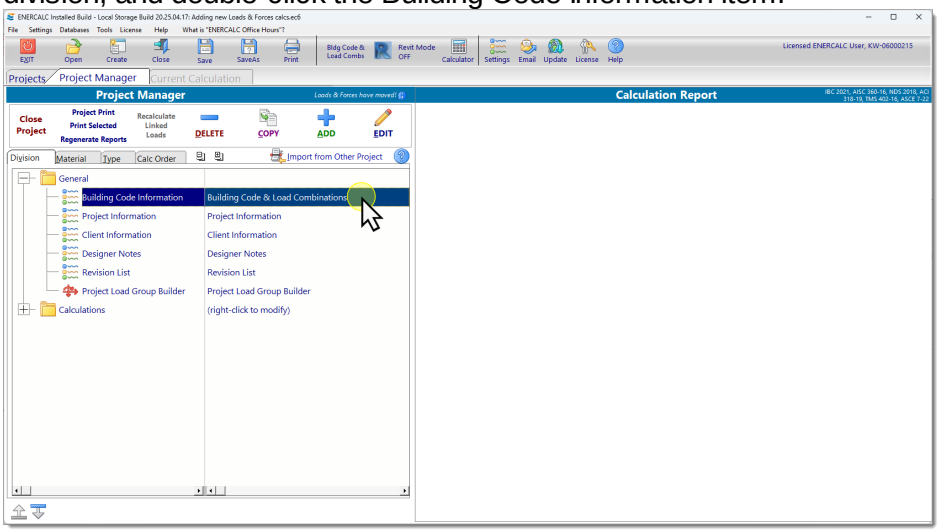
Starting with ASCE 7-22, Loads & Forces calculations are added to a Project File in the same way other calculations are added.

Video: Adding ASCE 7-22 Loads & Forces Modules to Project File

Make sure the project file is set to ASCE 7-22 or later by checking the ASCE 7 reference in the upper right-hand corner.

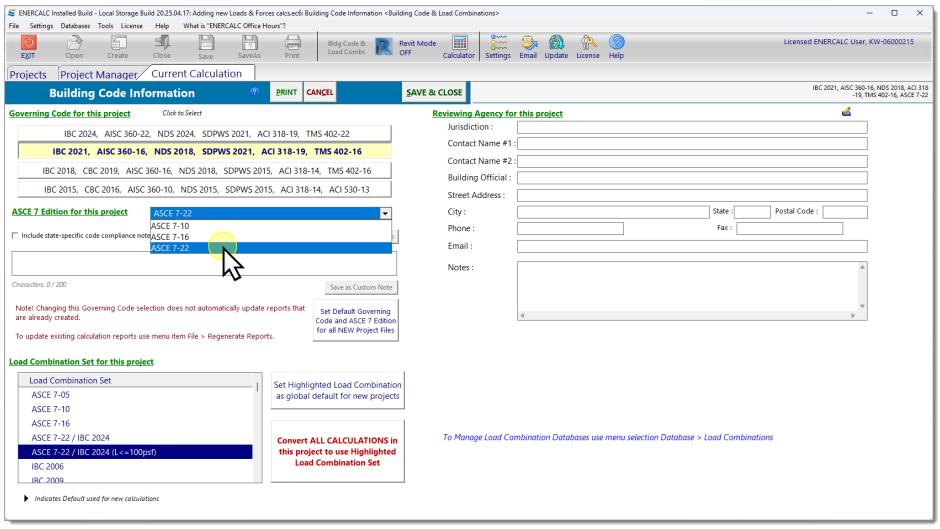


If the ASCE 7 edition is set to a version earlier than ASCE 7-22, expand the General division, and double-click the Building Code Information item.

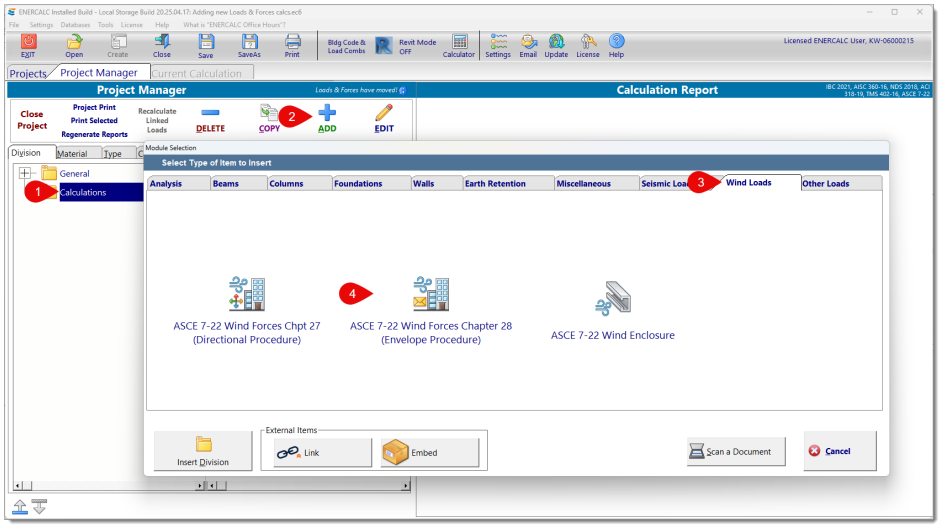


Use the ASCE 7 Edition selector to choose ASCE 7-22 or later, and then click Save & Close.

© 2026 ENERCALC, LLC

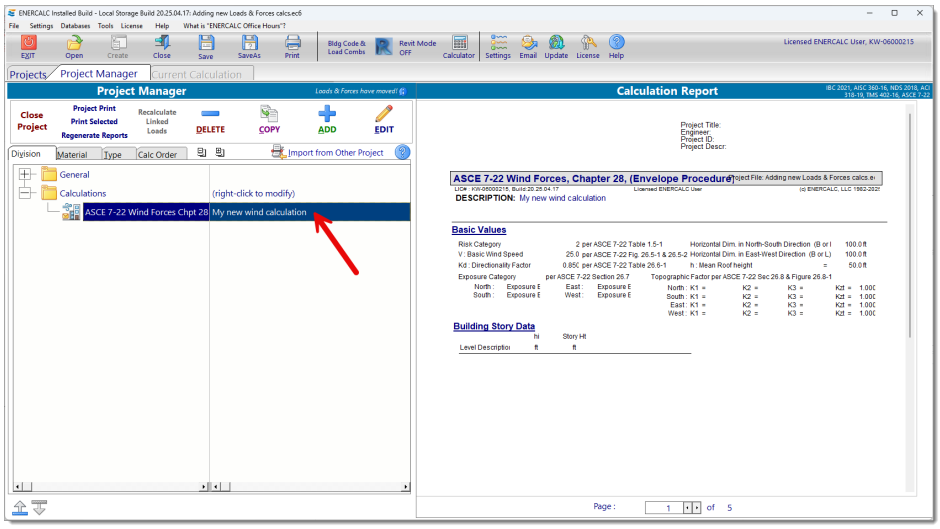


Click on the Calculations folder, then click Add, then choose the desired tab and calculation:



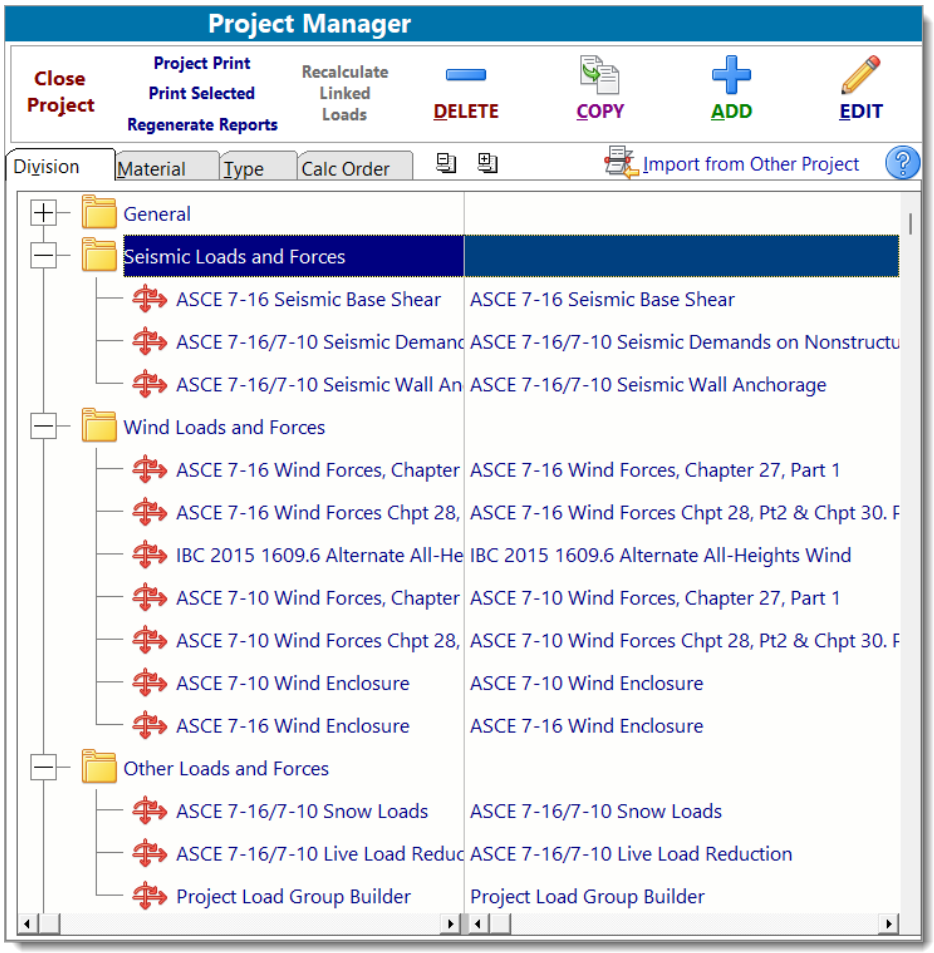
The screen will display the new calculation form, and when you save and close, the calculation will appear in the Project Manager:

© 2026 ENERCALC, LLC



Historical Note: When Loads & Forces calculations were first introduced, they were automatically included in the Project Manager when starting a new project file, appearing as a set of preloaded Loads & Forces divisions. These divisions and modules will continue to appear in ENERCALC SEL project files when the governing code is set to ASCE 7-10 or ASCE 7-16.

© 2026 ENERCALC, LLC



Each project file includes the following Loads & Forces divisions:

- Seismic Loads & Forces
- Wind Loads & Forces
- Other Loads & Forces

Each division contains modules for calculating the respective types of loads and forces. Use of these modules is optional, but they cannot be moved out of their assigned division. Likewise, other calculation modules cannot be added to or placed inside these divisions.

As **ENERCALC SEL** continues to evolve, these Loads & Forces divisions will include more tools to support the development of project load calculations.

13.2.1 ASCE Seismic Base Shear

This module is a presentation of equivalent lateral force procedure seismic provisions in ASCE 7.

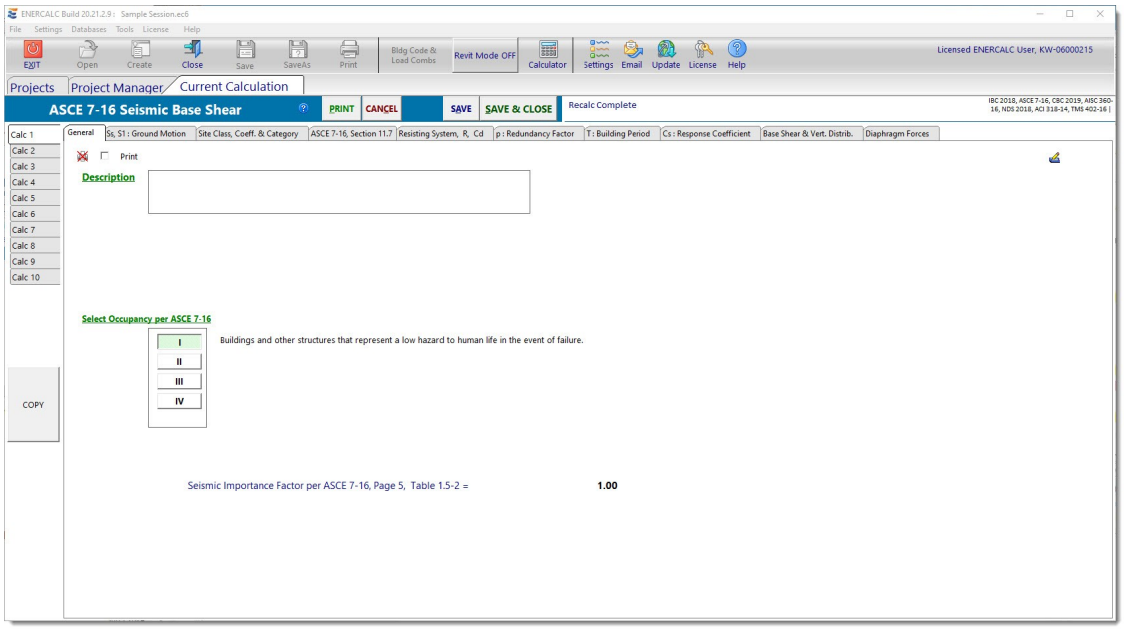
All of the references to ASCE 7 are given on the module screens. Above each screen capture are notes as needed to call attention to entries or explain usage.

© 2026 ENERCALC, LLC

The module is designed to allow you to work downwards through the tabs on the left side of the screen. ASCE 7 skips all around in its definitions of things to check. This module simplifies the process by properly ordering the items and the necessary decisions to be made in working toward a final seismic base shear value.

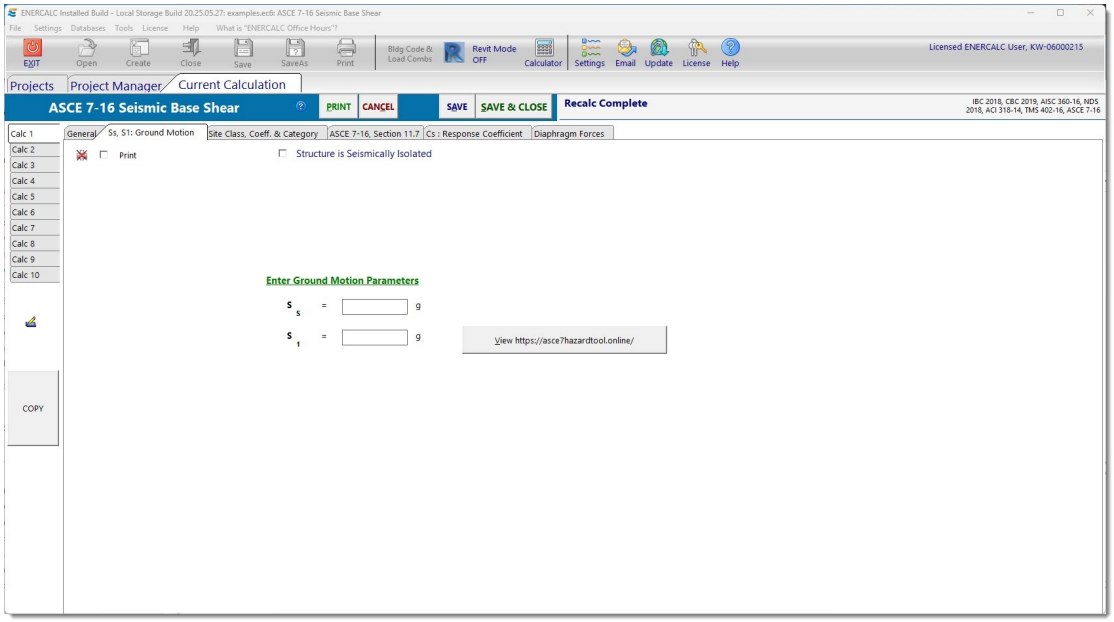
The program includes a link to the USGS databases of seismic ground motion specifically for use with ASCE 7-16.

General



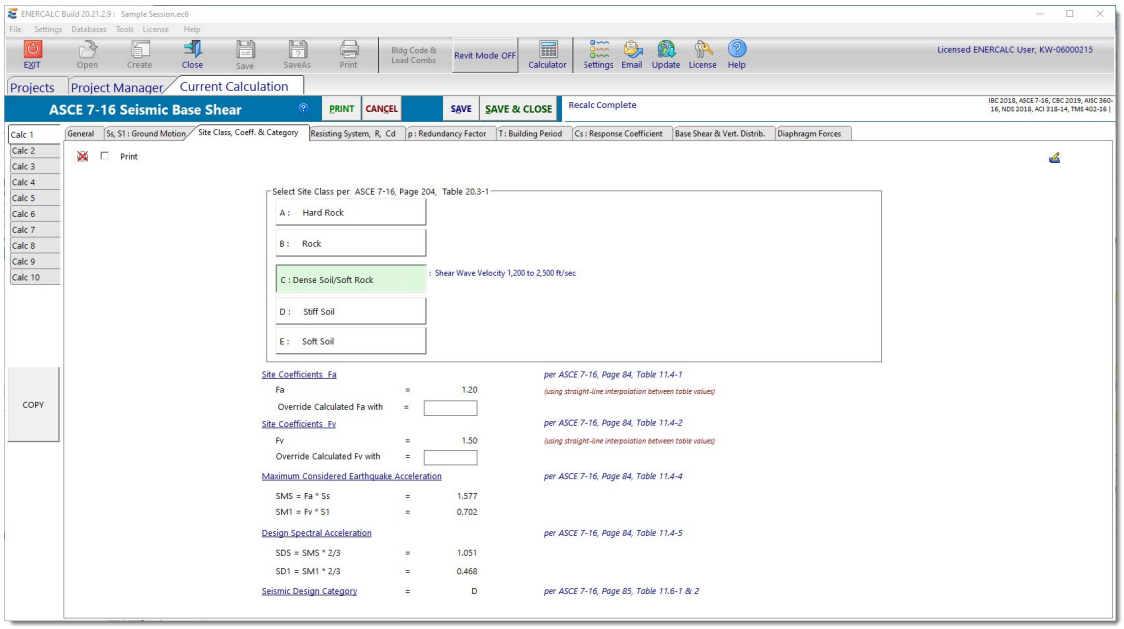
© 2026 ENERCALC, LLC

Ground Motion



Site Class, Site Coefficients and Seismic Design Category

Select the appropriate Site Class based on the geotechnical conditions.

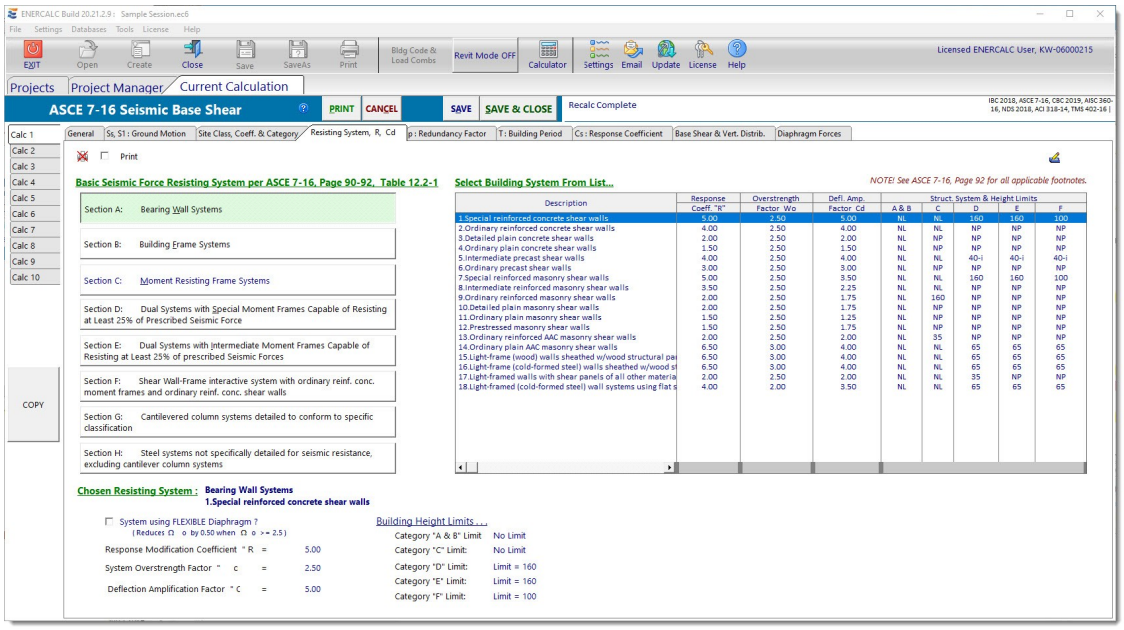


© 2026 ENERCALC, LLC

Selection of Seismic Force Resisting System

Click one of the large boxes at the top of the tab to load the table with specific selections for that general category of Seismic Force Resisting System.

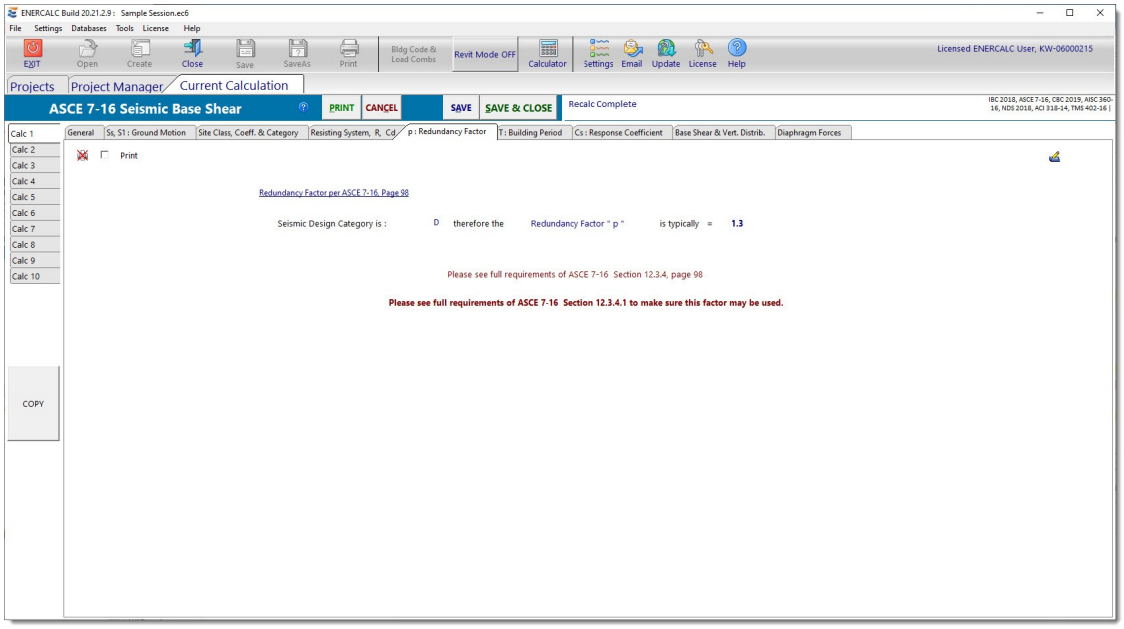
Note the checkbox for systems with flexible diaphragms. This may not apply to some building systems...check ASCE for details on the particular system of interest.



© 2026 ENERCALC, LLC

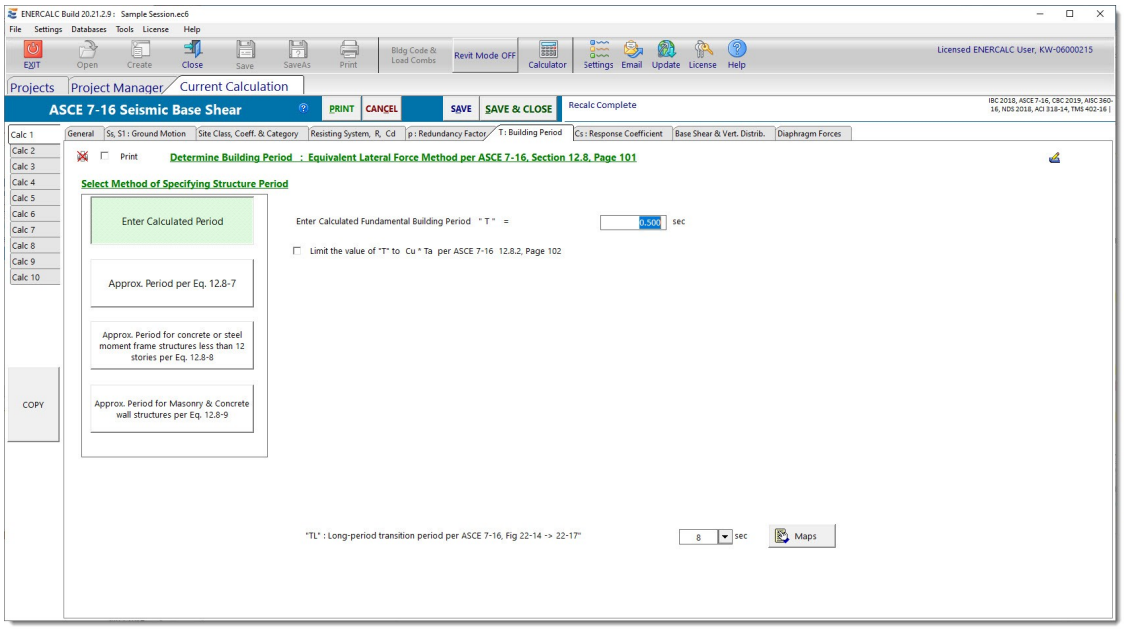
Redundancy Factor

Depending on the Seismic Design Category determined from the prior selections of Occupancy, Seismic Ground Motion and Building System, the Redundancy Factor per ASCE will be shown here.



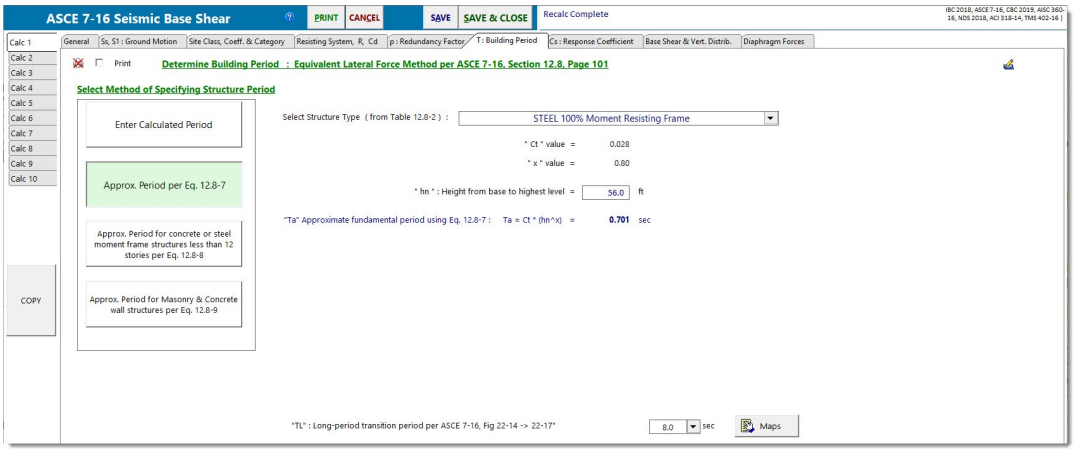
Specification of Building Period

All four ASCE 7 options for building period determination are available here.

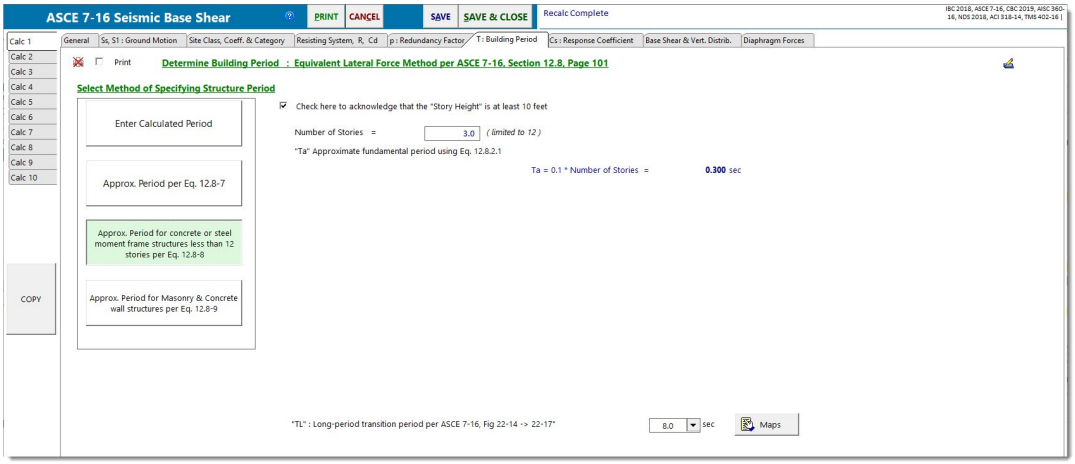


© 2026 ENERCALC, LLC

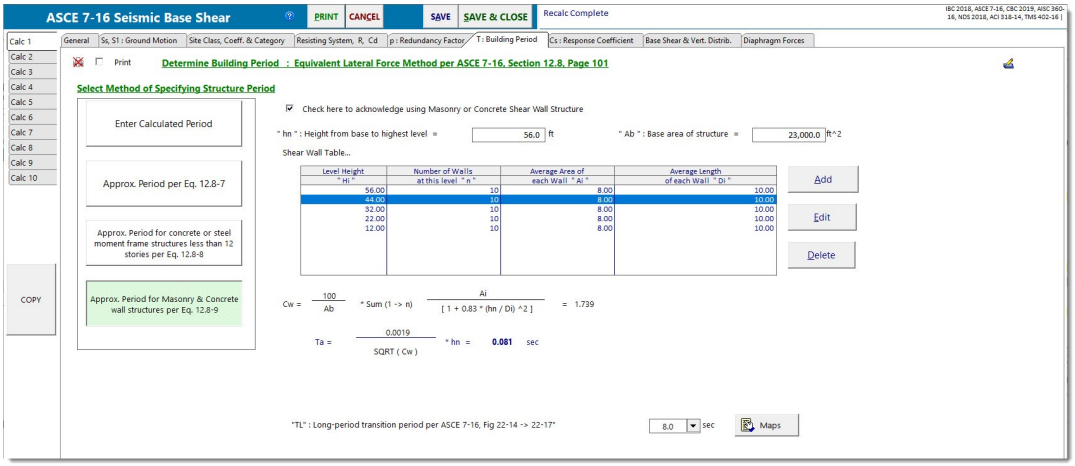
Evaluation of Approximate Building Period



Approximate Period for Steel & Concrete Frames



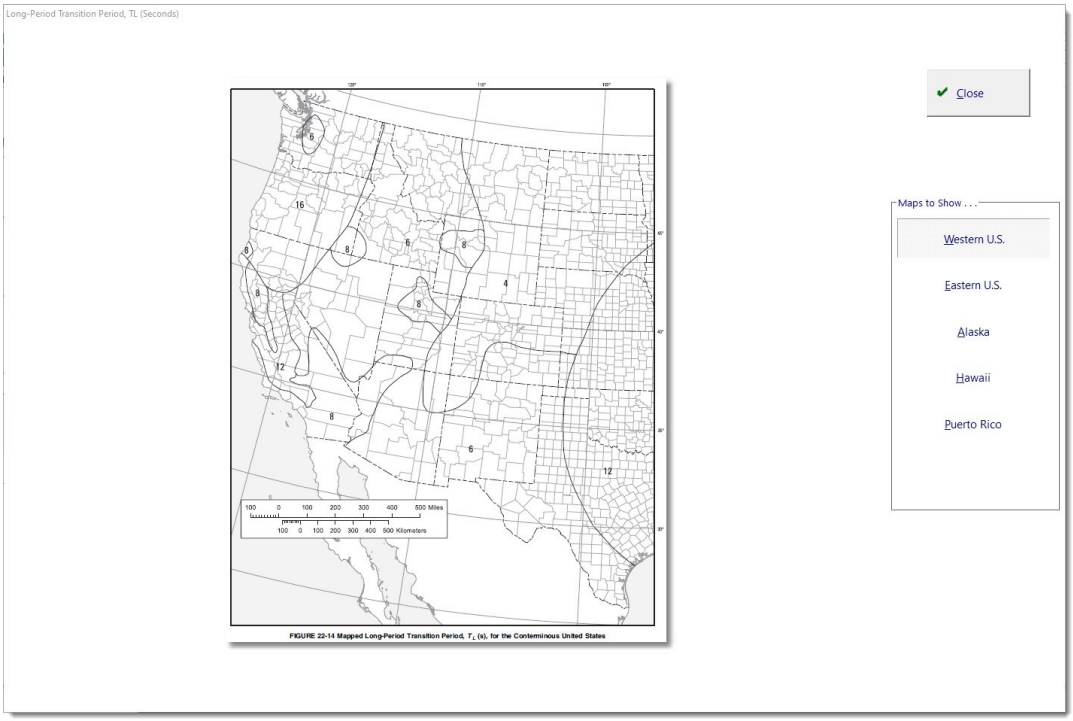
Approximate Period for Concrete or Masonry Shear Wall Structures



© 2026 ENERCALC, LLC

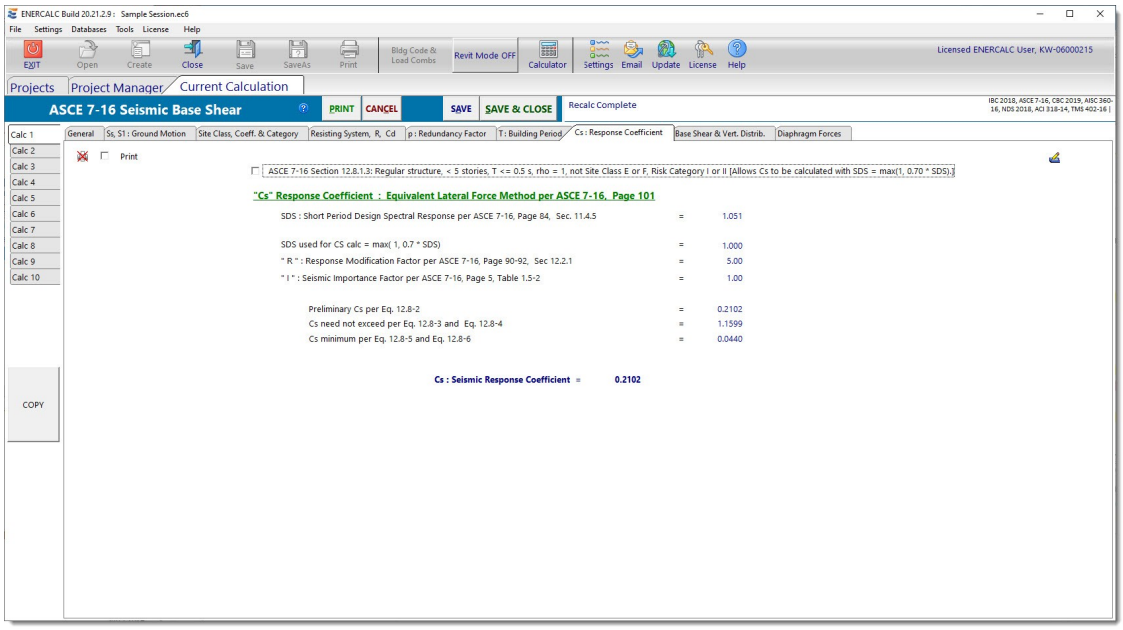
Long-Period Transition Period Reference Maps

Maps copyright ASCE.



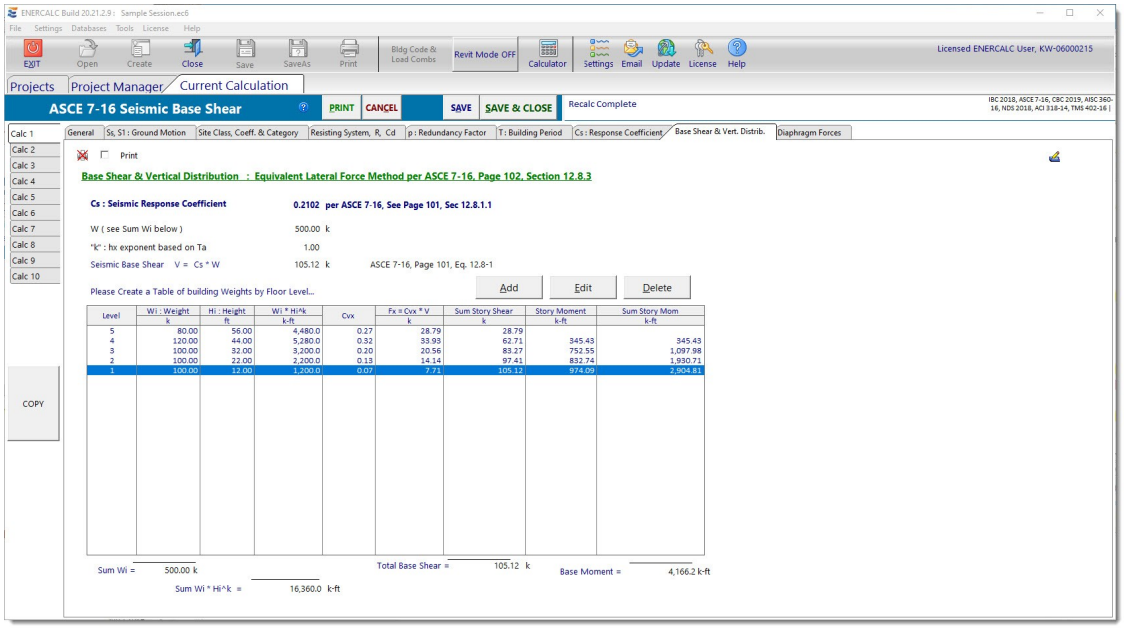
Cs: Seismic Response Coefficient

Please check ASCE 7 section references for details.



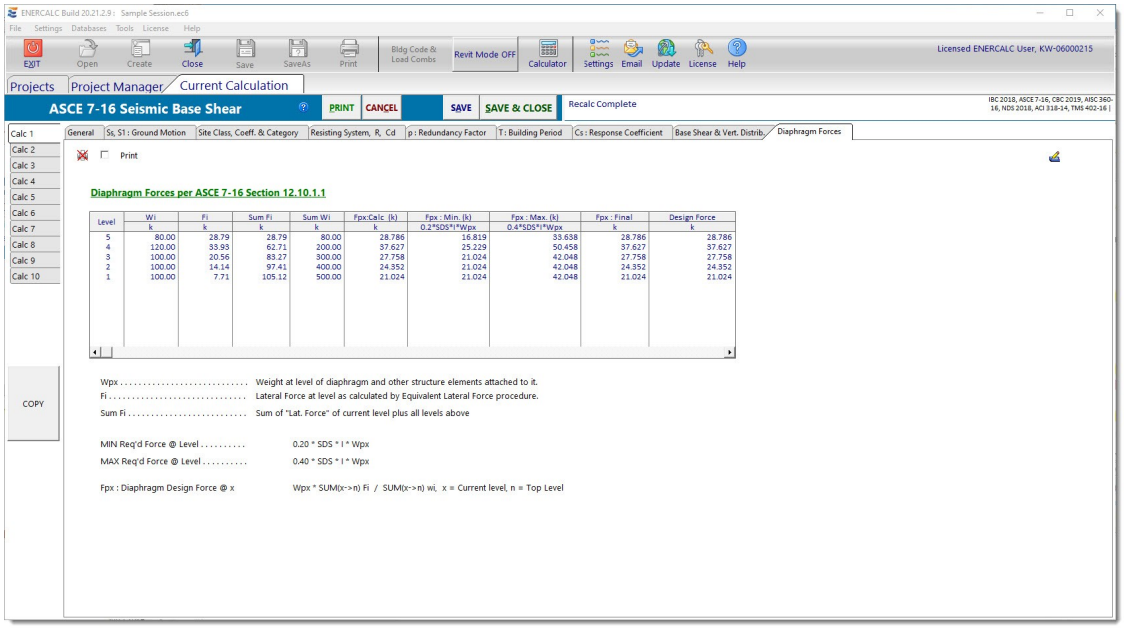
Vertical Distribution of Base Shear

This tab calculates the vertical distribution of seismic forces. Use the [**Add**] button for each new level. To edit information for a level, highlight the corresponding line and click [**Edit**].



Diaphragm Forces

Using the information entered above (on the prior tab) the force distribution results are shown here.



© 2026 ENERCALC, LLC

13.2.2 ASCE 7-22 Seismic Base Shear

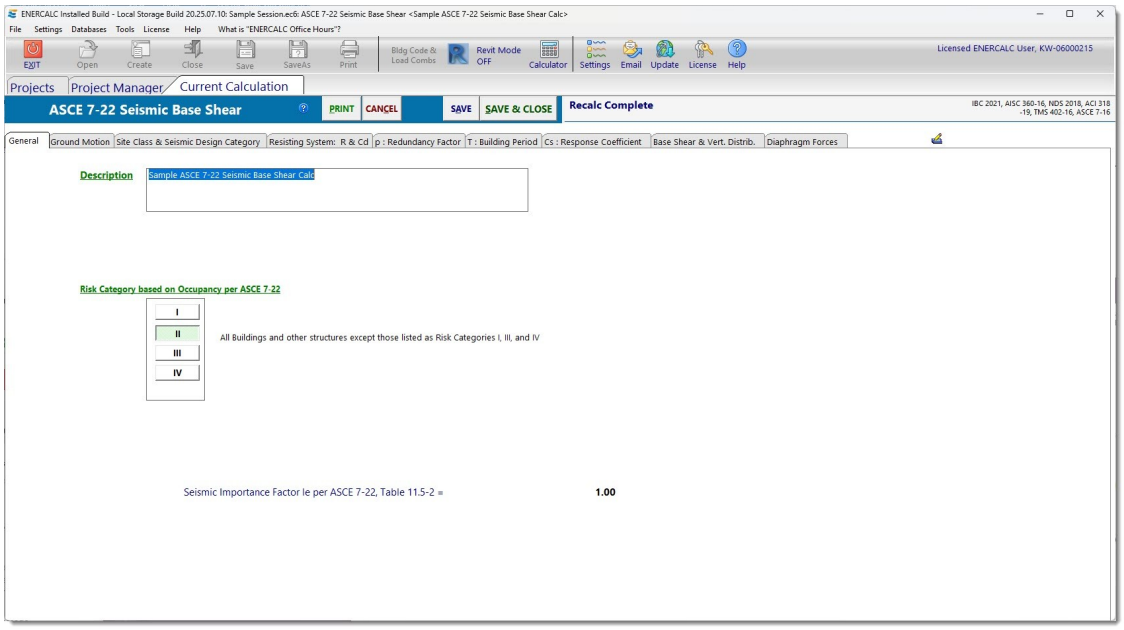
This module is a presentation of equivalent lateral force procedure seismic provisions in ASCE 7-22.

All of the references to ASCE 7 are given on the module screens. Above each screen capture are notes as needed to call attention to entries or explain usage.

The module is designed to allow you to work downwards through the tabs on the left side of the screen. ASCE 7 skips all around in its definitions of things to check. This module simplifies the process by properly ordering the items and the necessary decisions to be made in working toward a final seismic base shear value.

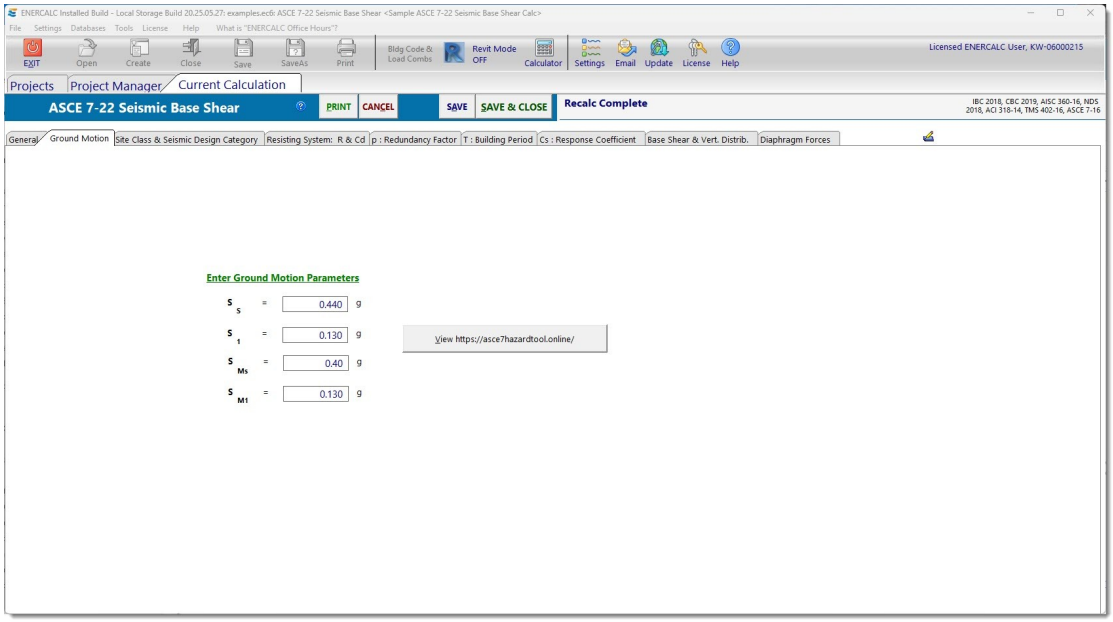
The program includes a link to the USGS databases of seismic ground motion specifically for use with ASCE 7-22.

General



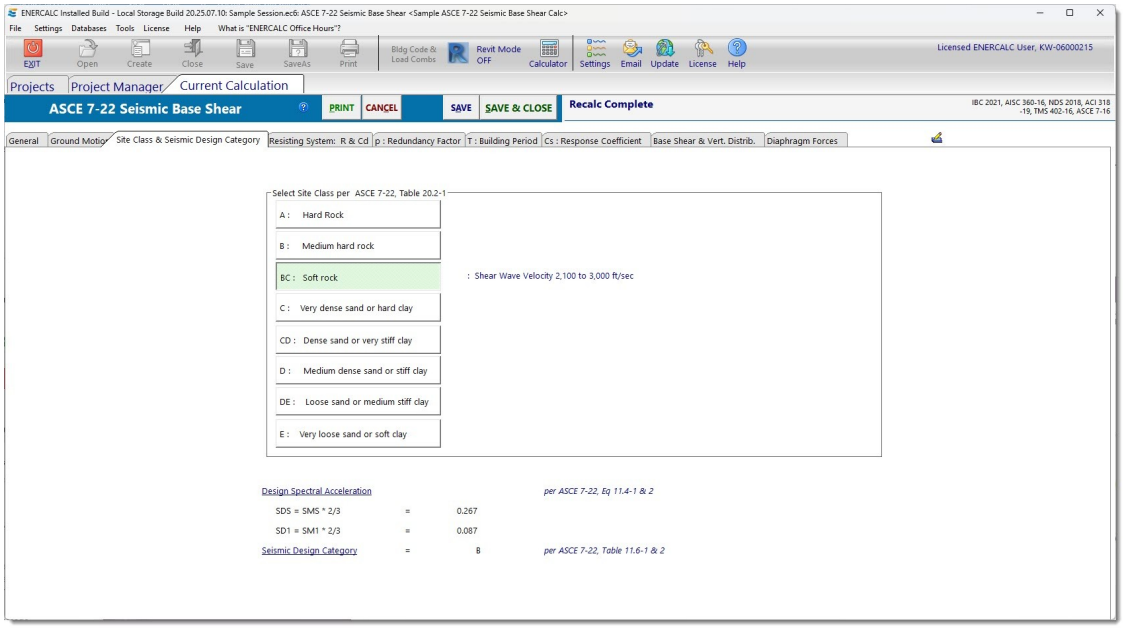
© 2026 ENERCALC, LLC

Ground Motion



Site Class, Site Coefficients and Seismic Design Category

Select the appropriate Site Class based on the geotechnical conditions.

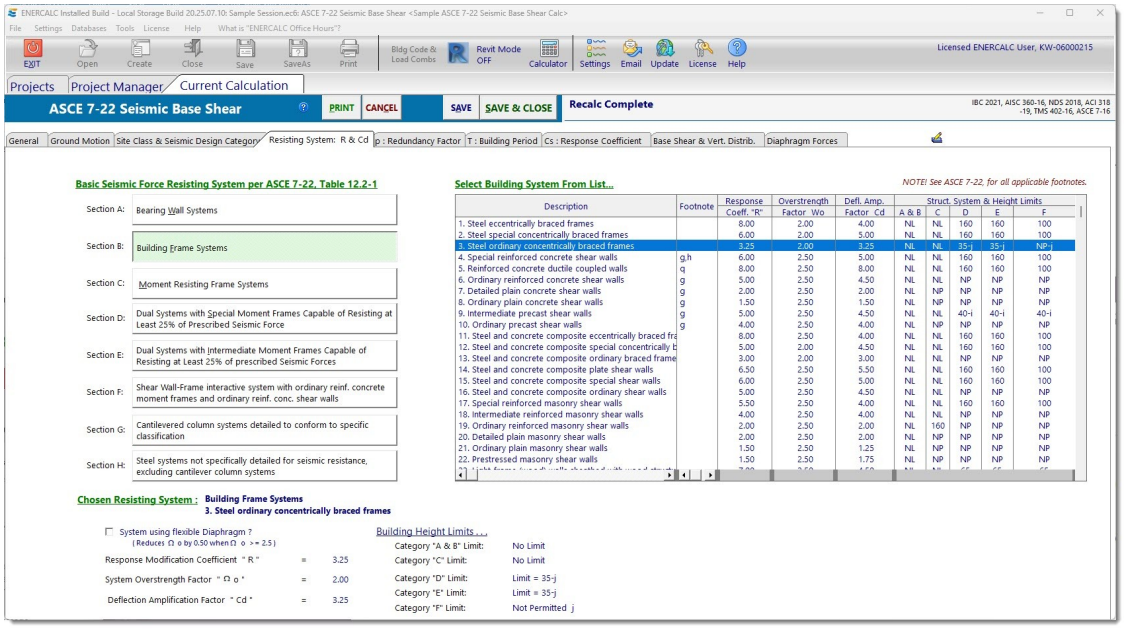


© 2026 ENERCALC, LLC

Selection of Seismic Force Resisting System

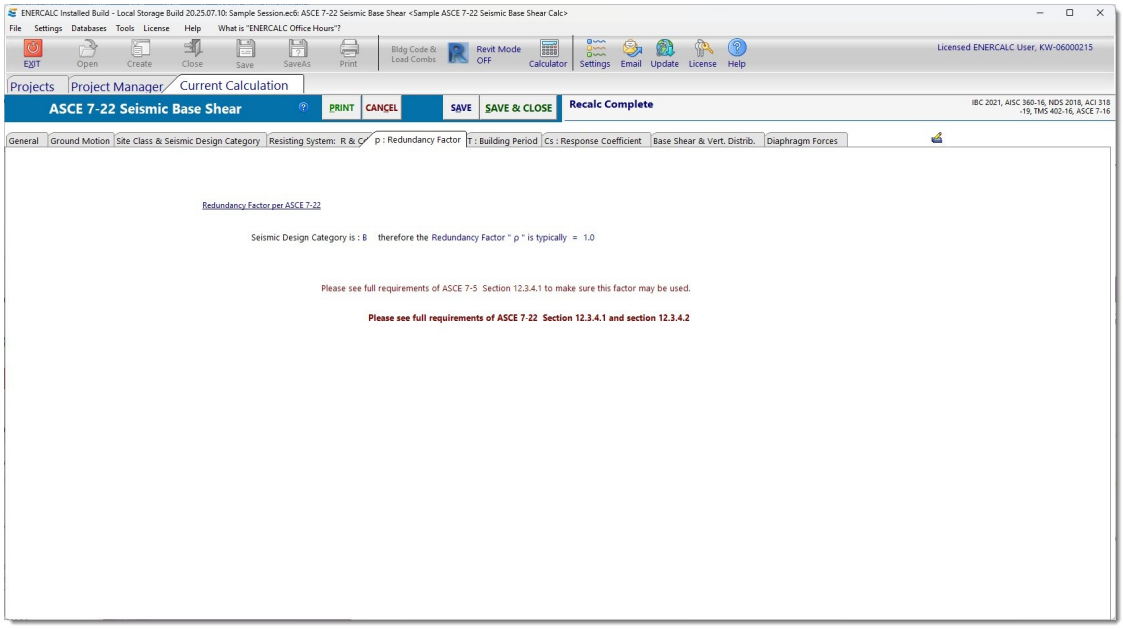
Click one of the large boxes at the top of the tab to load the table with specific selections for that general category of Seismic Force Resisting System.

Note the checkbox for systems with flexible diaphragms. This may not apply to some building systems...check ASCE for details on the particular system of interest.



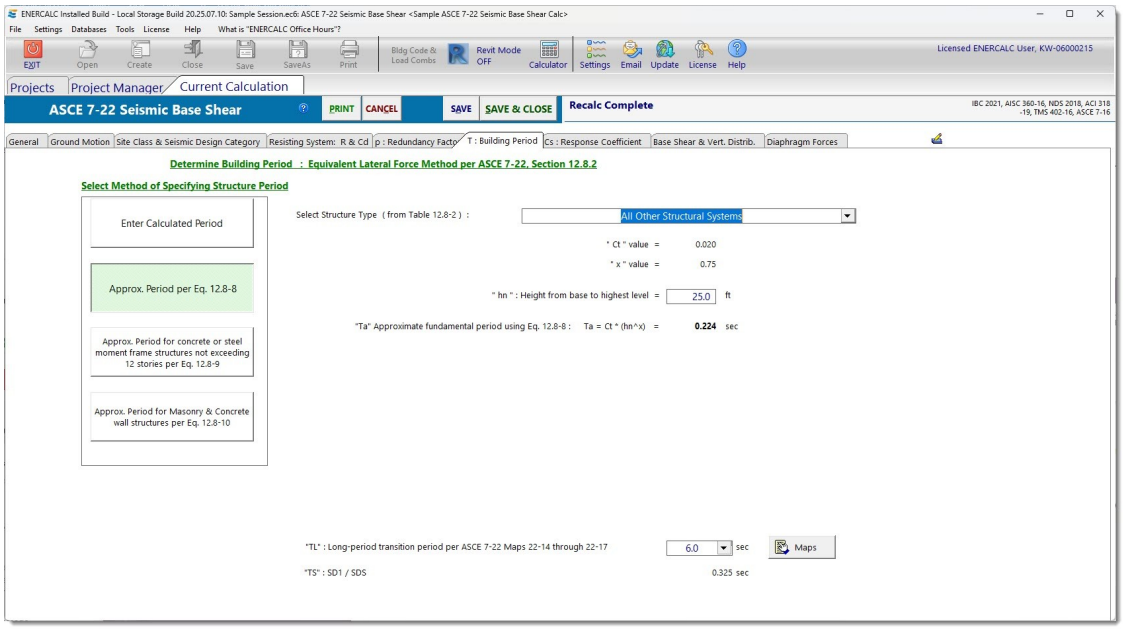
Redundancy Factor

Depending on the Seismic Design Category determined from the prior selections of Occupancy, Seismic Ground Motion and Building System, the Redundancy Factor per ASCE will be shown here.



Specification of Building Period

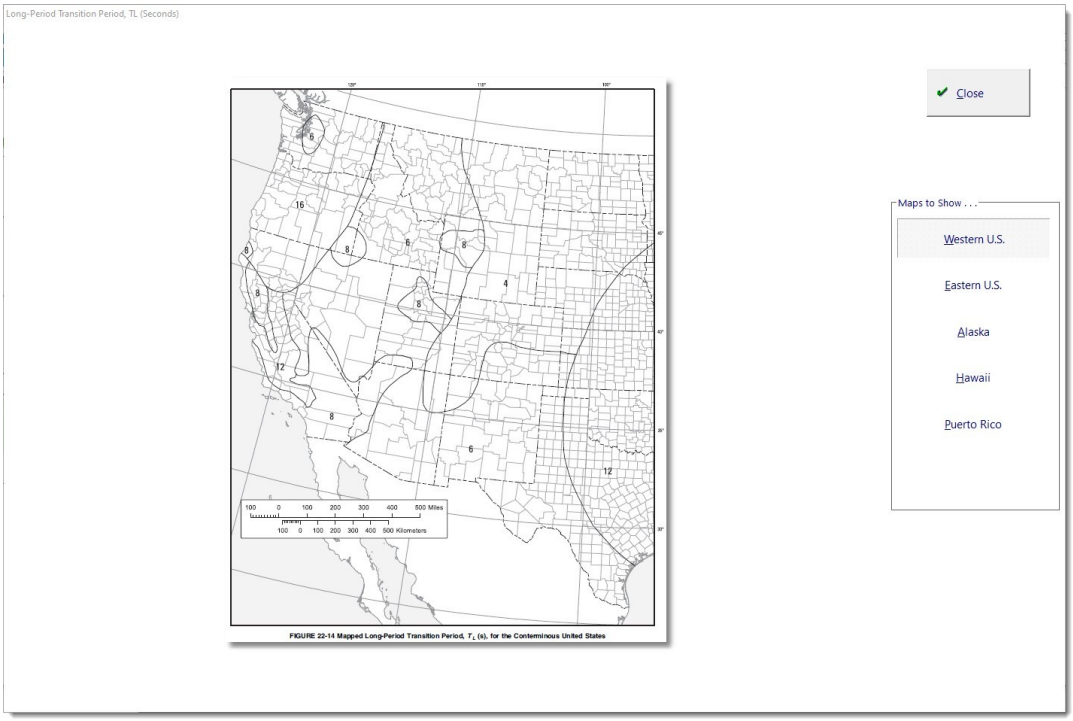
All four ASCE 7 options for building period determination are available here.



© 2026 ENERCALC, LLC

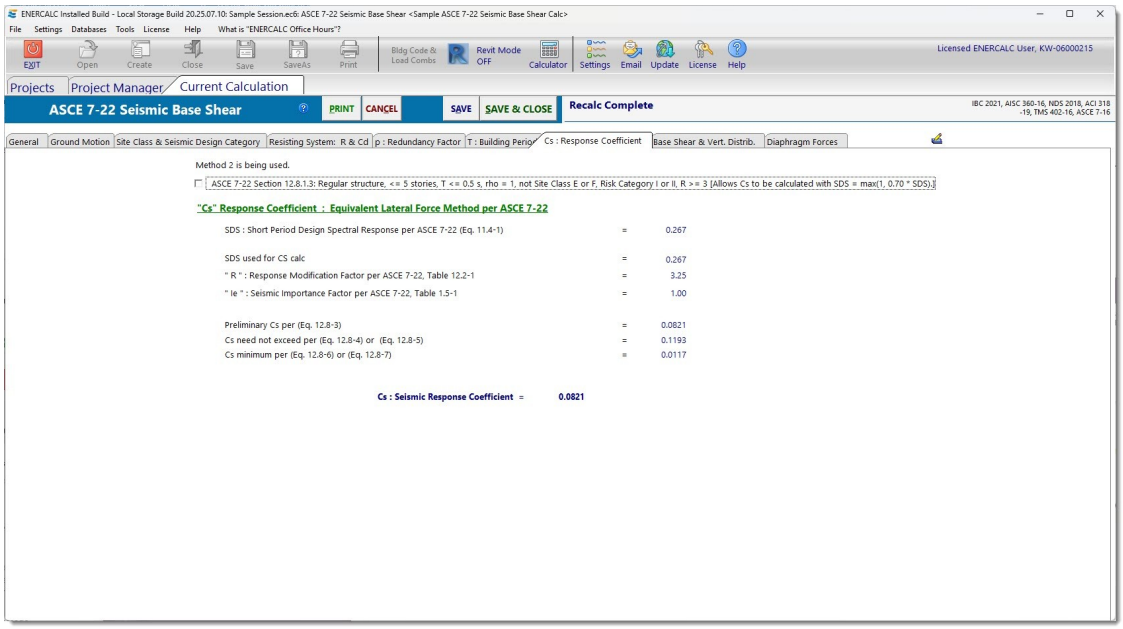
Long-Period Transition Period Reference Maps

Maps copyright ASCE.



Cs: Seismic Response Coefficient

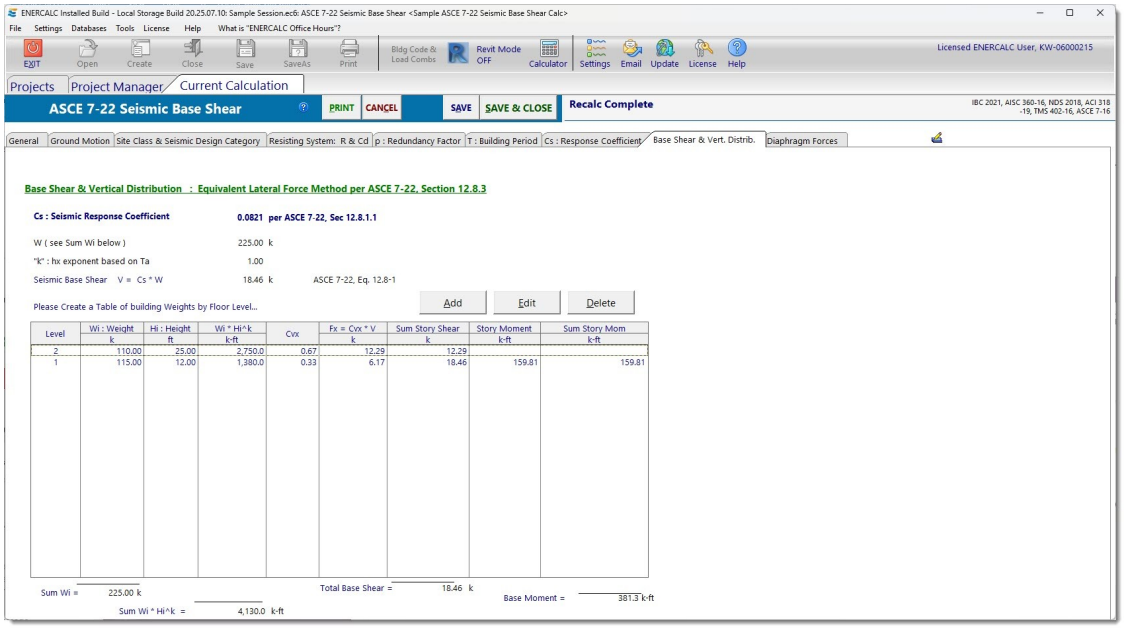
Please check ASCE 7 section references for details.



© 2026 ENERCALC, LLC

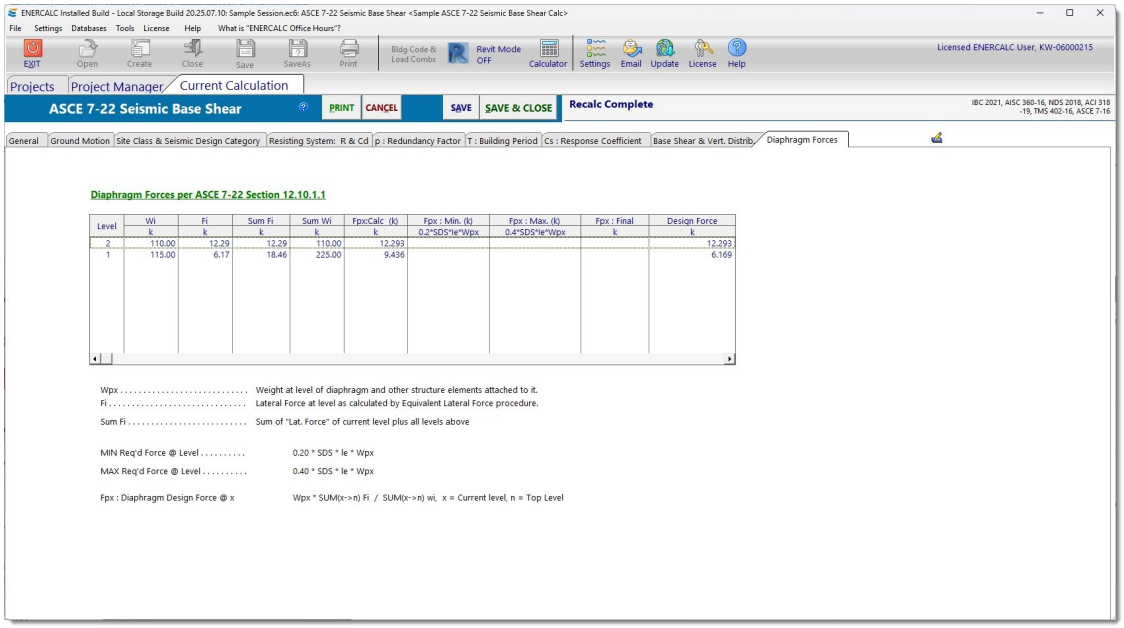
Vertical Distribution of Base Shear

This tab calculates the vertical distribution of seismic forces. Use the [**Add**] button for each new level. To edit information for a level, highlight the corresponding line and click [**Edit**].



Diaphragm Forces

Using the information entered above (on the prior tab) the force distribution results are shown here.

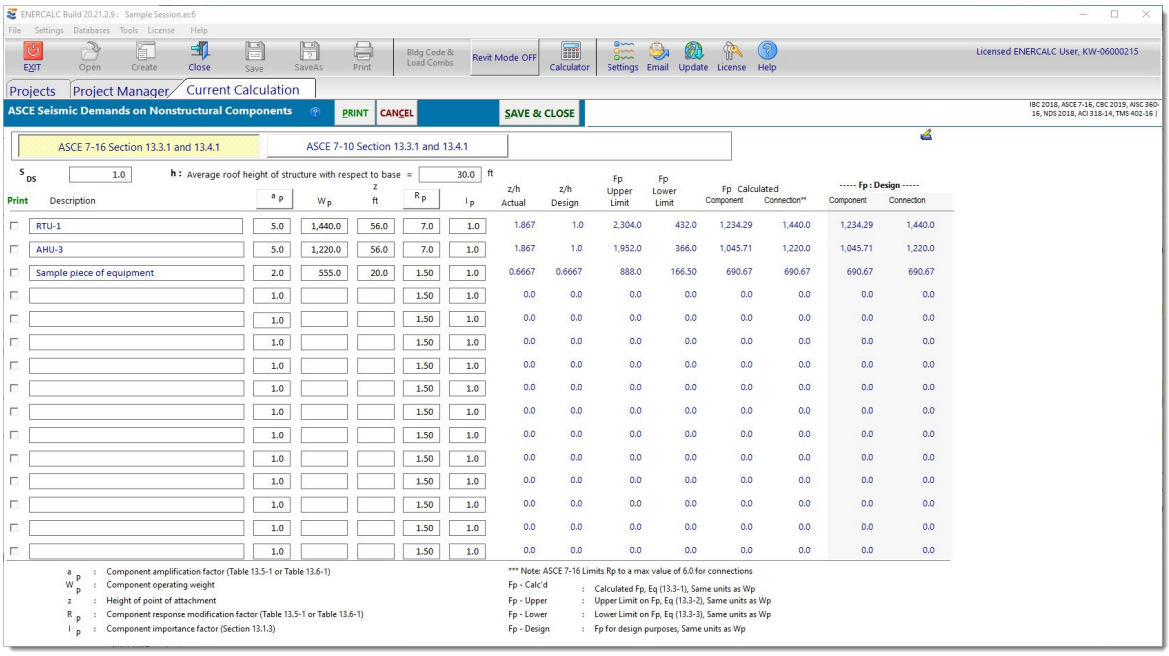


© 2026 ENERCALC, LLC

13.2.3 ASCE Seismic Demands on Nonstructural Components

This module is an implementation of the provisions for seismic demands on nonstructural components as per ASCE 7. The module currently supports calculations based on ASCE 7-10 and ASCE 7-16. Input parameters are taken directly from the nomenclature presented in ASCE 7.

This calculation produces both component and connection design forces using the defined parameters.

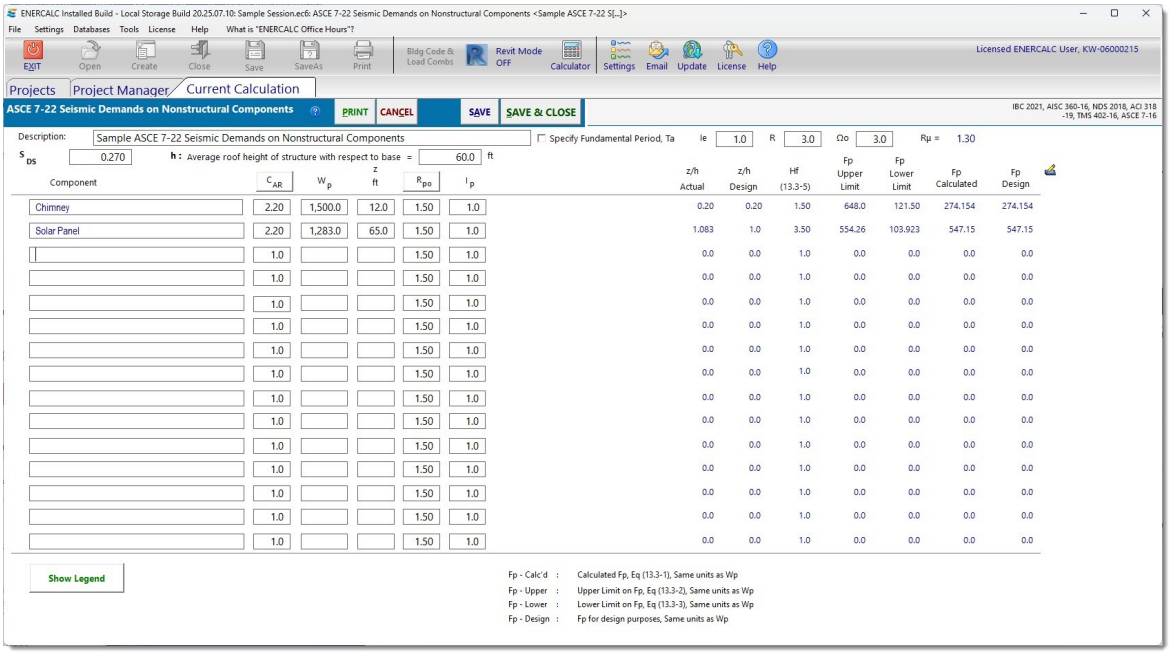


13.2.4 ASCE 7-22 Seismic Demands on Nonstructural Components

This module is an implementation of the provisions for seismic demands on nonstructural components as per ASCE 7-22. Input parameters are taken directly from the nomenclature presented in ASCE 7.

© 2026 ENERCALC, LLC

This calculation produces both component and connection design forces using the defined parameters.



13.2.5 ASCE Seismic Wall Anchorage

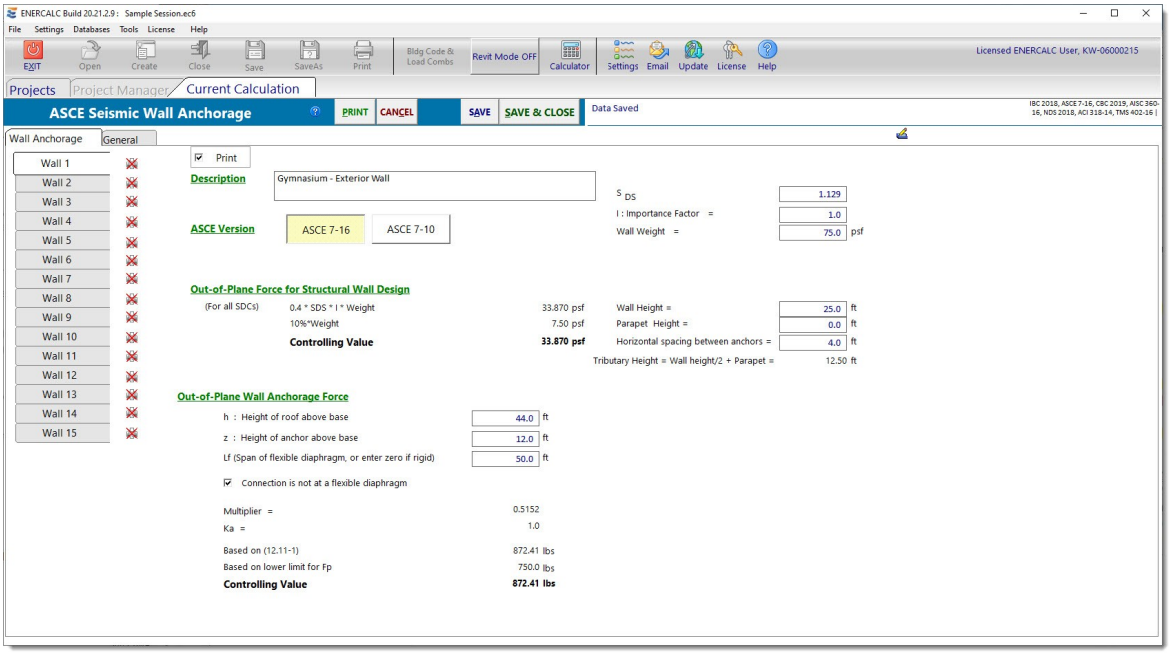
This module is an implementation of the provisions for design and anchorage of walls to resist seismic forces as per ASCE 7. The module currently supports calculations based on ASCE 7-10 and ASCE 7-16. Input parameters are taken directly from the nomenclature presented in ASCE 7.

The calculation produces:

- Out-of-Plane Force for Structural Wall Design
- Out-of-Plane Wall Anchorage Force

© 2026 ENERCALC, LLC

The module is configured to allow the definition of up to 20 different wall conditions. The user has the option to decide which (if any) of the calculations will be included in a Project Print.



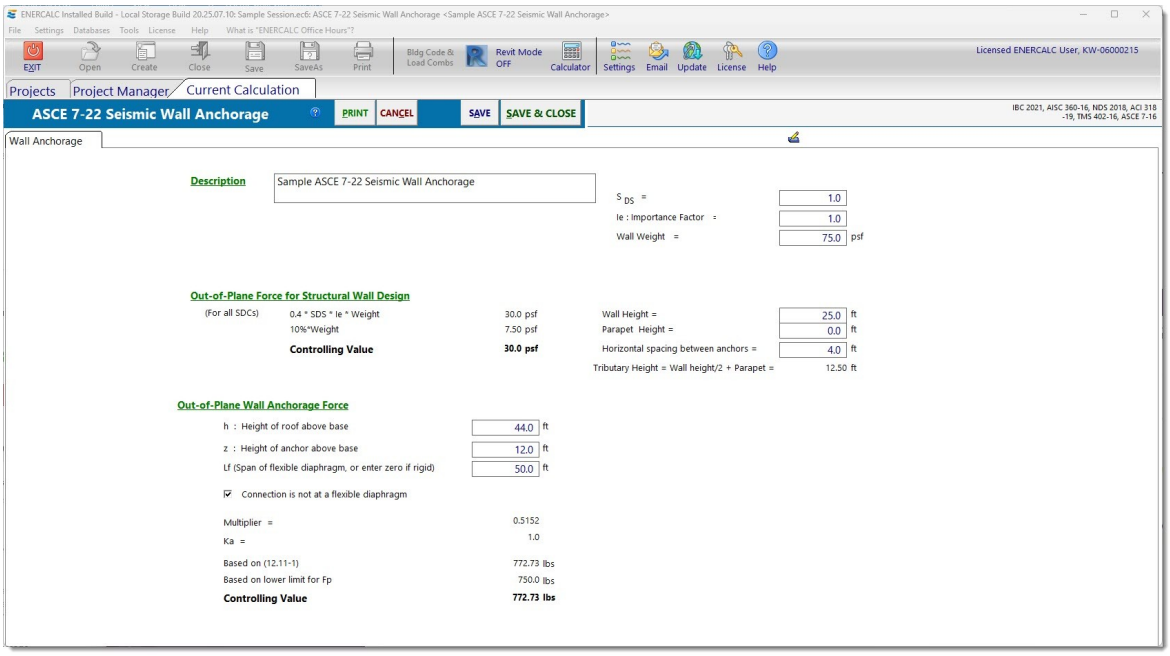
13.2.6 ASCE 7-22 Seismic Wall Anchorage

This module is an implementation of the provisions for design and anchorage of walls to resist seismic forces as per ASCE 7-22. Input parameters are taken directly from the nomenclature presented in ASCE 7.

The calculation produces:

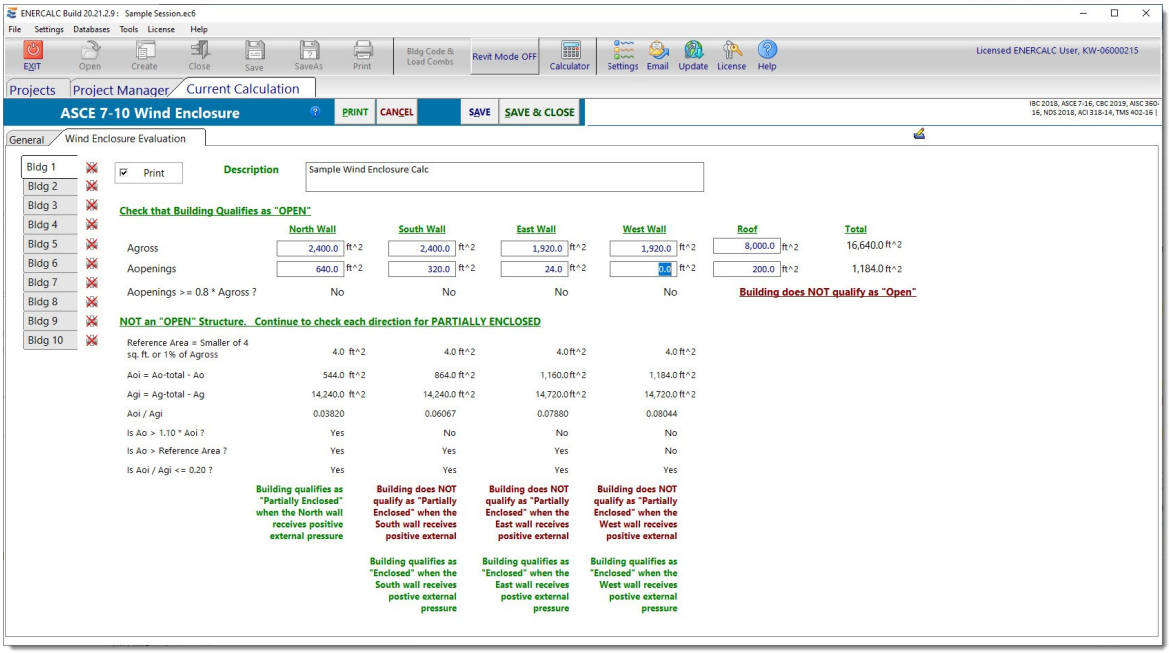
- Out-of-Plane Force for Structural Wall Design
- Out-of-Plane Wall Anchorage Force

© 2026 ENERCALC, LLC



13.2.7 ASCE Wind Enclosure

This module is an implementation of the provisions for determining the Enclosure Classification for the determination of Wind Loads as per ASCE 7.



The module currently supports calculations based on ASCE 7-22, ASCE 7-16 and ASCE 7-10. Input parameters are taken directly from the nomenclature presented in ASCE 7.

The ASCE 7-16 and ASCE 7-10 calculation sheet is set up to permit up to ten different analyses to be documented within a single Project File. The ASCE 7-22 calculation sheet

© 2026 ENERCALC, LLC

is set up to permit a single analysis per instance, but multiple instances can be added when a project includes multiple buildings, or in situations where multiple scenarios are being evaluated.

The data collection is presented in such a way as to guide the user through the module in a logical progression.

Values of gross area and area of openings are provided for walls on four sides of the building (arbitrarily designated "North", "South", "East", and "West"). Once these values are provided, the module can determine if the structure qualifies as "Open" or not. If the structure is determined to be "Open", then the module will indicate that result and stop there. If the structure does *not* qualify as an "Open" structure, then input fields for Roof Gross Area and Roof Opening Area will be displayed, and a set of four tabs will appear in the bottom half of the screen. Once the Roof areas are populated, the four tabs thoroughly present the calculations to determine whether the structure qualifies as "Enclosed" or "Partially Enclosed" when each of the four respective walls receives positive external pressure. The ASCE 7-22 version of the module also incorporates the newly added "Partially Open" enclosure classification.

The results of this determination are useful for project documentation requirements, as well as in guiding the user to read the appropriate results values from other modules within ENERCALC SEL.

Note: The provisions of the ASCE Wind Enclosure module are already incorporated into the ASCE 7 Wind Loads modules. So it is not necessary to run the standalone version of this module if the ASCE 7 Wind Loads modules will be used.

13.2.8 ASCE 7-10/16 Wind Forces, Chapter 27, Part 1

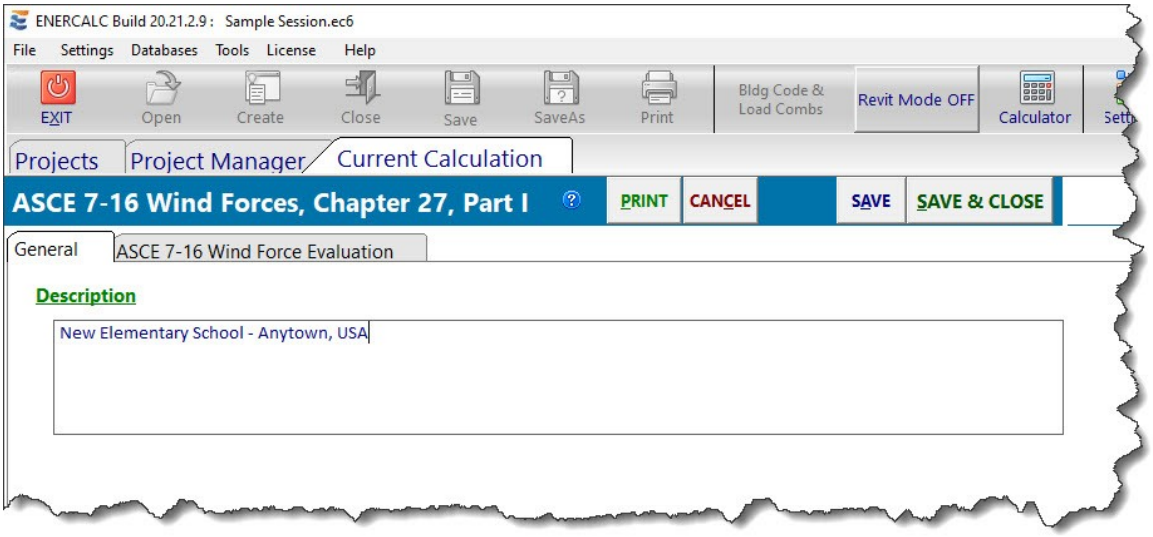
This module is a presentation of the Wind Forces provisions of Chapter 27, Part 1 of ASCE 7-10 and ASCE 7-16.

Limited documentation is provided here, because all of the references to ASCE 7 are given on the module screens.

© 2026 ENERCALC, LLC

General

The General tab provides an input field for a general description of the project and/or the wind calculations that are being performed.



ASCE 7-10 / 7-16 Wind Force Evaluation

This tab provides access to a series of Calculation tabs that can be used to store up to ten separate wind calculations. These might be useful for studying different project sites, different architectural concepts, or even evaluating separate buildings in a project.

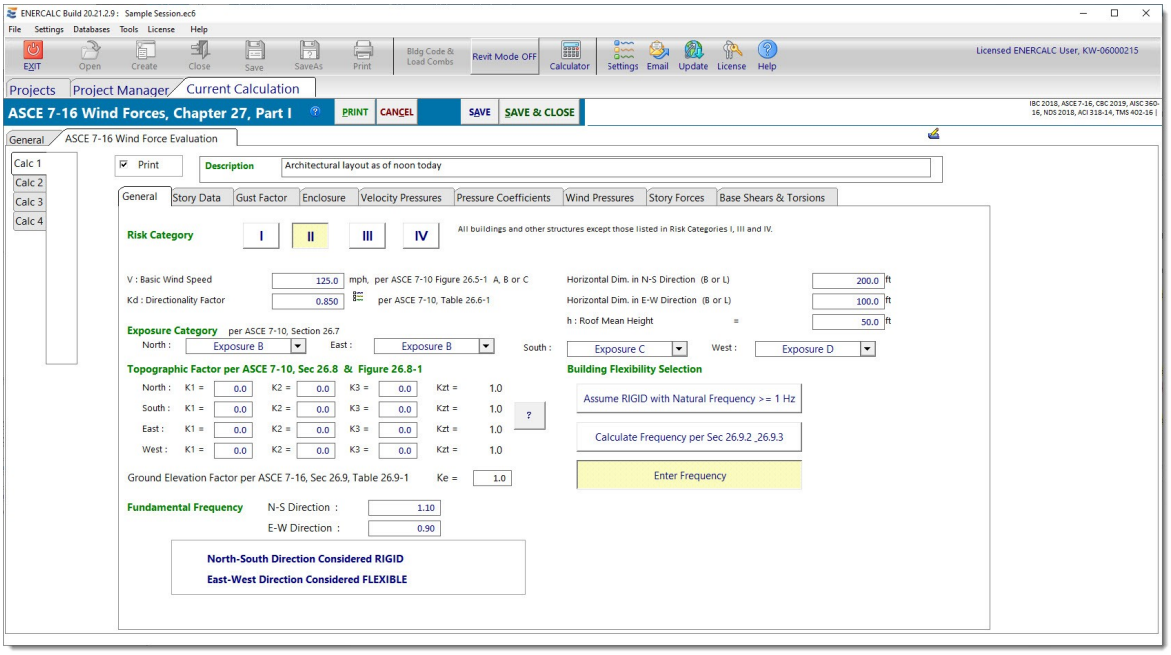
The Description field can be used to distinguish from among the different wind load calculations that might be defined in a single Project File.

The Print checkbox specifies whether or not the particular wind load calculation will or will not be included when a Print command is clicked or when a Project Print is performed.

© 2026 ENERCALC, LLC

General (sub-tab of the ASCE 7-10 / 7-16 Wind Force Evaluation tab)

This tab collects basic data such as Risk Category, Basic Wind Speed, Directionality Factor, Building Dimensions, Exposure Category, Topographic Factor, and information to determine how the building frequency will be determined.



The Risk Category is only for reference in documenting the design. It no longer influences the importance factor, but it does dictate which Wind Speed map to use to determine the Basic Wind Speed.

The Exposure Category is dependent upon the upwind characteristics. Therefore, as different building elevations become the windward face of the structure, it is possible that the Exposure Category will change. For this reason, the program allows the Exposure Category to be defined separately for each building face in turn becoming the windward face.

The Building Flexibility Selection offers three different options for defining the frequency of the building.

- The first option declares that the building will be assumed to be rigid (having a frequency greater than or equal to 1 Hz).
- The second option applies the prescriptive methods in Sections 26.9.2 and 26.9.3 of ASCE 7-10 to approximate the frequency of the building based on construction type/layout and mean roof height.
- The third option allows the user to explicitly specify the frequency in the North-South direction and in the East-West direction.

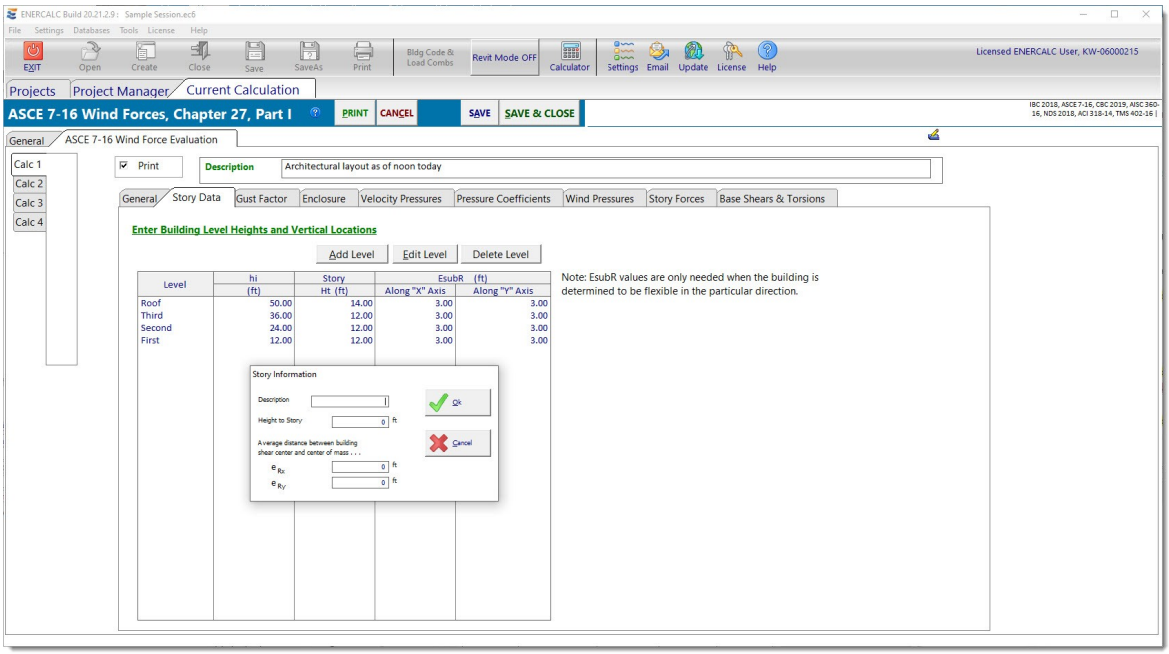
Note that if the second option is selected, a new tab named "Frequency" will be introduced between the "Story Data" tab and the "Gust Factor" tab.

© 2026 ENERCALC, LLC

This tab contains the only difference between ASCE 7-10 and ASCE 7-16, which is the introduction of Ke, the Ground Elevation Factor in ASCE 7-16.

Story Data

This tab collects the data required to define the vertical locations of the stories with respect to finished grade.



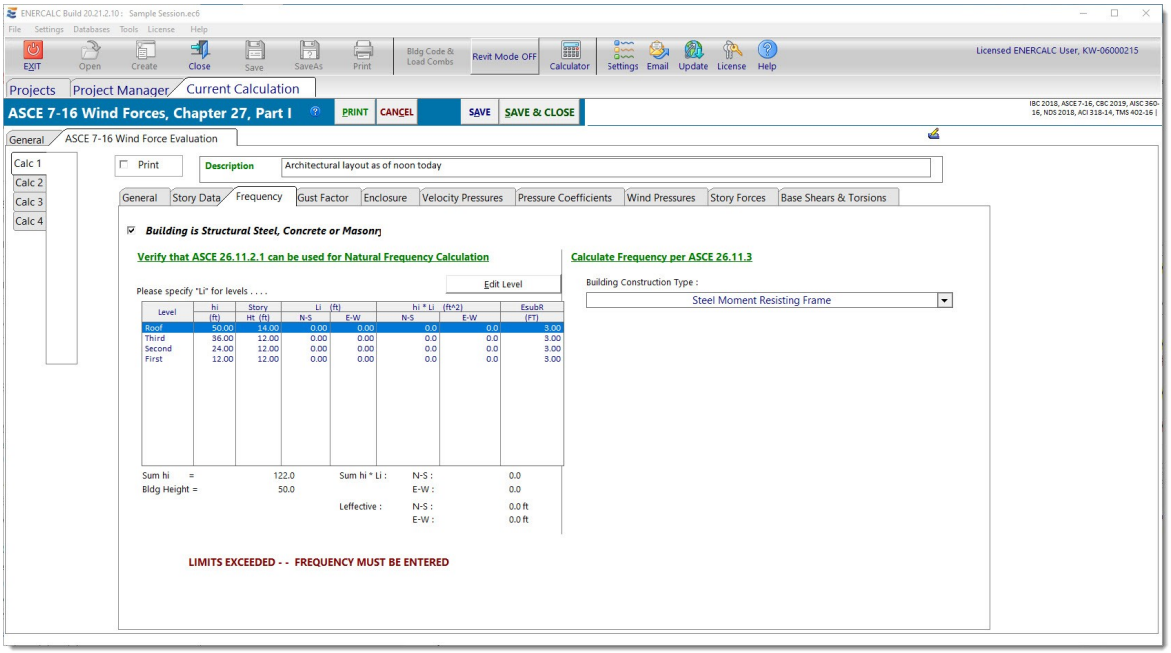
The development of the story data table is controlled by the three buttons: Add Level, Edit Level, and Delete Level, which perform their respective operations on the table of story data.

Clicking Add or Edit opens the Story Information pop-up dialog as shown above. Two items are worthy of note on this dialog. First, the Height to Story is always referenced from grade, so it is looking to collect the height from grade to the story of interest, not the story to story height. Second, the values of e_{Rx} and e_{Ry} are only used if a building is determined (or defined) to be "Flexible" (< 1 Hz) in a particular direction. So if a building turns out to be "Rigid" in one or both directions, then the values of e_{Rx} and e_{Ry} are not required in those directions, and the values can be left at zero or a value can be entered, but the program will ignore it.

© 2026 ENERCALC, LLC

Frequency (Only displayed if the option is selected on the General tab to "Calculate Frequency")

This tab displays the story data that was previously collected, and it requests the Building Construction Type and the values of Li necessary to complete the check to determine if the approximate methods of frequency determination are applicable.

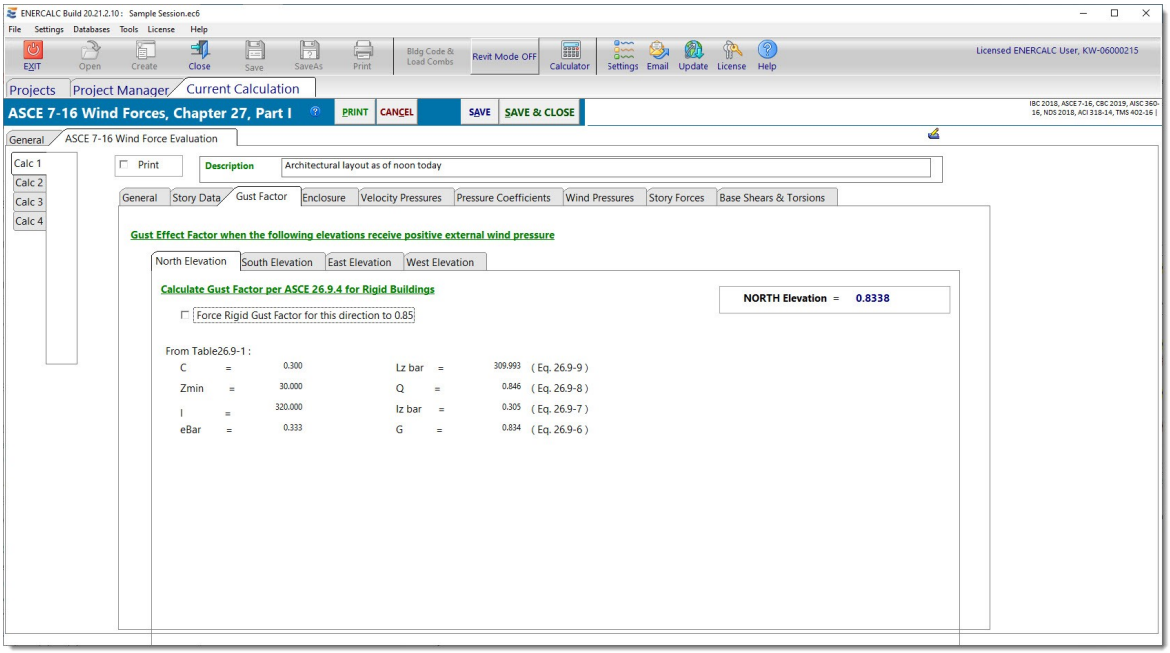


If the approximate methods of frequency determination are found to be applicable, this tab reports the approximate frequency. Otherwise, a message will be displayed to indicate that the approximate methods are not applicable, and that the building frequency must be determined another way.

© 2026 ENERCALC, LLC

Gust Factor

This tab displays the results of the Gust Factor determination for each of the four elevations, in turn, being the face of the building that receives positive external pressure.



The North, South, East, and West tabs present the calculation and the resulting Gust Factor that will be used in subsequent calculations when the respective building elevation receives positive external pressure.

When the building is categorized as "Rigid" in a particular direction, the program offers the checkbox option to force the value of the Gust Factor to 0.85 when the referenced elevation receives positive external pressure.

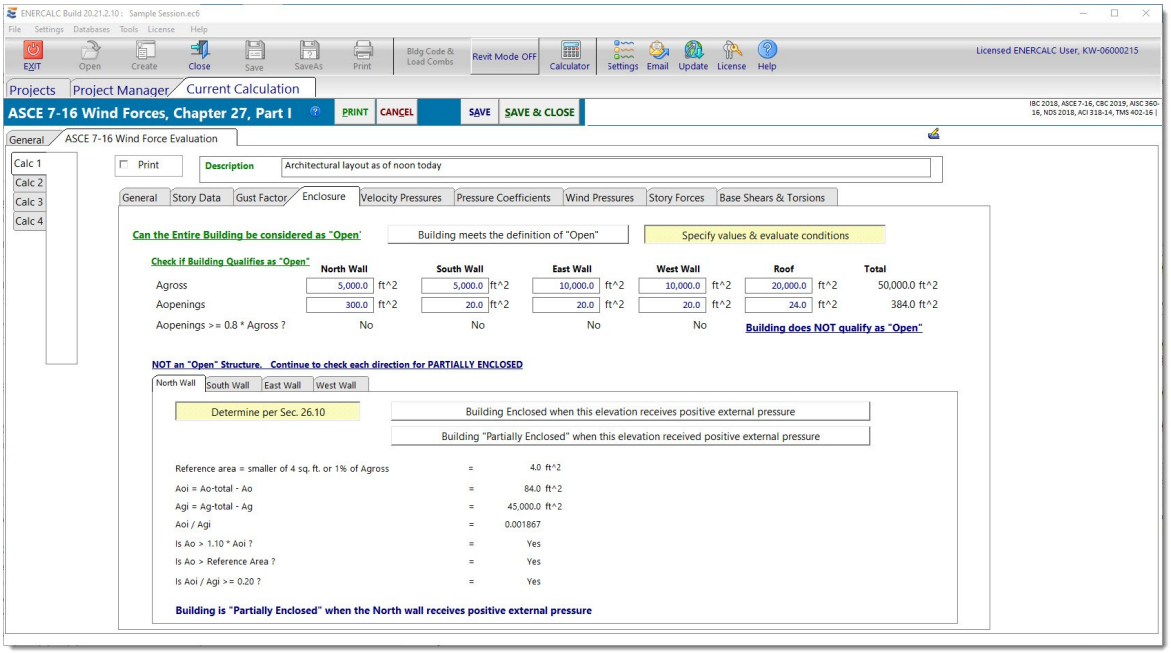
When the building is categorized as "Flexible" in a particular direction, the program requires one more value, which is the damping ratio. This value is collected in an input box on the respective direction tab when appropriate, and the value is incorporated into the Gust Factor to be used when the referenced elevation receives positive external pressure.

It is worth noting that many of the parameters used in the calculation of the Gust Factor are dependent upon the Exposure category. Since the Exposure category can vary for each of the four cardinal directions, it is possible that the Gust Factor for use when the ***North*** elevation of the building receives positive external pressure may actually be different than the Gust Factor for use when the ***South*** elevation of the building receives positive external pressure.

© 2026 ENERCALC, LLC

Enclosure

This tab displays the results of the Enclosure determination for each of the four elevations, in turn, being the face of the building that receives positive external pressure.



The upper half of this tab is dedicated to evaluating the building to determine whether or not it qualifies as an "Open" structure. The module collects the gross areas of each of the four walls along with the areas of opening in each of the four walls. Based on the data provided by the user, the module performs the calculations and checks the criteria to see if the building qualifies as an "Open" structure. If it does, then the module reports that result. If the building does NOT qualify as an "Open" structure" then additional input fields are displayed to collect the gross area of the Roof, and the area of openings in the roof, and the workflow continues to determine whether the building qualifies as "Enclosed" or "Partially Enclosed". This evaluation takes place four times, considering each of the four walls to be the windward wall, in turn. The intermediate calculations are performed and the results are reported on each of the four wall tabs.

It is worth noting that a building could potentially be considered "Partially Enclosed" when some of its elevations receive positive external pressure, and "Enclosed" when its other elevations receive positive external pressure.

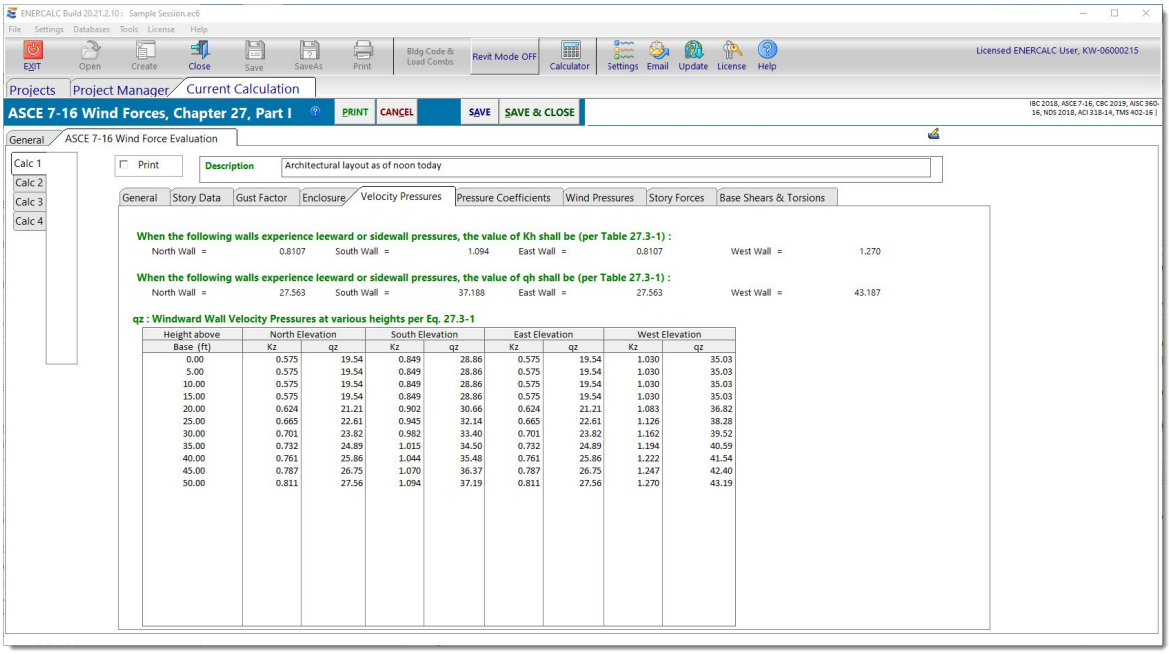
It is also worth noting that convenience buttons have been provided on each of the four wall tabs to allow the user to simply declare the building to be "Enclosed" or "Partially Enclosed" when the selected elevation receives positive external pressure. These have been implemented for situations where the evaluation has already been performed and/or the user is already confident in making a decision by judgment.

The Enclosure classification is used downstream to select appropriate values of GCpi to use when each of the four elevations becomes the windward wall.

© 2026 ENERCALC, LLC

Velocity Pressures

This tab displays the results of the Velocity Pressure determination for the various walls when each wall is under leeward, sidewall, and windward wall conditions.



Height above Base (ft)	North Elevation Kz	North Elevation qz	South Elevation Kz	South Elevation qz	East Elevation Kz	East Elevation qz	West Elevation Kz	West Elevation qz
0.00	0.575	19.54	0.849	28.86	0.575	19.54	1.030	35.03
5.00	0.575	19.54	0.849	28.86	0.575	19.54	1.030	35.03
10.00	0.575	19.54	0.849	28.86	0.575	19.54	1.030	35.03
15.00	0.575	19.54	0.849	28.86	0.575	19.54	1.030	35.03
20.00	0.624	21.21	0.902	30.66	0.624	21.21	1.083	36.82
25.00	0.665	22.61	0.945	32.14	0.665	22.61	1.126	38.28
30.00	0.701	23.82	0.982	33.40	0.701	23.82	1.162	39.52
35.00	0.732	24.89	1.015	34.50	0.732	24.89	1.194	40.59
40.00	0.761	25.86	1.044	35.48	0.761	25.86	1.222	41.54
45.00	0.787	26.75	1.070	36.37	0.787	26.75	1.247	42.40
50.00	0.811	27.56	1.094	37.19	0.811	27.56	1.270	43.19

The first row of data reports the values of Kh that are applicable when each of the walls experiences leeward or sidewall pressures.

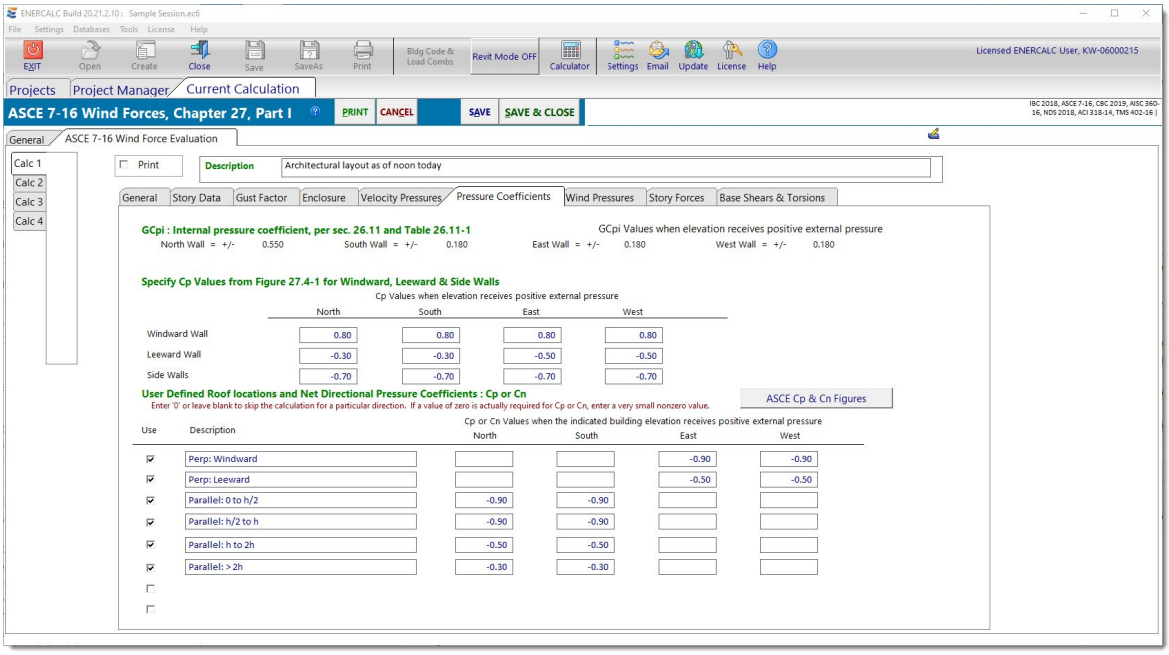
The second row of data reports the resulting values of qh that are applicable when each of the walls experiences leeward or sidewall pressures.

Next, a table is presented that reports the values of Kz and the resulting values of qz that are applicable, as a function of height, when each of the walls experiences windward wall pressures.

© 2026 ENERCALC, LLC

Pressure Coefficients

This tab reports the values of GCpi that are applicable when each of the respective elevations receives positive external pressure. The remainder of this tab is dedicated to collecting the values of Cp or Cn as appropriate for the various surfaces of the building.



Wall values are collected first, and the input fields collect values of Cp that will be used when each of the four walls is a windward wall, a sidewall, or a leeward wall.

At the bottom of the tab is a customizable table that is set up to receive many lines of Cp or Cn values pertinent to the roof. A convenience button is provided to display the figures that define the Cp or Cn values for various conditions. A glance at the tables reveals that the factors for roofs are frequently dependent upon the wind direction with respect to the ridge, and also dependent upon whether pressures are desired for the windward or leeward surfaces of the roof. So some planning will be required to set this table up to yield the desired results. To make the downstream results most applicable and easy to read, the program has been set up so that a roof pressure value will only be calculated for situations where nonzero values of Cp or Cn have been specified. To put this another way, referring to the table above, there are no values of Cp defined for the North or South elevations for the "Perpendicular: Windward" or "Perpendicular: Leeward" conditions. This is because the ridge is assumed to run in the North-South direction in the hypothetical building being considered. As such, it would not make sense to ask the program to report windward or leeward roof pressures when the wind acts in the north or south directions. So to avoid overpopulating the output with meaningless results, the blank fields will be interpreted by the program as an indication that the corresponding calculation is not required. We will see the benefit of this when we move to the Wind Pressures tab and see how concisely these results are reported.

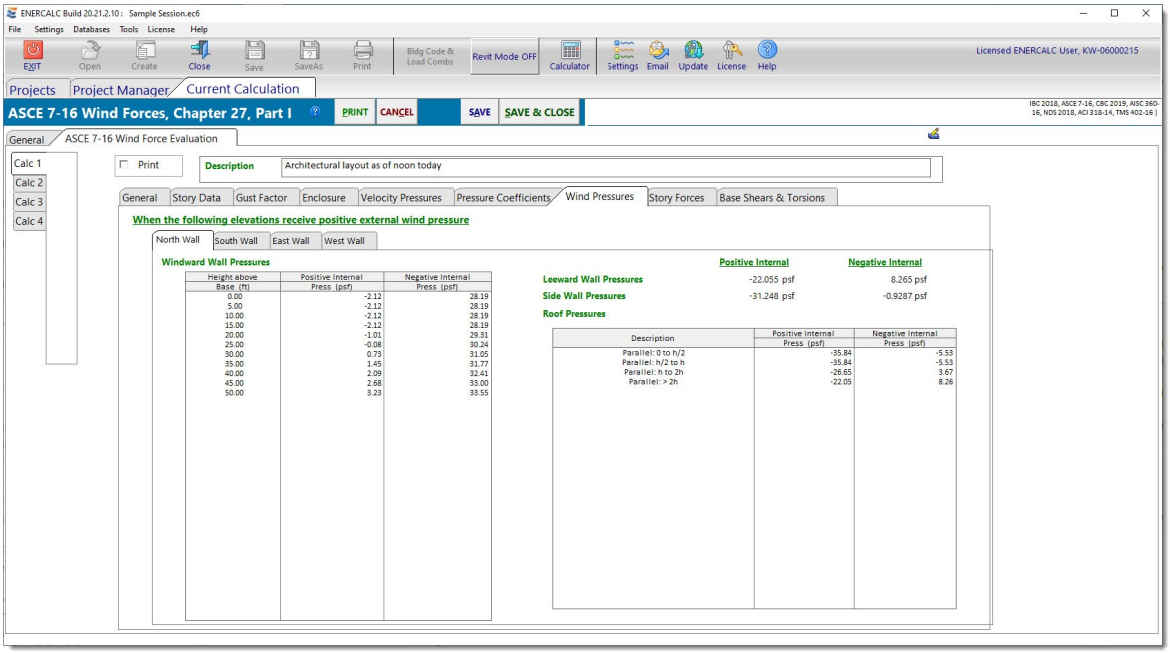
On a related note, it is worth mentioning that the Cp and Cn tables occasionally report values of zero for certain conditions. These are typically provided for interpolation

© 2026 ENERCALC, LLC

purposes. But if a situation is ever encountered where a value of zero for Cp or Cn is actually required for design purposes, the user is advised to enter a small nonzero value.

Wind Pressures

This tab reports the values of wind pressures that occur on the various surfaces of the building when the named elevation receives positive external pressure.



Looking at the screen capture above, we see that the North Wall tab is currently selected. Let's work through this tab thoroughly as an example. As indicated in the on-screen note, we interpret all of the results on this tab as being the pressures that occur on the named surface of the building **when the North Wall receives positive external wind pressure**. So when we are focused on the North Wall tab:

- The "Windward Wall Pressures" are those that would apply to the **North** Wall when the North Wall receives positive external pressure.
- The "Leeward Wall Pressures" are those that would apply to the **South** Wall when the North Wall receives positive external pressure.
- The "Sidewall Pressures" are those that would apply to the **East** and **West** Walls when the North Wall receives positive external pressure.
- The "Roof Pressures" are those that would apply to the identified areas measured from the **North** edge of the roof when the North Wall receives positive external pressure.

© 2026 ENERCALC, LLC

Just to drive home the proper interpretation of the values reported on the Wind Pressures tab, let's work through the East tab, so we can interpret some windward and leeward roof pressures. Remember that this hypothetical building is assumed to have a North-South oriented ridge. So when we are focused on the East Wall tab:

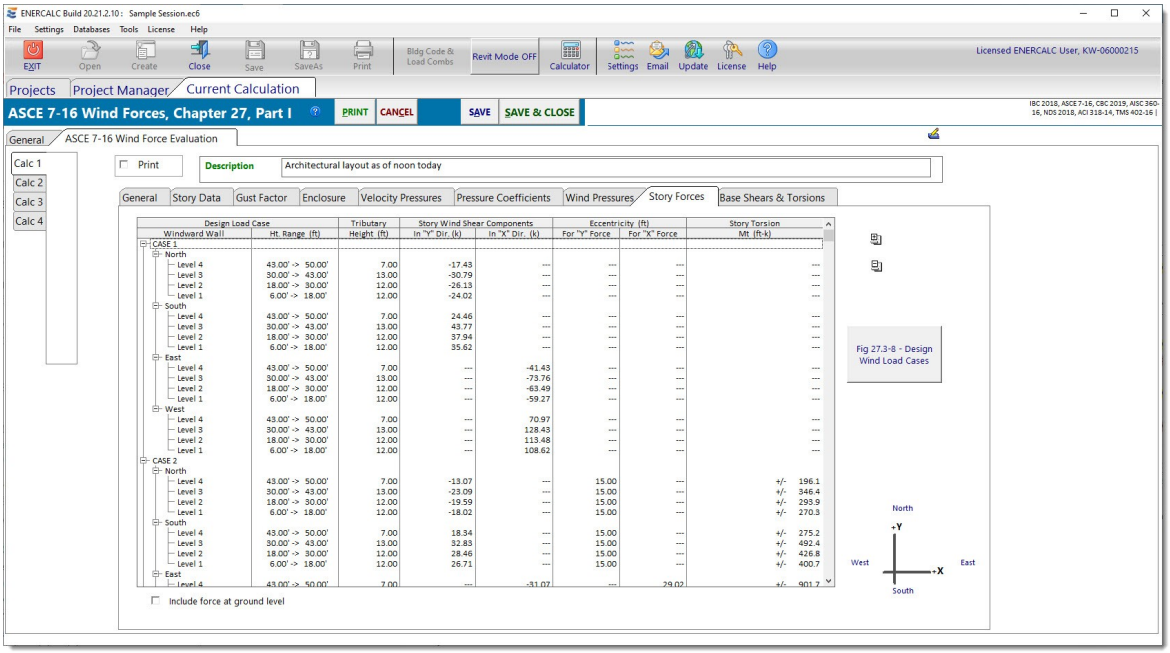
- The "Windward Wall Pressures" are those that would apply to the **East** Wall when the East Wall receives positive external pressure.
- The "Leeward Wall Pressures" are those that would apply to the **West** Wall when the East Wall receives positive external pressure.
- The "Sidewall Pressures" are those that would apply to the **North** and **South** Walls when the East Wall receives positive external pressure.
- The "Perp: Windward Roof Pressures" are those that would apply to the **windward (East)** portion of the roof when the East Wall receives positive external pressure.
- The "Perp: Leeward Roof Pressures" are those that would apply to the **leeward (West)** portion of the roof when the East Wall receives positive external pressure.

Note that all surfaces report pressures based on both the positive and the negative internal pressure conditions, and the algebraic sign convention follows that of ASCE 7, which is to say that positive values are interpreted to act toward the named surface and negative values act away from the named surface. Work this logic through an example such as a windward wall and see that it all makes sense. The negative internal pressure condition produces higher total pressures toward the windward wall, than the positive internal pressure condition does, because the negative internal pressure works in the same direction as the external pressure on a windward wall. Similar logic can be applied to all other surfaces to demonstrate that the mathematics are proper.

© 2026 ENERCALC, LLC

Story Forces

This tab reports the values of wind forces tributary to each story in the building.



Using the story heights determined on the Story Data tab, the Story Forces tab determines the tributary heights for each floor by assuming a simply-supported wall construction that spans between adjacent floor/roof levels. Wind pressures are applied to the tributary heights and multiplied by the perpendicular dimension of the building to arrive at forces for each story.

The option is provided to either display or hide the forces tributary to the lower half of the lowest level. In some construction types, this component of load is delivered to a slab on grade and is not a design consideration for the Main Wind Force Resisting System.

The program reports results for each of the four "Cases" as presented by Figure 27.4-8 of ASCE 7-10 (Figure 27.3-8 of ASCE 7-16). Cases 2 and 4 also incorporate a design torsional moment as defined in the figure. In all situations, the program reports the force magnitude. In situations where a torsional moment applies, the program reports the moment arm that is being considered for each component of force, as well as the resulting net moment.

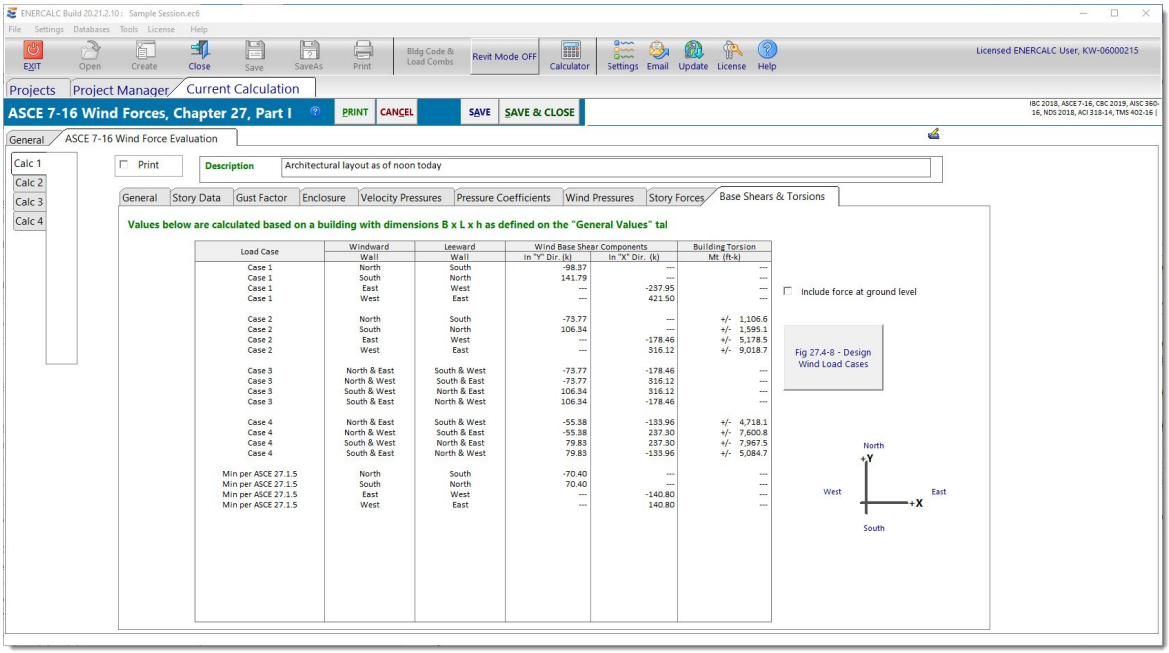
The final item in the results on this tab is based on the minimum required wind loads per Section 27.4.7 of ASCE 7-10 (Section 27.1.5 of ASCE 7-16). That section requires that the wind load to be used in the design of the Main Wind Force Resisting System for an enclosed or partially enclosed building shall not be less than 16 psf multiplied by the wall area of the building (and 8 psf multiplied by the roof area of the building projected onto a vertical plane normal to the assumed wind direction). So in this last item in the results list, the program reports story forces assuming 16 psf applied to each wall of the building. As of build 6.12.4.24, the program does not collect enough information to consider the 8 psf

© 2026 ENERCALC, LLC

on the projection of the roof area, so this additional load may need to be considered with supplemental hand calculations on buildings with other than flat or low-slope roofs.

Base Shears & Torsions

This tab reports the summation of the wind story forces and torsions for all levels in the building, for all four "Cases" and for the minimum required wind loads per Section 27.4.7.



Although "Base Shears" are not technically a part of the ASCE 7 design procedure for wind loads, the summation of all story forces is often of interest to designers for a variety of reasons. Some designers like to see how the wind "base shear" compares to the seismic base shear. For some designers, the summation of the wind forces is useful in the checking process to get a confidence level that the calculated pressures are reasonable. Whatever the reason, the values are reported on this tab if they are of interest.

As with the Story Forces tab, the Base Shears & Torsions tab provides the option to either include or exclude the component of shear and torsion that is tributary to the bottom half of the lowest level.

13.2.9 ASCE 7-10/16 Wind Forces Chapter 28 and 30

This module is a presentation of the Wind Forces provisions of Chapter 28, Part 2 and Chapter 30, Part 2 of ASCE 7-10 and ASCE 7-16.

Limited documentation is provided here, because all of the references to ASCE 7 are given on the module screens.

© 2026 ENERCALC, LLC

General

The General tab provides an input field for a general description of the project and/or the wind calculations that are being performed.

Applicability Checklists

The Applicability Checklists tab provides a quick listing of the requirements in order to be able to apply Chapter 28, Part 2 (MWFRS) and Chapter 30, Part 2 (Components & Cladding).

Analysis Values

The Analysis Values tab collects the input (on the top of the screen) and displays the results (on the bottom of the screen).

13.2.10 ASCE 7-22 Wind Forces, Chapter 27 (Directional Procedure)

This module is a presentation of the Wind Forces provisions of ASCE 7-22 Chapter 27 (Directional Procedure).

Limited documentation is provided here, because all of the references to ASCE 7 are given on the module screens.

13.2.11 ASCE 7-22 Wind Forces, Chapter 28 (Envelope Procedure)

This module is a presentation of the Wind Forces provisions of ASCE 7-22 Chapter 28 (Envelope Procedure).

Limited documentation is provided here, because all of the references to ASCE 7 are given on the module screens.

13.2.12 ASCE 7-22 Wind Forces, Chapter 30, Part 1 (Components & Cladding)

This module is a presentation of the Wind Forces provisions of ASCE 7-22 Chapter 30, Part 1 (Components & Cladding).

Limited documentation is provided here, because all of the references to ASCE 7 are given on the module screens.

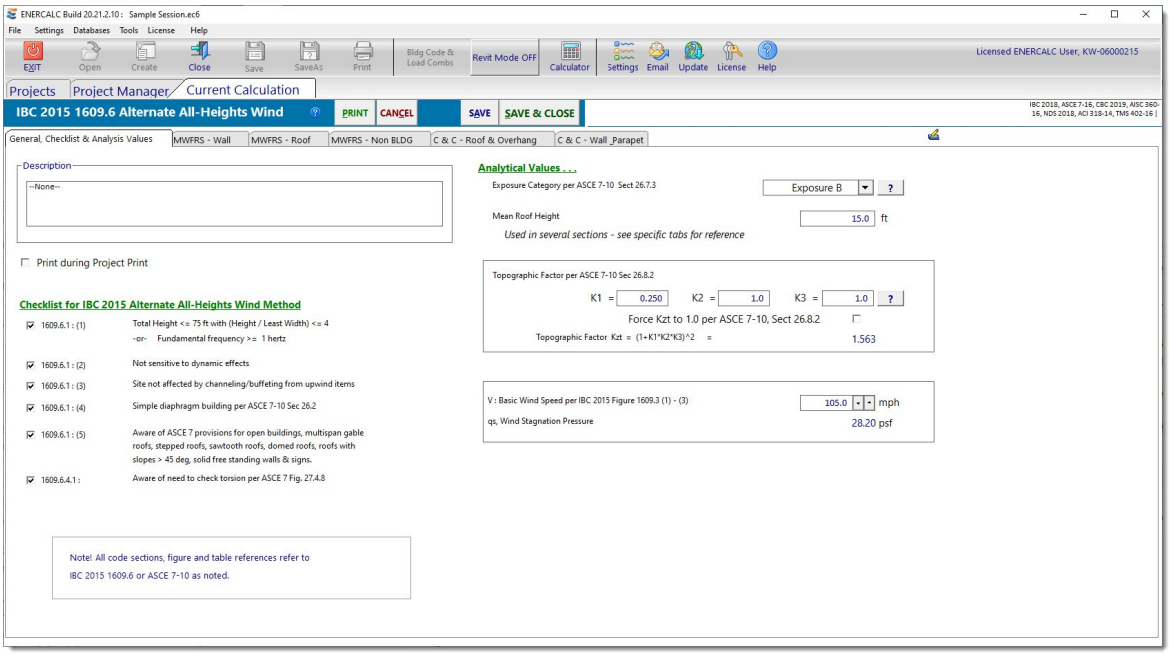
13.2.13 IBC Alternate All-Heights Wind Method

This module is an implementation of the IBC Alternate All-Heights Wind Method.

© 2026 ENERCALC, LLC

General, Checklist, & Analysis Values

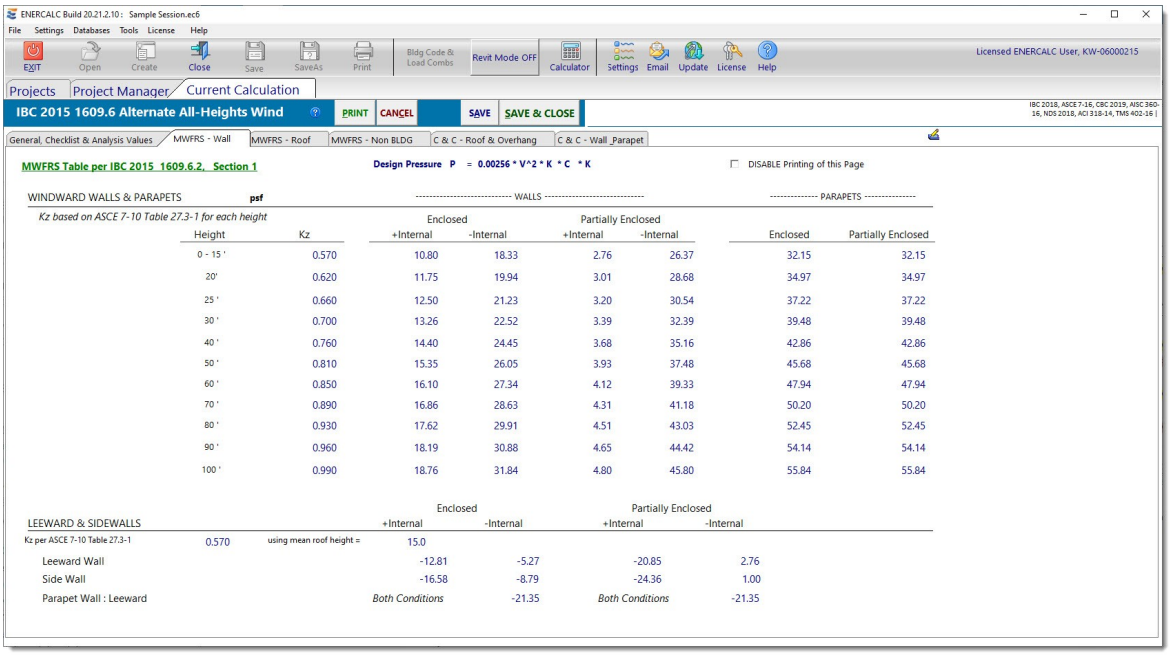
This tab offers a checklist of the criteria that must be satisfied in order to apply the IBC 2015 Alternate All-Heights Wind Method. Once all of the checkboxes are checked, the module will offer input fields for analytical values such as Exposure Category, Mean Roof Height, Topographic Factor parameters, Basic Wind Speed, Occupancy category, and options for the selection of Velocity Pressure Coefficients.



© 2026 ENERCALC, LLC

MWFRS - Wall

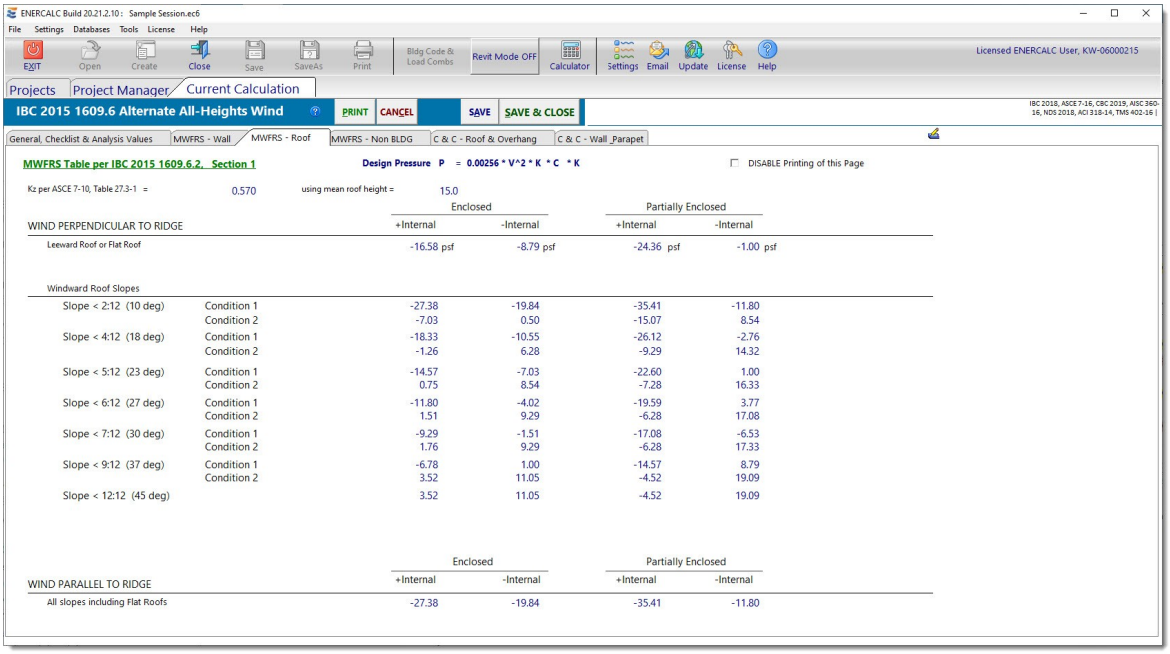
This tab reports design pressures on windward walls, leeward walls, sidewalls and parapets. Pressure values are distinguished for Enclosed and Partially Enclosed structures, and values are reported for the positive internal pressure condition and for the negative internal pressure condition.



© 2026 ENERCALC, LLC

MWFRS - Roof

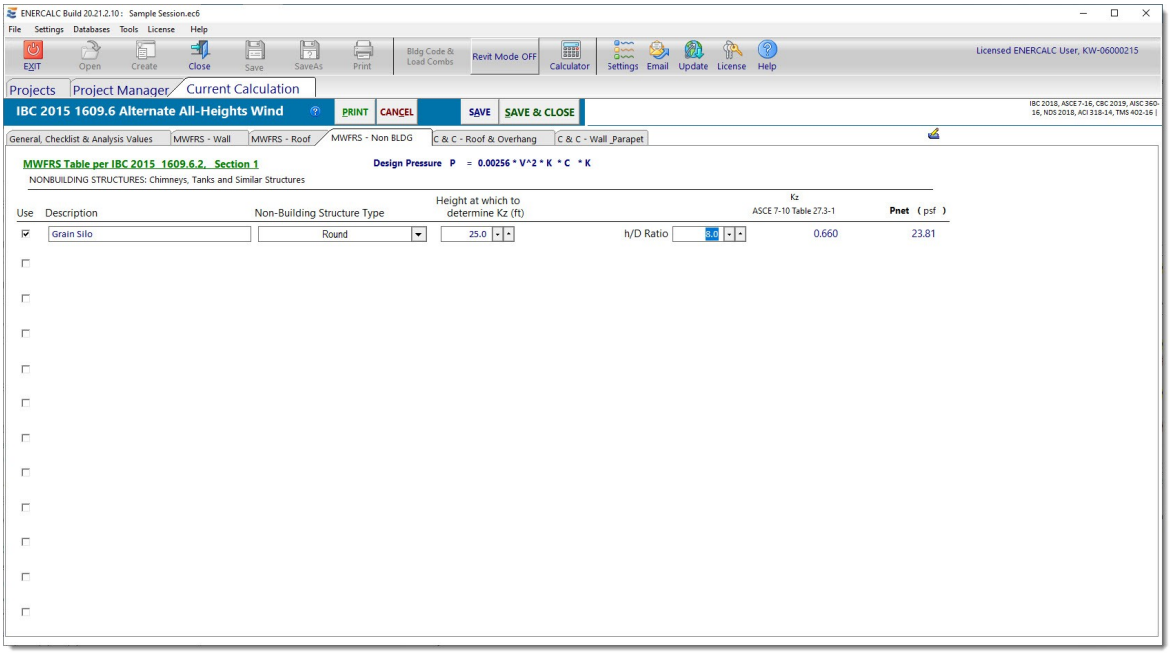
This tab reports design pressures on roof surfaces. Pressure values are distinguished for the conditions of wind perpendicular to the ridge and wind parallel to the ridge. In the case of wind perpendicular to the ridge, pressure values are reported for flat roofs as well as for various slope angles. In all cases, results are presented for Enclosed and for Partially Enclosed structures, and values are reported for the positive internal pressure condition and for the negative internal pressure condition.



© 2026 ENERCALC, LLC

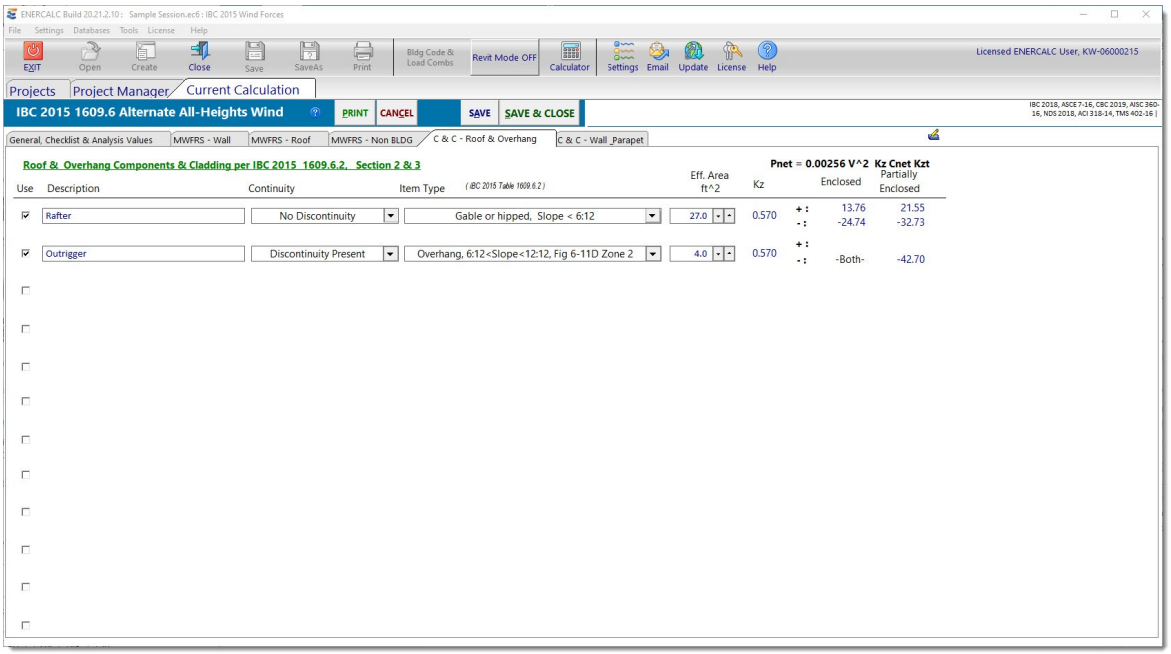
MWFRS - Non BLDG

This tab reports net design pressures on the projection of surfaces of non-building structures.



C&C - Roof & Overhang

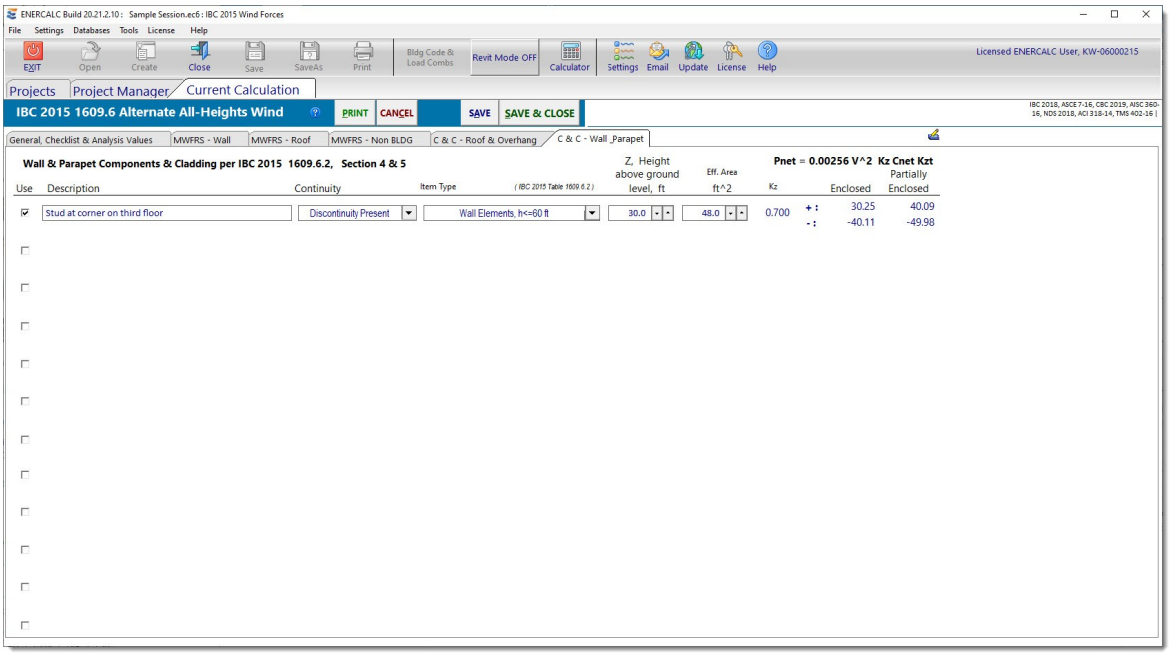
This tab reports design pressures on components and cladding within roof surfaces. Results are presented for Enclosed and for Partially Enclosed structures, and values are reported for the positive internal pressure condition and for the negative internal pressure condition.



© 2026 ENERCALC, LLC

C&C - Wall & Parapet

This tab reports design pressures on components and cladding within wall and parapet surfaces. Results are presented for Enclosed and for Partially Enclosed structures, and values are reported for the positive internal pressure condition and for the negative internal pressure condition.



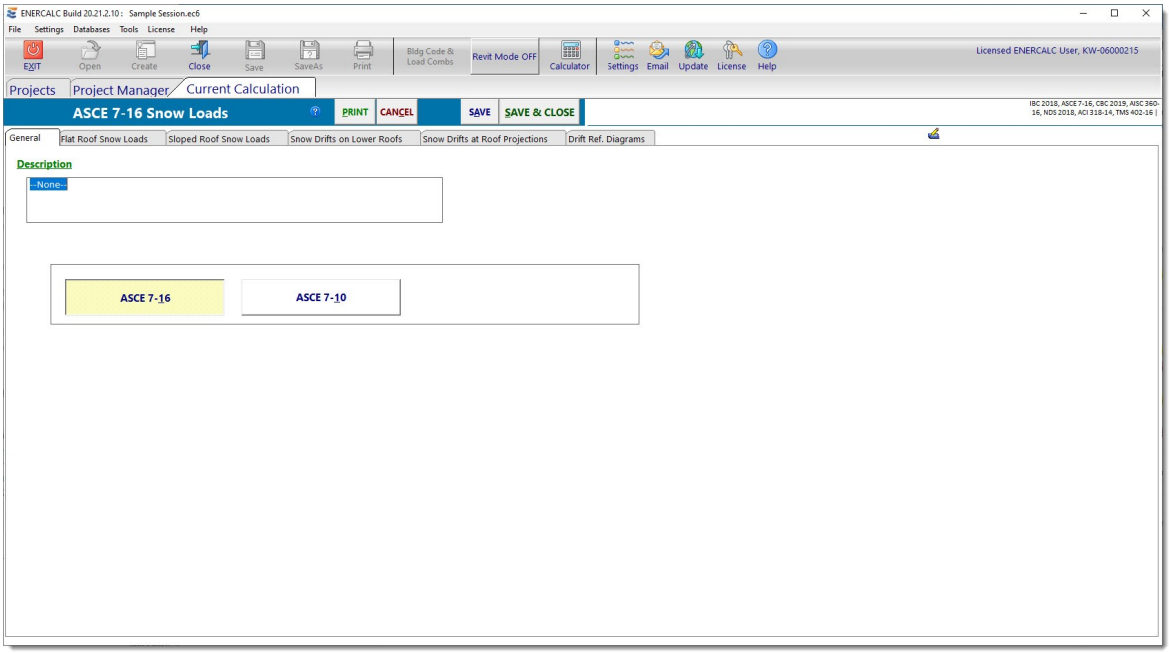
13.2.14 ASCE Snow Loads

This module is an implementation of the ASCE method of determining Flat Roof Snow Load, Sloped Roof Snow Load, and Snow Drifts on Lower Roofs, and Snow Drifts at Roof Obstructions. The module currently supports calculations based on ASCE 7-10 and ASCE 7-16. The Snow Loads module does ***not*** currently address the calculation of unbalanced snow loads, rain on snow surcharges, or sliding snow surcharges.

© 2026 ENERCALC, LLC

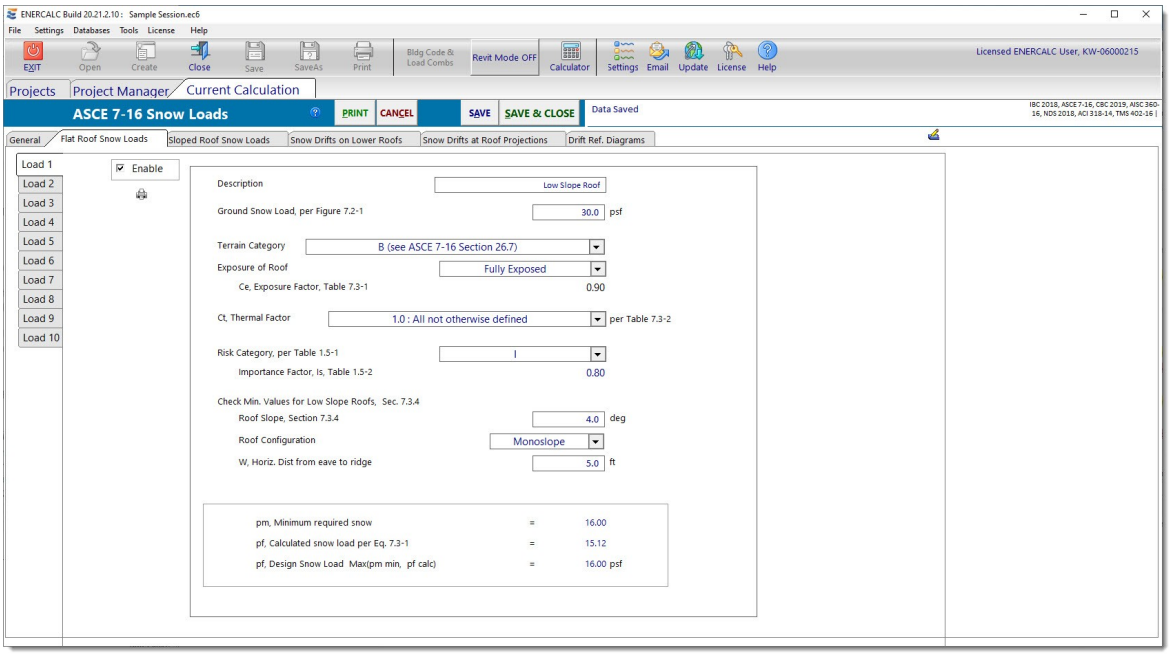
General

Provides a place to enter descriptive text.



Flat Roof Snow Loads

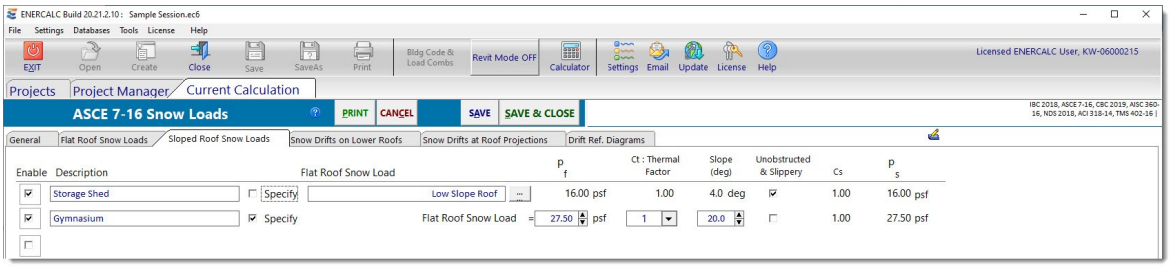
Allows up to 10 different flat roof snow loads to be calculated by entering values for all of the parameters in accordance with ASCE 7.



© 2026 ENERCALC, LLC

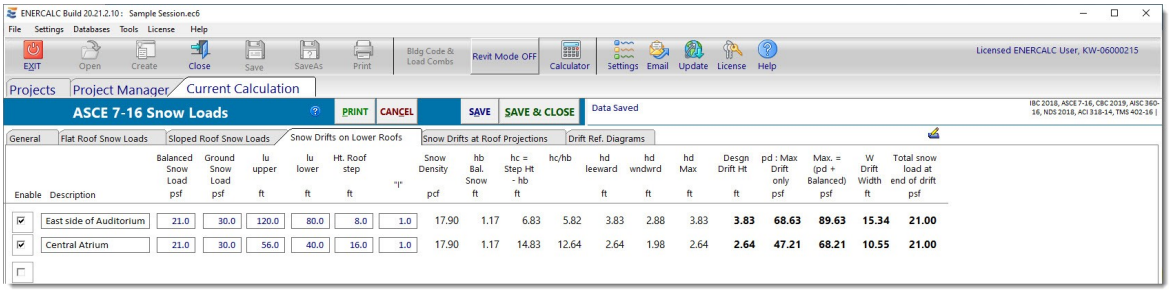
Sloped Roof Snow Loads

Allows up to 10 different sloped roof snow loads. The input form provides the option to refer to a previously defined Flat Roof Snow Load, or to explicitly specify a Flat Roof Snow Load value along with the remaining values for all of the parameters in accordance with ASCE 7.



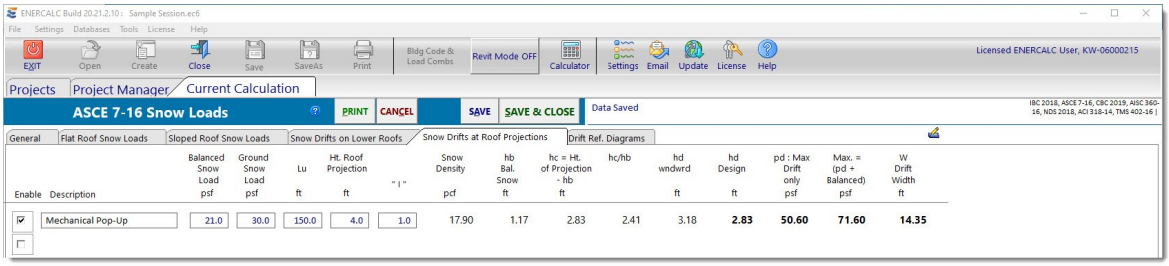
Snow Drifts on Lower Roofs

Allows up to 10 different conditions to be evaluated for snow drifts on lower roofs. The input form collects values for all of the necessary parameters to determine the windward and leeward drifts and selects the controlling case in accordance with ASCE 7.



Snow Drifts at Roof Projections

Allows up to 10 different conditions to be evaluated for snow drifts at roof projections. The input form collects values for all of the necessary parameters to determine the windward drift in accordance with ASCE 7.



13.2.15 ASCE 7-22 Snow Loads

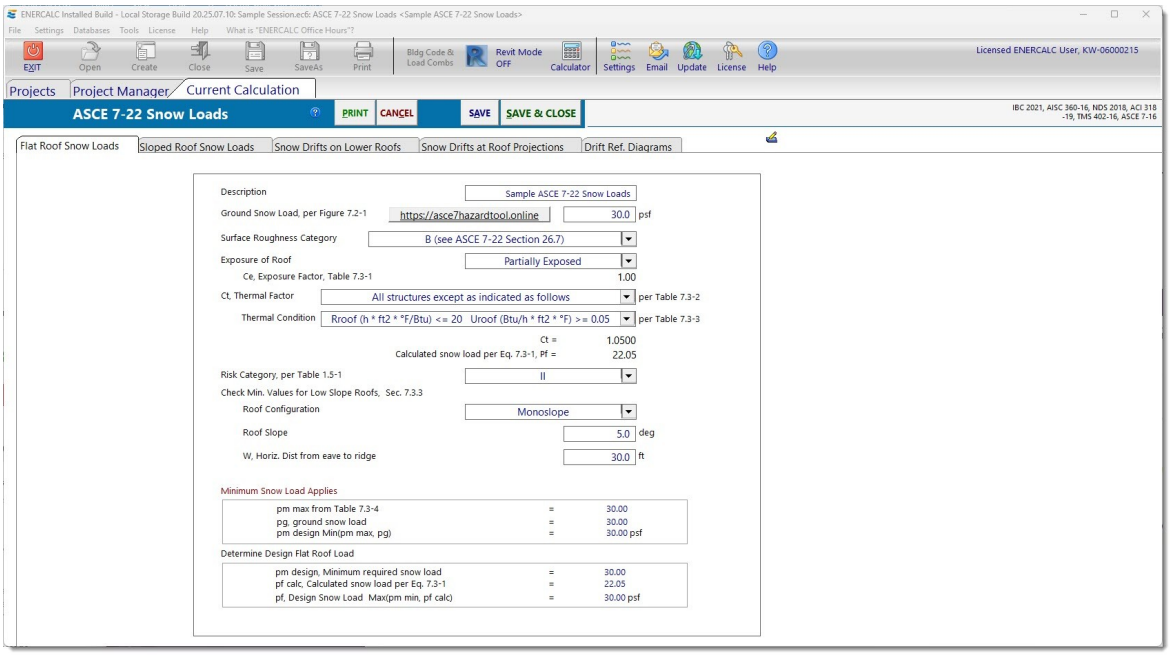
This module is an implementation of the ASCE 7-22 method of determining Flat Roof Snow Load, Sloped Roof Snow Load, and Snow Drifts on Lower Roofs, and Snow Drifts at

© 2026 ENERCALC, LLC

Roof Obstructions. The Snow Loads module does ***not*** currently address the calculation of unbalanced snow loads, rain on snow surcharges, or sliding snow surcharges.

Flat Roof Snow Loads

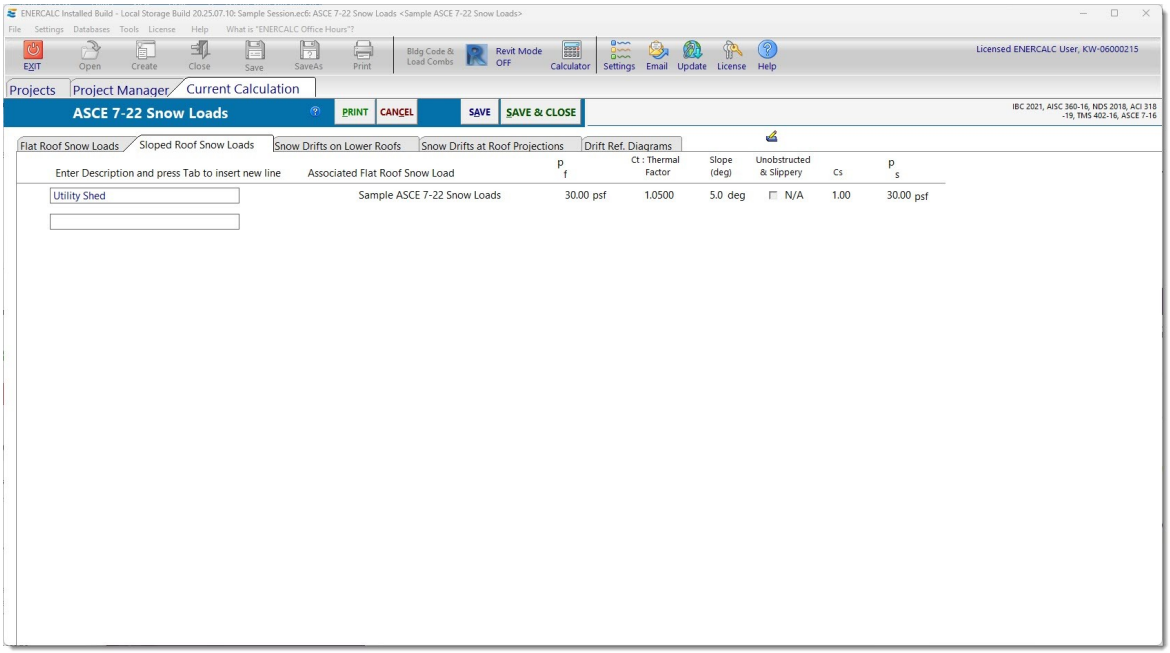
Collects values for determination of the flat roof snow load in accordance with ASCE 7.



© 2026 ENERCALC, LLC

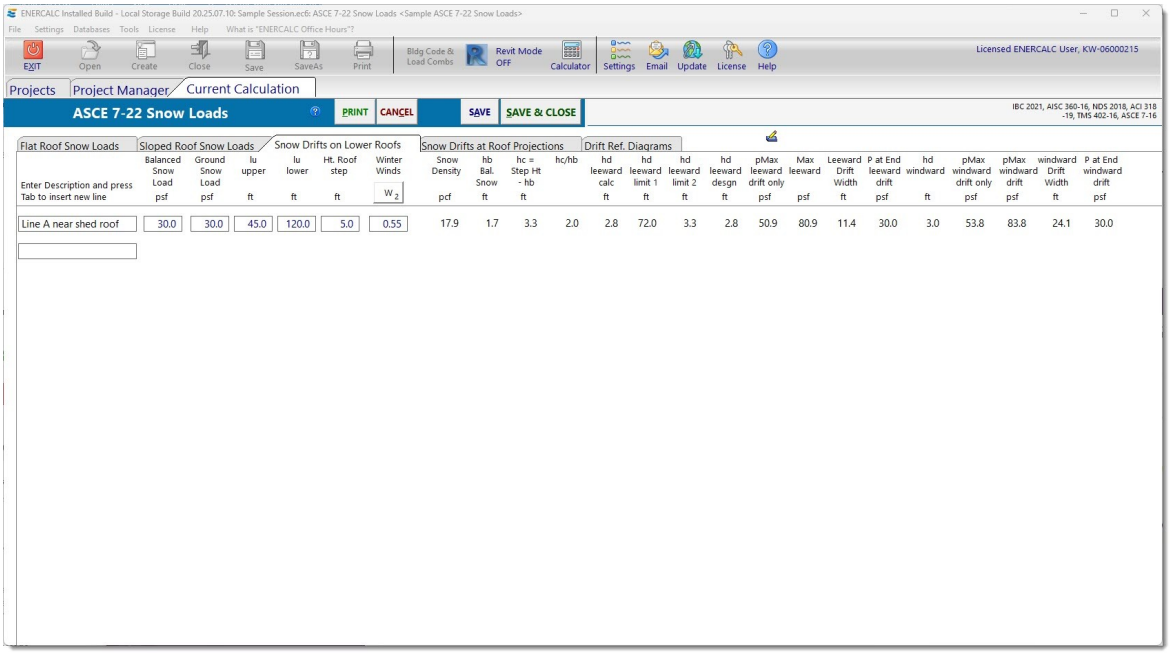
Sloped Roof Snow Loads

Collects parameters to calculate sloped roof snow loads in accordance with ASCE 7.



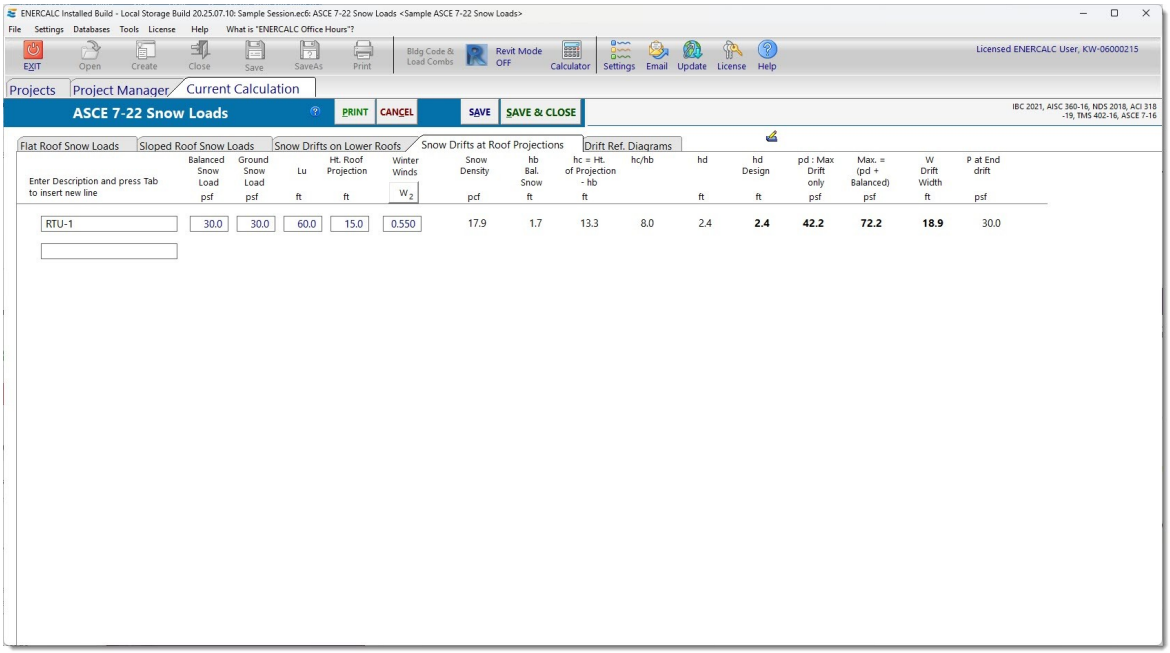
Snow Drifts on Lower Roofs

The input form collects values for all of the necessary parameters to determine the windward and leeward drifts in accordance with ASCE 7.



Snow Drifts at Roof Projections

The input form collects values for all of the necessary parameters to determine drifts at roof projections in accordance with ASCE 7.

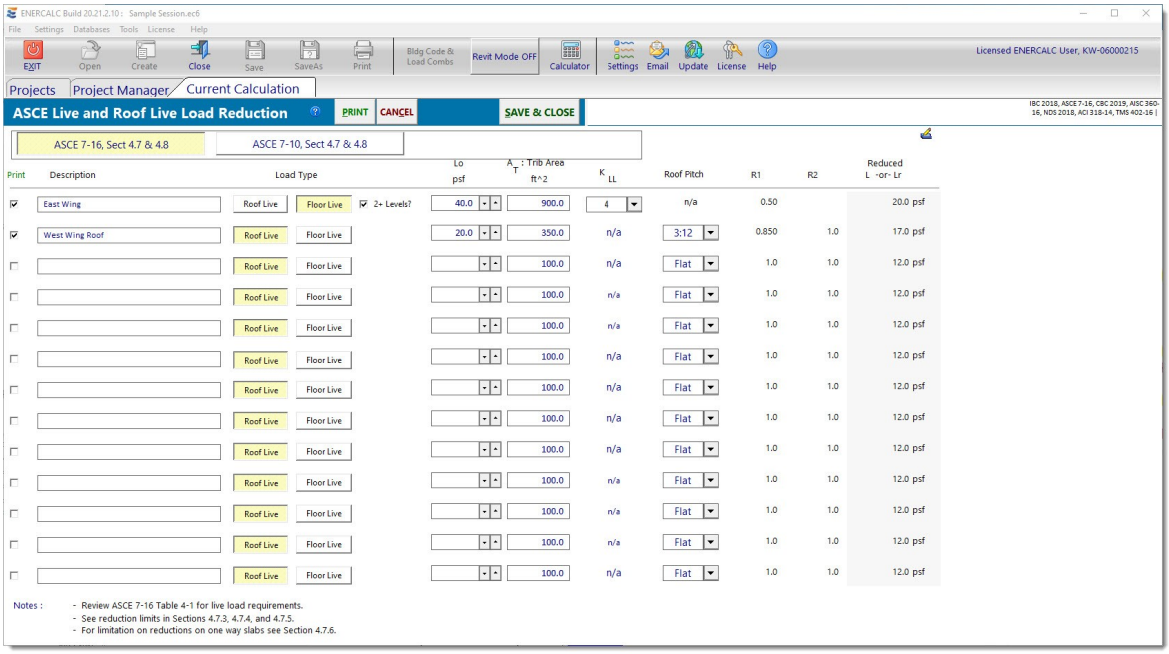


13.2.16 ASCE Live and Roof Live Load Reduction

This section enables the user to calculate reduced live and roof live loads for various parts of a project.

© 2026 ENERCALC, LLC

This section is a direct application of ASCE 7-10 and ASCE 7-16 Sections 4.7 & 4.8 concerning live and roof live load reductions. We refer the user to those sections of the referenced design standards for specific descriptions of the values used in each column seen below.

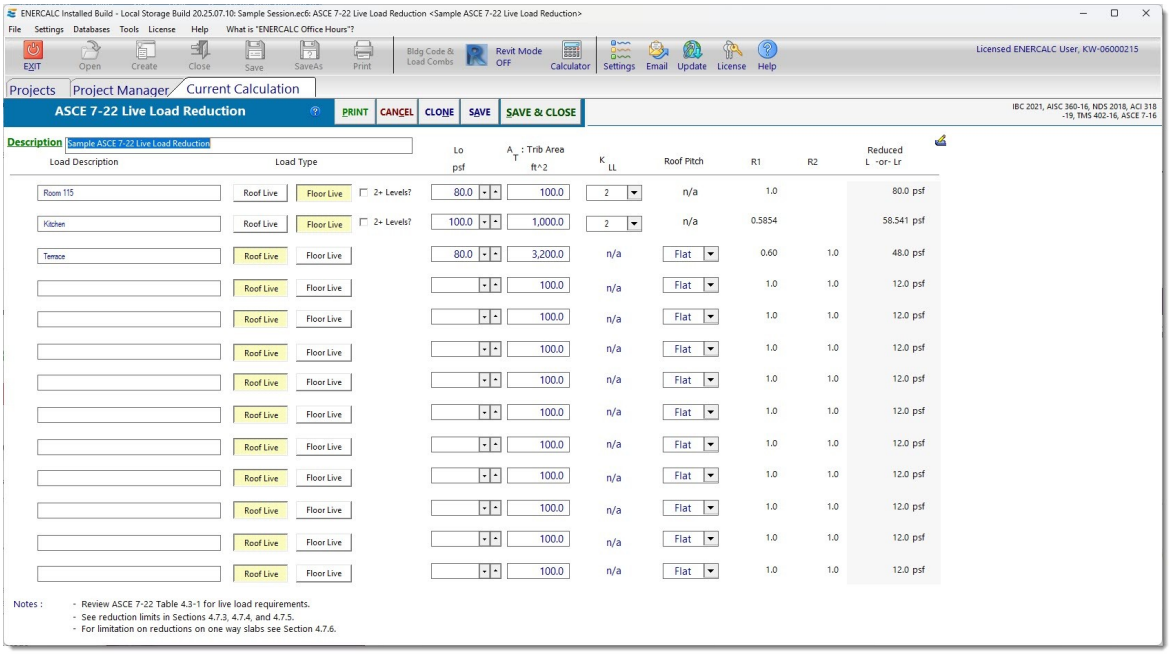


13.2.17 ASCE 7-22 Live and Roof Live Load Reduction

This section enables the user to calculate reduced live and roof live loads for various parts of a project.

© 2026 ENERCALC, LLC

This section is a direct application of ASCE 7-22 Sections 4.7 & 4.8 concerning live and roof live load reductions. We refer the user to those sections of the referenced design standards for specific descriptions of the values used in each column seen below.

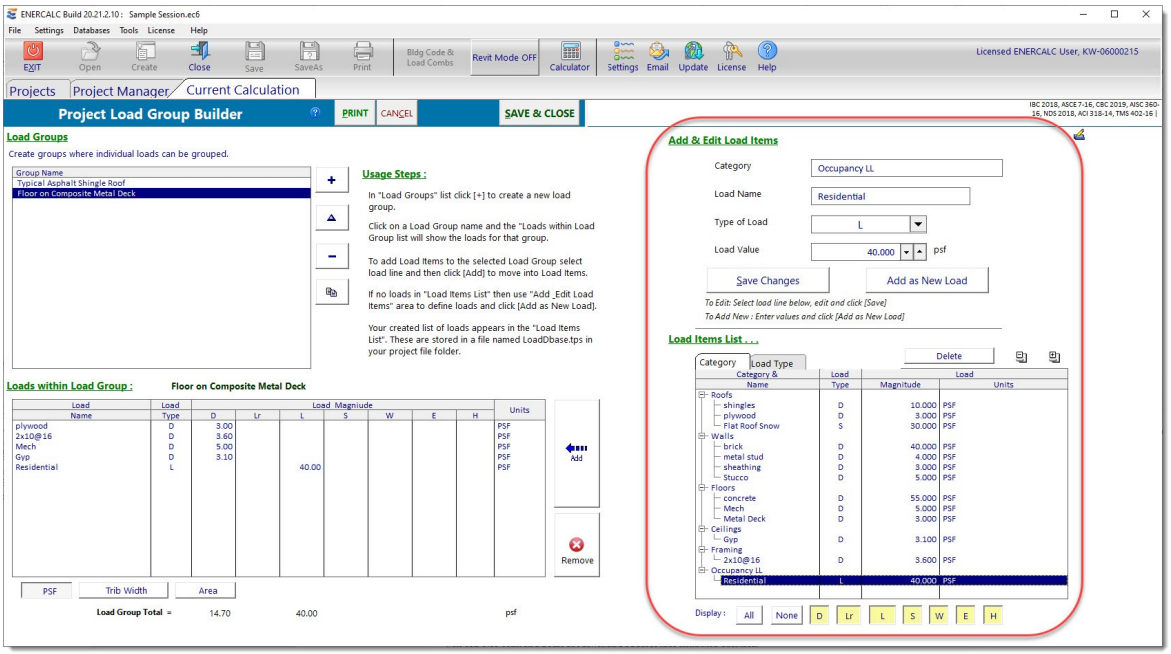


13.2.18 Project Load Group Builder

The Project Load Group Builder enables the user to store commonly used "Load Items" and then combine them into various Groups to calculate total loads for the defined Groups.

© 2026 ENERCALC, LLC

Load Items are defined and organized on the right-hand side of the screen as indicated in the screen capture below:



You can type in a Category name (discussed in more detail below), a Load Name, choose the Load Type, and specify the Load Value, and then click "Add as New Load". The newly added Load Item will appear in the Load Items List below. To edit an existing Load Item, click on the item of interest in the Load Items List, edit the values in the individual fields above, and then click "Save Changes"

Load Items are very flexible in their definition. A Load Item could represent the weight of a particular building construction material, such as the Dead Load of 3/4" plywood floor sheathing in units of PSF. A Load Item could also represent an allowance of 5 PSF Dead Load for ceiling & lights. But Load Items can also take the form of load types other than Dead Load as well. For instance, you could define a Load Item consisting of 100 PSF Live Load, or 30 PSF Snow Load. In this way, the Project Load Group Builder is capable of tracking loads of all types that may be applicable to a particular area of a building.

One of the powerful features of the Project Load Group Builder is that it takes the Load Item data that you enter and it saves that data in a file that is separate and distinct from your Project File. (The file is named LoadDbase.tps, and it is saved in the folder that you have identified as your default Project File location.) In this way, the system continues to accumulate more and more of your commonly specified materials and loads as you work on different projects. And that ever-growing database of your custom Load Items is always available to you on future projects.

© 2026 ENERCALC, LLC

In order to bring some organization to the many load items that you may eventually accumulate, the Project Load Group Builder introduces the concept of the "Category". A Category can contain one or more Load Items. Here again, the way you use Categories can be extremely flexible. Some users may prefer to organize their Load Items into Categories that represent different types of materials or loads. For example, the organizational hierarchy might take on a form like this where the bullet items represent Load Items within the indicated Category:

Category: Roof Sheathing
- 1/2" sheathing
- 5/8" sheathing
- 3/4" sheathing
- 7/8" sheathing
- 1" sheathing

Category: Roof Framing
- 2x6@16" o/c
- 2x8@16" o/c
- 2x10@16" o/c
- 2x12@16" o/c

Category: NW Concrete Slab
- 4.5" total with 2" deck
- 5.0" total with 2" deck
- 5.5" total with 2" deck
- 6.0" total with 2" deck
- 5.5" total with 3" deck
- 6.0" total with 3" deck
- 6.5" total with 3" deck

Category: LW Concrete Slab
- 4.5" total with 2" deck
- 5.0" total with 2" deck
- 5.25" total with 2" deck
- 5.5" total with 2" deck
- 6.0" total with 2" deck
- 5.5" total with 3" deck
- 6.0" total with 3" deck
- 6.25" total with 3" deck
- 6.5" total with 3" deck

© 2026 ENERCALC, LLC

Category: Live Load
- Classroom
- First Floor Corridor
- Office
- Stack Rooms

Category: Snow Load
- 30 PSF
- 32 PSF
- 34 PSF
- 36 PSF

And so on...

But remember, this is just one way that Categories could be used. Here is yet another idea:

Category: Typical classroom on structural floor
- DL of flooring
- DL of concrete slab on metal deck
- DL of composite metal deck
- DL allowance for structural steel framing
- DL allowance for ceiling & lights
- DL allowance for mechanical & misc.
- LL for classroom occupancy
- LL for partition allowance

Category: Typical corridor on structural floor
- DL of flooring
- DL of concrete slab on metal deck
- DL of composite metal deck
- DL allowance for structural steel framing
- DL allowance for ceiling & lights
- DL allowance for mechanical & misc.
- LL for corridor above first floor occupancy

Category: Typical roof structure
- DL of asphalt shingles
- DL allowance for second layer of asphalt shingles
- DL of metal roof deck
- DL structural steel framing
- DL allowance for ceiling & lights
- DL allowance for mechanical & misc.
- SL for sloped roof

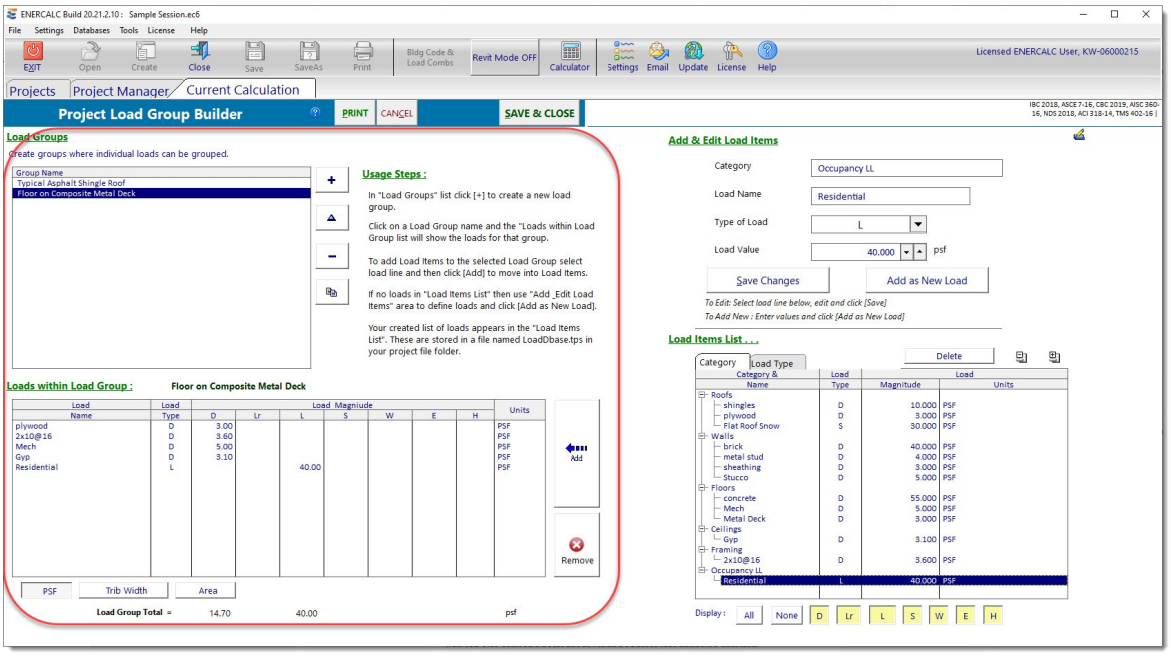
And so on...

© 2026 ENERCALC, LLC

So remember that your Load Items and your Categories are being stored in a separate file, and they are continuing to accumulate from one project to the next as you enter more data. Consider this as you decide on a system of organization that will work best for you.

Now that we have covered Load Items and Categories, we can introduce the *real* purpose for this module, which is the "Load Group". Very simply, a Load Group is a group of Load Items that will be summed together.

Load Groups are defined and manipulated on the right-hand side of the screen as indicated in the screen capture below:



The Load Groups in the current Project File are listed in the "Group Name" list in the upper left corner of the module. The toolbar buttons above the list perform the following functions:

Adds a new Load Group

Creates a copy of the selected Load Group

Allows the name of the selected Load Group to be edited

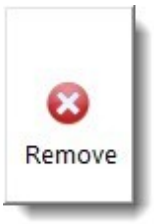
© 2026 ENERCALC, LLC

Deletes the selected Load Group

When a Load Group is selected in the "Group Name" list, the details of that Load Group are displayed in the lower left corner of the screen. (A newly created Load Group will display no detailed information until some Load Items are added to the Load Group.) The display will show the Load Items that have been added to the current Load Group including the Load Item name, the Load Type (Dead Live, Snow, etc.), the magnitude of the load, and the units.

So we have now covered Load Items and Load Groups. The final detail is...how do we add these Load Items to Load Groups? The answer lies in the buttons shown below:

Adds the selected Load Item to the current Load Group



Removes the selected Load Item from the current Load Group

Finally, the Print function will automatically include all defined Load Groups and provide a nice tabular summary of the Load Items that combine to produce each Load Group.

13.3 Analysis

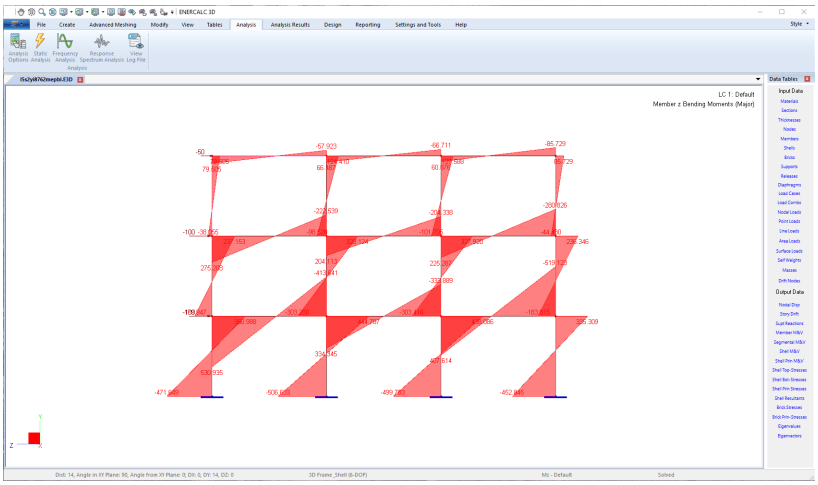
Please select a subtopic.

13.3.1 ENERCALC 3D

ENERCALC 3D

A Structural Analysis and Design Program

© 2026 ENERCALC, LLC



Accuracy, Reliability, Ease of Use

ENERCALC, INC.

13.3.1.1 License Agreement

OpenGL® is a registered trademark of Silicon Graphics, Inc. (SGI).
Windows® is a registered trademark of Microsoft Corporation.
ENERCALC 3D is a trademark of ENERCALC, Inc.

Copyright 2010-2020 by ENERCALC. All rights reserved.

13.3.1.2 Terms and Conventions

The convention for commands in this documentation is Main Menu > Sub-Menu. For example, Create > Line Loads means the Line Loads command from the Create item in the main ribbon.

Model View: A window in the program that contains the graphical display of the model.

Report View: A window in the program that contains the text or graphical report.

Structural Command: A command in the program that affects the results for a model.

Member: A beam or frame element. It also refers to a truss when the element has full moment releases at two ends. The term “beam element”, “frame element” and “member” are used interchangeably in this program.

Shell: a four-node shell finite element. It includes membrane action and plate bending action. It is sometimes called a plate.

Brick: an eight-node solid finite element.

Entity: A node, member, shell or brick.

Element: A member or finite element (shell or brick).

Object: A node or finite element (shell or brick) or its dependent.

Dependent: A structural entity whose existence depends upon the existence of another structural entity. For example, a support is a dependent of a node; a moment release is a dependent of a member (beam element). All loads are dependents of nodes or members or finite elements.

Parent: A structural entity which may have dependents. Nodes and elements may be parents. For example, a node may be a parent of a support or a member. A member may be a parent of a moment release.

Distance List: A comma separated list that specifies multiple distances. For example, a distance list of "12,2@14,3@10" will generate distances of 12, 14, 14, 10, 10, and 10 in length units.

Orphaned Node: A node that is not connected to any elements.

DOFs: Degrees of freedom.

64-bit floating point (double precision): The solver that uses 64-bit (8 bytes) floating-point arithmetic. The 64-bit floating point (double precision) is the standard solver in almost all structural analysis programs.

128-bit floating point (quad precision): The solver that uses 128-bit (16 bytes) floating-point arithmetic. The 128-bit floating point (quad precision) is extremely accurate and is uniquely available in ENERCALC 3D.

13.3.1.3 Related Links

Use the following links to access detailed info.

ENERCALC 3D Training Manual: https://enercalc.com/3d_training

ENERCALC 3D Verifications Manual: https://enercalc.com/ec3d_verification/

13.3.1.4 Introduction

Built from the ground up, ENERCALC 3D is a powerful structural design / finite element analysis software tool designed for structural engineers of all skill levels. ENERCALC 3D is reliable, easy to use, and affordable. The software is designed for accuracy and simplicity, allowing engineers to get the job done without being overwhelmed by useless features.

The program includes the following frame and finite elements:

- 2D and 3D beam and truss element (also called member). The element can be linear, tension-only or compression-only.
- 3D four-node shell element, with thick (MITC4) plate and thin (Kirchhoff) plate bending and plane membrane stress (compatible and incompatible) formulations.
- 3D eight-node solid element (brick) with compatible and incompatible formulations.
- Linear and nonlinear nodal, line, and surface spring elements.
- Rigid diaphragm.

© 2026 ENERCALC, LLC

The program includes the following analysis and design options:

- Static linear analysis.
- Geometric nonlinear (P-Delta) analysis.
- Standard 64-bit floating point (double precision) and extremely accurate 128-bit floating point (quad precision) skyline solvers.
- Lightning fast sparse solver based on Intel PARDISO solver.
- Nodal, point, line, and surface forces; point moments; self-weight.
- Forced displacements on supports.
- Member moment releases.
- Concrete beam, column and slab/wall designs according to ACI 318-02/05/08/11/14.
- Steel beam and column design according to AISC 15th Edition LRFD.

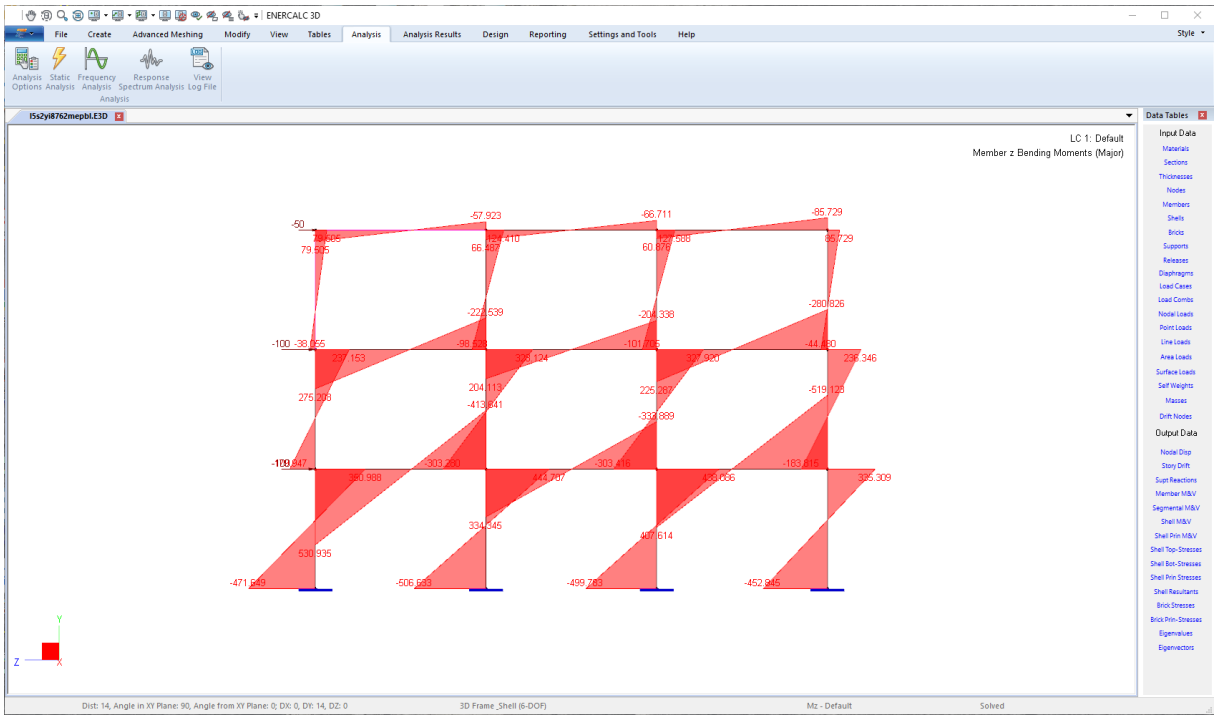
The program provides the following main user interface features:

- Multiple views with different display settings.
- Graphically drawing nodes, members and finite elements, area loads and rigid diaphragms via mouse-click.
- Versatile spreadsheets for input data and results.
- Powerful automatic model generations for continuous beams; 2D and 3D frames; 2D and 3D shells; arc beams and non-prismatic beams.
- Quality 3D graphical rendering with hidden line or surface removal based on OpenGL®
- Loading diagram; moment and shear diagram for members; contours for shells and solids; deflection diagram.
- Flexible editing features such as undo/redo, duplicate, move, scale, delete, revolve, extrude, splitting members, sub-mesh shells, node and element merging.
- Real time panning, zooming and rotating.
- Many different selection methods such as point/window/cross select, select by IDs, select by properties, with options to freeze or thaw parts of a model.
- Flexible annotations for input and results.
- Text and graphical reports in html format. Graphical report may contain multiple images. Text report may be saved in plain text format.
- Print previews for graphical and text reports

© 2026 ENERCALC, LLC

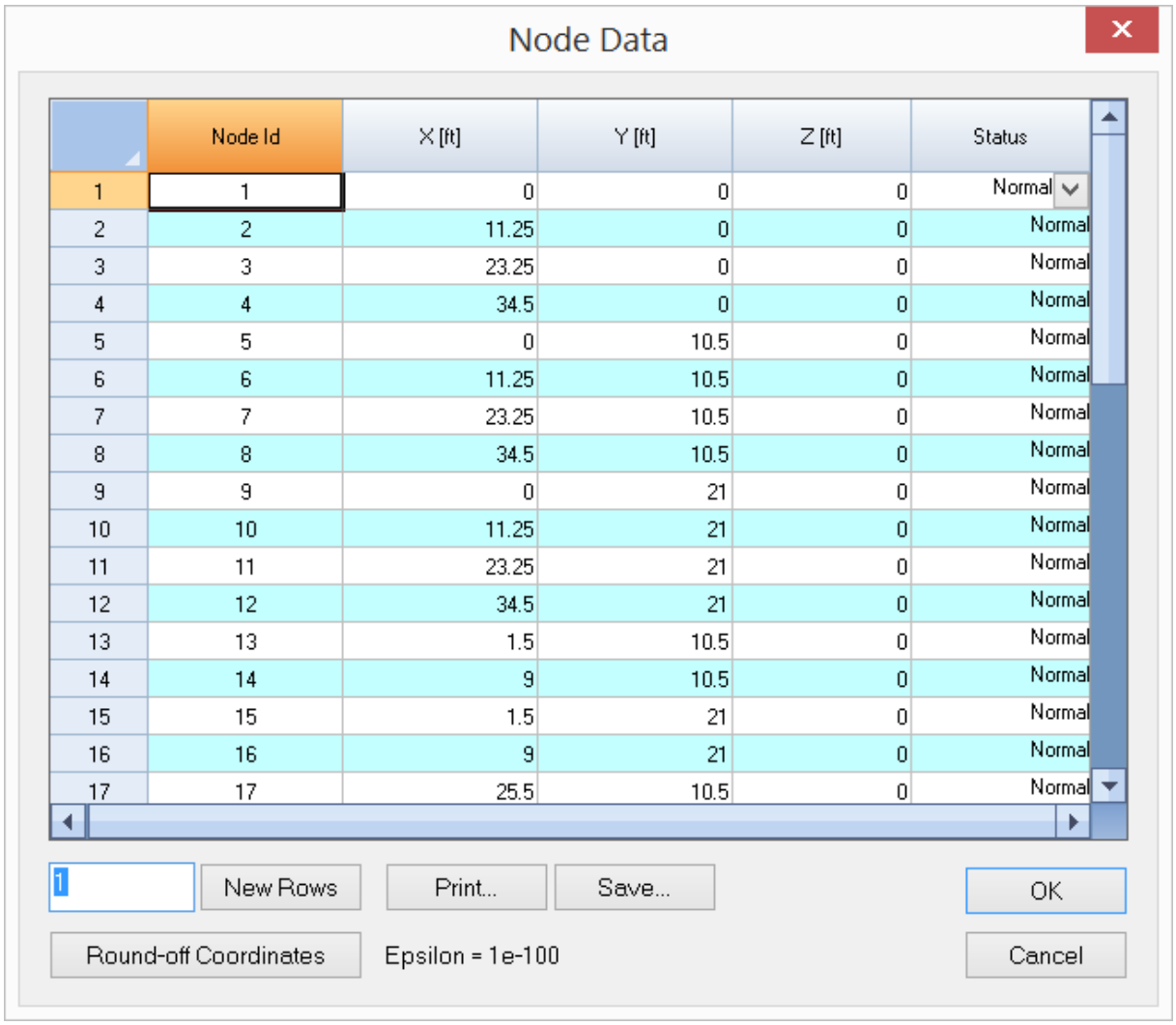
Graphical User Interface (GUI)

ENERCALC 3D has a modern graphical user interface. It includes ribbons, toolbars, multiple views and a status bar.



Spreadsheet Navigation

The program uses spreadsheets extensively for data input and output. It offers multiple ways to navigate within a spreadsheet as specified in the following table.



Key	Action
up arrow	Moves active cell up one row
down arrow	Moves active cell down one row
right arrow	Moves active cell right one column
left arrow	Moves active cell left one column
Shift+arrow	Extends selection in direction of arrow key

© 2026 ENERCALC, LLC

Key	Action
Page Up	Moves active cell one page up
Page Down	Moves active cell one page down
Ctrl+Page	Moves active cell one page left
Ctrl+Page	Moves active cell one page right
Home	Moves active cell to first cell in row
End	Moves active cell to last cell in row that contains data
Ctrl+Home	Moves active cell to first row, first column
Ctrl+End	Moves active cell to last row and column that contain data
Tab	Moves active cell to next cell to the right (or at end of row moves to beginning of next row)
Shift+Tab	Moves active cell to next cell to the left (or at beginning of row moves up to end of row above)
Shift+space	Selects current row
Ctrl+space	Selects current column
Shift+Ctrl+s	Selects entire sheet
Ctrl+X or	Cuts current selection or active cell's data to Clipboard

© 2026 ENERCALC, LLC

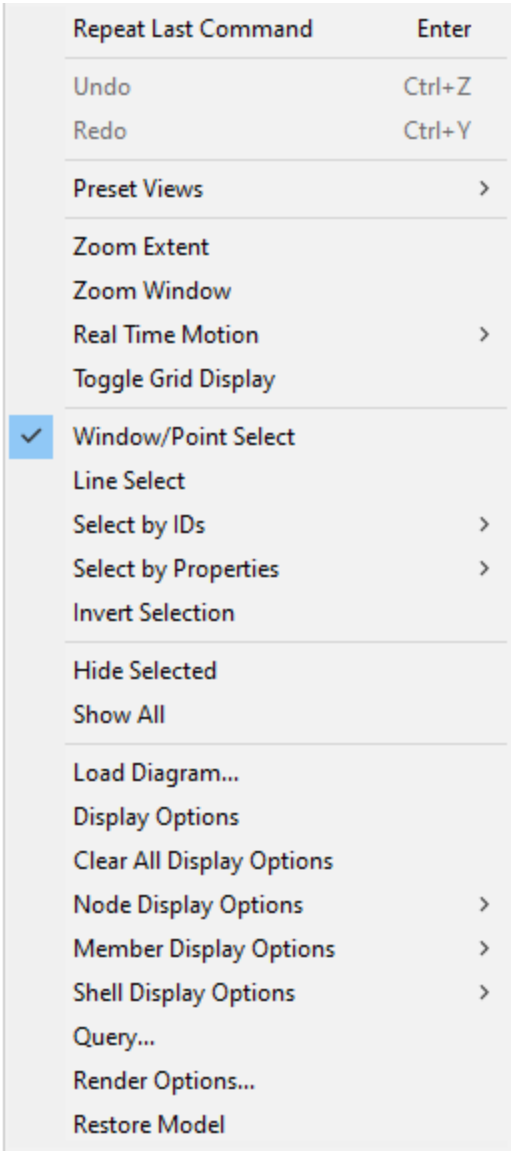
Key	Action
Ctrl+V or	Pastes Clipboard contents into active cell
Ctrl+C or	Copies current selection or active cell's data to Clipboard
Enter	Active cell moves down
Esc	If sheet is in edit mode, previous cell value replaces new value and edit mode is turned off
F2	If edit mode is on, cell value is cleared

13.3.1.5 Ribbons

© 2026 ENERCALC, LLC

Right-click Menu

One important feature of the graphical user interface is the right-click menu. It has been designed to put commonly used commands within close reach at all times. No matter what ribbon tab is displayed, right clicking on the main area of the screen will open the following menu:



Quick Access Toolbar

Pan

The Pan command in the Quick Access toolbar is the same as View > Pan.

© 2026 ENERCALC, LLC

Rotate

The Rotate command in the Quick Access toolbar is the same as View > Rotate.

Zoom

The Zoom command in the Quick Access toolbar is the same as View > Zoom.

Restore Model

The Restore Model command in the Quick Access toolbar is the same as View > Restore Model.

Node Display Options

Quick Access toolbar > Node Display Options opens a dialog that offers various options to control the display of data items related to nodes.

Member Display Options

Quick Access toolbar > Member Display Options opens a dialog that offers various options to control the display of data items related to members.

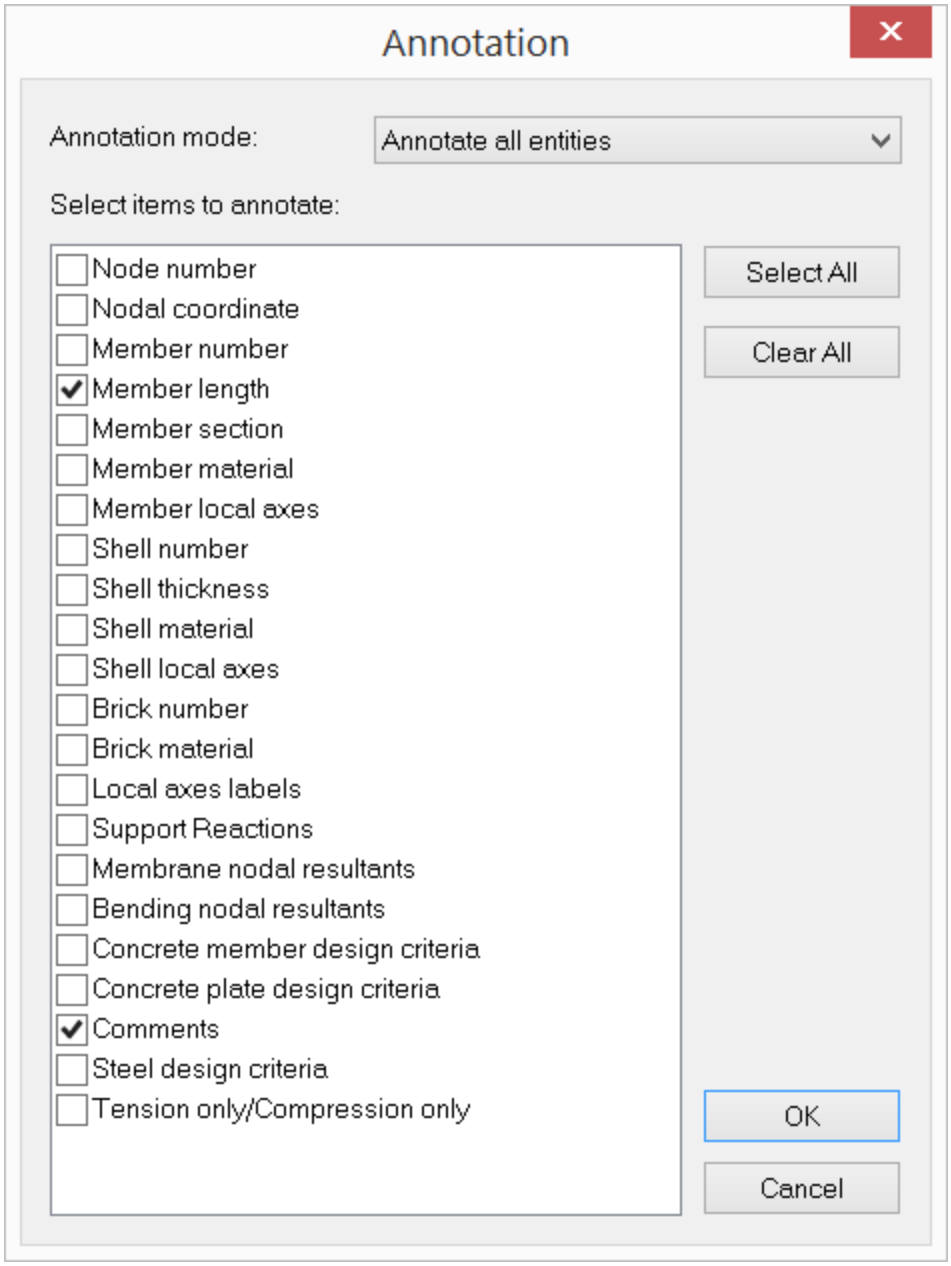
Shell Display Options

Quick Access toolbar > Shell Display Options opens a dialog that offers various options to control the display of data items related to shells.

© 2026 ENERCALC, LLC

Display Options

Quick Access toolbar > Display Options prompts you with the following dialog.



It allows you to view annotations for nodes and elements and their properties. The element local axes may also be displayed using this command. Three annotation modes are available. You may annotate all entities, annotate selected entities, and erase existing annotations. For performance reasons, it is recommended that the annotations be applied only for objects of interest.

© 2026 ENERCALC, LLC

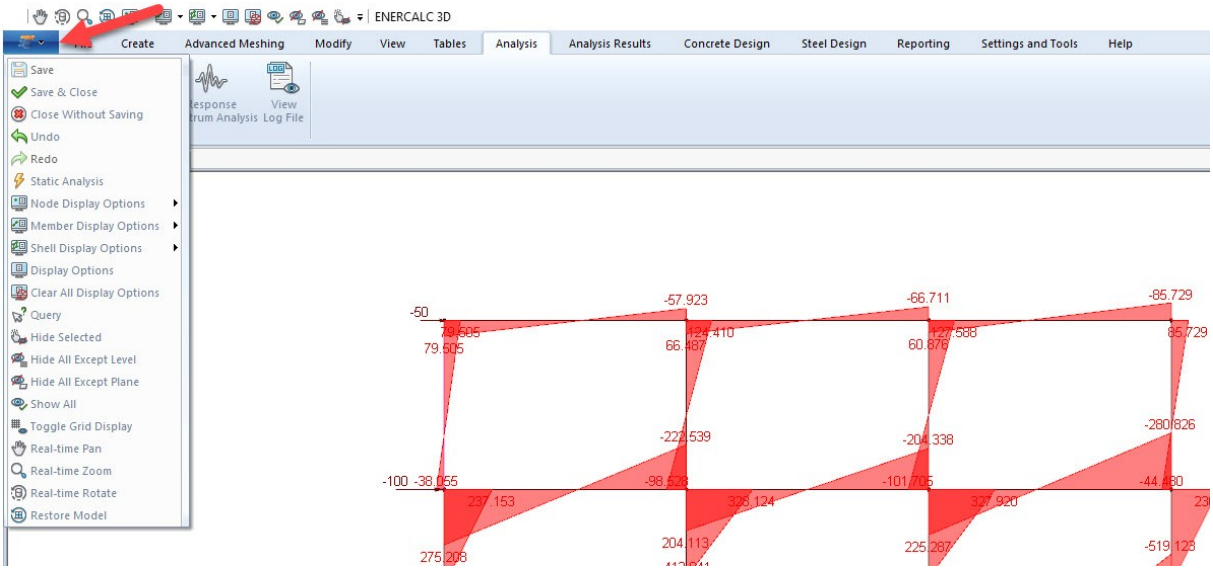
Clear All Display Options

Quick Access toolbar > Clear All Display Options is a convenient way to turn off all display options in one step.

Isometric (and other) View

The Isometric View command in the Quick Access toolbar is the same as View > Preset Views.

Quick Access Tab



Save

The Save command in the Quick Access tab is the same as File > Save.

Save & Close

The Save & Close command in the Quick Access tab is the same as File > Save & Close.

Close without Saving

The Close without Saving command in the Quick Access tab is the same as File > Close without Saving.

Undo

Quick Access tab > Undo undoes the previous structural command. By default, you may undo up to 10 levels. You may set a different number of undo levels by running Settings & Tools > Data Options. Non-structural commands such as zooming or panning may not be undone. More undo levels requires more computer memory.

Redo

Quick Access tab > Redo reverses the previous Undo command.

© 2026 ENERCALC, LLC

Static Analysis

The Static Analysis command in the Quick Access tab is the same as Analysis > Static Analysis.

Node Display Options

Quick Access tab > Node Display Options opens a dialog that offers various options to control the display of data items related to nodes.

Member Display Options

Quick Access tab > Member Display Options opens a dialog that offers various options to control the display of data items related to members.

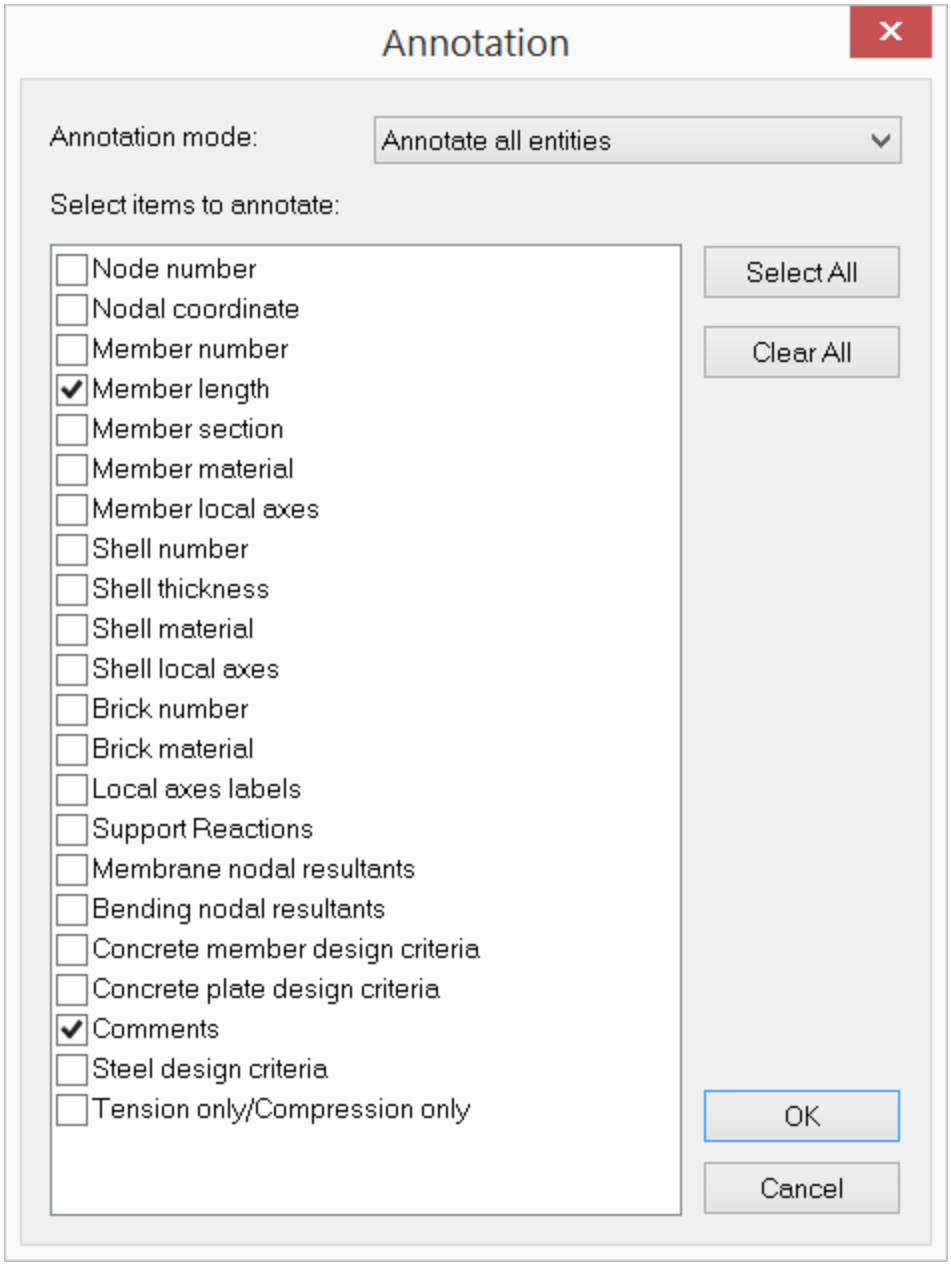
Shell Display Options

Quick Access tab > Shell Display Options opens a dialog that offers various options to control the display of data items related to shells.

© 2026 ENERCALC, LLC

Display Options

Quick Access tab > Display Options prompts you with the following dialog.



It allows you to view annotations for nodes and elements and their properties. The element local axes may also be displayed using this command. Three annotation modes are available. You may annotate all entities, annotate selected entities, and erase existing annotations. For performance reasons, it is recommended that the annotations be applied only for objects of interest.

© 2026 ENERCALC, LLC

Clear All Display Options

Quick Access tab > Clear All Display Options is a convenient way to turn off all display options in one step.

Query

The Query command in the Quick Access tab is the same as View > Query.

Hide Selected

The Hide Selected command in the Quick Access tab is the same as View > Hide Selected.

Hide All Except Level

The Hide All Except Level command in the Quick Access tab is the same as View > Hide All Except Level.

Hide All Except Plane

The Hide All Except Plane command in the Quick Access tab is the same as View > Hide All Except Plane.

Show All

The Show All command in the Quick Access tab is the same as View > Show All.

Toggle Grid Display

The Toggle Grid Display command in the Quick Access tab is the same as Create > Grids & Snaps > Toggle Grid Display and View > Drawing Grids > Toggle Grid Display.

Pan

The Pan command in the Quick Access tab is the same as View > Pan.

Zoom

The Zoom command in the Quick Access tab is the same as View > Zoom.

Rotate

The Rotate command in the Quick Access tab is the same as View > Rotate.

Restore Model

The Restore Model command in the Quick Access tab is the same as View > Restore Model.

File

The File ribbon provides commands that are related to files.

© 2026 ENERCALC, LLC

Save Only

When the current window is a model view, File > Save Only saves the model. If the model has not been saved before, the Save As dialog will be displayed prompting you to enter a file name.

Save & Close

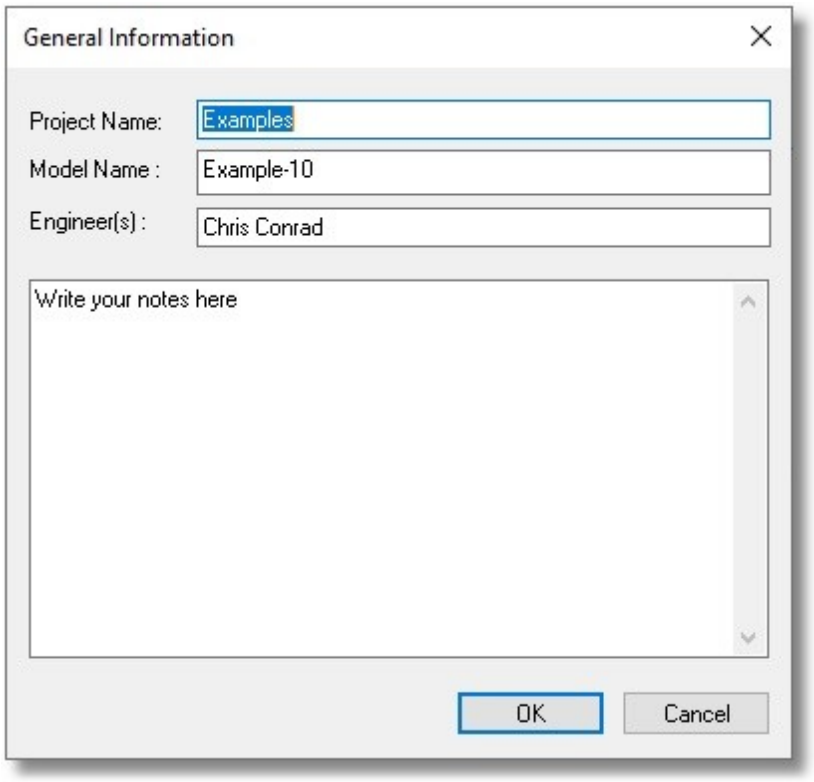
File > Save & Close saves the current model data, closes the model and redisplays the Model Manager window.

Close without Saving

File > Close without Saving provides a way to close the current model and return to the Model Manager without saving the current model.

General Information

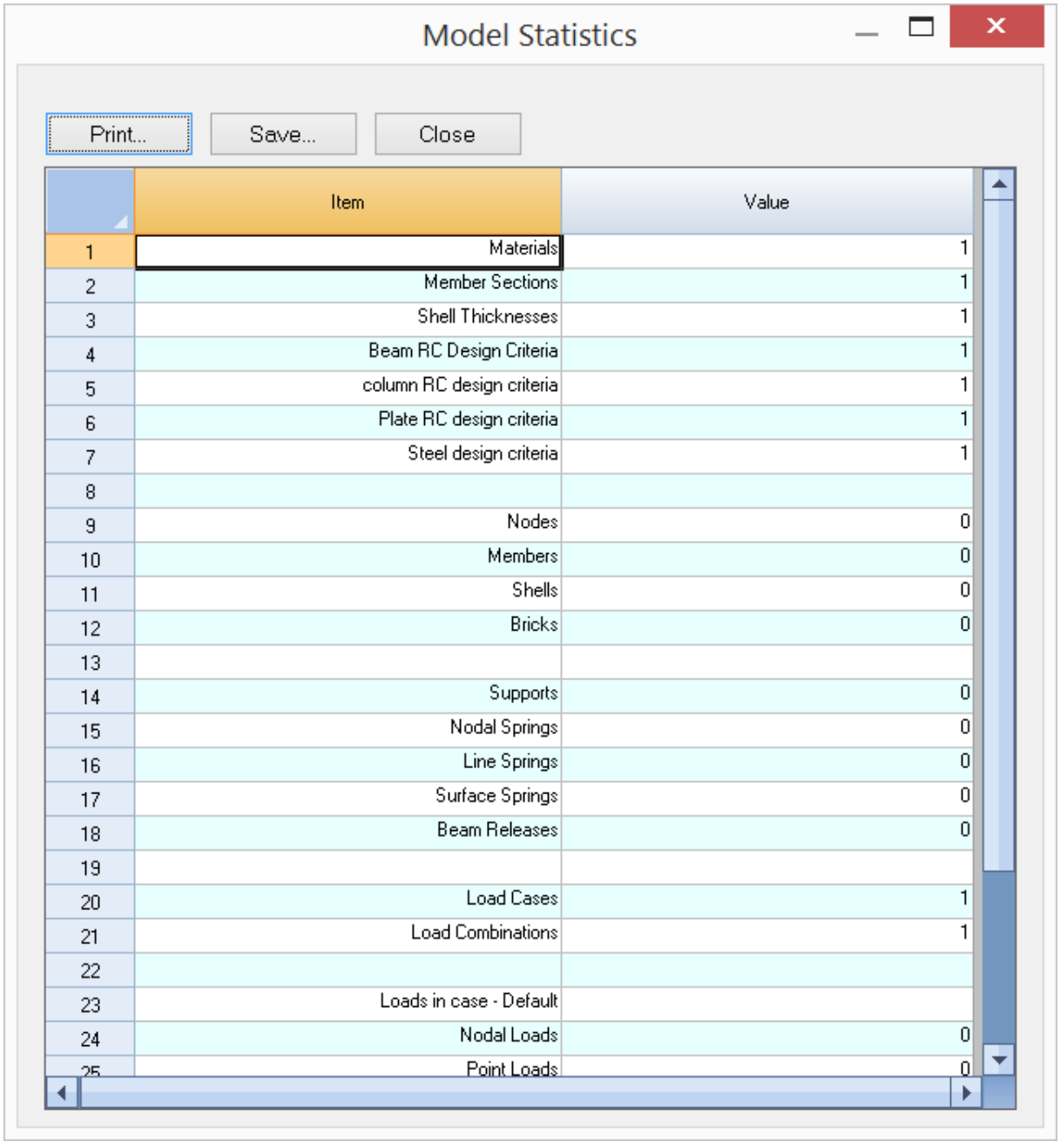
File > General Information prompts you with the following dialog that allows you to specify the Project Name, Model Name, and Engineer:



© 2026 ENERCALC, LLC

Model Statistics

File > Model Statistics displays the key statistics about the model.



	Item	Value
1	Materials	1
2	Member Sections	1
3	Shell Thicknesses	1
4	Beam RC Design Criteria	1
5	column RC design criteria	1
6	Plate RC design criteria	1
7	Steel design criteria	1
8		
9	Nodes	0
10	Members	0
11	Shells	0
12	Bricks	0
13		
14	Supports	0
15	Nodal Springs	0
16	Line Springs	0
17	Surface Springs	0
18	Beam Releases	0
19		
20	Load Cases	1
21	Load Combinations	1
22		
23	Loads in case - Default	
24	Nodal Loads	0
25	Point Loads	0

DXF Options

File > DXF Options allows for the import and export of DXF files.

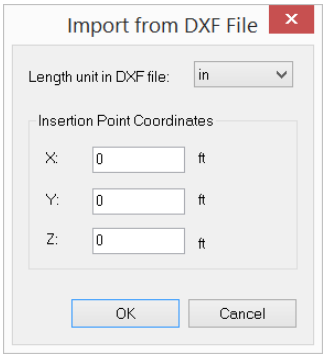
This Model > Import from DXF imports geometry data from a DXF file to the program. DXF stands for Drawing eXchange Format and is widely supported by many CAD and structural analysis programs.

© 2026 ENERCALC, LLC

ENERCALC 3D converts all lines into members and 3D faces to shells. Nodes are created as needed. The command prompts you with a file selection dialog. After selecting a DXF file, you are then prompted with the following dialog (Figure 1.3):

You have the option to select length unit in the DXF file. You may also specify an insertion point at which the imported geometry will be inserted. The program merges nodes and elements automatically after the importing.

You may use this command as many times as needed.



This Model > Export to DXF exports geometry data from the program to a DXF file.

ENERCALC 3D converts all members (beams or trusses) into lines and shells into 3D faces. Exporting bricks is not currently supported. Elements that are currently frozen will not be exported. You should set up the correct length unit from Settings > Units before exporting.

View Log File

File > View Log File allows you to view the log file generated during the solution process.

Create

The Create ribbon provides commands to draw individual nodes and elements based on a drawing grid or existing nodes and parametrically generate models of regular shape.

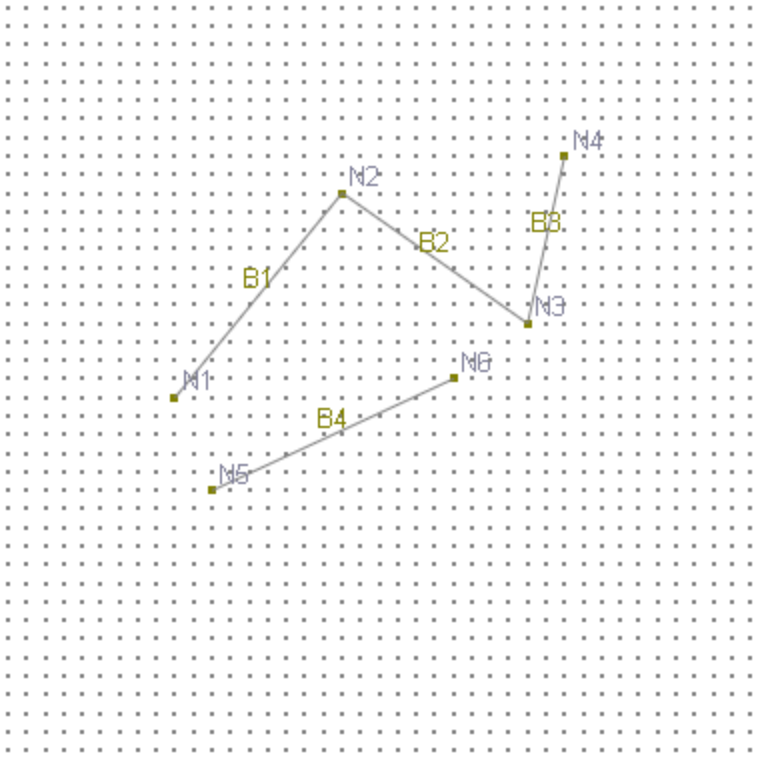
Nodes

Create > Nodes allows you to draw new nodes in the model. To draw a node, simply move the mouse, point to an intersection of the grid and click the left mouse button. You may also draw a node by entering nodal X, Y, and Z (optional) coordinates in the command window via the keyboard. This is very useful if you need to draw nodes outside the grid interactions. The command remains in effect until another command is selected, the right mouse button is clicked, or ESC is pressed.

© 2026 ENERCALC, LLC

Members

Create > Members allows you to draw new members in the model. To draw a member, simply move the mouse and click the left mouse button from point to point.

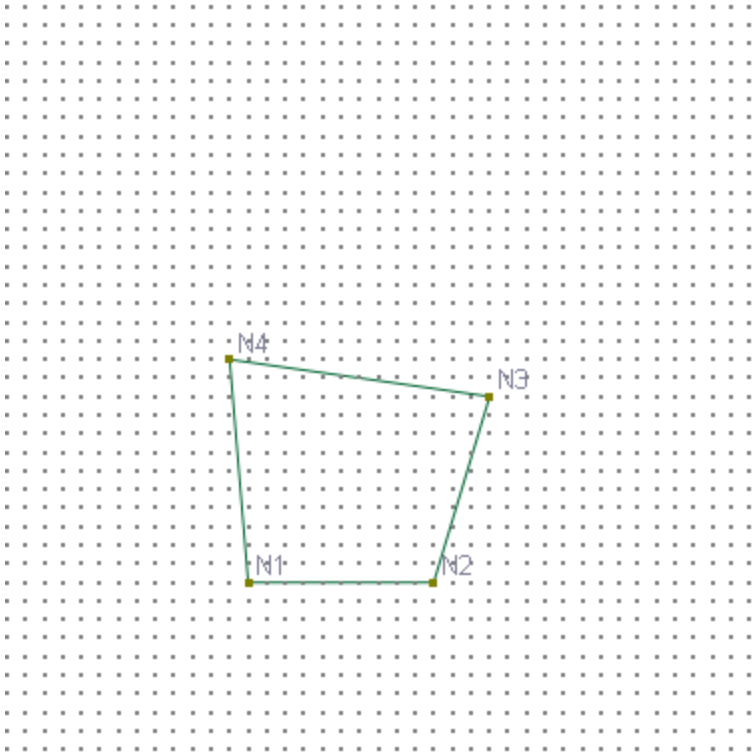


The clicked points must be intersections on the grid or existing nodes. These points become the element nodes. New nodes are created if necessary. Members are drawn continuously. Right clicking the mouse once lets you start drawing members from a new location. Remember, the start and end nodes determine the default local coordinate system. The members drawn have the current section and material properties. You may use the commands in the Modify ribbon to assign appropriate properties to them.

You may also specify a node by entering nodal X, Y and Z (optional) coordinates in the command window via the keyboard. This is very useful if you need to specify nodes outside the grid interactions. In addition, you may specify a node by entering an existing node number directly. You can combine the use of keyboard and mouse to draw members.

© 2026 ENERCALC, LLC

You may turn on annotations for nodes and members while drawing. To do that, use the Display Options buttons on the Quick Access Toolbar. The command remains in effect until another command is selected, the right mouse button is clicked twice, or ESC is pressed.



Shells

Create > Shells allows you to draw new shells in the model. To draw a shell, simply move the mouse and click the left mouse button from point to point. The clicked points must be intersections on the grid or existing nodes. These points become the element nodes. New nodes are created if necessary. Remember, the order of clicked points determines the default local coordinate system. The shells drawn have the current thickness and material properties. You may use the commands in the Create or Modify ribbons to assign appropriate properties to them.

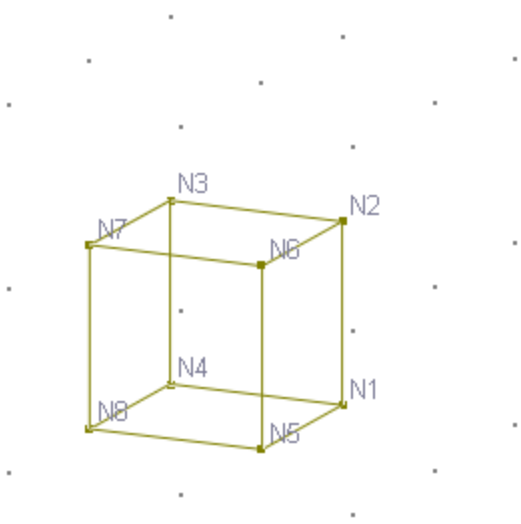
You may also specify a node by entering nodal X, Y, and Z (optional) coordinates in the command window via the keyboard. This is very useful if you need to specify nodes outside the grid interactions. In addition, you may specify a node by entering an existing node number directly. You can combine the use of keyboard and mouse to draw shells.

You may turn on annotations for nodes and shells while drawing. To do that, use the Display Options buttons on the Quick Access Toolbar. The command remains in effect until another command is selected, the right mouse button is clicked, or ESC is pressed.

© 2026 ENERCALC, LLC

Bricks

Create > Bricks allows you to draw new bricks in the model. To draw a brick, simply move the mouse and click the left mouse button from point to point.



The grid must be set up in 3 dimensions. The clicked points must be intersections on the grid or existing nodes. These points become the element nodes. New nodes are created if necessary. Remember, the order of clicked points must be such that the vector of the surface 1-2-3-4 points to the surface 5-6-7-8. The bricks drawn have the current material properties. You may use the commands in the Create or Modify ribbons to assign appropriate properties to them.

You may also specify a node by entering nodal X, Y, and Z coordinates in the command window via the keyboard. This is very useful if you need to specify nodes outside the grid interactions. In addition, you may specify a node by entering an existing node number directly. You can combine the use of keyboard and mouse to draw bricks.

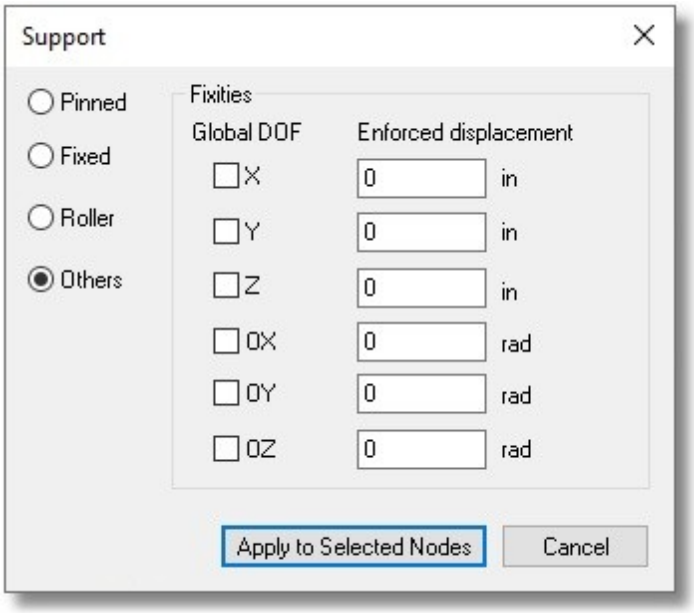
You may turn on annotations for nodes and bricks while drawing. To do that, use the Display Options buttons on the Quick Access Toolbar. The command remains in effect until another command is selected, the right mouse button is clicked, or ESC is pressed.

It is generally not easy to draw bricks in 3 dimensions due to visualization difficulty. You may generate bricks based on shells using the commands such as Modify > Extrude or Revolve. You may also use spreadsheets to input nodes and bricks by running Tables > Nodes or Bricks.

Boundary Conditions

© 2026 ENERCALC, LLC

Create > Boundary Conditions > Support prompts you with the following dialog.

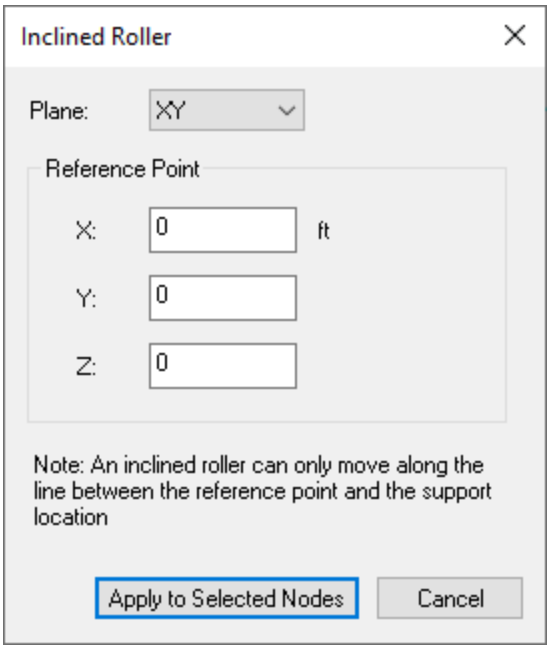


It allows you to assign supports (rigid boundary conditions) to selected nodes in the model. One or more of the six global degrees of freedom (DOFs) may be restrained. In addition, you may specify enforced displacements in the restrained DOFs. The enforced displacements may be used to model support settlements. You may regard them as special loads. For normal supports, enforced displacements in the restrained DOFs are zero. The program provides three commonly used supports, namely, pinned, fixed and roller. In order for support assignments to take place, nodes must be selected beforehand.

After clicking “Apply to Selected Nodes”, you can start to continuously assign supports by window-selecting nodes until you right click the mouse or press the ESC key.

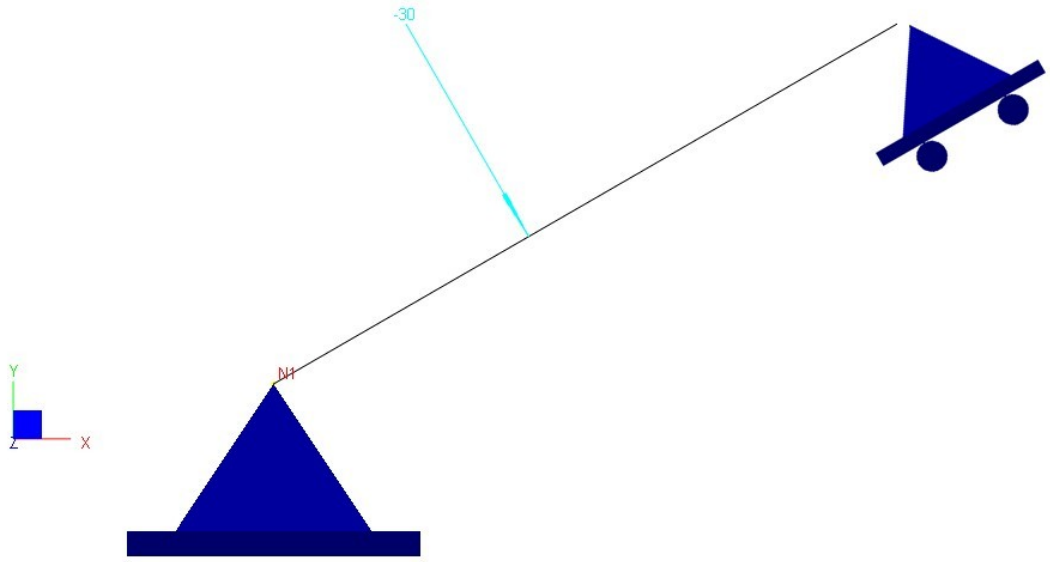
© 2026 ENERCALC, LLC

Create > Boundary Conditions > Inclined Rollers prompts you with the following dialog.



It allows you to define an inclined roller support on XY, YZ or XZ plane.

An inclined roller can only move along the line between the reference point (defined in the dialog) and the support location. For example in the following figure, the roller is located at coordinate (8.0, 5.0, 0) and is inclined 30 degrees from the X-axis.



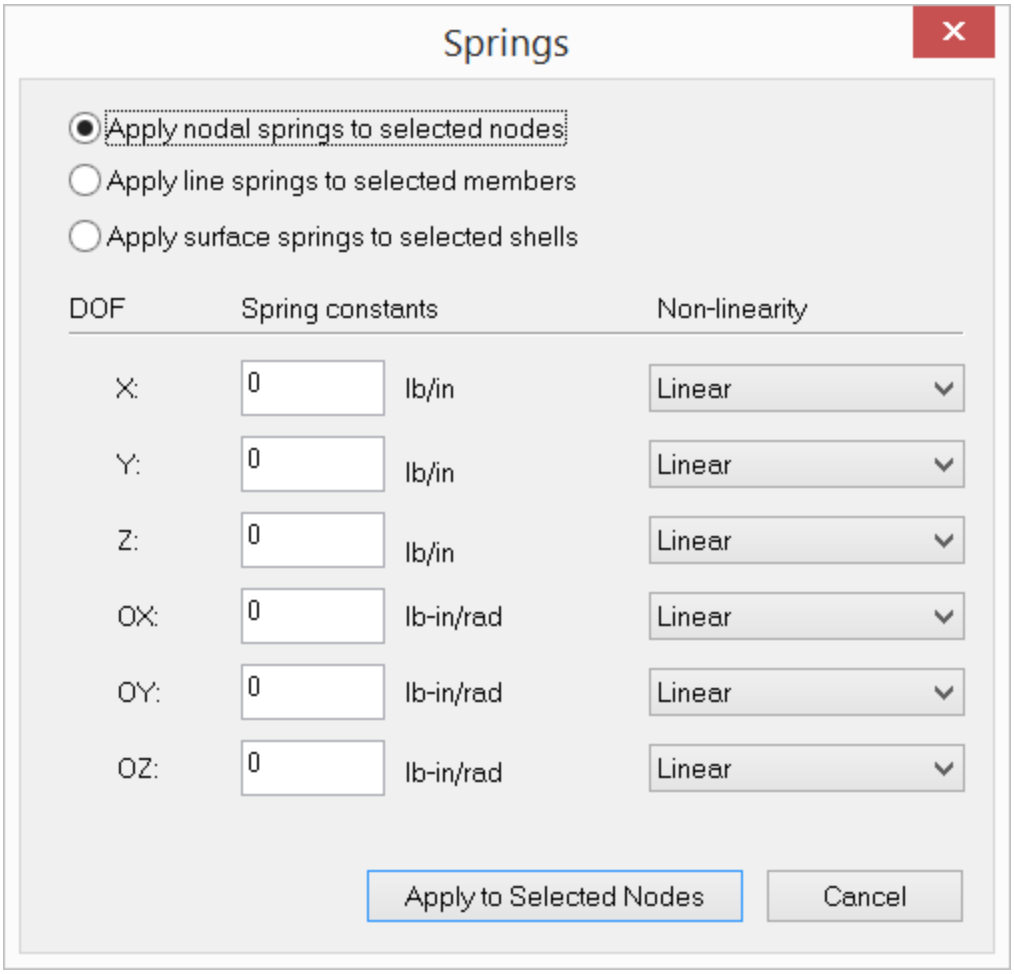
© 2026 ENERCALC, LLC

We can use the reference point (8.0 + 10 * cos30, 5 + 10 * sin30, 0) = (16.666, 10, 0) to constrain the support. An inclined roller is a type of multi-DOF constraint.

A regular support and multi-DOF constraints may be applied on the same node as long as the support/constrained directions do not interfere with each other.

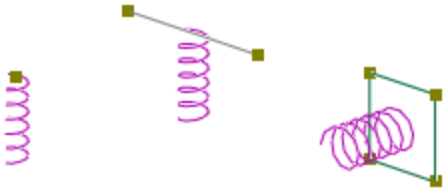
Multi-DOF constraint forces and moments are listed separately from the regular support reactions in the analysis results.

Create > Boundary Conditions > Springs prompts you with the following dialog.



It allows you to assign nodal, line, and surface springs (flexible boundary conditions) to selected nodes, members, or shells in the model.

© 2026 ENERCALC, LLC



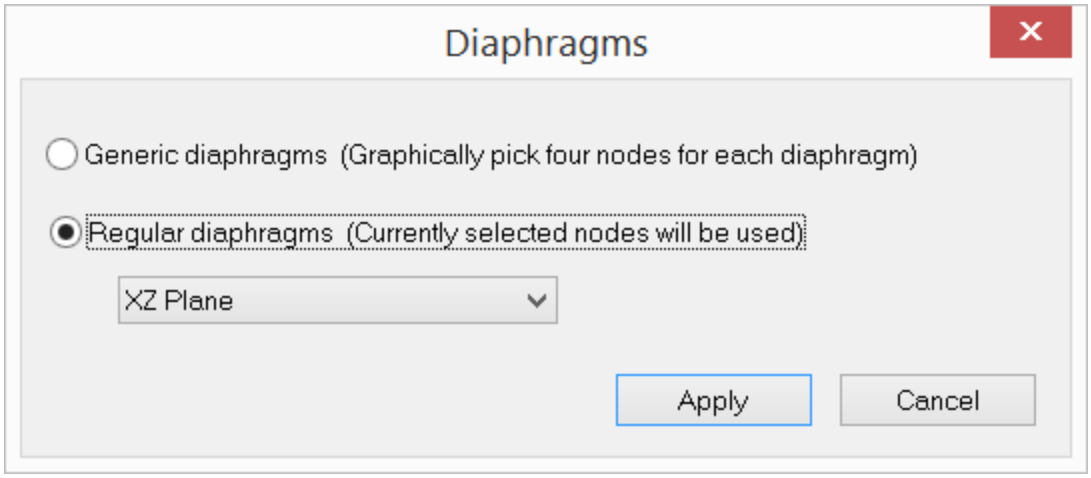
A nodal spring may be restrained in one or more of the six global DOFs (Dx, Dy, Dz, Dox, Doy and Doz). A line or surface spring may be restrained in one or more of the three global translational DOFs (Dx, Dy and Dz). To qualify to be a valid flexible restraint, the corresponding spring constant must be specified.

A restraint may be designated as linear, compression-only or tension only. A compression-only restraint is active only when the nodal displacement in the restrained direction is negative. A tension-only restraint is active only when the nodal displacement in the restrained direction is positive. The presence of tension only or compression only springs makes the model nonlinear and requires iterative solution for each load combination.

After clicking “Assign”, you can start to continuously assign nodal, line or surface springs by window-selecting nodes, members or shells until you right click the mouse or press the ESC key.

Constraints

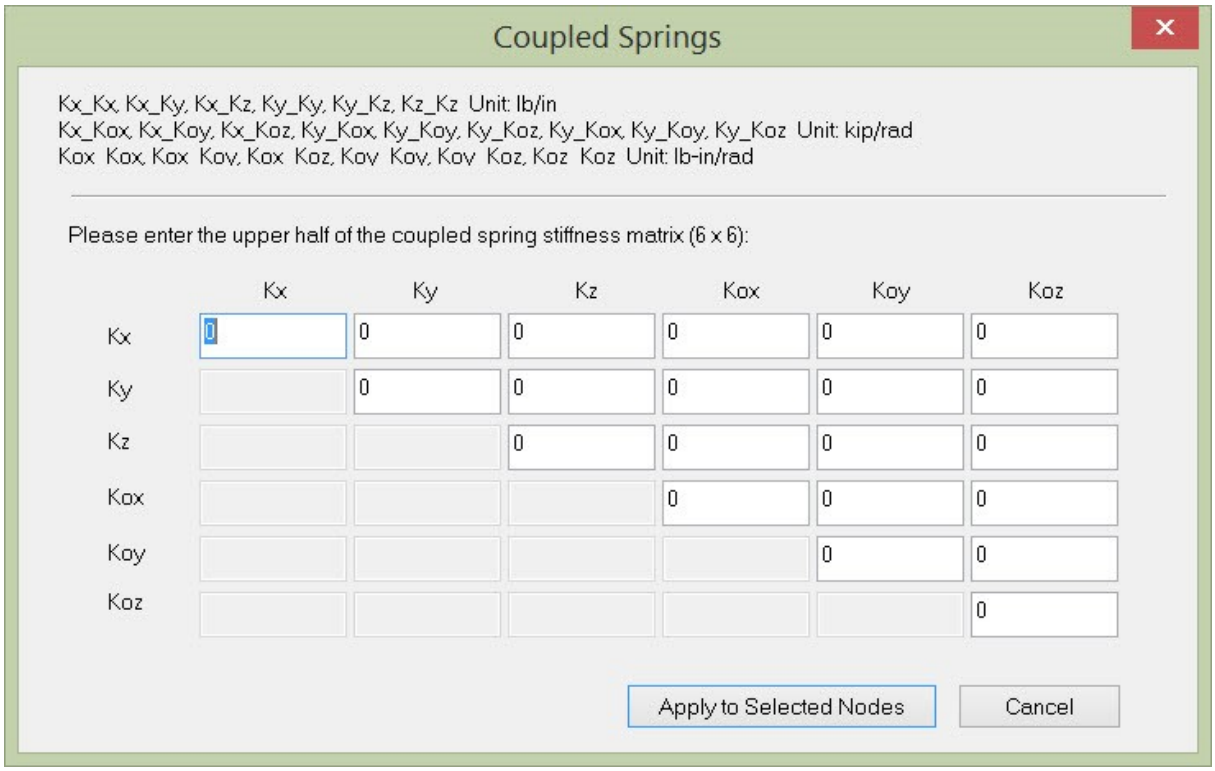
Create > Constraints > Diaphragms prompts you with the following dialog.



It allows you to define regular or generic rigid diaphragms (in-plane) in a 3D model. For example, to model horizontal concrete floors, you may select one node on each floor and apply regular diaphragms to the selected nodes in XZ plane (with normal in the global Y direction). Instead of using plate elements, rigid diaphragms allow you to model stiff in-plane actions quickly. The program further provides the option to ignore the rigid diaphragm actions as an analysis option (Analysis > Analysis Options).

© 2026 ENERCALC, LLC

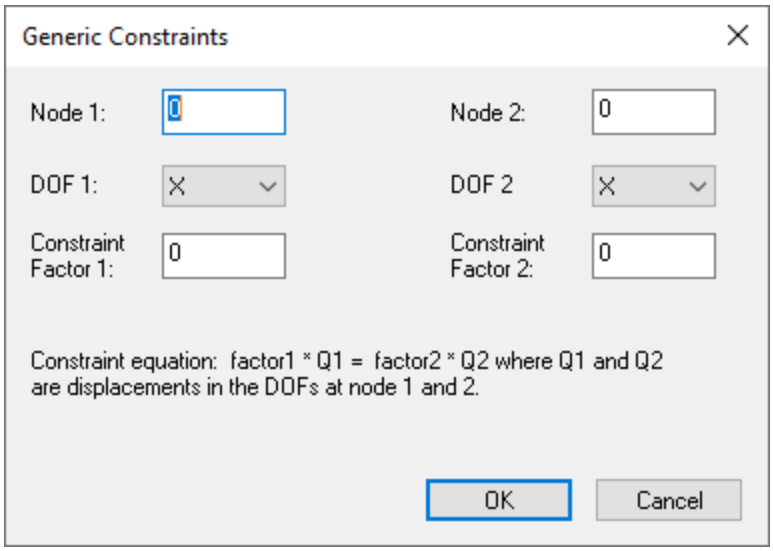
Create > Constraints > Coupled Springs prompts you with the following dialog.



It allows you to assign coupled springs to selected nodes. Coupled springs are useful in modeling/simplifying substructures such as bridge foundations. It is important to enter the coupled spring stiffness matrix in the appropriate units.

© 2026 ENERCALC, LLC

Create > Constraints > Generic Constraints prompts you with the following dialog.



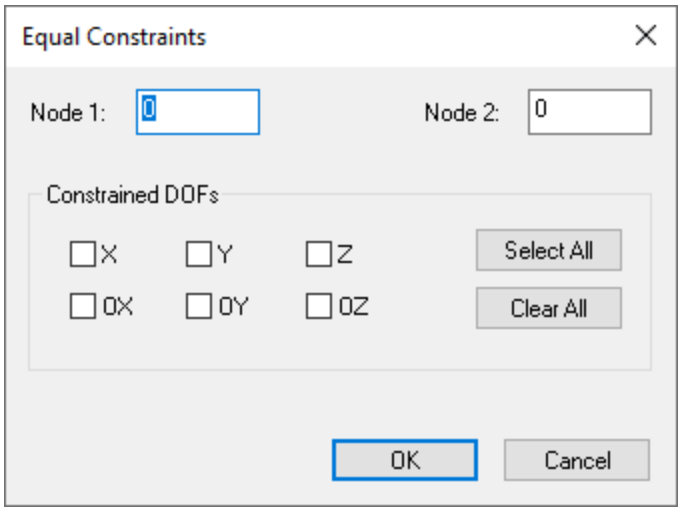
It allows you to define a generic constraint at one or two nodes. If the constraint is applied to the same node, the constraint DOFs must be different. Constrained DOFs must be compatible: Q1 and Q2 must be both translational or rotational. Constraint factors must be non-zero.

A regular support and multi-DOF constraints may be applied on the same node as long as the support/constrained directions do not interfere with each other.

Multi-DOF constraint forces and moments are listed separately from the regular support reactions in the analysis results.

© 2026 ENERCALC, LLC

Create > Constraints > Equal Constraints prompts you with the following dialog.



It allows you to define a generic constraint at one or two nodes. An equal displacement constraint is one type of multi-DOF constraint.

A regular support and multi-DOF constraints may be applied on the same node as long as the support/constrained directions do not interfere with each other.

Multi-DOF constraint forces and moments are listed separately from the regular support reactions in the analysis results.

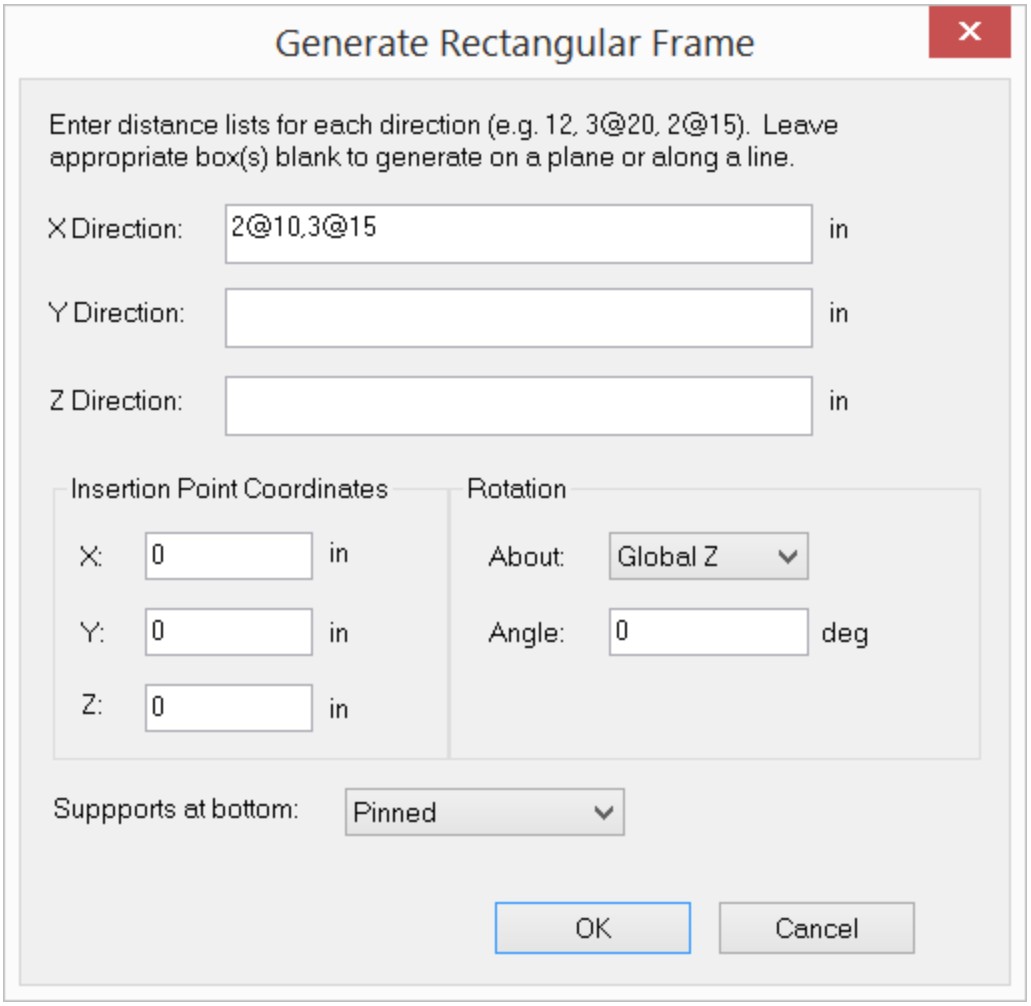
Model Generators

Create > Model Generators provides commands to quickly generate commonly used structural components in a model. These commands may be used multiple times to generate different parts in the model.

Create > Model Generators > 2D Truss/Frame provides a variety of common truss and frame templates for quick generation of these common structural assemblies.

© 2026 ENERCALC, LLC

Create > Model Generators > Rectangular Frames prompts you with the following dialog.

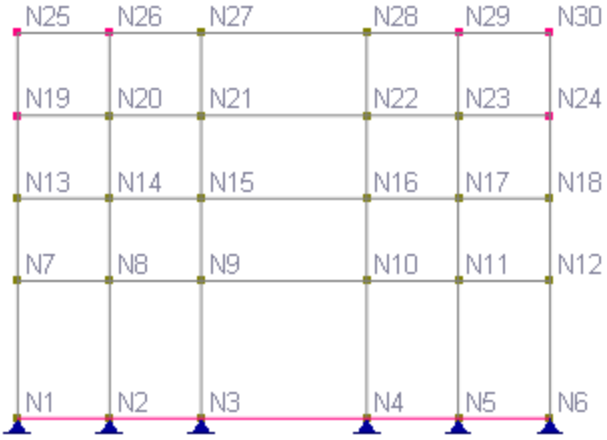


It allows you to quickly generate 1D frames (continuous beams), 2D frames (plane frame or grillage) or 3D frames (space frames). The distance list is a comma separated list that specifies multiple distances. For example, a distance list of “12, 2@14, 3@10” will generate distances of 12, 14, 14, 10, 10 and 10 in length unit. You may leave appropriate distance list(s) blank to generate on a plane or along a line. You may specify pinned or fixed supports at the bottom. The generated members have the default section and material properties. You may assign them appropriate properties using commands in the Modify ribbon.

© 2026 ENERCALC, LLC

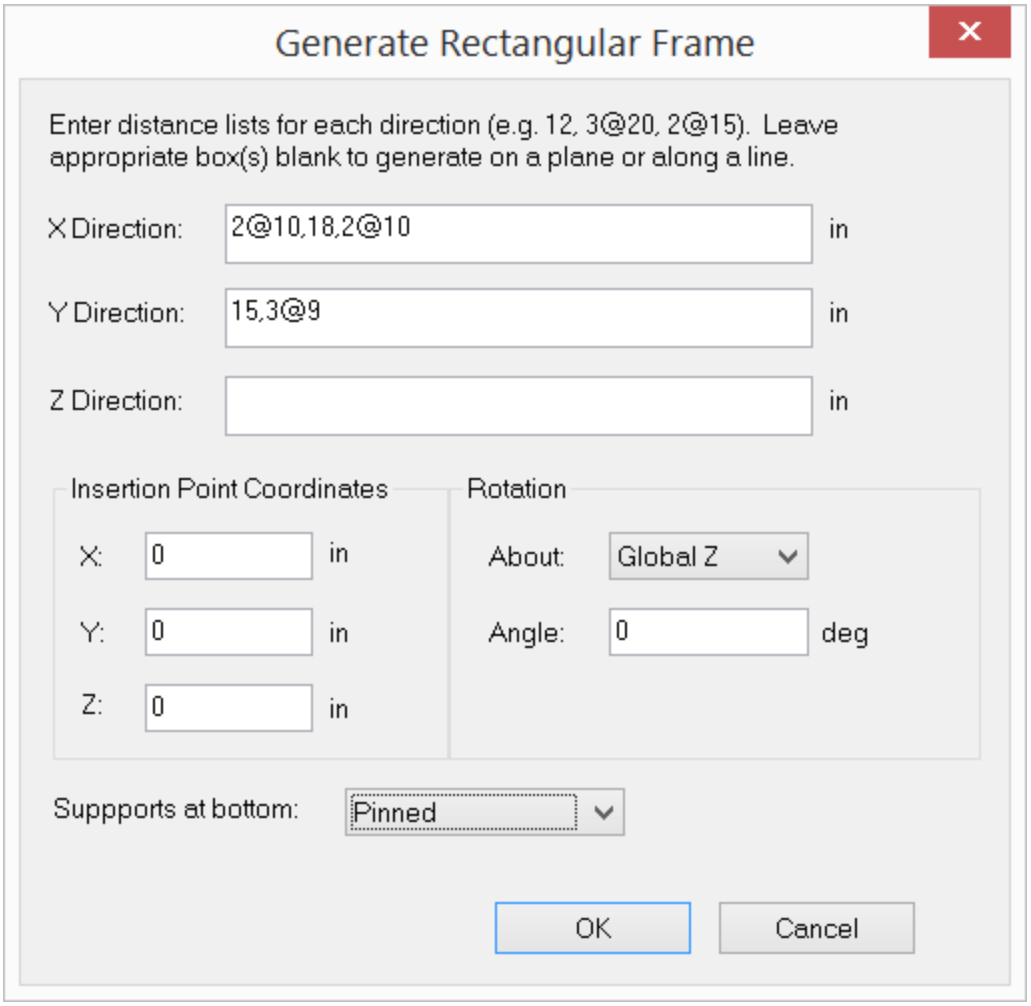
The following three examples show the uses of this command. The first example is a continuous beam in the X direction generated using the input from the dialog above. The first two spans are of 10 ft and the last three spans are of 15 ft. The pinned supports are also generated automatically.

The second example is a 2D frame on the XY plane with horizontal spans 10, 10, 18, 10, 10 ft and vertical spans 15, 8, 8, 8 ft as shown in the following.



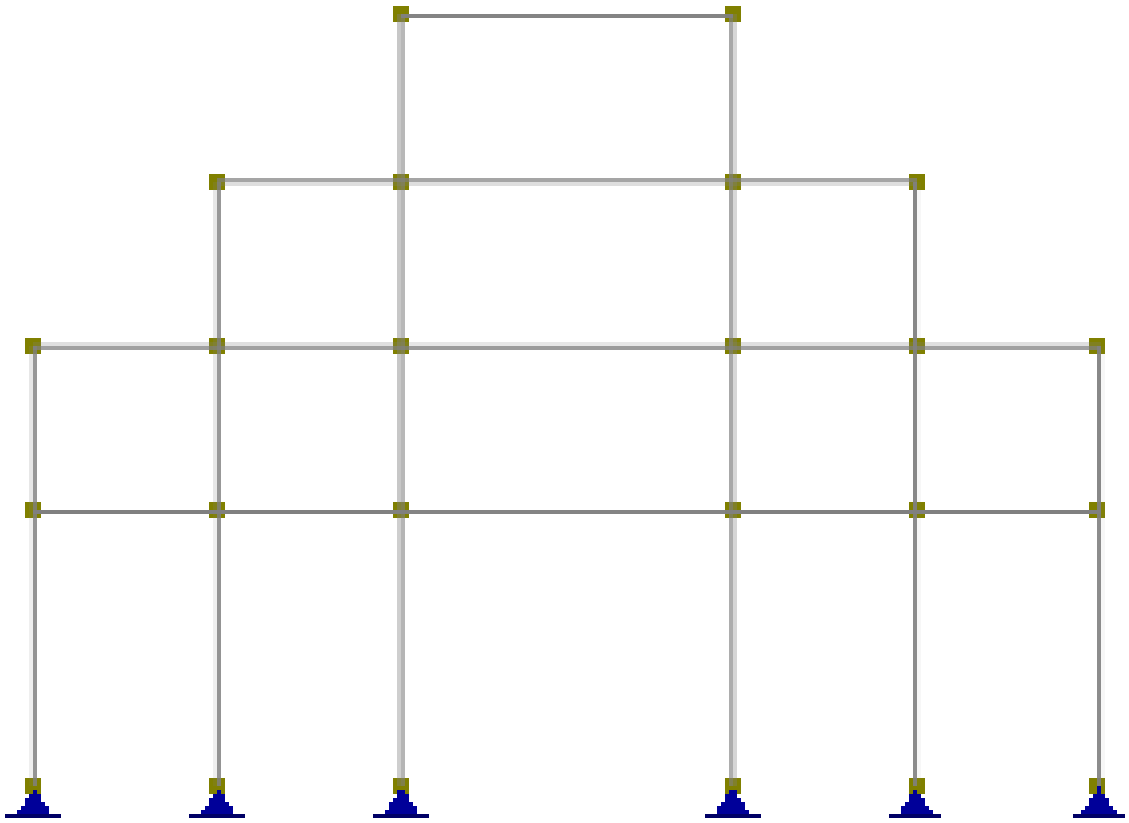
© 2026 ENERCALC, LLC

A rectangular frame is first generated using the input from the following dialog.



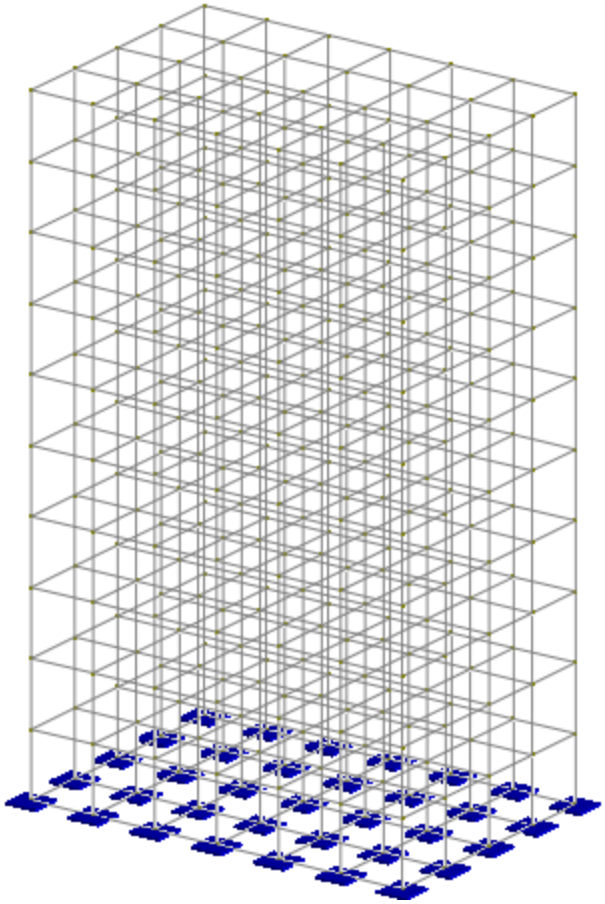
© 2026 ENERCALC, LLC

The frame is then modified by selecting and deleting nodes 19, 24, 25, 26, 29, 30 and horizontal members at the bottom. Notice when a node is deleted, elements (and their dependents such as loads) connected to that node are automatically deleted also.



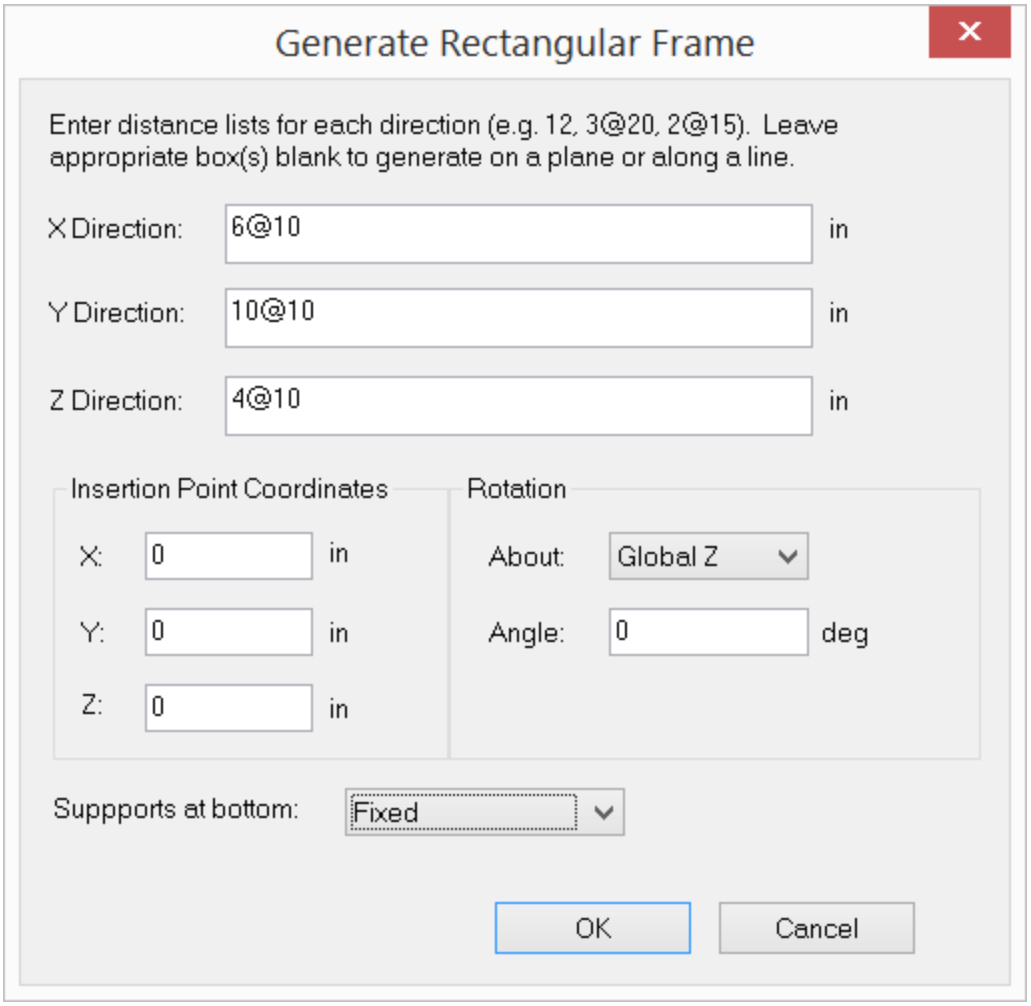
© 2026 ENERCALC, LLC

The third example is a 3D frame with 6 spans in the X direction, 4 spans in the Z direction, and 10 spans in the Y direction. All spans are 10 ft. The frame is fixed at the bottom.



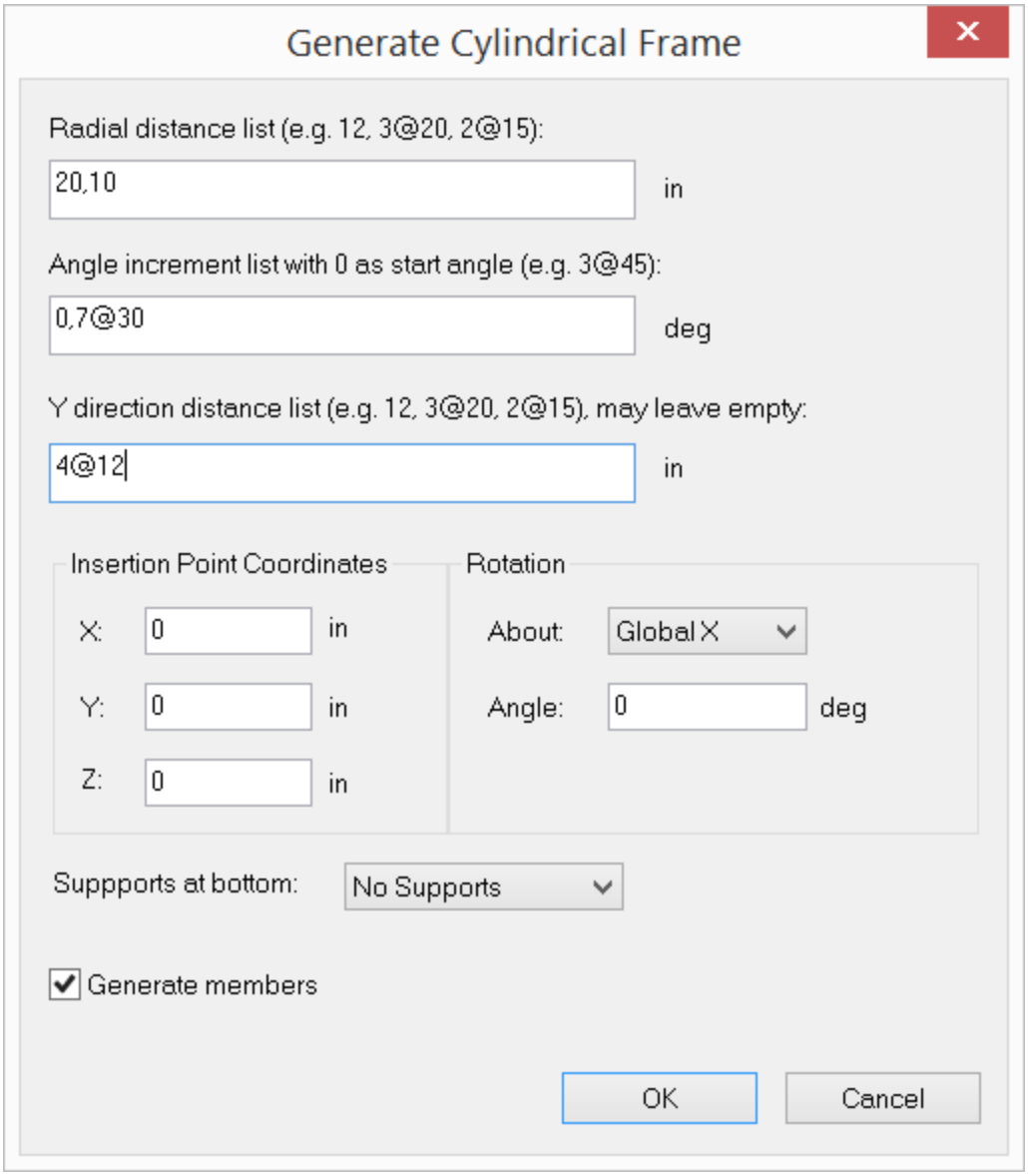
© 2026 ENERCALC, LLC

It is generated using the following input.



© 2026 ENERCALC, LLC

Create > Model Generators > Cylindrical Frames prompts you with the following dialog.

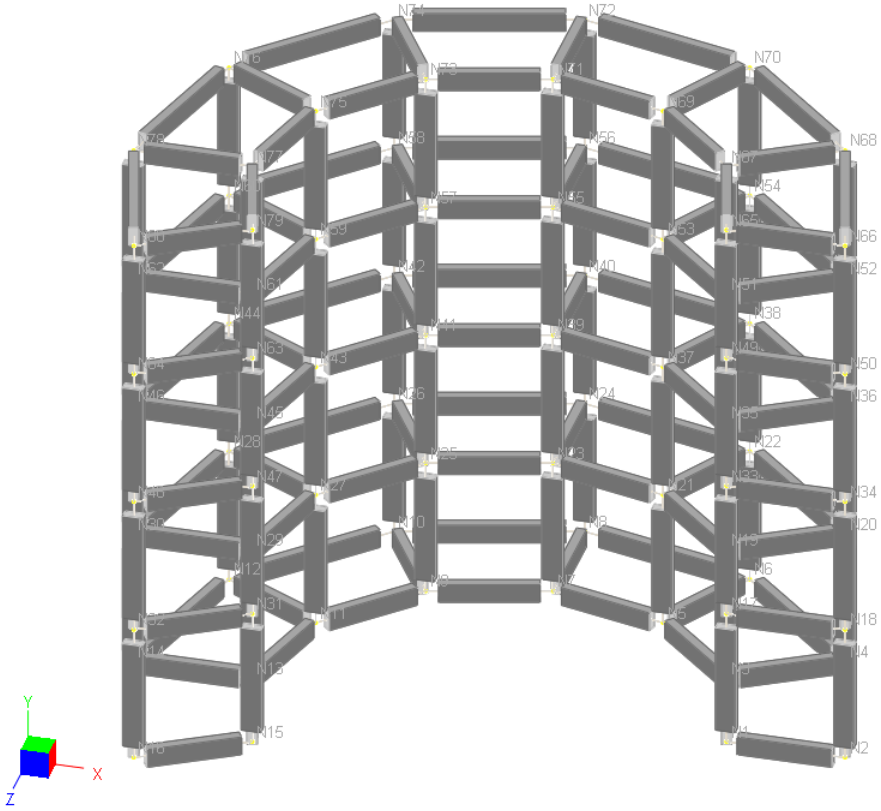


It allows you to quickly generate 2D or 3D cylindrical frames. The distance list is a comma separated list that specifies multiple distances. For example, a distance list of “12, 2@14, 3@10” will generate distances of 12, 14, 14, 10, 10 and 10 in length unit. You may leave the Y direction distance list empty, in which case, a plane cylindrical frame will be generated. You may specify pinned or fixed supports at the bottom. The generated members have the default section and material properties. You may assign them appropriate properties using commands in the Modify ribbon.

You have the option not to generate members. In this way, you can generate nodes on cylindrical system first. Then you may use the command Create > Shells (Bricks) by Nodes to generate a system of shell (brick) elements.

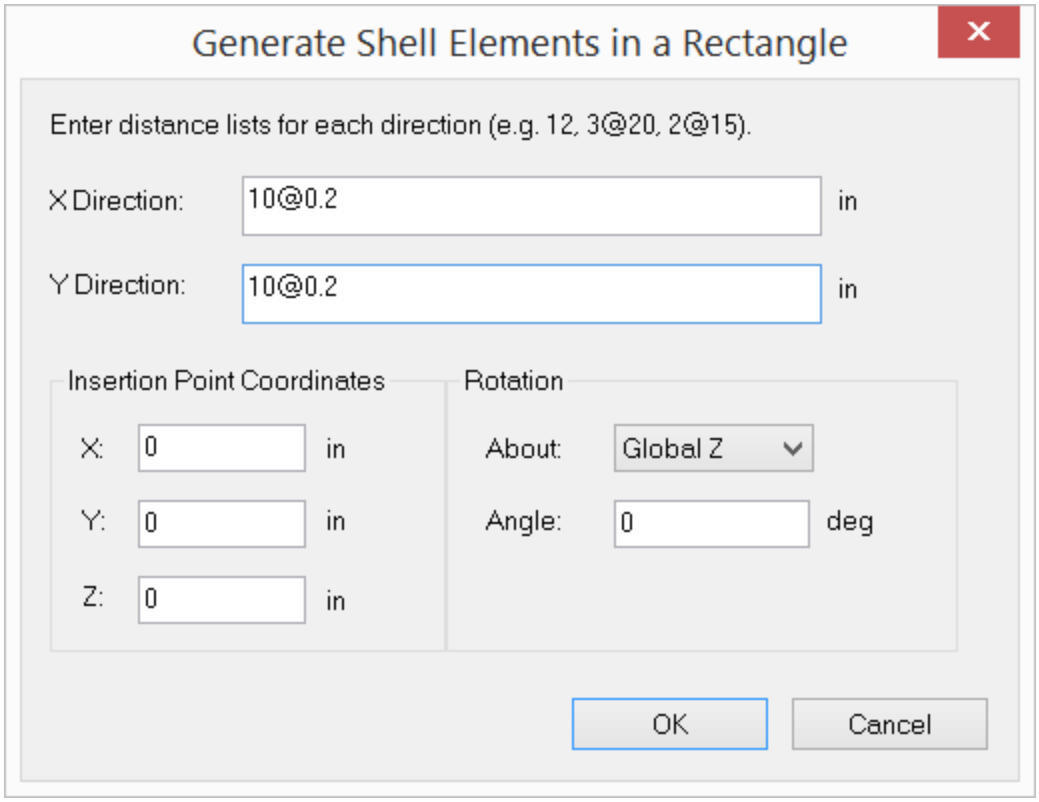
© 2026 ENERCALC, LLC

Using the input from the screen capture above, the following 3D cylindrical frame was generated. You may need to set the element local angles for columns for correct orientation.



© 2026 ENERCALC, LLC

Create > Model Generators > Rectangular Shells prompts you with the following dialog.

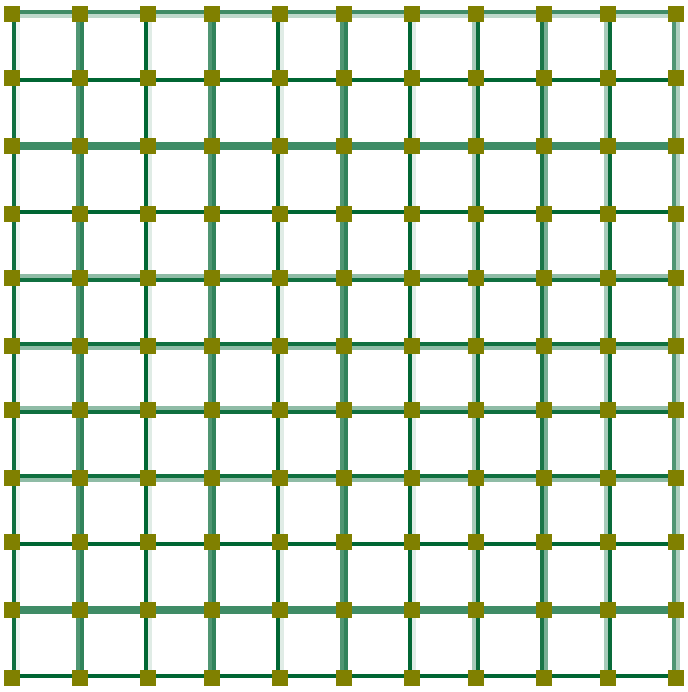


It allows you to quickly generate shells in a rectangle. The distance list is a comma separated list that specifies multiple distances. For example, a distance list of "12,2@14,3@10" will generate distances of 12, 14, 14, 10, 10 and 10 in length units. You may specify an insertion point to translate and rotation parameters to rotate the generated shells. The generated shells have the default thickness and material properties. You may assign them appropriate properties using commands in the Modify ribbon.

This command can only generate shells in a rectangle. To generate shells in a general quadrilateral, you may first define one quadrilateral shell and then use the command Modify > Sub-Mesh Shells to sub-divide it.

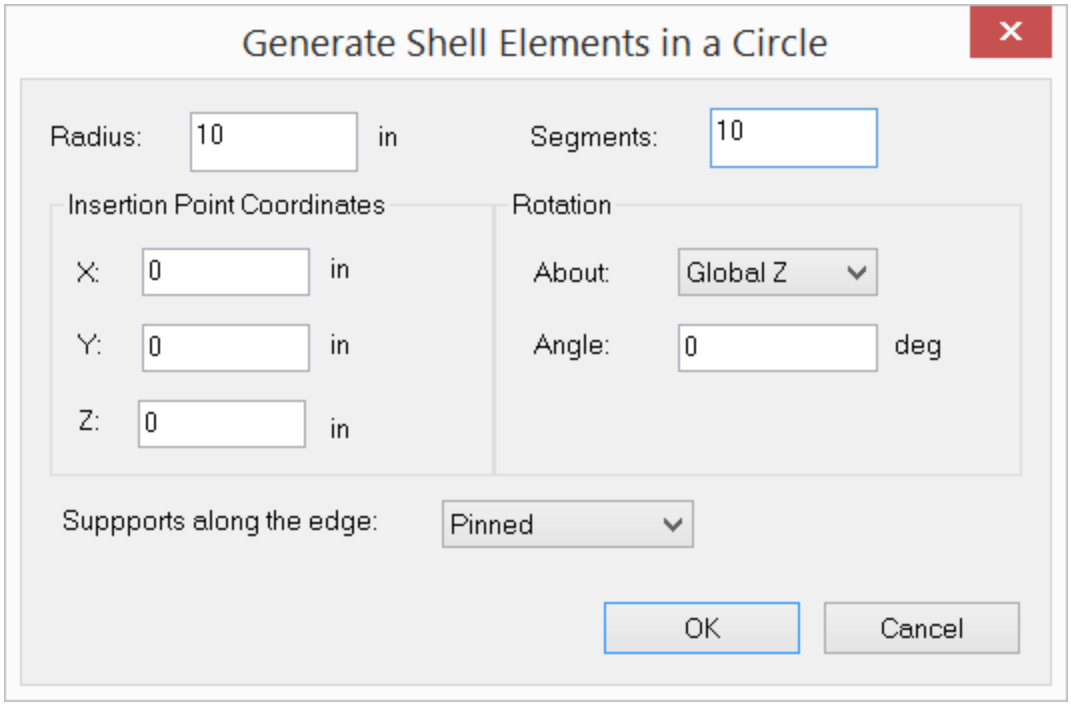
© 2026 ENERCALC, LLC

The following example shows a 10x10 rectangular mesh of shells generated using the input from the screen capture above.



© 2026 ENERCALC, LLC

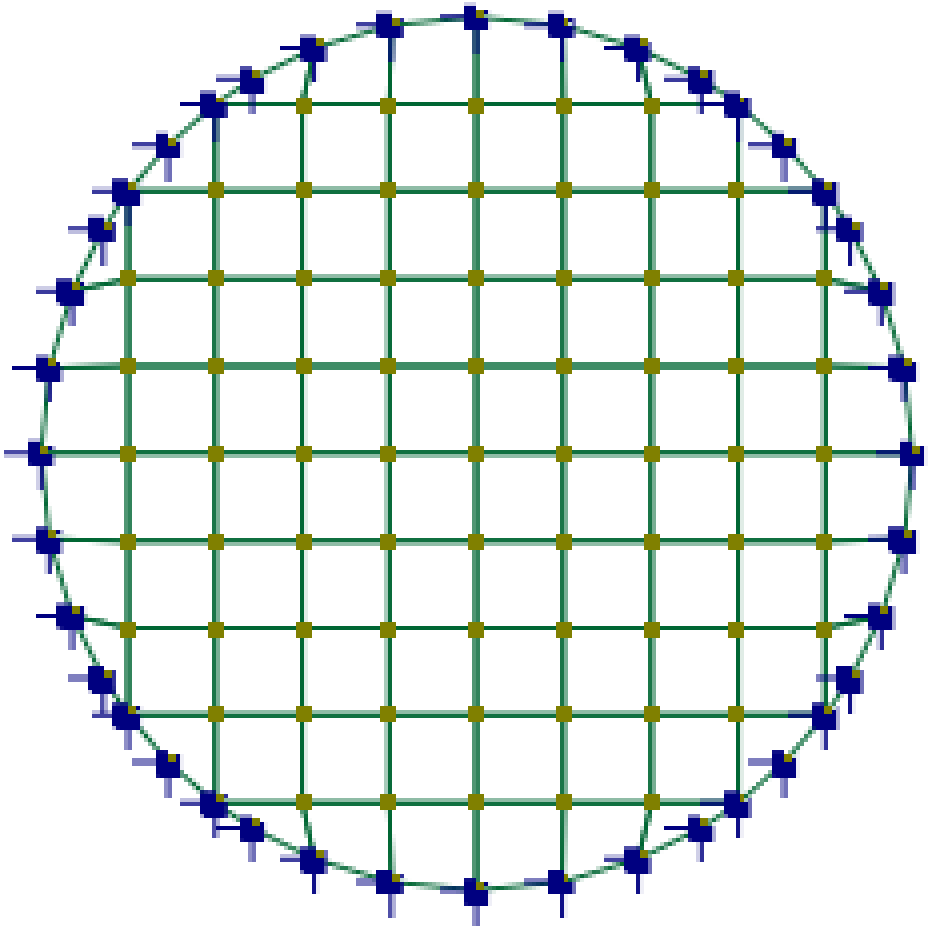
Create > Model Generators > Circular Shells prompts you with the following dialog.



It allows you to quickly generate shells in a circle. You may specify the number of segments to control the fineness of the mesh. Generally speaking, a relatively fine mesh is recommended to minimize the discretization error along the curved edge. The generated shells are mostly rectangular in shape, with some general quadrilaterals along the edge. You should not use rectangular thin plate formulation in the analysis. You may specify an insertion point to translate and rotation parameters to rotate the generated shells. The generated shells have the default thickness and material properties. You may assign them appropriate properties using commands in the Modify ribbon.

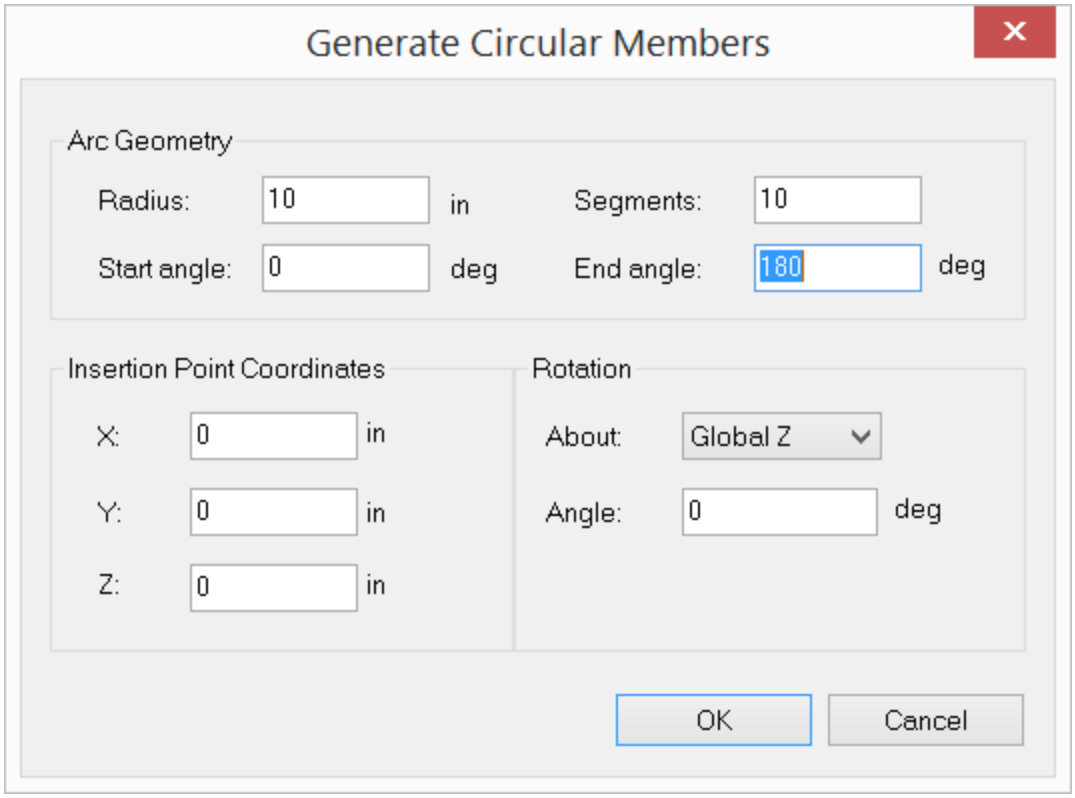
© 2026 ENERCALC, LLC

The following example shows shells generated in a circle using the input from the screen capture above.



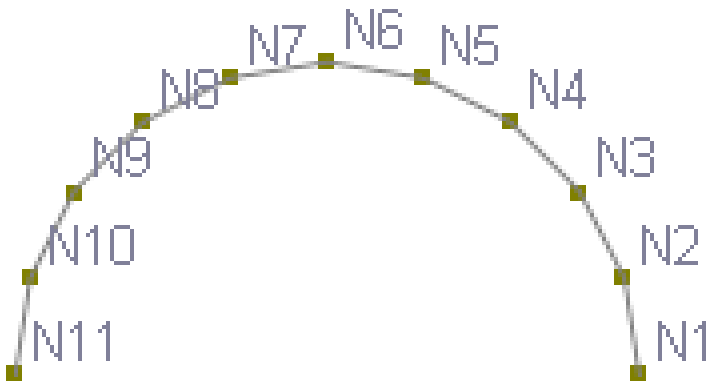
© 2026 ENERCALC, LLC

Create > Model Generators > Arc Members prompts you with the following dialog.



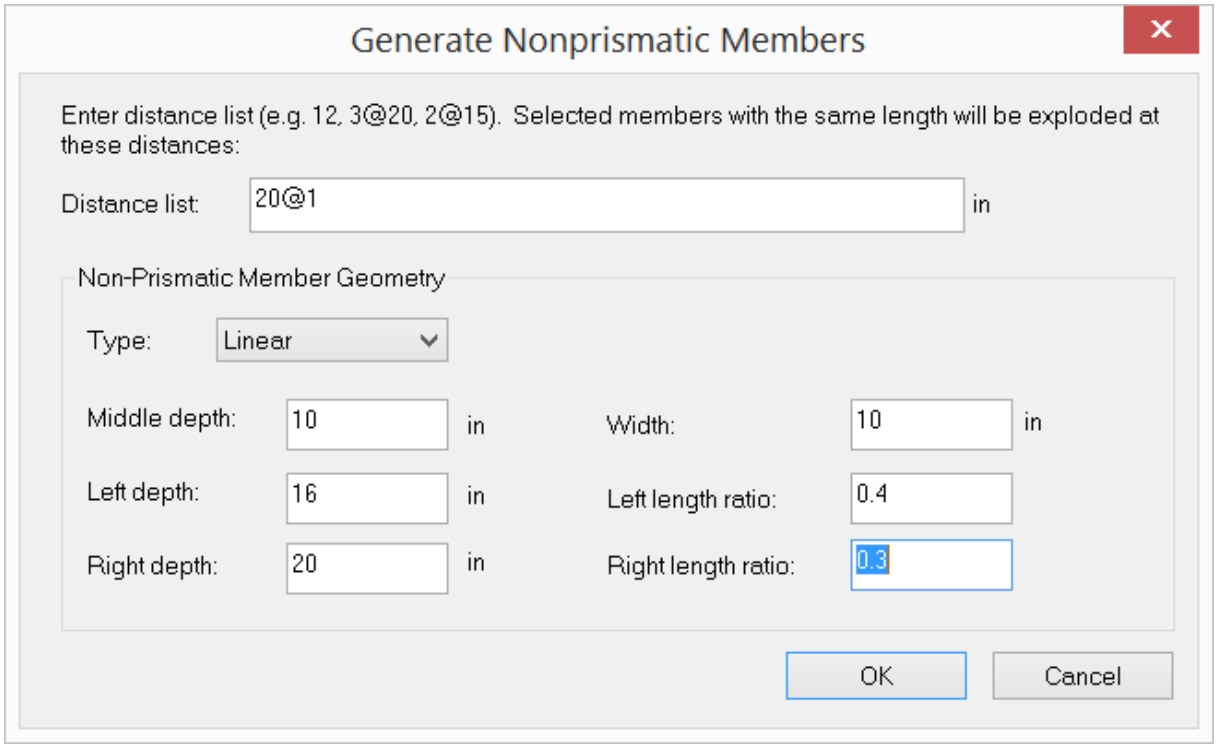
It allows you to quickly generate members along an arc. You may specify an arc radius, the start and end angles, and the number of segments. You may specify an insertion point to translate and rotation parameters to rotate the generate shells. The generated members have the default section and material properties. You may assign them appropriate properties using commands in the Modify ribbon.

The following example shows members generated along an arc using the input from the screen capture above.



© 2026 ENERCALC, LLC

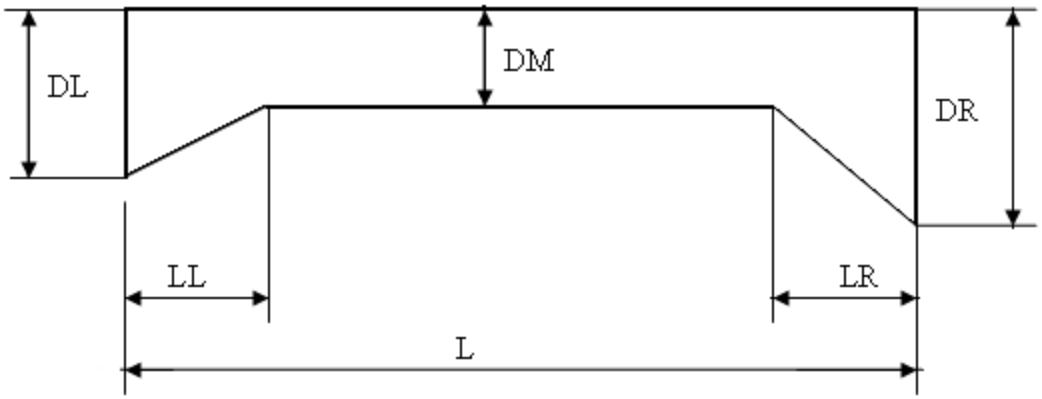
Create > Model Generators > Non-Prismatic Members prompts you with the following dialog.



It allows you to quickly convert each of the selected prismatic members into multiple prismatic members to approximate a non-prismatic member. The distance list is a comma separated list that specifies multiple distances. For example, a distance list of “12, 2@14, 3@10” will generate distances of 12, 14, 14, 10, 10 and 10 in length units. The lengths of the selected prismatic members must be consistent with the distance list.

© 2026 ENERCALC, LLC

The left and right haunches of the non-prismatic members may be of type linear, parabolic or straight. You must define the geometry including middle depth (DM), left depth (DL), right depth (DR), width, left length ratio (LL / L), right length ratio (LR / L). Each of the selected prismatic members will be exploded into multiple prismatic members to approximate the non-prismatic member's behavior. Appropriate member sections will be automatically added and assigned in the model. Existing loads on the selected prismatic members will be assigned to the new members appropriately.



Create > Model Generators > Rectangular Tank provides a quick way to model an open-topped rectangular tank with parametric input.

Entities from other Entities

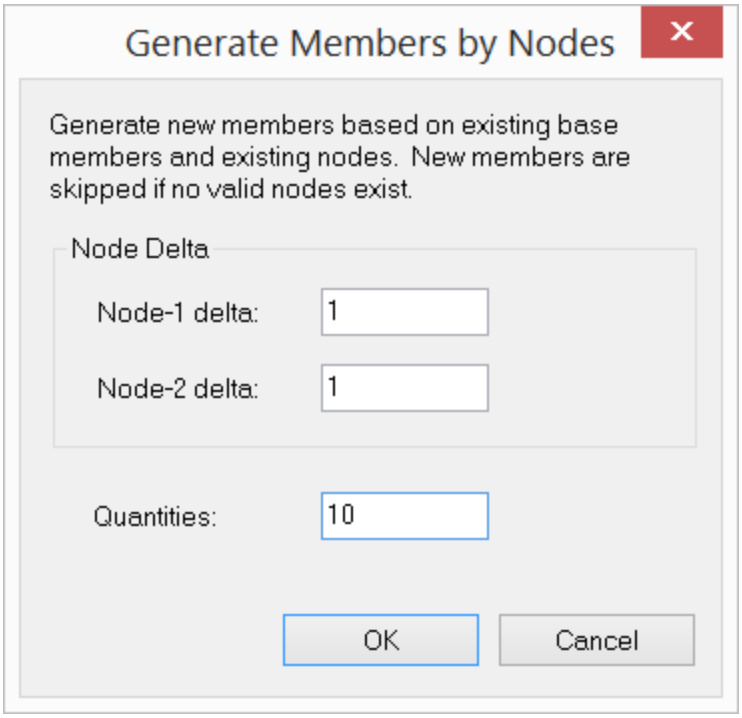
Create > Entities from other Entities provides commands to quickly generate elements using other elements that already exist in a model.

Create > Entities from other Entities > Nodes from Grid will convert all grid points to nodes in one sweep. You may use this command as many times as you like. You must merge nodes manually if necessary. The generated nodes may then be used to generate members, shells, or bricks using the following three commands.

© 2026 ENERCALC, LLC

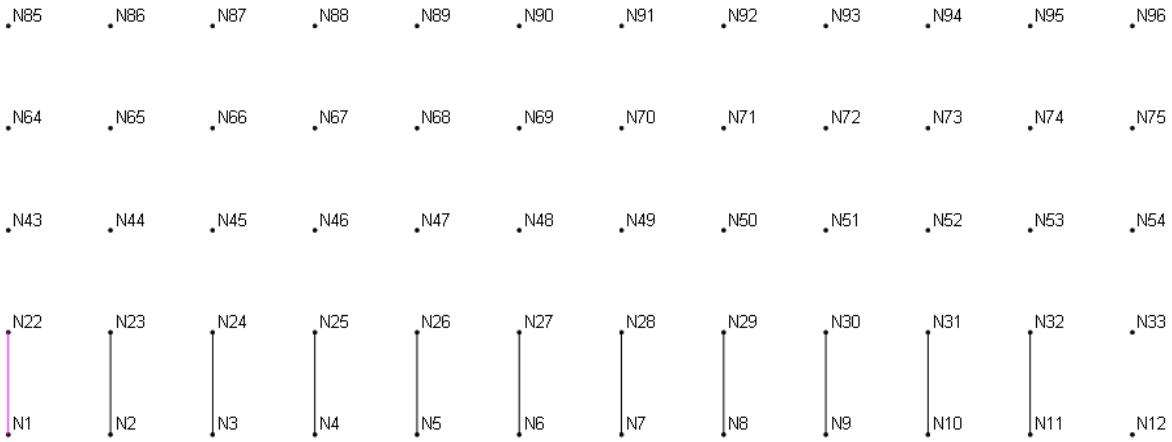
Create > Entities from other Entities > Members by Nodes allows you to quickly generate new members based on selected base members and existing nodes. New members are skipped if no valid nodes exist. The following five figures show how this command may be used.

Given one selected member and existing nodes in the screen capture above and using the input in the figure below:

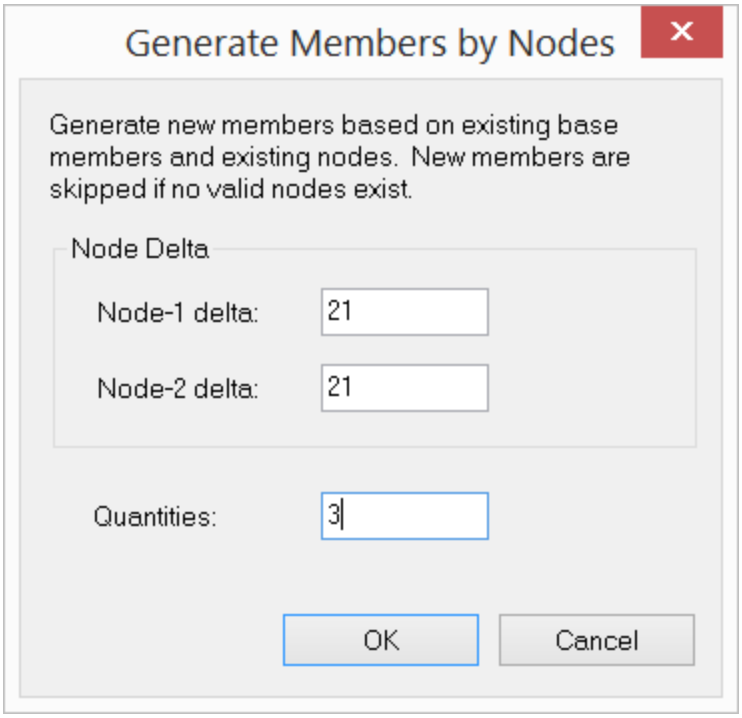


© 2026 ENERCALC, LLC

10 new members in the figure below are generated:

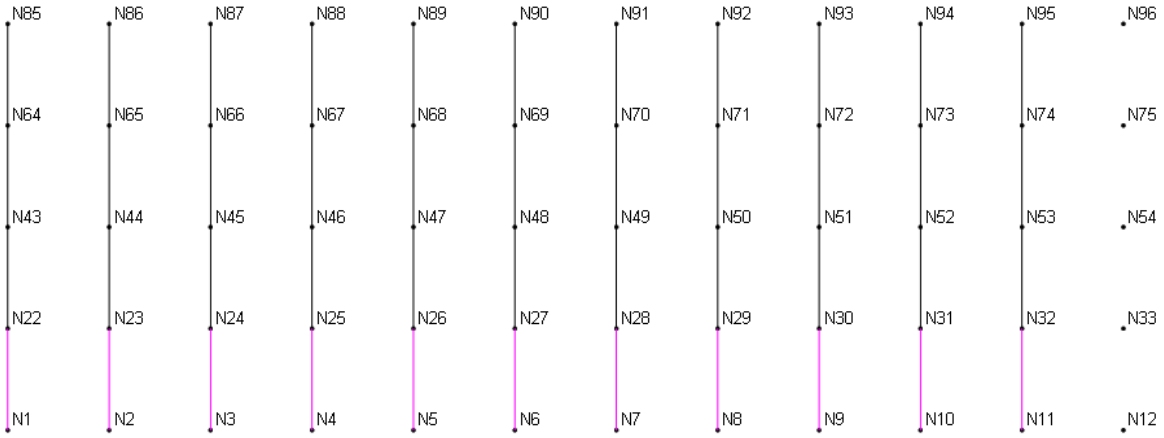


By selecting all 11 members in the screen capture above and using the input in the following figure:



© 2026 ENERCALC, LLC

33 more members are generated in the following figure:



Create > Entities from other Entities > Shells by Nodes allows you to quickly generate new shells based on selected base shells and existing nodes. New shells are skipped if no valid nodes exist. The concept in this command is similar to Generate > Members by Nodes.

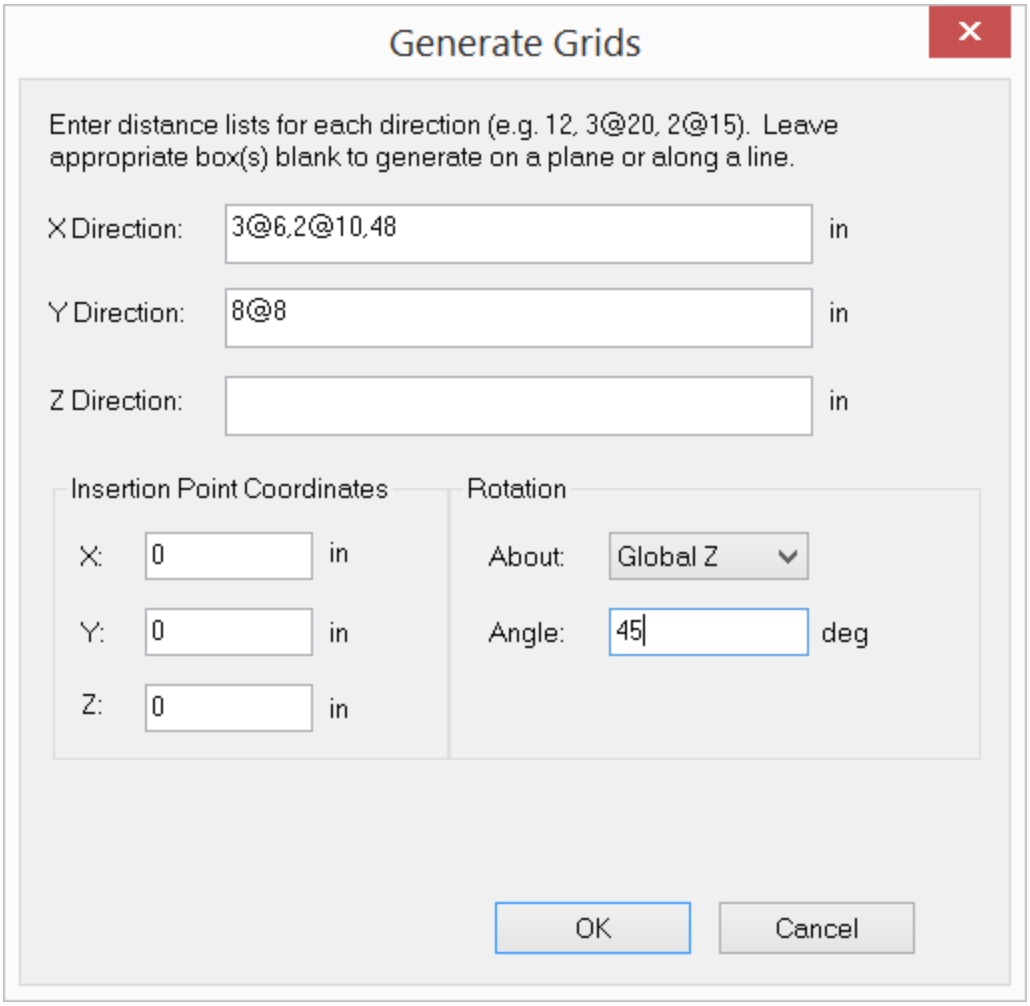
Create > Entities from other Entities > Bricks by Nodes allows you to quickly generate new bricks based on selected base bricks and existing nodes. New bricks are skipped if no valid nodes exist. The concept in this command is similar to Generate > Members by Nodes.

Grids & Snaps

Use Create > Grids & Snaps to create dimensional control for precision drawing.

© 2026 ENERCALC, LLC

Create > Grids & Snaps > Drawing Grid Setup offers the controls to configure the grid used for snapping and dimensional control. The command prompts you with the following dialog.

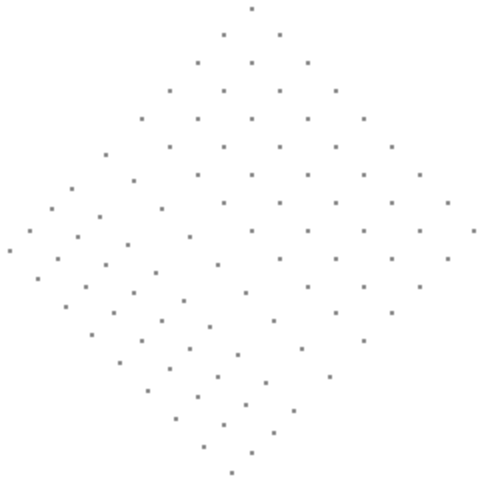


It allows you to generate a 1D, 2D or 3D rectangular grid for drawing or guidance. The distance list is a comma separated list that specifies multiple distances. For example, a distance list of “12, 2@14, 3@10” will generate distances of 12, 14, 14, 10, 10 and 10 in length units. You may specify a distance list for the X, Y, or Z direction or any combination of them.

You may specify an insertion point to translate and rotation parameters to rotate the grid. The drawing grid may be turned on or off by running the command View > Grids & Snaps > Drawing Grid or by simply pressing F7. You may regard the grid as a user defined coordinate system that can be changed at any time. The coordinates of the grid intersection under the mouse are displayed in the status bar. It helps you to identify correct points when drawing nodes or elements.

© 2026 ENERCALC, LLC

The following example shows the use of this command.



Create > Grids & Snaps > Toggle Grid Display turns the grid on and off.

Create > Grids & Snaps > Snap to Points controls the number of snap points along every member.

Create > Grids & Snaps > Perpendicular Point causes the cursor to snap to points on members that lie on perpendicular lines from the last point to the member.

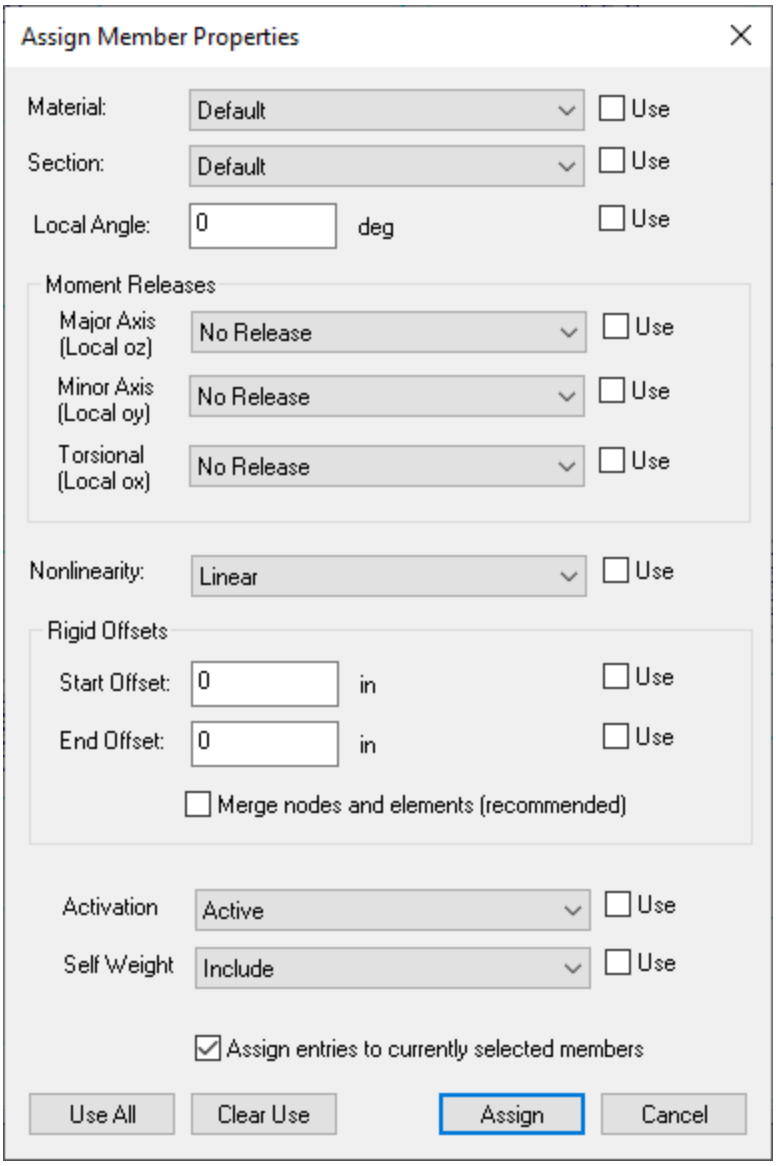
Create > Grids & Snaps > Clear Snap Points resets the snap points.

Member Properties

Create (or Modify) > Member Properties contains the commands used to assign and/or review member properties.

© 2026 ENERCALC, LLC

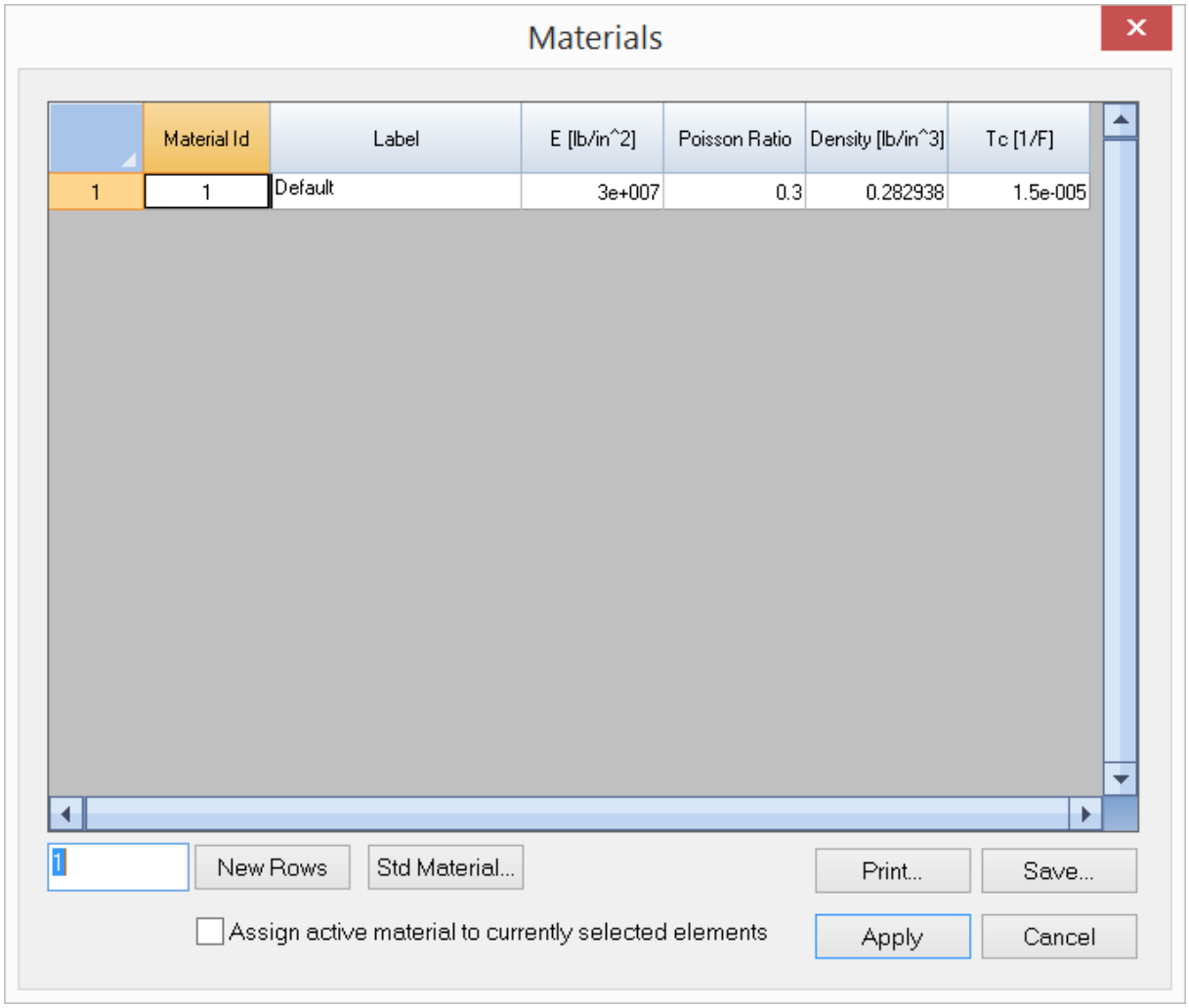
Create (or Modify) > Member Properties > Member Properties prompts you with the following dialog.



It allows you to assign one or more properties such as material, section etc to members. Make sure the “Use” checkbox by each property is set correctly. After clicking “Assign”, you can start to continuously assign all checked properties by window-selecting members until you right click the mouse or press the ESC key.

© 2026 ENERCALC, LLC

Create (or Modify) > Member Properties > Materials prompts you with the following dialog.



It allows you to define and/or assign materials to selected elements in the model. An ID is assigned automatically to each material by the program and may not be changed. You may assign a label with 127 maximum characters to each material for easy identification. The material properties include:

- Young's modulus (E),
- Poisson ratio ($0 \le v < 0.5$),
- Weight density,
- Temperature coefficients.

The shear modulus (G) is calculated automatically.

These material properties are used in the structural analysis of the model. Material properties related to design may be set from design ribbons. For example, concrete strength f'c and reinforcement strengths fy and fys may be defined from Design > RC Materials.

© 2026 ENERCALC, LLC

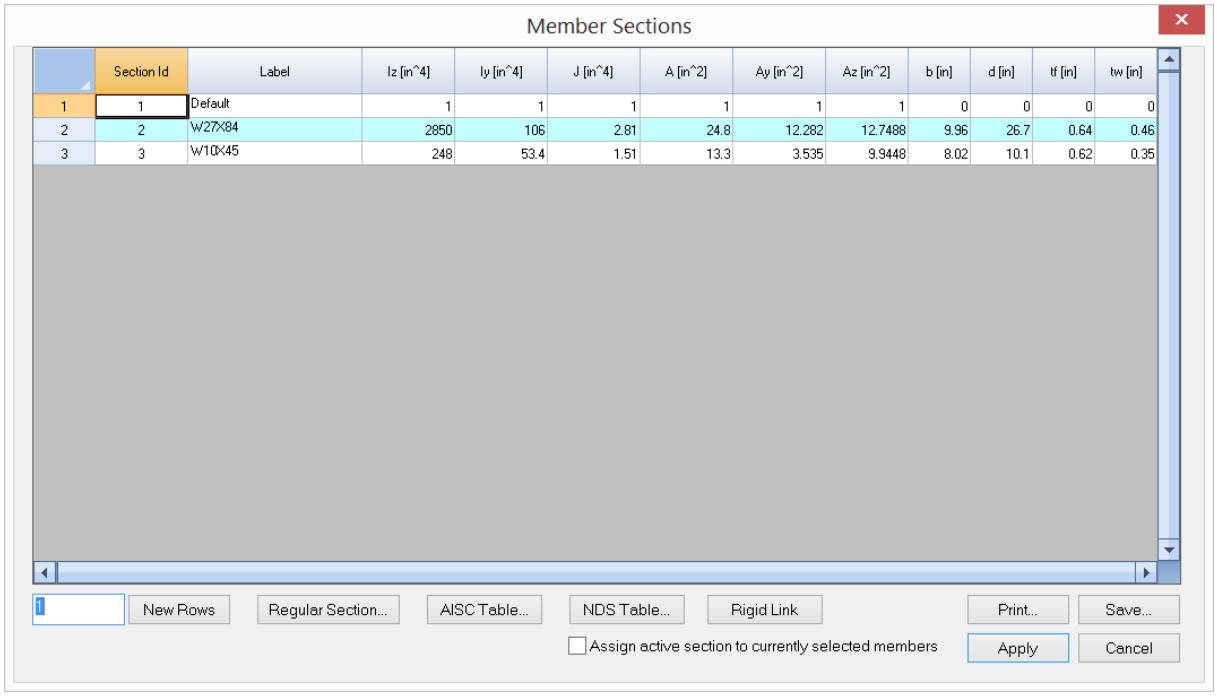
You may add standard steel and concrete materials by clicking the “Std Materials” button. A standard material label starts with “Steel” or “Concrete”.

You may add one or more materials by clicking the “New Rows” button. You may also print all materials in the list by clicking the “Print” button. The “Assign active material to currently selected elements” checkbox may be used to assign the active material to selected elements. The active material refers to the one that currently has focus in the list in the dialog. In order for material assignments to take place, members, shells or bricks must be selected beforehand.

*A more flexible way to assign material and other properties to entities is to use **Modify > Member Properties** or **Modify > Shell Properties** command, which allows you to continuously assign one or more properties to entities.*

The program always has a default material labeled “Default”. You may not delete this material or change its label. You may, however, change its properties.

Create (or Modify) > Member Properties > Member Sections prompts you with the following dialog.



© 2026 ENERCALC, LLC

It allows you to define and/or assign sections to selected members in the model. An ID is assigned automatically to each section by the program and may not be changed. You may assign a label with 127 maximum characters to each section for easy identification. The section properties include:

- moment of inertia about major axis (I_z)
- moment of inertia about minor axis (I_y)
- torsional moment of inertia (J)
- section area (A)
- shear area in the local y direction (A_y)
- shear area in the local z direction (A_z).

These properties are used in the analysis. Other properties (B, H, T_f and T_w) are dimensions for regular sections such as rectangular, circular, wide flange sections. These dimensional properties are used for graphic rendering only (not used in analysis).

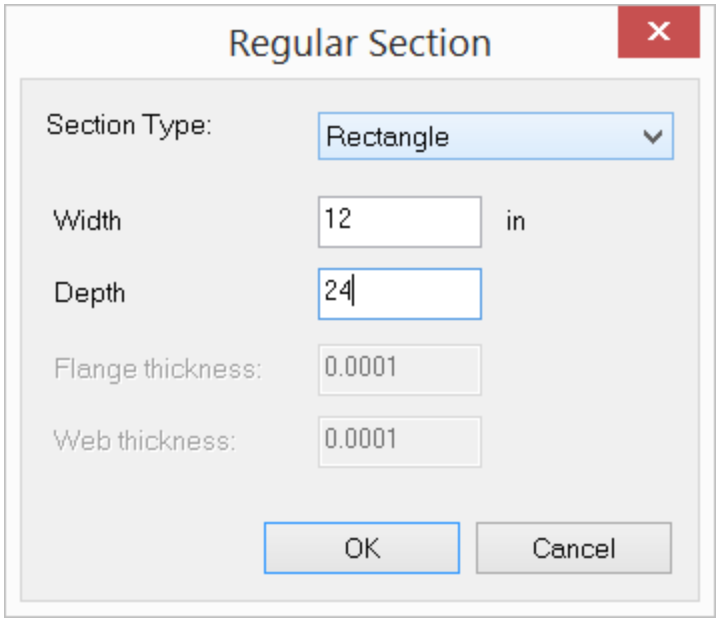
You may add one or more sections by clicking the “New Rows” button. You may also print all sections in the list by clicking the “Print” button. The “Assign active section to currently selected members” checkbox may be used to assign the active section to selected members. The active section refers to the one that currently has focus in the list in the dialog. In order for section assignments to take place, members must be selected beforehand.

*A more flexible way to assign member properties is to use **Modify > Member Properties** command, which allows you to continuously assign one or more properties to members.*

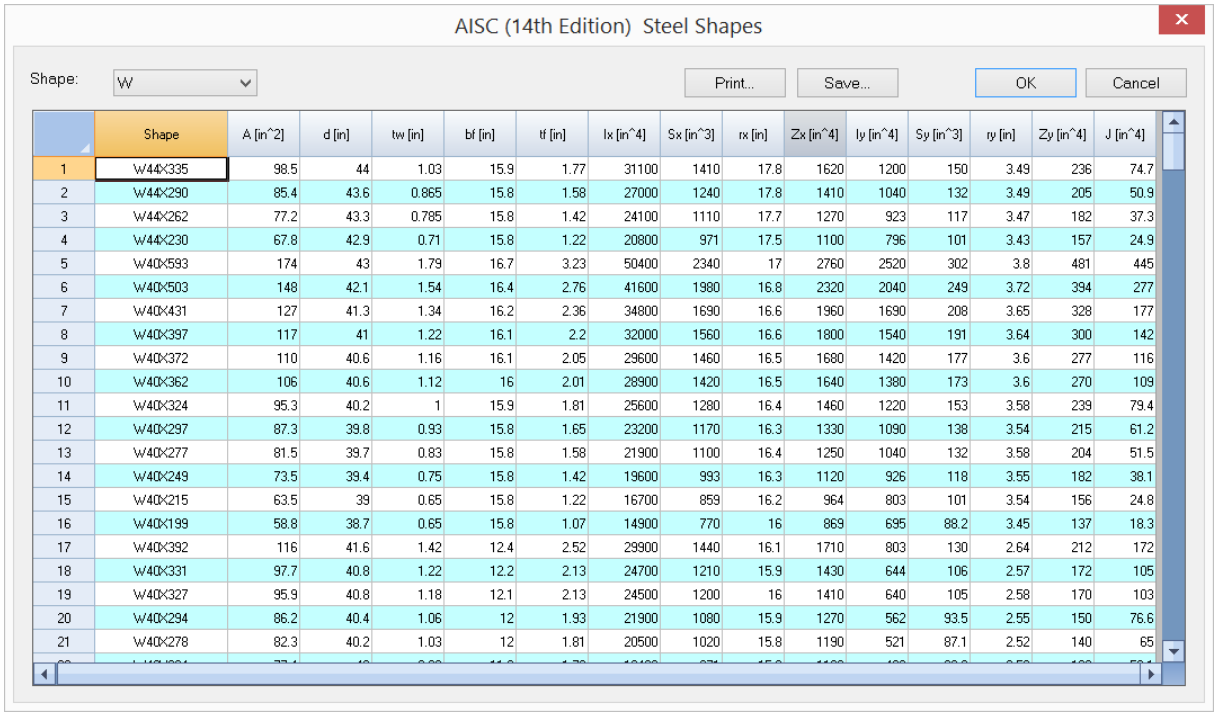
Note: *The section property input does not include Ixy – the product of inertia. For a section where Ixy is not zero such as an angle, the principal axes of a cross section are different from its geometric axes. In such situations, you should enter section properties in its principal axes and adjust the member element local angle accordingly.*

© 2026 ENERCALC, LLC

You may add a regular section by clicking the “Regular Section” button. The program displays the following dialog.



Three regular sections are currently provided by the program, namely rectangular, circular, wide flange and Tee sections. The properties of these sections are calculated automatically.



	Shape	A [in^2]	d [in]	tw [in]	bf [in]	tf [in]	Ix [in^4]	Sx [in^3]	rx [in]	Zx [in^4]	Iy [in^4]	Sy [in^3]	ry [in]	Zy [in^4]	J [in^4]
1	W44X335	98.5	44	1.03	15.9	1.77	31100	1410	17.8	1620	1200	150	3.49	236	74.7
2	W44X290	85.4	43.6	0.865	15.8	1.58	27000	1240	17.8	1410	1040	132	3.49	205	50.9
3	W44X262	77.2	43.3	0.785	15.8	1.42	24100	1110	17.7	1270	923	117	3.47	182	37.3
4	W44X230	67.8	42.9	0.71	15.8	1.22	20800	971	17.5	1100	796	101	3.43	157	24.9
5	W40X593	174	43	1.79	16.7	3.23	50400	2340	17	2760	2520	302	3.8	481	445
6	W40X503	148	42.1	1.54	16.4	2.76	41600	1980	16.8	2320	2040	249	3.72	394	277
7	W40X431	127	41.3	1.34	16.2	2.36	34800	1690	16.6	1960	1690	208	3.65	328	177
8	W40X397	117	41	1.22	16.1	2.2	32000	1560	16.6	1800	1540	191	3.64	300	142
9	W40X372	110	40.6	1.16	16.1	2.05	29600	1460	16.5	1680	1420	177	3.6	277	116
10	W40X362	106	40.6	1.12	16	2.01	28900	1420	16.5	1640	1380	173	3.6	270	109
11	W40X324	95.3	40.2	1	15.9	1.81	25600	1280	16.4	1460	1220	153	3.58	239	79.4
12	W40X297	87.3	39.8	0.93	15.8	1.65	23200	1170	16.3	1330	1090	138	3.54	215	61.2
13	W40X277	81.5	39.7	0.83	15.8	1.58	21900	1100	16.4	1250	1040	132	3.58	204	51.5
14	W40X249	73.5	39.4	0.75	15.8	1.42	19600	993	16.3	1120	926	118	3.55	182	38.1
15	W40X215	63.5	39	0.65	15.8	1.22	16700	859	16.2	964	803	101	3.54	156	24.8
16	W40X199	58.8	38.7	0.65	15.8	1.07	14900	770	16	869	695	88.2	3.45	137	18.3
17	W40X392	116	41.6	1.42	12.4	2.52	29900	1440	16.1	1710	803	130	2.64	212	172
18	W40X331	97.7	40.8	1.22	12.2	2.13	24700	1210	15.9	1430	644	106	2.57	172	105
19	W40X327	95.9	40.8	1.18	12.1	2.13	24500	1200	16	1410	640	105	2.58	170	103
20	W40X294	86.2	40.4	1.06	12	1.93	21900	1080	15.9	1270	562	93.5	2.55	150	76.6
21	W40X278	82.3	40.2	1.03	12	1.81	20500	1020	15.8	1190	521	87.1	2.52	140	65

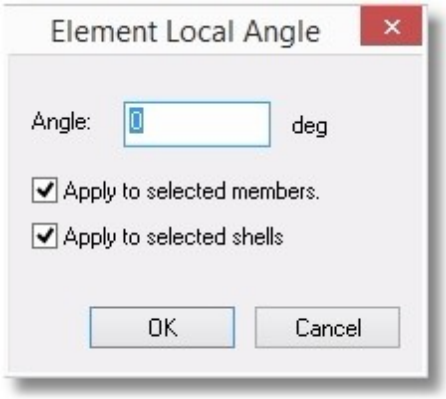
© 2026 ENERCALC, LLC

You may also add sections from the AISC steel shape table (above) or NDS wood shape table. *You should not modify an AISC or NDS shape label or its properties.*

You may create one and only one rigid link section for use in the model by simply click “Rigid Link” button. A rigid link is a member that has very large sectional properties (A, Ay, Az, Iz, Iy and J). There can only be one rigid link section defined in the model and it must be named as “RIGID_LINK”. The properties for the RIGID_LINK section must be set to 0’s on the member section dialog. The program will appropriately calculate A, Ay, Az, Iz, Iy and J during the solution process. **Self weight for rigid links will be ignored by the program.**

The program always has a default section labeled “Default”. You may not delete this section or change its label. You may, however, change its properties.

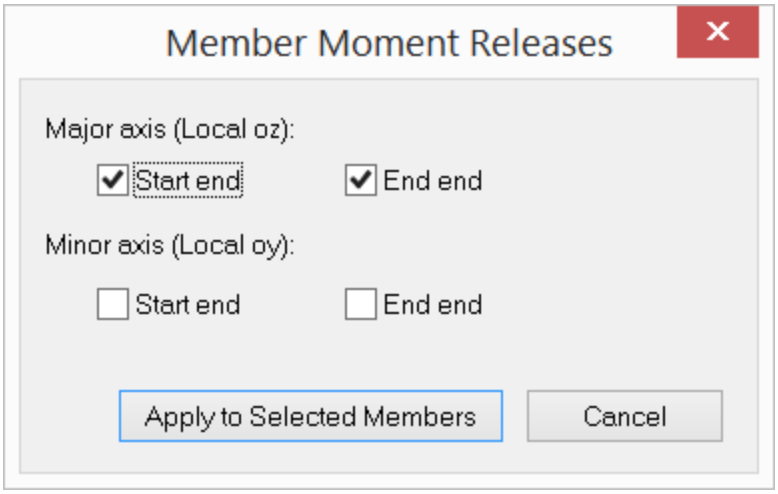
Create (or Modify) > Member Properties > Local Angles prompts you with the following dialog to assign local angles to the selected members and/or shells.



The element local angle is used to change the element local coordinate system.

© 2026 ENERCALC, LLC

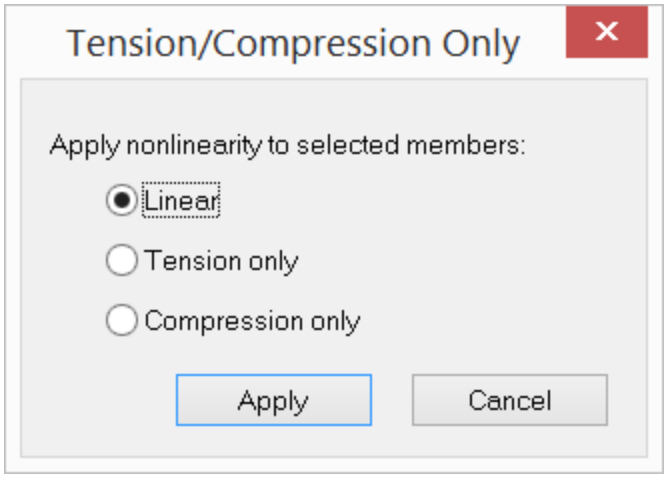
Create (or Modify) > Member Properties > Moment Releases prompts you with the following dialog.



It allows you to assign moment releases to selected members in the model. Major or minor moment releases may be applied to the start and/or end ends of the members. Trusses are members with moments fully released at both ends. The program assigns appropriate moment releases automatically if the model type is of “2D Truss” or “3D Truss”. However, if the model contains both trusses and beams, you should use the model type “2D Frame” or “3D Frame”, and assign appropriate moment releases to members. In order for moment release assignments to take place, members must be selected beforehand.

© 2026 ENERCALC, LLC

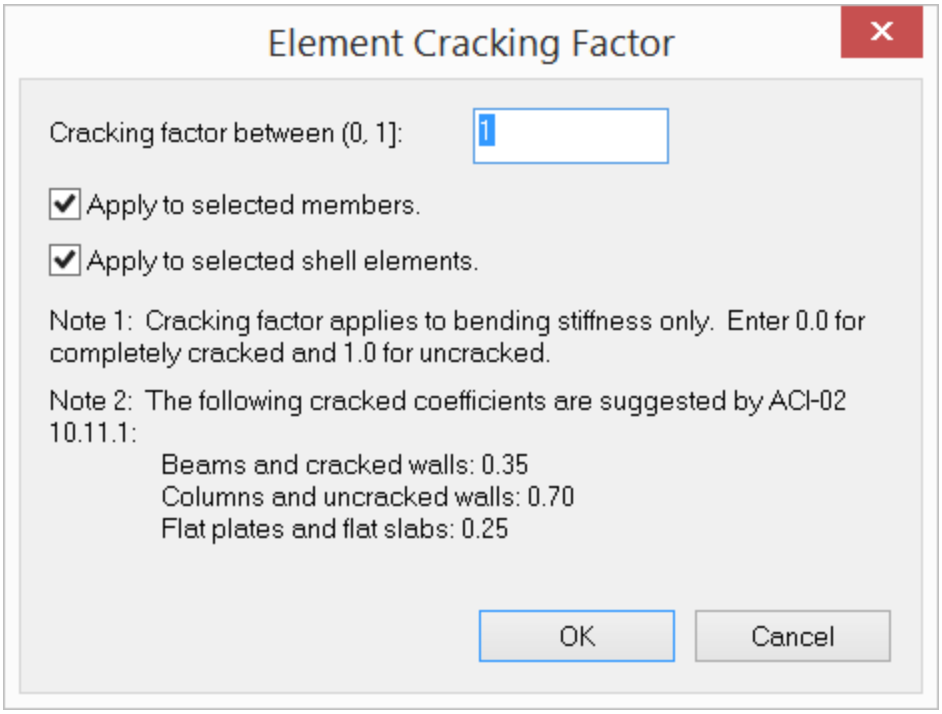
Create (or Modify) > Member Properties > Tension/Compression Only prompts you with the below dialog.



It allows you to assign nonlinearity (linear, tension only or compression only) to the selected members. The member stiffness will be ignored if a tension only member is subjected to compressive forces or if a compression only member is subjected to tensile forces. The presence of tension only or compression only members makes the model nonlinear and requires iterative solution for each load combination.

© 2026 ENERCALC, LLC

Create (or Modify) > Member Properties > Cracking Factors prompts you with the following dialog.

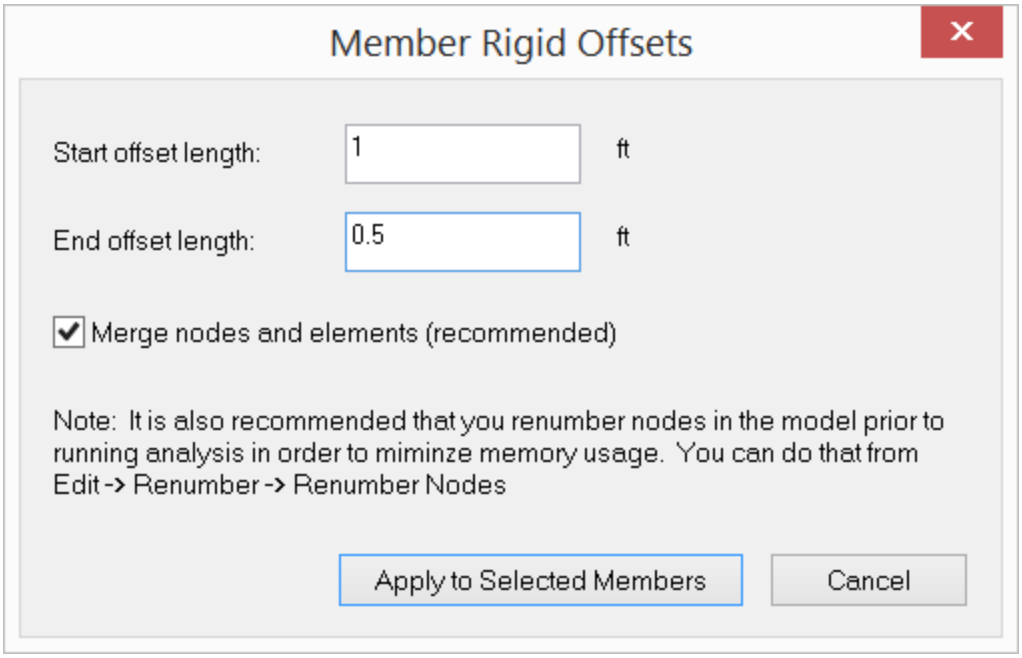


It allows you to assign cracking factors to selected beams, columns, and plates. Cracking factors apply only to bending stiffness of members and shell elements.

Note: *Cracking factors are not considered by the program unless you check the option "Use cracked section properties (Icr) for members and finite elements" in Analysis > Analysis Options. Analysis results are cleared after assignment of cracking factors.*

© 2026 ENERCALC, LLC

Create (or Modify) > Member Properties > Rigid Offsets prompts you with the following dialog.



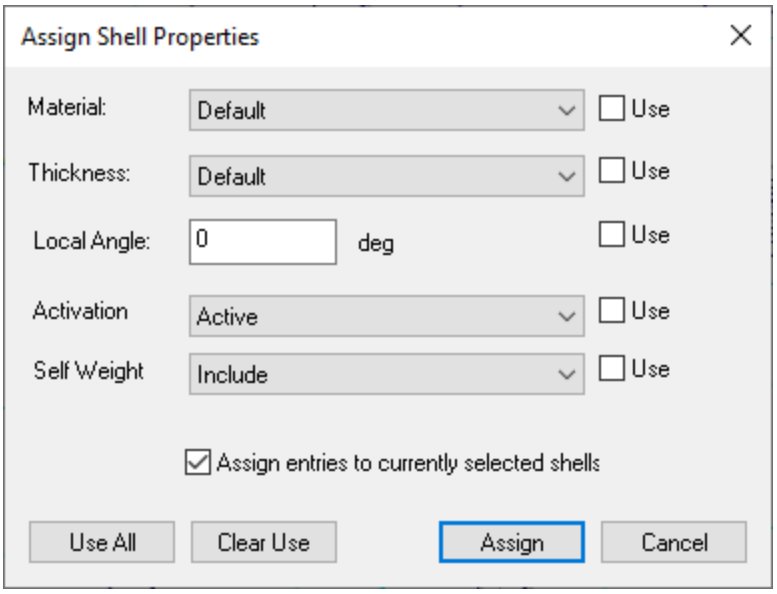
It allows you to assign rigid offsets to the selected members. This command will effectively break each selected member into two or three members, with either or both ends being rigid links.

Shell Properties

Create (or Modify) > Shell Properties contains the commands used to assign and/or review member properties.

© 2026 ENERCALC, LLC

Create (or Modify) > Shell Properties > Shell Properties prompts you with the following dialog.

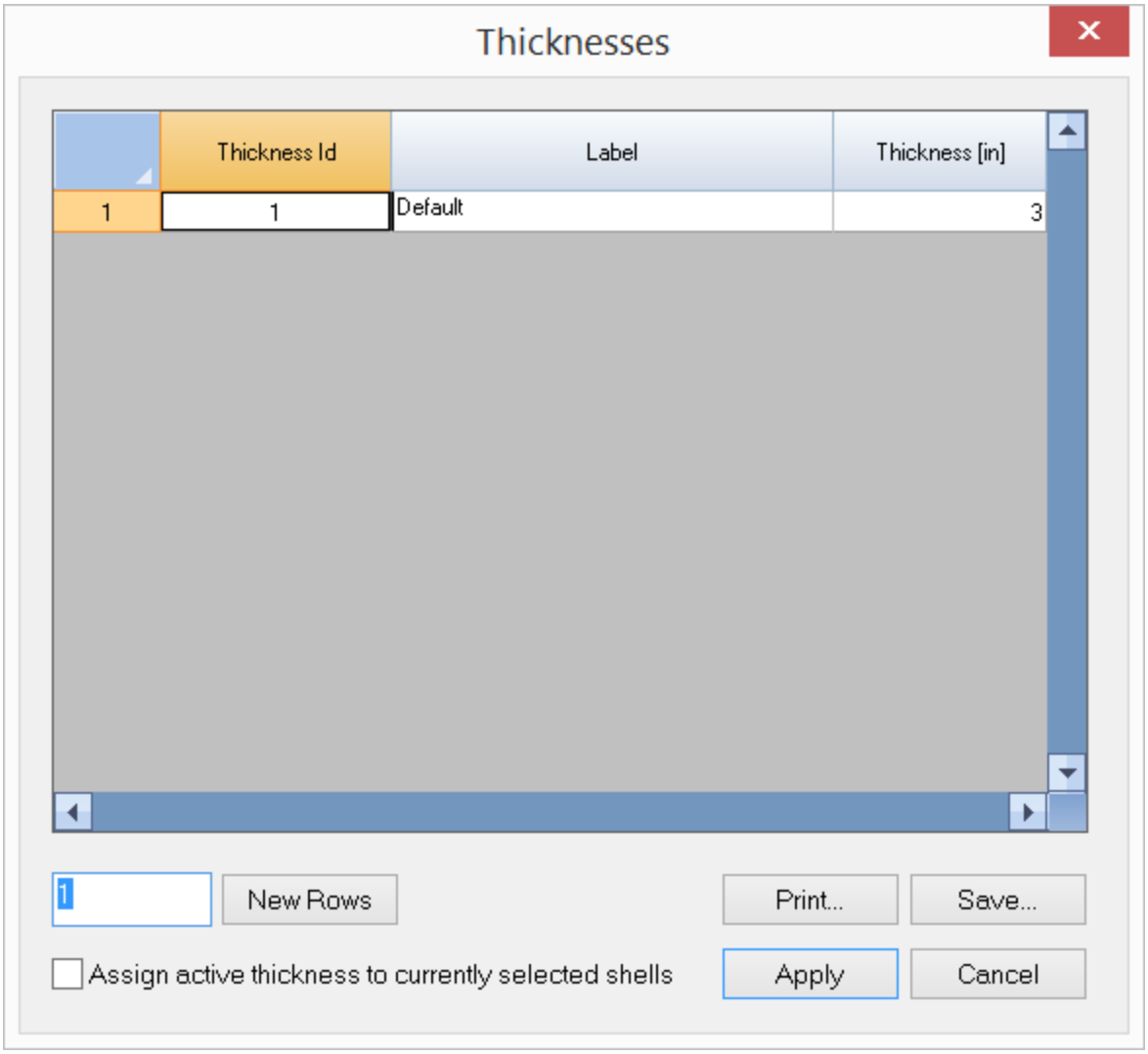


It allows you to assign one or more properties such as material, thickness etc to shells. Make sure the “Use” checkbox by each property is set correctly. After clicking “Assign”, you can start to continuously assign all checked properties by window-selecting shells until you right click the mouse or press the ESC key.

Create (or Modify) > Shell Properties > Materials prompts you with the same dialog as for Materials for members. It is provided in this ribbon for convenience.

© 2026 ENERCALC, LLC

Create (or Modify) > Shell Properties > Thicknesses prompts you with the following dialog.



It allows you to define and/or assign thicknesses to selected shells in the model. An ID is assigned automatically to each thickness by the program and may not be changed. You may assign a label with 127 maximum characters to each thickness for easy identification. The thickness properties include thickness only.

You may add one or more thicknesses by clicking the “New Rows” button. You may also print all thicknesses in the list by clicking the “Print” button. The “Assign active thickness to currently selected shells” checkbox may be used to assign the active thickness to selected shells. The active thickness refers to the one that currently has focus in the list in the dialog. In order for thickness assignments to take place, shells must be selected beforehand.

*A more flexible way to assign shell properties is to use **Modify > Shell Properties** command, which allows you to continuously assign one or more properties to shells.*

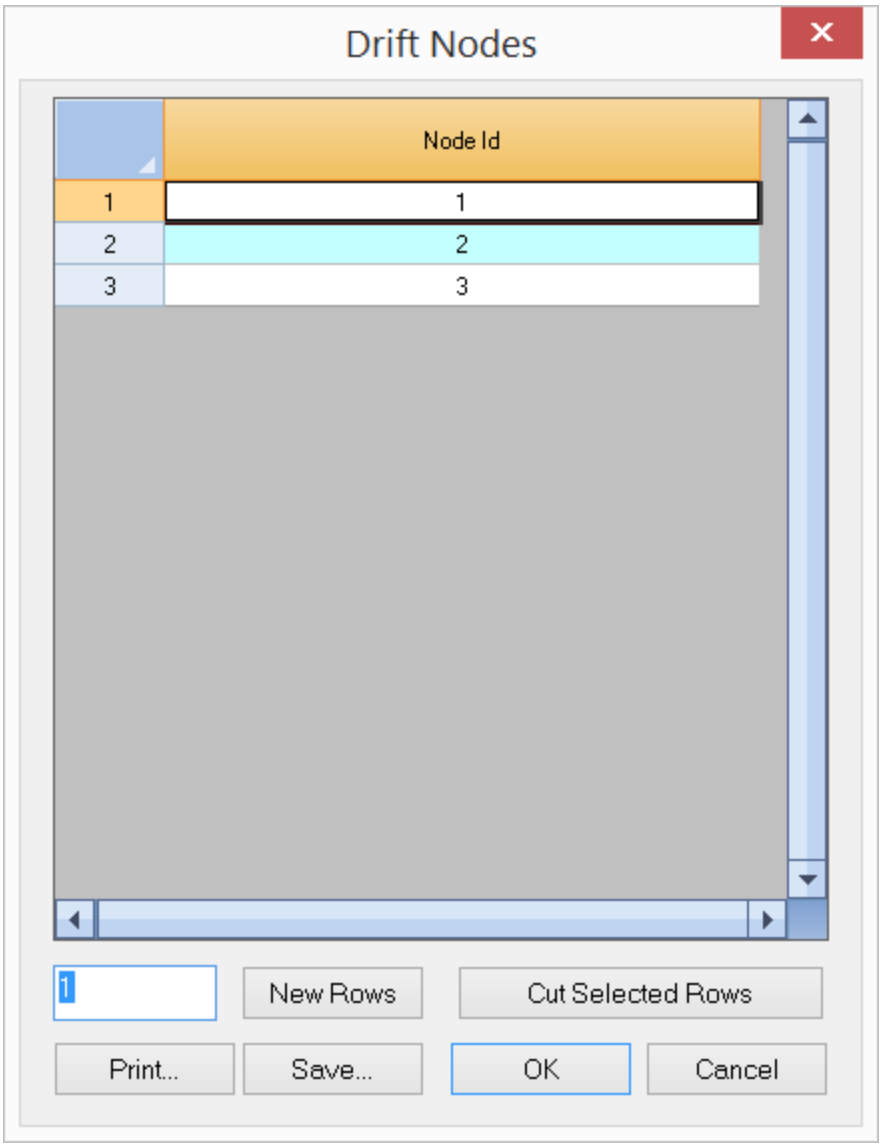
© 2026 ENERCALC, LLC

The program always has a default thickness labeled “Default”. You may not delete this thickness or change its label. You may, however, change its properties.

Create (or Modify) > Shell Properties > Element Local Angles prompts you with the same dialog as for Members. It is provided in this ribbon for convenience.

Story Drift Nodes

Create > Story Drift Nodes prompts you with the following dialog.

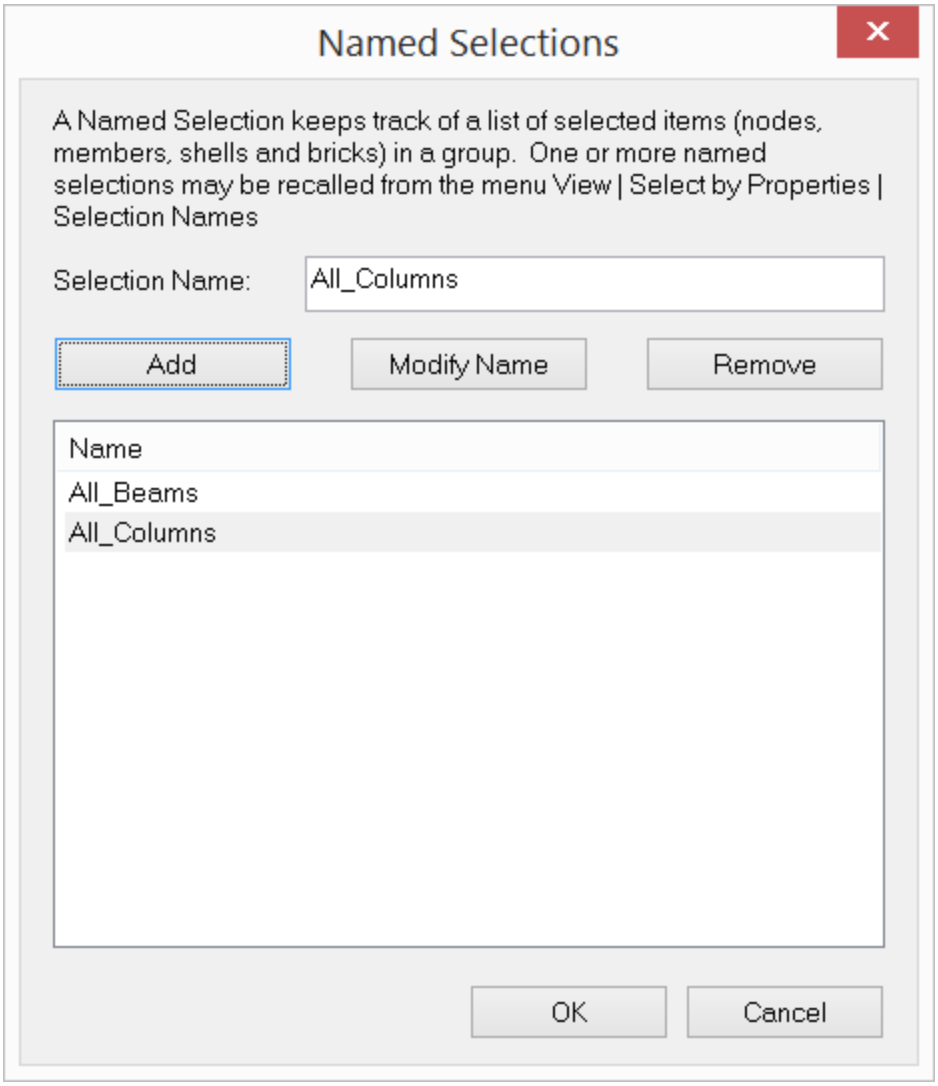


It allows you to enter nodes that will be used for floor drift calculation. An empty row is allowed if all rows below it do not contain any non-empty fields. Selected rows (whole row must be selected) may be cut by clicking the button “Cut Selected Rows”.

© 2026 ENERCALC, LLC

Named Selection

Create > Named Selection prompts you with the following dialog.

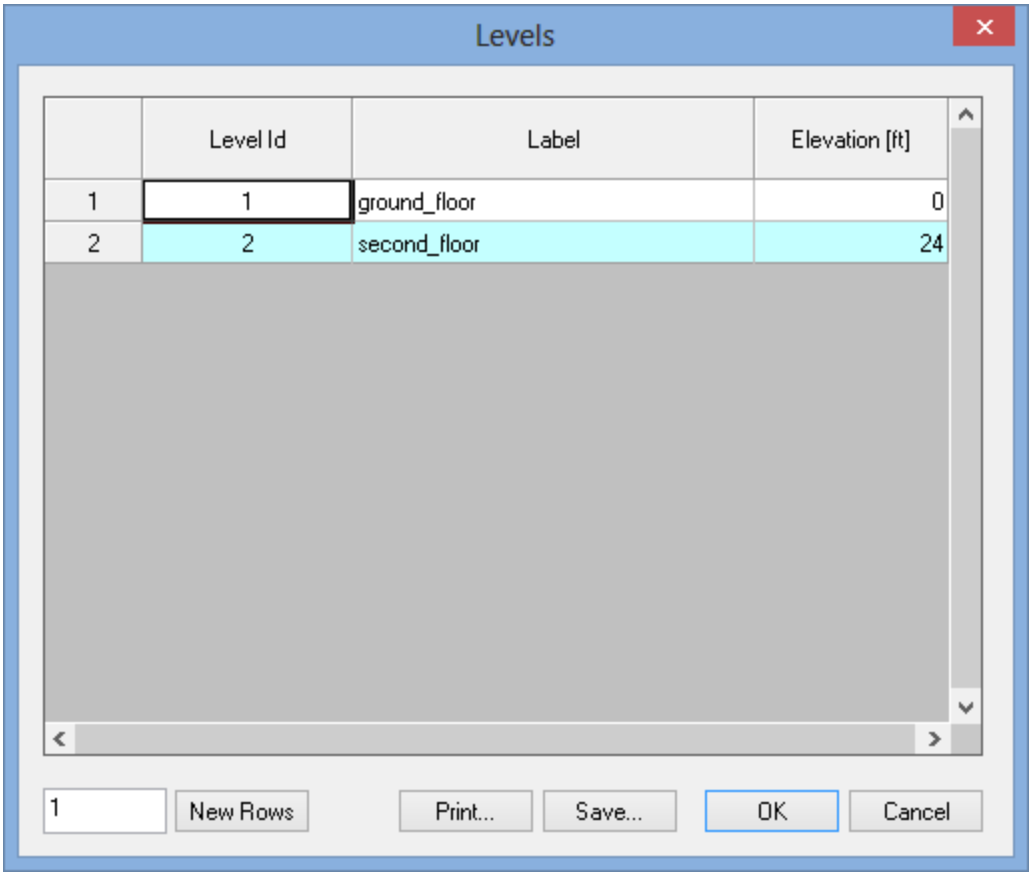


It allows you to save the currently selected items to a named group. You may use the command Create > Select by Properties > Select by Selection Names to recall the previously saved named selections. This command is very useful to group related items.

© 2026 ENERCALC, LLC

Levels

Create > Levels prompts you with the following dialog.



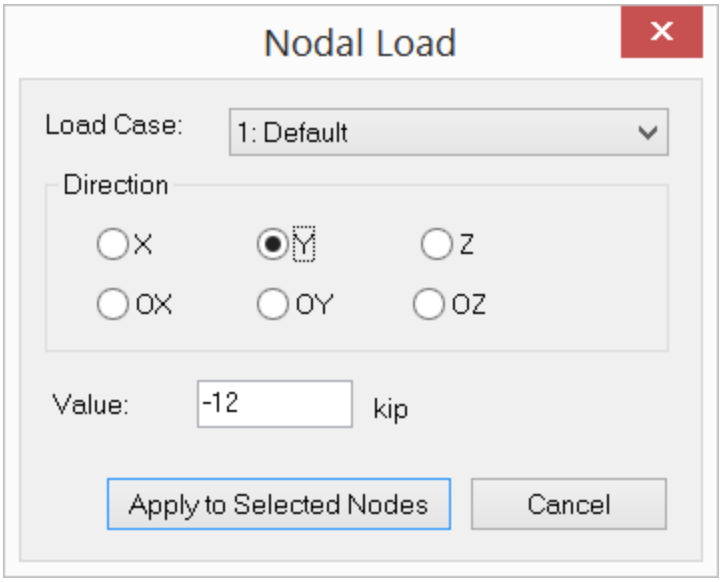
It allows you to define physical levels in the model. Once levels are defined, you are able to view a level plan by using the command View > Hide All Except Level. To reveal the hidden parts of the model, click View > Show All.

Draw Loads

Create > Draw Loads contains the commands used to apply loads to the model.

© 2026 ENERCALC, LLC

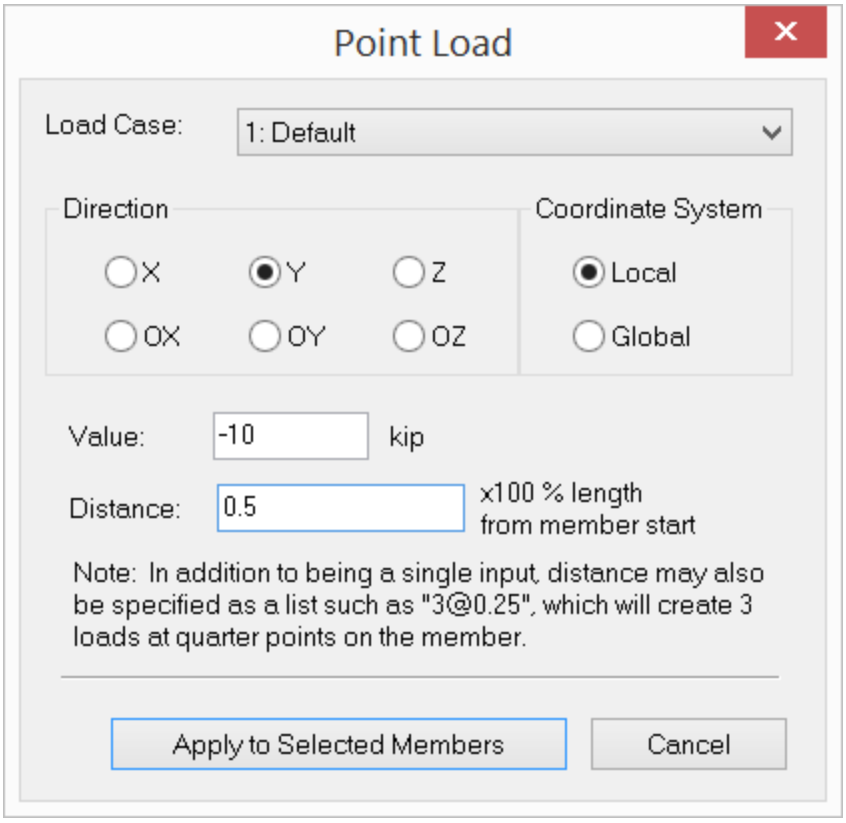
Create > Draw Loads > Nodal Loads prompts you with the following dialog.



It allows you to assign nodal loads to selected nodes in the model. You must select a load case to which the nodal loads belong. Nodal loads are specified in the global coordinate system. The loads are nodal forces in the X, Y, or Z direction if radio button “X”, “Y”, or “Z” is selected. The loads are nodal moments in the X, Y, or Z direction if radio button “OX”, “OY”, or “OZ” is selected. The load magnitude may be any non-zero value.

© 2026 ENERCALC, LLC

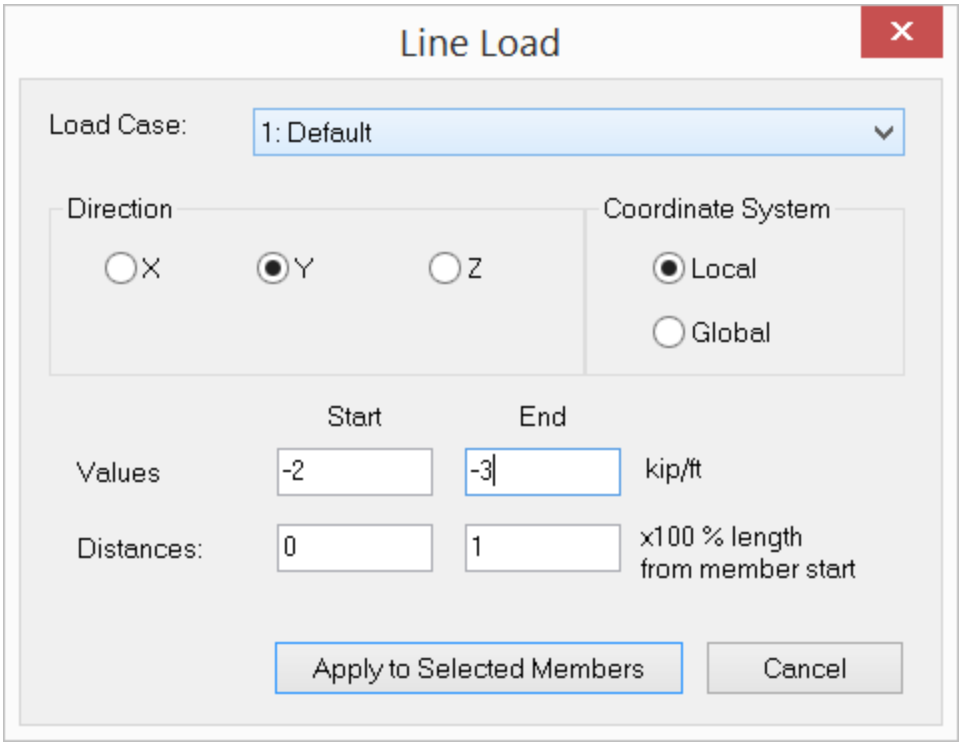
Create > Draw Loads > Point Loads prompts you with the following dialog.



It allows you to assign point loads to selected members in the model. You must select a load case to which the point loads belong. Point loads may be specified in either the local or global coordinate system. The loads are point forces in the X, Y, or Z direction if radio button “X”, “Y”, or “Z” is selected. The loads are point moments in the X, Y, or Z direction if radio button “OX”, “OY”, or “OZ” is selected. The load magnitude may be any non-zero value. The load distance is the ratio of the load location (measured from the member start) to the member length. A distance of 0.5 places the load at the middle of each selected member.

© 2026 ENERCALC, LLC

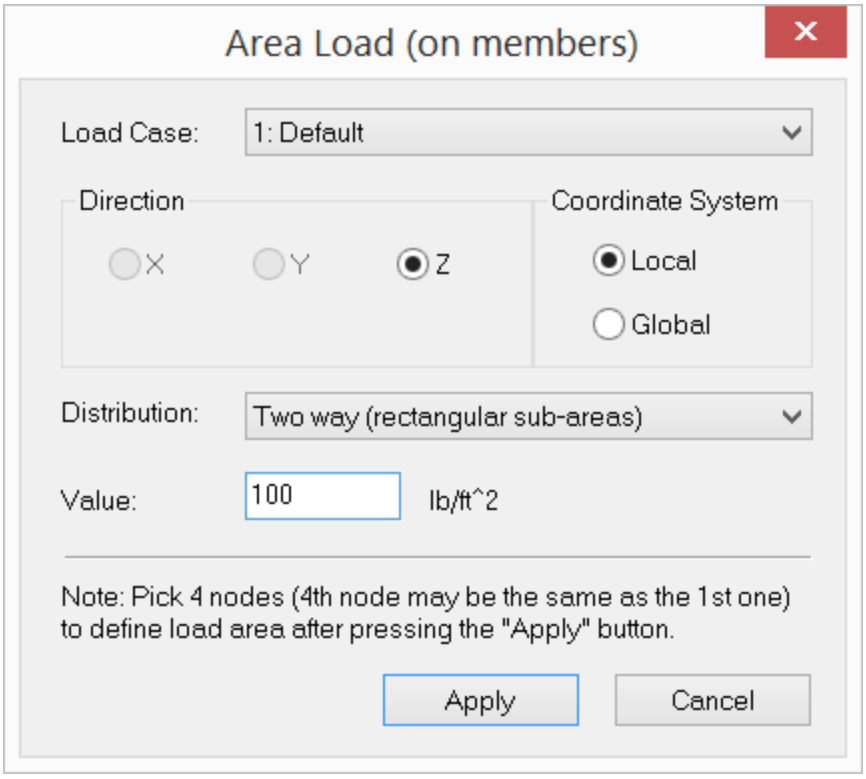
Create > Draw Loads > Line Loads prompts you with the following dialog.



It allows you to assign line loads to selected members in the model. You must select a load case to which the line loads belong. Line loads may be specified in either the local or global coordinate system. The loads are line forces in the X, Y, or Z direction. The start and end magnitudes of the load may be zero for either end but not for both. The load distances are the ratios of the load start and end locations (measured from the member start) to the member length. A start distance of 0.0 and an end distance of 1.0 place the line load on the entire span of each selected member.

© 2026 ENERCALC, LLC

Create > Draw Loads > Area Loads prompts you with the following dialog.



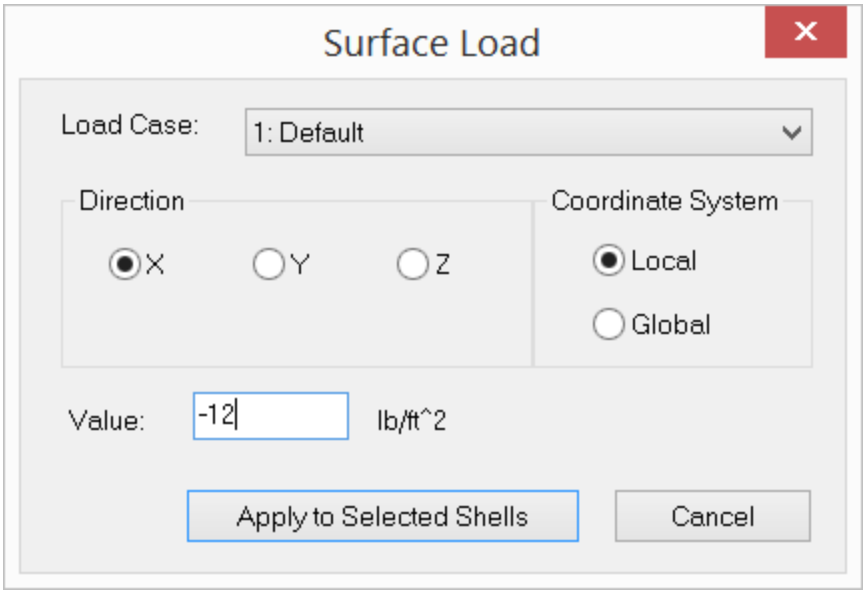
It allows you to assign area loads to enclosed areas of members in the model. You must select a load case to which the area loads belong. Area loads may be specified in either the local or global coordinate system. Global area loads may be in the global X, Y, or Z direction. Local area loads may only be in the local z direction, which is perpendicular to the load area.

A load area is defined by specifying three or four coplanar nodes. The area load is then distributed as line loads to perimeter members of enclosed areas within the load area prior to static or dynamic solution. Various area load distribution methods are available. It is recommended that area loads be defined in their own load cases. In this way, you will find it easier to identify, edit, and delete area loads later on.

The program also allows you to convert area loads to line loads automatically. This feature lets you see how the program would convert the area loads prior to the solution. For more information on the load conversion, see Tables > Area Loads.

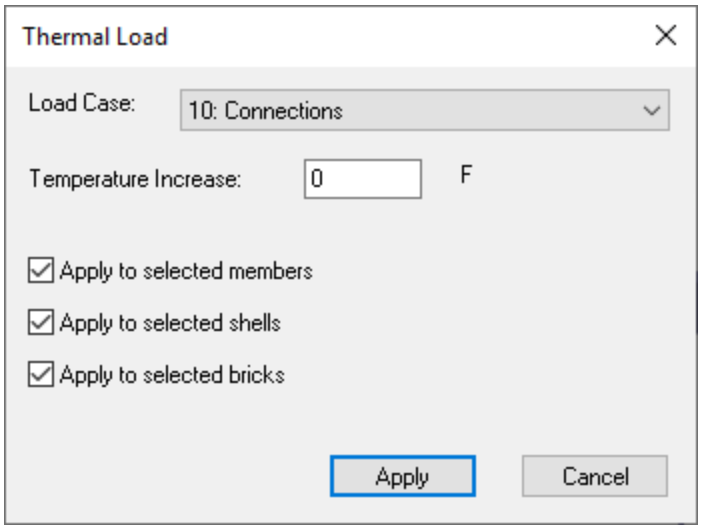
© 2026 ENERCALC, LLC

Create > Draw Loads > Surface Loads prompts you with the following dialog.



It allows you to assign surface loads to selected shells in the model. You must select a load case to which the surface loads belong. Surface loads may be specified in either the local or global coordinate system. The loads are surface forces in the X, Y, or Z direction. Surface load applies to the entire surface of a shell element.

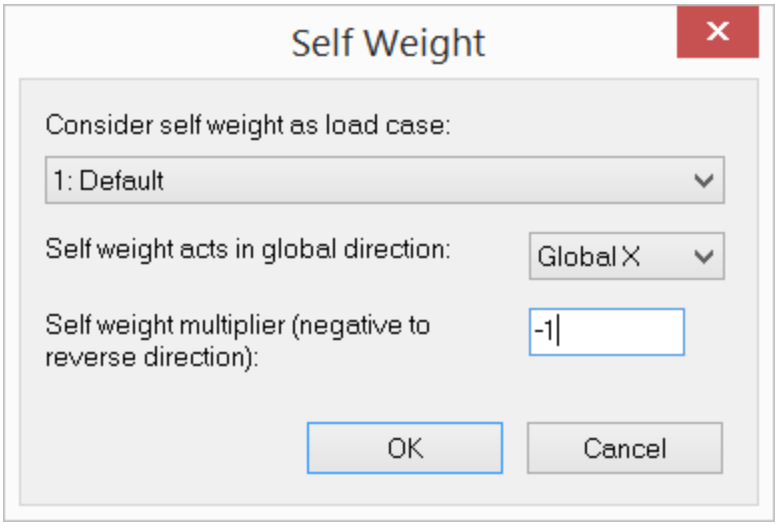
Create > Draw Loads > Thermal Loads prompts you with the following dialog.



It allows you to assign thermal loads to selected elements in the model. You must select a load case to which the surface loads belong. Currently, ENERCALC 3D considers thermal effect in longitudinal direction of members, membrane directions of shells, and bricks. It does not consider thermal gradients in members or shells.

© 2026 ENERCALC, LLC

Create > Draw Loads > Self Weights prompts you with the following dialog.



It allows you to define how the program computes self weights for all elements in the model. You must select a load case to which self weights belong. The self weights may act in the global X, Y, or Z direction. By default, self weights act in the global Y direction. You may specify a self weight multiplier (applied to material densities). A zero multiplier ignore self weights altogether.

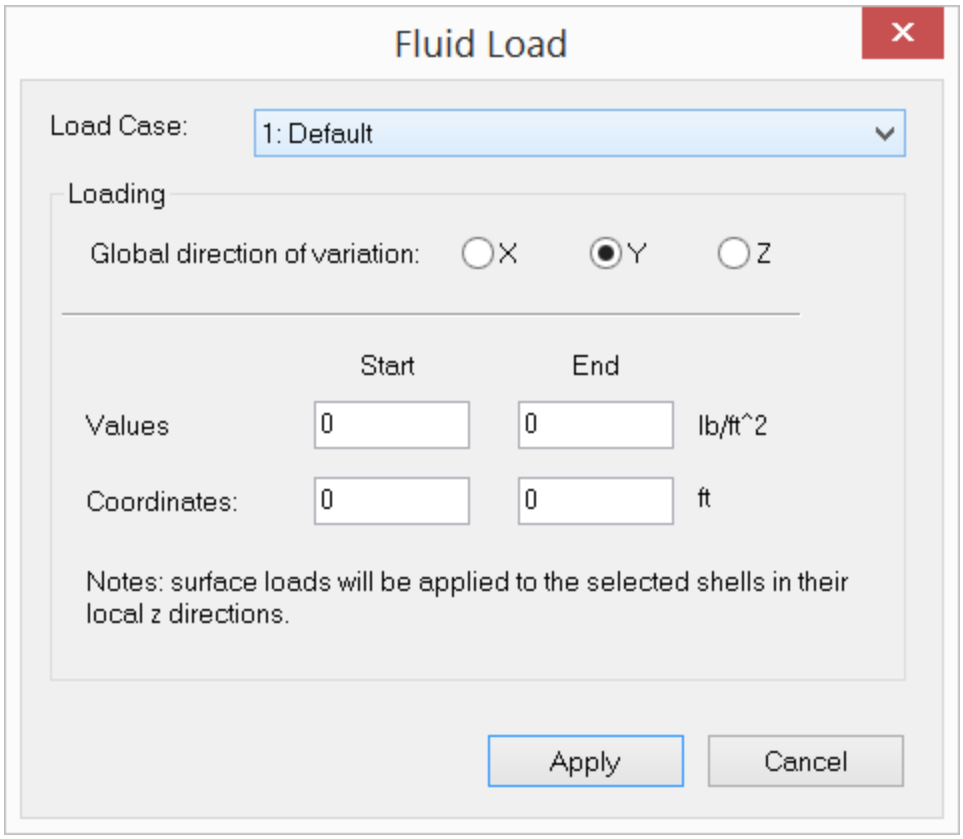
Create > Draw Loads > Self Weight Exclusion allows self weight to be considered or ignored for selected members, shells, and bricks.

Generate Loads

Create > Generate Loads contains the commands used to automatically generate certain types of loads in the model.

© 2026 ENERCALC, LLC

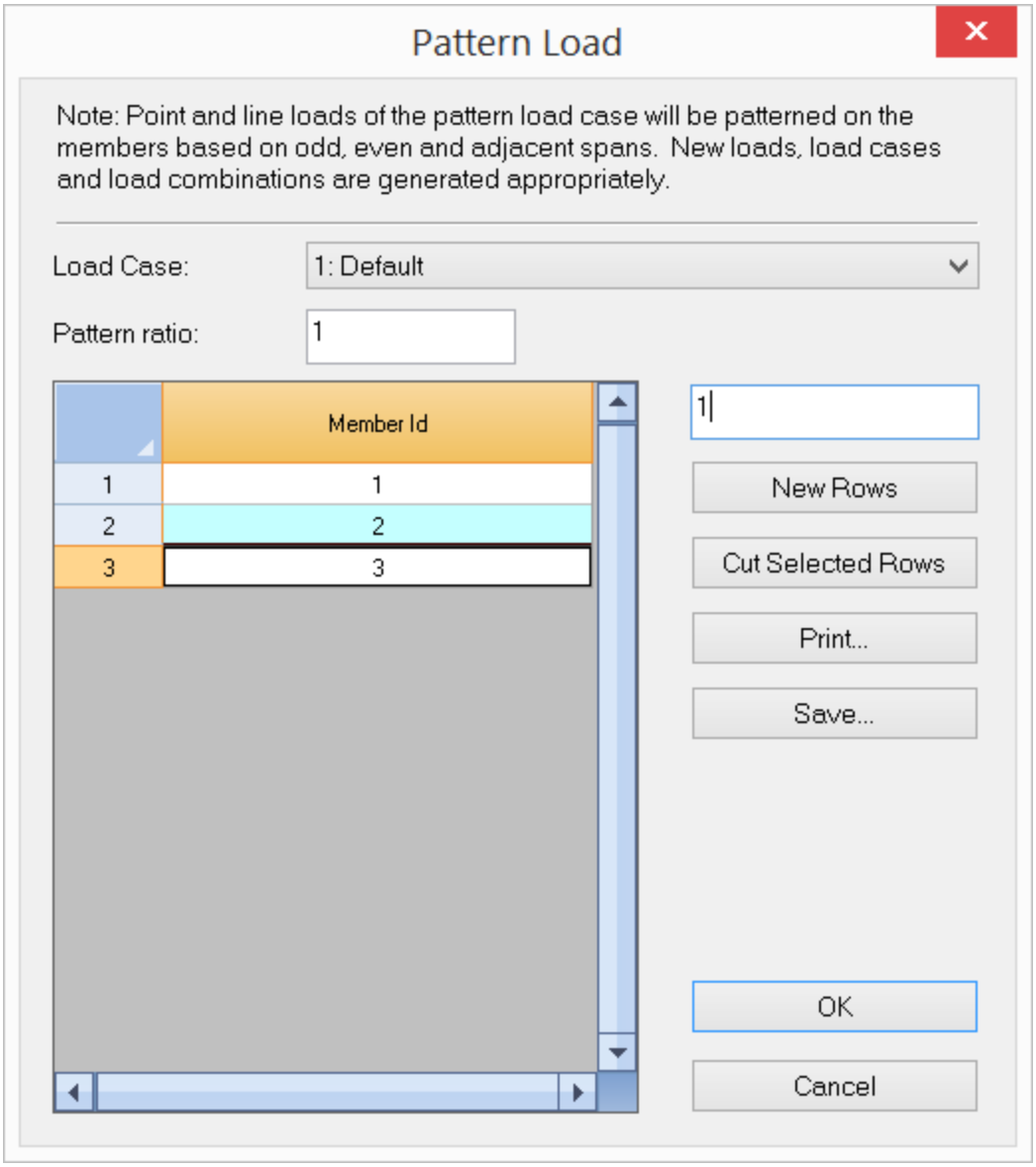
Create > Generate Loads > Fluid Loads prompts you with the dialog.



It allows you to generate fluid loads applied to selected shells in the model. You must select a load case to which the fluid loads belong. Fluid loads are applied in the local coordinate system. The load variation must be in global X, Y or Z direction.

© 2026 ENERCALC, LLC

Create > Generate Loads > Pattern Loads prompts you with the following dialog.



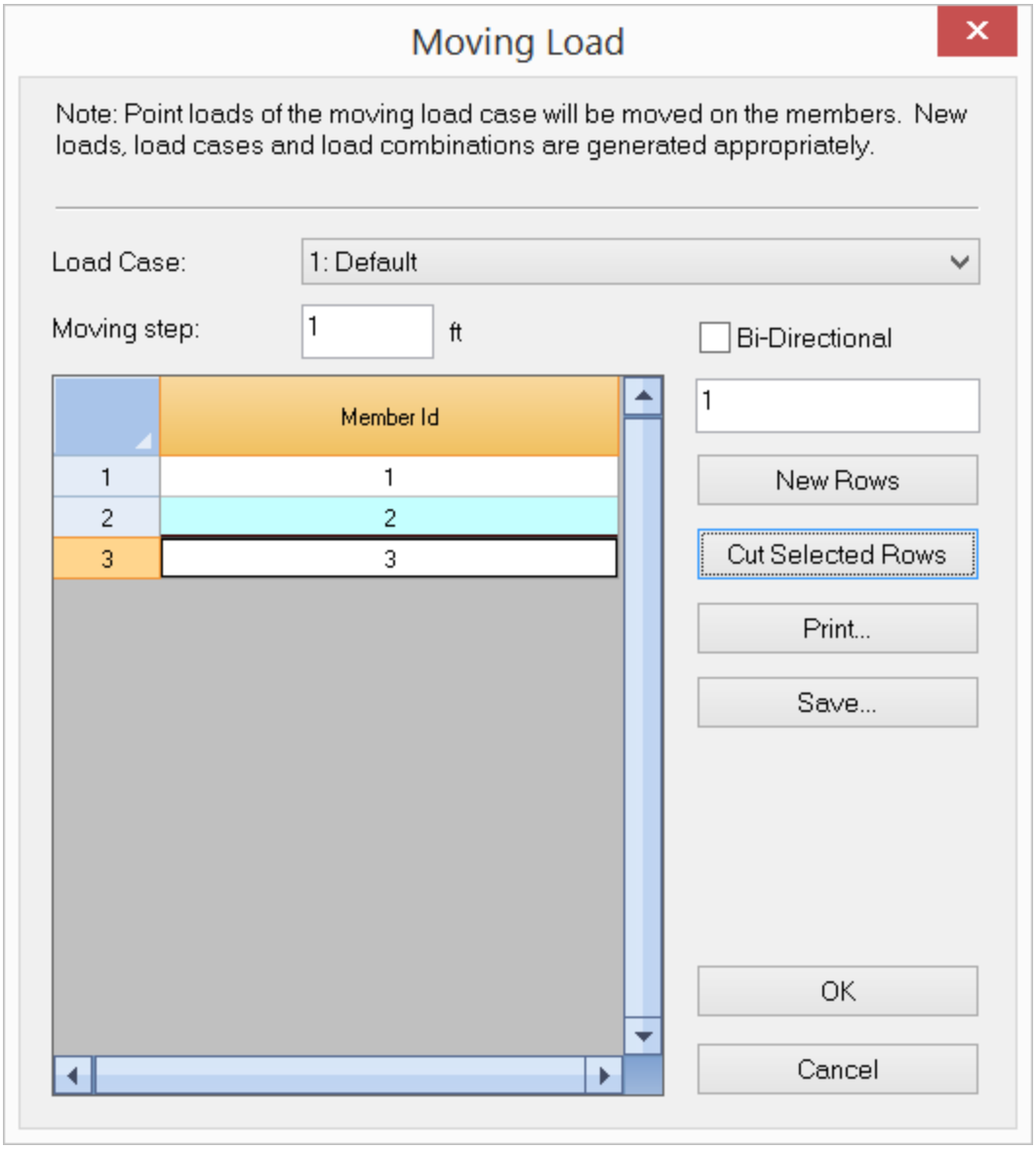
It allows you to generate pattern loads applied to specified members in the model. You must select a load case (generally live case) that contains loads to be patterned. A pattern ratio is also available (to address provisions such as ACI 318-19 Section 6.4.3.3, where a factor is applied in certain skip loading situations). Load patterning allows us to generate maximum positive and negative moment at each span, maximum positive and negative moment at each support as well as maximum shear at each support. The existing point and line loads in the load case will be patterned based on odd, even and adjacent/alternate spans. These patterned loads are assigned to their own load cases. The program automatically generates additional load cases and load combinations based on the load patterning.

© 2026 ENERCALC, LLC

It should be pointed out that the program does not consider support conditions for pattern load generation. One pattern load case cannot be used in more than one load combination prior to load pattern generation.

© 2026 ENERCALC, LLC

Create > Generate Loads > Moving Loads prompts you with the following dialog.



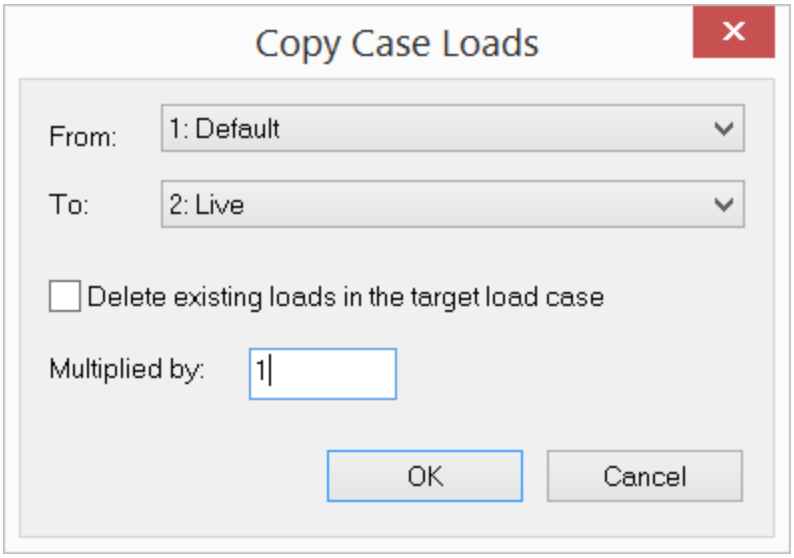
It allows you to generate moving loads applied to specified members in the model. You must select a load case that contains moving loads. Only point loads on the specified members in the load case will be moved. These moving loads are assigned to their own load cases. The program automatically generates additional load cases and load combinations based on the moving step size.

If the Bi-Directional option is deselected, generation steps will take place in the positive global axis direction.
If the Bi-Directional option is selected, generation steps will take place in the positive AND negative global axis direction.

© 2026 ENERCALC, LLC

It should be pointed out that one moving load case cannot be used in more than one load combination prior to moving load generation.

Create > Generate Loads > Case-Copy Loads prompts you with the following dialog.



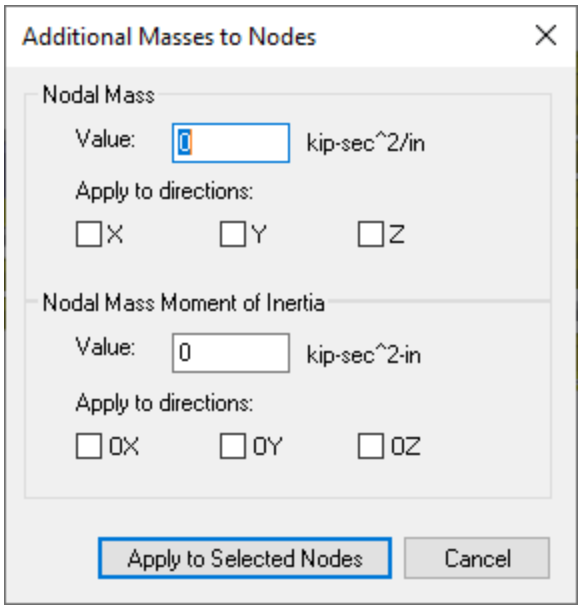
It allows you to copy all loads from one load case to another. You have the option to delete existing loads in the target load case. The loads copied may also be multiplied by a factor. At least two load cases must exist in the model in order to use this command.

Create > Generate Loads > Convert Area Loads to Line Loads will convert all area loads to line loads in every load case. This is useful in checking how area loads would be converted during the solution process. You can always undo the area loads to line loads conversion.

Create > Generate Loads > Convert Local Loads to Global Loads will convert all member point loads, line loads and shell surface loads from local coordinate systems to global coordinate system in every load case. This is useful for data transfer from ENERCALC 3D to Revit Structure using FastFrame Revit Link. You can always undo the local loads to global loads conversion.

© 2026 ENERCALC, LLC

Create > Generate Loads > Additional Masses prompts you with the dialog below.

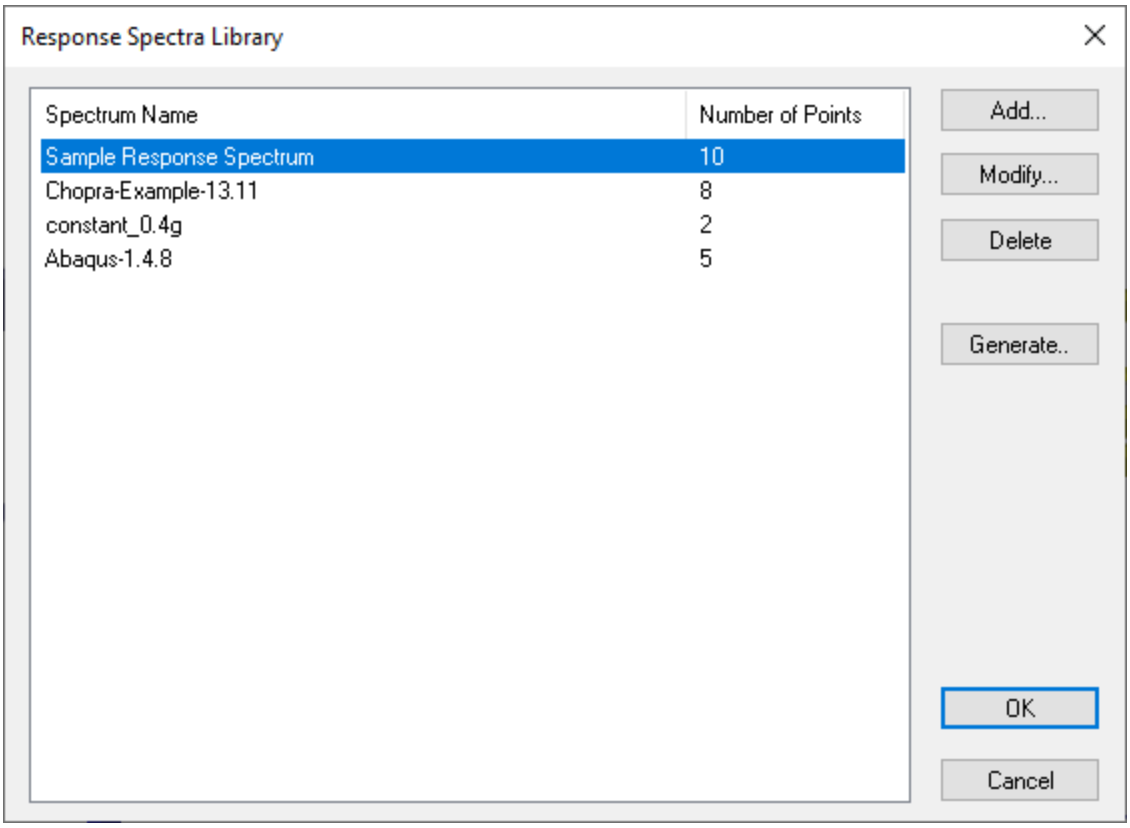


It allows you to assign additional masses and mass moment of inertia to selected nodes. The mass can be applied to X, Y and/or Z directions while the mass moment of inertia can be applied to OX, OY and/or OZ directions. Additional Masses are added to the mass calculated from the load combination for frequency analysis (see the command: Analysis > Frequency Analysis). Mass moment of inertia values can only be input using the Additional Masses command.

The mass unit is a force unit divided by the acceleration of gravity, while the mass moment of inertia has units of mass times length squared. The acceleration of gravity is taken as 386.09 in/sec^2 or 9.8 m/sec^2.

© 2026 ENERCALC, LLC

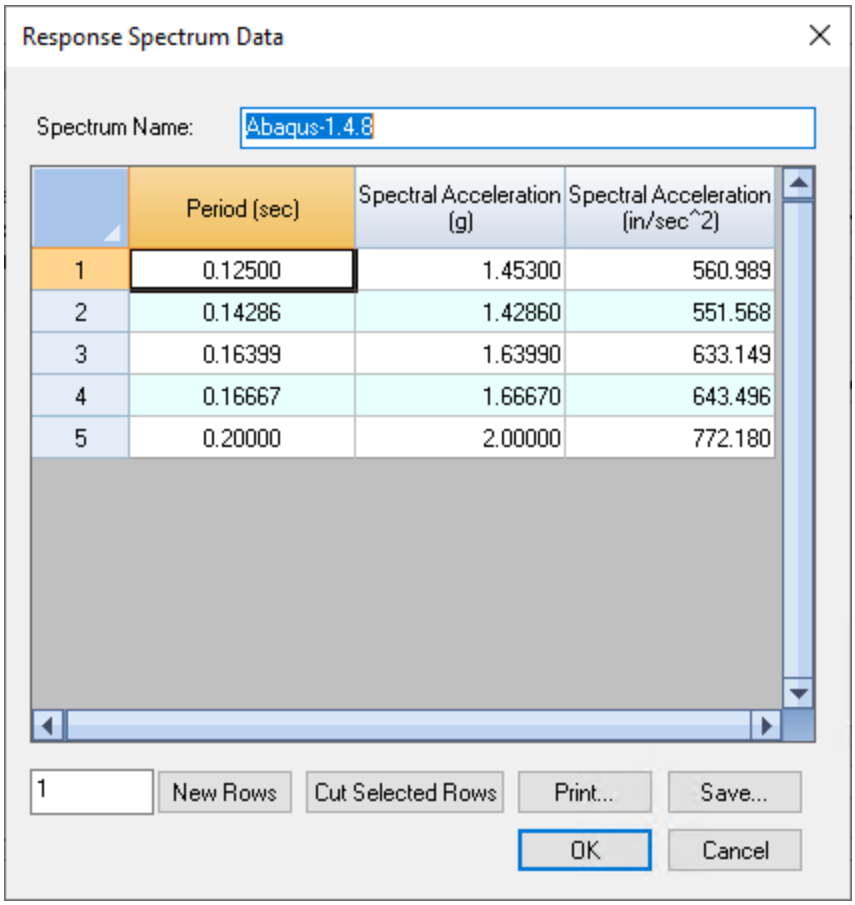
Create > Generate Loads > Response Spectra Library prompts you with the dialog below.



It allows you to define spectra for current and future projects. You can then use one or more spectra in Analysis > Response Spectrum Analysis.

You may view/modify a user-defined spectrum by double clicking the spectrum.

© 2026 ENERCALC, LLC

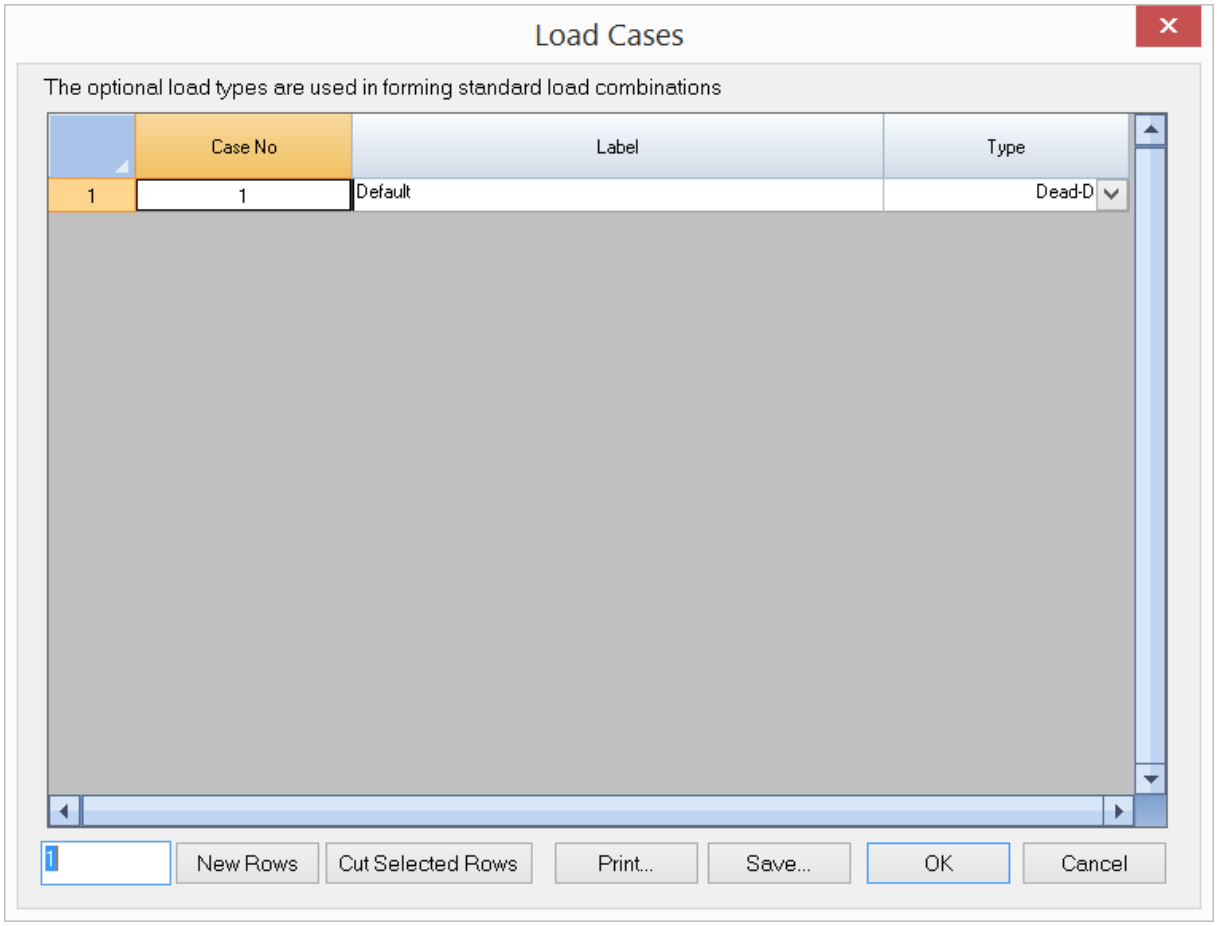


The first spectrum can not be edited or deleted. Spectra generated based on building codes cannot be edited but can be deleted.

© 2026 ENERCALC, LLC

Load Cases

Create > Load Cases prompts you with the following dialog.



It allows you to define load cases to be used for loads and load combinations. A number is assigned to each load case automatically by the program. You may assign a label with 127 maximum characters to each load case for easy identification. Duplicate labels in load cases are not allowed. A load type specifies the characteristics of the load case. Examples are DEAD, LIVE, WIND, EARTHQUAKE. They are used to generate standard load combinations in Create > Load Combinations.

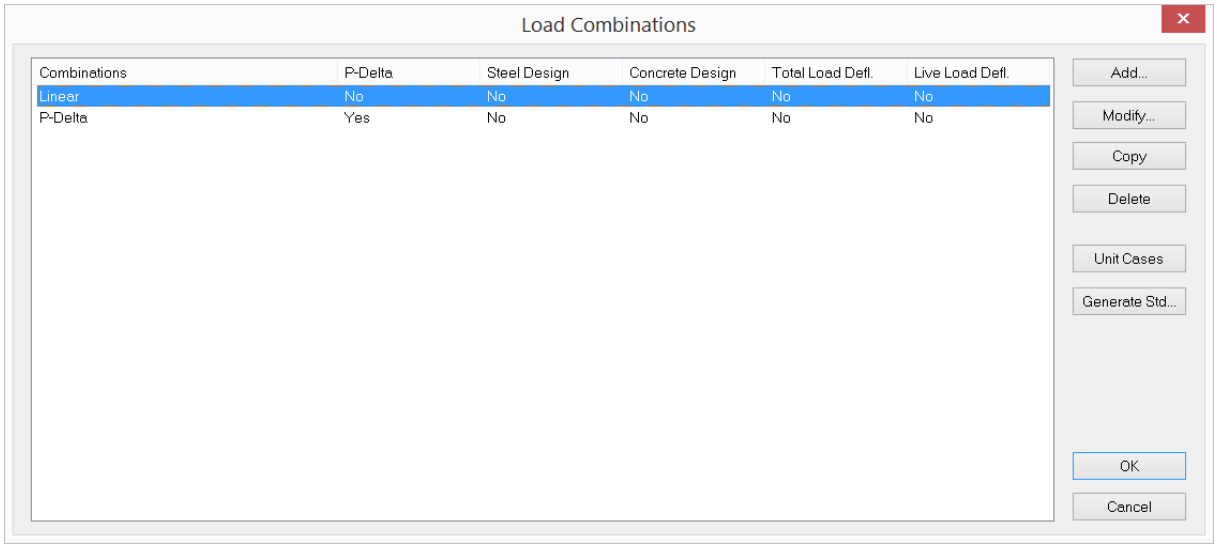
You may add one or more load cases by clicking the “New Rows” button. You may also print all load cases in the list by clicking the “Print” button.

The program always has a default load case labeled “Default”. You may not delete this load case or change its label. You may however change its type.

© 2026 ENERCALC, LLC

Load Combinations

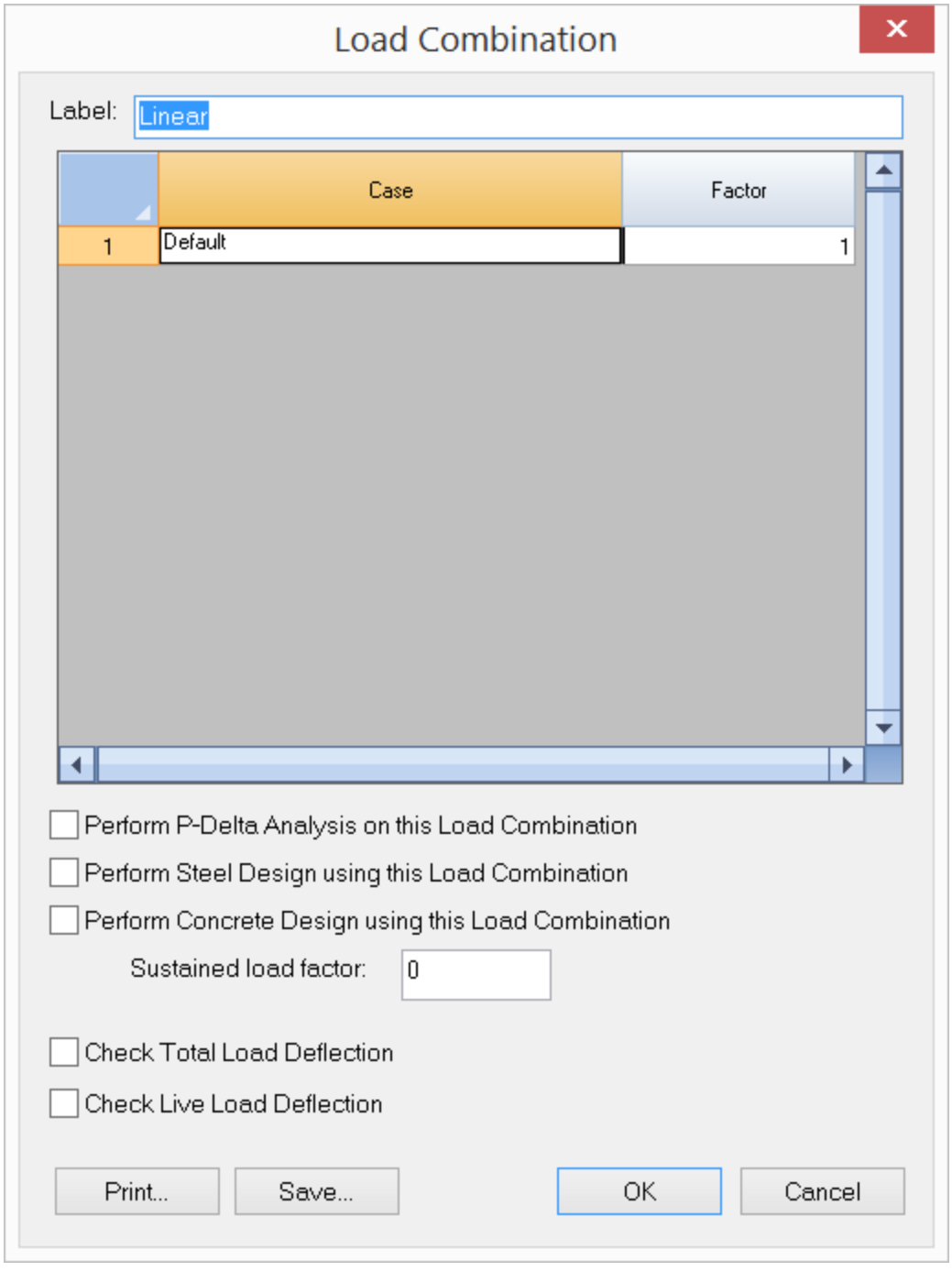
Create > Load Combinations prompts you with the following dialog.



It allows you to define combinations of existing load cases in the model. The program solves for load combinations but not for load cases. You may assign a label with 127 maximum characters to each load combination for easy identification. Duplicate labels in load combinations are not allowed.

© 2026 ENERCALC, LLC

You may add one load combination by clicking the “Add” button. You may then define the new load combination in the following dialog.



The definition includes a label with 127 maximum characters, a load factor for each load case, and a P-Delta flag. A load factor of zero excludes the respective load case from participating in the load combination. You may print all load cases and their corresponding load factors in the list by clicking the “Print” button.

© 2026 ENERCALC, LLC

If you need to design concrete beams, columns and/or plates, check “Perform Concrete Design using this Load Combination” for each load combination that is appropriate for concrete design.

When concrete columns are to be designed, the program needs to calculate the infamous β_d value. The easiest way to facilitate this calculation is to create a load combination specifically for the purpose of defining the *sustained loads*. (It might even be helpful to name this load combination "Sustained".)

In some models, this sustained load combination might just consist of 1.0 times the dead load case. In other situations, the sustained load combination might consist of 1.0 times the dead load case and 1.0 times a self-weight load case. In still other cases, the sustained load combination might consist of 1.0 times the dead load case and some percentage of the live load case. It depends entirely on how the load cases were set up for the given model.

Then, a sustained load factor must also be entered. This can only be a single factor, but it must be representative of the constituent load cases (like a weighted average), because this one factor will be applied to the entire sustained load combination to calculate the maximum factored sustained axial load.

Finally, this sustained load combination must be designated in Concrete Design > RC Model Design Criteria before performing concrete column design.

If you need to design steel members, check or uncheck “Perform Steel Design using this Load Combination”. If you need to use the load combination to check total or live load deflection, check or uncheck appropriate boxes.

You may modify, copy, or delete a load combination by clicking the “Modify”, “Copy”, or “Delete” button. You may also create a load combination for every load case with a unit load factor for the load case but zeros for the rest of the load cases. To do that, click the “Unit Cases” button.

You may also generate standard load combinations based on design codes such as ACI 318-02/05/08/11/14 by clicking the button “Generate Std”.

There must be at least one load combination in a model.

© 2026 ENERCALC, LLC

Line Select

Create > Line Select allows you to line-select elements by clicking-dragging the left mouse button. The line-selection selects or unselects a group of entities that the drawn line intersects.

Select versus Reverse Select

By default, the selection mode is REVERSE SELECT, that is, entities picked will be selected if they are currently unselected and will be unselected if they are currently selected. However, the selection mode will be SELECT if CTRL is pressed while selecting, that is, entities picked will always be selected.

The selection of nodes and elements is an important activity in the program. Most of the commands apply only to selected nodes or elements. For this reason, many selection methods are provided in the program and are explained in the following sections.

Window/Point Select

Create > Window/Point Select allows you to window-select or point-select nodes and elements by clicking or clicking-dragging the left mouse button. The Window/Point Select command is the default selection command in the program.

Point versus Window

The point-selection selects or unselects at most one node or element. The window-selection selects or unselects a group of nodes or elements within a rectangular selection window.

Window Drawn Right-to-Left versus Left-to-Right

If the selection window is drawn from left to right, then only entities that are entirely within the selection rectangle will be selected/unselected.

If the selection window is drawn from right to left, then all entities that are crossed by the selection rectangle will also be selected or unselected.

Select versus Reverse Select

By default, the selection mode is REVERSE SELECT, that is, entities picked will be selected if they are currently unselected and will be unselected if they are currently selected. However, the selection mode will be SELECT if CTRL is pressed while selecting, that is, entities picked will always be selected.

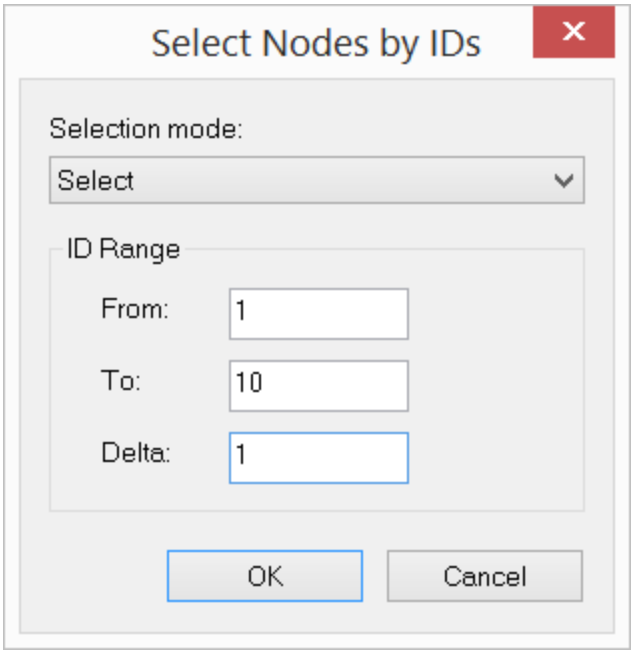
The selection of nodes and elements is an important activity in the program. Most of the commands apply only to selected nodes or elements. For this reason, many selection methods are provided in the program and are explained in the following sections.

© 2026 ENERCALC, LLC

Select

Create > Select offers a variety of quick options for selecting specific nodes, members or shells.

Create > Select > Select Nodes prompts you with the following dialog.



It allows you to select nodes by specifying a range of node IDs.

Three selection modes are provided: “Select”, “Unselect”, “Reverse Select”. The “Select” mode will select nodes. The “Unselect” mode will unselect nodes. The “Reverse Select” mode will select the unselected nodes and unselect the selected nodes.

Create > Select > Select Members prompts you with a dialog to select members by specifying a range of member IDs.

Three selection modes are provided: “Select”, “Unselect”, “Reverse Select”. The “Select” mode will select members. The “Unselect” mode will unselect members. The “Reverse Select” mode will select the unselected members and unselect the selected members.

Create > Select > Select Shells prompts you with a dialog to select shells by specifying a range of shell IDs.

Three selection modes are provided: “Select”, “Unselect”, “Reverse Select”. The “Select” mode will select shells. The “Unselect” mode will unselect shells. The “Reverse Select” mode will select the unselected shells and unselect the selected shells.

© 2026 ENERCALC, LLC

Create > Select > Select Bricks prompts you with a dialog to select bricks by specifying a range of brick IDs.

Three selection modes are provided: “Select”, “Unselect”, “Reverse Select”. The “Select” mode will select bricks. The “Unselect” mode will unselect bricks. The “Reverse Select” mode will select the unselected bricks and unselect the selected bricks.

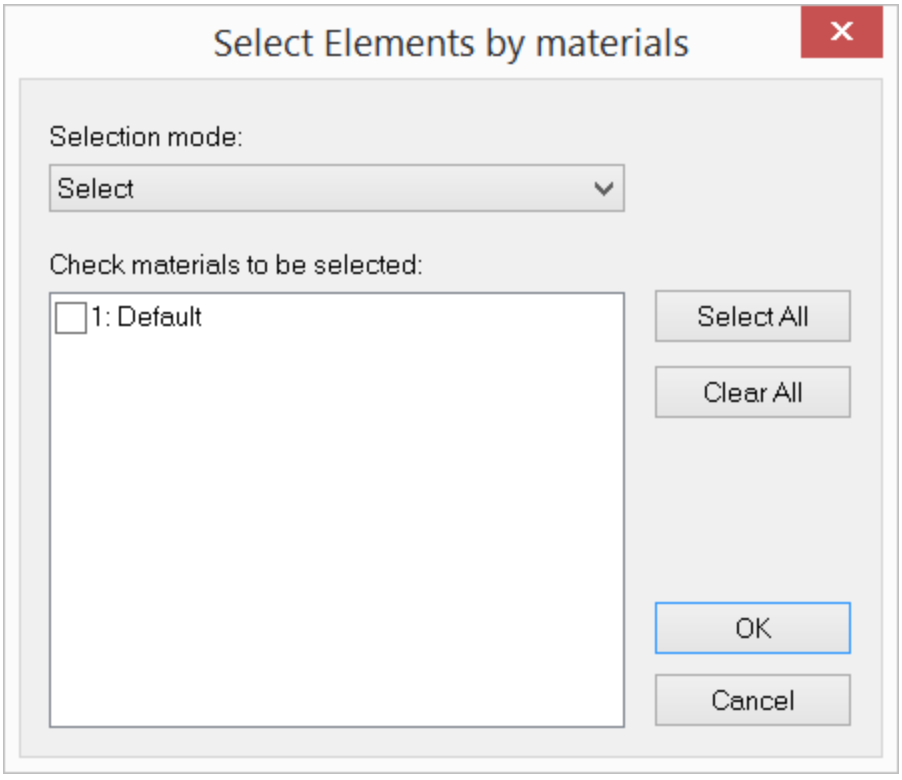
Create > Select > Select All selects all nodes and elements. You may use this command by pressing CTRL+A.

Create > Select > Unselect All unselects all nodes and elements. You may use this command by pressing ESC. If you are in the middle of another command such as zooming, press ESC twice to unselect all.

Select by Properties

Create > Select by Properties offers a variety of quick options for selecting specific nodes, members or shells by their properties.

Create > Select by Properties > Select by Materials prompts you with the following dialog.

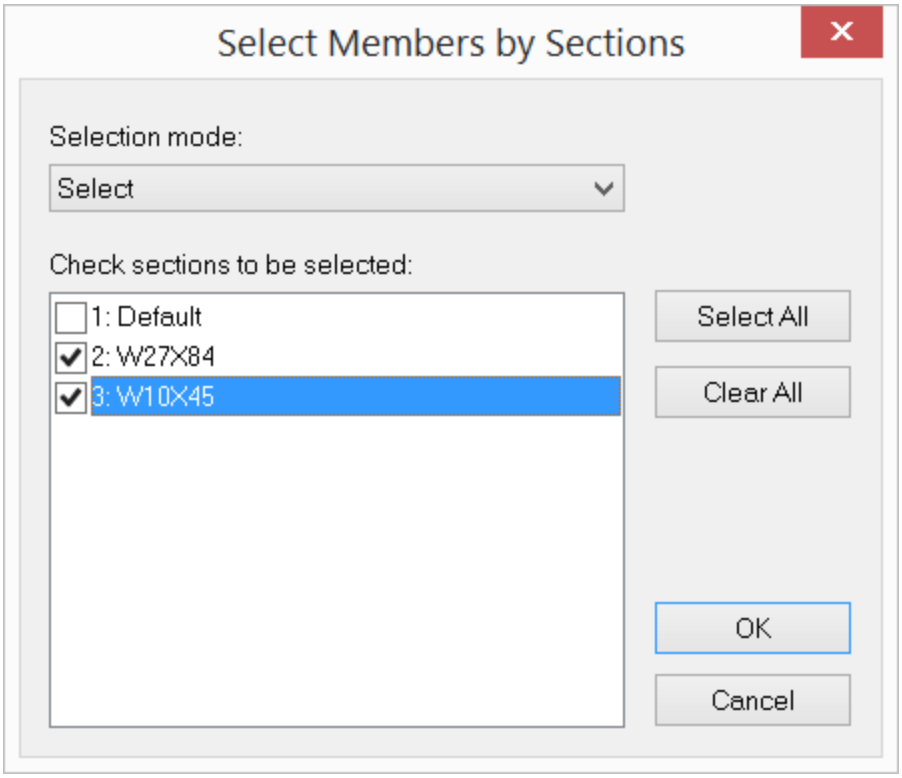


It allows you to select/unselect elements that use the specified materials.

Three selection modes are provided: “Select”, “Unselect”, “Reverse Select”. The “Select” mode will select elements. The “Unselect” mode will unselect elements. The “Reverse Select” mode will select the unselected elements and unselect the selected elements.

© 2026 ENERCALC, LLC

Create > Select by Properties > Select by Member Sections prompts you with the following dialog.

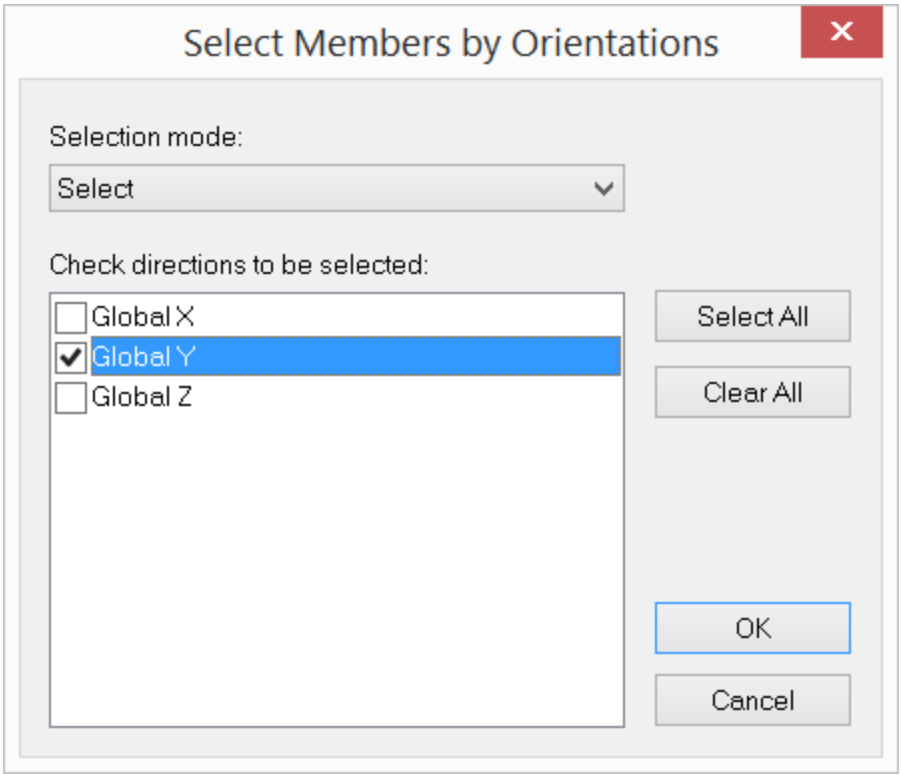


It allows you to select/unselect members that use the specified sections.

Three selection modes are provided: “Select”, “Unselect”, “Reverse Select”. The “Select” mode will select elements. The “Unselect” mode will unselect elements. The “Reverse Select” mode will select the unselected elements and unselect the selected elements.

© 2026 ENERCALC, LLC

Create > Select by Properties > Select by Member Orientations prompts you with the following dialog.

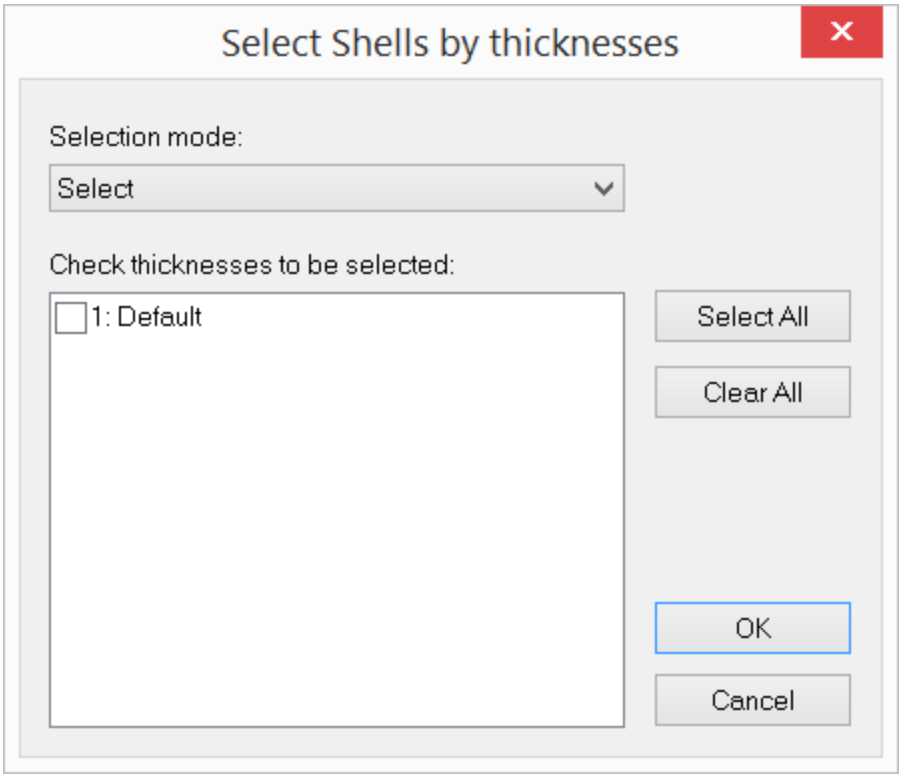


It allows you to select/unselect members based on their orientations to the three global axes. For example, you may select/unselect all vertical columns by checking the global Y direction. The selection modes are similar to the ones used in previous sections and are not repeated here.

Create > Select by Properties > Select Tension/Compression Only Members allow you to select all tension-only or compression-only members.

© 2026 ENERCALC, LLC

Create > Select by Properties > Select by Shell Thicknesses prompt you with the following dialog.



It is similar to the one used in View > Select by Properties > Member Sections. View > Select by Properties > Shell Thicknesses applies to shells only. Three selection modes are similar to the ones used in the previous section and are not repeated here.

Create > Select by Properties > Select Orphaned Nodes selects all orphaned nodes.

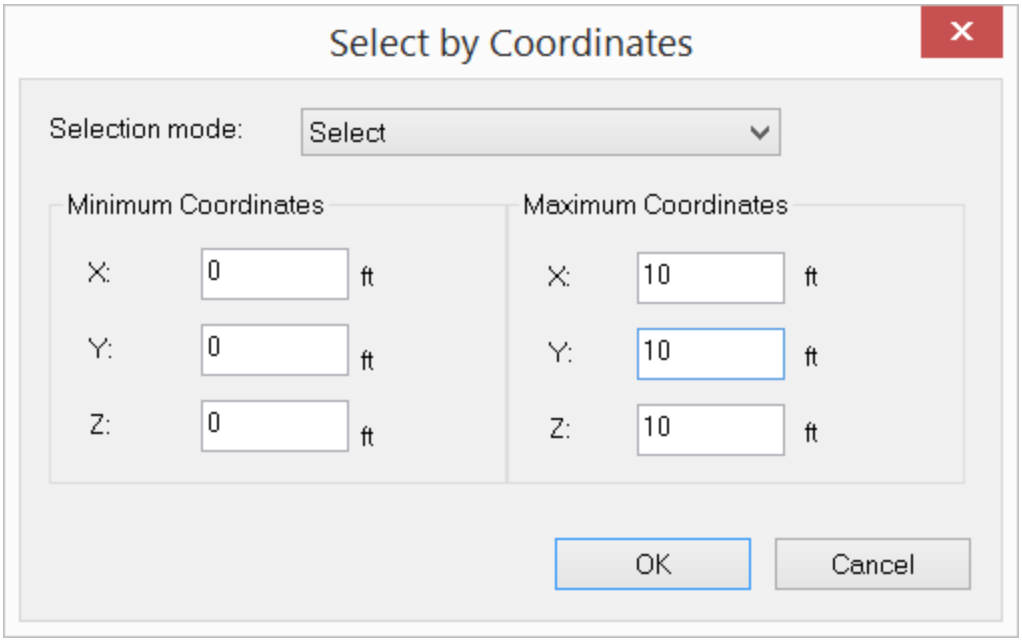
Create > Select by Properties > Select Elements with Self Weight Excluded selects all members whose self weight has been excluded.

Create > Select by Properties > Select Inactive Elements selects all inactive elements.

Create > Select by Properties > Select Elements by Nodes selects elements that are connected to currently selected nodes.

© 2026 ENERCALC, LLC

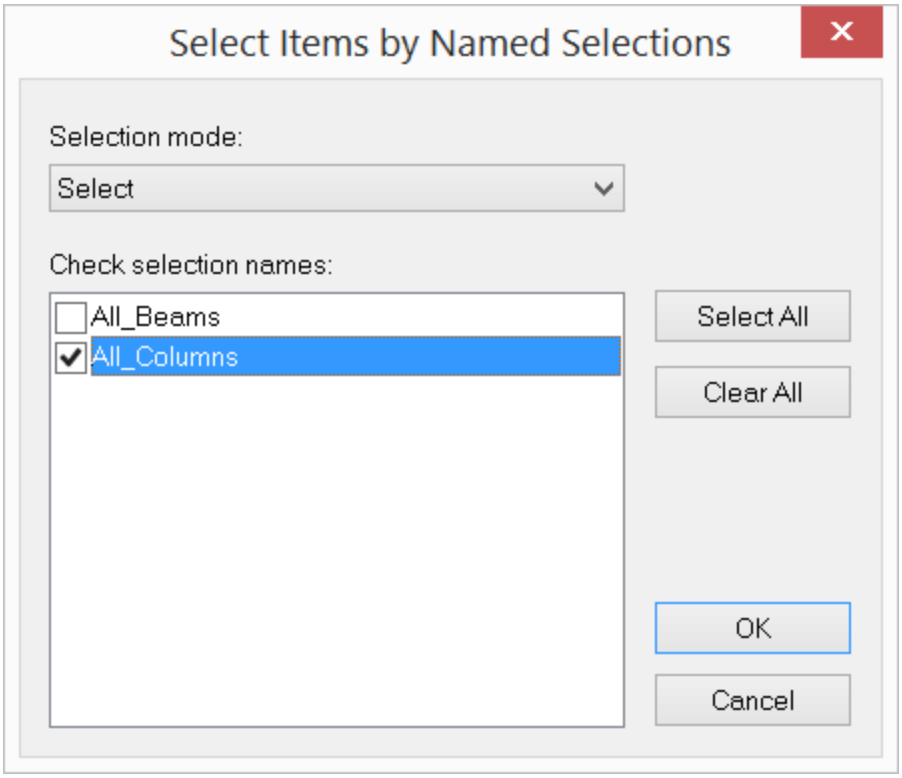
Create > Select by Properties > Select by Coordinates prompts you with the following dialog.



It allows you to select/unselect nodes and elements based on nodal coordinates. Nodes are selected/unselected if their coordinates are within the boundary of the minimum and maximum coordinates. Elements are selected/unselected if coordinates of their nodes are within the boundary of minimum and maximum coordinates. The selection modes are similar to the ones used in previous sections and are not repeated here.

© 2026 ENERCALC, LLC

Create > Select by Properties > Select by Selection Names allows you to select/unselect nodes and elements based on saved named selections. You may assign (or save) named selections from the Create ribbon.



Create > Select by Properties > Select by RC Beam Design Criteria allows you to select beam members based on their design criteria.

Create > Select by Properties > Select by RC Column Design Criteria allows you to select column members based on their design criteria.

Create > Select by Properties > Select by RC Plate Design Criteria allows you to select plate elements based on their design criteria.

Create > Select by Properties > Select by Steel Design Criteria allows you to select members based on a steel design criteria.

Create > Select by Properties > Select by Unity Check Ratios allows you to select members based on their unity check ratio.

Invert Selection

Create > Invert Selection inverts the current selection state of all nodes and elements.

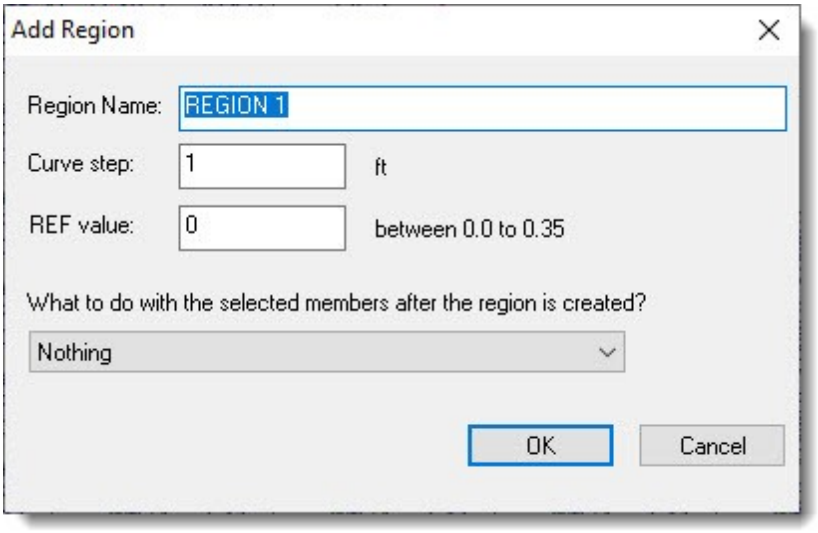
© 2026 ENERCALC, LLC

Advanced Meshing

The Advanced Meshing ribbon provides commands to create complex meshes. It offers the ability to incorporate multiple regions of varying mesh density, holes, control points and control lines (called "Trees").

Add Region

Advanced Meshing > Add Region displays the following dialog.

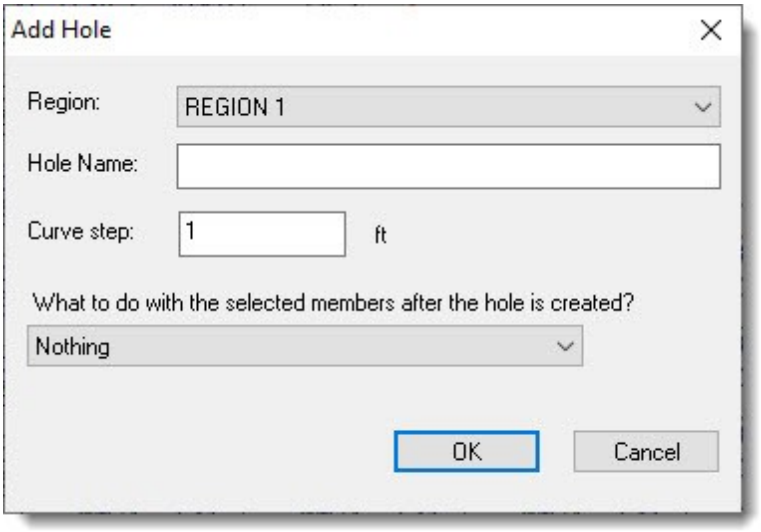


It allows you to add a region to mesh model. You must first select beams as the boundary (closed polygon) of the region. You have the option to deactivate or delete the selected beams after the mesh region is created. This ensures that the selected beams will not be part of the structural model.

© 2026 ENERCALC, LLC

Add Hole

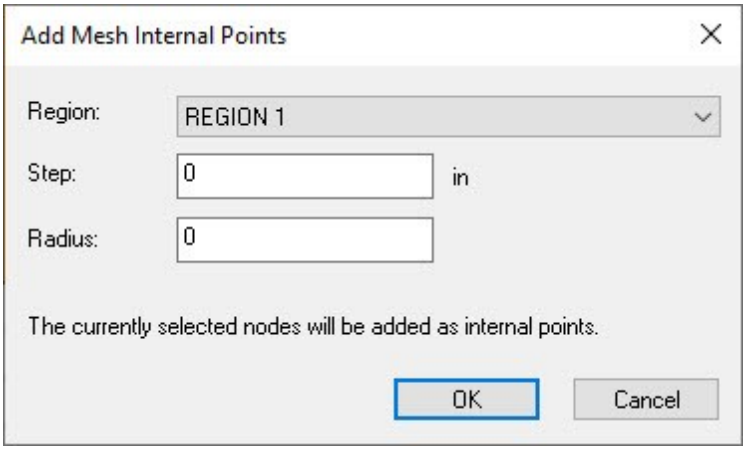
Advanced Meshing > Add Hole displays the following dialog.



It allows you to add a hole to a mesh region. You must first select beams as the boundary (closed polygon) of the hole. You have the option to deactivate or delete the selected beams after the hole is created. This ensures that the selected beams will not be part of the structural model.

Add Internal Points

Advanced Meshing > Add Internal Points displays the following dialog.

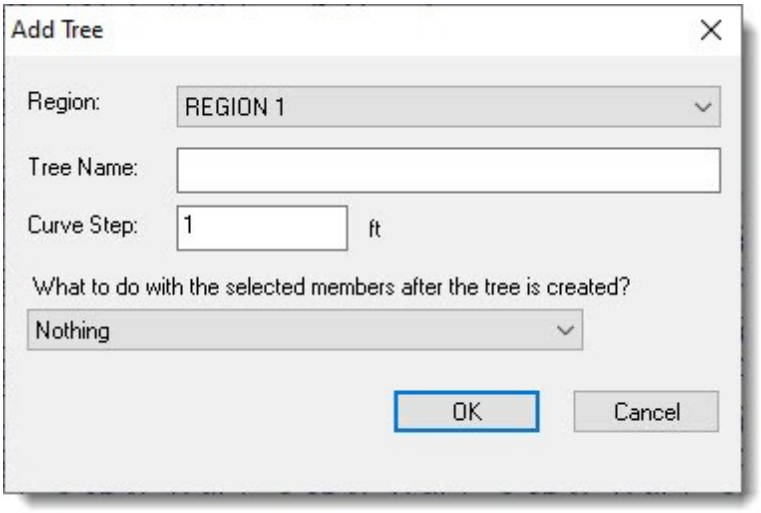


It allows you to add internal points to a mesh region. You must first select existing nodes as the location of the internal points in the mesh region.

© 2026 ENERCALC, LLC

Add Tree

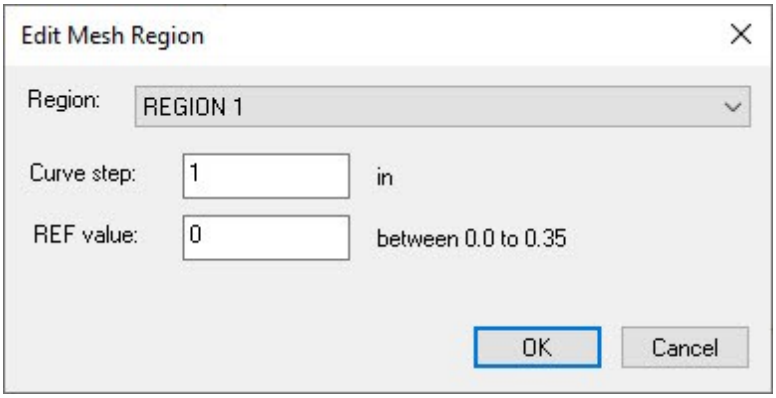
Advanced Meshing > Add Tree displays the following dialog.



It allows you to add a tree to a mesh region. You must first select beams as the branches of the tree. Please note that disconnected branches are not allowed. You have the option to deactivate or delete the selected beams after the tree is created. This ensures that the selected beams will not be part of the structural model.

Edit Region

Advanced Meshing > Edit Region displays the following dialog.

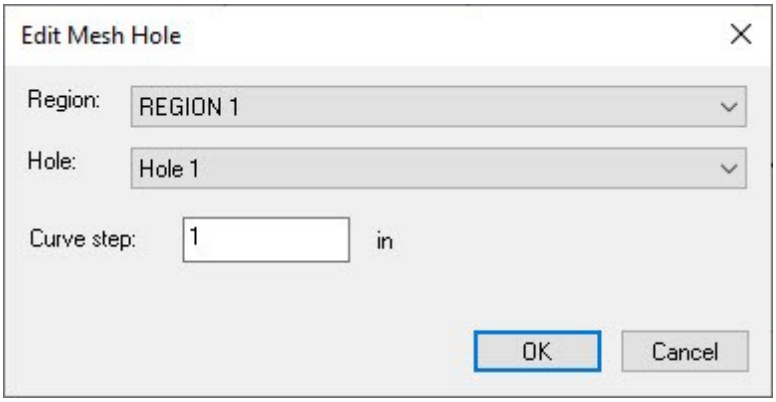


It allows you to edit an existing mesh region.

© 2026 ENERCALC, LLC

Edit Hole

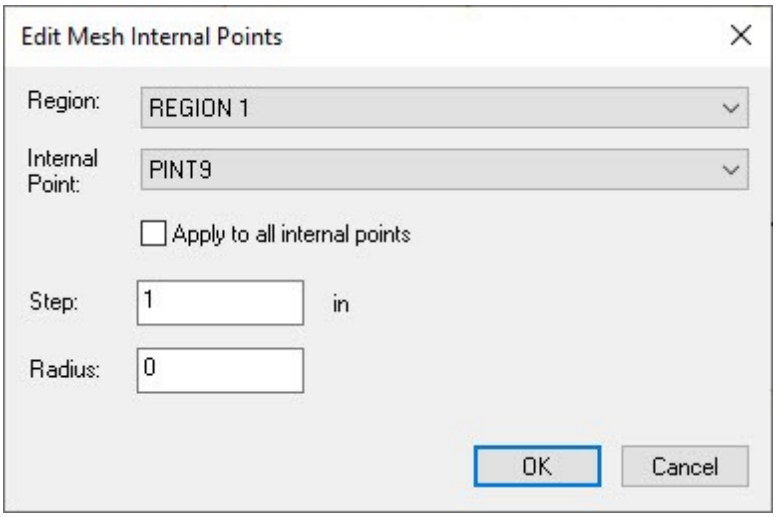
Advanced Meshing > Edit Hole the following dialog



It allows you to edit an existing hole in a region.

Edit Internal Points

Advanced Meshing > Edit Internal Points displays the following dialog.

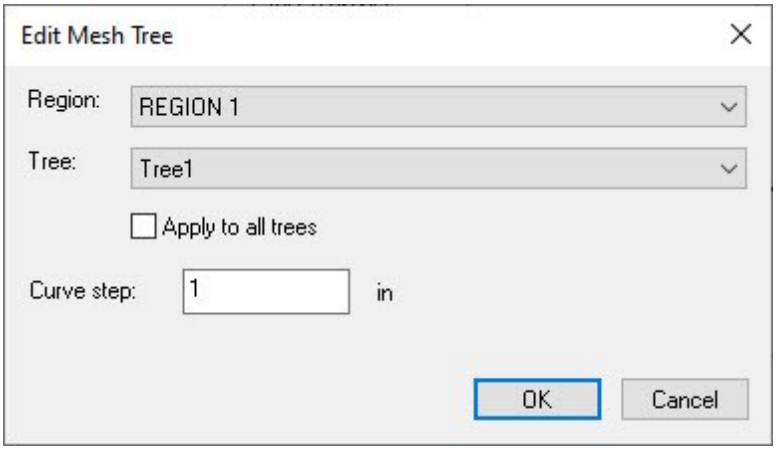


It allows you to edit internal points in a region. You have the option to edit a single internal point or all internal points within a region.

© 2026 ENERCALC, LLC

Edit Tree

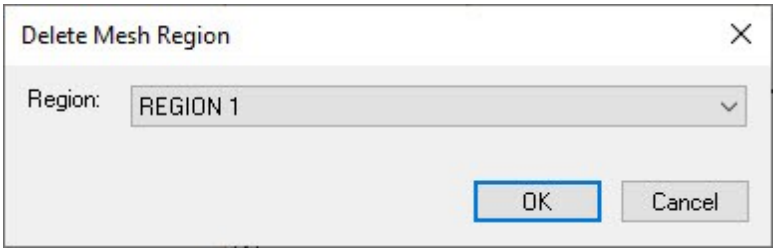
Advanced Meshing > Edit Tree displays the following dialog.



It allows you to edit an existing tree in a region.

Delete Region

Advanced Meshing > Delete Region displays the following dialog.

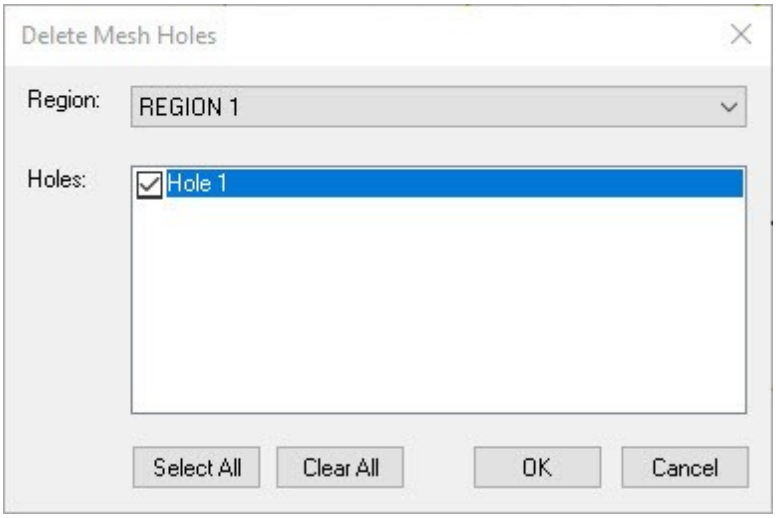


It allows you to delete an existing region.

© 2026 ENERCALC, LLC

Delete Hole

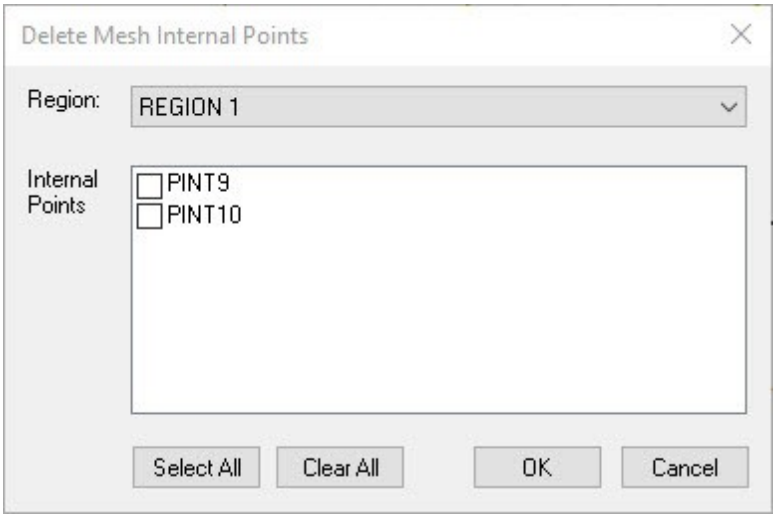
Advanced Meshing > Delete Hole displays the following dialog.



It allows you to delete one or more holes in a region.

Delete Internal Points

Advanced Meshing > Delete Internal Points displays the following dialog.

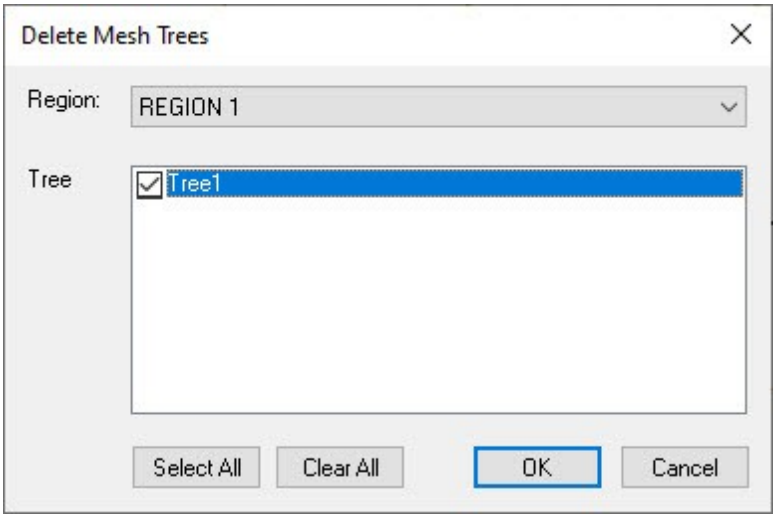


It allows you to delete one or more internal points in a region.

© 2026 ENERCALC, LLC

Delete Tree

Advanced Meshing > Delete Tree displays the following dialog.



It allows you to delete one or more trees in a region.

Clear Mesh Model

Advanced Meshing > Clear Mesh Model allows you to clear the mesh model. This will delete all regions and its dependents such as holes, internal points and trees.

Load Mesh Model from File

Advanced Meshing > Load Mesh Model from File allows you to load a previously saved mesh model file (*.SUR) and therefore replace the existing mesh model.

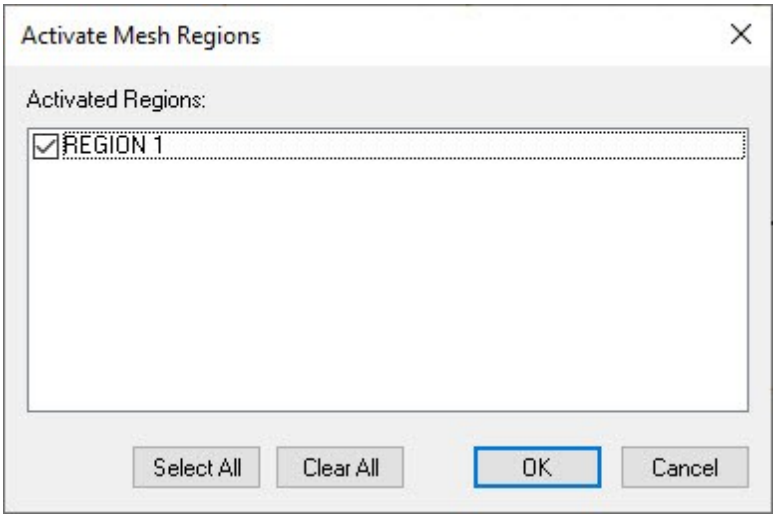
Save Mesh Model to File

Advanced Meshing > Save Mesh Model to File allows you to save the existing mesh model to a file (*.SUR). The saved mesh file can be edited, loaded later.

© 2026 ENERCALC, LLC

Activate Regions

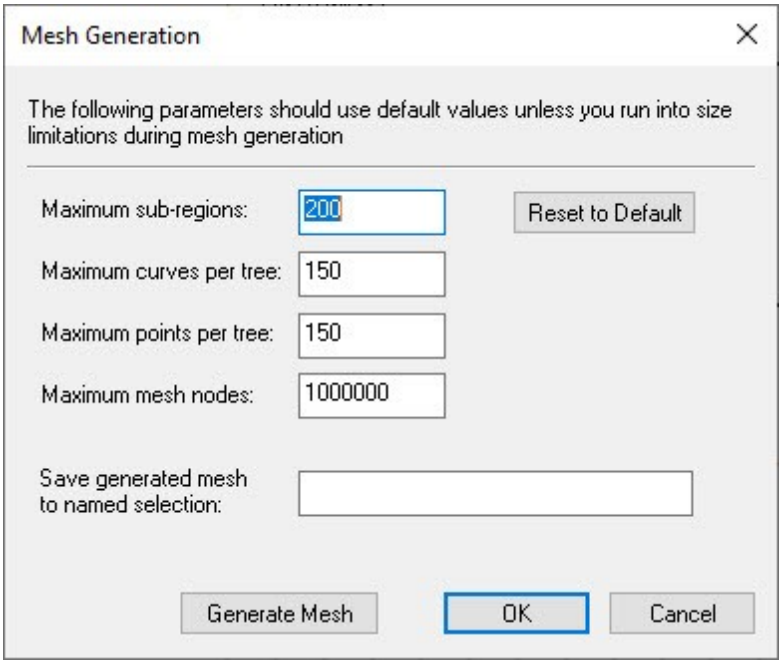
Advanced Meshing > Activate Regions displays the following dialog.



It allows you to deactivate/activate one or more regions before generating mesh.

Generate Mesh

Advanced Meshing > Generate Mesh displays the following dialog.



It allows you to generate mesh. You have the option to save the generated mesh to a named selection.

© 2026 ENERCALC, LLC

Generate Mesh from File

Advanced Meshing > Generate Mesh from File allows you to generate mesh from a previously saved mesh model file (*.SUR).

View Mesh Model

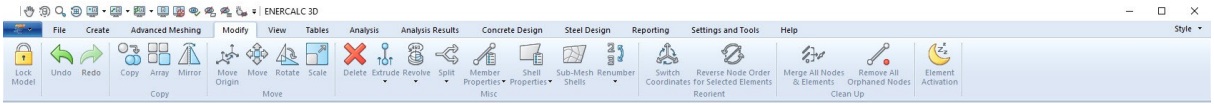
Advanced Meshing > View Mesh Model shows or hides the mesh model.

Annotate Mesh Model

Advanced Meshing > Annotate Mesh Model allows you to annotate mesh model objects. Also see Display Options on the Quick Access Toolbar.

Modify

The Modify ribbon provides commands to edit or modify the model.



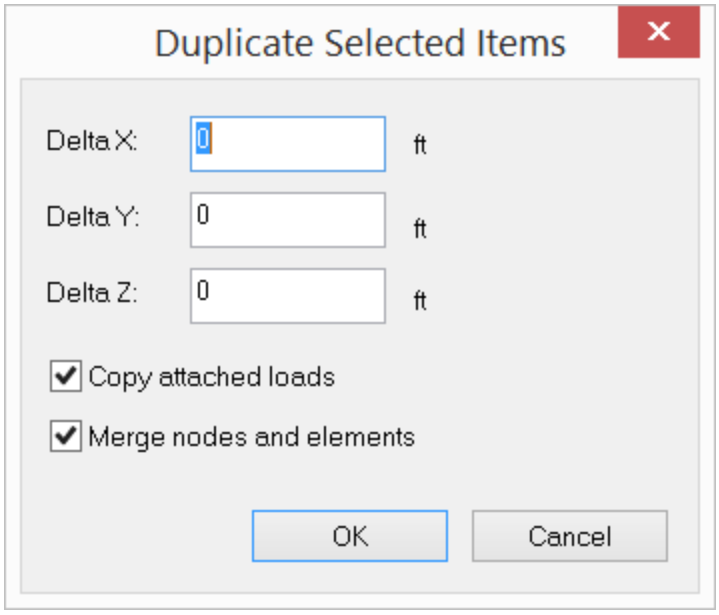
Lock Model

Modify > Lock Model locks the model so that you cannot modify it. You may still access non-structural commands such as zooming and panning while the model is locked. The model may be automatically locked after an analysis is performed successfully. To do that, just click Settings & Tools > Preferences.

© 2026 ENERCALC, LLC

Copy

Modify > Copy prompts you with the following dialog.

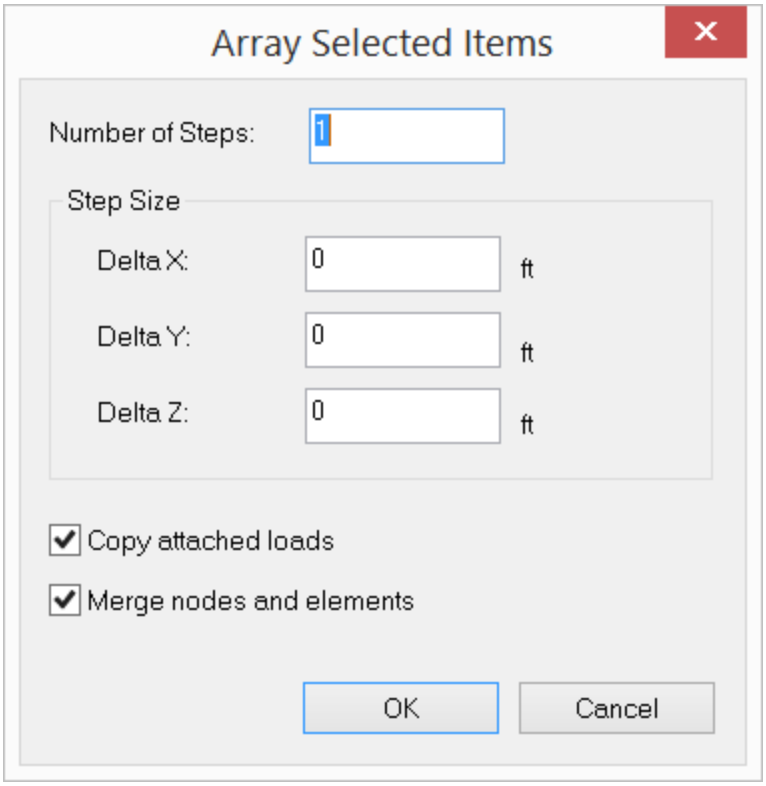


It allows you make one copy of the selected parts of a model to a different location. You may specify copy distances along the X, Y or Z directions. Nodal or element dependents (except loads) are copied together with their parents automatically. For example, when a member is copied, moment releases on that member are copied also. You have the option to copy the loads attached to the selected nodes or elements. You have the option to automatically merge nodes and elements after copying. You should check this option unless duplicate nodes are explicitly permitted.

© 2026 ENERCALC, LLC

Array

Modify > Array prompts you with the following dialog.

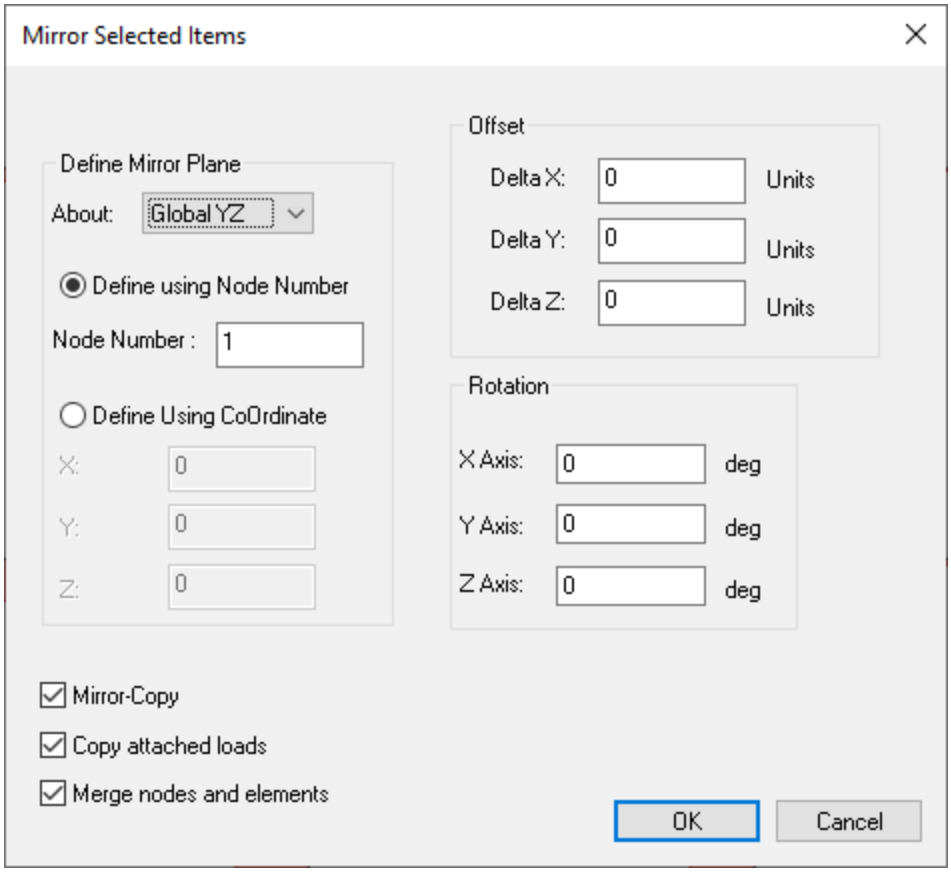


It allows you make one or more copy of the selected parts of a model to a different location. You may specify the step size along the X, Y or Z directions. Nodal or element dependents (except loads) are copied together with their parents automatically. For example, when a member is copied, moment releases on that member are copied also. You have the option to copy the loads attached to the selected nodes or elements. You have the option to automatically merge nodes and elements after copying. You should check this option unless duplicate nodes are explicitly permitted.

© 2026 ENERCALC, LLC

Mirror

Modify > Mirror prompts you with the following dialog.

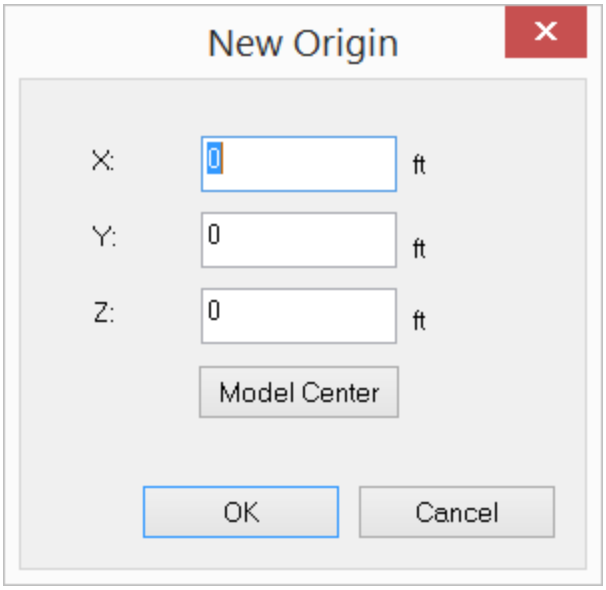


It allows you to mirror the selected parts of a model to a different location by defining a mirror plane. Nodal or element dependents (except loads) are mirrored together with their parents automatically. For example, when a member is mirrored, moment releases on that member are mirrored also. You have the option to copy the loads attached to the selected nodes or elements. You have the option to automatically merge nodes and elements after copying. You should check this option unless duplicate nodes are explicitly permitted.

© 2026 ENERCALC, LLC

Move Origin

Modify > Move Origin prompts you with the following dialog.

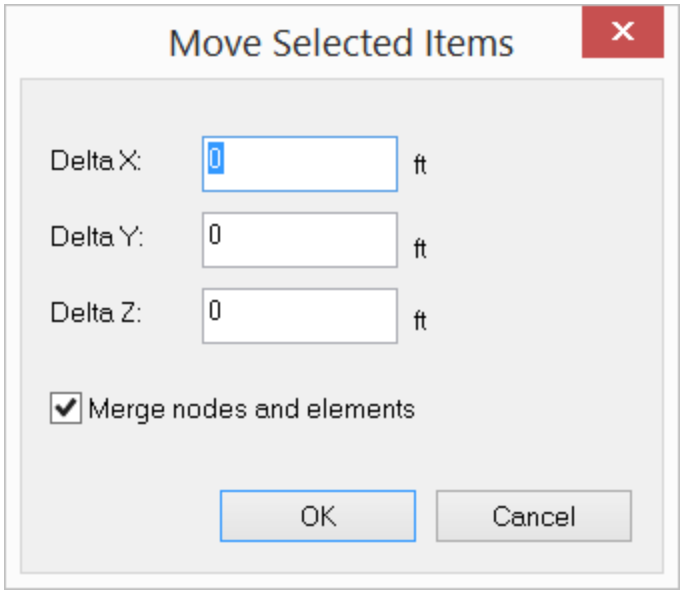


It allows you to reset the model origin. In particular, the origin may be set at the current model center. This allows you to center the model so its view may be rotated more smoothly.

© 2026 ENERCALC, LLC

Move

Modify > Move prompts you with the following dialog.



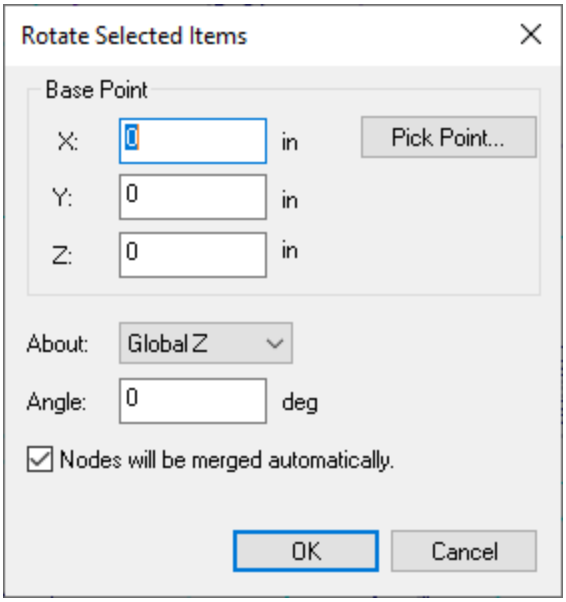
It lets you move selected parts of a model to a different location. You may specify move distances along the X, Y or Z directions. Nodal or element dependents such as loads are moved together with selected nodes or elements automatically. You have the option to automatically merge nodes and elements after moving. You should check this option unless duplicate nodes are explicitly permitted.

Note: When using this command, be sure to select the intended entities AND THEIR NODES, before executing the command. If the entities are selected without their nodes, the command will not work.

© 2026 ENERCALC, LLC

Rotate

Modify > Rotate prompts you with the following dialog.



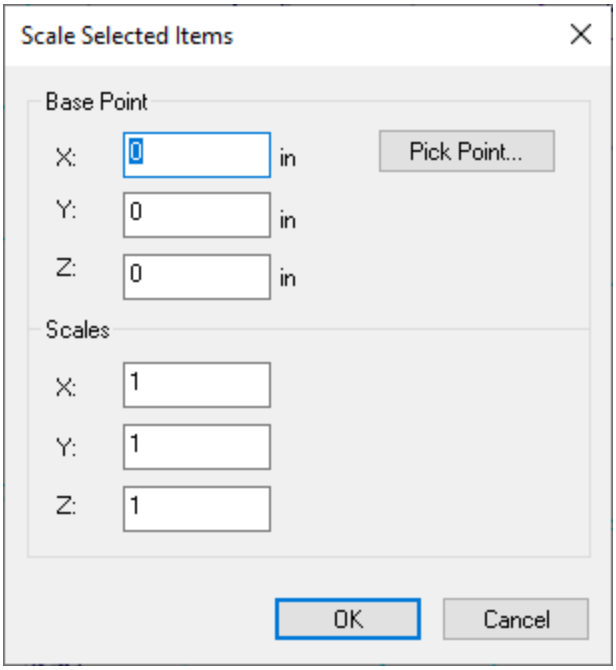
It lets you rotate selected parts of a model by an angle about one of the global axes. Nodal or element dependents such as loads are moved together with the elements. You have the option to automatically merge nodes and elements after rotating. You should check this option unless duplicate nodes are explicitly permitted.

Note: When using this command, be sure to select the intended entities AND THEIR NODES, before executing the command. If the entities are selected without their nodes, the command will not work.

© 2026 ENERCALC, LLC

Scale

Modify > Scale prompts you with the following dialog.



It lets you scale selected parts of a model in the X, Y or Z directions. You may specify the coordinates of a base point and scales for the three global directions.

The following formula is used to perform the scaling in the program.

$X_{new} = X_{base} + (X_{old} - X_{base}) * scale$

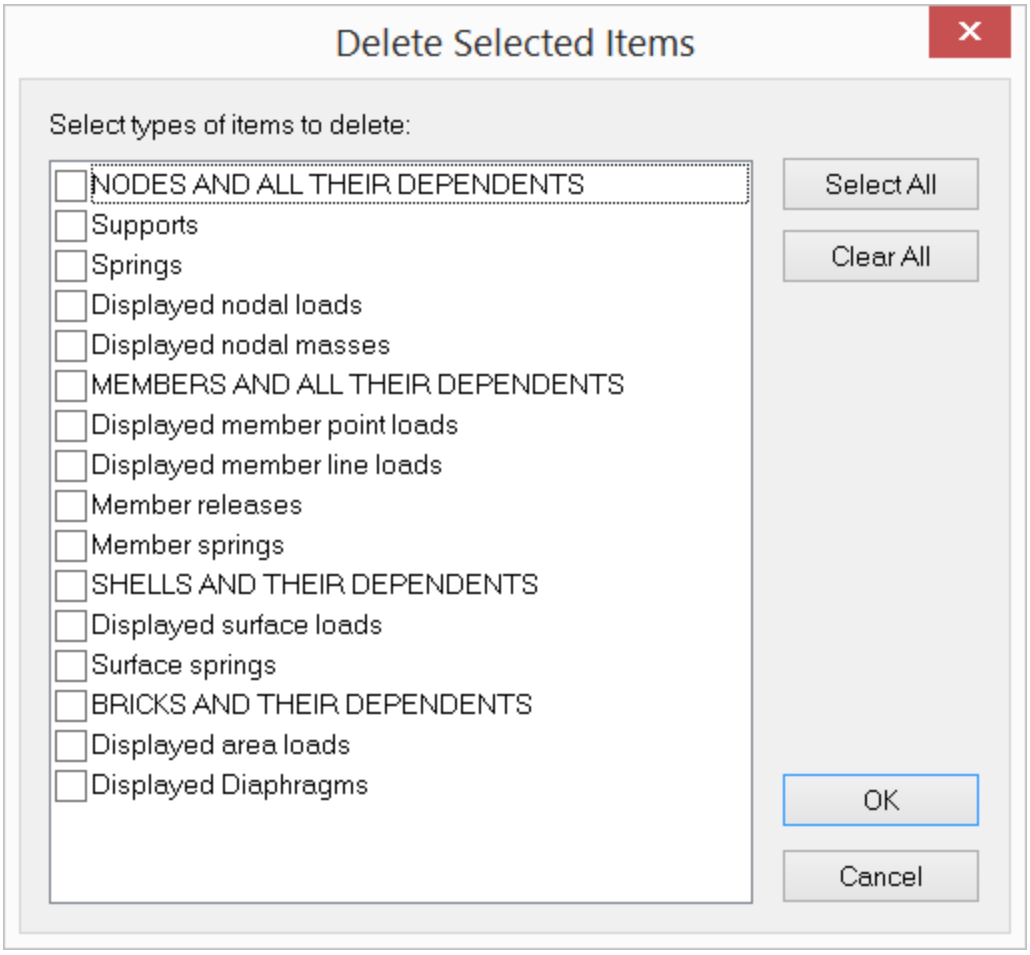
Where X_{new} represents the nodal coordinates after scaling, X_{old} represents the nodal coordinates before scaling and X_{base} represents coordinates of the base point.

Note: When using this command, be sure to select the intended entities AND THEIR NODES, before executing the command. If the entities are selected without their nodes, the command will not work.

© 2026 ENERCALC, LLC

Delete

Modify > Delete prompts you with the following dialog.



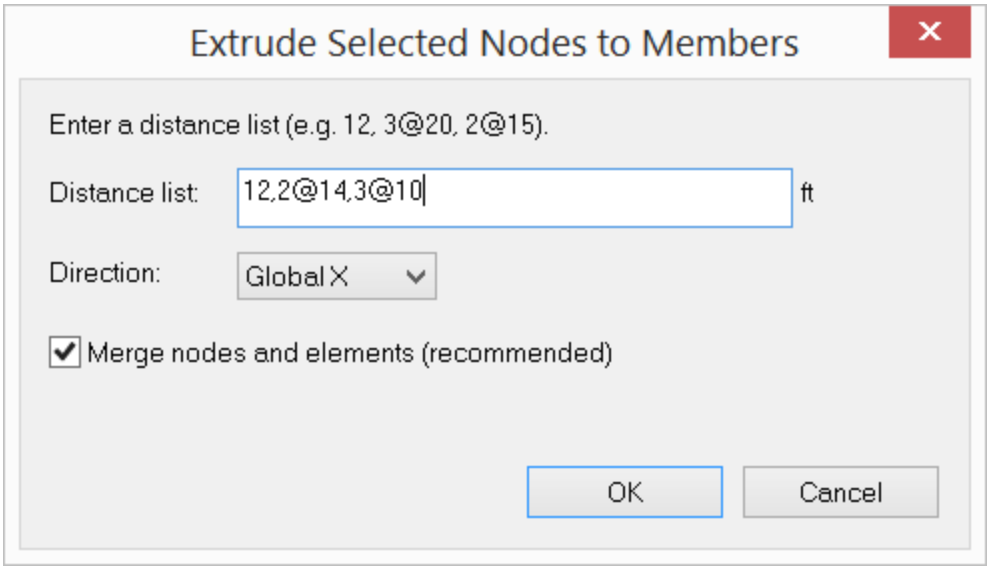
It allows you to delete selected nodes or elements or their dependents. Loads are deleted based on their visibilities in the model view. Dependents such as loads will be deleted if their parent nodes or elements are deleted.

Extrude

Modify > Extrude provides various methods of creating entities by starting with a selected set and extruding them by specified distances.

© 2026 ENERCALC, LLC

Modify > Extrude > Extrude Nodes to Members prompts you with the following dialog.



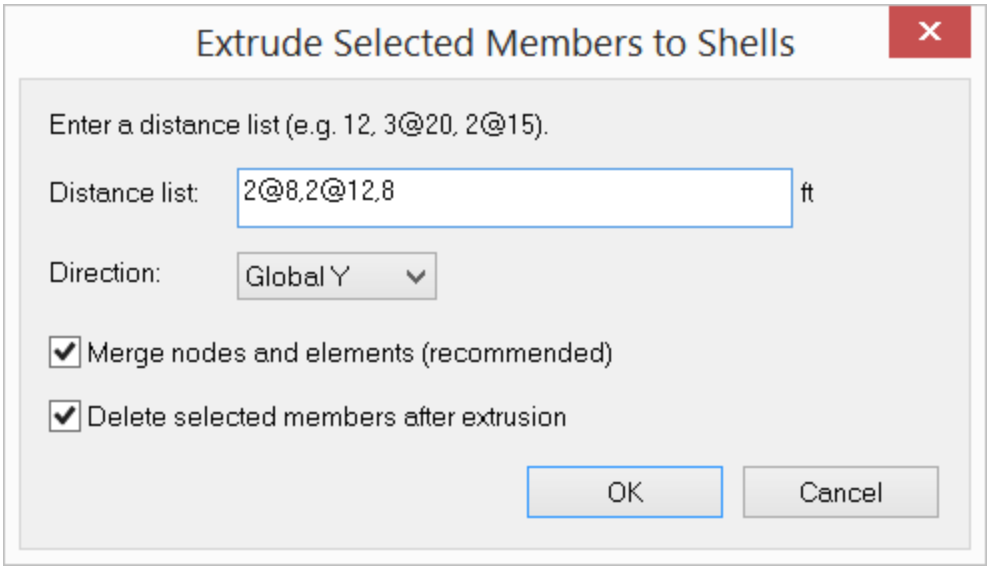
It generates a series of members by extruding selected nodes along a global direction.

You may specify a distance list and an extrusion direction for the generation of beams. The distance list is a comma separated list that specifies multiple distances. For example, a distance list of “12,2@14,3@10” will generate distances of 12, 14, 14, 10, 10 and 10 in length units. You have the option to automatically merge nodes and elements after extrusion. You should check this option unless duplicate nodes are explicitly permitted.

The following screen capture illustrates the members that are generated by extruding one node (the first node) using the input from the dialog above.

© 2026 ENERCALC, LLC

Modify > Extrude > Extrude Members to Shells prompts you with the following dialog.

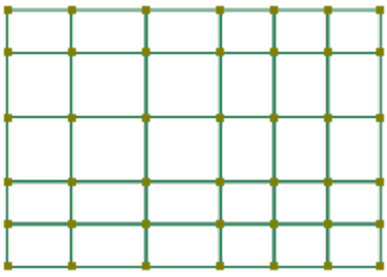


It generates a series of shells by extruding selected members along a global direction. You may specify a distance list and an extrusion direction for the generation of shells. The extrusion direction must not be parallel to the selected members. You have the option to automatically merge nodes and elements after extrusion. You should check this option unless duplicate nodes are explicitly permitted. You also have the option to automatically delete selected members after the extrusion.

The input in the above dialog is applied to the members in the screen capture below...

members used for extruding

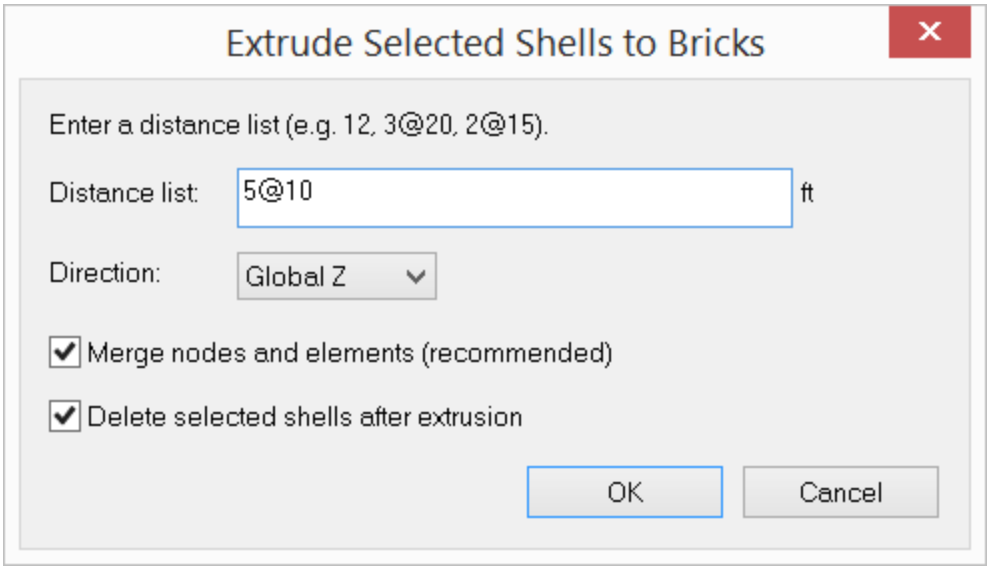
...to generate the shells in following screen capture by extruding.



shells generated

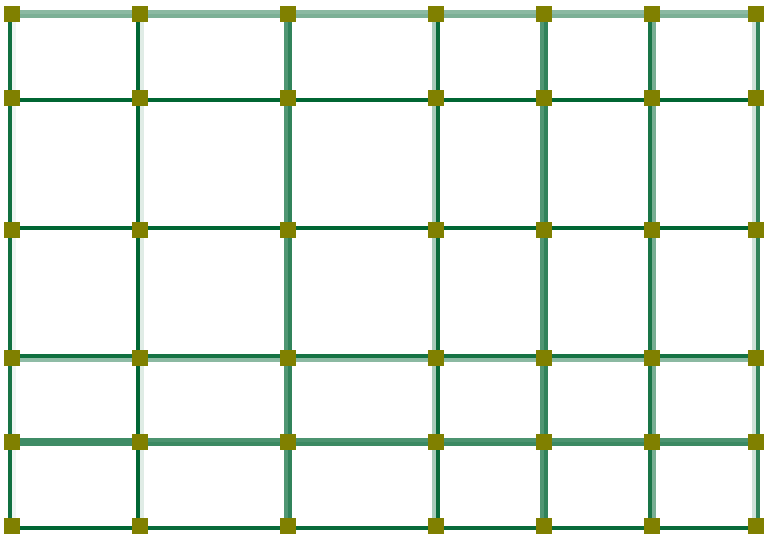
© 2026 ENERCALC, LLC

Modify > Extrude > Extrude Shells to Bricks prompts you with the following dialog.



It generates a series of bricks by extruding selected shells along a global direction. You may specify a distance list and an extrusion direction for the generation of shells. The extrusion direction must not be parallel to the selected shells. You have the option to automatically merge nodes and elements after extrusion. You should check this option unless duplicate nodes are explicitly permitted. You also have the option to automatically delete selected shells after the extrusion.

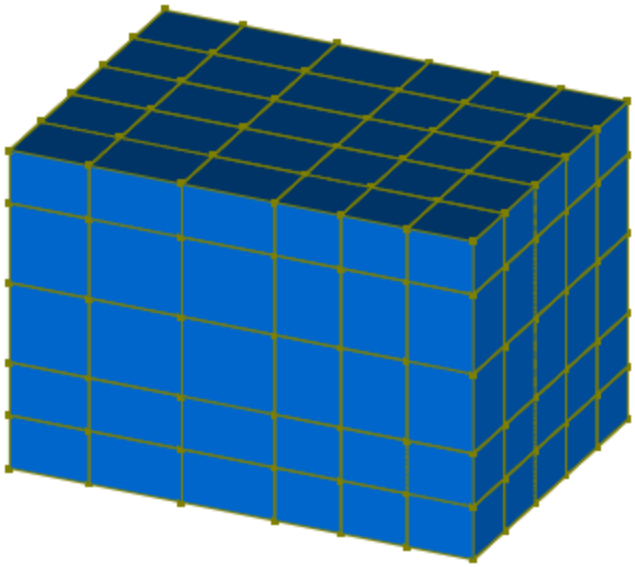
The input in the above dialog is applied to the shells in the screen capture below...



shells used for extruding

© 2026 ENERCALC, LLC

...to generate the bricks in the following screen capture by extruding.



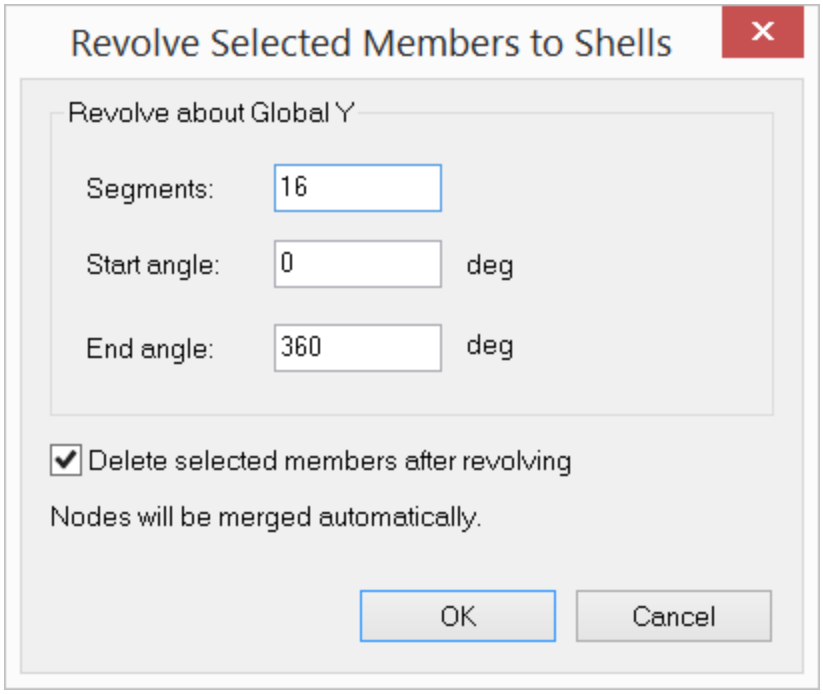
bricks generated

Revolve

Modify > Revolve provides various methods of creating entities by starting with a selected set and revolving them through specified angles.

© 2026 ENERCALC, LLC

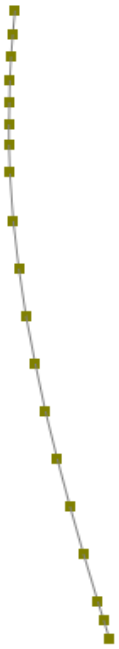
Modify > Revolve > Revolve Members to Shells prompts you with the following dialog.



It generates a series of shells by revolving selected members about the global Y axis. You may specify the number of segments, start and end angles for revolving. The program will merge nodes automatically. You have the option to automatically delete the selected members after revolving.

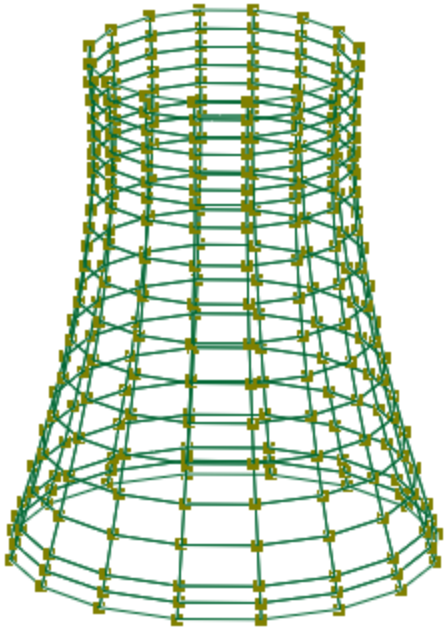
© 2026 ENERCALC, LLC

The input in the above dialog is applied to the members in the screen capture below...



members used for revolving

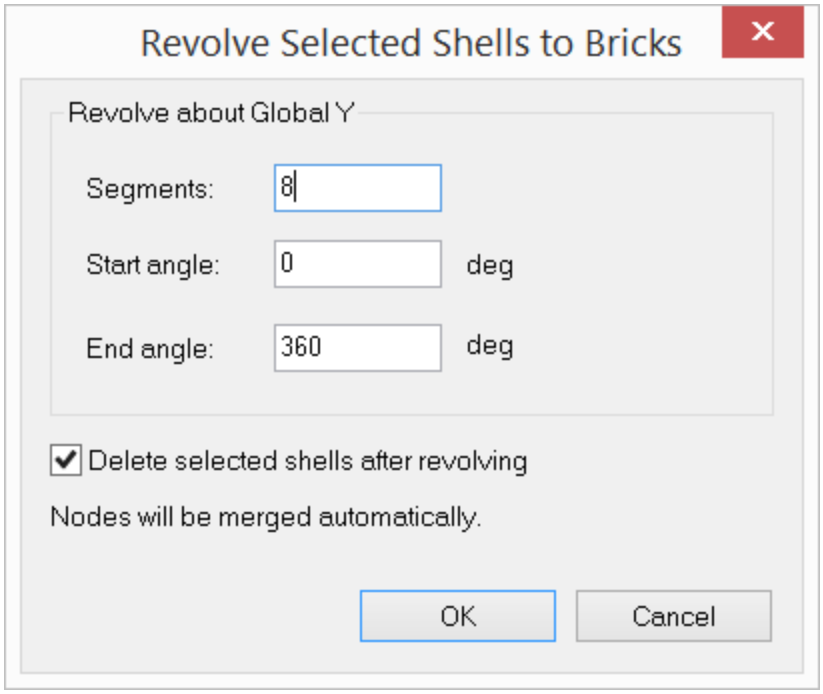
...to generate the shells in the screen capture below by revolving.



shells generated

© 2026 ENERCALC, LLC

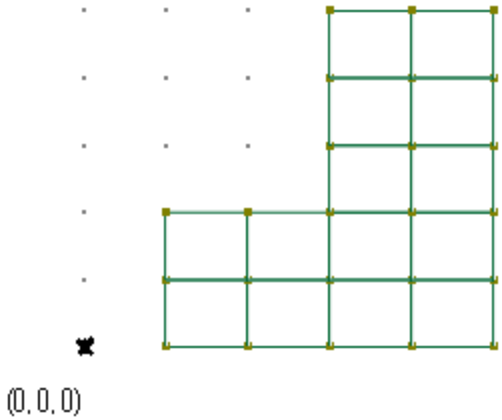
Modify > Revolve > Revolve Shells to Bricks prompts you with the following dialog.



It generates a series of bricks by revolving selected shells about the global Y axis. You may specify the number of segments, start and end angles for revolving. The program will merge nodes automatically. You have the option to automatically delete the selected shells after revolving.

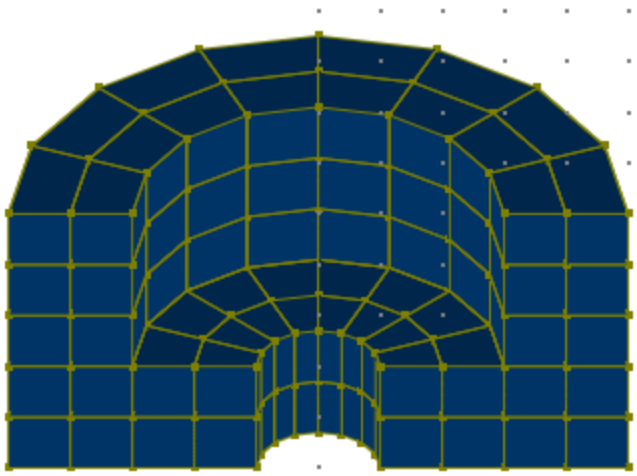
© 2026 ENERCALC, LLC

The input in the dialog above is applied to the shells in the screen capture below...



shells used for revolving

...to generate the bricks in the screen capture below by revolving. Bricks are rendered in the figure.



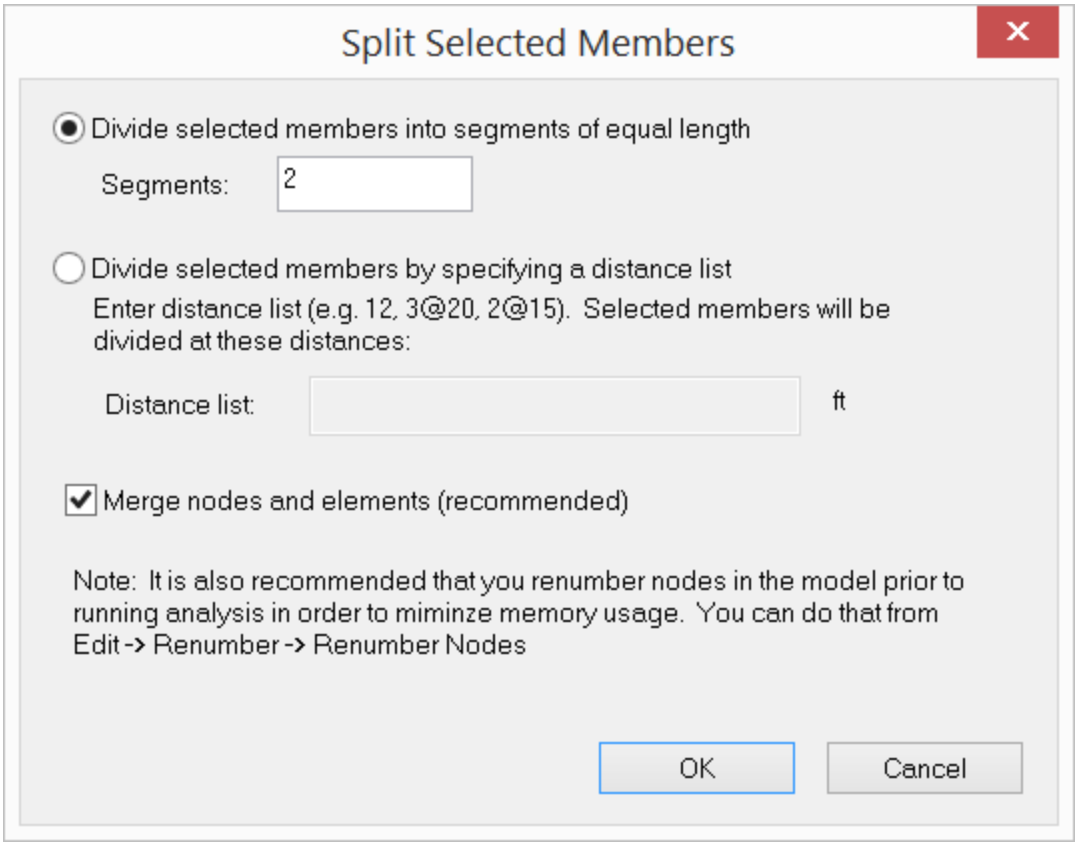
bricks generated

Split

Modify > Split provides tools for subdividing members.

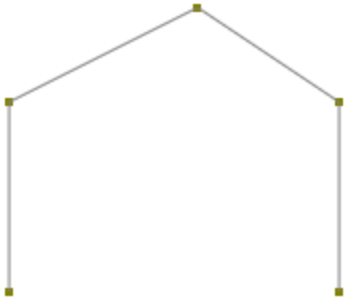
© 2026 ENERCALC, LLC

Modify > Split > Split Members prompts you with the following dialog.



It allows you to divide selected members by specifying 2 or more segments of equal length or by providing a distance list. Loads on the original members are assigned automatically to the generated members after splitting. You have the option to automatically merge nodes and elements after splitting.

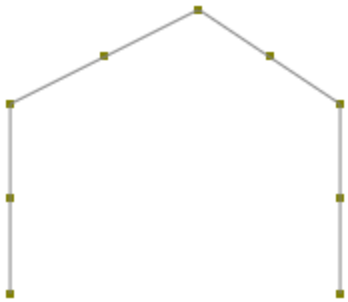
The input in the dialog above is applied to the members in the screen capture below...



members before splitting

© 2026 ENERCALC, LLC

...to generate the members in the screen capture below by splitting.



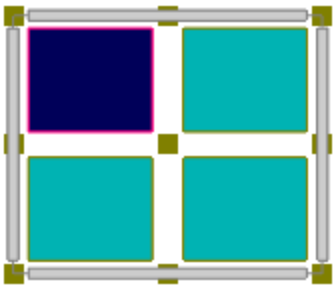
members after splitting

Modify > Split > Insert Nodes at Intersections of Selected Members allows you to insert nodes at all intersections of the selected members. The newly created nodes are likely isolated (orphaned) nodes, meaning they are not attached to intersecting members. Generally speaking, you should explode the members at these nodes. The program prompts you to do so at the end of this command.

Modify > Split > Split Selected Members at Nodes allows you to split selected members at nodes which are located on but are not connected to these members.

The following figure shows a shell with edge members on four sides:

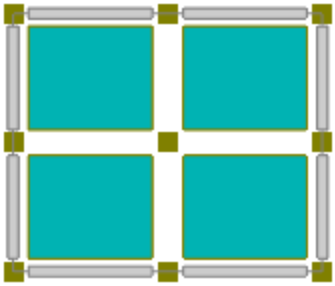
The following figure shows shells generated by sub-meshing (2x2) the shell from above:



Notice in the figure above, the middle nodes on the edge are on but not yet connected to the members.

© 2026 ENERCALC, LLC

The following figure shows members generated by splitting members at nodes.



The figures are rendered using the command View > Render, with the rendering ratio of 80% for both members and shells.

Member Properties

Modify > Member Properties provides tools for assigning properties to members.

Modify > Member Properties > Member Properties initiates the same command as Create > Member Properties > Member Properties. It opens the dialog that allows assignment of member properties.

Modify > Member Properties > Materials initiates the same command as Create > Member Properties > Materials. It opens the dialog that allows you to define and/or assign materials to selected elements in the model.

Modify > Member Properties > Sections initiates the same command as Create > Member Properties > Sections. It opens the dialog that allows you to modify sections in the model.

Modify > Member Properties > Element Local Angles initiates the same command as Create > Member Properties > Local Angles. It allows a member local angle to be modified.

Modify > Member Properties > 3-Point Member Orientation prompts you with a dialog to specify an X, Y, and Z coordinate. The orientation of selected members will be changed so the local z-axis of all selected members is perpendicular to the plane formed by the two member end coordinates and the specified X, Y, Z coordinate.

Modify > Member Properties > Moment Releases initiates the same command as Create > Member Properties > Moment Releases. It allows moment releases to be modified.

Modify > Member Properties > Tension/Compression Only initiates the same command as Create > Member Properties > Tension/Compression Only. It allows the Tension/Compression Only assignment to be modified.

Modify > Member Properties > Convert Members to Rigid Links will convert selected members to rigid links.

Modify > Member Properties > Cracking Factors initiates the same command as Create > Member Properties > Cracking Factors. It allows cracking factors to be modified.

Modify > Member Properties > Rigid Offsets initiates the same command as Create > Member Properties > Rigid Offsets. It allows rigid offsets to be modified.

© 2026 ENERCALC, LLC

Shell Properties

Modify > Shell Properties provides tools for assigning properties to shells.

Modify > Shell Properties > Shell Properties initiates the same command as Create > Shell Properties > Shell Properties. It opens the dialog that allows assignment of shell properties.

Modify > Shell Properties > Materials initiates the same command as Create > Shell Properties > Materials. It opens the dialog that allows you to define and/or assign materials to selected shells in the model.

Modify > Shell Properties > Thicknesses initiates the same command as Create > Shell Properties > Thicknesses. It opens the dialog that allows modification of shell thicknesses.

Modify > Shell Properties > Element Local Angles initiates the same command as Create > Shell Properties > Element Local Angles. It allows a shell local angle to be modified.

Modify > Shell Properties > Match Local x-Axes with Source prompts you with a dialog to specify a source shell. The orientation of the local x-axes of the selected shells will be changed to match the orientation of the local x-axis of the source shell.

Modify > Shell Properties > Match Local z-Axes for Shells prompts you with a dialog to specify a source shell. The orientation of the local z-axes of the selected shells will be changed to match the orientation of the local z-axis of the source shell.

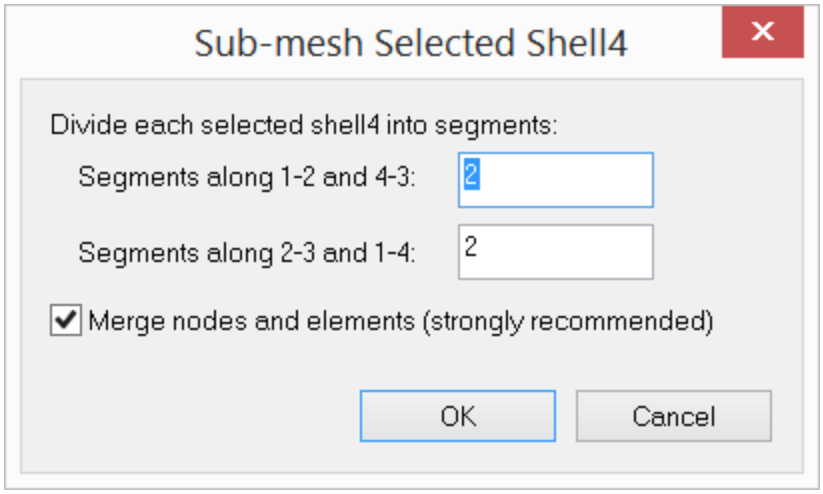
Modify > Shell Properties > Align Local z-Axes with Reference Point prompts you with a dialog to specify a reference point. The orientation of the local z-axes of the selected shells will be changed to match the vector from the reference point to the center of the selected shells.

Modify > Shell Properties > Align Local y-Axes with Reference Point prompts you with a dialog to specify a reference point. The orientation of the local y-axes of the selected shells will be changed to match the vector from the reference point to the center of the selected shells.

© 2026 ENERCALC, LLC

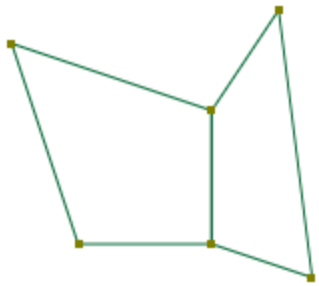
Sub-Mesh Shells

Modify > Sub-Mesh Shells prompts you with the following dialog.



It allows you to sub-mesh selected shells by specifying 2 or more segments along sides 1-2 & 4-3 and sides 2-3 & 1-4. Loads on the original shells are assigned automatically to the generated shells after sub-meshing. You have the option to merge nodes and elements after sub-meshing. You should check this option unless duplicate nodes are explicitly permitted.

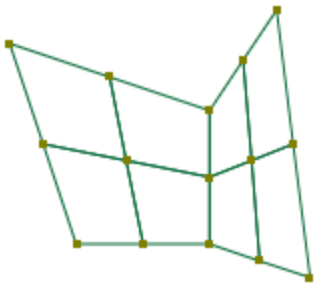
The input in the dialog above is applied to the shells in screen capture below...



shells before sub-meshing

...to generate the shells in the screen capture below by meshing.

© 2026 ENERCALC, LLC

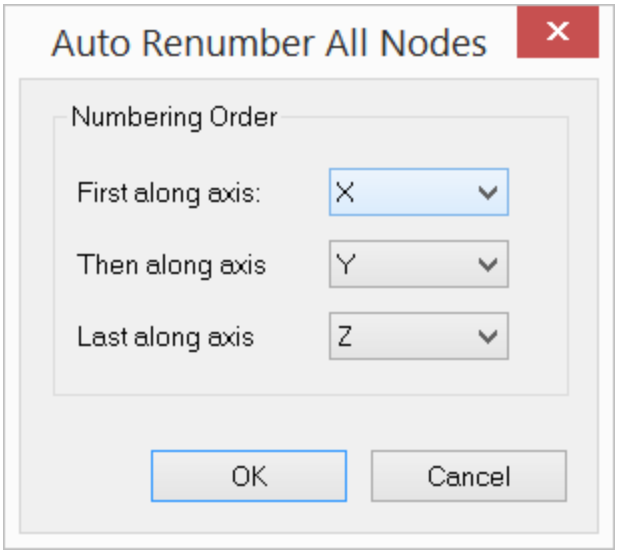


shells after sub-meshing

Renumber

Modify > Renumber offers options to renumber entities in the model.

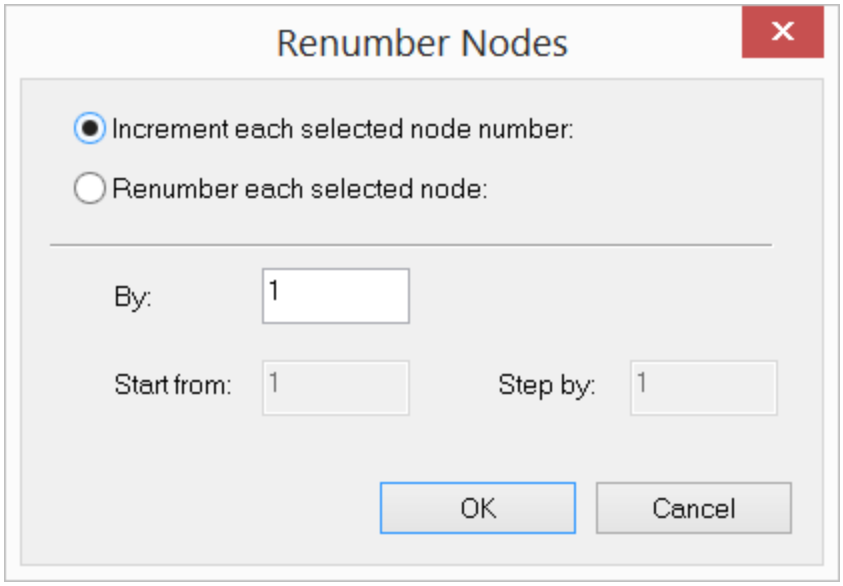
Modify > Renumber > Auto Renumber All Nodes prompts you with the following dialog.



It allows you to renumber all nodes sequentially based on nodal coordinates.

© 2026 ENERCALC, LLC

Modify > Renumber > Renumber Selected Nodes prompts you with the following dialog.



It allows you to Renumber selected nodes based on the following two modes:

a). Increment each selected node number by a delta (may be positive or negative). For example, if we have selected node numbers 2, 5, 8 and a delta of 2, the new node numbers will be 4, 7, and 10.

b). Renumber each selected node from a new start number (must be positive) and a step (may be positive or negative). For example, if we have selected node numbers 2, 5, 8, a new start number of 1000 and a step of 2, the new node numbers will be 1000, 1002, 1004.

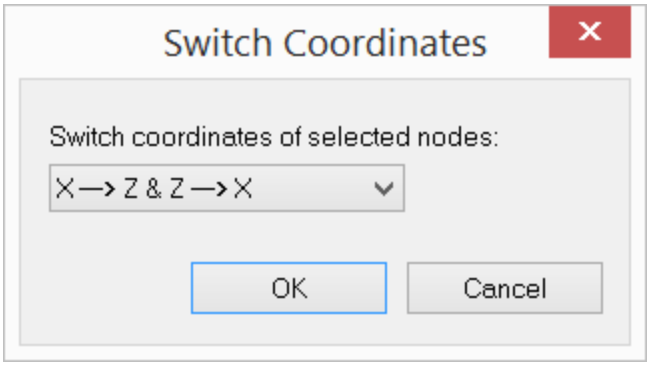
If renumbering is successful, nodal dependents such as loads, masses and springs will be renumbered automatically. The renumbering will be undone automatically if any errors are encountered.

Modify > Renumber > Renumber Members (or Shells, or Bricks) command is similar to Renumber Selected Nodes.

© 2026 ENERCALC, LLC

Switch Coordinates

Modify > Switch Coordinates prompts you with the following dialog.



It allows you to switch the X and Z, or the Y and Z coordinates of the selected nodes. For example, you may generate a floor system on the XY (vertical) plane and then switch coordinates to place the floor on the XZ (horizontal) plane.

Reverse Node Order for Selected Elements

Modify > Reverse Node Order for Selected Elements allows you to reverse the nodes' order for selected elements. For members and shells, this command in effect changes their local coordinate systems. For bricks, this command may be used to rectify a wrong nodal ordering which results in negative diagonals in element stiffness.

It is important to point out that dependents on the elements are not reversed or changed accordingly. After running this command, you should check on these dependents such as loads and moment releases on members, loads on shells, etc.

Merge All Nodes & Elements

Modify > Merge All Nodes & Elements merges all nodes that are located within a distance tolerance between two or more nodes, and merges all elements that share the same nodes. You may set the distance tolerance using the command Settings > Data Options.

Remove All Orphaned Nodes

Modify > Remove All Orphaned Nodes removes all nodes that are not connected to any elements. Orphaned nodes make the model unstable. They must be removed prior to the solution.

Element Activation

Modify > Element Activation allows members, shells, and/or bricks to be selectively activated or deactivated.

This allows these modeling entities to remain in the model while studying the effects of ignoring their structural contributions to the model.

© 2026 ENERCALC, LLC

View

The View ribbon provides commands to graphically view inputs such as geometry and loading; perform selections by various methods; and display outputs such as shear and moment diagrams for members, and contours for shells and bricks.

Drawing Grids

View > Drawing Grids > Drawing Grid Setup is the same as Create > Grids & Snaps > Drawing Grid Setup.

View > Drawing Grids > Toggle Grid Display shows or hides the drawing grid.

Redraw

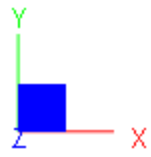
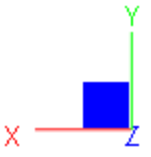
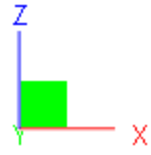
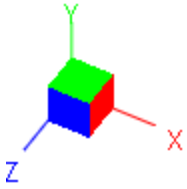
View > Redraw regenerates and redraws all graphics in the model view.

Restore Model

View > Restore restores original settings for the model view. These settings include zooming = 1.0, panning = 0, rotation = 0.

Preset Views

You may place the model view in a preset orientation by selecting one of these commands:

Front	Back	Left	Right
			
Top	**Bottom**	**Isometric**	
			

Named Views

View > Named Views allows you to save the current view settings such as zooming factor, panning distance or rotation angles so that you may recall the same view settings later on.

Zoom Extent

View > Zoom Extent displays the entire model in the view.

© 2026 ENERCALC, LLC

Zoom Window

View > Zoom Window zooms in on a specific part of the model by clicking and dragging the left mouse button. The command remains in effect until another command is selected, the right mouse button is clicked, or ESC is pressed.

Zoom Object

View > Zoom Object zooms in on a specific node, member, shell or brick. The command remains in effect until another command is selected, the right mouse button is clicked, or ESC is pressed.

Zoom Previous

View > Zoom Previous lets you zoom back to the previous view. You may use this command after you zoom in or pan to view a portion of your model in greater detail.

Zoom In

View > Zoom In zooms in on the model by a preset factor (1.25). You may use this command by pressing CTRL+UP arrow or CTRL+RIGHT arrow.

Zoom Out

View > Zoom Out zooms out on the model by a preset factor (1.25). You may use this command by pressing CTRL+DOWN arrow or CTRL+LEFT.

Pan

The Pan controls allow the view of the model to be moved.

View > Pan > Pan Left pans the model to the left by a preset screen distance. You may use this command by pressing CTRL+LEFT arrow.

View > Pan > Pan Right pans the model to the right by a preset screen distance. You may use this command by pressing CTRL+RIGHT arrow.

View > Pan > Pan Up pans the model to the top by a preset screen distance. You may use this command by pressing CTRL+UP arrow.

View > Pan > Pan Down pans the model to the bottom by a preset screen distance. You may use this command by pressing CTRL+DOWN arrow.

View > Pan > Pan Screen pans (moves) the model by clicking and dragging the left mouse button. The command remains in effect until another command is selected, the right mouse button is clicked, or ESC is pressed.

Rotate

The Rotate commands allow the view of the model to be spun around various axes.

View > Rotate +X rotates the model view about X by a preset positive angle (5 degrees). You may use this command by pressing SHIFT+DOWN arrow.

View > Rotate -X rotates the model view about X by a preset negative angle (5 degrees). You may use this command by pressing SHIFT+UP arrow.

© 2026 ENERCALC, LLC

View > Rotate +Y rotates the model view about Y by a preset positive angle (5 degrees). You may use this command by pressing SHIFT+RIGHT arrow.

View > Rotate -Y rotates the model view about Y by a preset negative angle (5 degrees). You may use this command by pressing SHIFT+LEFT arrow.

View > Rotate +Z rotates the model view about Z by a preset positive angle (5 degrees). You may use this command by pressing CTRL+SHIFT+UP arrow or CTRL+SHIFT+RIGHT arrow.

View > Rotate -Z rotates the model view about Z by a preset negative angle (5 degrees). You may use this command by pressing CTRL+SHIFT+DOWN arrow or CTRL+SHIFT+LEFT arrow.

Pan

View > Pan allows you to pan the model view in real time by clicking and dragging the left mouse button. The command remains in effect until another command is selected, the right mouse button is clicked, or ESC is pressed.

Zoom

View > Zoom allows you to zoom in or out on the model view in real time by clicking and dragging the left mouse button up or down. The command remains in effect until another command is selected, the right mouse button is clicked, or ESC is pressed.

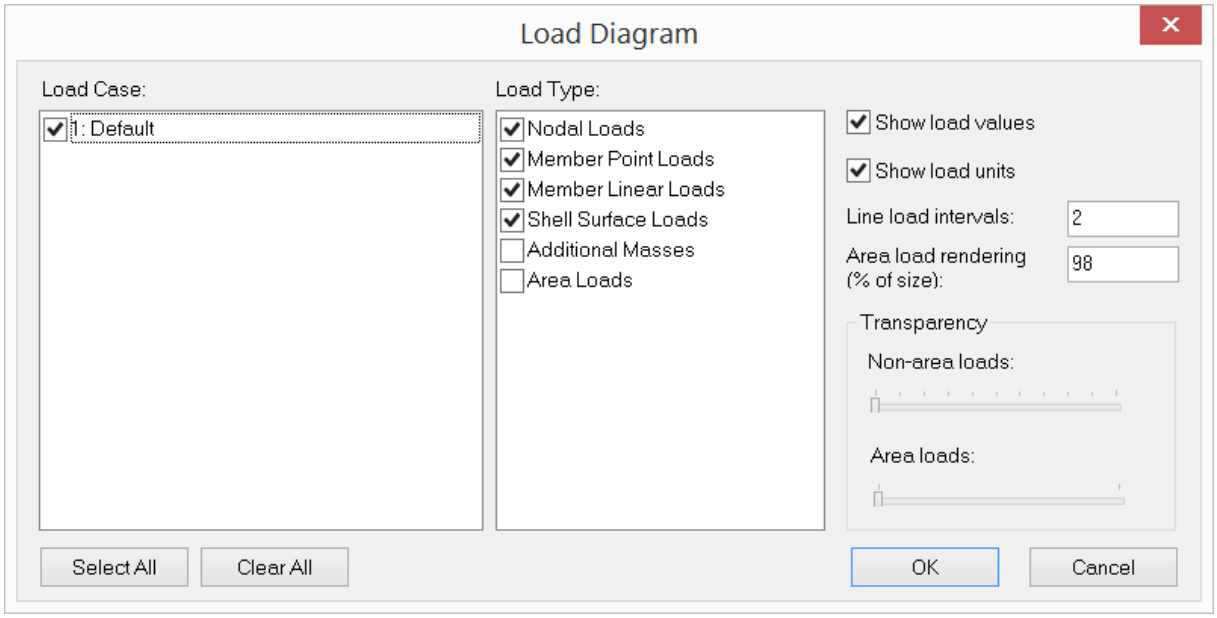
Rotate

View > Rotate allows you to rotate the model view in real time by clicking and dragging the left mouse button. The command remains in effect until another command is selected, the right mouse button is clicked, or ESC is pressed.

© 2026 ENERCALC, LLC

Load Diagram

View > Load Diagram prompts you with the following dialog.

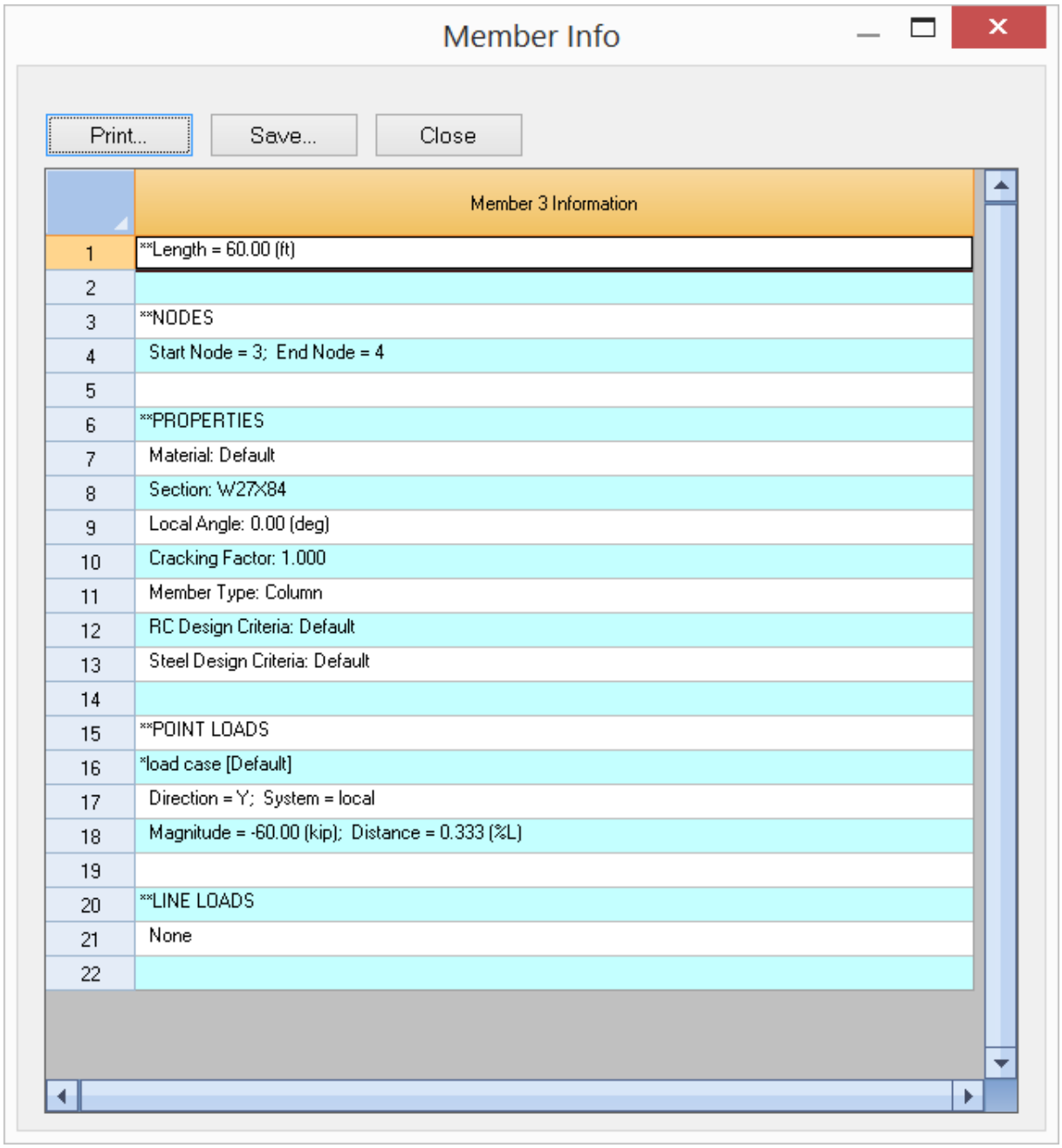


It allows you to view loads of selected types in selected load cases. You may have the options to show load magnitudes or units. The line load intervals may vary between 1 and 16. An interval between 2 to 6 is recommended. Transparency may be set for non-area loads and area loads so you can see objects underneath the loads. The displayed loads may be deleted. The loads not displayed cannot be deleted unless their parent nodes or elements are deleted.

© 2026 ENERCALC, LLC

Query

View > Query lets you query extensive input and output information for a single node, member or finite element.



For example, a node query will list node id, nodal coordinates and nodal loads. It will also list relevant support or nodal spring information. If there are analysis results, it will list nodal displacements, support or spring reactions as well.

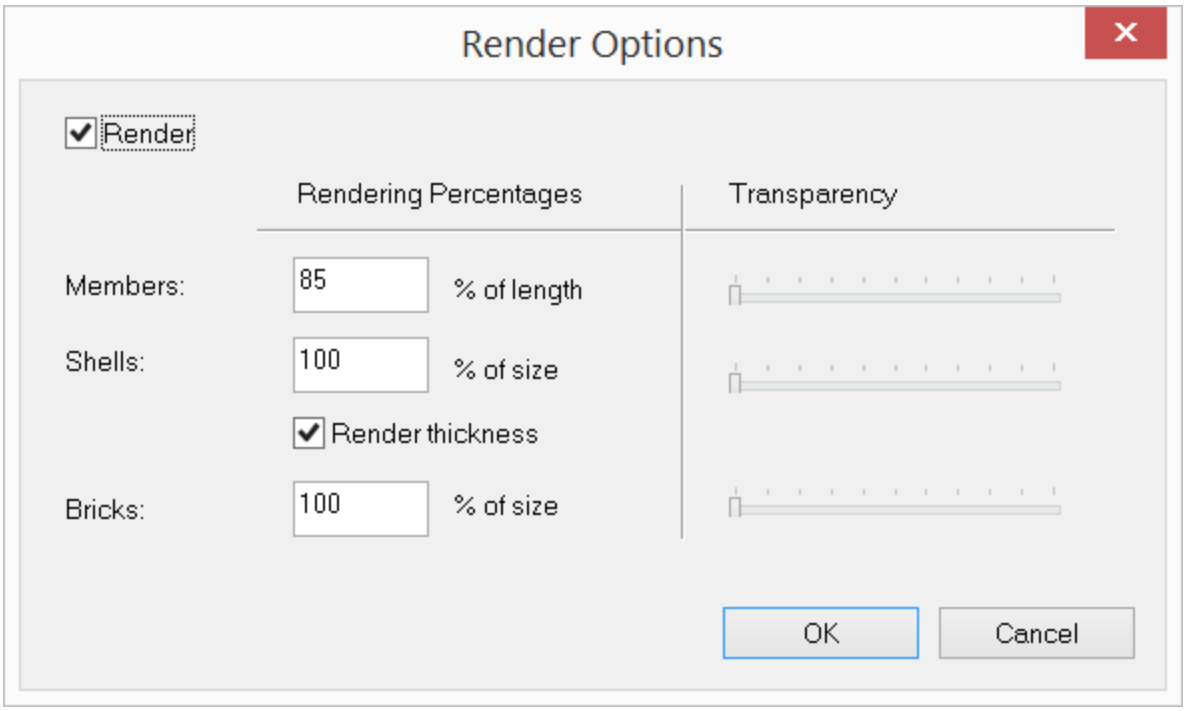
© 2026 ENERCALC, LLC

Measure

View > Measure lets you click two locations and obtain the distance between the two coordinates.

Render Options

View > Render Options prompts you with the following dialog.



It allows you to turn on or off the shading of the surfaces of members, shells and bricks as though they were illuminated from multiple light sources. It provides a way for you to realistically visualize the image of the model. For shells, you have the option to render thickness as well as surface.

You have the option to apply different rendering percentages to different elements. Enter 100% for full rendering, 0% for no rendering, and anything in-between for partial rendering. The partial rendering (e.g. 50-80%) may be useful in identifying connectivity of elements to nodes.

Note: *Rendering a model can be expensive in terms of memory and time usage by the program. You should turn off the rendering when it is not necessary.*

Quick Render

View > Quick Render is a toggle that switches between the wireframe display and the rendered view. When shells are present in the model, it actually acts as a 3-way toggle that introduces a third view that displays plates rendered and with actual thickness displayed.

You may use this command by pressing F8.

© 2026 ENERCALC, LLC

Diaphragm Render

View > Diaphragm Render is a toggle. It controls the rendering of diaphragms.

Global Axes

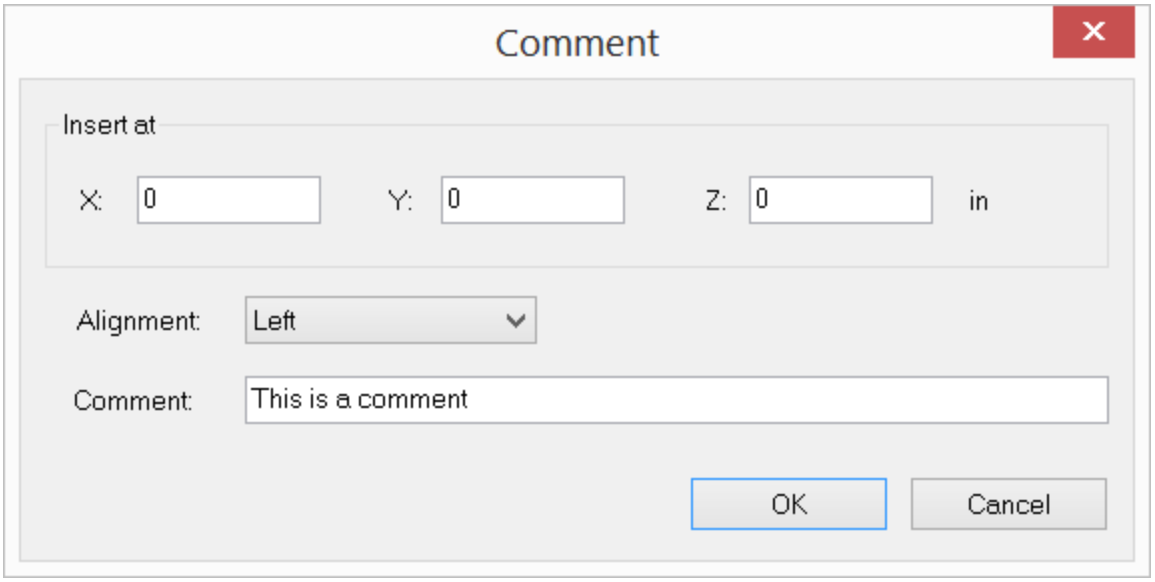
View > Global Axes shows or hides the legend of the global axes in the bottom-left corner of the window. You may use this command by pressing F5.

Contour Legend

View > Contour Legend shows or hides the contour legend. The results must exist for the contour legend to be displayed.

Comment

View > Comment allows you to insert a comment at a specified location.



The comment must be less than 256 characters in length. To remove an existing comment, go to Tables > Comments and delete the comment entry.

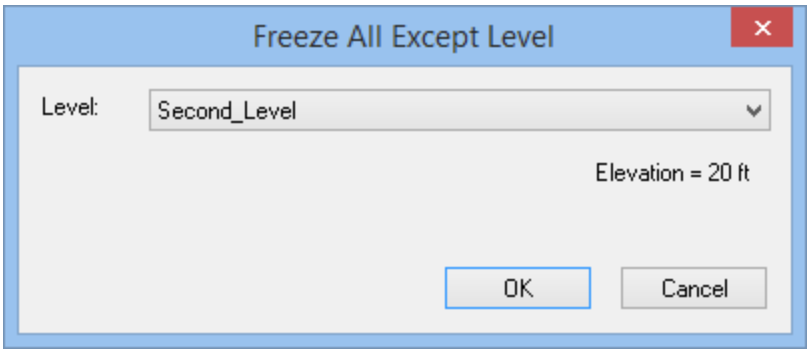
Hide Selected

View > Hide Selected hides the selected nodes, elements and their dependents. The hidden nodes or elements are not displayed and are not modifiable unless the model integrity is at stake. This command allows you to focus on some particular parts of the model. For example, if you want to work on a particular floor of a three dimensional building, you may select the floor, invert the selection and hide the selected elements.

© 2026 ENERCALC, LLC

Hide All Except Level

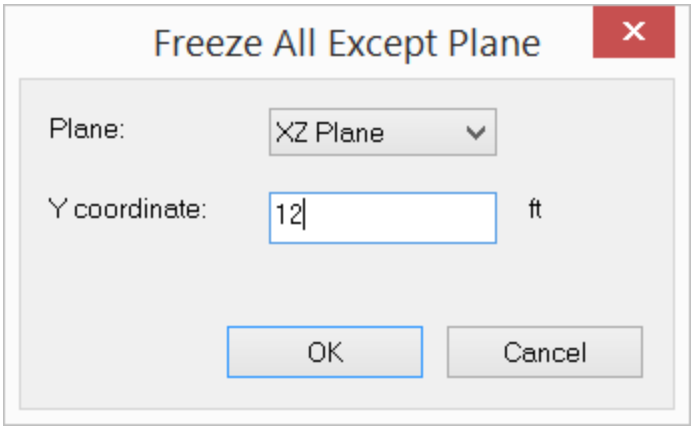
View > Hide All Except Level hides the all nodes, elements and their dependents except those on the specified level.



This command provides a shortcut to the previous command when you would like to focus on elements of the model in a horizontal plan view. You must have levels defined (through Create > Levels) before running this command.

Hide All Except Plane

View > Hide All Except Plane hides all nodes, elements and their dependents except those on the specified plane.



This command provides a shortcut to the previous command when you would like to focus on elements of the model in a plan view.

Show All

View > Show All displays all nodes, elements and shells (even if they were previously hidden).

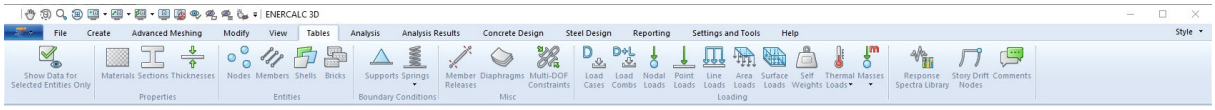
Display Options

View > Display Options allows control over what is and is not displayed on the screen.

© 2026 ENERCALC, LLC

Tables

The Tables ribbon provides commands to create or modify all input data of a model using spreadsheets.



Spreadsheets provide an alternative method to the graphic input described in earlier chapters. You may combine both methods to create a model quickly.

The spreadsheets support the common clipboard actions such as “CTRL+X” to cut, “CTRL+C” to copy and “CTRL+V” to paste data. You may even share data between the spreadsheets in the ENERCALC 3D and other spreadsheet programs such as Microsoft Excel. For example, you may generate node data in an Excel spreadsheet, copy the nodal coordinate data and paste to the Nodes spreadsheet in the program. In this way, you can take advantage of the more powerful data manipulation functions in the Excel.

In each spreadsheet, you may add one or more rows by clicking the “New Row” button. You may also print data in the spreadsheet by clicking the “Print” button. You have the option to view only the selected data. To do that, click Tables > Show Data for Selected Entities Only. You may not modify the data in the spreadsheet when this option is chosen.

The "Fill Down" Command

When working in the tables, it is often convenient to fill a column or a specific range of cells with the same value. The "Fill Down" command makes this very easy.

The command works in two modes:

- To fill an ***entire*** column with a specific value, right click on the cell containing the desired value and click OK.
- To fill a specific ***set*** of cells with a specific value, highlight the set of cells to be changed, then right-click on the cell containing the desired value (can be anywhere in the current table) and click OK.

Show Data for Selected Entities Only

Tables > Show Data for Selected Entities Only is a toggle to control what is displayed when tables are viewed.

Materials

The Tables > Materials command opens the Material Data table to display the materials that have been defined in the current model. It also offers assignment options.

Sections

The Tables > Sections command opens the Section Data table to display the sections that have been defined in the current model. It also offers assignment options.

© 2026 ENERCALC, LLC

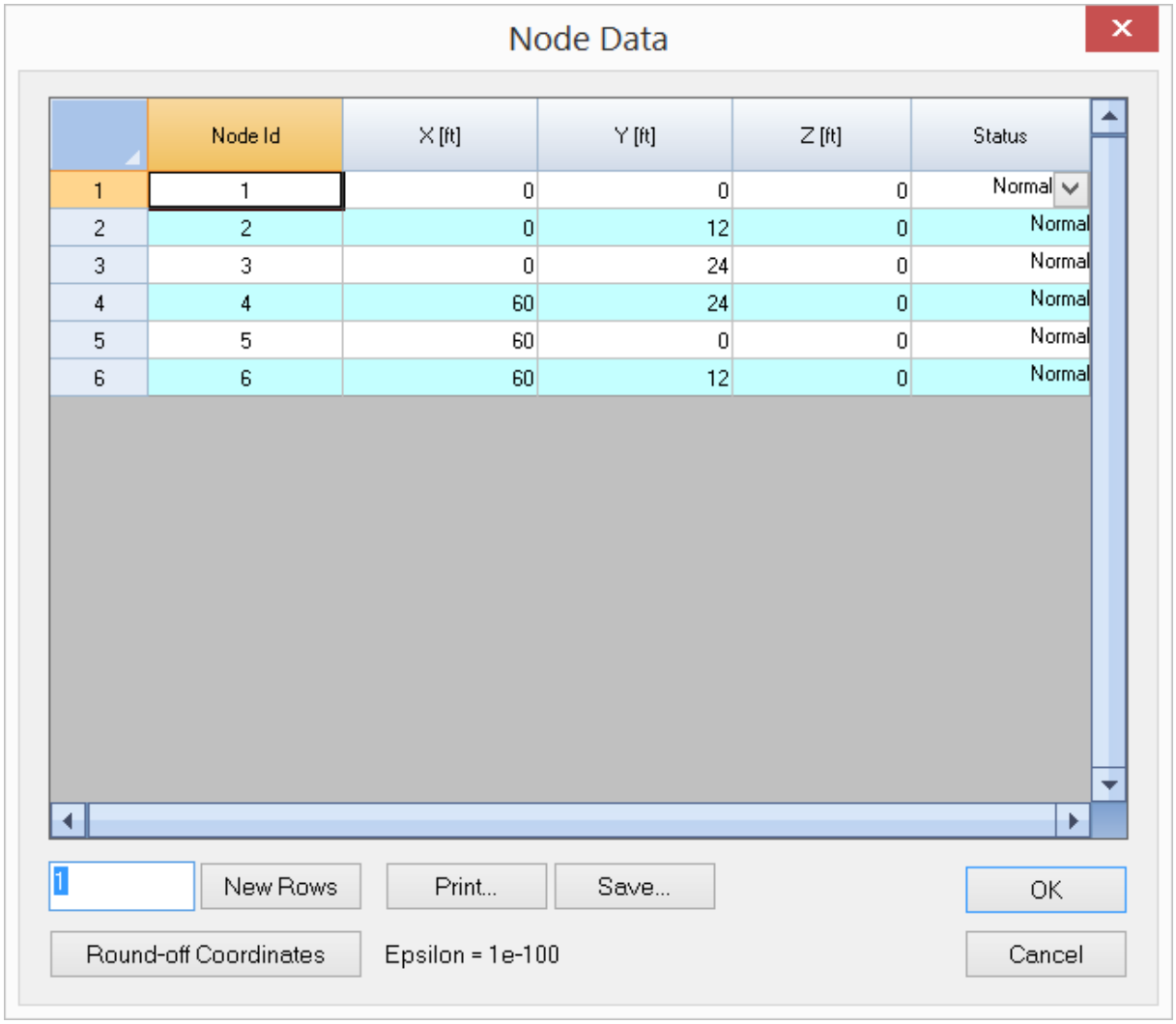
Thicknesses

The Tables > Thicknesses command opens the Shell Thickness Data table to display the thicknesses that have been defined in the current model. It also offers assignment options.

© 2026 ENERCALC, LLC

Nodes

Tables > Nodes prompts you with the following dialog.



It allows you to enter nodes in a spreadsheet. Each node includes the nodal coordinates and the selection status. You may not modify the node IDs.

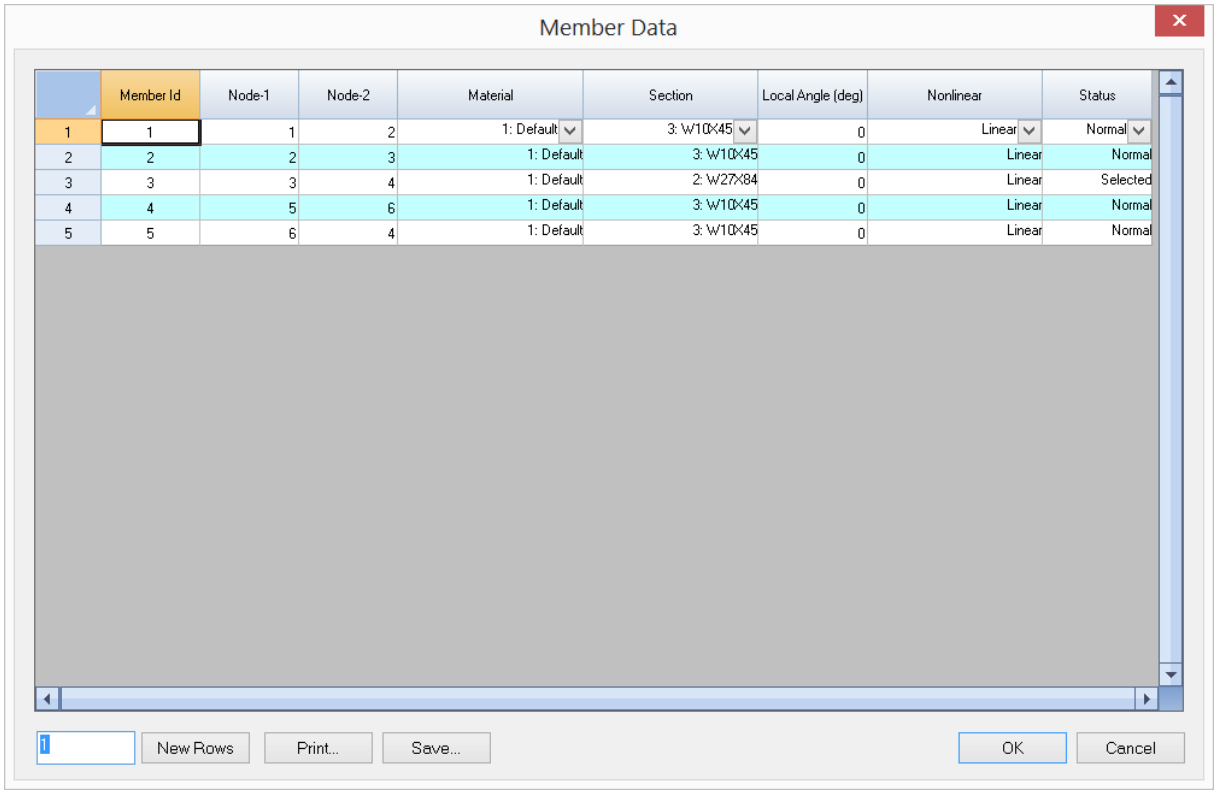
An empty row is allowed if all rows below it are empty (except the node ID and status fields). You may not delete the existing nodes in the dialog. To delete the existing nodes, you must dismiss this dialog and click Modify > Delete.

Due to machine inaccuracy of floating point values, some commands (such as Modify > Rotate) may cause the presence of very small numerical coordinate values. You may round off these tiny values to be zeros by clicking the Round-off Coordinates button. The epsilon used for the round-off may be set from Settings & Tools > Data Options. The default epsilon value is 1e-10 and must be less than or equal to 1e-6.

© 2026 ENERCALC, LLC

Members

Tables > Members prompts you with the following dialog.



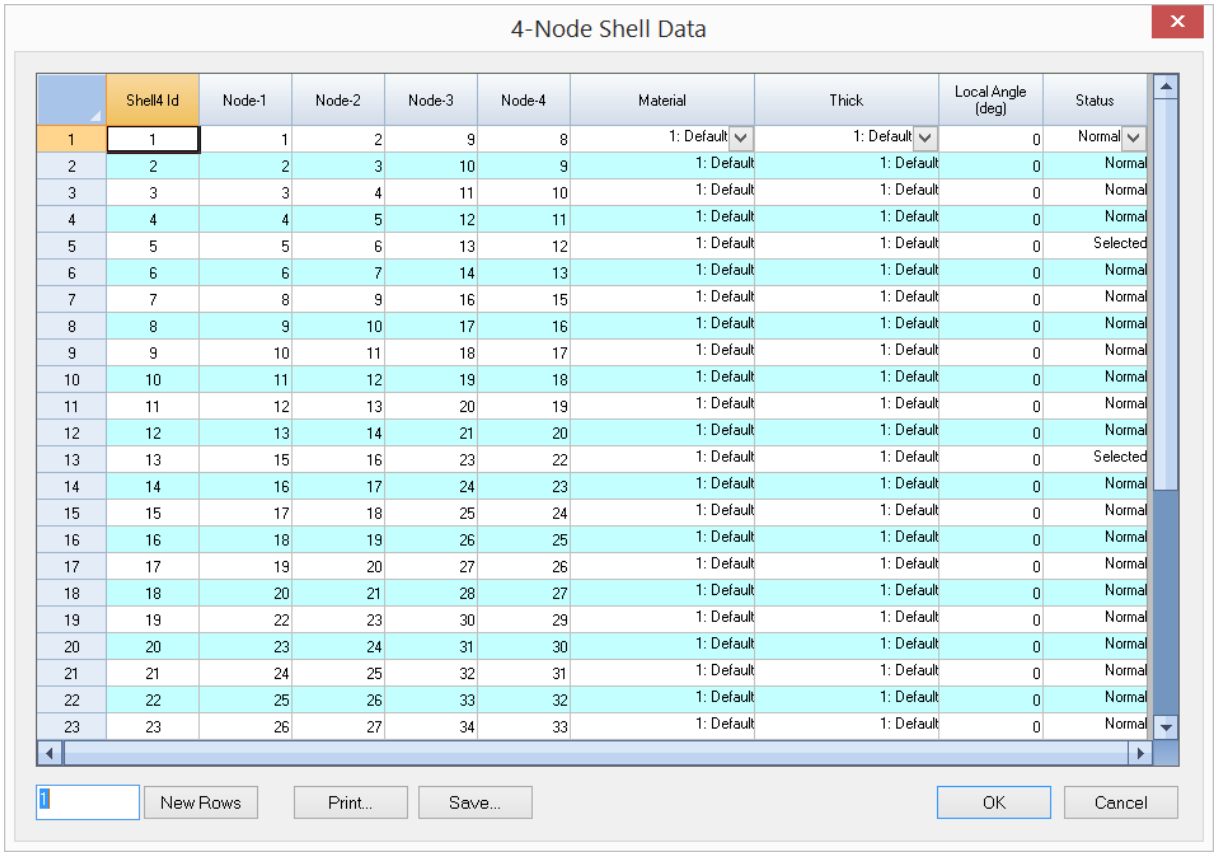
It allows you to enter members in a spreadsheet. Each member includes the IDs of start and end nodes, the material and section IDs, the element local angle, and the selection status. You may not modify the member ID. All other IDs must be valid (defined). Material and section combo boxes are provided for you to correctly pick and apply proper material and section IDs to selected members.

An empty row is allowed if all rows below it are empty (except the member ID and status fields). You may not delete the existing members in this dialog. To delete the existing members, you must dismiss this dialog and click Modify > Delete.

© 2026 ENERCALC, LLC

Shells

Tables > Shells prompts you with the following dialog.



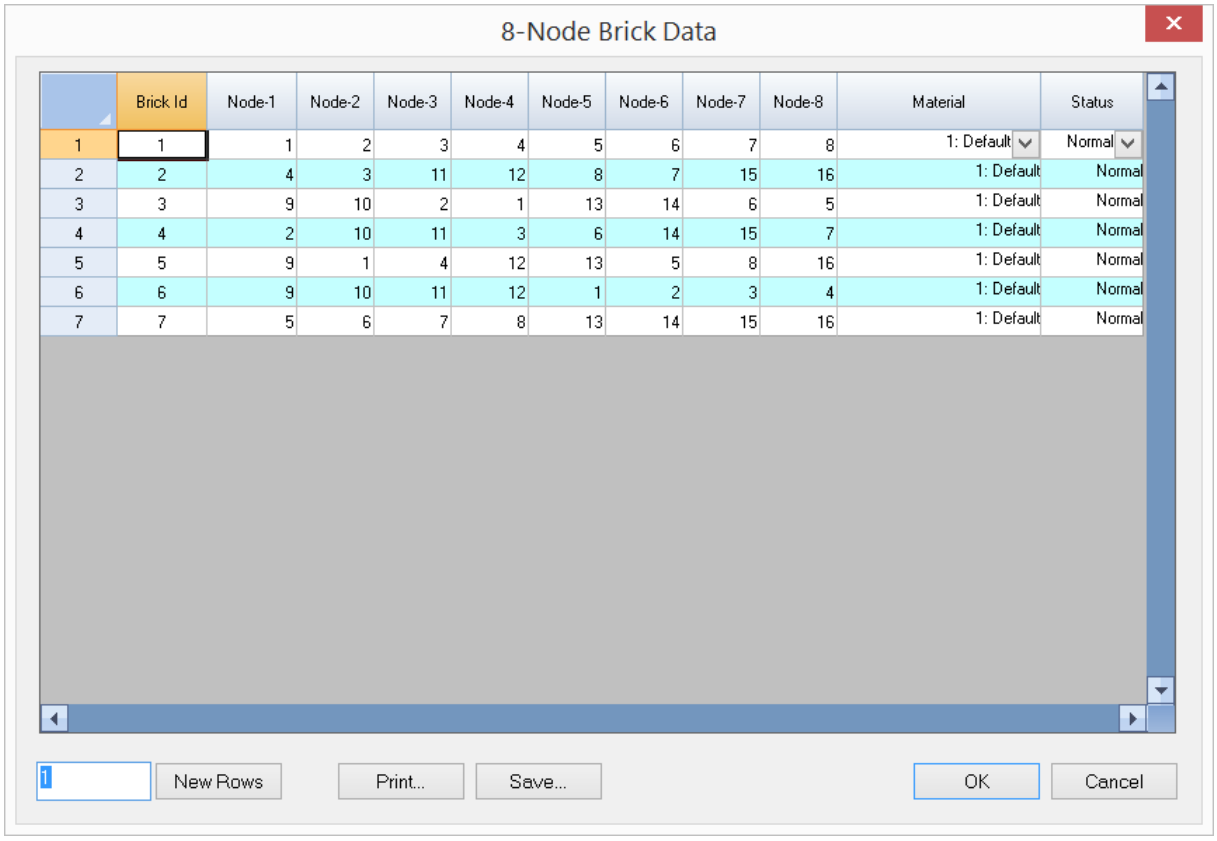
It allows you to enter shells in a spreadsheet. Each shell includes the IDs of four element nodes, the material and thickness IDs, the element local angle, and selection status. You may not modify the shell IDs. All other IDs must be valid (defined). Material and thickness combo boxes are provided for you to correctly pick and apply proper material and thickness IDs to selected shells.

An empty row is allowed if all rows below it are empty (except the shell ID and status fields). You may not delete the existing shells in this dialog. To delete the existing shells, you must dismiss this dialog and click Modify > Delete.

© 2026 ENERCALC, LLC

Bricks

Tables > Bricks prompts you with the following dialog.

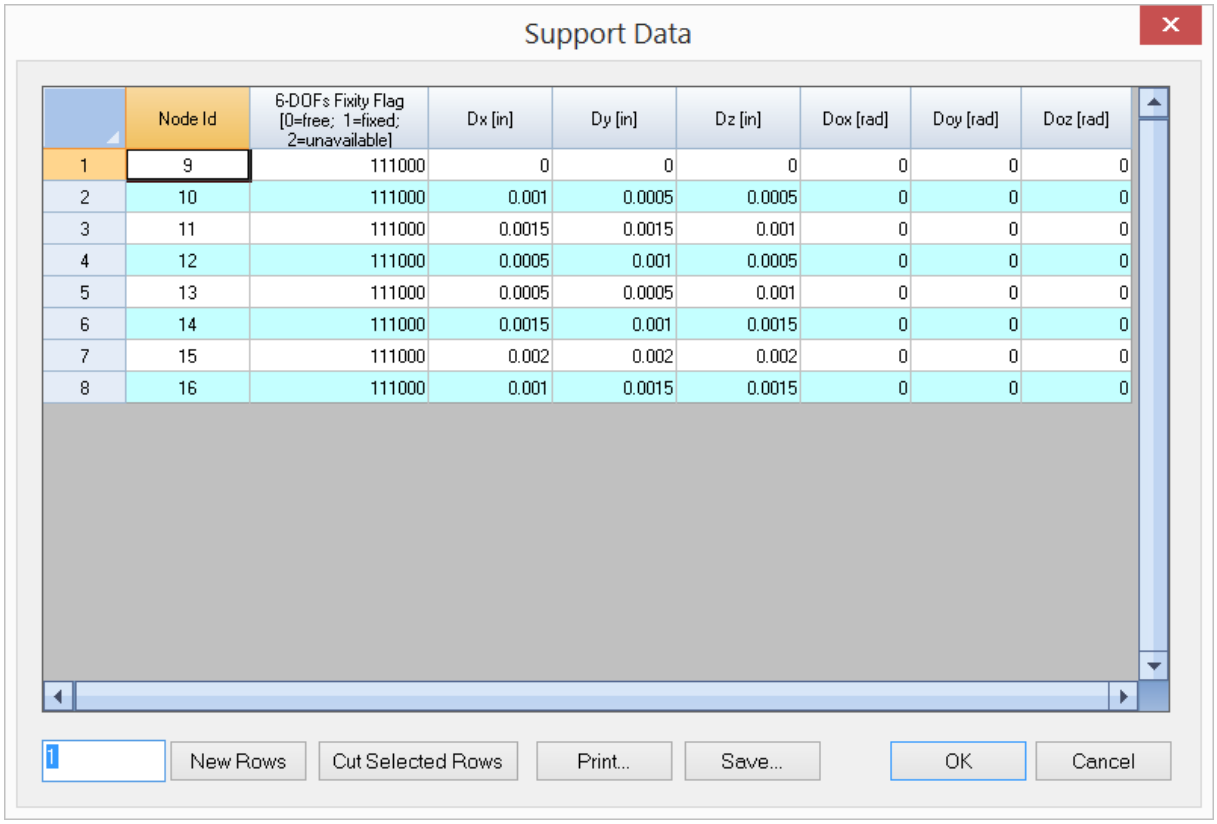


It allows you to enter bricks in a spreadsheet. Each brick includes the IDs of eight element nodes, the material ID and selection status. You may not modify the brick IDs. All other IDs must be valid (defined). The material combo box is provided for you to correctly pick and apply proper material IDs to selected bricks.

© 2026 ENERCALC, LLC

Supports

Tables > Supports prompts you with the following dialog.



It allows you to enter supports in a spreadsheet. Each support includes the node ID, the fixity flag, and enforced displacements for all restrained DOFs. The node IDs must be valid (defined).

The fixity flag is a string of 6 characters representing restrained DOFs in D_x, D_y … D_{oz}. For each character in the flag, enter '1' if the DOF is restrained and '0' if unrestrained. For example, "111111" represents a fixed support while "111000" represents a pinned support. Enforced displacements may be applied to the restrained DOFs. They may be regarded as special loads and can be used to model known support settlements. For normal support, they are 0s. Enforced displacements applied to unrestrained DOFs will be discarded.

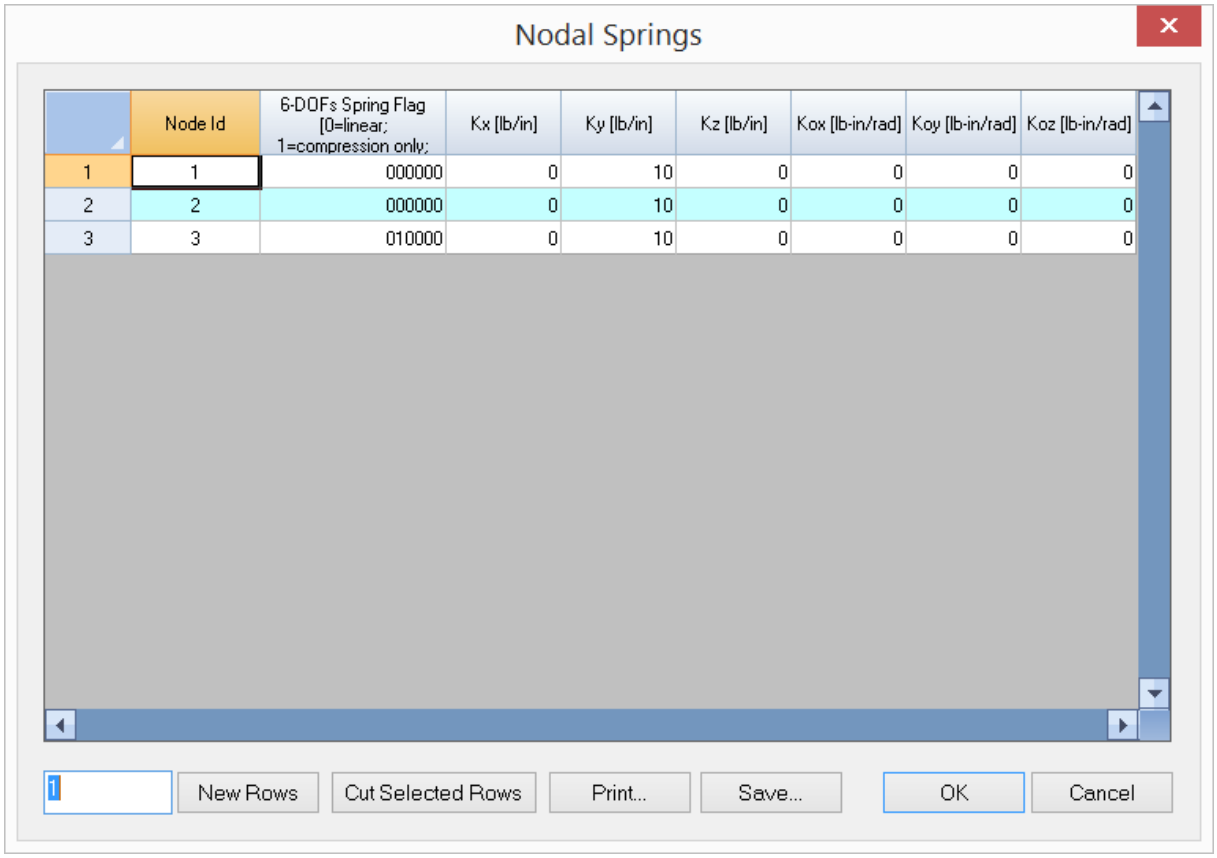
An empty row is allowed if all rows below it are empty. Selected rows (whole row must be selected) may be cut by clicking the button "Cut Selected Rows".

Springs

Tables > Springs offers options to access the tables for Nodal Springs, Line Springs and Surface Springs.

© 2026 ENERCALC, LLC

Tables > Springs > Nodal Springs prompts you with the following dialog.



It allows you to enter nodal springs in a spreadsheet. Each nodal spring includes the node ID, the spring non-linearity flag and six spring coefficients K_x, K_y …, K_{oz}. The node ID must be valid (defined).

The spring flag is a string of 6 characters representing the spring non-linearity in three translational DOFs (D_x, D_y, D_z) and three rotational DOFs (D_{ox}, D_{oy}, D_{oz}). For each character in the flag, enter '0' if the restrained DOF is linear, '1' if compression-only and '2' if tension only. For the unrestrained DOFs, just enter 0s for the corresponding spring coefficients.

An empty row is allowed if all rows below it are empty. Selected rows (whole row must be selected) may be cut by clicking the button "Cut Selected Rows".

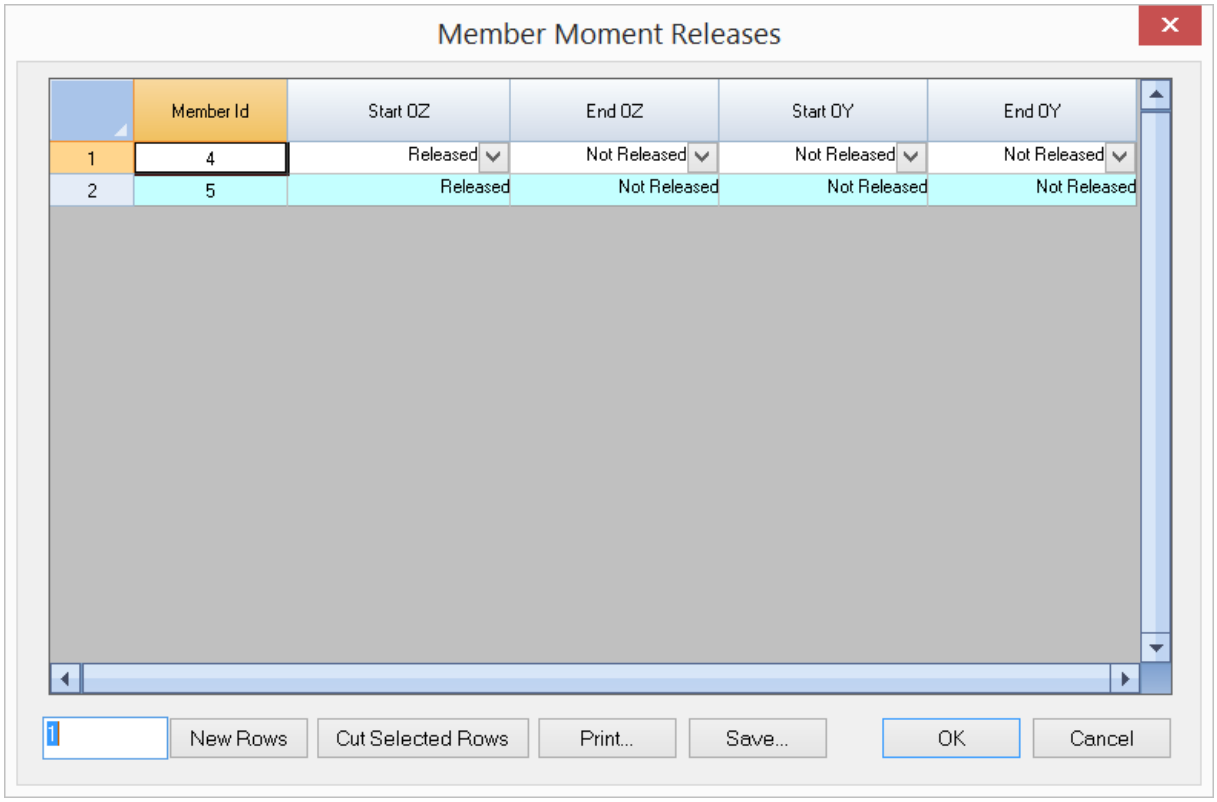
Tables > Springs > Line Springs offers options to access the table for Line Springs.

Tables > Springs > Surface Springs prompts you with dialogs similar to that in Tables > Springs > Nodal Springs. However, only three translational DOFs D_x, D_y, D_z are available for line or surface spring coefficients.

© 2026 ENERCALC, LLC

Member Releases

Tables > Member Moment Releases prompts you with the following dialog.



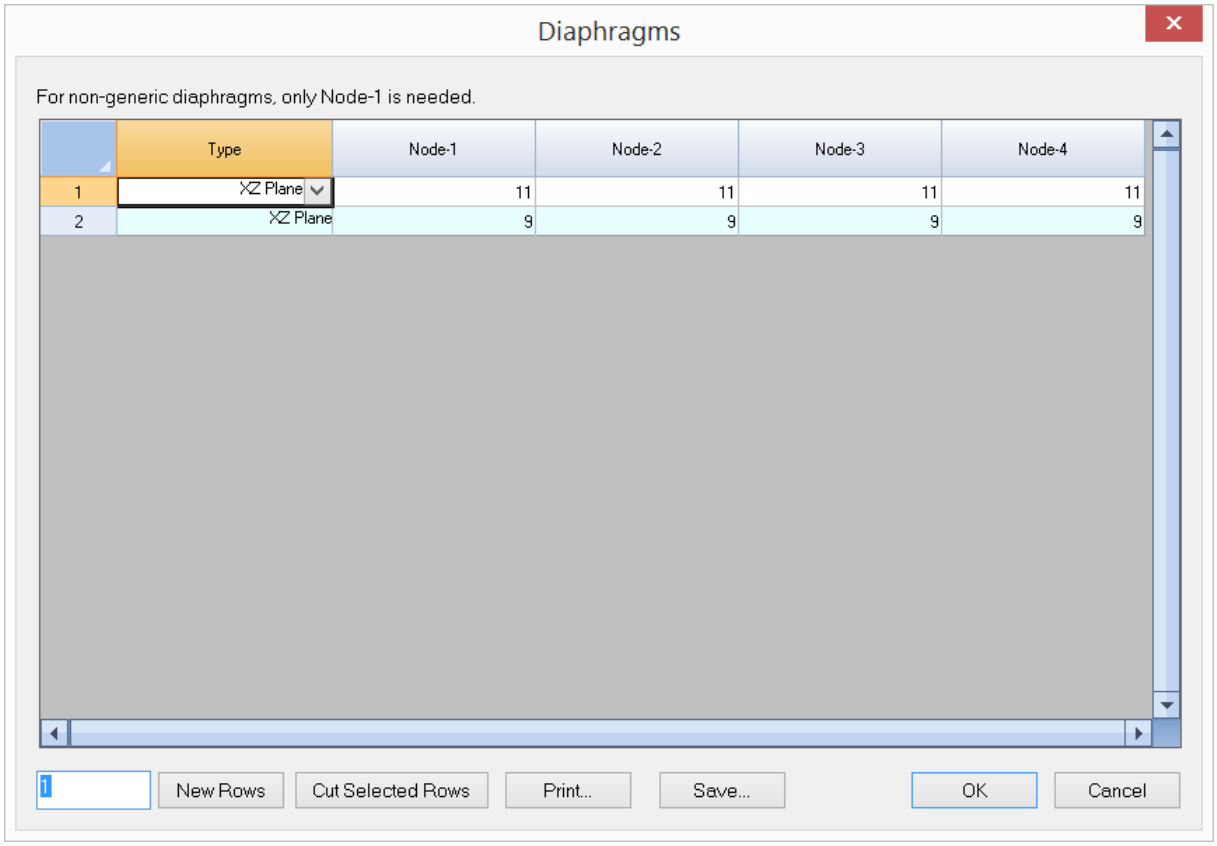
It allows you to enter member moment releases in a spreadsheet. Each moment release includes the member ID, four 1-character release codes for major moment release (the local oz), and minor moment release (the local oy) for both ends of members. Enter '1' for the released local DOF and '0' for the un-released local DOF. The member ID must be valid.

An empty row is allowed if all rows below it are empty. Selected rows (whole row must be selected) may be cut by clicking the button "Cut Selected Rows".

© 2026 ENERCALC, LLC

Diaphragms

Tables > Diaphragms prompts you with the following dialog.



It allows you to enter generic or regular rigid diaphragms in a spreadsheet. For a generic diaphragm, four distinct nodes are required to define the diaphragm plane. For a regular diaphragm in global XZ, YZ and XY plane, only the first node is required to define the diaphragm plane.

An empty row is allowed if all rows below it are empty. Selected rows (whole row must be selected) may be cut by clicking the button “Cut Selected Rows”.

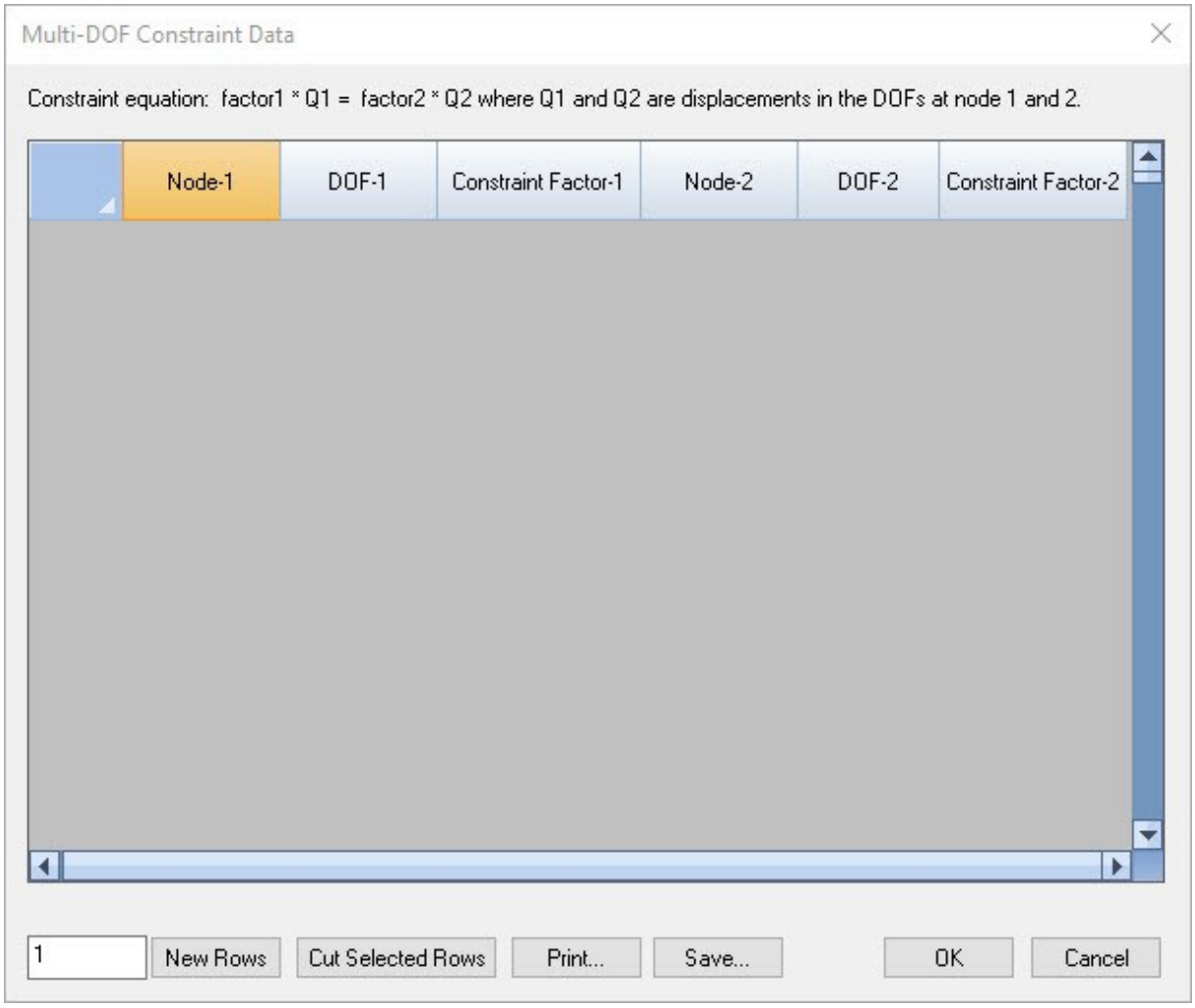
Rigid diaphragms may be used instead of plate finite elements to model stiff in-plane actions such as concrete floors. Internally, the program creates multiple in-plane rigid links for each diaphragm prior to static or frequency analysis. A rigid link is simply a member with very large sectional properties that can be adjusted with the diaphragm stiffness factor (see Settings > Data Options). The larger the diaphragm stiffness factor, the stronger the in-plane rigid diaphragm action is. The presence of rigid links with large diaphragm stiffness factor (say 1E10) could create numerical difficulties during the solution if 64-bit floating point solver is used. However, the unique 128-bit floating point solver in ENERCALC 3D makes this problem nonexistent in that much larger diaphragm stiffness factor (say 1E20) may be used without creating numerical difficulties during solution.

© 2026 ENERCALC, LLC

The program further provides the option to ignore the rigid diaphragm actions as an analysis option (Analysis > Analysis Options).

Multi-DOF Constraints

Tables > Multi-DOF Constraints provides access to the table where multi-DOF constraint data is reported.



Load Cases

Tables > Load Cases command is identical to the one found in the Create ribbon. It is provided here for convenience.

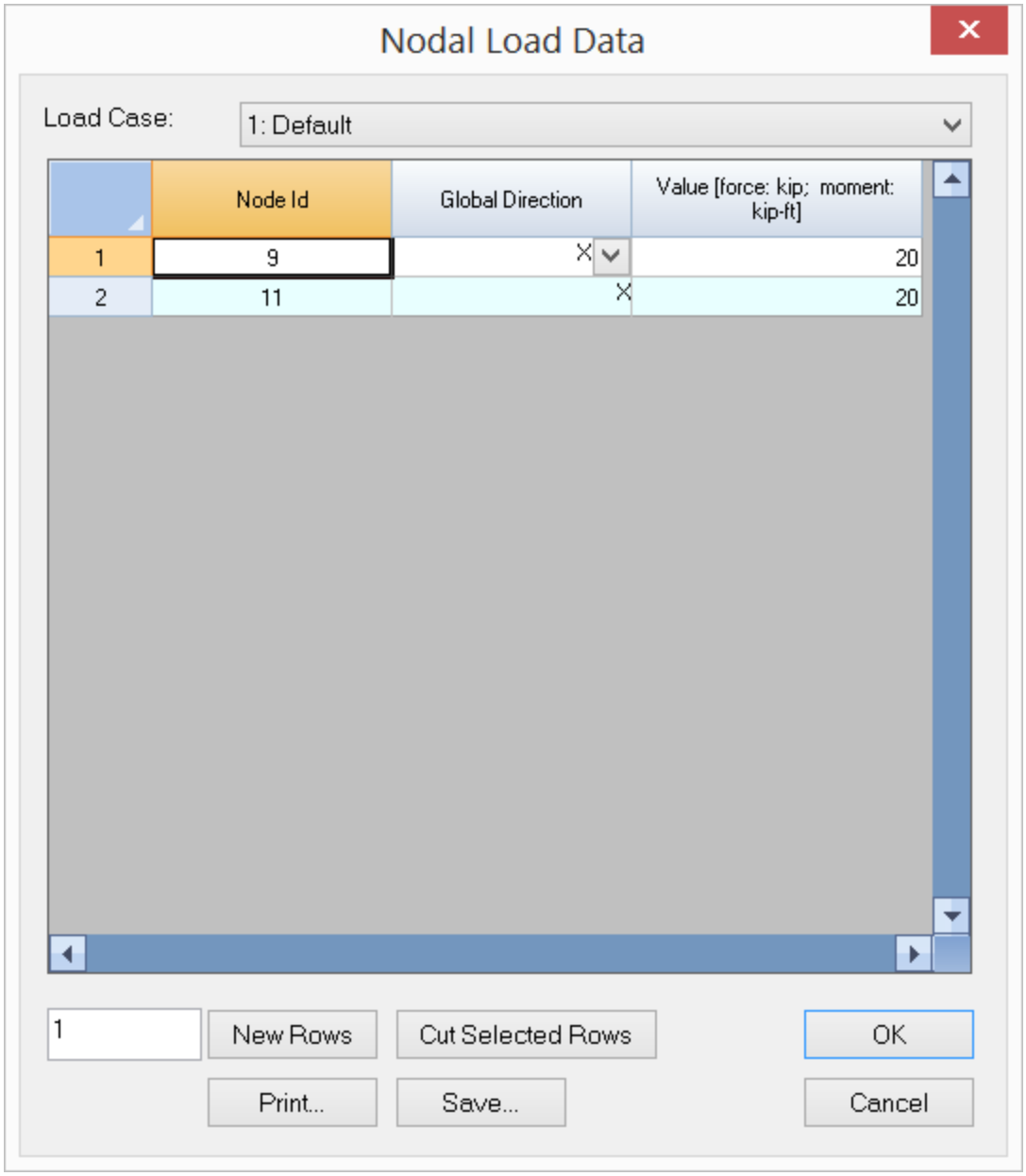
Load Combinations

Tables > Load Combinations command is identical to the one found in the Create ribbon. It is provided here for convenience.

© 2026 ENERCALC, LLC

Nodal Loads

Tables > Nodal Loads prompts you with the following dialog.



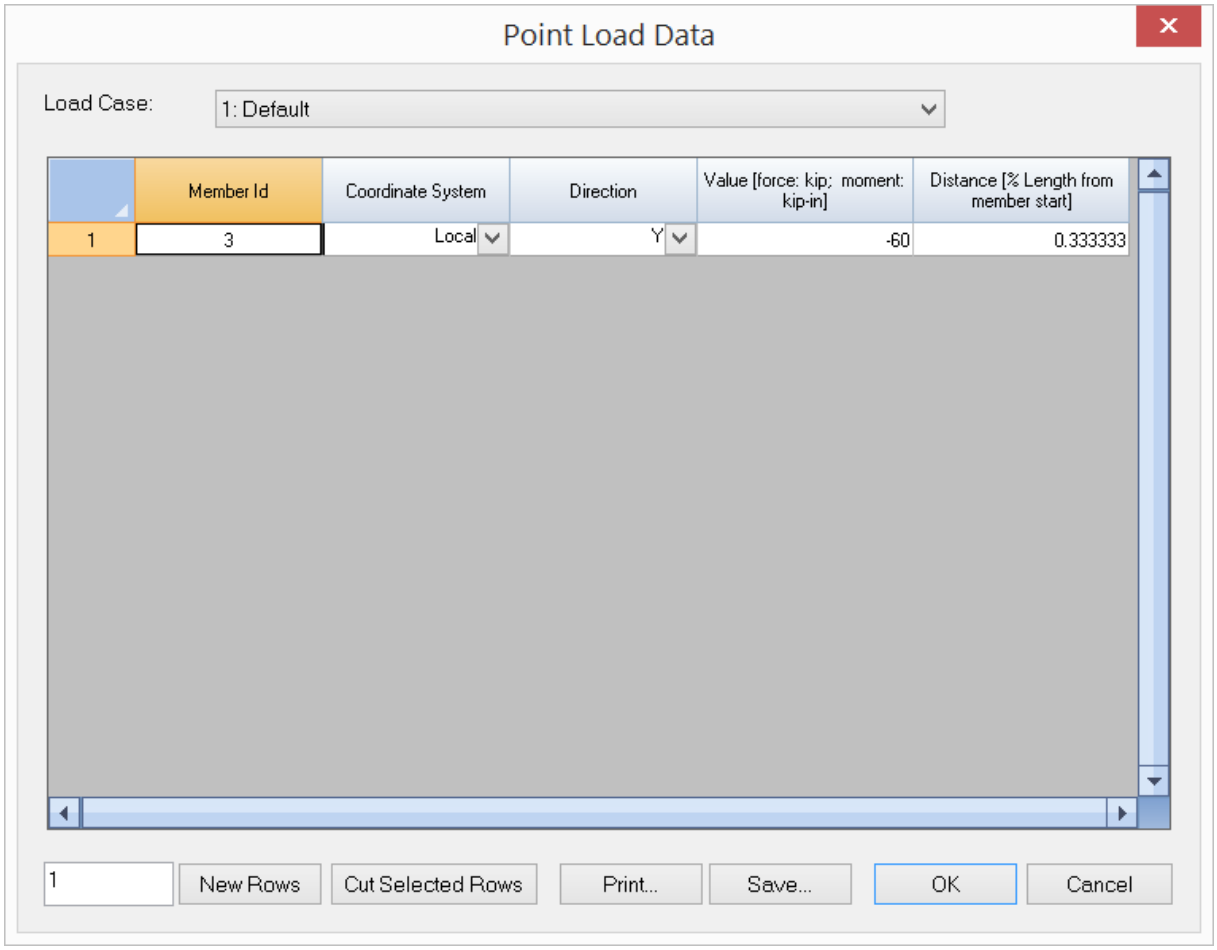
It allows you to enter nodal loads in a spreadsheet. Each nodal load includes the node ID, the load direction, and magnitude. The load direction is specified in the global coordinate system. The load is a force if the load direction is in the X, Y or Z direction and moment if in the OX, OY or OZ. The node ID must be valid (defined).

An empty row is allowed if all rows below it do not contain any non-empty fields. Selected rows (whole row must be selected) may be cut by clicking the button “Cut Selected Rows”.

© 2026 ENERCALC, LLC

Point Loads

Tables > Point Loads prompts you with the following dialog.



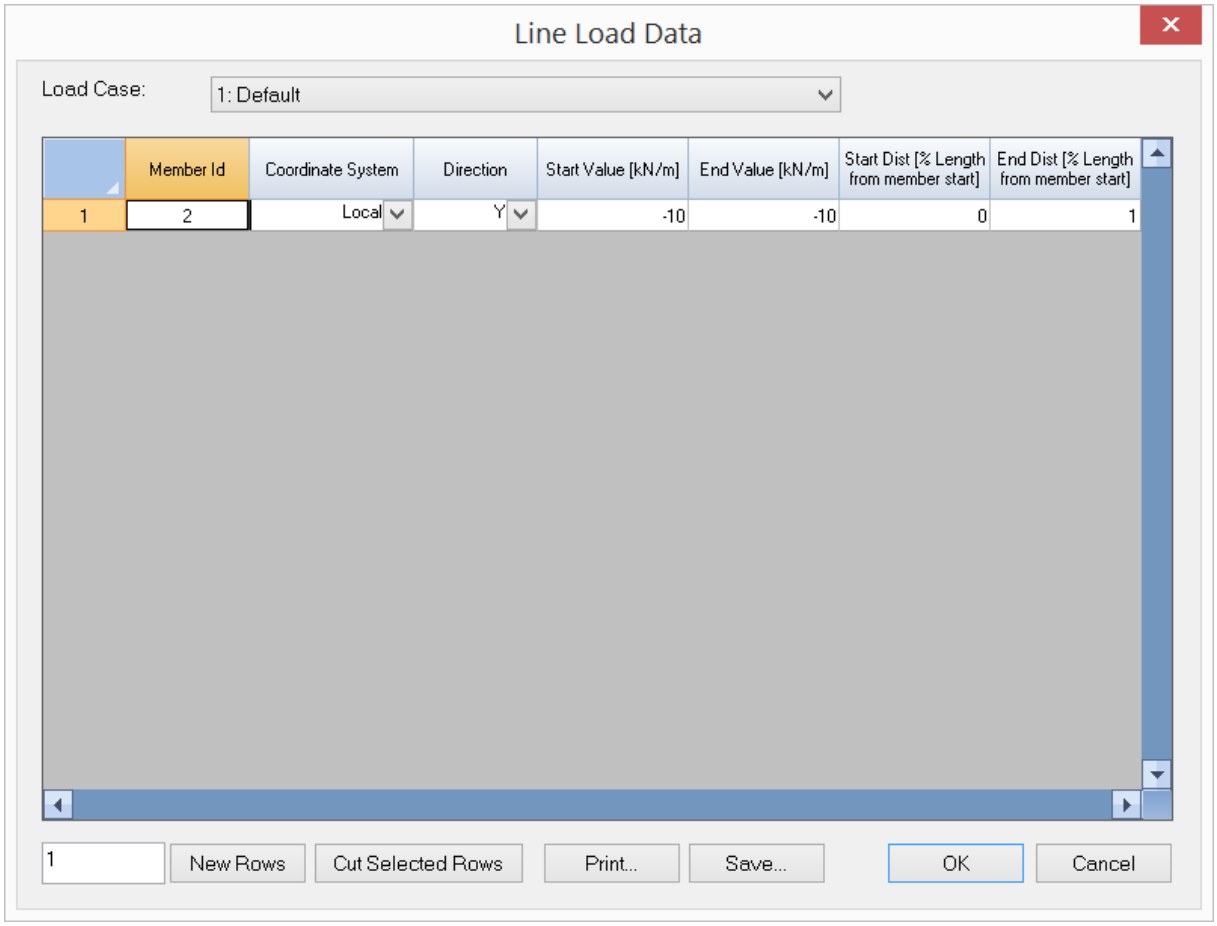
It allows you to enter member point loads in a spreadsheet. Each point load includes the member ID, the load coordinate system, direction, magnitude, and distance. The load is a force if the load direction is in the X, Y or Z direction and moment if in the OX, OY or OZ. The load distance is the ratio of load location (measured from the start of the member) to member length. The member ID must be valid (defined).

An empty row is allowed if all rows below it are empty. Selected rows (whole row must be selected) may be cut by clicking the button “Cut Selected Rows”.

© 2026 ENERCALC, LLC

Line Loads

Tables > Line Loads prompts you with the following dialog.



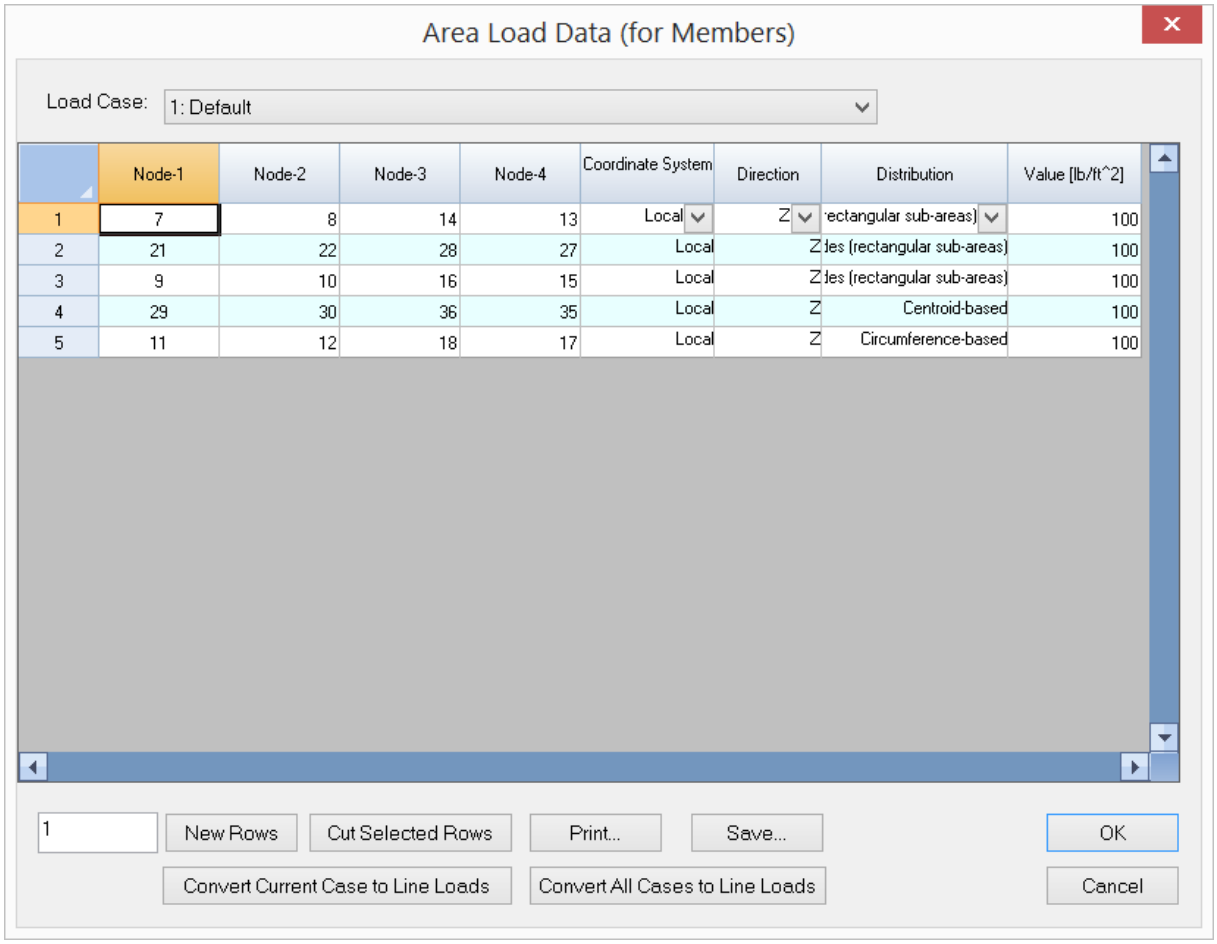
It allows you to enter member line loads in a spreadsheet. Each line load includes the member ID, the load coordinate system, direction, start and end magnitudes, and the start and end distances. The load is a force in the local or global X, Y, Z directions. The load distances are the ratio of load start and end locations (measured from the start of the member) to the member length. The member ID must be valid (defined).

An empty row is allowed if all rows below it are empty. Selected rows (whole row must be selected) may be cut by clicking the button “Cut Selected Rows”.

© 2026 ENERCALC, LLC

Area Loads

Tables > Area Loads prompts you with the following dialog.



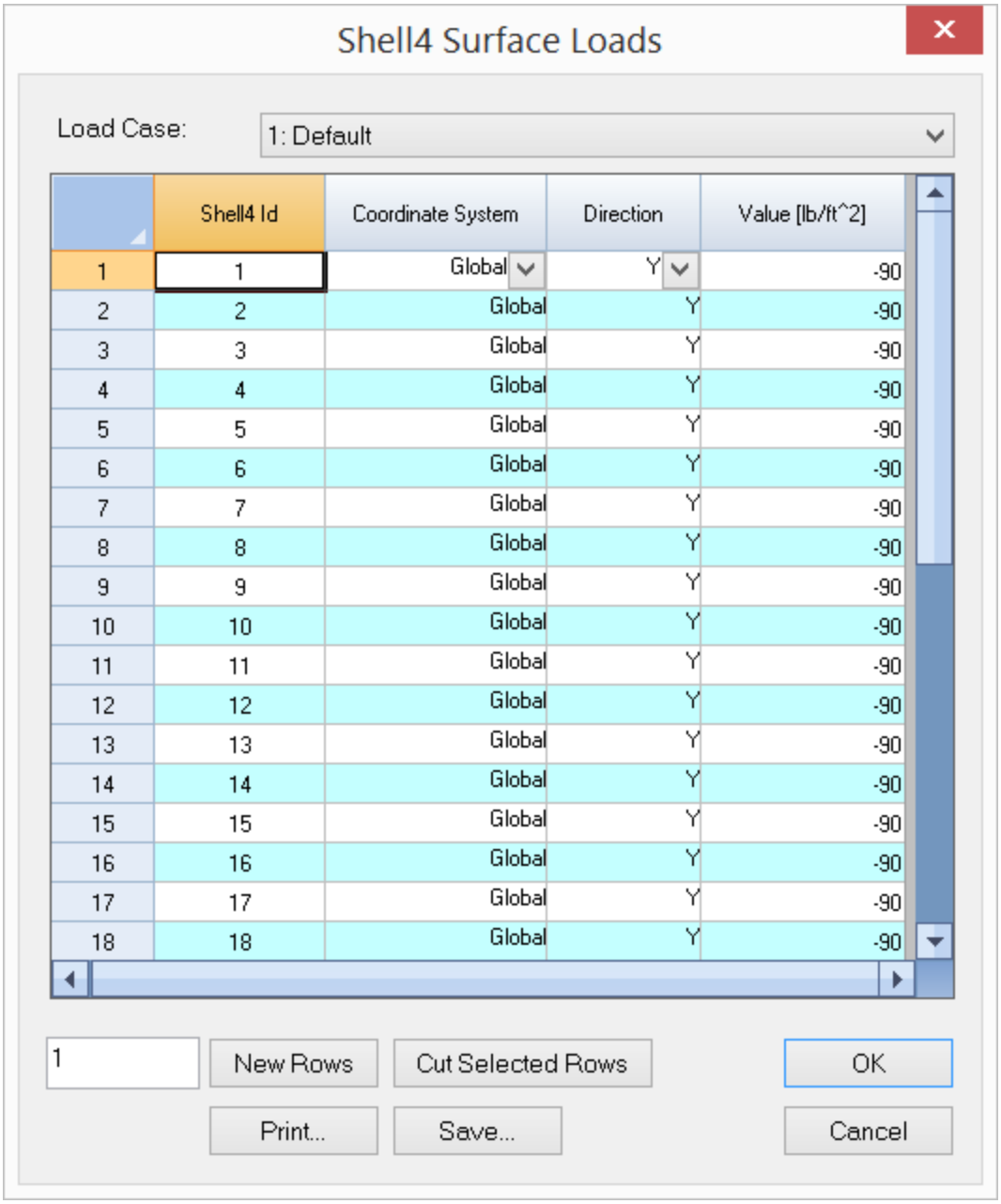
It allows you to enter member area loads in a spreadsheet. Each area load includes four node IDs, the load coordinate system, direction, load distribution, and load magnitude. The area load may be in global X, Y or Z direction, or in local Z direction. The four nodes form a quadrilateral load area and must be in the same plane. Node-4 may also be the same as Node-1, in which case the load area is a triangle. Area loads in one or all load cases may be converted to lines loads in their respective load cases.

An empty row is allowed if all rows below it are empty. Selected rows (whole row must be selected) may be cut by clicking the button “Cut Selected Rows”.

© 2026 ENERCALC, LLC

Surface Loads

Tables > Surface Loads prompts you with the following dialog.



It allows you to enter shell surface loads in a spreadsheet. Each surface load includes the shell ID, the load coordinate system, direction, and magnitude. The load is always a force in the local or global X, Y or Z directions. The shell ID must be valid (defined).

An empty row is allowed if all rows below it are empty. Selected rows (whole row must be selected) may be cut by clicking the button “Cut Selected Rows”.

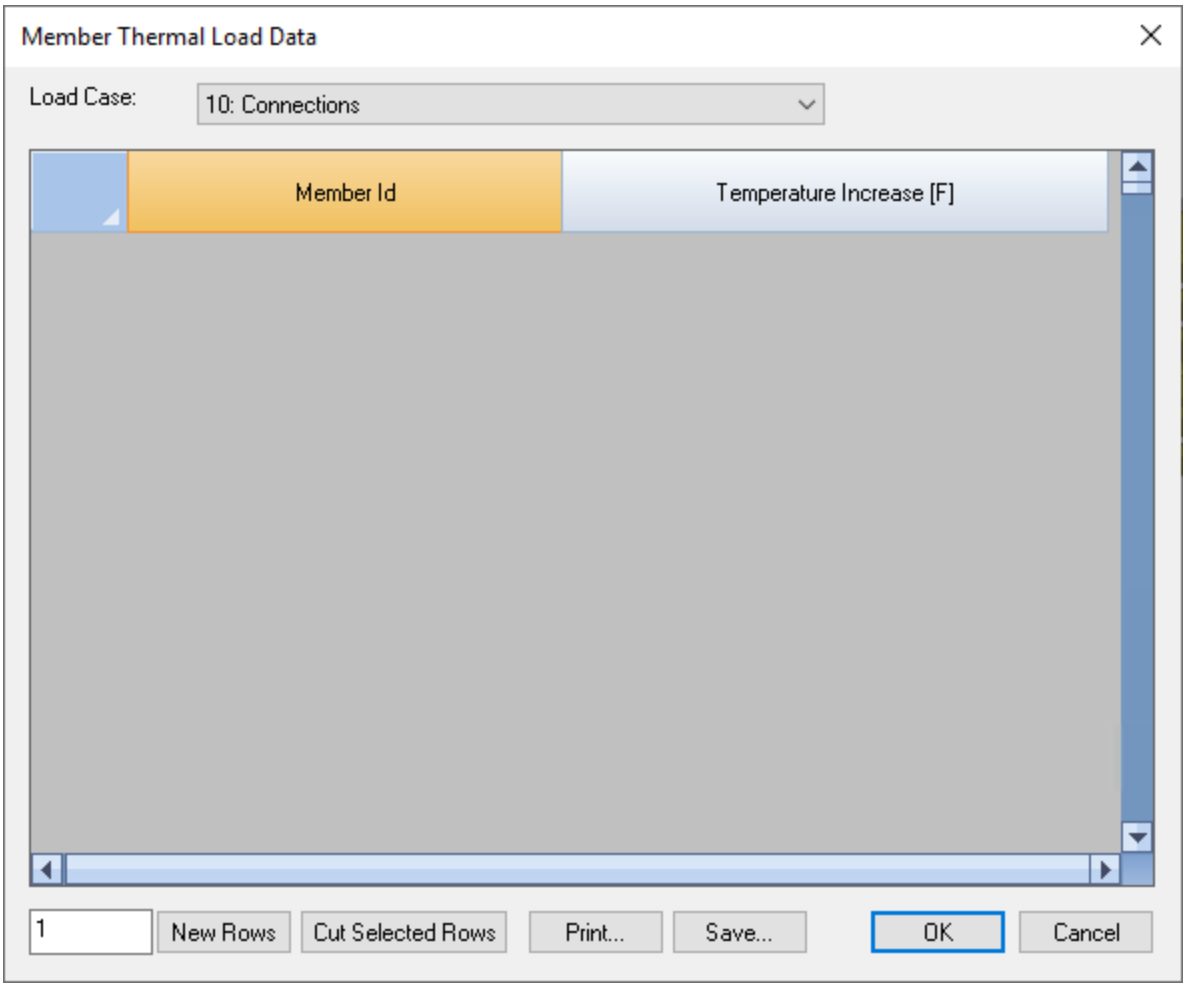
© 2026 ENERCALC, LLC

Self Weights

The Tables > Self Weights command is identical to the one found in the Create ribbon. It is provided here for convenience only.

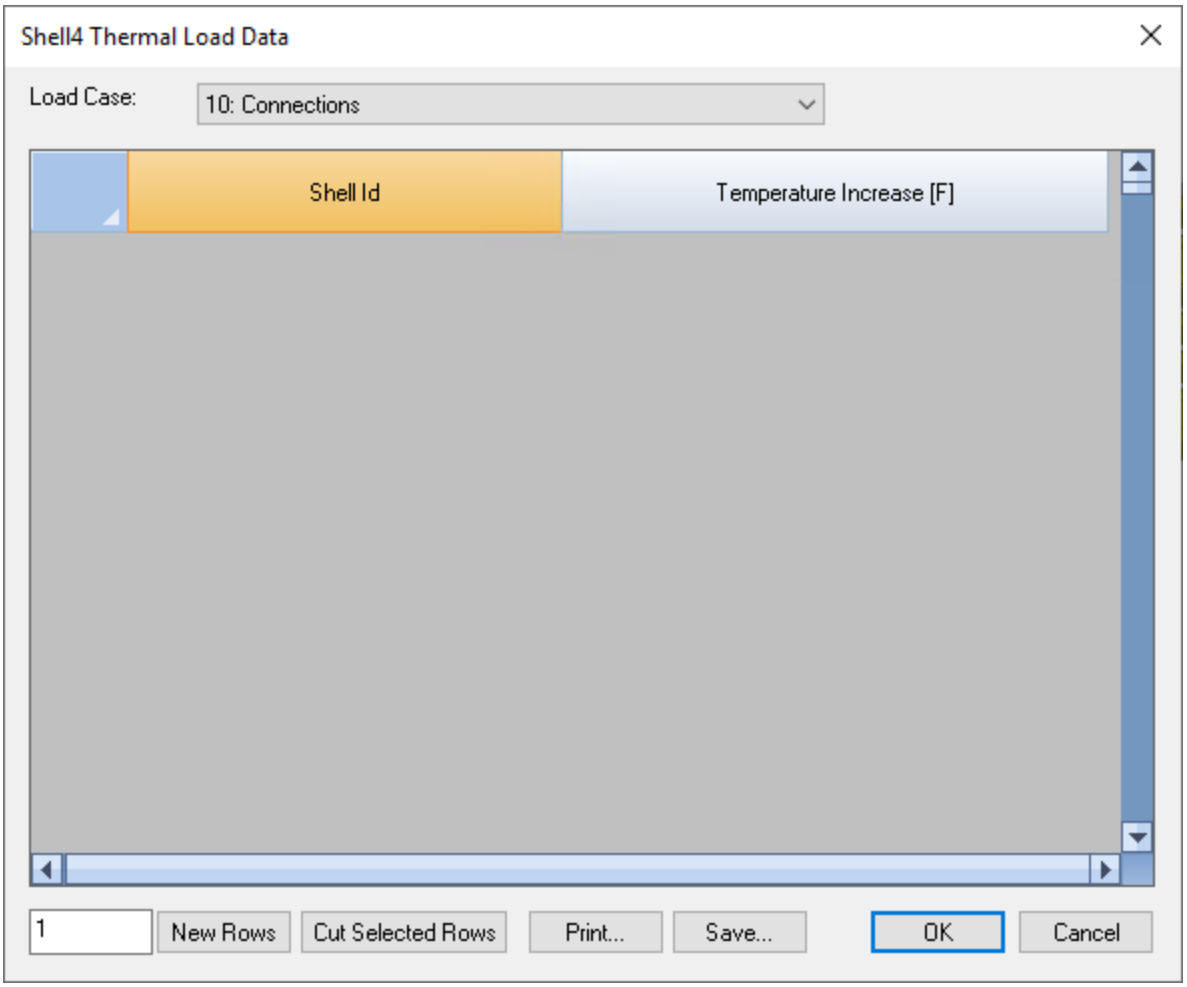
Thermal Loads

Tables > Thermal Loads > Member Thermal Loads displays a table of members and their temperature increase.



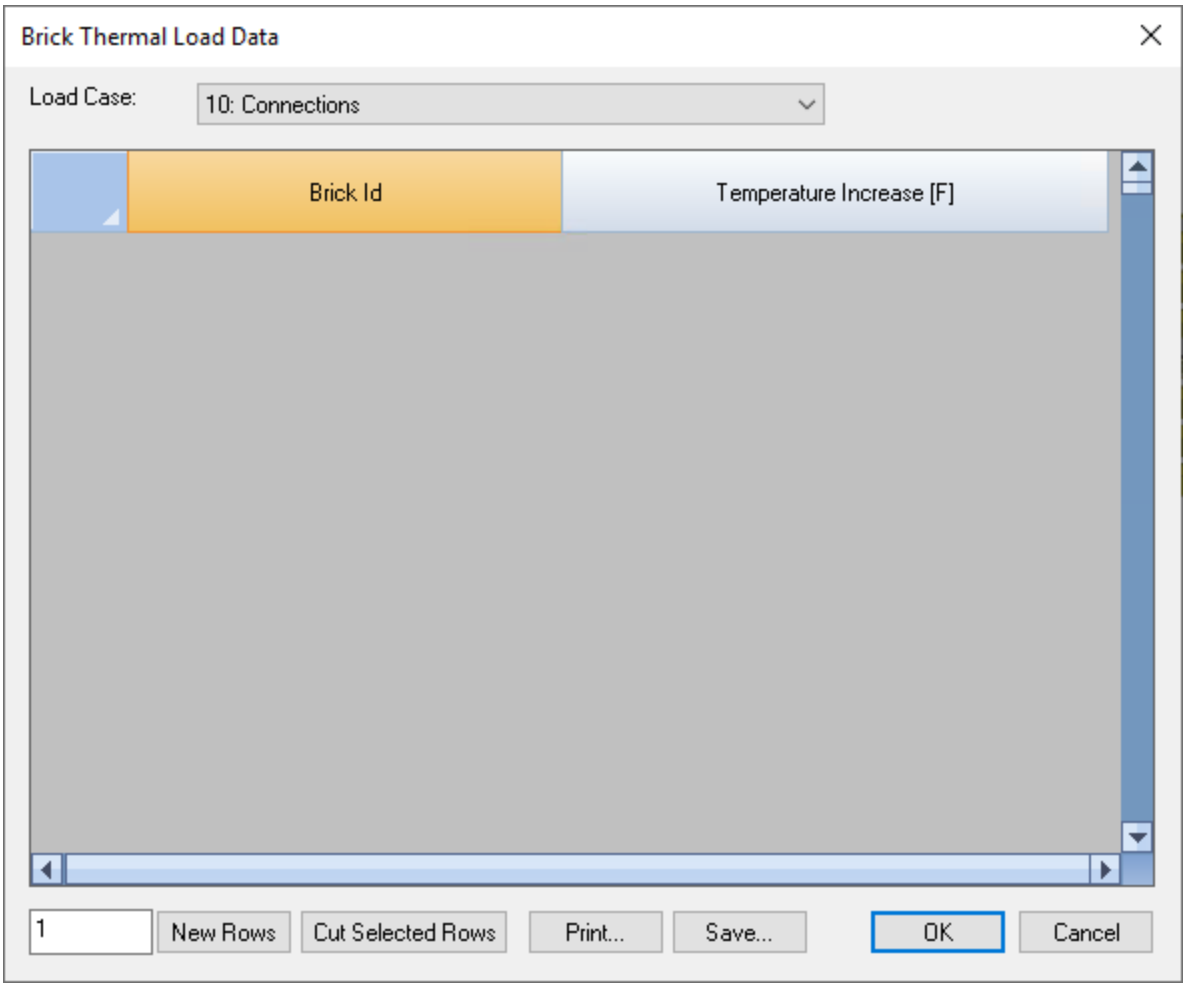
© 2026 ENERCALC, LLC

Tables > Thermal Loads > Shell Thermal Loads displays a table of shells and their temperature increase.



© 2026 ENERCALC, LLC

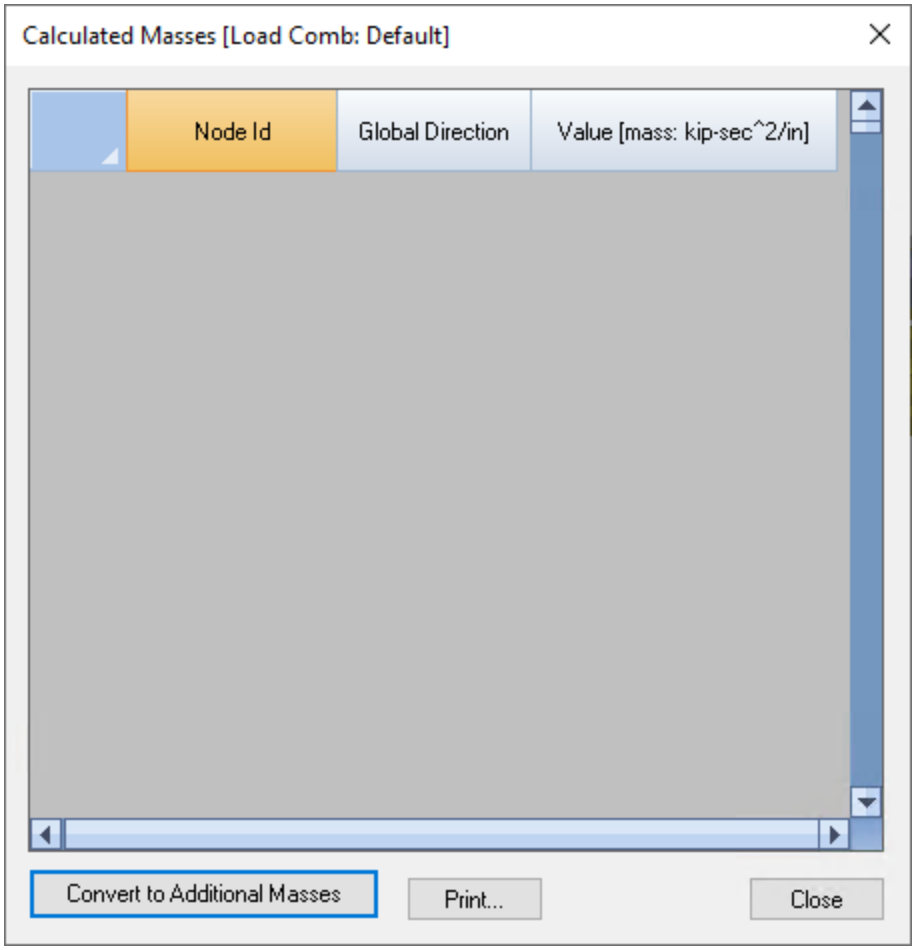
Tables > Thermal Loads > Brick Thermal Loads displays a table of bricks and their temperature increase.



© 2026 ENERCALC, LLC

Masses

Tables > Masses > Calculated Masses prompts you with the following dialog.

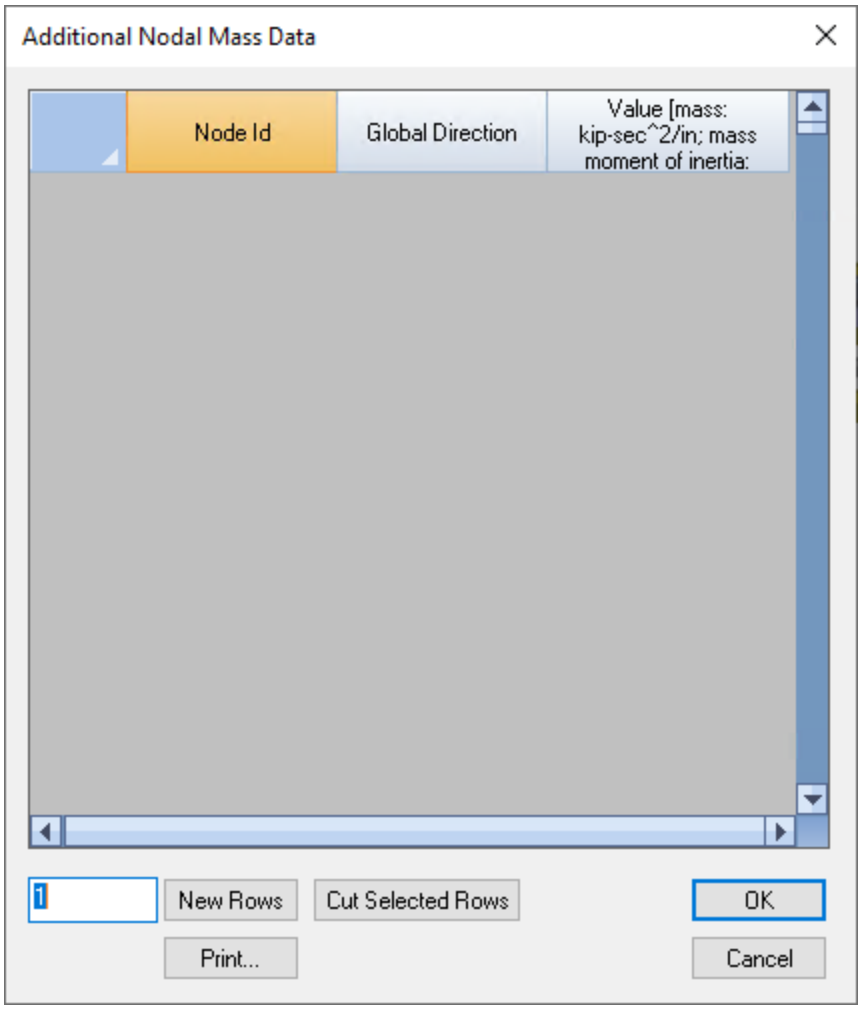


It allows you to view the masses calculated from the load combination for frequency analysis set in Analysis > Frequency Analysis. The program will automatically convert all forces (not moments) in the positive or negative gravity direction to masses and apply them in all available mass degrees of freedom.

Obviously, the calculated mass values cannot be modified. However, you may convert all the calculated masses to additional masses. In this case, the “Convert loads to masses” option in Analysis > Frequency Analysis will be disabled. This technique may be useful when you want to account for the influence of axial loads on frequencies.

© 2026 ENERCALC, LLC

Tables > Masses > Additional Masses prompts you with the following dialog.



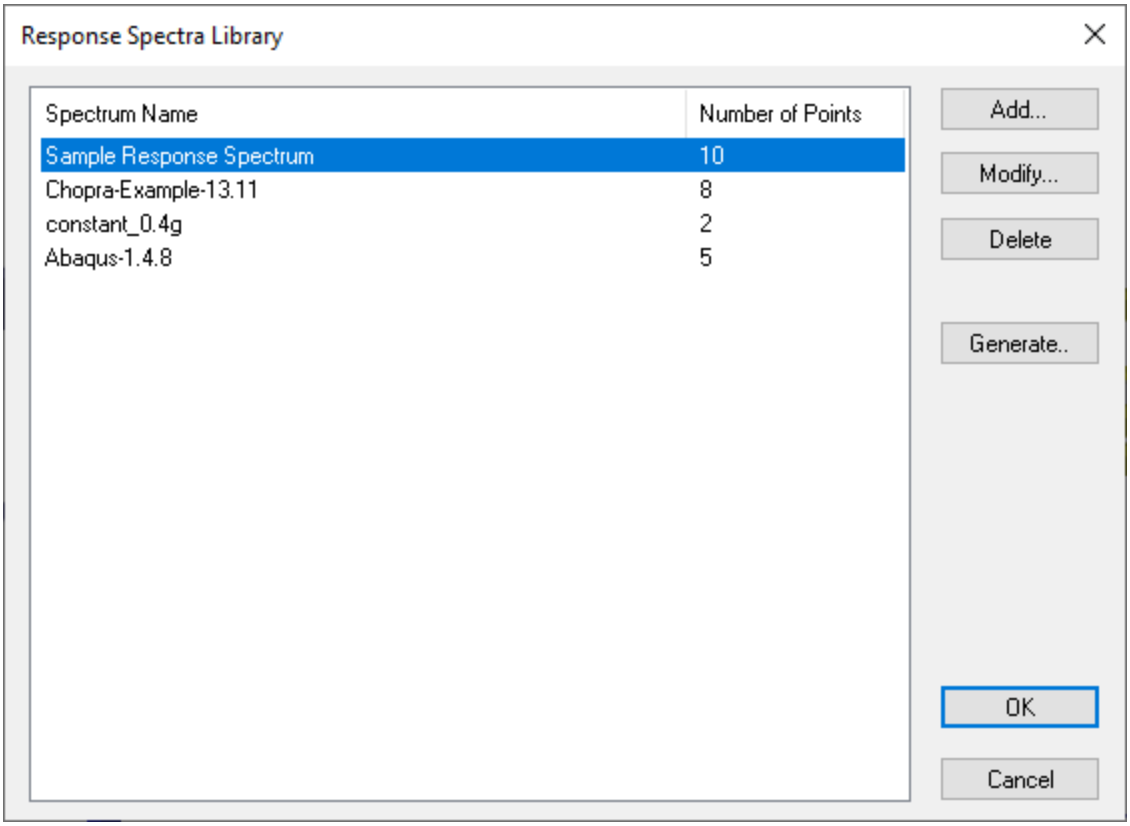
It allows you to enter additional nodal mass and nodal mass moment of inertia values. Each nodal mass or mass moment of inertia includes the node ID, the mass direction, and magnitude. The mass direction is specified in the global coordinate system. The unit of measurement for mass is force divided by the acceleration of gravity. For mass moment of inertia, the unit of measurement is mass times length units squared. The acceleration of gravity is generally takes as a constant value of 386.09 in/sec^2 or 9.8 m/sec^2.

An empty row is allowed if all rows below it do not contain any non-empty fields. Selected rows (whole row must be selected) may be cut by clicking the button “Cut Selected Rows”.

© 2026 ENERCALC, LLC

Response Spectra Library

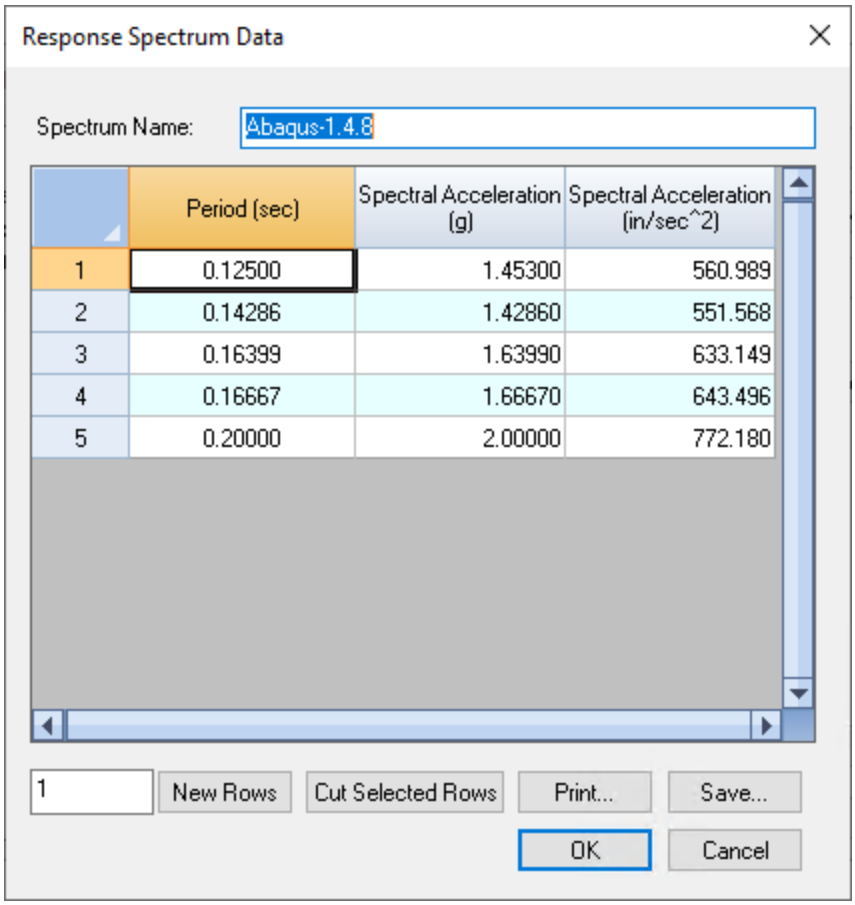
Tables > Response Spectra Library prompts you with the dialog below.



It allows you to define spectra for current and future projects. You can then use one or more spectra in Analysis > Response Spectrum Analysis.

You may view/modify a user-defined spectrum by double clicking the spectrum as shown in the screen capture below.

© 2026 ENERCALC, LLC

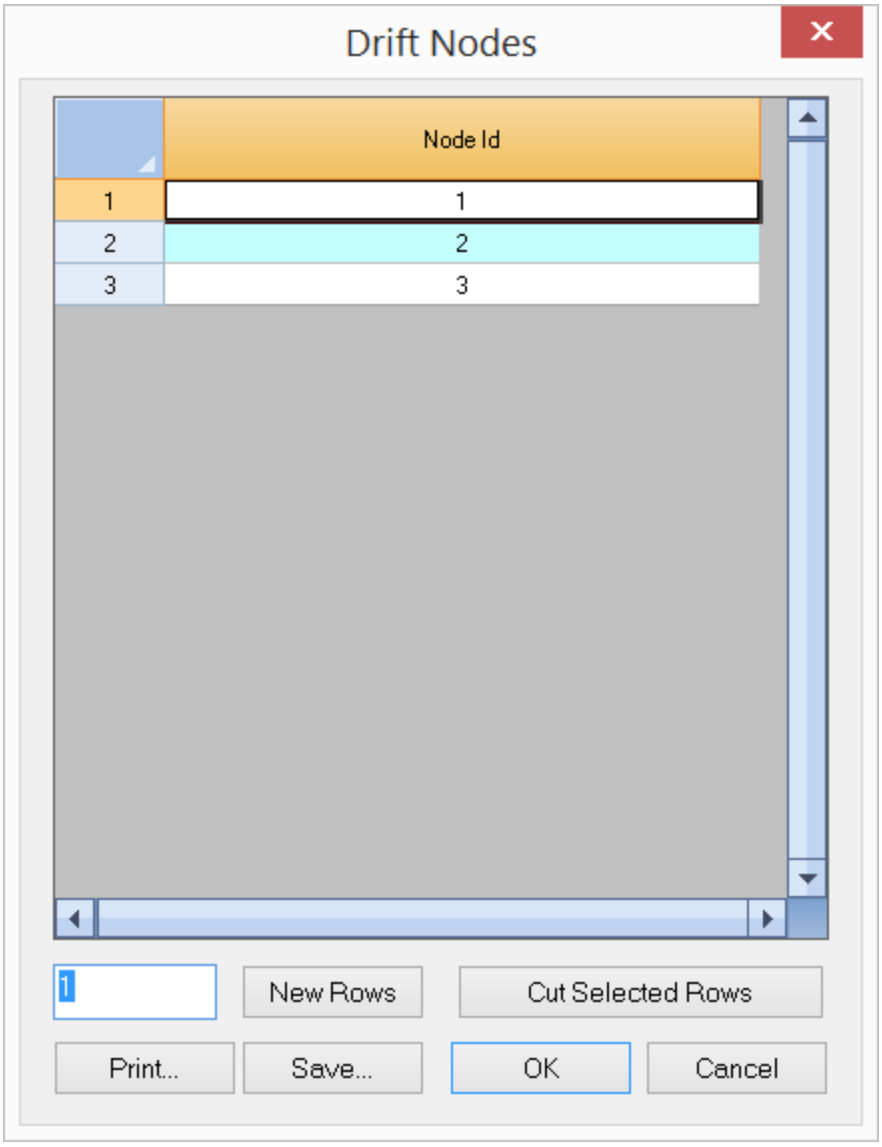


The first spectrum can not be edited or deleted. Spectra generated based on building codes cannot be edited but can be deleted.

© 2026 ENERCALC, LLC

Story Drift Nodes

Tables > Story Drift Nodes prompts you with the following dialog.

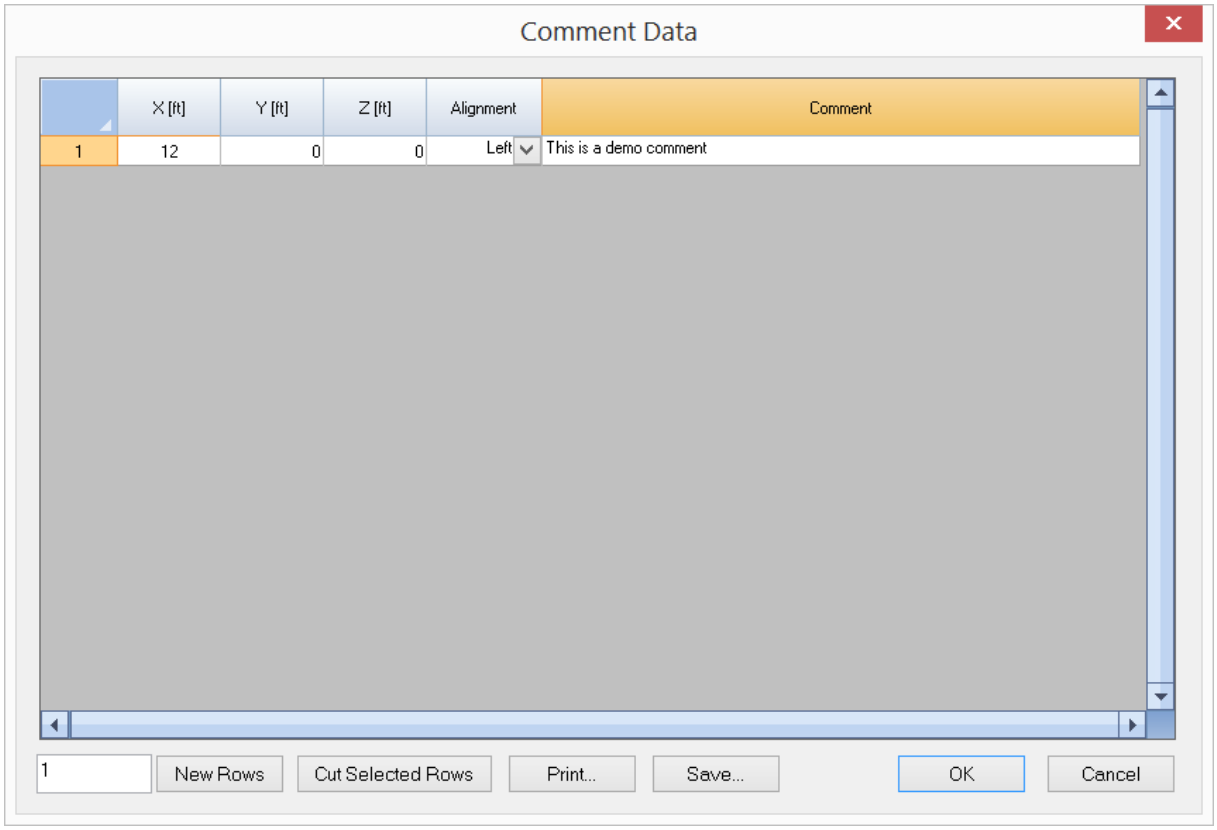


It allows you to enter nodes that will be used for floor drift calculation. An empty row is allowed if all rows below it do not contain any non-empty fields. Selected rows (whole row must be selected) may be cut by clicking the button “Cut Selected Rows”.

© 2026 ENERCALC, LLC

Comments

Tables > Comments prompts you with the following dialog.



It allows you to add or delete comments at different locations in the model. You may also add an individual comment using View > Comments. A comment must be less than 256 characters in length.

An empty row is allowed if all rows below it do not contain any non-empty fields. Selected rows (whole row must be selected) may be cut by clicking the button “Cut Selected Rows”.

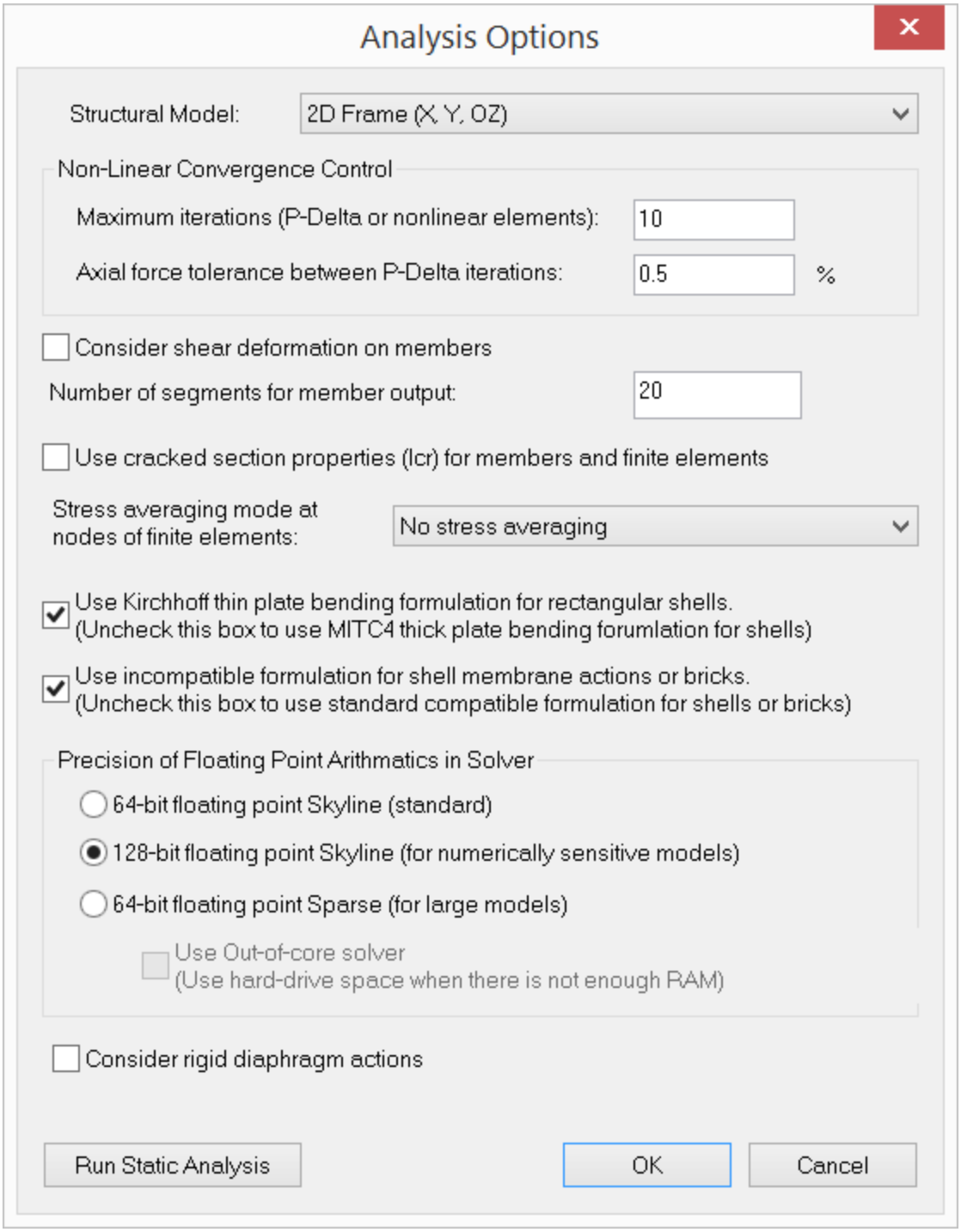
Analysis

The Analysis ribbon provides commands to set the analysis options and to perform analyses. For more information on analysis, please refer to "Technical Topics".

© 2026 ENERCALC, LLC

Analysis Options

Analysis > Analysis Options prompts you with the following dialog.



It allows you to set important options before performing analysis on the model.

© 2026 ENERCALC, LLC

The “Model type” determines the type of the model to analyze. Model type “3D Frame & Shell” is the most general. It has all six degrees of freedom (DOFs) available to every node in the model. Any model may be analyzed with this model type. However, computer memory or time may be wasted if a simpler model can be used.

Model type “2D Frame” may be used to model a 2D frame (beams and trusses) structure in the XY plane. Only three DOFs (D_x, D_y and D_{oz}) are available to every node in the model. The rest of the DOFs are suppressed.

Model type “3D Truss” may be used to model 3D truss structures. Only three DOFs (D_x, D_y, and D_z) are available to every node in the model. The rest of the DOFs are suppressed. If the model contains both 3D trusses and beams, “3D Frame & Shell” model type must be used and appropriate moment releases assigned.

Model type “2D Truss” may be used to model 2D truss structures in the XY plane. Only two DOFs (D_x and D_y) are available to every node in the model. The rest of the DOFs are suppressed. If the model contains both 2D trusses and beams, “2D Frame” model type must be used and appropriate moment releases assigned.

Model type “2D Plate Bending” may be used to model 2D plate bending structures such as flat slabs or mat foundations in the XY plane. It uses only the plate bending action of the shell formulation. Only three DOFs (D_z, D_{ox} and D_{oy}) are available to every node in the model. The rest of the DOFs are suppressed. The self weight should be in either –Z or +Z direction depending on your sign convention for loads.

Model type “2D Plane Stress” may be used to model 2D plane stress structures such as shear walls in the XY plane. It uses only the membrane action of the shell formulation. Only two DOFs (D_x and D_y) are available to every node in the model. The rest of the DOFs are suppressed.

Model type “3D Brick” may be used to model 3D solid structures. Only three DOFs (D_x, D_y, D_z) are available to every node in the model. The rest of the DOFs are suppressed.

Non-linear convergence control includes the maximum iterations and axial force tolerance between adjacent P-Delta iterations. The maximum iterations apply to both P-Delta analysis and analysis involving non-linear springs. It is provided to avoid excessive number of nonlinear iterations during the solution. A default value of 10 is usually sufficient. Axial force tolerance between adjacent P-Delta iterations reflects the actual convergence of the P-Delta analysis. The default value of 0.5% should be good for most cases. It is a good idea to perform a linear analysis before the P-Delta analysis. In this way, you may identify any problems in the model before the more rigorous analysis option is undertaken.

By default, the program considers shear deformations on members in the model. You must also set shear areas of member sections for this option to take effect. To do that, click Create (or Modify) > Member Properties > Member Sections. You may ignore member shear deformations by unchecking “Consider shear deformation on members”. Generally, shear deformations on members are insignificant. However, you should check this option when

© 2026 ENERCALC, LLC

members are of relatively great depths. *Shear deformation, when considered, applies to both the element stiffness matrix and local (segmental) deflections.*

The number of segments for member segmental output may be set from 1 to 127. A value of 20 segments is recommended in most cases. More segments produce more accurate results, but require more usage of computer memory. The accuracy may be reflected in the smoothness of moment, shear and deflection diagrams. Since member local deflection is computed based on the moment and shear diagrams, a value of more than 20 segments may be needed if very accurate local deflection is desired.

You may specify whether to use cracked section properties for the analysis. The cracking factors are specified in Create (or Modify) > Member Properties > Cracking Factors or Concrete Design > Cracking Factors. Cracking factors will not be applied until “Use cracked section properties (Icr) for members and finite elements” is checked here. This option is given so that you do not need to reenter cracking factors if you decide to use gross section properties in a different analysis run.

Usually, stresses in finite elements are not continuous across element boundary. You may average them for adjacent shells/bricks at nodes. This usually makes the results more accurate and the contours smoother. However, it may also disguise insufficient convergence for an unsatisfactory (coarse) finite element mesh. *Obviously, stress averaging can only apply to adjacent elements that have compatible local coordinate systems. For planar elements, stress averaging should only apply to adjacent elements that share the same local coordinate systems. Special attention should be given to shear stress averaging at supports since shears of adjacent elements may have opposite signs.*

By default, the program uses the MITC4 for shell bending formulation. The MITC4 is a thick plate formulation and accounts for the out-of-plane shear deformation. However, if the shell elements are all rectangular, you may use the classical Kirchhoff plate bending formulation. The Kirchhoff plate element is a thin plate formulation and ignores the out-of-plane shear deformation.

For the membrane formulation of the shell element or of the brick element, incompatible modes may be added to the standard isoparametric (compatible) formulation. The incompatible shell element models the in-plane bending more accurately than the standard compatible element. The incompatible brick element, which produces much more accurate results than the compatible one, should almost always be preferred.

There are three kinds of solvers available in ENERCALC 3D:

Double precision 64-bit floating point Skyline solver:
- standard solver similar to many analysis programs in the market
- enough for most structures

Double precision 64-bit floating point Sparse solver:
- fastest, but lacks informative messages when something goes wrong during a solution
- can only be used for static analysis
- useful to solve extremely large structural models
- offers an out-of-core approach to minimize the requirement of computer memory

© 2026 ENERCALC, LLC

Quad precision 128-bit floating point Skyline solver:

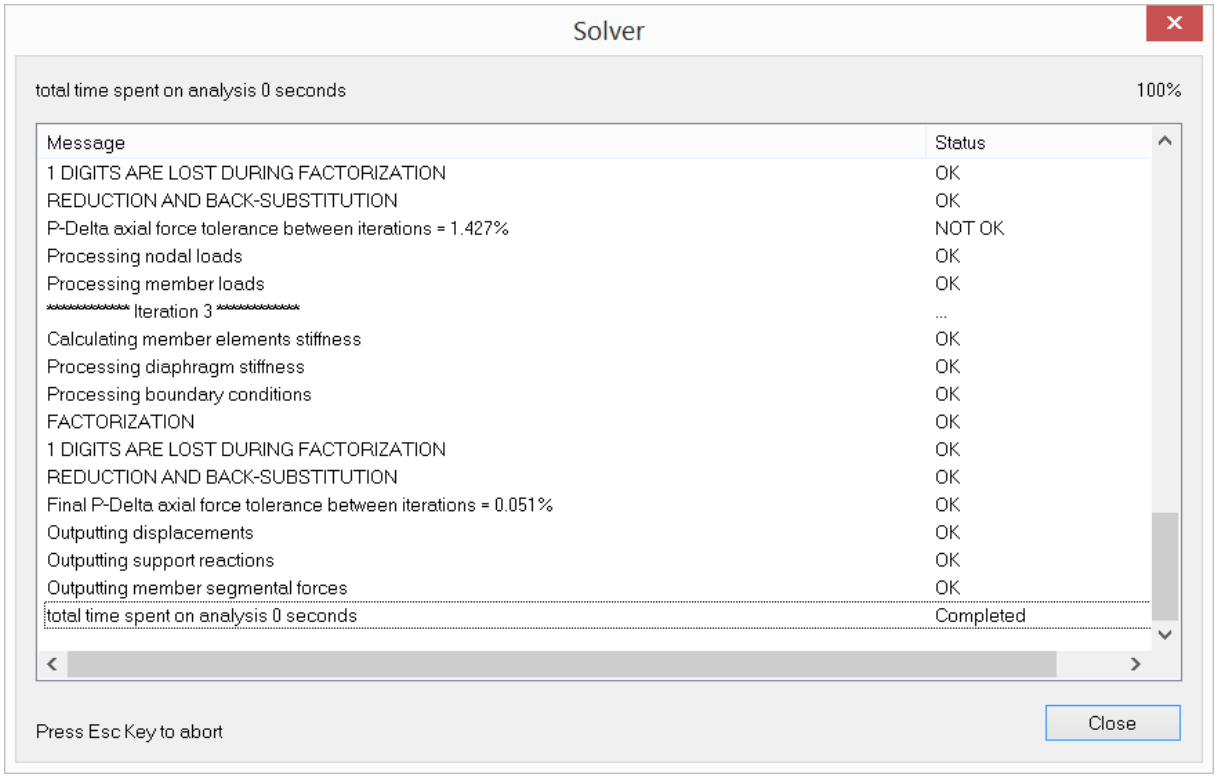
- provides an invaluable alternative for some large or numerically sensitive models where the 64-bit floating point (double precision) solver may fail
- extremely stable and accurate, but relatively slow
- the recommended solver if the model contains rigid diaphragms to avoid numerical difficulties.

You have the option to consider rigid diaphragm actions during the solution. This is useful to ignore rigid diaphragms without deleting the existing diaphragms.

Static Analysis

Analysis > Static Analysis performs the static analysis of the model. You should set the appropriate analysis options before running this command. To do that, just click Analysis > Analysis Options.

The following dialog is displayed showing the progress of the solver.



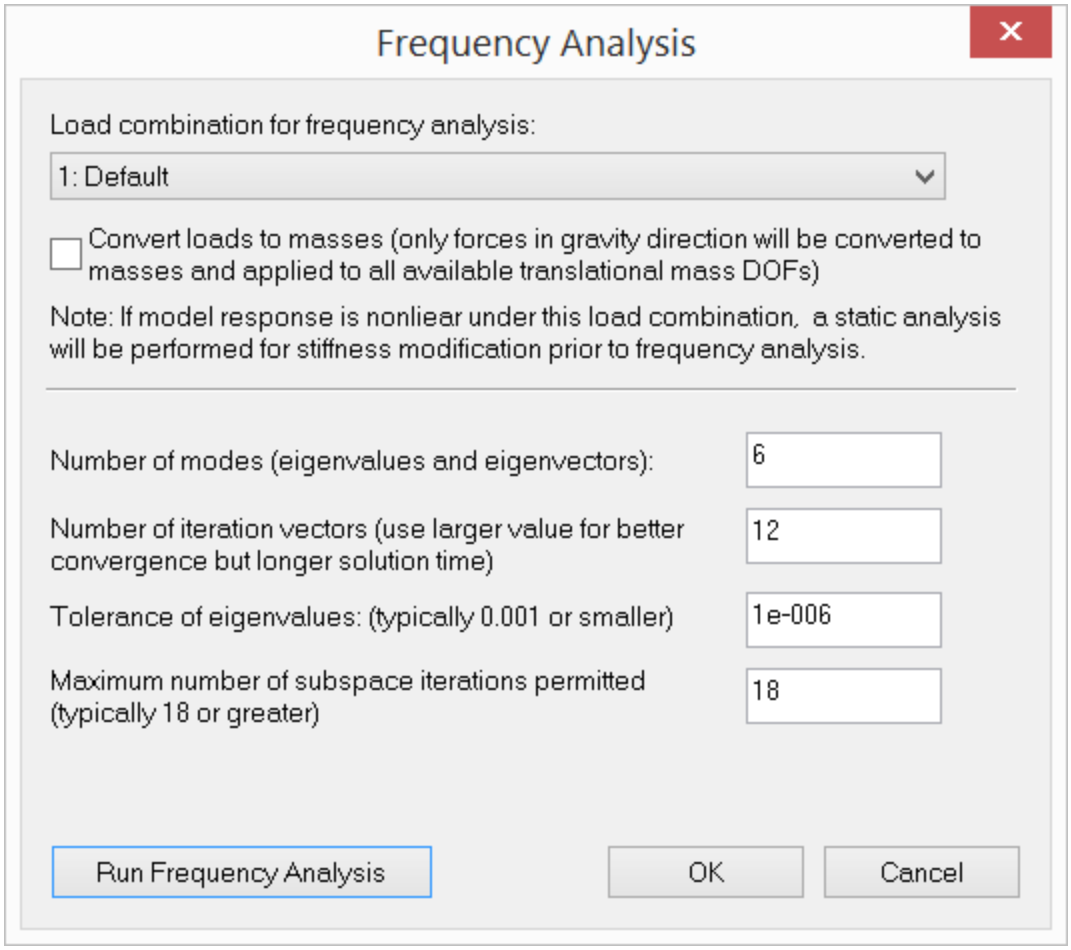
The solver first solves all linear load combinations, then all nonlinear load combinations. If the model contains compression-only or tension-only springs, every load combination is a nonlinear load combination. Otherwise, only P-Delta load combinations are nonlinear. The nonlinear load combinations are solved iteratively and therefore may require considerable solution time.

You may abort the solving process by pressing ESC if a non-sparse solver is used.

© 2026 ENERCALC, LLC

Frequency Analysis

Analysis > Frequency Analysis prompts you with the following dialog.



It allows you to set important options before performing frequency analysis on the model.

The load combination for mass and stiffness must be specified. If you want to input nodal mass and/or mass moment of inertia directly, you may do so from Create > Generate Loads > Additional Masses or Tables > Masses > Additional Masses. The load combination is also used to form the correct stiffness matrix if the model response is not linear.

The number of modes is used to determine how many frequencies and mode shapes are to be computed. This value must be less than the total number of mass degrees of freedom. Practically speaking, only the lowest eigen modes are used for design purposes.

The number of iteration vectors q is normally set as the minimum of (2 * p, p + 8), where p is the number of modes requested [Ref. 1]. A higher convergence rate can be achieved by using more iteration vectors. This may be necessary if some eigen modes are missing after the solution or if the solution becomes unstable.

© 2026 ENERCALC, LLC

The tolerance of eigenvalues is used to measure the convergence of eigenvalues during each successive solver iteration. It is expressed as the following:

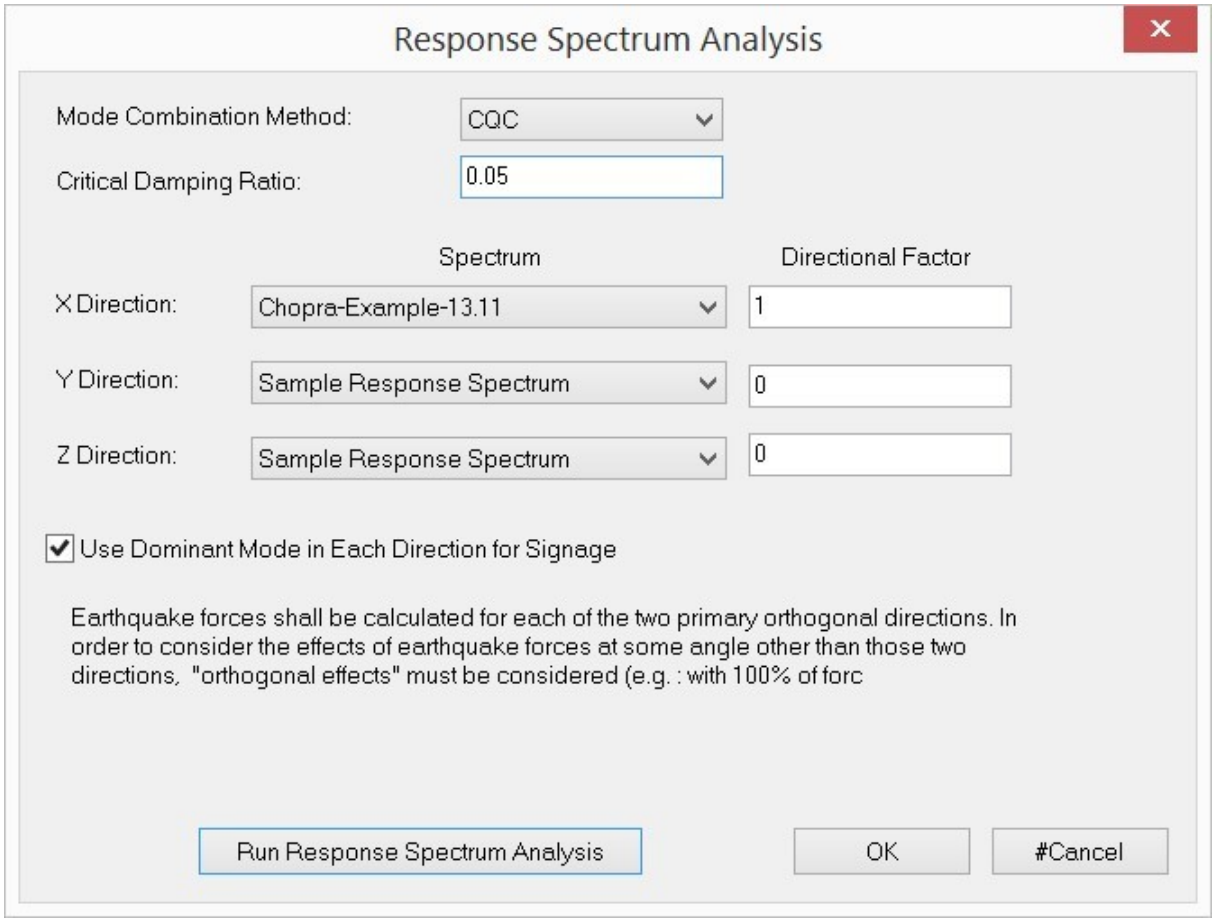
(i = 1, 2, … number of requested modes)

where k is the subspace iteration counter.

A maximum number of subspace iterations is used to prevent excessive solution time. If the solver reaches this limit without convergence, the eigen results should not be trusted.

Response Spectrum Analysis

Analysis > Response Spectrum Analysis prompts you with the dialog below.



It allows you to perform response spectrum analysis in global X, Y and/or Z directions. There are three mode combination methods available in the program: CQC (complete quadratic combination), SRSS (Square root of sum of squares) and ABSSUM (absolute sum). CQC method for modal combination is applicable to a wider class of structures and is therefore recommended method. Critical damping ratio (0 <= damp < 1.0) affects CQC results. When critical damping ratio is 0, CQC method is the same as SRSS method.

© 2026 ENERCALC, LLC

You must first run from Analysis > Frequency Analysis prior to running this command. Response spectra can be defined from Tables > Response Spectra Library. Inertia forces in global direction X, Y or/and Z from response spectrum analysis will be calculated and then converted to nodal loads.

These nodal forces will be placed in respective load cases such as INERTIA_LOADCASE_X_MODE_1, INERTIA_LOADCASE_X_MODE_2 etc. Existing loads in these load cases will be deleted prior to the load conversion.

In addition, response spectrum load combinations **INERTIA_LOADCOMB_X_MODE_1, INERTIA_LOADCOMB_X_MODE_2 etc.** will be created or recreated.

Static analysis will be performed on spectrum load combinations (as well as normal user-defined load combinations) automatically. Modal combinations will be subsequently calculated for results such as displacements, forces and stresses etc. using CQC, SRSS or ABSSUM on the response spectrum load combinations. Normally, modal combination results are all positive due to the sign lost during SRSS, CQC and ABSSUM procedures. However, you can choose to use signage for modal combination results based on the dominant mode (with maximum participation factor) in each global direction.

Modal combination results is done in each global direction first. Using directional factors, these directional modal results will be combined into final modal combination results, which can be added to any user-defined load combination results if non-zero response spectrum load factor is specified in the load combination definition (see Loads > Load Combinations).

View Log File

Analysis > View Log File initiates the same command as File > View Log File.

Analysis Results

The Analysis Results ribbon provides commands to view and print analysis results for each load combination.

Show Results for Selected Entities Only

Analysis Results > Show Results for Selected Entities Only is a toggle to control what is displayed when results are viewed.

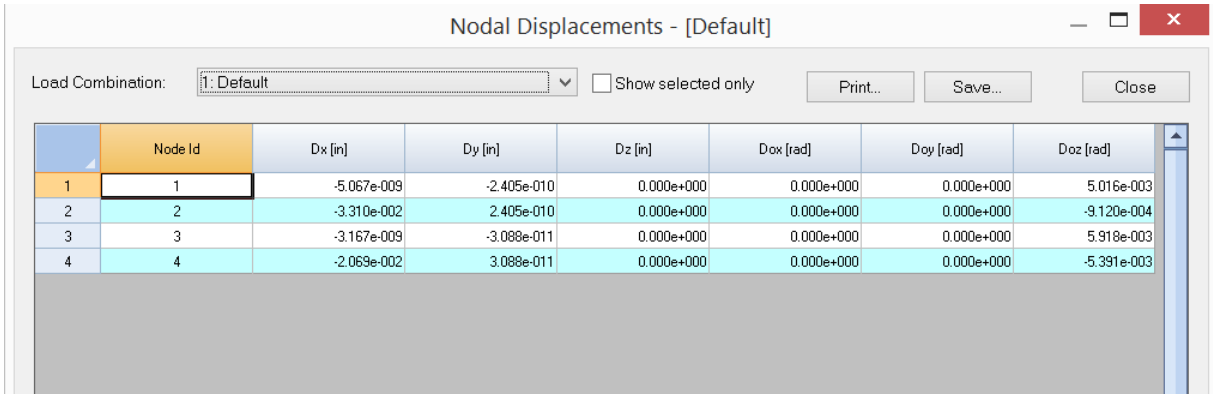
Results Load Combination

Analysis Results > Results Load Combination is a dropdown selector that establishes which load combination will be used for the display of results.

© 2026 ENERCALC, LLC

Nodal Displacements

Analysis Results > Nodal Displacements displays the following dialog.

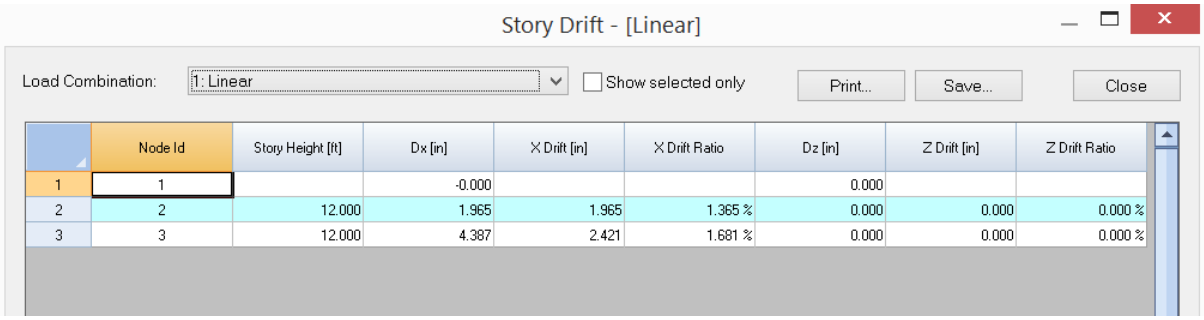


	Node Id	Dx [in]	Dy [in]	Dz [in]	Dox [rad]	Doy [rad]	Doz [rad]
1	1	-5.067e-009	-2.405e-010	0.000e+000	0.000e+000	0.000e+000	5.016e-003
2	2	-3.310e-002	2.405e-010	0.000e+000	0.000e+000	0.000e+000	-9.120e-004
3	3	-3.167e-009	-3.088e-011	0.000e+000	0.000e+000	0.000e+000	5.918e-003
4	4	-2.069e-002	3.088e-011	0.000e+000	0.000e+000	0.000e+000	-5.391e-003

It allows you to view nodal displacements for each load combination. The displacements for each node include three translational components D_x, D_y and D_z and three rotational components D_{ox}, D_{oy} and D_{oz}. You have the option to view the displacements for the selected nodes only.

Story Drifts

Analysis Results > Story Drifts displays the following dialog.



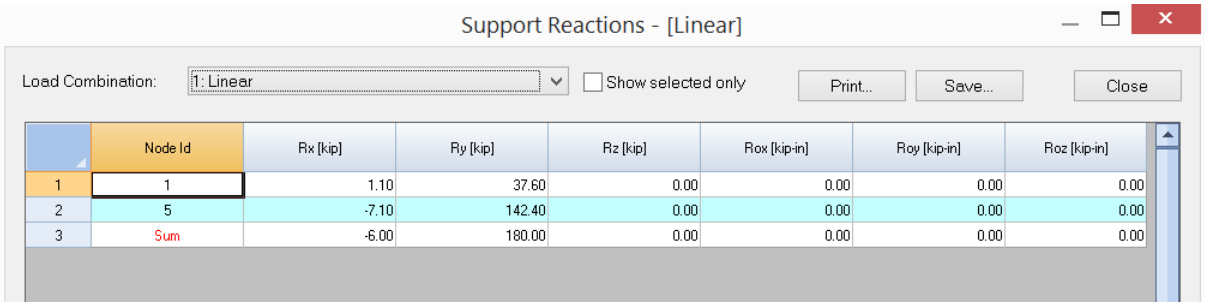
	Node Id	Story Height [ft]	Dx [in]	X Drift [in]	X Drift Ratio	Dz [in]	Z Drift [in]	Z Drift Ratio
1	1		-0.000			0.000		
2	2	12.000	1.965	1.965	1.365 %	0.000	0.000	0.000 %
3	3	12.000	4.387	2.421	1.681 %	0.000	0.000	0.000 %

It allows you to view story drifts for each load combination. You have the option to view the story drifts for the selected nodes only.

© 2026 ENERCALC, LLC

Support Reactions

Analysis Results > Support Reactions displays the following dialog.

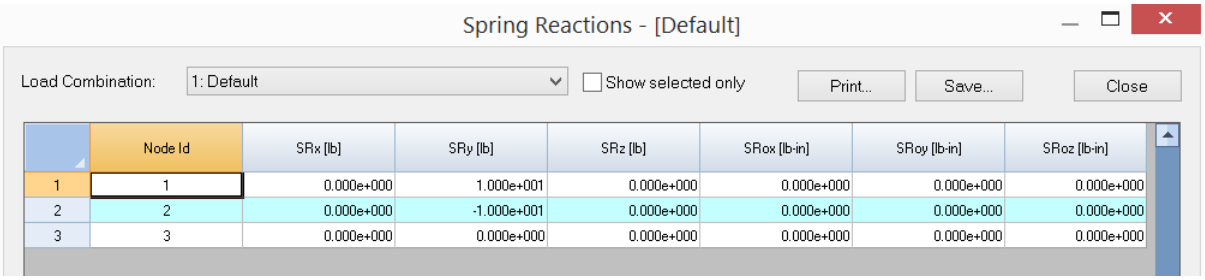


	Node Id	Rx [kip]	Ry [kip]	Rz [kip]	Rox [kip-in]	Roy [kip-in]	Roz [kip-in]
1	1	1.10	37.60	0.00	0.00	0.00	0.00
2	5	-7.10	142.40	0.00	0.00	0.00	0.00
3	Sum	-6.00	180.00	0.00	0.00	0.00	0.00

It allows you to view support reactions for each load combination. The reactions for each support include three force components R_x, R_y and R_z and three moment components R_{ox}, R_{oy} and R_{oz}. You have the option to view the reactions for the selected supports only.

Nodal Spring Reactions

Analysis Results > Nodal Spring Reactions displays the following dialog.



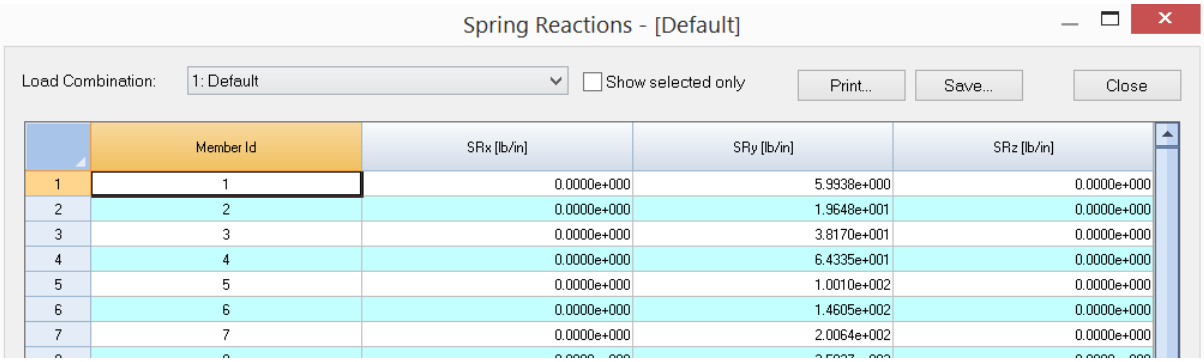
	Node Id	SRx [lb]	SRy [lb]	SRz [lb]	SRox [lb-in]	SRoy [lb-in]	SRoz [lb-in]
1	1	0.000e+000	1.000e+001	0.000e+000	0.000e+000	0.000e+000	0.000e+000
2	2	0.000e+000	-1.000e+001	0.000e+000	0.000e+000	0.000e+000	0.000e+000
3	3	0.000e+000	0.000e+000	0.000e+000	0.000e+000	0.000e+000	0.000e+000

It allows you to view nodal spring reactions for each load combination. The reactions for each nodal spring include three force components SR_x, SR_y and SR_z and three moment components SR_{ox}, SR_{oy} and SR_{oz}. You have the option to view the reactions for the selected nodal springs only.

© 2026 ENERCALC, LLC

Line Spring Reactions

Analysis Results > Line Spring Reactions displays the following dialog.

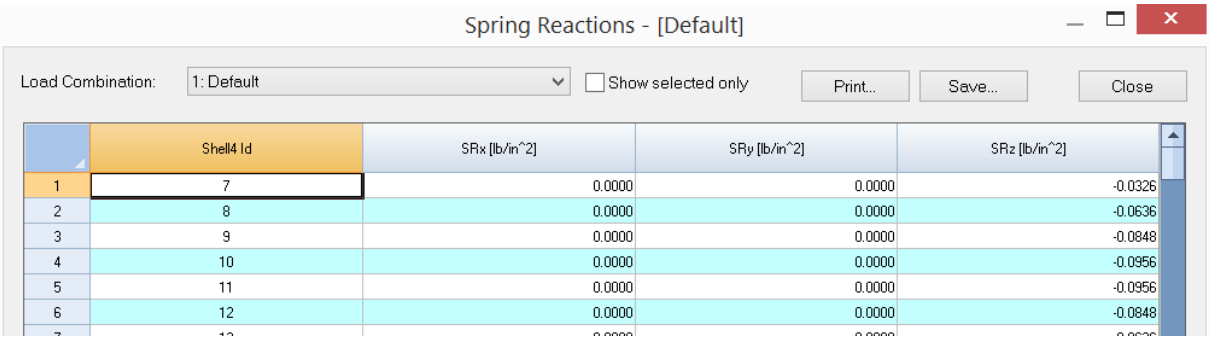


	Member Id	SRx [lb/in]	SRy [lb/in]	SRz [lb/in]
1	1	0.0000e+000	5.9938e+000	0.0000e+000
2	2	0.0000e+000	1.9648e+001	0.0000e+000
3	3	0.0000e+000	3.8170e+001	0.0000e+000
4	4	0.0000e+000	6.4335e+001	0.0000e+000
5	5	0.0000e+000	1.0010e+002	0.0000e+000
6	6	0.0000e+000	1.4605e+002	0.0000e+000
7	7	0.0000e+000	2.0064e+002	0.0000e+000

It allows you to view line spring reactions for each load combination. The reactions for each line spring include three force components SRx, SRy and SRz. You have the option to view the reactions for the selected line springs only.

Surface Spring Reactions

Analysis Results > Surface Spring Reactions displays the following dialog.



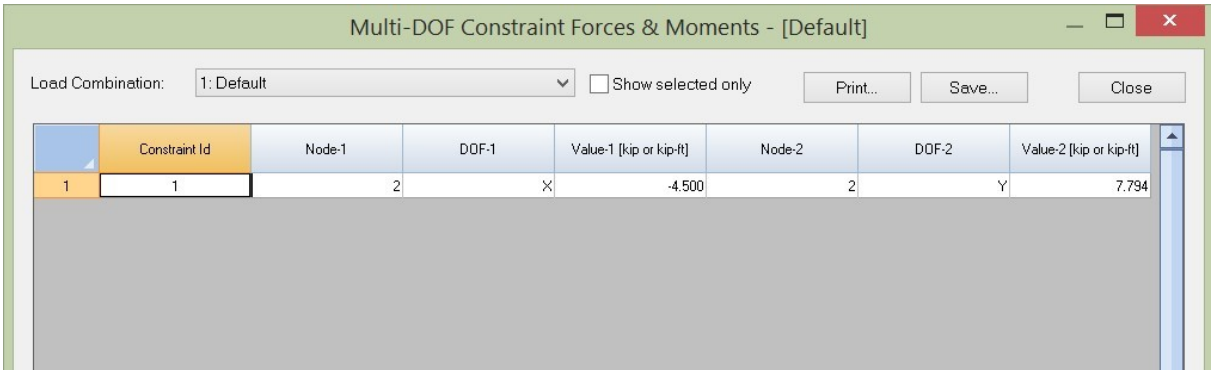
	Shell4 Id	SRx [lb/in^2]	SRy [lb/in^2]	SRz [lb/in^2]
1	7	0.0000	0.0000	-0.0326
2	8	0.0000	0.0000	-0.0636
3	9	0.0000	0.0000	-0.0848
4	10	0.0000	0.0000	-0.0956
5	11	0.0000	0.0000	-0.0956
6	12	0.0000	0.0000	-0.0848

It allows you to view surface spring reactions for each load combination. The reactions for each surface spring include three force components SR_x, SR_y and SR_z. You have the option to view the reactions for the selected surface springs only.

© 2026 ENERCALC, LLC

Multi-DOF Constraint Forces & Moments

Analysis Results > Multi-DOF Constraint Forces & Moments displays the following dialog.

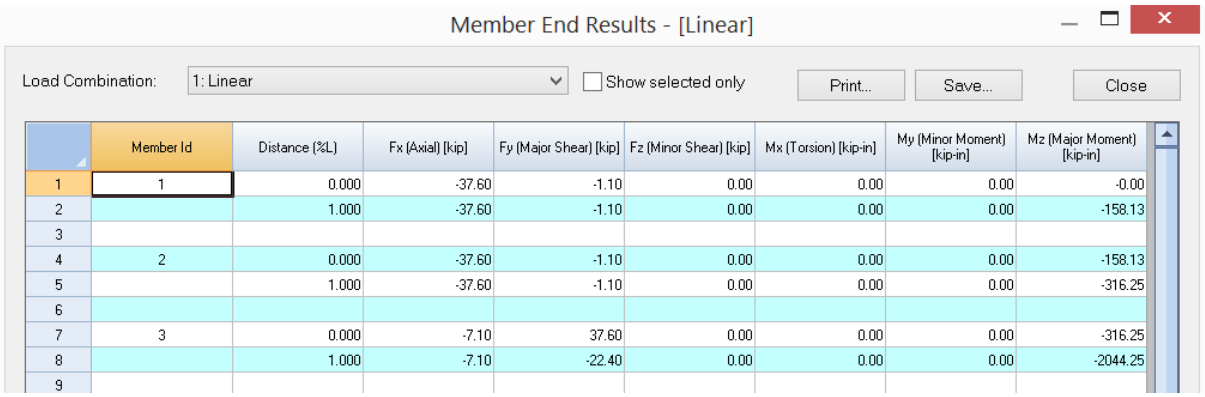


	Constraint Id	Node-1	DOF-1	Value-1 [kip or kip-ft]	Node-2	DOF-2	Value-2 [kip or kip-ft]
1	1	2	X	-4.500	2	Y	7.794

It allows you to view constraint forces and moments for each load combination. The value columns can be either constraint forces (for DOF X, Y and Z) or moments (for DOF OX, OY and OZ).

Member End Forces & Moments

Analysis Results > Member End Forces & Moments displays the following dialog.



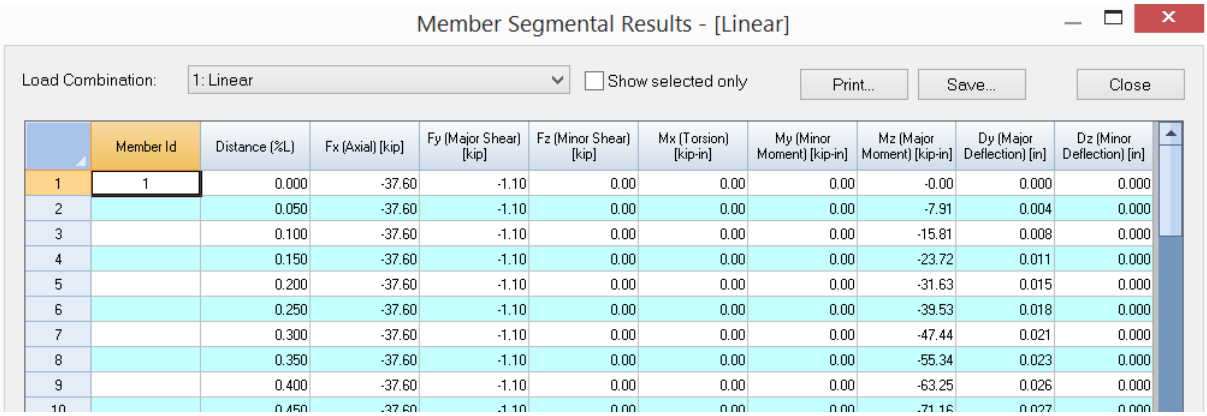
	Member Id	Distance (%L)	Fx (Axial) [kip]	Fy (Major Shear) [kip]	Fz (Minor Shear) [kip]	Mx (Torsion) [kip-in]	My (Minor Moment) [kip-in]	Mz (Major Moment) [kip-in]
1	1	0.000	-37.60	-1.10	0.00	0.00	0.00	-0.00
2		1.000	-37.60	-1.10	0.00	0.00	0.00	-158.13
3								
4	2	0.000	-37.60	-1.10	0.00	0.00	0.00	-158.13
5		1.000	-37.60	-1.10	0.00	0.00	0.00	-316.25
6								
7	3	0.000	-7.10	37.60	0.00	0.00	0.00	-316.25
8		1.000	-7.10	-22.40	0.00	0.00	0.00	-2044.25
9								

It allows you to view forces and moments at the ends of each member for each load combination. The end forces and moments include axial force F_x, major shear F_y, minor shear F_z, torsion M_x, minor moment M_y, and major moment M_z. You have the option to view the end forces and moments for the selected members only.

© 2026 ENERCALC, LLC

Member Segmental Results

Analysis Results > Member Segmental Results displays the following dialog.



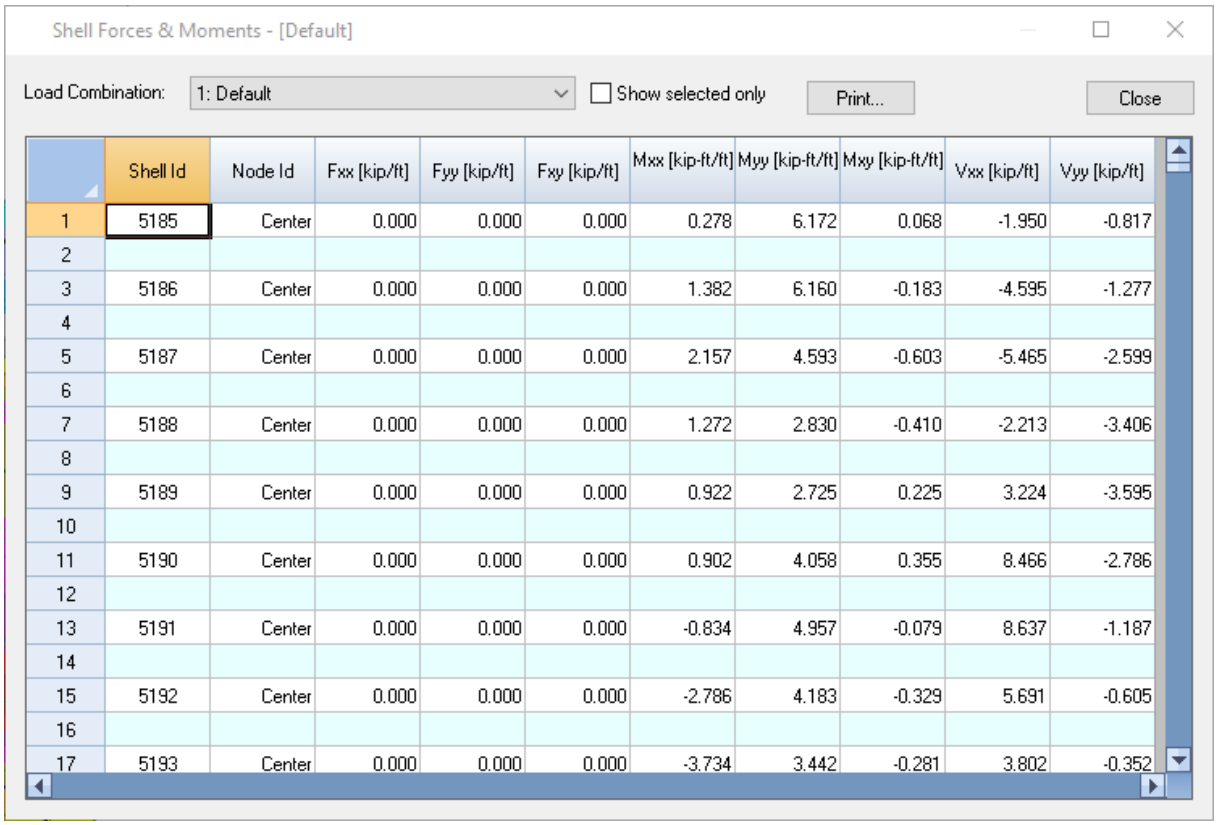
	Member Id	Distance (%L)	Fx (Axial) [kip]	Fy (Major Shear) [kip]	Fz (Minor Shear) [kip]	Mx (Torsion) [kip-in]	My (Minor Moment) [kip-in]	Mz (Major Moment) [kip-in]	Dy (Major Deflection) [in]	Dz (Minor Deflection) [in]
1	1	0.000	-37.60	-1.10	0.00	0.00	0.00	-0.00	0.000	0.000
2		0.050	-37.60	-1.10	0.00	0.00	0.00	-7.91	0.004	0.000
3		0.100	-37.60	-1.10	0.00	0.00	0.00	-15.81	0.008	0.000
4		0.150	-37.60	-1.10	0.00	0.00	0.00	-23.72	0.011	0.000
5		0.200	-37.60	-1.10	0.00	0.00	0.00	-31.63	0.015	0.000
6		0.250	-37.60	-1.10	0.00	0.00	0.00	-39.53	0.018	0.000
7		0.300	-37.60	-1.10	0.00	0.00	0.00	-47.44	0.021	0.000
8		0.350	-37.60	-1.10	0.00	0.00	0.00	-55.34	0.023	0.000
9		0.400	-37.60	-1.10	0.00	0.00	0.00	-63.25	0.026	0.000
10		0.450	-37.60	-1.10	0.00	0.00	0.00	-71.16	0.027	0.000

It allows you to view member segmental results for each load combination. The segmental results are shown at each segmental point designated by a distance along the member. They include axial force F_x, major shear F_y, minor shear F_z, torsion M_x, minor moment M_y, major moment M_z, major local deflection D_y, and minor local deflection D_z. You have the option to view the segmental forces and moments for the selected members only.

© 2026 ENERCALC, LLC

Shell Forces, Moments & Stresses

Analysis Results > Shell Forces, Moments & Stresses > Shell Forces & Moments displays the following dialog.

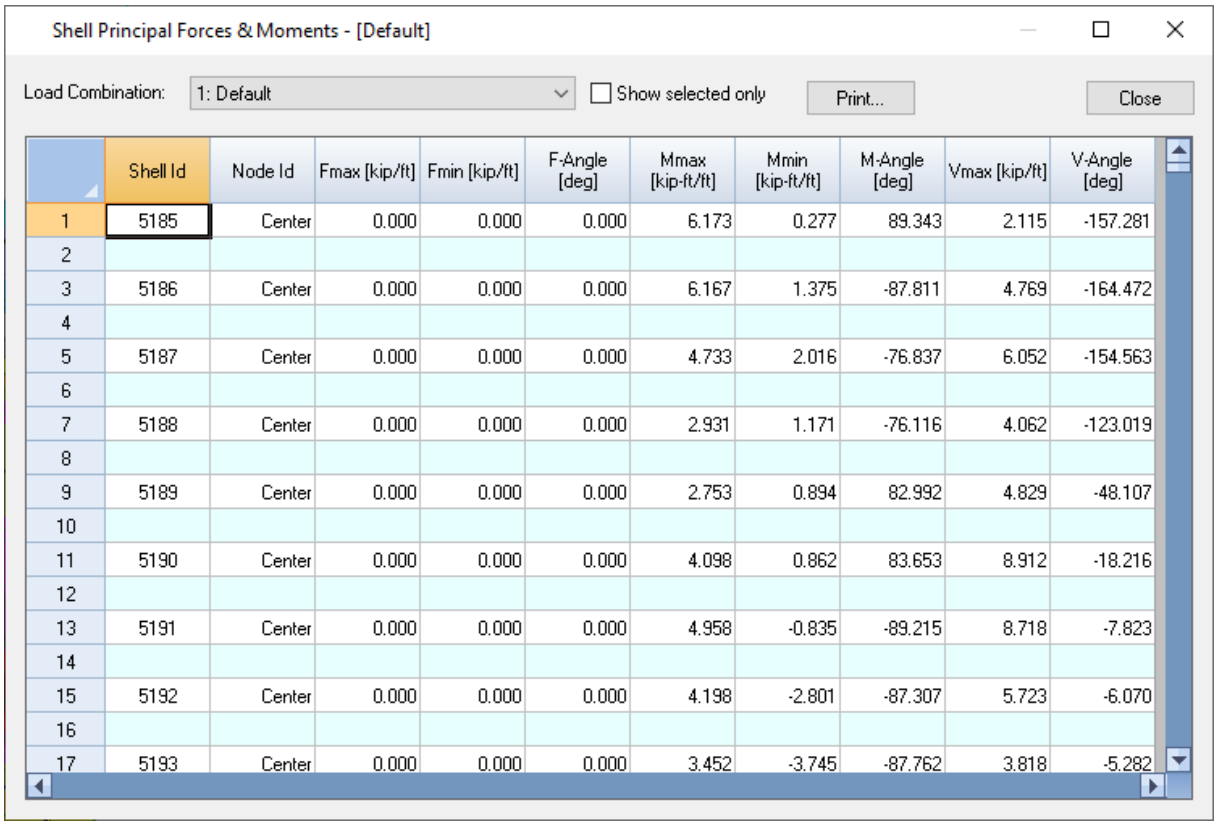


	Shell Id	Node Id	Fxx [kip/ft]	Fyy [kip/ft]	Fxy [kip/ft]	Mxx [kip-ft/ft]	Myy [kip-ft/ft]	Mxy [kip-ft/ft]	Vxx [kip/ft]	Vyy [kip/ft]
1	5185	Center	0.000	0.000	0.000	0.278	6.172	0.068	-1.950	-0.817
2										
3	5186	Center	0.000	0.000	0.000	1.382	6.160	-0.183	-4.595	-1.277
4										
5	5187	Center	0.000	0.000	0.000	2.157	4.593	-0.603	-5.465	-2.599
6										
7	5188	Center	0.000	0.000	0.000	1.272	2.830	-0.410	-2.213	-3.406
8										
9	5189	Center	0.000	0.000	0.000	0.922	2.725	0.225	3.224	-3.595
10										
11	5190	Center	0.000	0.000	0.000	0.902	4.058	0.355	8.466	-2.786
12										
13	5191	Center	0.000	0.000	0.000	-0.834	4.957	-0.079	8.637	-1.187
14										
15	5192	Center	0.000	0.000	0.000	-2.786	4.183	-0.329	5.691	-0.605
16										
17	5193	Center	0.000	0.000	0.000	-3.734	3.442	-0.281	3.802	-0.352

It allows you to view shell forces and moments for each load combination. The shell forces and moments include in-plane normal forces F_{xx}, F_{yy} and shear force F_{xy}; out-of-plane bending moments M_{xx}, M_{yy}, M_{xy}; and out-of-plane shear forces V_{xx}, V_{yy}. You may specify the force and moment locations to be at the nodes and/or the center of each shell by running Settings & Tools > Data Options. You have the option to view the forces and moments for the selected shells only.

© 2026 ENERCALC, LLC

Analysis Results > Shell Forces, Moments & Stresses > Shell Principal Forces & Moments displays the following dialog.



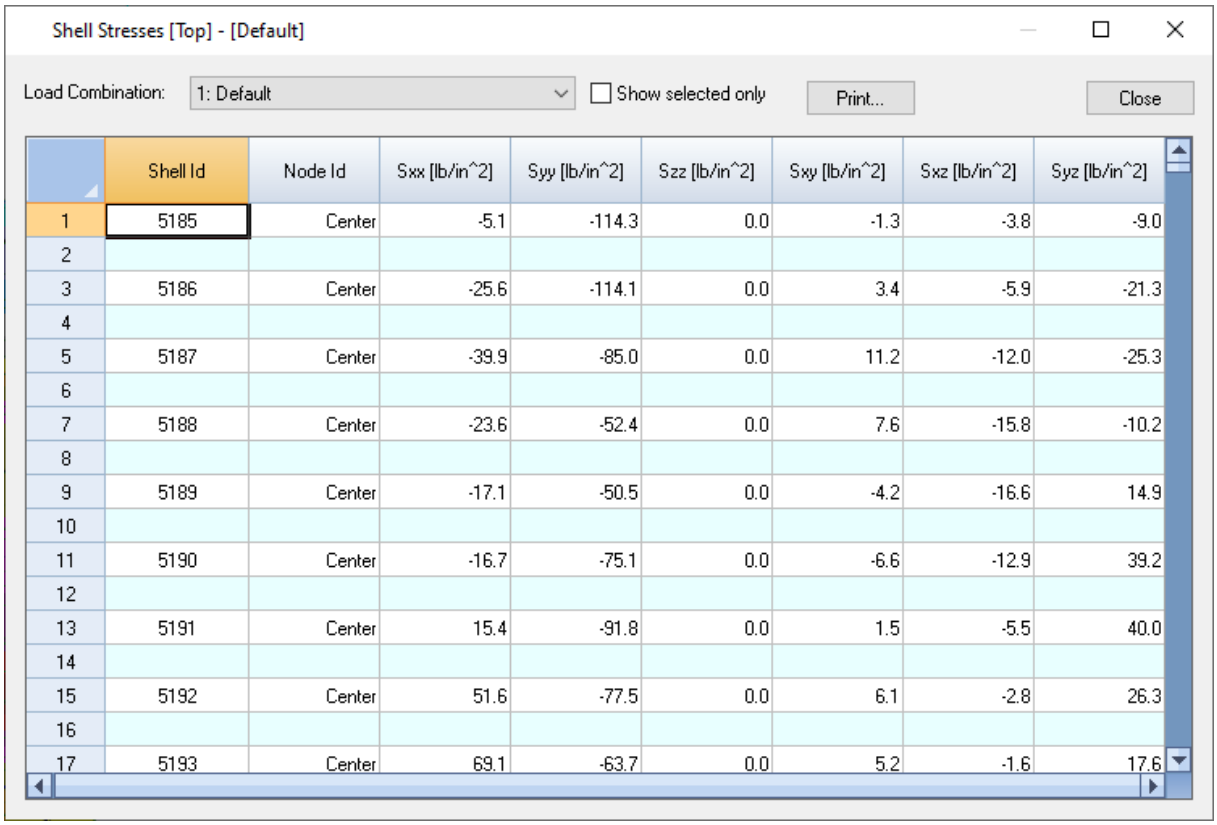
	Shell Id	Node Id	Fmax [kip/ft]	Fmin [kip/ft]	F-Angle [deg]	Mmax [kip-ft/ft]	Mmin [kip-ft/ft]	M-Angle [deg]	Vmax [kip/ft]	V-Angle [deg]
1	5185	Center	0.000	0.000	0.000	6.173	0.277	89.343	2.115	-157.281
2										
3	5186	Center	0.000	0.000	0.000	6.167	1.375	-87.811	4.769	-164.472
4										
5	5187	Center	0.000	0.000	0.000	4.733	2.016	-76.837	6.052	-154.563
6										
7	5188	Center	0.000	0.000	0.000	2.931	1.171	-76.116	4.062	-123.019
8										
9	5189	Center	0.000	0.000	0.000	2.753	0.894	82.992	4.829	-48.107
10										
11	5190	Center	0.000	0.000	0.000	4.098	0.862	83.653	8.912	-18.216
12										
13	5191	Center	0.000	0.000	0.000	4.958	-0.835	-89.215	8.718	-7.823
14										
15	5192	Center	0.000	0.000	0.000	4.198	-2.801	-87.307	5.723	-6.070
16										
17	5193	Center	0.000	0.000	0.000	3.452	-3.745	-87.762	3.818	-5.282

It allows you to view shell principal forces and moments for each load combination. The shell principal forces and moments include in-plane principal forces F_{max}, F_{min} and the principal angle F-angle; out-of-plane principal moments M_{max}, M_{min} and the principal angle M-angle; out-of-plane principal shear force V_{max} and the principal angle V-angle. You may specify the principal force and moment locations to be at the nodes and/or the center of each shell by running Settings & Tools > Data Options. You have the option to view the principal forces and moments for the selected shells only.

© 2026 ENERCALC, LLC

Shell Stresses [Top & Bottom]

Analysis Results > Shell Stresses [Top] or Shell Stresses [Bottom] displays the following dialog.



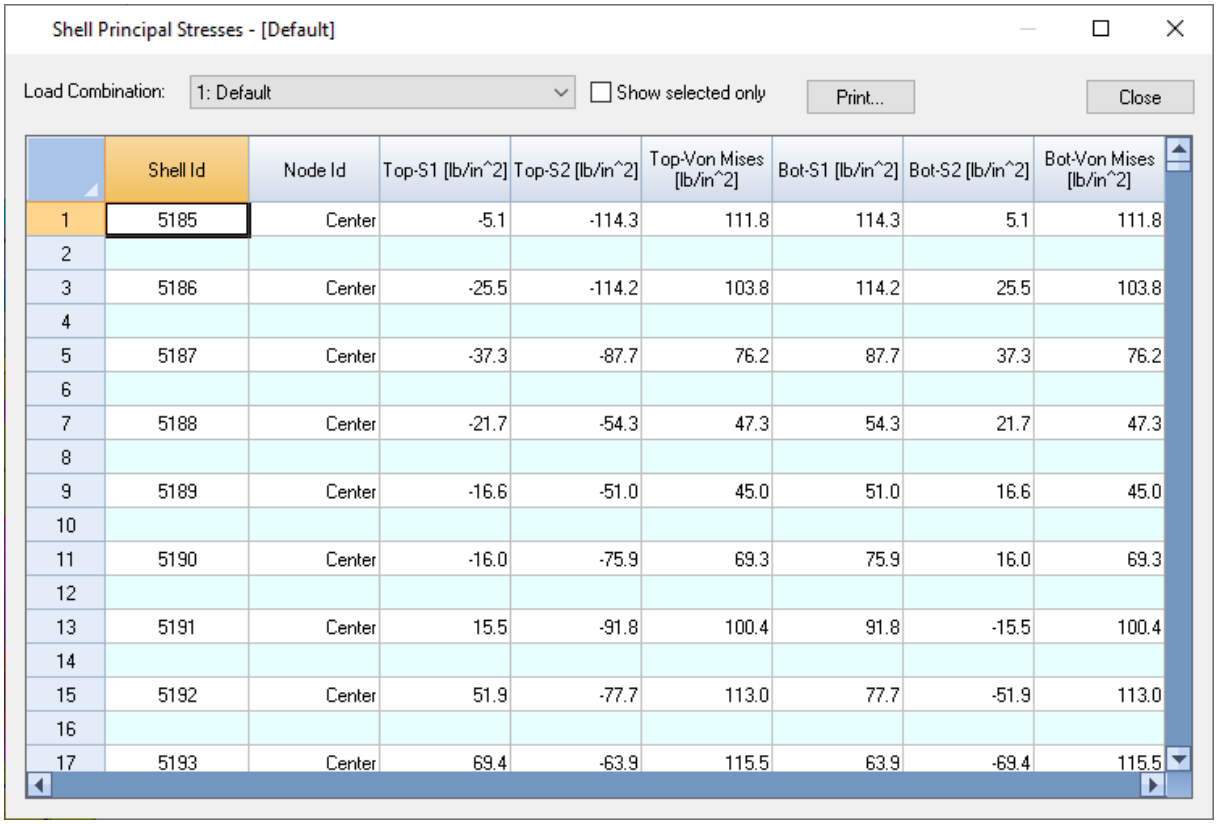
	Shell Id	Node Id	Sxx [lb/in^2]	Syy [lb/in^2]	Szz [lb/in^2]	Sxy [lb/in^2]	Sxz [lb/in^2]	Syz [lb/in^2]
1	5185	Center	-5.1	-114.3	0.0	-1.3	-3.8	-9.0
2								
3	5186	Center	-25.6	-114.1	0.0	3.4	-5.9	-21.3
4								
5	5187	Center	-39.9	-85.0	0.0	11.2	-12.0	-25.3
6								
7	5188	Center	-23.6	-52.4	0.0	7.6	-15.8	-10.2
8								
9	5189	Center	-17.1	-50.5	0.0	-4.2	-16.6	14.9
10								
11	5190	Center	-16.7	-75.1	0.0	-6.6	-12.9	39.2
12								
13	5191	Center	15.4	-91.8	0.0	1.5	-5.5	40.0
14								
15	5192	Center	51.6	-77.5	0.0	6.1	-2.8	26.3
16								
17	5193	Center	69.1	-63.7	0.0	5.2	-1.6	17.6

It allows you to view shell top or bottom stresses for each load combination. The shell stresses include three normal components S_{xx}, S_{yy}, S_{zz} and three shear components S_{xy}, S_{xz} and S_{yz}. You may specify the stress locations to be at the nodes and/or the center of each shell by running Settings > Data Options. You have the option to view the stresses for the selected shells only.

© 2026 ENERCALC, LLC

Shell Principal Stresses

Analysis Results > Shell Principal Stresses displays the following dialog.



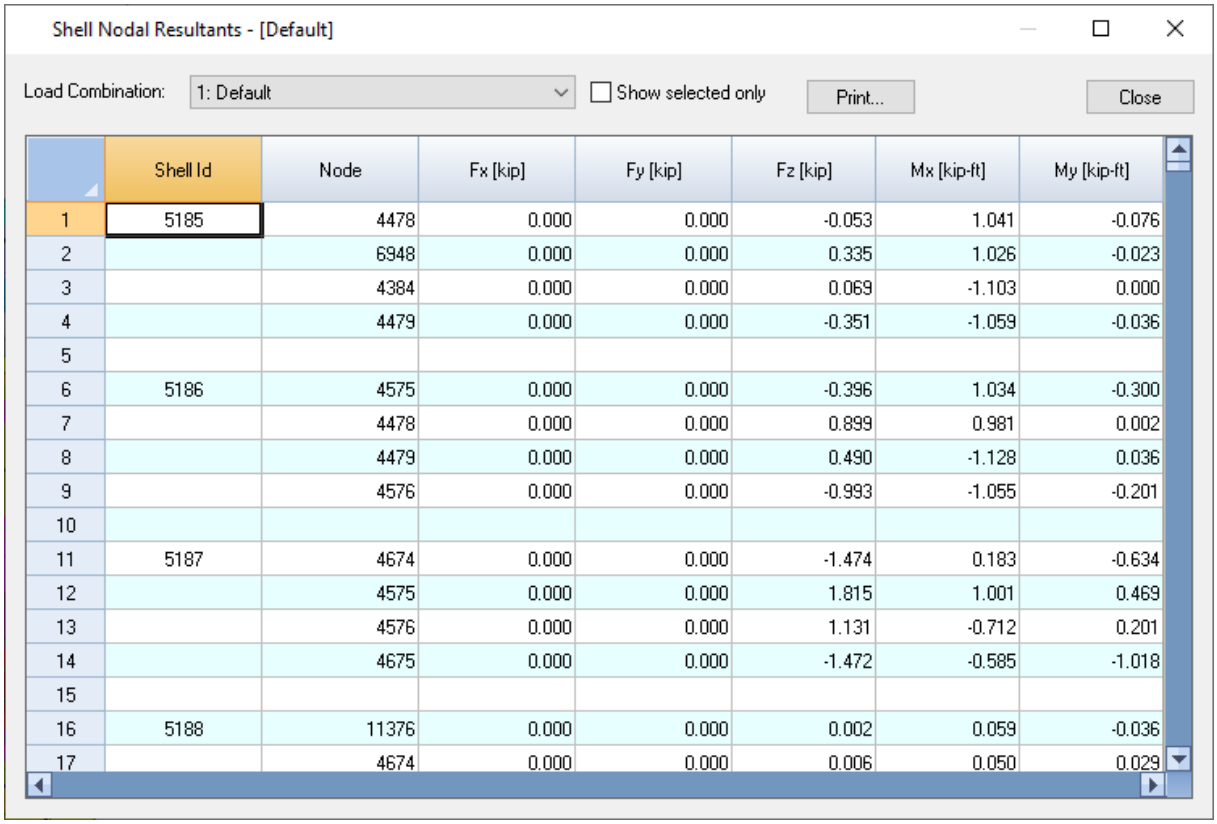
	Shell Id	Node Id	Top-S1 [lb/in^2]	Top-S2 [lb/in^2]	Top-Von Mises [lb/in^2]	Bot-S1 [lb/in^2]	Bot-S2 [lb/in^2]	Bot-Von Mises [lb/in^2]
1	5185	Center	-5.1	-114.3	111.8	114.3	5.1	111.8
2								
3	5186	Center	-25.5	-114.2	103.8	114.2	25.5	103.8
4								
5	5187	Center	-37.3	-87.7	76.2	87.7	37.3	76.2
6								
7	5188	Center	-21.7	-54.3	47.3	54.3	21.7	47.3
8								
9	5189	Center	-16.6	-51.0	45.0	51.0	16.6	45.0
10								
11	5190	Center	-16.0	-75.9	69.3	75.9	16.0	69.3
12								
13	5191	Center	15.5	-91.8	100.4	91.8	-15.5	100.4
14								
15	5192	Center	51.9	-77.7	113.0	77.7	-51.9	113.0
16								
17	5193	Center	69.4	-63.9	115.5	63.9	-69.4	115.5

It allows you to view shell top or bottom principal stresses and Von Mises stresses for each load combination. You may specify the stress locations to be at the nodes and/or the center of each shell by running Settings > Data Options. You have the option to view the stresses for the selected shells only.

© 2026 ENERCALC, LLC

Shell Nodal Resultants

Analysis Results > Shell Nodal Resultants displays the following dialog.



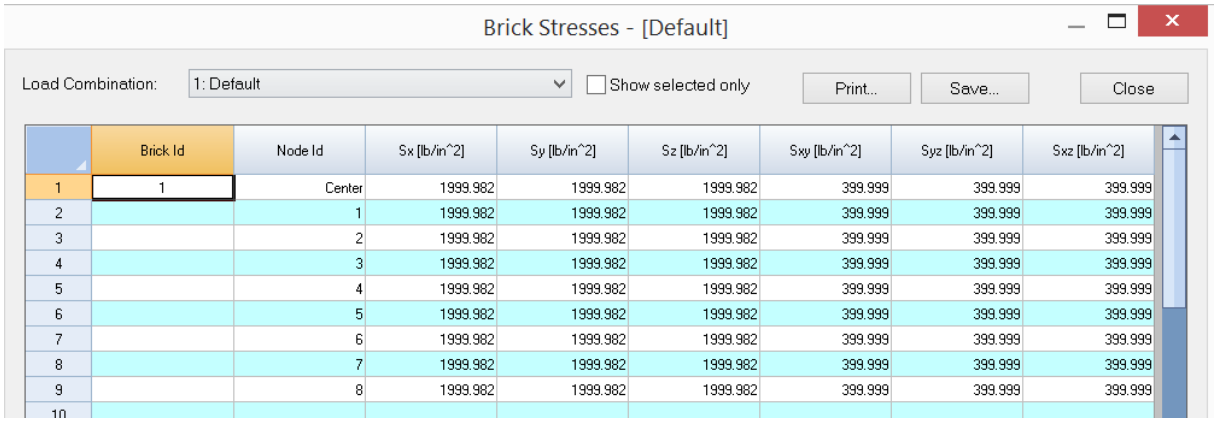
	Shell Id	Node	Fx [kip]	Fy [kip]	Fz [kip]	Mx [kip-ft]	My [kip-ft]
1	5185	4478	0.000	0.000	-0.053	1.041	-0.076
2		6948	0.000	0.000	0.335	1.026	-0.023
3		4384	0.000	0.000	0.069	-1.103	0.000
4		4479	0.000	0.000	-0.351	-1.059	-0.036
5							
6	5186	4575	0.000	0.000	-0.396	1.034	-0.300
7		4478	0.000	0.000	0.899	0.981	0.002
8		4479	0.000	0.000	0.490	-1.128	0.036
9		4576	0.000	0.000	-0.993	-1.055	-0.201
10							
11	5187	4674	0.000	0.000	-1.474	0.183	-0.634
12		4575	0.000	0.000	1.815	1.001	0.469
13		4576	0.000	0.000	1.131	-0.712	0.201
14		4675	0.000	0.000	-1.472	-0.585	-1.018
15							
16	5188	11376	0.000	0.000	0.002	0.059	-0.036
17		4674	0.000	0.000	0.006	0.050	0.029

It allows you to view shell nodal resultants for each load combination. The nodal resultants are concentrated forces and moments at element nodes that keep individual elements in equilibrium. They are expressed in the local coordinate system. You have the option to view the nodal resultants for the selected shells only.

© 2026 ENERCALC, LLC

Bricks

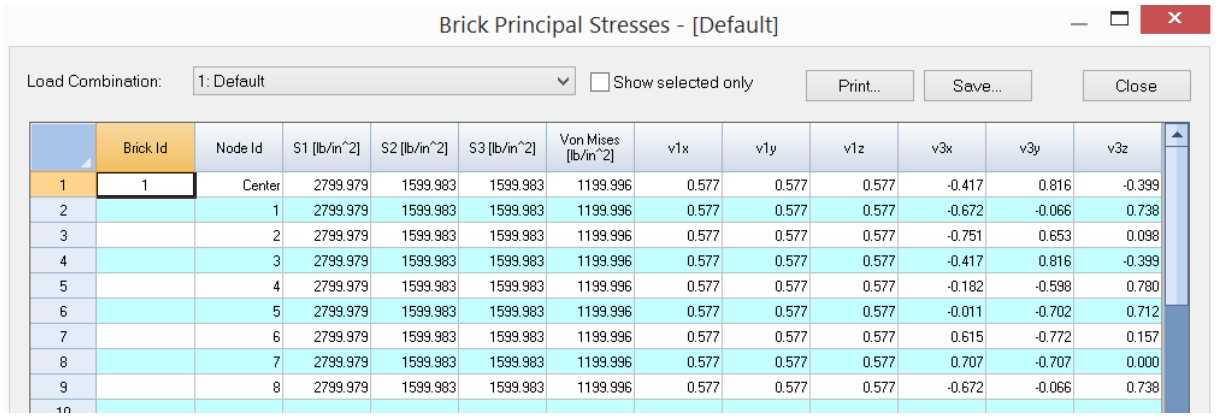
Analysis Results > Bricks > Brick Stresses displays the following dialog.



	Brick Id	Node Id	Sx [lb/in^2]	Sy [lb/in^2]	Sz [lb/in^2]	Sxy [lb/in^2]	Syz [lb/in^2]	Sxz [lb/in^2]
1	1	Center	1999.982	1999.982	1999.982	399.999	399.999	399.999
2		1	1999.982	1999.982	1999.982	399.999	399.999	399.999
3		2	1999.982	1999.982	1999.982	399.999	399.999	399.999
4		3	1999.982	1999.982	1999.982	399.999	399.999	399.999
5		4	1999.982	1999.982	1999.982	399.999	399.999	399.999
6		5	1999.982	1999.982	1999.982	399.999	399.999	399.999
7		6	1999.982	1999.982	1999.982	399.999	399.999	399.999
8		7	1999.982	1999.982	1999.982	399.999	399.999	399.999
9		8	1999.982	1999.982	1999.982	399.999	399.999	399.999
10								

It allows you to view brick stresses for each load combination. The brick stresses include three normal components S_{xx}, S_{yy}, S_{zz} and three shear components S_{xy}, S_{yz} and S_{xz}. You may specify the stress locations to be at the nodes and/or the center of each brick by running Settings > Data Options. You have the option to view the stresses for the selected bricks only.

Analysis Results > Bricks > Brick Principal Stresses displays the following dialog.



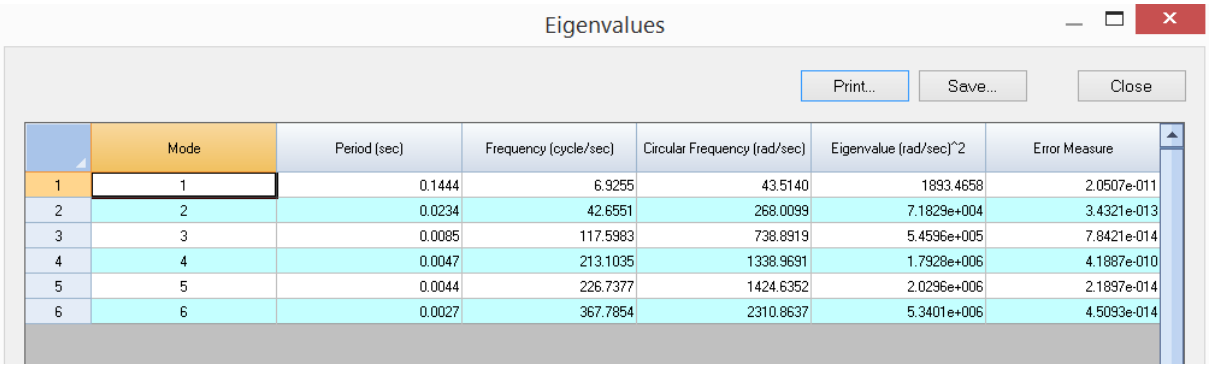
	Brick Id	Node Id	S1 [lb/in^2]	S2 [lb/in^2]	S3 [lb/in^2]	Von Mises [lb/in^2]	v1x	v1y	v1z	v3x	v3y	v3z
1	1	Center	2799.979	1599.983	1599.983	1199.996	0.577	0.577	0.577	-0.417	0.816	-0.399
2		1	2799.979	1599.983	1599.983	1199.996	0.577	0.577	0.577	-0.672	-0.066	0.738
3		2	2799.979	1599.983	1599.983	1199.996	0.577	0.577	0.577	-0.751	0.653	0.098
4		3	2799.979	1599.983	1599.983	1199.996	0.577	0.577	0.577	-0.417	0.816	-0.399
5		4	2799.979	1599.983	1599.983	1199.996	0.577	0.577	0.577	-0.182	-0.598	0.780
6		5	2799.979	1599.983	1599.983	1199.996	0.577	0.577	0.577	-0.011	-0.702	0.712
7		6	2799.979	1599.983	1599.983	1199.996	0.577	0.577	0.577	0.615	-0.772	0.157
8		7	2799.979	1599.983	1599.983	1199.996	0.577	0.577	0.577	0.707	-0.707	0.000
9		8	2799.979	1599.983	1599.983	1199.996	0.577	0.577	0.577	-0.672	-0.066	0.738
10												

It allows you to view brick principal stresses and Von Mises stresses for each load combination. Brick principal stresses are shown at locations of nodes and/or center of each shell. You may control the locations by running Settings > Data Options. Brick principal stresses include three principal components S_1, S_2, S_3 and direction vectors (V_{1x}, V_{1y} and V_{1z}) and (V_{3x}, V_{3y} and V_{3z}). You have the option to view the principal stresses for selected bricks only.

Dynamics

© 2026 ENERCALC, LLC

Analysis Results > Dynamics > Eigenvalues displays the following dialog.



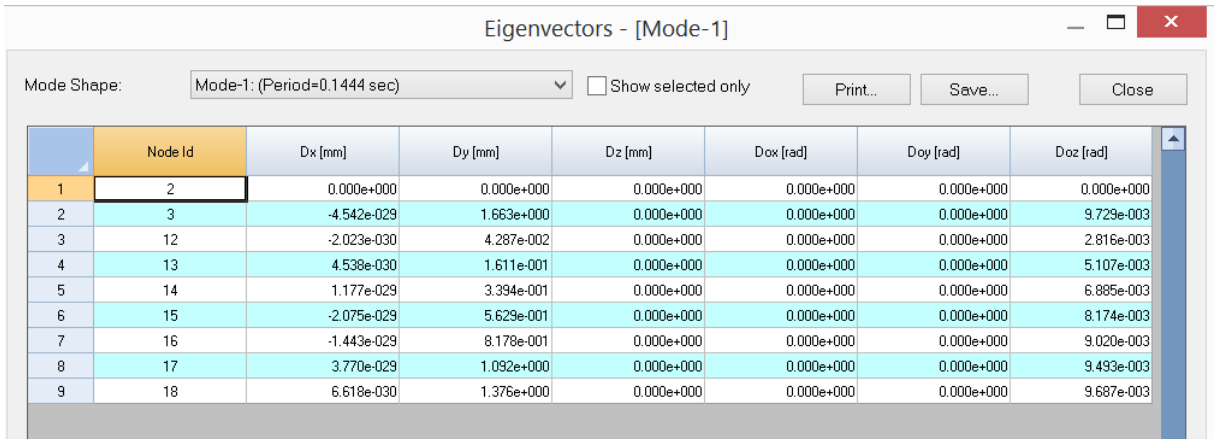
	Mode	Period (sec)	Frequency (cycle/sec)	Circular Frequency (rad/sec)	Eigenvalue (rad/sec)^2	Error Measure
1	1	0.1444	6.9255	43.5140	1893.4658	2.0507e-011
2	2	0.0234	42.6551	268.0099	7.1829e+004	3.4321e-013
3	3	0.0085	117.5983	738.8919	5.4596e+005	7.8421e-014
4	4	0.0047	213.1035	1338.9691	1.7928e+006	4.1887e-010
5	5	0.0044	226.7377	1424.6352	2.0296e+006	2.1897e-014
6	6	0.0027	367.7854	2310.8637	5.3401e+006	4.5093e-014

It allows you to view eigenvalues (λ) and their derivatives such as periods (T), frequencies (f), and circular frequencies (ω). In addition, an error measure is calculated for each eigenvalue according to the following (see Ref. 1):

$$\text{Error Measure} = \sqrt{1 - \frac{(\lambda_i^{(k)})^2}{(q_i^{(k)})^T (q_i^{(k)})}}$$

Where $q_i^{(k)}$ is the vector in the matrix $Q^{(k)}$ corresponding to $\lambda_i^{(k)}$.

Analysis Results > Dynamics > Eigenvectors displays the following dialog.

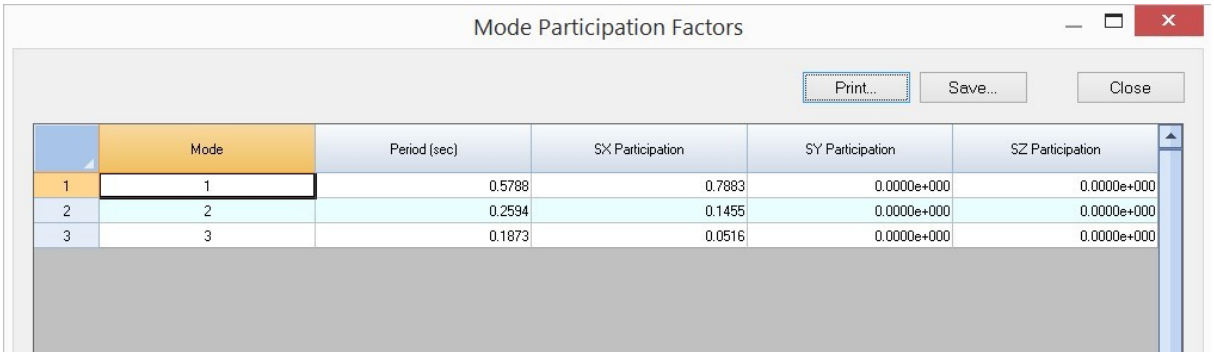


	Node Id	Dx [mm]	Dy [mm]	Dz [mm]	Dox [rad]	Doy [rad]	Doz [rad]
1	2	0.000e+000	0.000e+000	0.000e+000	0.000e+000	0.000e+000	0.000e+000
2	3	-4.542e-029	1.663e+000	0.000e+000	0.000e+000	0.000e+000	9.729e-003
3	12	-2.023e-030	4.287e-002	0.000e+000	0.000e+000	0.000e+000	2.816e-003
4	13	4.538e-030	1.611e-001	0.000e+000	0.000e+000	0.000e+000	5.107e-003
5	14	1.177e-029	3.394e-001	0.000e+000	0.000e+000	0.000e+000	6.885e-003
6	15	-2.075e-029	5.629e-001	0.000e+000	0.000e+000	0.000e+000	8.174e-003
7	16	-1.443e-029	8.178e-001	0.000e+000	0.000e+000	0.000e+000	9.020e-003
8	17	3.770e-029	1.092e+000	0.000e+000	0.000e+000	0.000e+000	9.493e-003
9	18	6.618e-030	1.376e+000	0.000e+000	0.000e+000	0.000e+000	9.687e-003

It allows you to view eigenvectors (mode shapes) for each mode of vibration. It is worthwhile to point out that eigenvectors are meaningful only in their relative values.

© 2026 ENERCALC, LLC

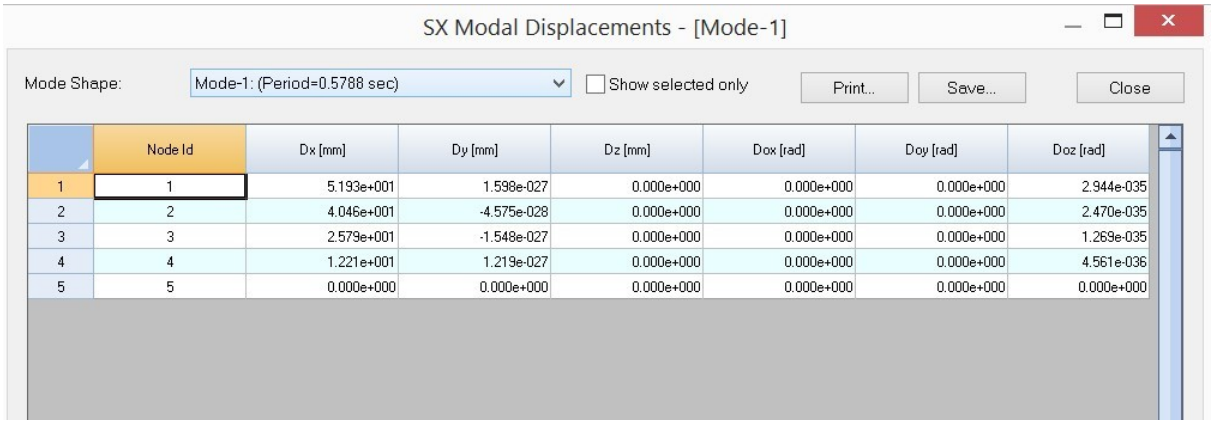
Analysis Results > Dynamics > Mode Participation Factors displays the following dialog.



	Mode	Period (sec)	SX Participation	SY Participation	SZ Participation
1	1	0.5788	0.7883	0.0000e+000	0.0000e+000
2	2	0.2594	0.1455	0.0000e+000	0.0000e+000
3	3	0.1873	0.0516	0.0000e+000	0.0000e+000

It allows you to view mode participation factors for each mode in global X, Y and Z directions. You must perform response spectrum analysis beforehand.

Analysis Results > Dynamics > Modal Displacements SX, SY and SZ displays the following dialog.

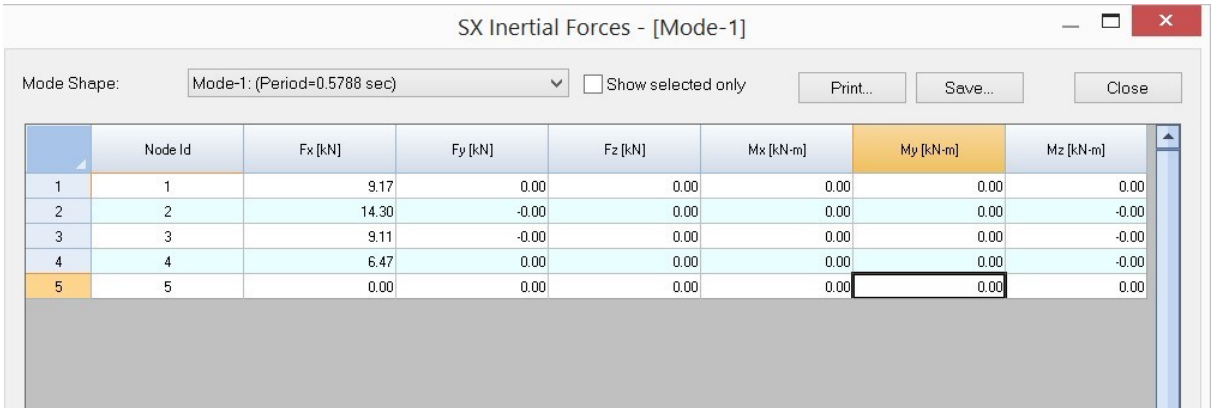


	Node Id	Dx [mm]	Dy [mm]	Dz [mm]	Dox [rad]	Doy [rad]	Doz [rad]
1	1	5.193e+001	1.598e-027	0.000e+000	0.000e+000	0.000e+000	2.944e-035
2	2	4.046e+001	-4.575e-028	0.000e+000	0.000e+000	0.000e+000	2.470e-035
3	3	2.579e+001	-1.548e-027	0.000e+000	0.000e+000	0.000e+000	1.269e-035
4	4	1.221e+001	1.219e-027	0.000e+000	0.000e+000	0.000e+000	4.561e-036
5	5	0.000e+000	0.000e+000	0.000e+000	0.000e+000	0.000e+000	0.000e+000

It allows you to view modal displacements for each mode as well as their SRSS combination in global X, Y and Z directions. You must perform response spectrum analysis beforehand.

© 2026 ENERCALC, LLC

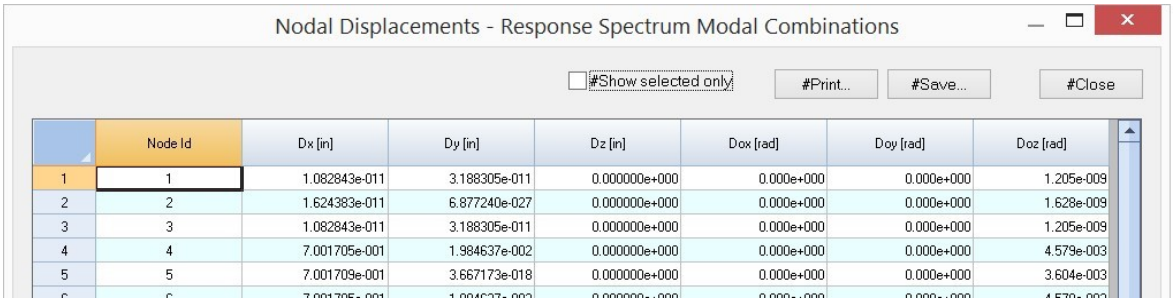
Analysis Results > Dynamics > Inertial Forces SX, SY and SZ displays the following dialog.



	Node Id	Fx [kN]	Fy [kN]	Fz [kN]	Mx [kN-m]	My [kN-m]	Mz [kN-m]
1	1	9.17	0.00	0.00	0.00	0.00	0.00
2	2	14.30	-0.00	0.00	0.00	0.00	-0.00
3	3	9.11	-0.00	0.00	0.00	0.00	-0.00
4	4	6.47	0.00	0.00	0.00	0.00	-0.00
5	5	0.00	0.00	0.00	0.00	0.00	0.00

It allows you to view inertia forces for each mode in global X, Y and Z directions. You must perform response spectrum analysis beforehand. Inertia forces are converted to nodal loads in different inertia load cases automatically during response spectrum analysis.

Analysis Results > Dynamics > Modal Nodal Displacements displays the following dialog.

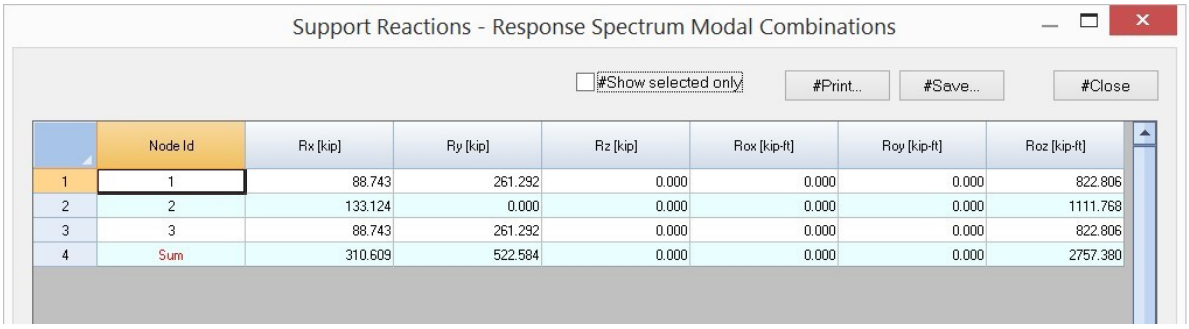


	Node Id	Dx [in]	Dy [in]	Dz [in]	Dox [rad]	Doy [rad]	Doz [rad]
1	1	1.082843e-011	3.188305e-011	0.000000e+000	0.000e+000	0.000e+000	1.205e-009
2	2	1.624383e-011	6.877240e-027	0.000000e+000	0.000e+000	0.000e+000	1.628e-009
3	3	1.082843e-011	3.188305e-011	0.000000e+000	0.000e+000	0.000e+000	1.205e-009
4	4	7.001705e-001	1.984637e-002	0.000000e+000	0.000e+000	0.000e+000	4.579e-003
5	5	7.001709e-001	3.667173e-018	0.000000e+000	0.000e+000	0.000e+000	3.604e-003

It allows you to view nodal displacements in modal combinations (including directional combinations). You must perform response spectrum analysis beforehand.

© 2026 ENERCALC, LLC

Analysis Results > Dynamics > Modal Support Reactions displays the following dialog.



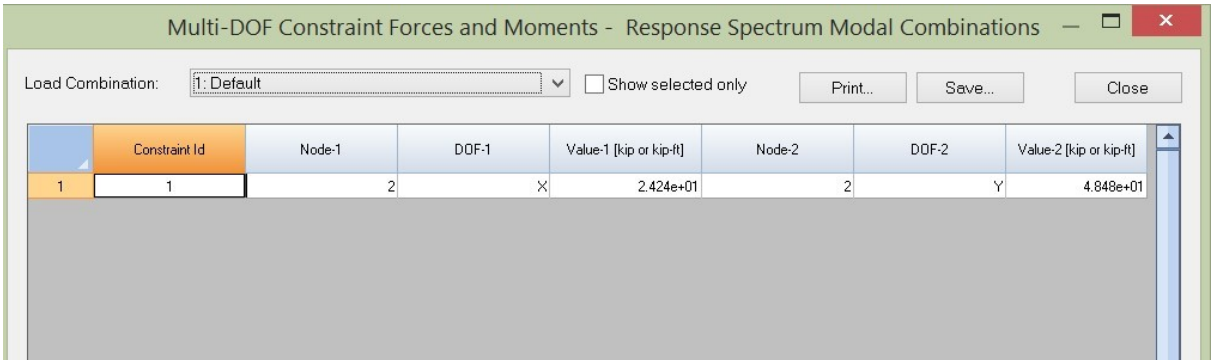
	Node Id	Rx [kip]	Ry [kip]	Rz [kip]	Rox [kip-ft]	Roy [kip-ft]	Roz [kip-ft]
1	1	88.743	261.292	0.000	0.000	0.000	822.806
2	2	133.124	0.000	0.000	0.000	0.000	1111.768
3	3	88.743	261.292	0.000	0.000	0.000	822.806
4	Sum	310.609	522.584	0.000	0.000	0.000	2757.380

It allows you to view support reactions in modal combinations (including directional combinations). You must perform response spectrum analysis beforehand.

Note: The support reactions do not include multi-DOF constraint forces and moments.

Analysis Results > Dynamics > Modal Nodal, Line, Surface Spring Reactions open dialogs similar to Modal Support Reactions. You must perform response spectrum analysis beforehand.

Analysis Results > Dynamics > Modal Multi-DOF Constraint Forces & Moments displays the following dialog.

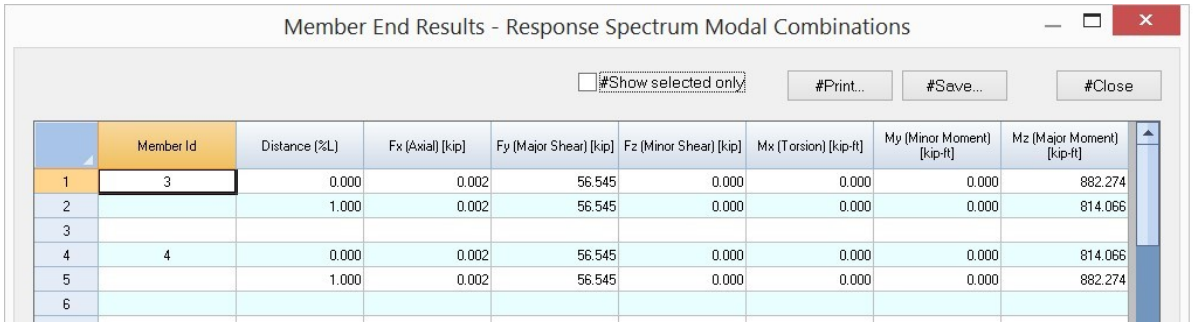


	Constraint Id	Node-1	DOF-1	Value-1 [kip or kip-ft]	Node-2	DOF-2	Value-2 [kip or kip-ft]
1	1	2	X	2.424e+01	2	Y	4.848e+01

It allows you to view multi-DOF constraint forces & moments in modal combinations (including directional combinations). You must perform response spectrum analysis beforehand.

© 2026 ENERCALC, LLC

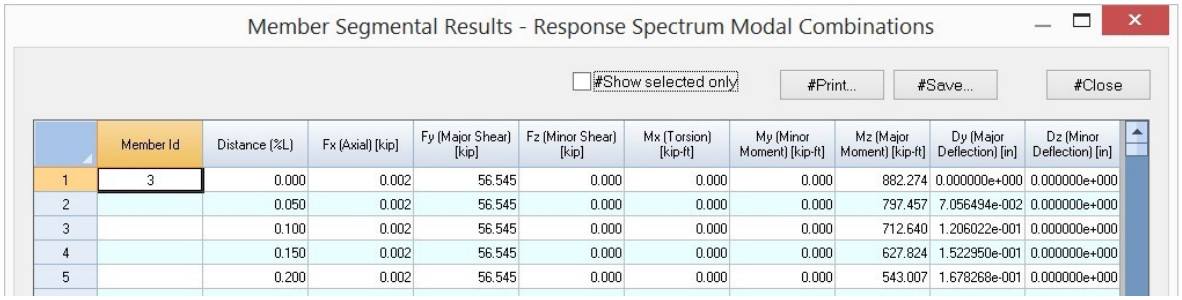
Analysis Results > Dynamics > Modal Member End Forces & Moments displays the following dialog.



	Member Id	Distance (%L)	Fx (Axial) [kip]	Fy (Major Shear) [kip]	Fz (Minor Shear) [kip]	Mx (Torsion) [kip-ft]	My (Minor Moment) [kip-ft]	Mz (Major Moment) [kip-ft]
1	3	0.000	0.002	56.545	0.000	0.000	0.000	882.274
2		1.000	0.002	56.545	0.000	0.000	0.000	814.066
3								
4	4	0.000	0.002	56.545	0.000	0.000	0.000	814.066
5		1.000	0.002	56.545	0.000	0.000	0.000	882.274
6								

It allows you to view member end forces and moments in modal combinations (including directional combinations). You must perform response spectrum analysis beforehand.

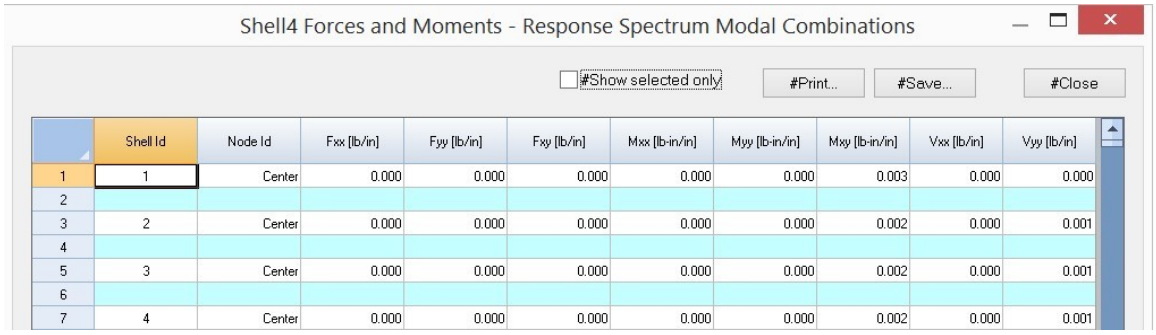
Analysis Results > Dynamics > Modal Member Segmental Results displays the following dialog.



	Member Id	Distance (%L)	Fx (Axial) [kip]	Fy (Major Shear) [kip]	Fz (Minor Shear) [kip]	Mx (Torsion) [kip-ft]	My (Minor Moment) [kip-ft]	Mz (Major Moment) [kip-ft]	Dy (Major Deflection) [in]	Dz (Minor Deflection) [in]
1	3	0.000	0.002	56.545	0.000	0.000	0.000	882.274	0.000000e+000	0.000000e+000
2		0.050	0.002	56.545	0.000	0.000	0.000	797.457	7.056494e-002	0.000000e+000
3		0.100	0.002	56.545	0.000	0.000	0.000	712.640	1.206022e-001	0.000000e+000
4		0.150	0.002	56.545	0.000	0.000	0.000	627.824	1.522950e-001	0.000000e+000
5		0.200	0.002	56.545	0.000	0.000	0.000	543.007	1.678268e-001	0.000000e+000

It allows you to view member segmental results in modal combinations (including directional combinations). You must perform response spectrum analysis beforehand.

Analysis Results > Dynamics > Modal Shell Forces & Moments displays the following dialog.

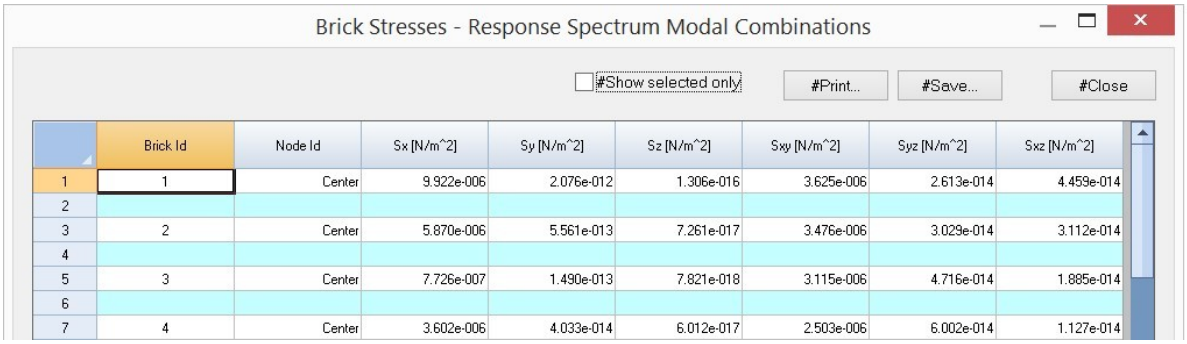


	Shell Id	Node Id	Fxx [lb/in]	Fyy [lb/in]	Fxy [lb/in]	Mxx [lb-in/in]	Myy [lb-in/in]	Mxy [lb-in/in]	Vxx [lb/in]	Vyy [lb/in]
1	1	Center	0.000	0.000	0.000	0.000	0.000	0.003	0.000	0.000
2										
3	2	Center	0.000	0.000	0.000	0.000	0.000	0.002	0.000	0.001
4										
5	3	Center	0.000	0.000	0.000	0.000	0.000	0.002	0.000	0.001
6										
7	4	Center	0.000	0.000	0.000	0.000	0.000	0.002	0.000	0.001

It allows you to view shell forces and moments in modal combinations (including directional combinations). You must perform response spectrum analysis beforehand.

© 2026 ENERCALC, LLC

Analysis Results > Dynamics > Modal Brick Stresses displays the following dialog.



	Brick Id	Node Id	Sx [N/m^2]	Sy [N/m^2]	Sz [N/m^2]	Sxy [N/m^2]	Syz [N/m^2]	Sxz [N/m^2]
1	1	Center	9.922e-006	2.076e-012	1.306e-016	3.625e-006	2.613e-014	4.459e-014
2								
3	2	Center	5.870e-006	5.561e-013	7.261e-017	3.476e-006	3.029e-014	3.112e-014
4								
5	3	Center	7.726e-007	1.490e-013	7.821e-018	3.115e-006	4.716e-014	1.885e-014
6								
7	4	Center	3.602e-006	4.033e-014	6.012e-017	2.503e-006	6.002e-014	1.127e-014

It allows you to view brick stresses in modal combinations (including directional combinations). You must perform response spectrum analysis beforehand.

Analysis Results > Dynamics > Modal Base Shears displays the following results window.

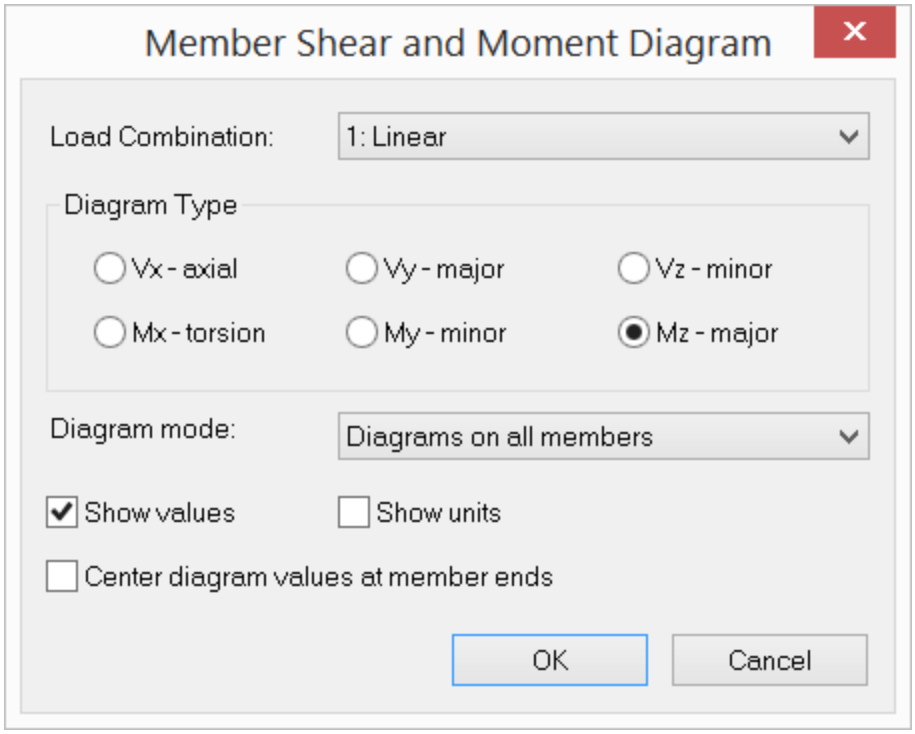
It allows you to view modal base shears in X and Z directions. You can compare them with the base shears computed by equivalent lateral force procedure using relevant code such as ASCE 7-10. You must perform response spectrum analysis beforehand.

Base Shears are not computed by the program if the model contains multi-DOF constraints.

© 2026 ENERCALC, LLC

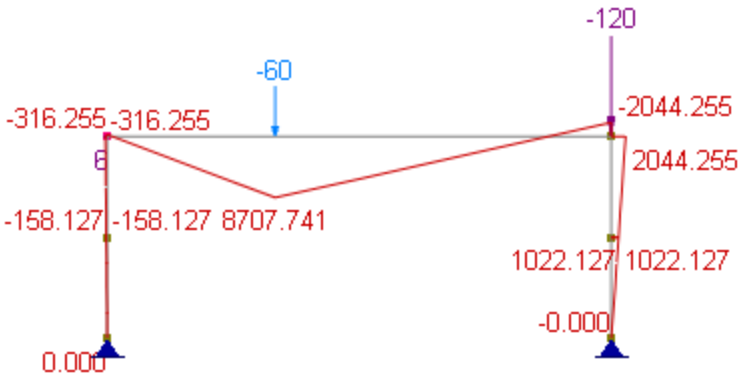
Shear & Moment Diagram

Analysis Results > Shear and Moment Diagram prompts you with the following dialog.



It allows you to view the member shear (including axial force) diagram or moment (including torsion) diagram for the selected load combination. Only one shear or moment diagram for the selected load combination may be displayed per window. However, you may display different shear or moment diagrams in multiple windows. To open a new window, click Settings & Tools > New Window.

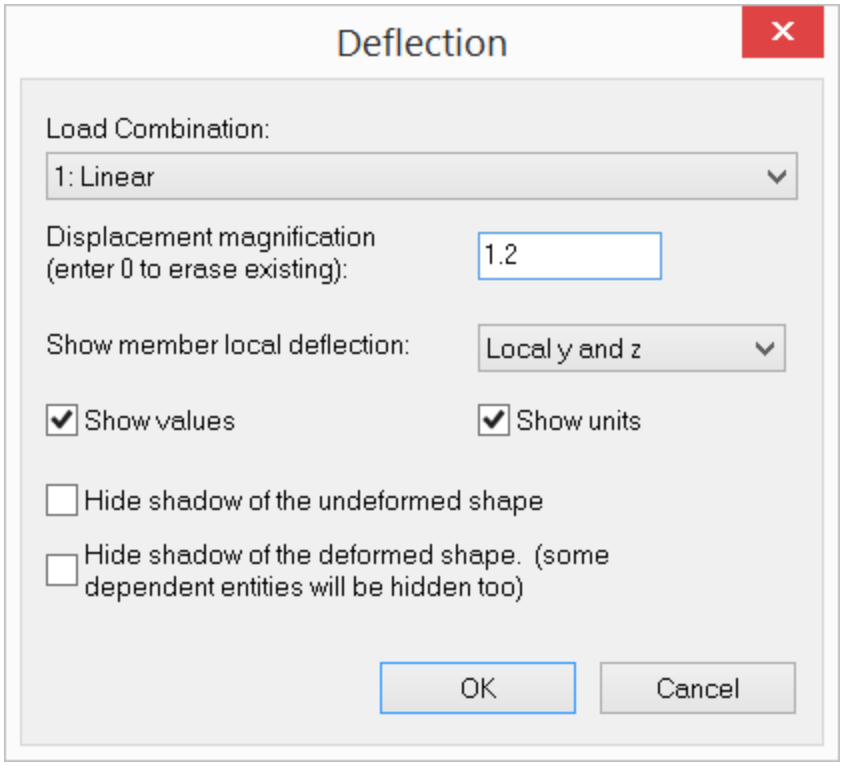
You have the option to show values and units for the diagram. You may show diagrams on all members or on selected members only. You may also erase existing diagrams. By default, no diagram is displayed even if an analysis has been performed successfully.

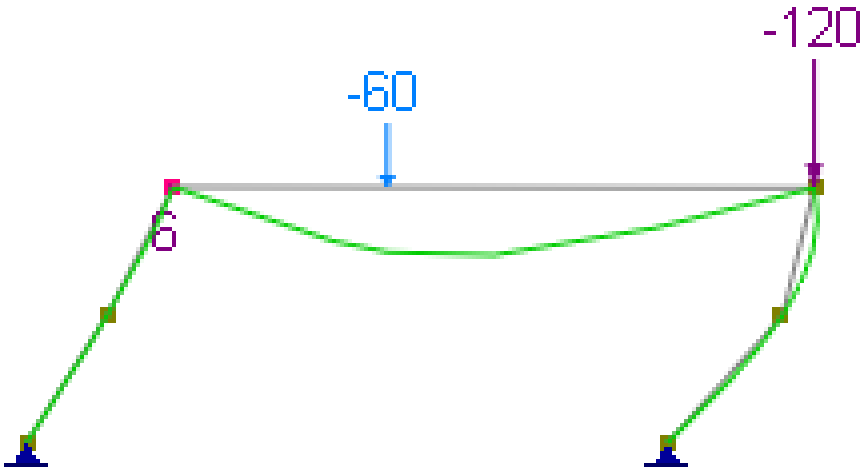


© 2026 ENERCALC, LLC

Deflection Diagram

Analysis Results > Deflection Diagram prompts you with the following dialog.





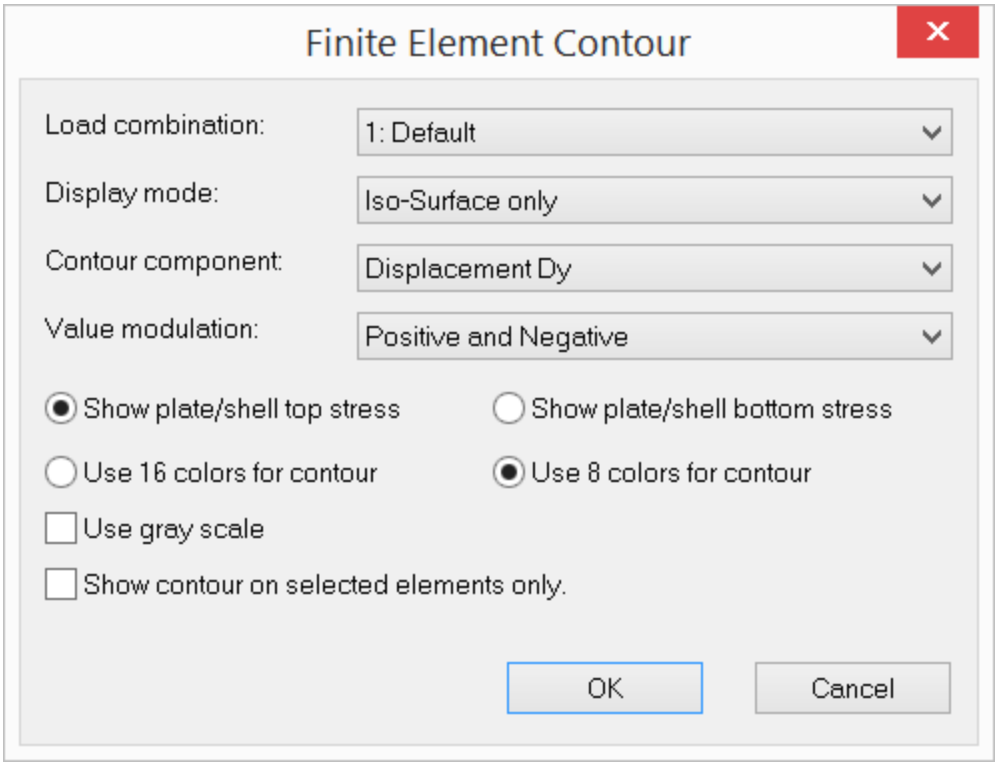
© 2026 ENERCALC, LLC

It allows you to view the deflected shape of the model for the selected load combination. The deflected shape is constructed by adding nodal displacements to nodal coordinates. For members, you have the option to show the local y and/or z deflection as well. You need to adjust the displacement magnification to view the deflection properly. The deflection displayed is for the selected load combination only. However, you may display deflections for different load combinations in multiple windows. To open a new window, click Window > New Window. The deflection values and units may be shown for the member local deflections. You may choose to have shadows of the undeformed shape and deformed shape hidden.

You cannot perform mouse selection while the deflected shape is shown. However, you may open another window with the undeformed shape and perform mouse selection as usual.

Contour Diagram

Analysis Results > Contour Diagram prompts you with the following dialog.



It allows you to view a result contour for shells and/or bricks for the selected load combination.

Four display modes are available. They are Iso-Surface and Value, Iso-Surface only, Value only, None or Erase. The Iso-Surface provides color bands for the contour component in different ranges. The number of ranges (or colors) may be either 16 or 8. Either top or bottom stresses may be specified for plate/shell elements. The Value (the absolute maximum) of the contour component may be shown for each element. The contour may be displayed in colors or gray scale. The latter is useful for people with color-impaired visions.

© 2026 ENERCALC, LLC

The contour components include:

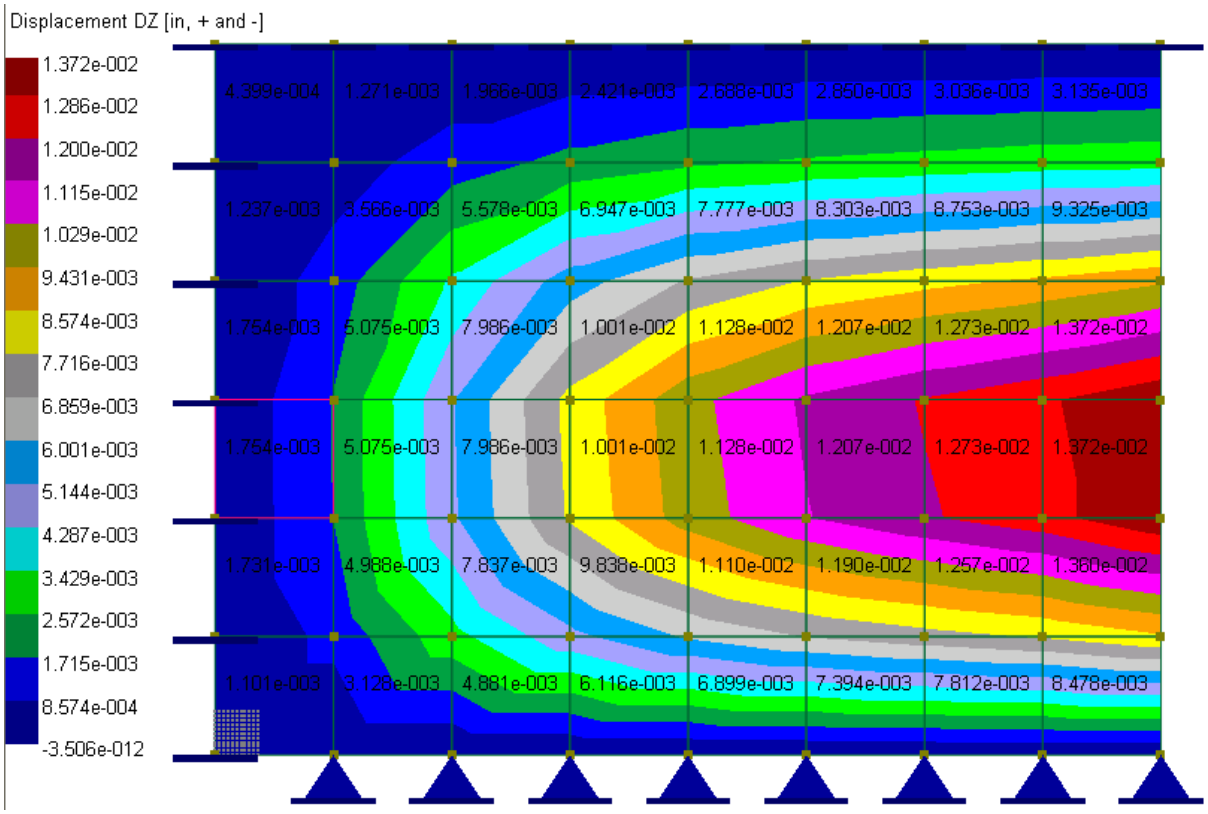
- nodal displacements (D_x, D_y, D_z, D_{ox}, D_{oy}, and D_{oz})
- shell bending moments (M_{xx}, M_{yy}, M_{xy}) and shears (V_{xx}, V_{yy})
- shell membrane normal forces (F_{xx}, F_{yy}) and in-plane shears (F_{xy})
- shell and brick stresses (S_{xx}, S_{yy}, S_{zz}, S_{xy}, S_{xz}, S_{yz})
- surface spring reactions (SR_x, SR_y, SR_z)
- shell principal moments (M_{max}, M_{min}) and shear (V_{max})
- shell principal membrane forces (F_{max}, F_{min})
- principal stresses (S_1, S_2, S_3) for shells and bricks
- Von Mises stresses for shells and bricks

Only one contour component for the selected load combination may be displayed per window. However, you may display different contour components in multiple windows. To open a new window, click Settings & Tools > New Window.

Four different modulations may be applied to values of the contour component. They are “Positive and Negative”, “Absolute”, “Positive Only”, “Negative Only”. For example, you may choose the contour component “Mxx” and the modulation “Negative Only” to view only the negative moments Mxx of the plates.

© 2026 ENERCALC, LLC

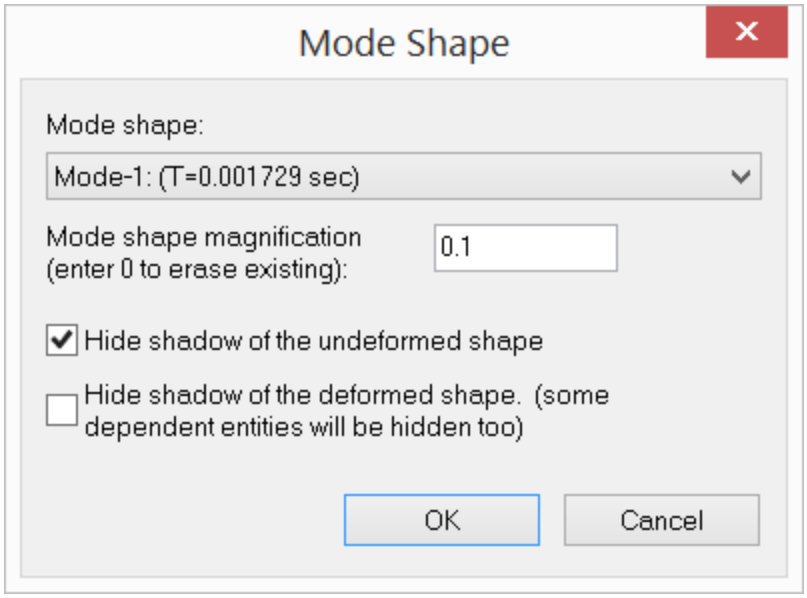
The following screen capture shows a displacement (D_Z) contour for a plate, with display mode “Iso-Surface and Values” and value modulation of “Positive and Negative”.



© 2026 ENERCALC, LLC

Mode Shape

Analysis Results > Mode Shape prompts you with the following dialog.



It allows you to view a mode shape after a frequency analysis is performed. You need to adjust the mode shape magnification to view the deflected shape properly. Mode shape diagrams are generated for a single mode; multiple mode shapes can be viewed by using multiple view windows. To open a new window, click Settings & Tools > New Window.

Mode Shape diagrams can be created with the 'shadow' of the undeformed shape, allowing for more concise presentation of the deflected shape. Like deformation diagrams from static analyses, the Mode Shape diagrams are 'locked' so that the model cannot be edited. To edit the model, either remove the diagram or open another window with the undeformed shape displayed.

Response Animation

Analysis Results > Response Animation toggles on or off structural responses (such as deflection, moment shear diagrams or stress contours) animation. Animation parameters may be set in Settings & Tools > Preferences.

© 2026 ENERCALC, LLC

Concrete Design

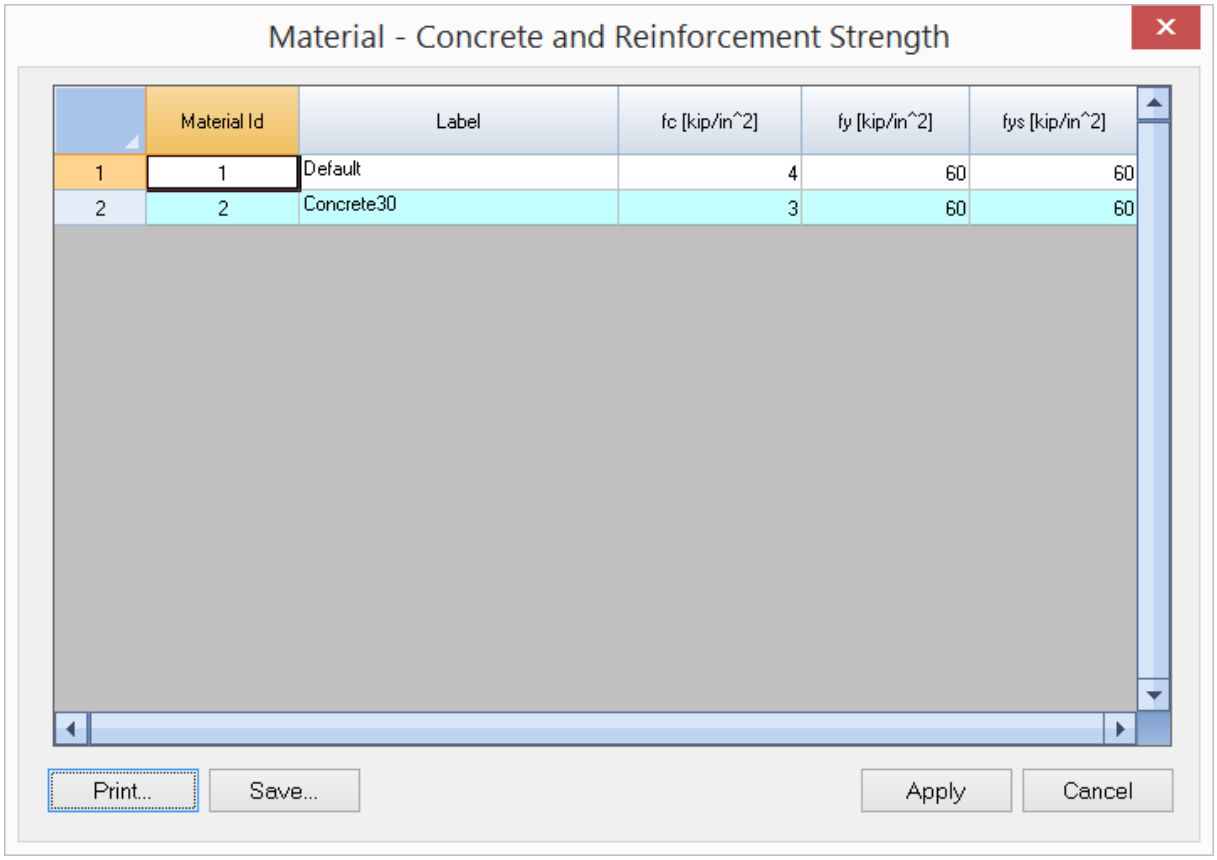
The Concrete Design ribbon provides commands related to concrete design for beams, columns, and slabs. This includes tools to input design parameters, execute the design process and review the design results in graphical and tabular form.

With the exception of the Cracking Factors command, the commands in the Concrete Design ribbon do not affect the analysis results. A static analysis must be done successfully before design can be performed.

© 2026 ENERCALC, LLC

RC Materials

Concrete Design > RC Materials prompts you with the following dialog.



It allows you to define concrete and reinforcement strength properties for the existing materials. The strength properties include:

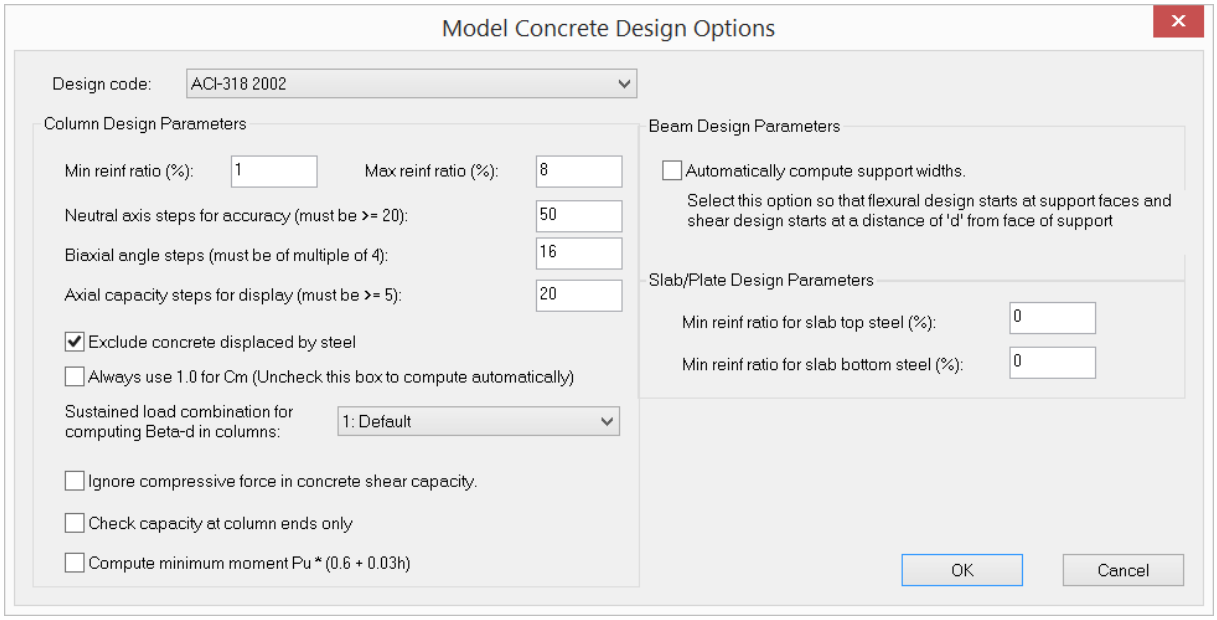
- Concrete compressive strength fc
- Concrete reinforcement strength fy
- Concrete stirrup or tie strength fys

If standard materials are used in Create > Materials, these strength properties will be set automatically. You may override these properties prior to performing concrete design. No concrete design will be performed on a member or shell if its f'c is zero. **You should not modify materials that are not concrete on this dialog.**

© 2026 ENERCALC, LLC

RC Model Design Criteria

Concrete Design > RC Design Criteria prompts you with the following dialog.



It allows you to enter global options for concrete design. You are encouraged to read the method of solution in the technical part of this document in order to understand these options. You should use this command before performing concrete design.

Design Code	Specifies design codes. Currently the program supports ACI 318-02/05/08/11/14 only.
Column min and max reinforcement ratios	Column minimum and maximum reinforcement ratios are used to generate column sections. They should be set between 1% and 8%. For all practical purposes, the maximum reinforcement ratio should be less than 4% to avoid rebar congestion.
Neutral axis steps for accuracy	Neutral axis steps affect the solution accuracy and speed. A value of 250 ~ 500 for neutral axis steps is sufficiently accurate for most sections. The adequacy of neutral axis steps can be determined by smoothness of the P-M_x and/or P-M_y interaction diagrams.
Biaxial angle steps	Biaxial angle steps affects the solution accuracy and speed. For biaxial problems, steps must be multiple of 4. A value of 16 or 32 is sufficiently accurate for most sections. The adequacy of biaxial angle steps can be determined by smoothness of the M_x-M_y interaction diagram. For uniaxial problems, biaxial angle steps should always be set to 4. This will give P-M_x (+) at 0 degree angle, P-M_x (-) at 180 degrees angle, P-M_y (+) at 90 degree angle, P-M_y (-) at 270 degrees angle.

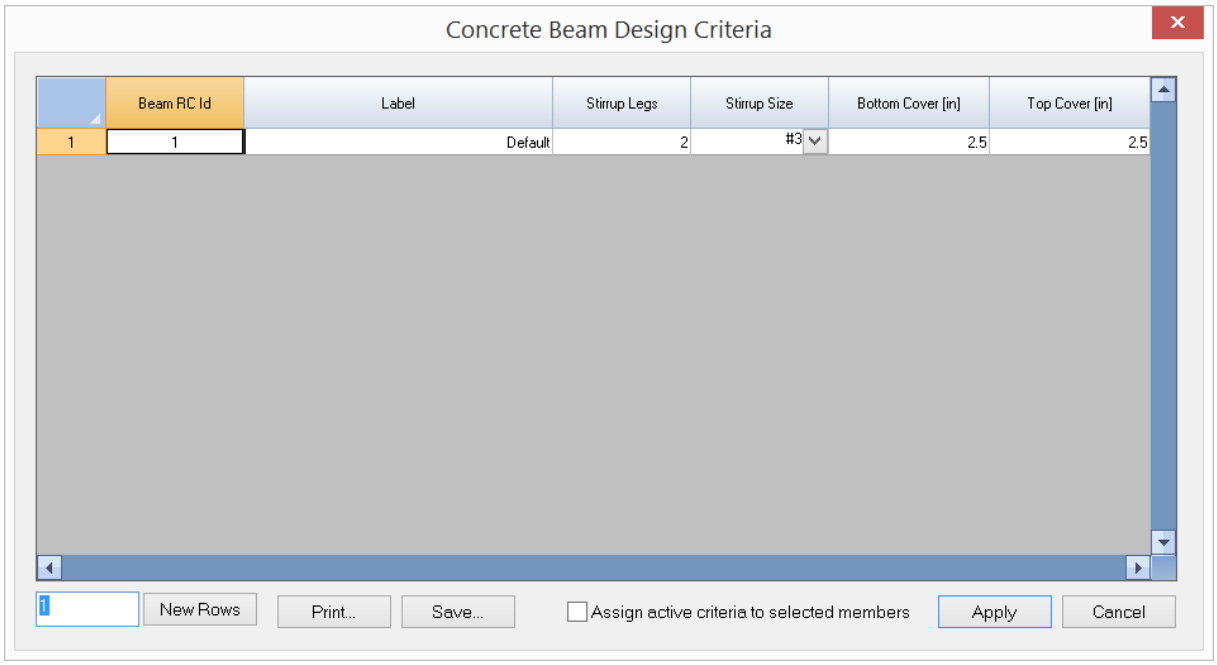
© 2026 ENERCALC, LLC

Design Code	Specifies design codes. Currently the program supports ACI 318-02/05/08/11/14 only.
Axial capacity steps for display	Specifies the number of axial steps for the display of interaction diagrams / surfaces and result data in the spreadsheet. This value should be smaller than neutral axis steps. A value of 20 to 50 is usually adequate.
Exclude concrete displaced by steel	Should almost always be checked. This option is provided for verifications with textbooks only!
Always use 1.0 for Cm	If this option is checked, Cm = 1.0 will be used for all concrete columns. Otherwise, the program will compute Cm automatically based on moment curvature and the existence of transverse loading on the column.
Sustained load combination for computing Beta-d in columns	Specify the load combination that contains the all sustained load cases with each case load factor equal to 1.0.
Ignore compressive force in concrete shear capacity	Specify whether or not to ignore the increase in column concrete shear capacity due to the influence of compressive force. Axial forces are ignored on concrete beams.
Check capacity at column ends only	If this option is checked, column capacity is checked at its ends only. Otherwise, column capacity is checked at every station along the column that analysis outputs.
Compute minimum moment	If this option is checked, a minimum moment Pu * (0.6 + 0.03h) is used for design
Automatically compute support widths	Select this option so that flexural design starts at the support faces and shear design starts at a distance of ‘d’ from the face of support.
Slab/plate min. reinf. ratios	Specify slab/plate minimum top and bottom reinforcement ratios for design. According to ACI 318-02/05/08/11 7.12.2.1, area of shrinkage and temperature reinforcement shall provide at least 0.18% of gross concrete area.

© 2026 ENERCALC, LLC

RC Design Criteria

Concrete Design > RC Design Criteria > RC Beam Design Criteria prompts you with the following dialog.



It allows you to define and/or assign different design criteria to selected concrete beams. An ID is assigned automatically to each design criterion by the program and may not be changed. You may assign a label with 127 maximum characters to each design criterion for easy identification. The beam design criteria include:

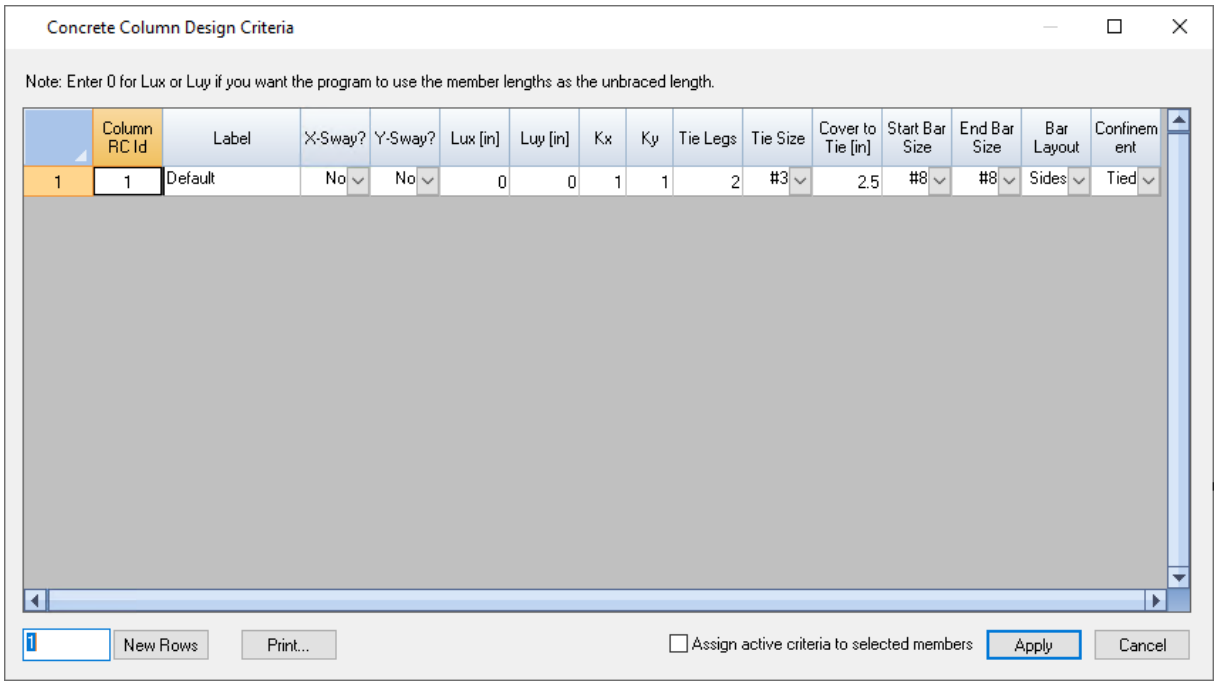
- Number of stirrup legs.
- Stirrup bar size.
- Bottom and top concrete covers measured from section edge to the centroid of longitudinal bars.

You may add one or more criteria by clicking the “New Rows” button. You may also print all design criteria in the list by clicking the “Print” button. The “Assign active criteria to selected members checkbox may be used to assign the active beam design criterion to selected beams. The active criterion refers to the one that currently has focus in the list in the dialog. In order for beam design criteria assignments to take place, beams must be selected beforehand.

The program always has a default beam design criterion labeled “Default”. You may not delete this criterion or change its label. You may however change its properties.

© 2026 ENERCALC, LLC

Concrete Design > RC Design Criteria > RC Column Design Criteria prompts you with the following dialog.



It allows you to define and/or assign different design criteria to selected concrete columns. An ID is assigned automatically to each design criterion by the program and may not be changed. You may assign a label with 127 maximum characters to each design criterion. The column design criteria include:

- Sway flags in x and y directions.
- Unbraced lengths in x and y directions. You may enter zero if you want the program to use the member lengths as the unbraced lengths.
- Effective length factors in x and y directions.
- Number of tie legs.
- Tie bar size.
- Concrete cover to the outside surface of ties. Since different longitudinal bar sizes may be used during automatic section generation, the program computes concrete cover to bar center based on the following formula: “cover to tie” + “tie diameter” + one half of “longitudinal bar diameter” *Note: This is different from concrete cover for beams.*
- Start and end bar trial sizes for section generation.
- Bar layout:
  All Patterns: Patterns will be tried with bars placed on Major Sides, Minor Sides, and Equal Sides of the section as described below.
  Major Sides: Bars will be placed only on the sides parallel to the section major axis.
  Minor Sides: Bars will be placed only on the sides parallel to the section minor axis.
  Equal Sides: Bars will be distributed equally on all sides.
- Confinement: Confining reinforcement can be either tied or spiral. Spiral applies to circular sections only.

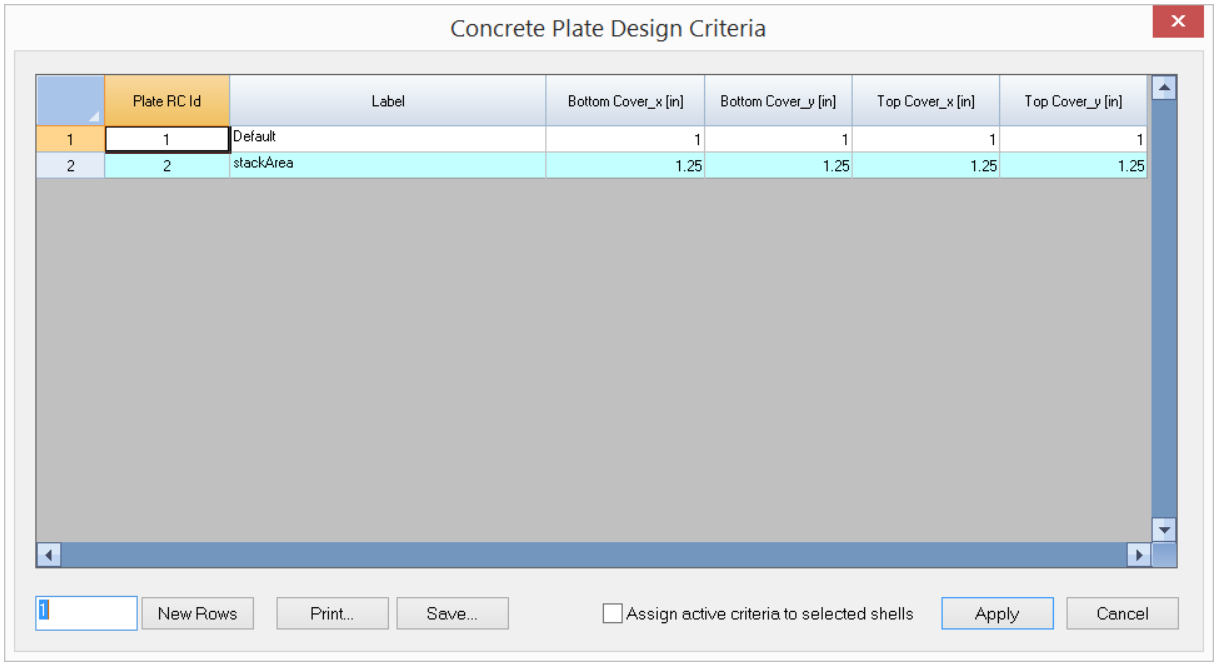
© 2026 ENERCALC, LLC

You may add one or more criteria by clicking the “New Rows” button. You may also print all design criteria in the list by clicking the “Print” button. The “Assign active criteria to selected members” checkbox may be used to assign the active column design criterion to selected columns. The active criterion refers to the one that currently has focus in the list in the dialog. In order for column design criteria assignments to take place, columns must be selected beforehand.

The program always has a default column design criterion labeled “Default”. You may not delete this criterion or change its label. You may however change its properties.

© 2026 ENERCALC, LLC

Concrete Design > RC Design Criteria > RC Plate Design Criteria prompts you with the following dialog.



It allows you to define and/or assign different design criteria to selected concrete plates. An ID is assigned automatically to each design criterion by the program and may not be changed. You may assign a label with 127 maximum characters to each design criterion. The plate design criteria include:

- Bottom-x, bottom-y, top-x and top-y concrete covers measured from plate edge to the centroid of bars

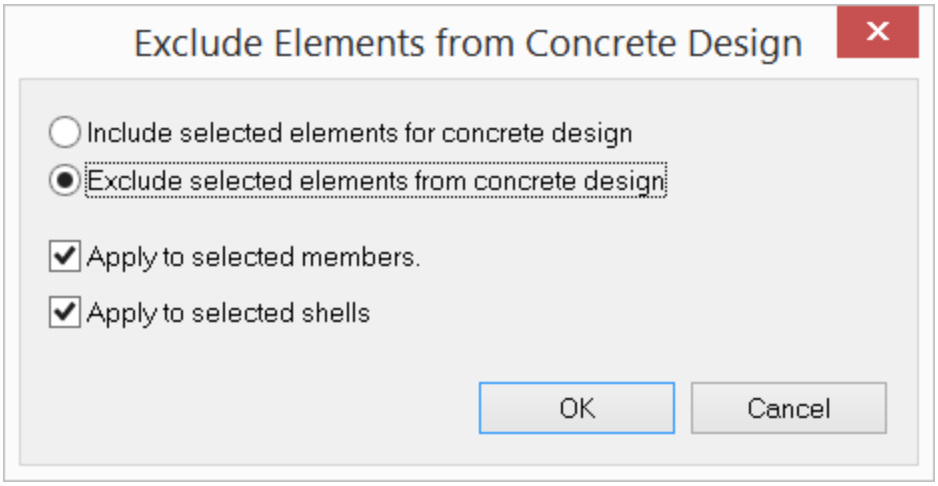
You may add one or more criteria by clicking the “New Rows” button. You may also print all design criteria in the list by clicking the “Print” button. The “Assign active criteria to selected shells” checkbox may be used to assign the active plate design criterion to selected plates. The active criterion refers to the one that currently has focus in the list in the dialog. In order for plate design criteria assignments to take place, plates must be selected beforehand.

The program always has a default plate design criterion labeled “Default”. You may not delete this criterion or change its label. You may however change its properties.

© 2026 ENERCALC, LLC

Exclude Concrete Elements

Concrete Design > Exclude Concrete Elements prompts you with the following dialog.

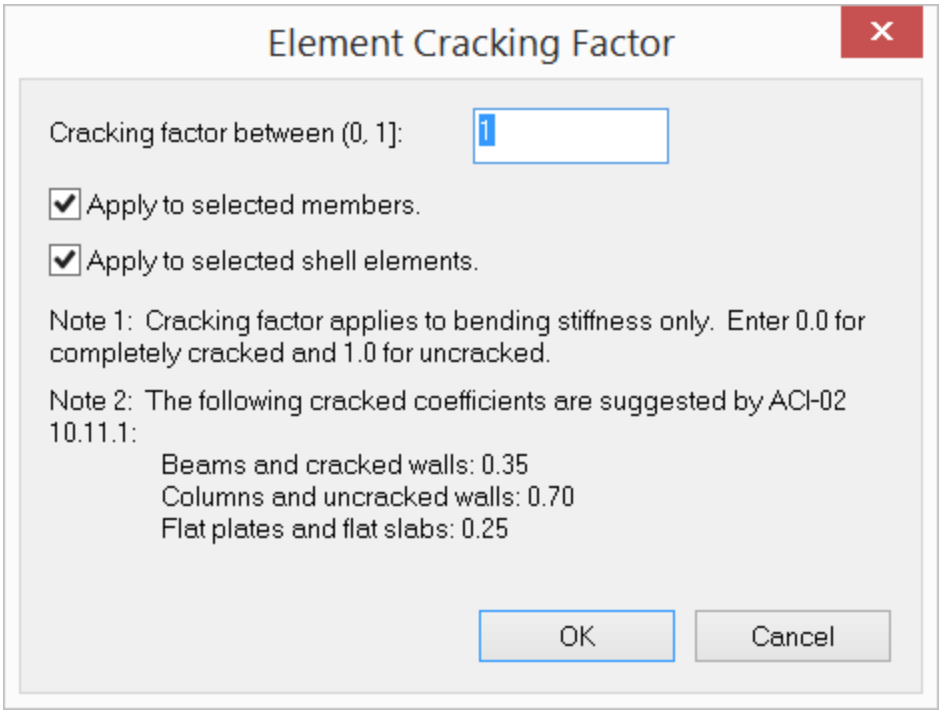


It allows you to include or exclude concrete design for selected beams, columns, and plates. For example, you might want to exclude some plate elements (such as those near supports) from concrete design where large stress spikes are present. Plate envelope contours do not include the excluded shell elements. This makes the contour bands appear more distinct from each other.

© 2026 ENERCALC, LLC

Cracking Factors

Concrete Design > Cracking Factors prompts you with the following dialog.



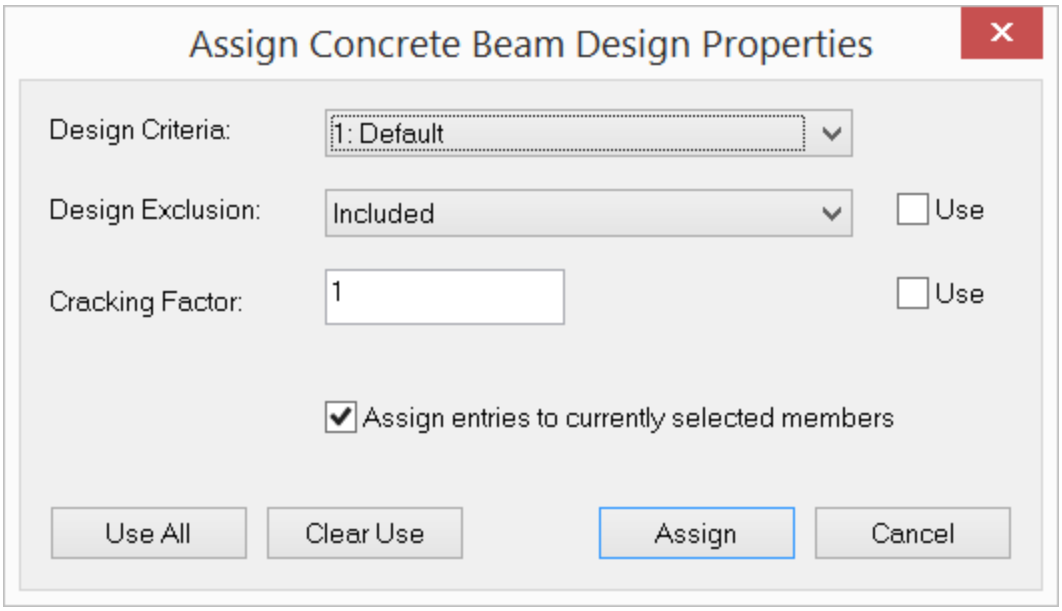
This command is also available from Create (or Modify) > Member Properties > Cracking Factors. It allows you to assign cracking factors to selected beams, columns, and plates. Cracking factors apply only to bending stiffness of members and shell elements.

Note: *Cracking factors are not considered by the program unless you check the option "Use cracked section properties (Icr) for members and finite elements" in Analysis > Analysis Options. Analysis results are cleared after assignment of cracking factors.*

© 2026 ENERCALC, LLC

RC Design Properties

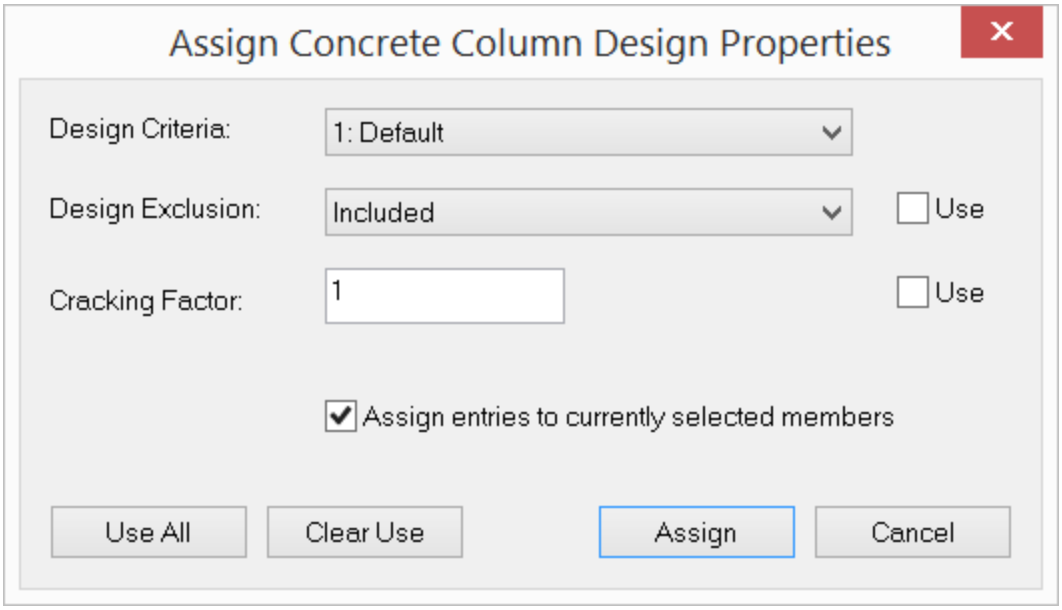
Concrete Design > RC Design Properties > RC Beam Design Properties prompts you with the following dialog.



It allows you to *continuously* assign concrete beam design properties to members. After clicking “Assign”, you can start to *continuously* assign concrete beam design properties by window-selecting members until you right click the mouse or press the ESC key.

© 2026 ENERCALC, LLC

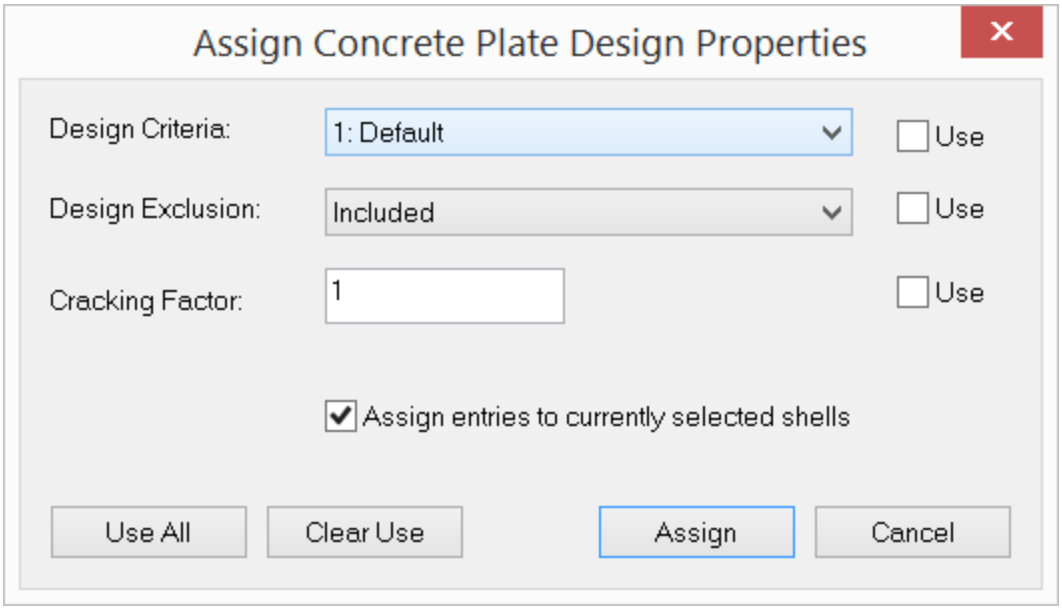
Concrete Design > RC Design Properties > RC Column Design Properties prompts you with the following dialog.



It allows you to *continuously* assign concrete column design properties to members. After clicking "Assign", you can start to *continuously* assign concrete column design properties by window-selecting members until you right click the mouse or press the ESC key.

© 2026 ENERCALC, LLC

Concrete Design > RC Design Properties > RC Plate Design Properties prompts you with the following dialog.

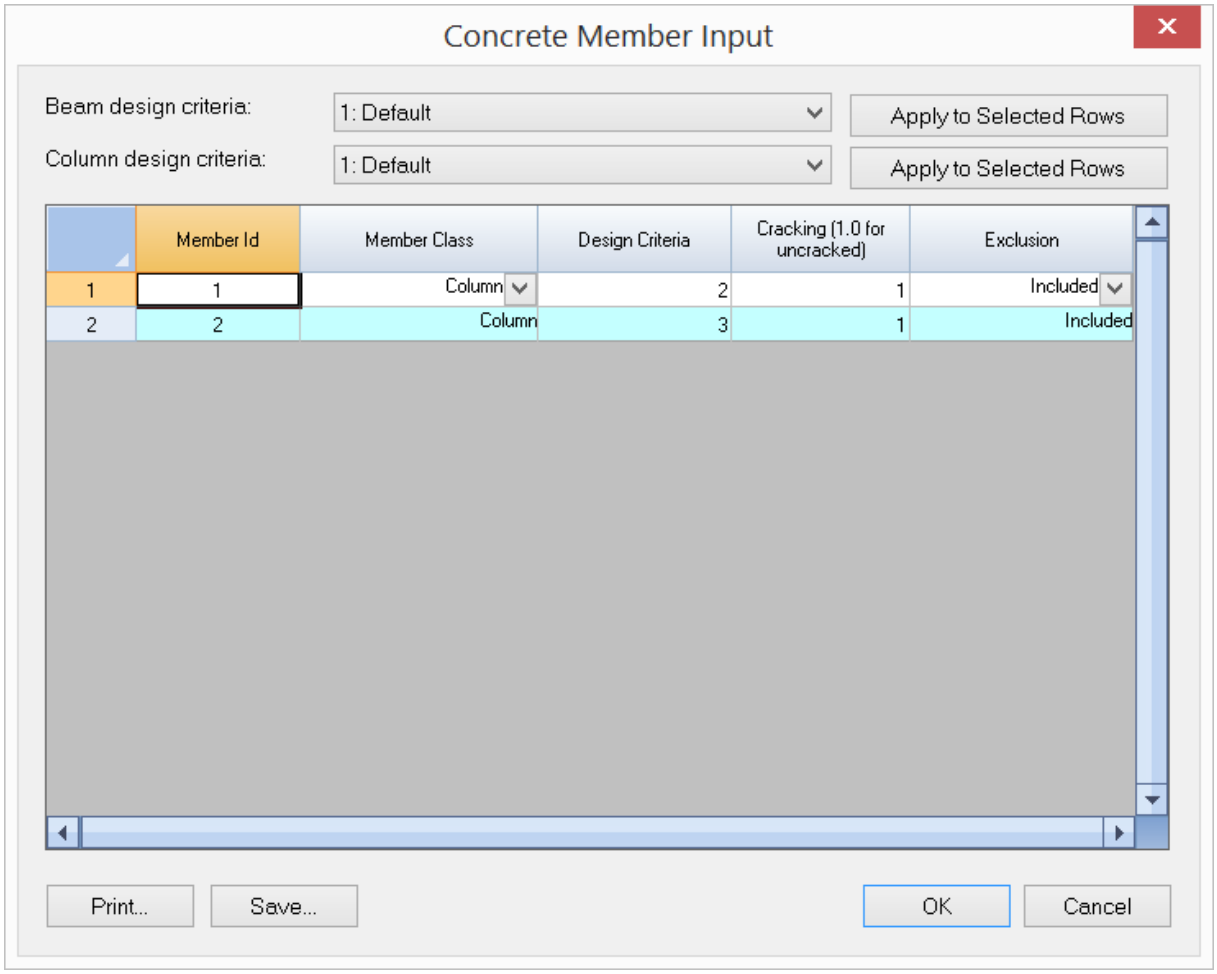


It allows you to *continuously* assign concrete plate design properties to shells. After clicking "Assign", you can start to *continuously* assign concrete plate design properties by window-selecting shells until you right click the mouse or press the ESC key.

© 2026 ENERCALC, LLC

RC Member Input

Concrete Design > RC Member Input prompts you with the following dialog.

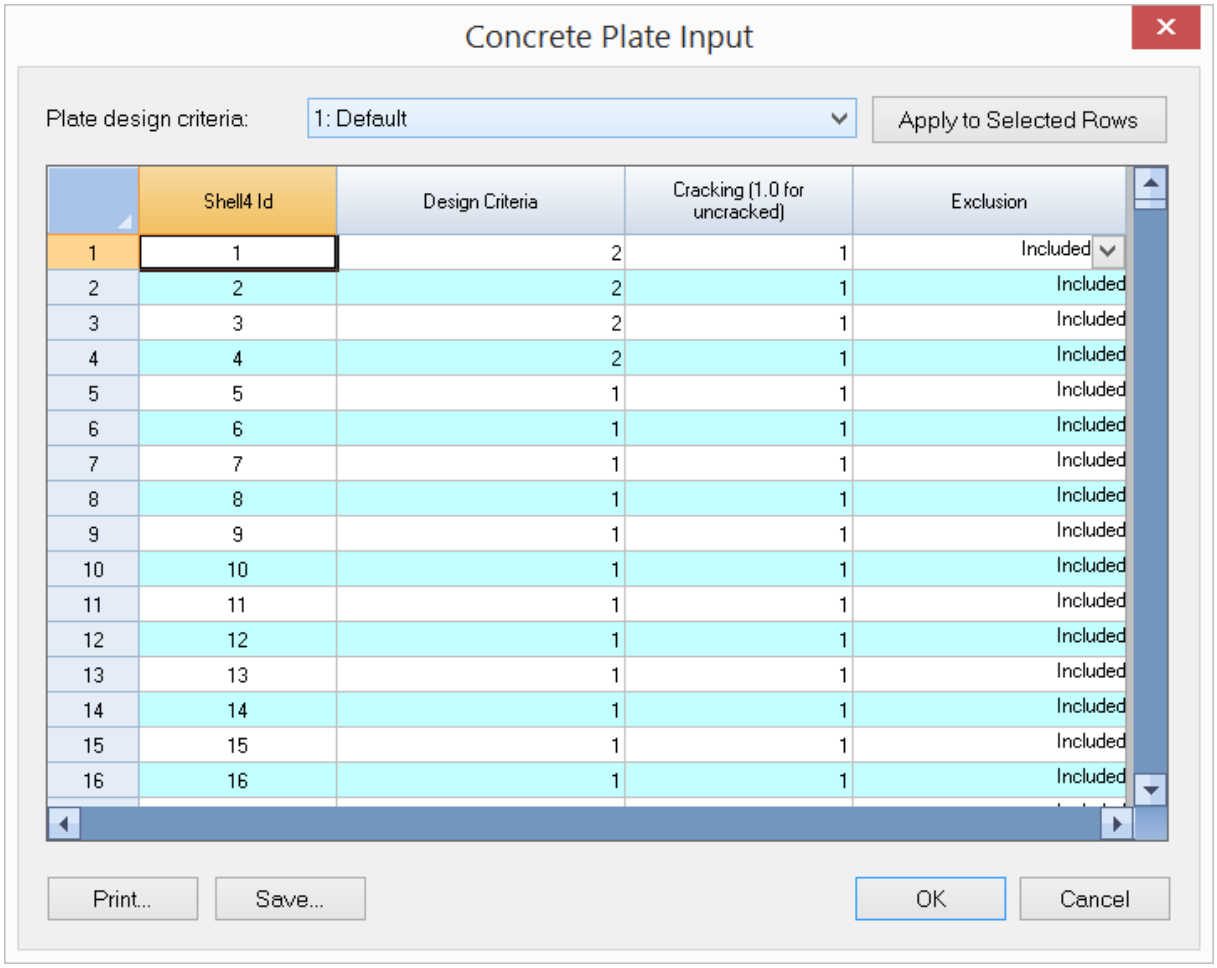


It allows you to enter beams and columns for concrete design in a spreadsheet. Each element includes the class ("B" for beam, "C" for column), design criteria ID, cracking factor, and exclusion design flag (0 for included, 1 for excluded). The element cracking factor with a value between 0 (fully cracked) and 1 (uncracked) applies to the moments of inertia of member elements. You may not modify the member ID. Design criteria IDs must be valid (defined). Beam and column design criteria dialogs are provided for you to correctly pick and apply proper element class and design criteria to selected members.

© 2026 ENERCALC, LLC

RC Plate Input

Concrete Design > RC Plate Input prompts you with the following dialog.



It allows you to enter plates for concrete design in a spreadsheet. Each element includes design criteria ID, cracking factor, and exclusion design flag (0 for included, 1 for excluded). The element cracking factor with a value between 0 (fully cracked) and 1 (uncracked) applies to the moments of inertia of member elements. You may not modify the plate (shell) ID. Design criteria IDs must be valid (defined). Plate design criteria combo box is provided for you to correctly pick and apply proper element design criteria to selected shells.

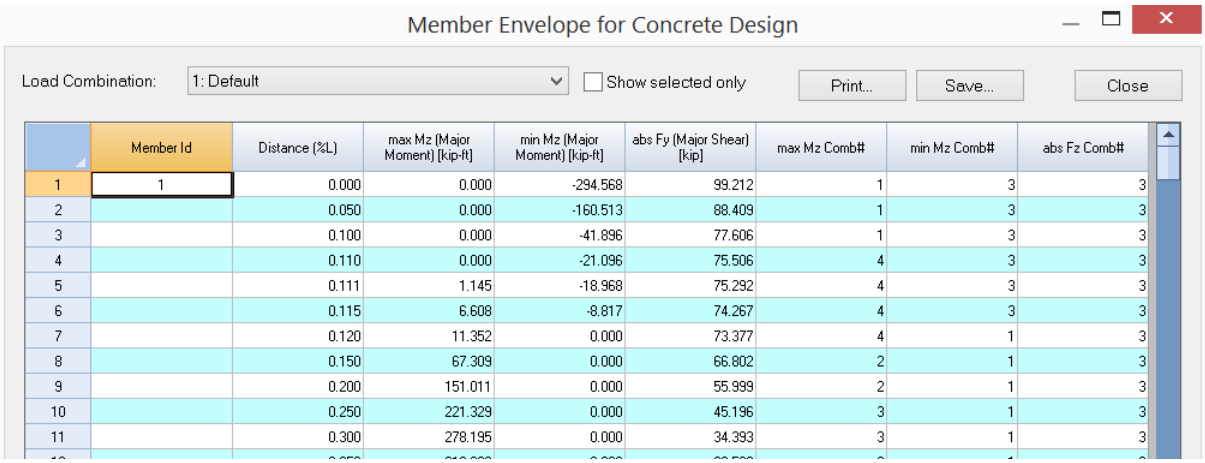
Perform Concrete Design

Concrete Design > Perform Concrete Design performs the concrete design based on the design criteria and design input. You must run the analysis successfully prior to running this command.

© 2026 ENERCALC, LLC

Concrete Design Output

Concrete Design > Concrete Design Output > RC Analysis Envelope displays the following dialog.

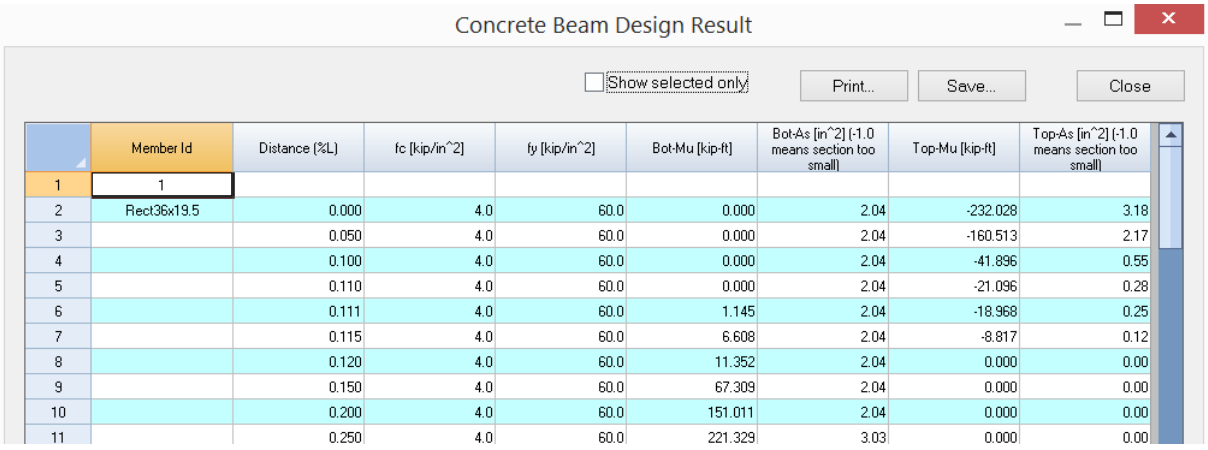


	Member Id	Distance (%L)	max Mz (Major Moment) [kip-ft]	min Mz (Major Moment) [kip-ft]	abs Fy (Major Shear) [kip]	max Mz Comb#	min Mz Comb#	abs Fz Comb#
1	1	0.000	0.000	-294.568	99.212	1	3	3
2		0.050	0.000	-160.513	88.409	1	3	3
3		0.100	0.000	-41.896	77.606	1	3	3
4		0.110	0.000	-21.096	75.506	4	3	3
5		0.111	1.145	-18.968	75.292	4	3	3
6		0.115	6.608	-8.817	74.267	4	3	3
7		0.120	11.352	0.000	73.377	4	1	3
8		0.150	67.309	0.000	66.802	2	1	3
9		0.200	151.011	0.000	55.999	2	1	3
10		0.250	221.329	0.000	45.196	3	1	3
11		0.300	278.195	0.000	34.393	3	1	3

It allows you to view the negative and positive moment envelope as well as the shear envelope for concrete design. You have the option to view the envelope for the selected beams/columns only.

Note: *The envelope only considers the load combinations that are designated for concrete design.*

Concrete Design > Concrete Design Output > RC Beam Results displays the following dialog.

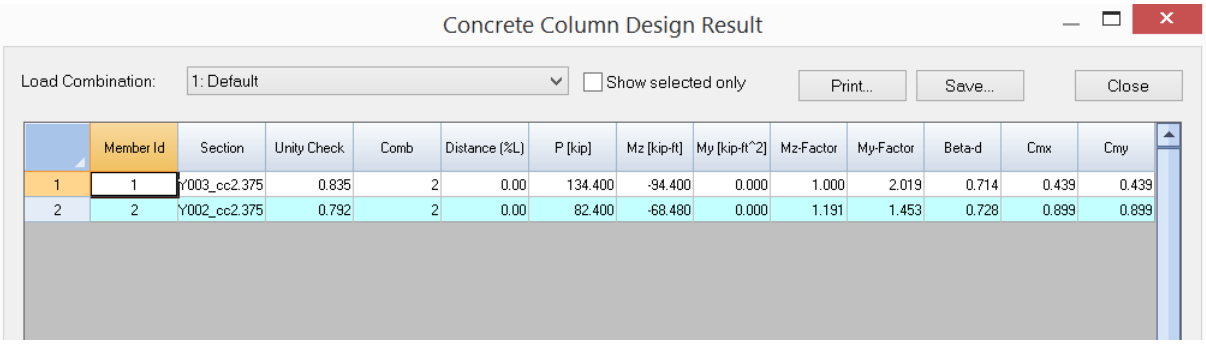


	Member Id	Distance (%L)	fc [kip/in^2]	fy [kip/in^2]	Bot-Mu [kip-ft]	Bot-As [in^2] (-1.0 means section too small)	Top-Mu [kip-ft]	Top-As [in^2] (-1.0 means section too small)
1	1							
2	Rect36x19.5	0.000	4.0	60.0	0.000	2.04	-232.028	3.18
3		0.050	4.0	60.0	0.000	2.04	-160.513	2.17
4		0.100	4.0	60.0	0.000	2.04	-41.896	0.55
5		0.110	4.0	60.0	0.000	2.04	-21.096	0.28
6		0.111	4.0	60.0	1.145	2.04	-18.968	0.25
7		0.115	4.0	60.0	6.608	2.04	-8.817	0.12
8		0.120	4.0	60.0	11.352	2.04	0.000	0.00
9		0.150	4.0	60.0	67.309	2.04	0.000	0.00
10		0.200	4.0	60.0	151.011	2.04	0.000	0.00
11		0.250	4.0	60.0	221.329	3.03	0.000	0.00

It allows you to view top and bottom required steel for flexure and their corresponding design moments at every analysis output station along the member. You have the option to view the RC beam results for the selected beams only.

© 2026 ENERCALC, LLC

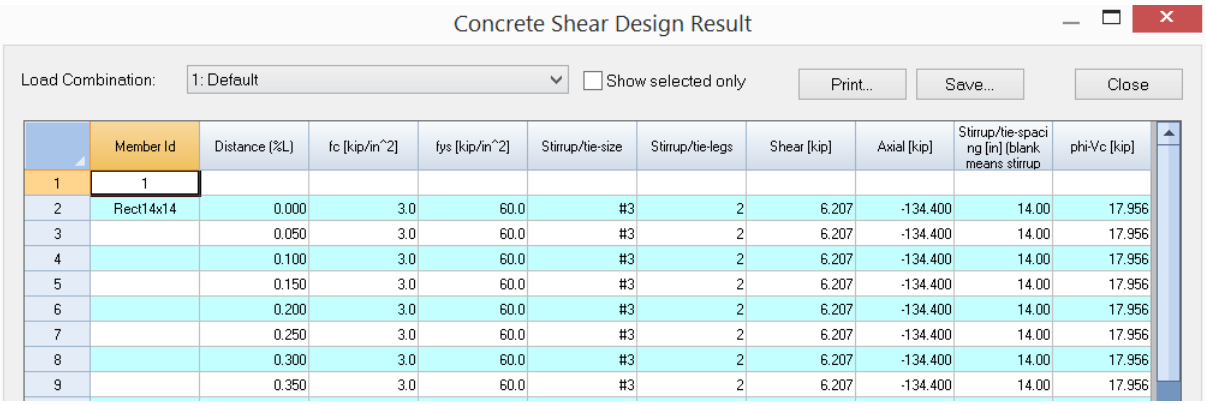
Concrete Design > Concrete Design Output > RC Column Results displays the following dialog.



	Member Id	Section	Unity Check	Comb	Distance (%L)	P [kip]	Mz [kip-ft]	My [kip-ft^2]	Mz-Factor	My-Factor	Beta-d	Cmx	Cmy
1	1	Y003_cc2.375	0.835	2	0.00	134.400	-94.400	0.000	1.000	2.019	0.714	0.439	0.439
2	2	Y002_cc2.375	0.792	2	0.00	82.400	-68.480	0.000	1.191	1.453	0.728	0.899	0.899

It allows you to view the final column design sections and their capacity ratios. Some intermediate results such as moment magnification factors (labeled "Mz-Factor" and "My-Factor"), Beta-d and Cm's are output as well. You have the option to view the RC column results for the selected columns only.

Concrete Design > Concrete Design Output > Member Shear Results displays the following dialog.

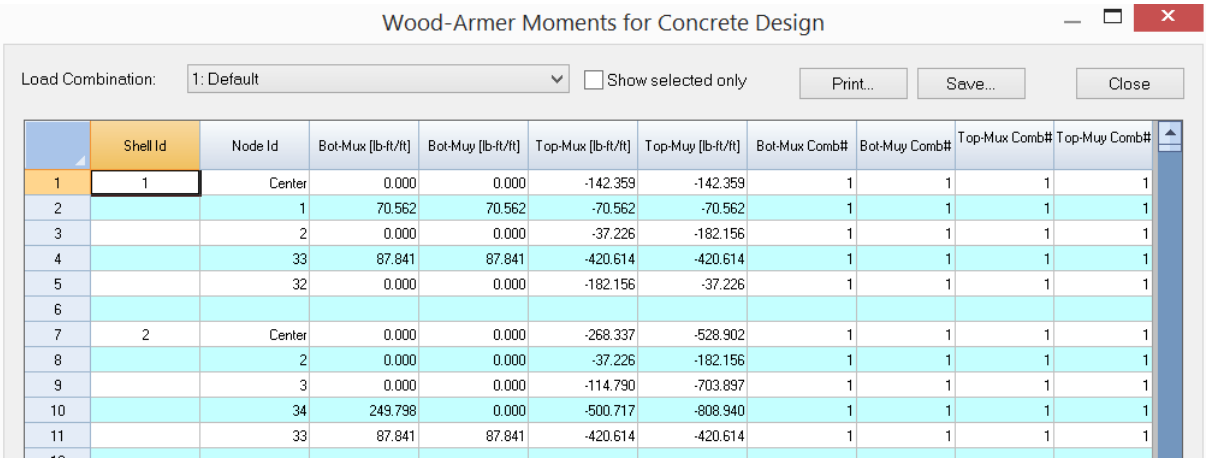


	Member Id	Distance (%L)	fc [kip/in^2]	fys [kip/in^2]	Stirrup/tie-size	Stirrup/tie-legs	Shear [kip]	Axial [kip]	Stirrup/tie-spacing [in] (blank means stirrup	phi-Vc [kip]
1	1									
2	Rect14x14	0.000	3.0	60.0	#3	2	6.207	-134.400	14.00	17.956
3		0.050	3.0	60.0	#3	2	6.207	-134.400	14.00	17.956
4		0.100	3.0	60.0	#3	2	6.207	-134.400	14.00	17.956
5		0.150	3.0	60.0	#3	2	6.207	-134.400	14.00	17.956
6		0.200	3.0	60.0	#3	2	6.207	-134.400	14.00	17.956
7		0.250	3.0	60.0	#3	2	6.207	-134.400	14.00	17.956
8		0.300	3.0	60.0	#3	2	6.207	-134.400	14.00	17.956
9		0.350	3.0	60.0	#3	2	6.207	-134.400	14.00	17.956

It allows you to view the required stirrup (tie) spacing for concrete beam and column shear design. You have the option to view the shear design results for the selected beams and columns only.

© 2026 ENERCALC, LLC

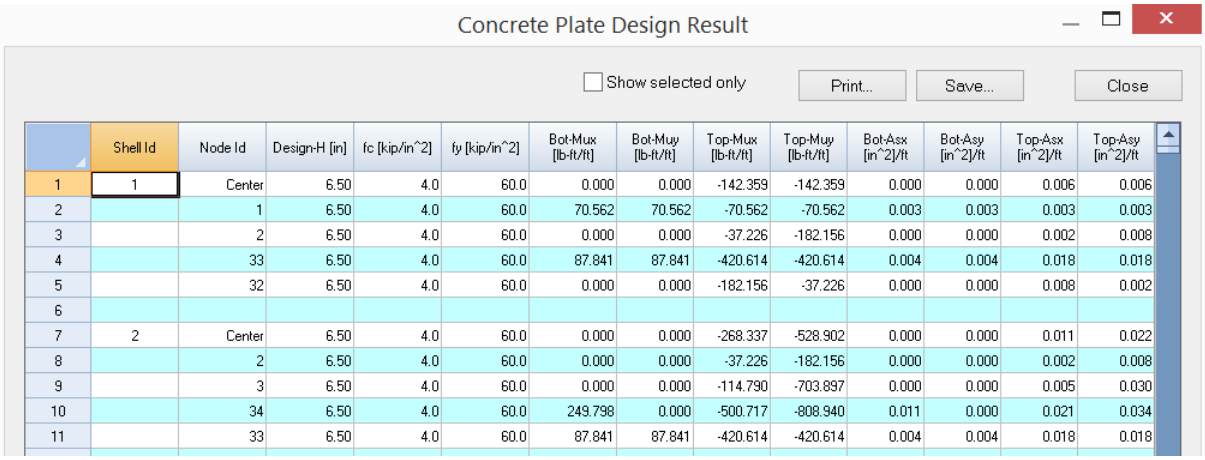
Concrete Design > Concrete Design Output > Wood-Armer Moments displays the following dialog.



	Shell Id	Node Id	Bot-Mux [lb-ft/ft]	Bot-Muy [lb-ft/ft]	Top-Mux [lb-ft/ft]	Top-Muy [lb-ft/ft]	Bot-Mux Comb#	Bot-Muy Comb#	Top-Mux Comb#	Top-Muy Comb#
1	1	Center	0.000	0.000	-142.359	-142.359	1	1	1	1
2		1	70.562	70.562	-70.562	-70.562	1	1	1	1
3		2	0.000	0.000	-37.226	-182.156	1	1	1	1
4		33	87.841	87.841	-420.614	-420.614	1	1	1	1
5		32	0.000	0.000	-182.156	-37.226	1	1	1	1
6										
7	2	Center	0.000	0.000	-268.337	-528.902	1	1	1	1
8		2	0.000	0.000	-37.226	-182.156	1	1	1	1
9		3	0.000	0.000	-114.790	-703.897	1	1	1	1
10		34	249.798	0.000	-500.717	-808.940	1	1	1	1
11		33	87.841	87.841	-420.614	-420.614	1	1	1	1

It allows you to view the critical Wood-Armer moments (top and bottom, local-x and local y directions) and the corresponding load combinations for concrete plates (shells). These moments are used directly in computing the required plate reinforcement areas. You have the option to view the Wood-Armer moments for the selected plates only.

Concrete Design > Concrete Design Output > RC Plate Results displays the following dialog.



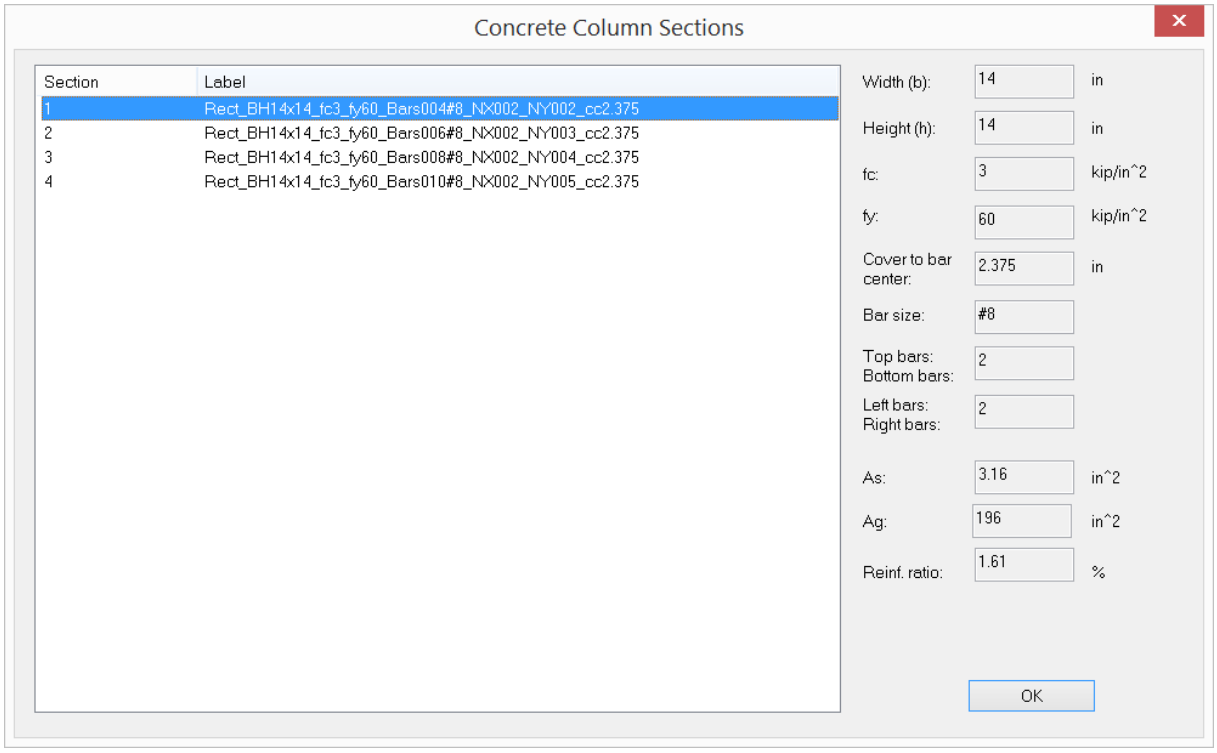
	Shell Id	Node Id	Design-H [in]	fc [kip/in^2]	fy [kip/in^2]	Bot-Mux [lb-ft/ft]	Bot-Muy [lb-ft/ft]	Top-Mux [lb-ft/ft]	Top-Muy [lb-ft/ft]	Bot-Asx [in^2]/ft	Bot-Asy [in^2]/ft	Top-Asx [in^2]/ft	Top-Asy [in^2]/ft
1	1	Center	6.50	4.0	60.0	0.000	0.000	-142.359	-142.359	0.000	0.000	0.006	0.006
2		1	6.50	4.0	60.0	70.562	70.562	-70.562	-70.562	0.003	0.003	0.003	0.003
3		2	6.50	4.0	60.0	0.000	0.000	-37.226	-182.156	0.000	0.000	0.002	0.008
4		33	6.50	4.0	60.0	87.841	87.841	-420.614	-420.614	0.004	0.004	0.018	0.018
5		32	6.50	4.0	60.0	0.000	0.000	-182.156	-37.226	0.000	0.000	0.008	0.002
6													
7	2	Center	6.50	4.0	60.0	0.000	0.000	-268.337	-528.902	0.000	0.000	0.011	0.022
8		2	6.50	4.0	60.0	0.000	0.000	-37.226	-182.156	0.000	0.000	0.002	0.008
9		3	6.50	4.0	60.0	0.000	0.000	-114.790	-703.897	0.000	0.000	0.005	0.030
10		34	6.50	4.0	60.0	249.798	0.000	-500.717	-808.940	0.011	0.000	0.021	0.034
11		33	6.50	4.0	60.0	87.841	87.841	-420.614	-420.614	0.004	0.004	0.018	0.018

It allows you to view the required plate reinforcement areas (top and bottom, local-x and local y directions) and the corresponding Wood-Armer moments for concrete plates (shells). You have the option to view the plate design results for the selected plates only.

Flexural/Axial Interaction

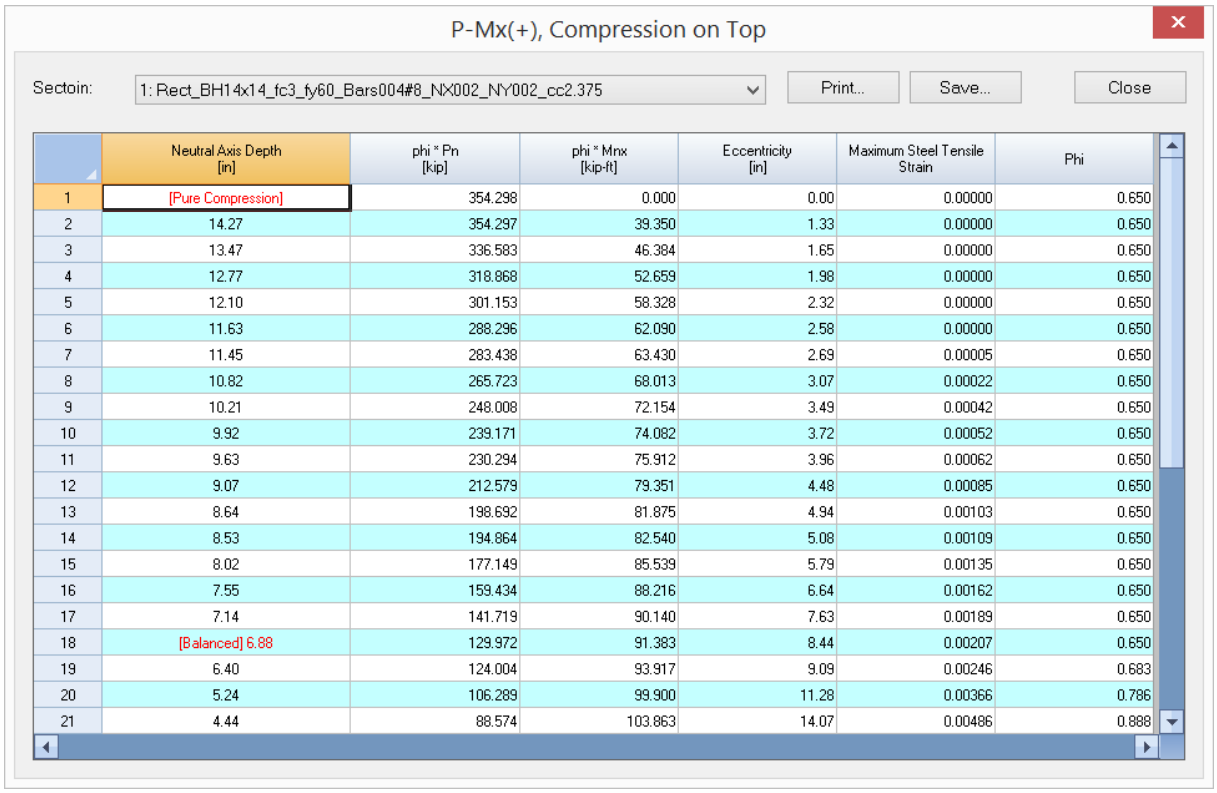
© 2026 ENERCALC, LLC

Concrete Design > Flexural/Axial Interaction > Sections displays all column sections generated by the program based on the input of material, section and column design criteria.



© 2026 ENERCALC, LLC

Concrete Design > Flexural/Axial Interaction > P-Mx (+) displays the P-M_x result data in a spreadsheet, with positive moment about the section major axis (at biaxial angle of 0 degree).



	Neutral Axis Depth [in]	phi * Pn [kip]	phi * Mnx [kip-ft]	Eccentricity [in]	Maximum Steel Tensile Strain	Phi
1	[Pure Compression]	354.298	0.000	0.00	0.00000	0.650
2	14.27	354.297	39.350	1.33	0.00000	0.650
3	13.47	336.583	46.384	1.65	0.00000	0.650
4	12.77	318.868	52.659	1.98	0.00000	0.650
5	12.10	301.153	58.328	2.32	0.00000	0.650
6	11.63	288.296	62.090	2.58	0.00000	0.650
7	11.45	283.438	63.430	2.69	0.00005	0.650
8	10.82	265.723	68.013	3.07	0.00022	0.650
9	10.21	248.008	72.154	3.49	0.00042	0.650
10	9.92	239.171	74.082	3.72	0.00052	0.650
11	9.63	230.294	75.912	3.96	0.00062	0.650
12	9.07	212.579	79.351	4.48	0.00085	0.650
13	8.64	198.692	81.875	4.94	0.00103	0.650
14	8.53	194.864	82.540	5.08	0.00109	0.650
15	8.02	177.149	85.539	5.79	0.00135	0.650
16	7.55	159.434	88.216	6.64	0.00162	0.650
17	7.14	141.719	90.140	7.63	0.00189	0.650
18	[Balanced] 6.88	129.972	91.383	8.44	0.00207	0.650
19	6.40	124.004	93.917	9.09	0.00246	0.683
20	5.24	106.289	99.900	11.28	0.00366	0.786
21	4.44	88.574	103.863	14.07	0.00486	0.888

Concrete Design > Flexural/Axial Interaction > P-Mx (-) displays the P-M_x result data in a spreadsheet, with negative moment about the section major axis (at biaxial angle of 180 degrees).

Concrete Design > Flexural/Axial Interaction > P-My (+) displays the P-M_y result data in a spreadsheet, with positive moment about the section minor axis (at biaxial angle of 90 degrees).

Concrete Design > Flexural/Axial Interaction > P-My (-) displays the P-M_y result data in a spreadsheet, with negative moment about the section minor axis (at biaxial angle of 270 degrees).

Concrete Design > Flexural/Axial Interaction > P-Mx-My displays the P-M_x-M_y result data in a spreadsheet at each biaxial angle step and axial capacity step.

© 2026 ENERCALC, LLC

Concrete Design > Flexural/Axial Interaction > Print Diagrams allows you to view and print the interaction diagrams for each column section.

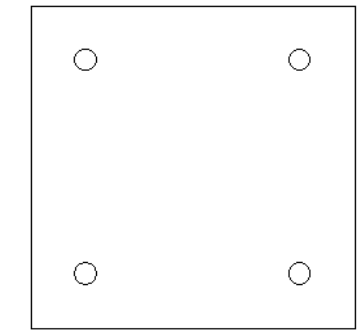
Company/Project: /

Engineer:

Date/Time: 01/31/16 18:30:44

FastFrame-3D V11.01 (1108.0)
(C) ENERCALC INC.

1: Rect_BH14x14_fc3_fy60_Bars004#8_NX002_NY002_cc2.375



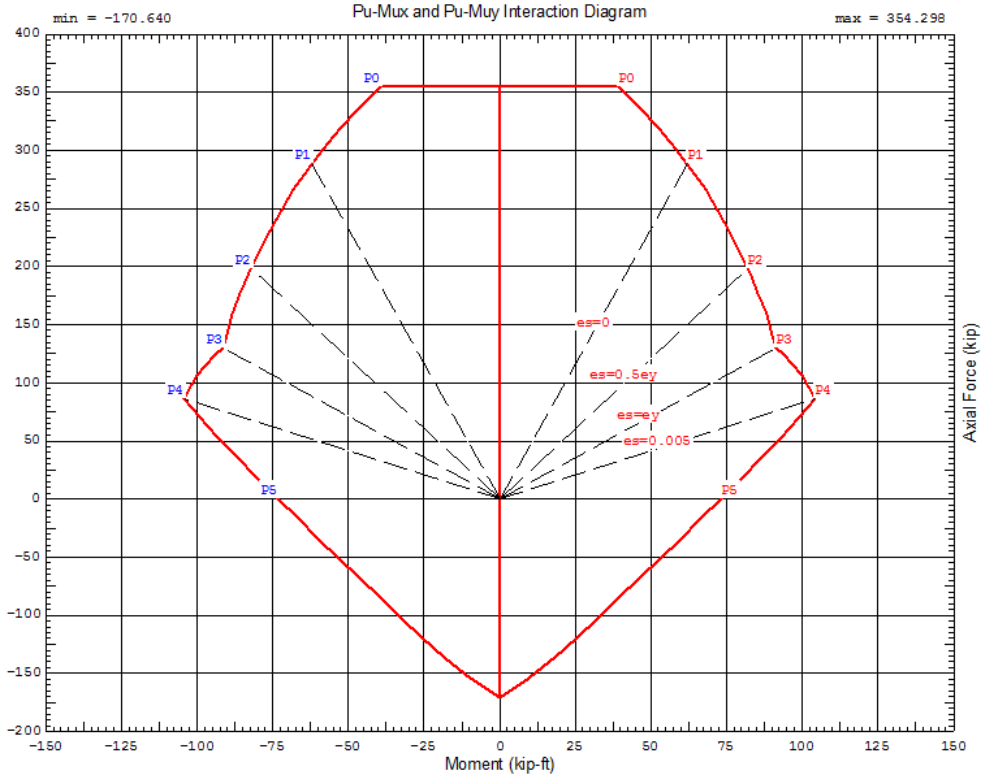
Code: ACI 318-02
Neutral axis steps: 50
Biaxial angle steps: 16
Excl. displ'd concrete: Yes

Size: 14 x 14 in
fc: 3.00 kip/in^2
fy: 60.00 kip/in^2
Bars: 4-#8
Cover(c.c): 2.375 in
Ag: 196.00 in^2
As: 3.16 in^2
Roh: 1.61%
Ix: 3201.33 in^4
Iy: 3201.33 in^4
Clear spacing: 8.25 in

Control Points (kip, kip-ft)

PT	Pu	Mux(+)	max es
P0	354.30	39.35	0.00000
P1	288.30	62.09	0.00000
P2	198.69	81.87	0.00103
P3	129.97	91.38	0.00207
P4	86.42	104.26	0.00500
P5	-0.00	73.30	0.01015

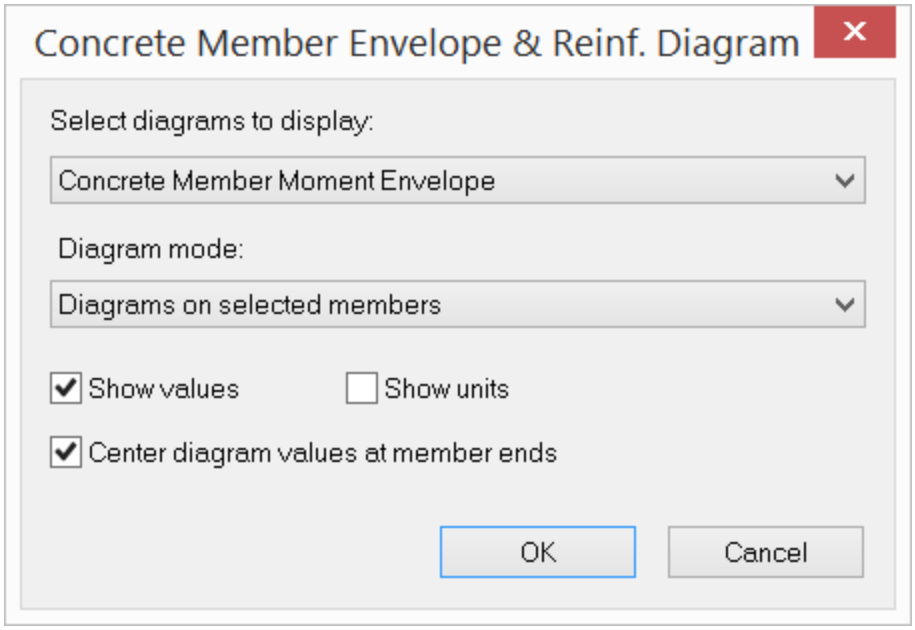
PT	Pu	Muy(-)	max es
P0	354.30	-39.35	0.00000
P1	288.30	-62.09	0.00000
P2	198.69	-81.87	0.00103
P3	129.97	-91.38	0.00207
P4	86.42	-104.26	0.00500
P5	-0.00	-73.30	0.01015



The red and blue lines are the interaction diagrams about section major and minor axes respectively. A sketch of the section and the key control points are listed above the diagrams as well.

Concrete Design Diagrams

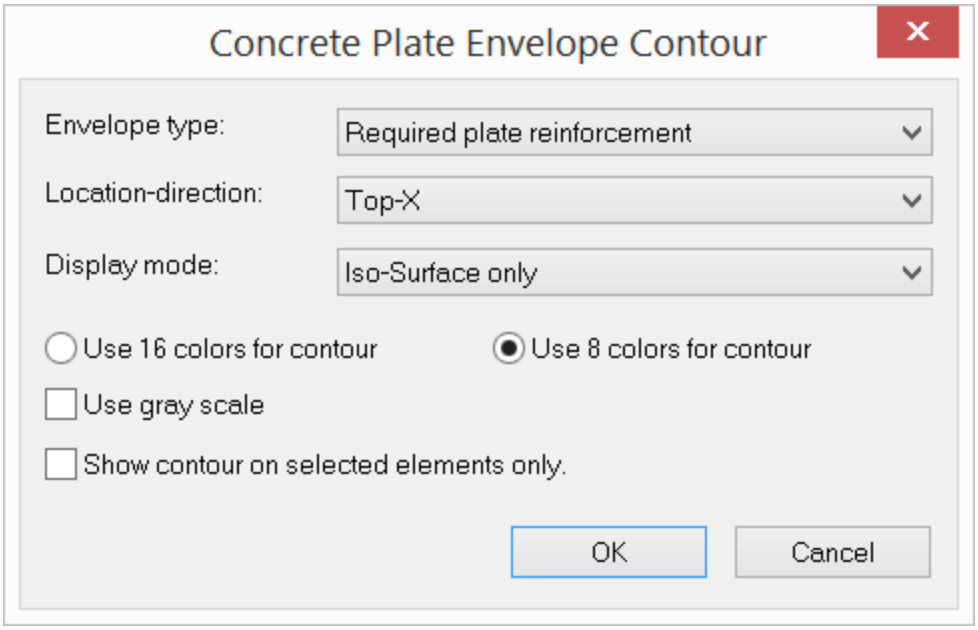
Concrete Design > Concrete Design Diagrams > RC Member Envelope Diagram displays the following dialog.



It allows you to view the required flexural reinforcement as well as the moment and shear envelope used for designing concrete beams. It also allows you to view required stirrup or tie spacing for concrete beams and columns. You have the option to view the member envelope diagrams for the selected members only.

© 2026 ENERCALC, LLC

Concrete Design > Concrete Design Diagrams > RC Plate Envelope Contour displays the following dialog.

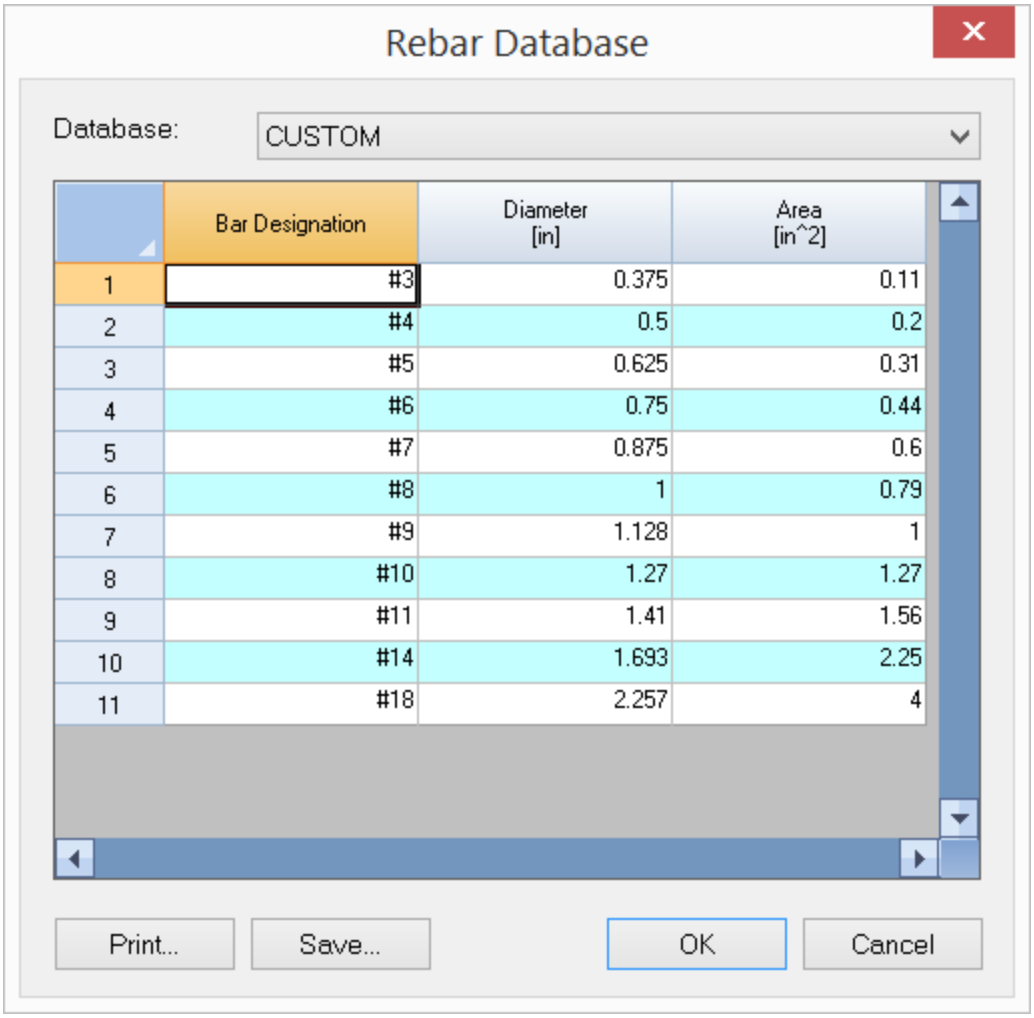


It allows you to view the required flexural reinforcement as well as the Wood-Armer moments (top and bottom, local x and y directions) for concrete plates (shells). You have the option to view the plate envelope contours for the selected plates only.

© 2026 ENERCALC, LLC

Concrete Design Tools

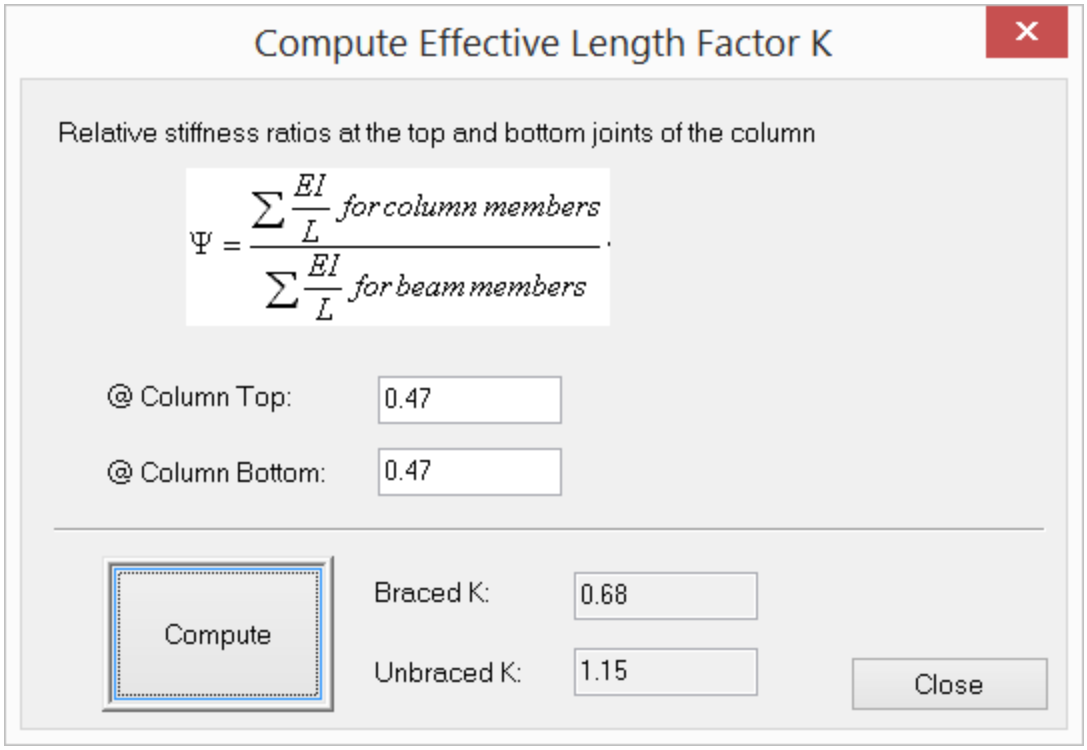
Concrete Design > Concrete Design Tools > Rebar Database displays the following dialog.



It allows you to select different rebar databases for use in concrete design.

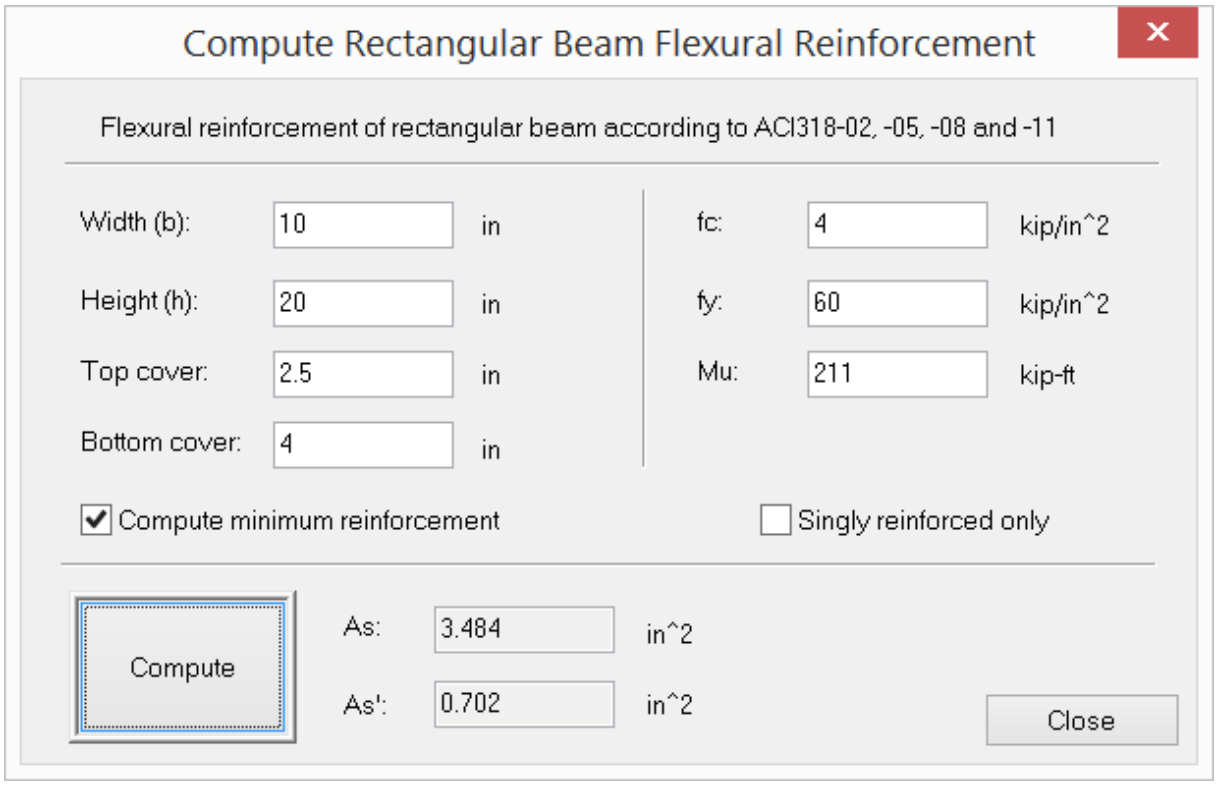
© 2026 ENERCALC, LLC

Concrete Design > Concrete Design Tools > K Calculator allows you to accurately calculate effective length factors (braced and unbraced Ks) based on the beam and column relative stiffness input.



© 2026 ENERCALC, LLC

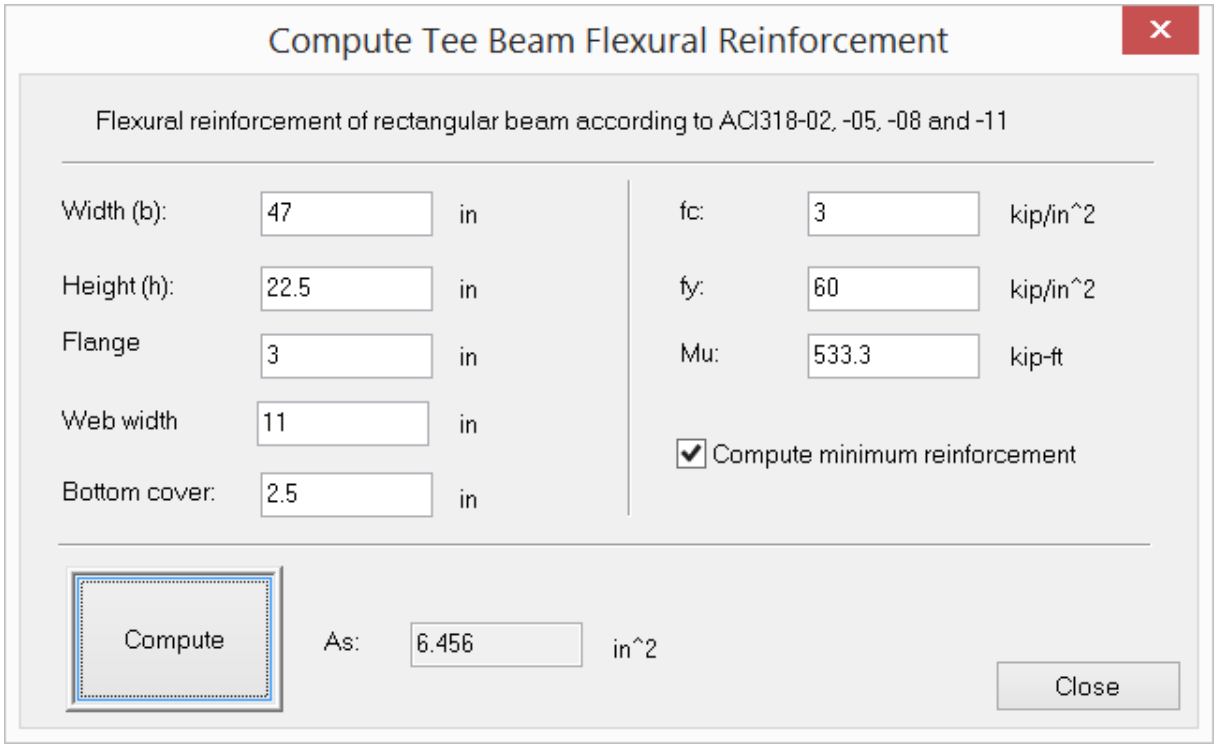
Concrete Design > Concrete Design Tools > Quick R-Beam Flexural Design allows you to quickly design a rectangular concrete beam according to ACI 318-02/05/08/11/14.



Minimum reinforcement may be optionally computed. You have the option to design the rectangular beam as singly or doubly reinforced. A negative reinforcement area means the design fails.

© 2026 ENERCALC, LLC

Concrete Design > Concrete Design Tools > Quick T-Beam Flexural Design allows you to quickly design a concrete tee beam according to ACI 318-02/05/08/11/14.



Minimum reinforcement may be optionally computed. The tee beam is always designed as singly reinforced. A negative reinforcement area means the design fails.

Unity Check

Concrete Design > Unity Check displays a graphical result of the pass/fail status of all designed members. Passing members are highlighted in blue. Failing members are highlighted in red.

Steel Design

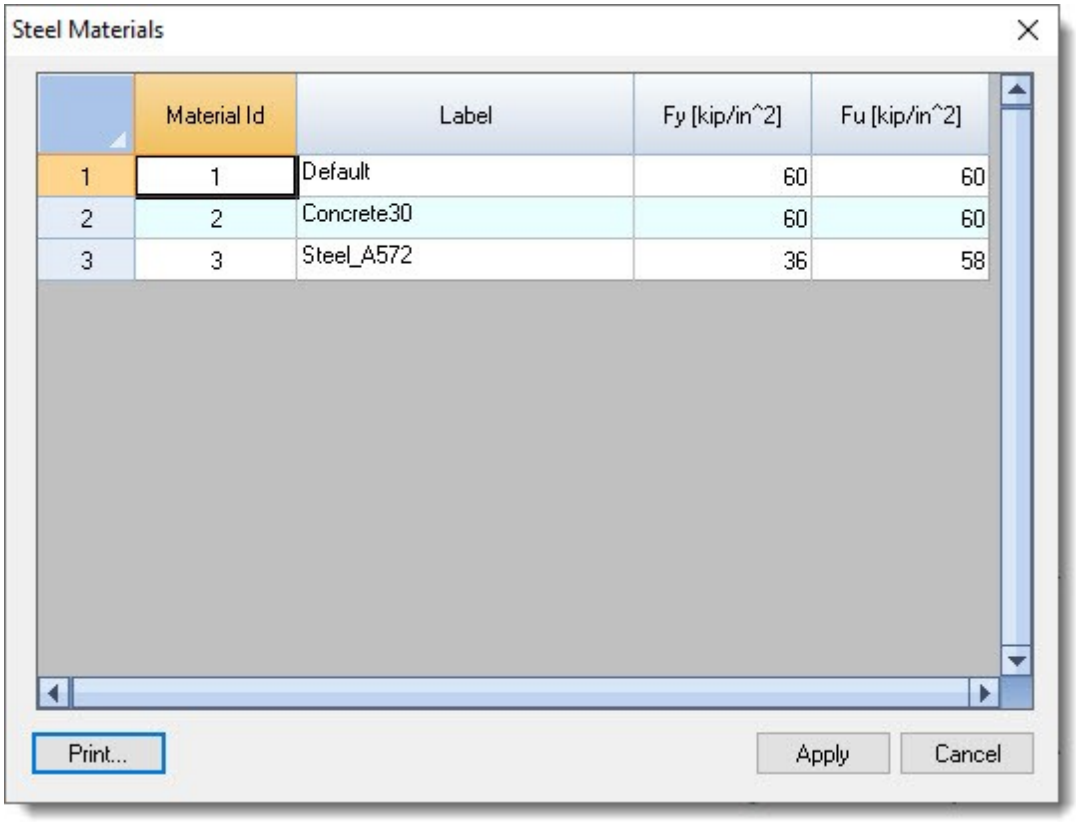
The Steel Design ribbon provides commands related to steel design for beams and columns. This includes tools to input design parameters, execute the design process and review the design results in graphical and tabular form.

The commands in the Steel Design ribbon do not affect the analysis results. A static analysis must be done successfully before design can be performed.

© 2026 ENERCALC, LLC

Steel Materials

Steel Design > Steel Materials prompts you with the following dialog.



It allows you to define steel strength properties for the existing materials. The strength properties include:

- Steel yield stress Fy
- Steel rupture stress Fu

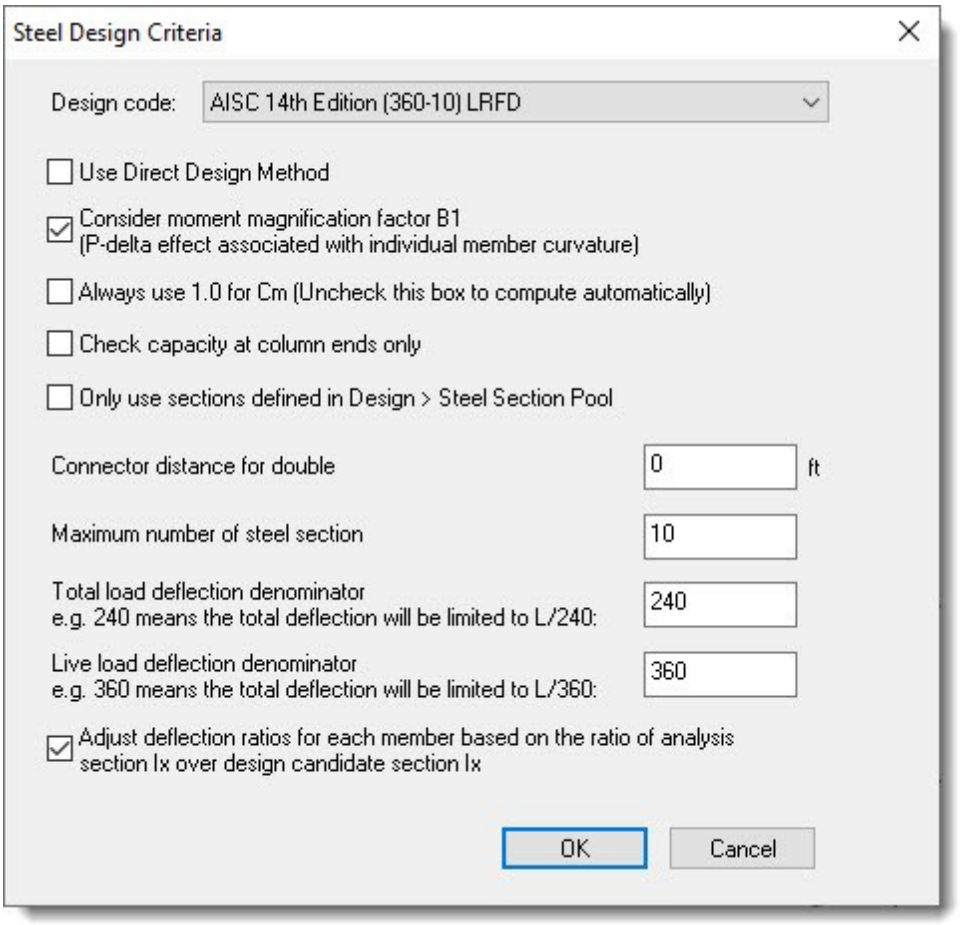
If standard materials are used in Tables > Material Data, these strength properties will be set automatically. You may override these properties prior to performing steel design. No steel design will be performed on a member if its modulus is not close (within 10%) to 29E3 ksi. You should not modify materials that are not steel on this dialog.

© 2026 ENERCALC, LLC

Steel Design Criteria

© 2026 ENERCALC, LLC

Steel Design > Steel Design Criteria > Steel Design Criteria prompts you with the following dialog.



Use Direct Design Method currently only affects how the moment magnification factor B1 is calculated. You also have the option not to consider moment magnification factor B1 altogether. Please be advised that P-Delta analyses should be performed on load combinations that are used for steel design.

To be conservative, you can always use 1.0 for Cm that accounts for nonuniform moment. Uncheck the Always use 1.0 for Cm" if you would like the program to calculate Cm for automatically.

You have the option to only use sections defined in Design > Steel Section Pool during the design process. This is useful if you do not want the program to use too many steel section sizes for the entire model.

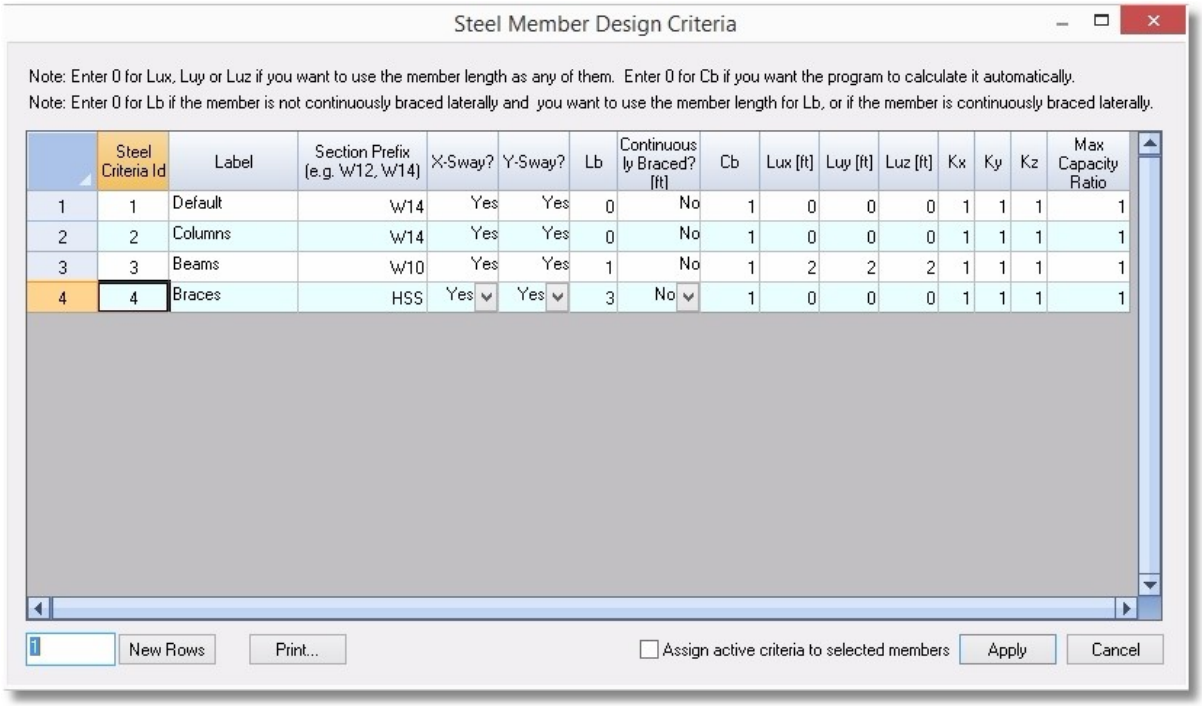
Connector distance for double angles is used for sections that are double angles.

The default number of section candidates designed for each member is 10.

You can also specify limits for total load deflection and live load deflection.

© 2026 ENERCALC, LLC

Steel Design > Steel Design Criteria > Steel Member Design Criteria prompts you with the following dialog.



It allows you to define and assign design criteria for members.

An ID is assigned automatically to each design criterion by the program and may not be changed. You may assign a label with 127 maximum characters to each design criterion.

© 2026 ENERCALC, LLC

The column design criteria include:

- Section Prefix, which is a comma delimited list. For example, if you want the member section to be with W10 or W12 size, enter the prefix as “w10,w12”. You also specify the prefix as the exact AISC shapes. Use the prefix “Default” if you do not want the member section changed from the original shape.

- Sway flags in x and y directions.

- The length between points that are braced against lateral displacement of compression flange Lb. Currently, the program only supports equal Lb along the member length. For non-continuously braced, the program will use the member length for Lb if the value is entered 0. For continuously braced, 0 must be entered for Lb.

- Lateral-torsional buckling modification factor for nonuniform moment diagrams Cb. The program will automatically calculate Cb if the value is entered 0. You can always use Cb = 1.0 for conservative reasons.

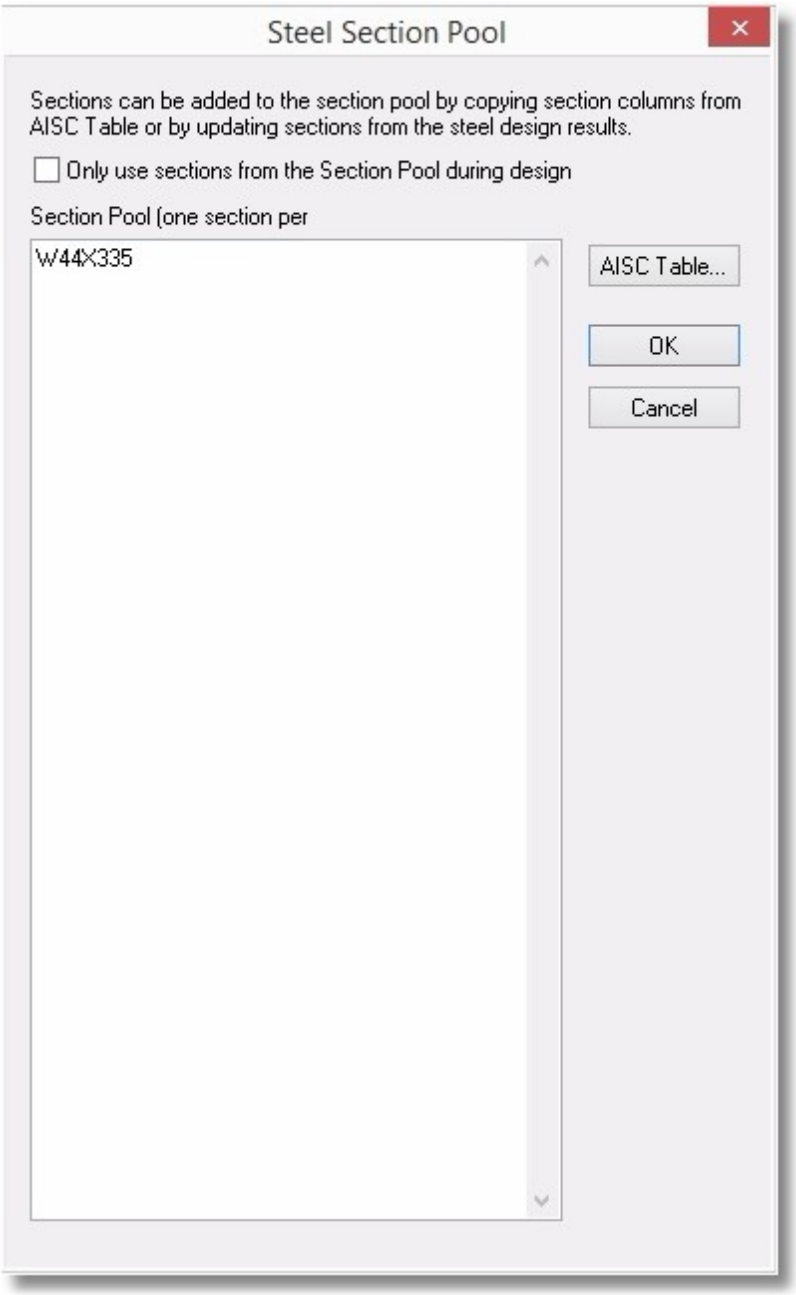
- Unbraced lengths in x, y and z directions. You may enter zero if you want the program to use the member lengths as the unbraced lengths.

- Effective length factors in x, y and z directions.

- You have the option to set the maximum capacity ratio. By default, this ratio is 1.0. You can set a value less than 1.0 (but greater than 0.0) for conservative reasons.

© 2026 ENERCALC, LLC

Steel Design > Steel Design Criteria > Steel Section Pool prompts you with the following dialog.

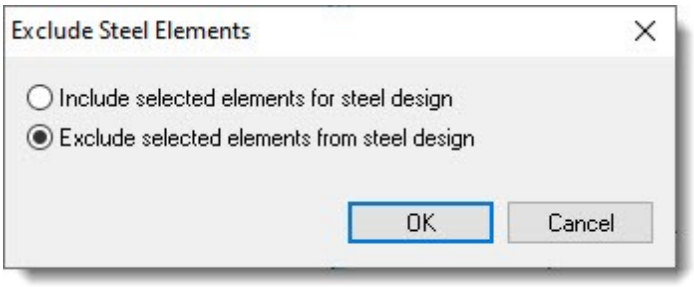


It allows you to define a list of sections that may be used exclusively for design.

You can copy a list of section labels from the AISC table. You can also enter sections manually. Each line in the section pool box can only contain one section. Furthermore, you can automatically add all the section candidates to the section pool (from Design > Steel Design Results).

© 2026 ENERCALC, LLC

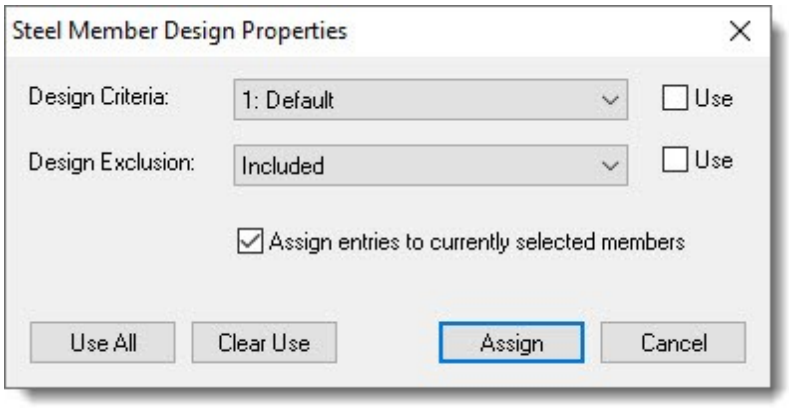
Steel Design > Steel Design Criteria > Exclude Steel Elements prompts you with the following dialog.



It allows you to include or exclude steel design for selected members.

Steel Member Design Properties

Steel Design > Steel Member Design Properties prompts you with the following dialog.

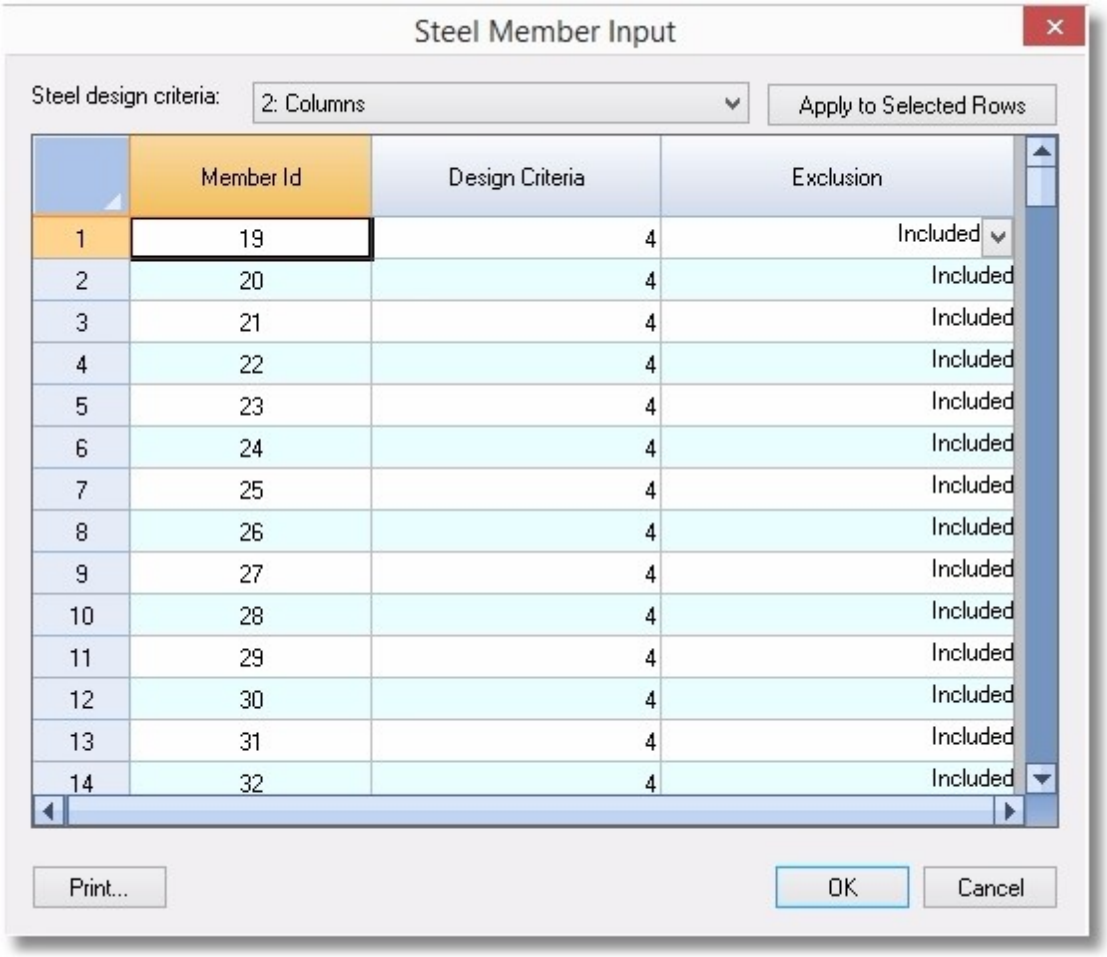


It allows you to continuously assign steel design properties to members. After clicking "Assign", you can start to continuously assign steel design properties by window-selecting members until you right click the mouse or press the ESC key.

© 2026 ENERCALC, LLC

Steel Member Input

Steel Design > Steel Member Input prompts you with the following dialog.



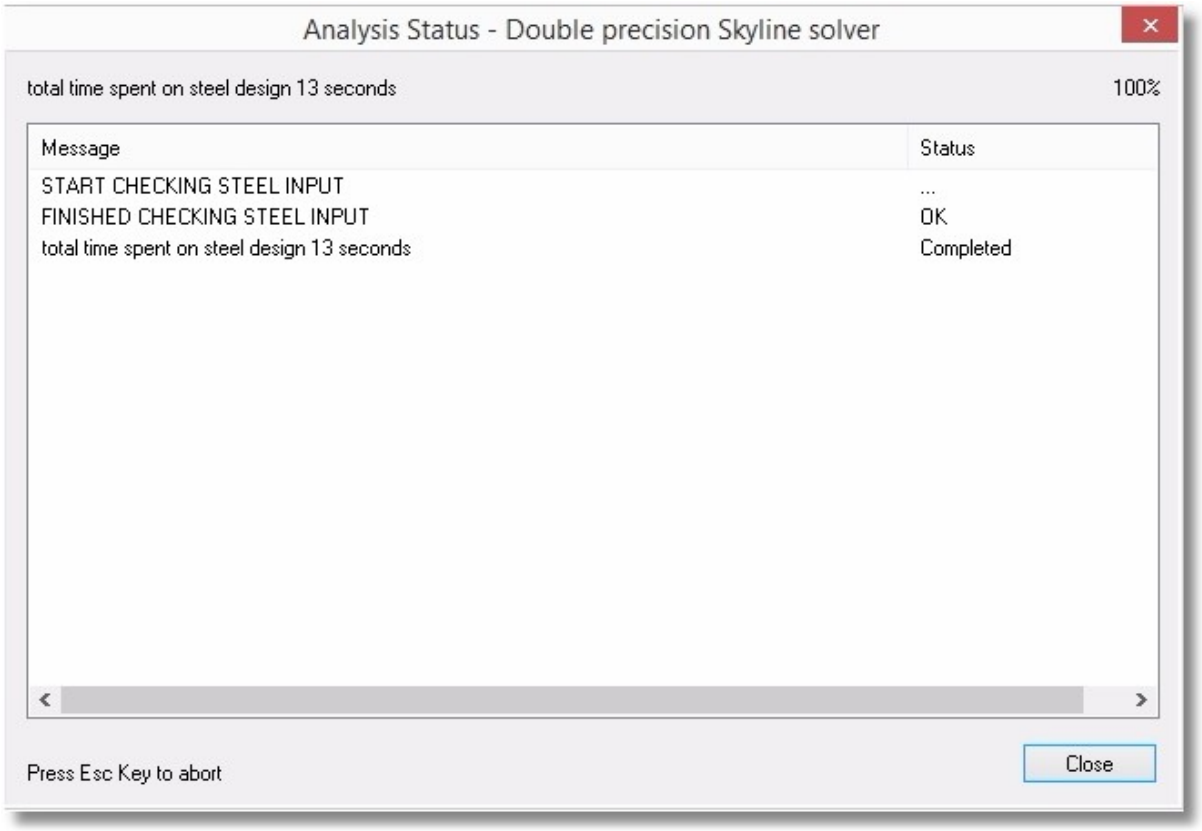
It allows you to enter members for steel design in a spreadsheet. Each element includes the design criteria ID, and exclusion design flag (0 for included, 1 for excluded). You may not modify the member ID. Design criteria IDs must be valid (defined). Steel design criteria combo box is provided for you to correctly pick and apply proper steel design criteria to selected members.

© 2026 ENERCALC, LLC

Perform Steel Design

Click Steel Design > Perform Steel Design to initiate a Steel Design process. (Static analysis results must be current in order to run a steel design.)

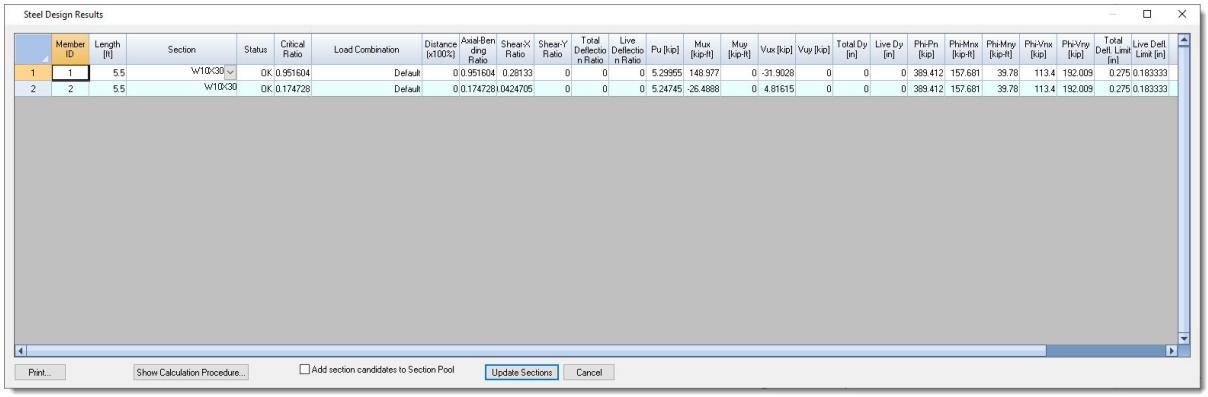
The message box will indicate when the design is complete.



© 2026 ENERCALC, LLC

Steel Design Results

Steel Design > Steel Design Results allows you to view the steel design results.



It also allows you to update member sections.

The section column in the screen capture above includes a list box that contains the member original section (first entry in the list box) and designed section candidates (second or more entries in the list box). You can change the member sections by picking the proper section candidates. Please be advised you need to re-analyze and design after one or more member sections are updated.

In addition, you can add all the section candidates to Section Pool, which can be used in the next round of design.

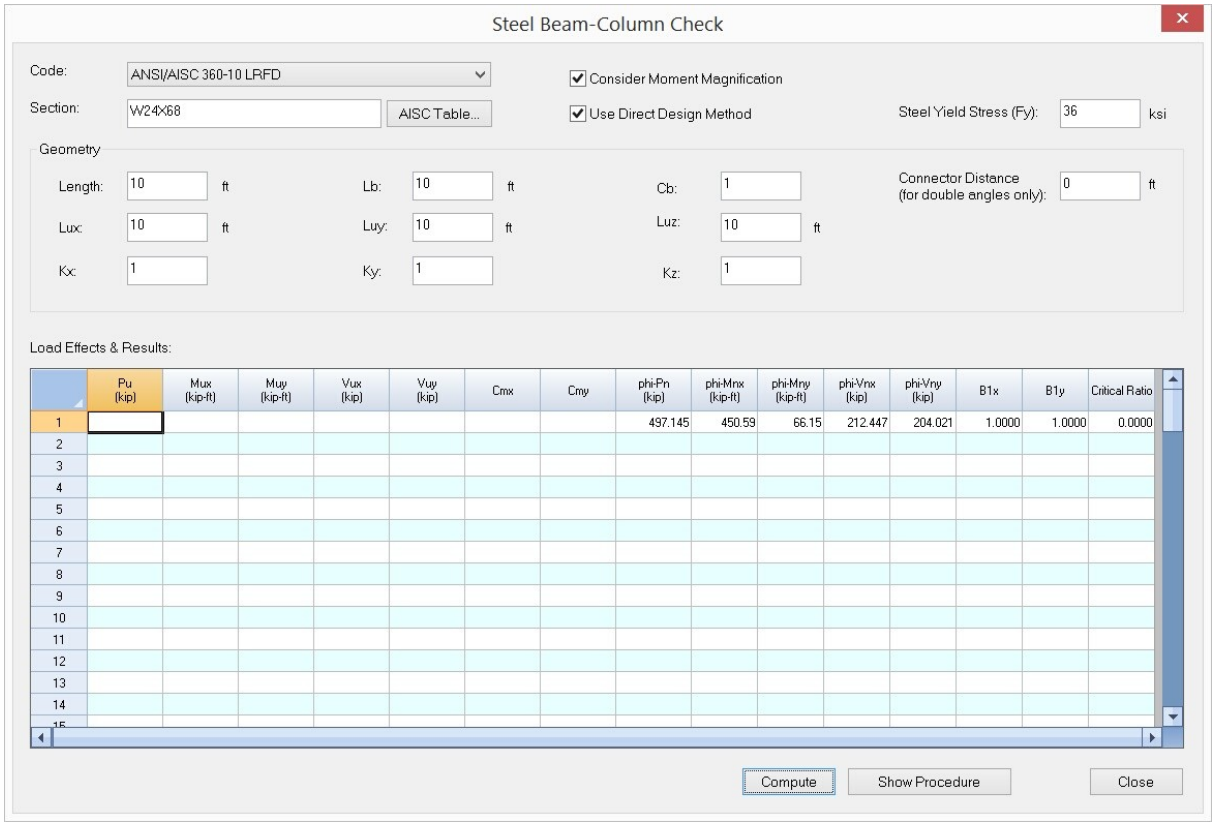
Finally, you can view the detailed calculation procedure for each member for the most critical load condition.

© 2026 ENERCALC, LLC

Steel Design Tools

Steel Design > Steel Design Tools > K Calculator offers a tool to calculate the K value of columns by entering the ratios of the sum of the EI/L values at the top and bottom of the column segment.

Steel Design > Steel Design Tools > Steel Section Check allows you to perform steel section capacity check.



Lux, Luy and Luz are unbraced lengths in local x, y and z directions. Kx, Ky and Kz are unbraced length factors in local x, y and z directions.

Lb is the unbraced lateral length. Cb is the lateral-torsional buckling modification factor for non-uniform moment diagrams. It should be greater or equal to 1.0. You can use 1.0 for Cb conservatively.

Connector Distance is used for double angles only.

Pu, Mux, Muy, Vux, Vuy are required axial, major moment, minor moment, major shear and minor shear. For Pu, the compressive force is positive while tensile force is negative. Moment Mux is positive when section top most fiber is under compression. Moment Muy is positive when section rightmost fiber is under compression. Moment magnification may be optionally considered to account for the P-delta (P-δ) effect.

© 2026 ENERCALC, LLC

If direct design method is chosen, the program will calculate stiffness reduction parameter based on Eq. C2-2a and C2-2b of the code.

Cmx, Cmy are coefficients accounting for non-uniform moments when computing moment magnification. You can use 1.0 for Cmx and Cmy conservatively. If Cmx or Cmy is 0, 1.0 is used in the computation instead.

Results include axial capacity (phi-Pn), moment capacity (phi-Mnx, phi-Mny), shear capacity (phi-Vnx, phi-Vny), moment magnification factors (B1x, B1y) and critical ratio. The section is deemed safe to resist a load if the critical ratio is less than 1.0, otherwise, the section is deemed unsafe. Please note that for single angle, the moment capacity is given about the principal axes (w-w and z-z).

© 2026 ENERCALC, LLC

08/21/14 19:10:38

W12X65 - using AISC 360-10 LRFD Method

Section Input

Section W12X65
A = Ag = 19.1 in^2; bf = 12 in; tf = 0.605 in; tw = 0.39 in; d = 12.1 in; h / tw = 24.9 ; Cw = 5780 in^6; h0 = 11.5 in; rts = 3.38 in;
Zx = 96.8 in^3; Sx = 87.9 in^3; Ix = 533 in^4; rx = 5.28 in; Zy = 44.1 in^3; Sy = 29.1 in^3; Iy = 174 in^4; ry = 3.02 in; J = 2.18 in^4;
Using Direct Design Method; Consider Multiplier B1 for P-δ Effect
Pu = Pr = 15 kips; Mux = Mxr = 180 kip-ft; Muy = Myr = 5 kip-ft; Cmx = 1 ; Cmy = 1 ; Vux = 2.3 kips; Vuy = 1.1 kips;
Fy = 36 ksi; Cb = 1 ; Lb = 12 ft; Kx = 1 ; Ky = 1 ; Kz = 1 ; Lx = 12 ft; Ly = 12 ft; Lz = 12 ft;

Axial Capacity Calculation

b = bf / 2
Unstiffened b / tf = 9.91736

$$0.56\sqrt{\frac{E}{F_y}}$$

= 15.8941

$$\frac{b}{t} \le 0.56\sqrt{\frac{E}{F_y}}$$

$$Q_s = 1.0 \qquad \text{(E7-4)}$$

= 1

Stiffened b / t = h / tw = 24.9

$$\lambda_r = 1.49\sqrt{\frac{E}{F_y}}$$

= 42.2896

The section has non-slender stiffened element
Qa = 1

Compressive strength to account for Flexural Buckling

$$\frac{K_x L_x}{r_x}$$

© 2026 ENERCALC, LLC

= 27.2727

$$\frac{K_y L_y}{r_y}$$

= 47.6821

$$\frac{KL}{r} = \max\left(\frac{K_x L_x}{r_x} , \frac{K_y L_y}{r_y} \right)$$

= 47.6821

$$F_e = \frac{\pi^2 E}{\left(\frac{KL}{r} \right)^2} \qquad \text{(E3-4)}$$

= 125.889 ksi

$$4.71\sqrt{\frac{E}{F_y}}$$

= 133.681

$$\frac{KL}{r} \le 4.71\sqrt{\frac{E}{F_y}}$$

$$F_{cr} = \left[0.658^{\frac{F_y}{F_e}} \right] F_y \qquad \text{(E3-2)}$$

= 31.939 ksi

$$P_n = F_{cr} A_g \qquad \text{(E3-1)}$$

= 610.035 kips

Flexural Buckling Controls: Pn = 610.035 kips

$$\phi_c P_n$$

= 549.031 kips

Moment Magnification Calculation

α = 1.00 (LRFD)

Pr / Py = 0.021815

$\alpha P_r / P_y \le 0.5$

© 2026 ENERCALC, LLC

$\tau_b = 1.0$ (C2-2a)

= 1

Moment magnifier B1 for P-delta effects in local x direction
Pr / Py = 0.021815

$\alpha P_r / P_y \le 0.5$

$\tau_b = 1.0$ (C2-2a)

= 1

$EI^* = 0.8 \tau_b EI$

= 1.23656e+007 ksi

$$P_{e1} = \frac{\pi^2 EI^*}{(K_1 L)^2} \quad \text{(A-8-5)}$$

= 5885.59 kips

$$B_1 = \frac{C_m}{1 - \alpha P_r / P_{e1}} \ge 1 \quad \text{(A-8-3)}$$

= 1.00256

Magnified Mux = Mux * B1 = 180.46 kip-ft

Moment magnifier B1 for P-delta effects in local y direction
Pr / Py = 0.021815

$\alpha P_r / P_y \le 0.5$

$\tau_b = 1.0$ (C2-2a)

= 1

$EI^* = 0.8 \tau_b EI$

= 4.0368e+006 ksi

$$P_{e1} = \frac{\pi^2 EI^*}{(K_1 L)^2} \quad \text{(A-8-5)}$$

= 1921.37 kips

$$B_1 = \frac{C_m}{1 - \alpha P_r / P_{e1}} \ge 1 \quad \text{(A-8-3)}$$

= 1.00787

Magnified Muy = Muy * B1 = 5.03934 kip-ft

© 2026 ENERCALC, LLC

Mrx = Mux; Mry = Muy

Major Flexural Capacity Calculation

Web compactness:

$$\lambda = \frac{h_c}{t_w}$$

= 24.9

$$\lambda_{pw} = 3.76\sqrt{\frac{E}{F_y}}$$

= 106.717

$$\lambda_{rw} = 5.70\sqrt{\frac{E}{F_y}}$$

= 161.779

Web is compact

Flange compactness:

$$\lambda = \frac{b_f}{2t_f}$$

= 9.91736

$$\lambda_{pf} = 0.38\sqrt{\frac{E}{F_y}}$$

= 10.7853

$$\lambda_{rf} = 1.0\sqrt{\frac{E}{F_y}}$$

= 28.3823

Flange is compact

Mnx to account for Yielding

$$M_n = M_p = F_y Z_x \qquad \text{(F2-1)}$$

= 290.4 kip-ft

Mnx to account for Flange Local Buckling

$$\lambda < \lambda_{pf}$$

© 2026 ENERCALC, LLC

$$M_n = M_p$$

= 290.4 kip-ft

Mnx to account for Lateral-Torsional Buckling

$$L_p = 1.76 r_y \sqrt{\frac{E}{F_y}} \qquad \text{(F2-5)}$$

= 12.5715 ft

For I section, c = 1

$$L_r = 1.95 r_{ts} \frac{E}{0.7F_y} \sqrt{\frac{Jc}{S_x h_o} + \sqrt{\left(\frac{Jc}{S_x h_o}\right)^2 + 6.76\left(\frac{0.7F_y}{E}\right)^2}} \qquad \text{(F2-6)}$$

= 45.9286 ft

$$M_n = M_p = F_y Z_x \qquad \text{(F2-1)}$$

= 290.4 kip-ft

Lb < Lp

$$M_n = M_p$$

= 290.4 kip-ft

Therefore Mnx = 290.4 kip-ft

$$M_{cx} = \phi_b M_{nx}$$

= 261.36 kip-ft

Minor Flexural Capacity Calculation

Mny to account for Yielding

$$M_n = M_p = F_y Z_y \le 1.6 F_y S_y \qquad \text{(F6-1)}$$

= 132.3 kip-ft

Mny to account for Lateral-Torsional Buckling

$$\lambda < \lambda_{pf}$$

$$M_n = M_p$$

= 132.3 kip-ft

Therefore Mny = 132.3 kip-ft

$$M_{cy} = \phi_b M_{ny}$$

© 2026 ENERCALC, LLC

= 119.07 kip-ft

Flexural and Axial Interaction Calculation

$$\frac{P_r}{P_c} = \frac{P_u}{\phi_c P_n}$$

= 0.0273209

$$\frac{P_r}{P_c} < 0.2$$

$$\frac{P_r}{2P_c} + \left(\frac{M_{rx}}{M_{cx}} + \frac{M_{ry}}{M_{cy}} \right) \leq 1.0 \qquad \text{(H1-1b)}$$

= 0.746448

Axial-Flexural Strength: OK

Major Shear Capacity Calculation

$$A_w = dt_w$$

$$k_v = 5$$

$$h/t_w$$

= 24.9

$$2.24\sqrt{E/F_y}$$

= 63.5764

$$h/t_w \leq 2.24\sqrt{E/F_y}$$

$$C_v = 1.0 \qquad \text{(G2-2)}$$

$$V_n = 0.6 F_y A_w C_v \qquad \text{(G2-1)}$$

= 101.93 kips

$$h/t_w \leq 2.24\sqrt{E/F_y}$$

$$\phi_v = 1.00$$

$$\phi_v V_n$$

= 101.93 kips

© 2026 ENERCALC, LLC

$$\frac{V_u}{\phi_v V_n}$$

= 0.0225644

Shear Strength (Major Axis): OK

Minor Shear Capacity Calculation

$A_w = 2 b_f t_f$
$k_v = 1.2$
$h/t_w = b/t_f$
= 9.91736

$1.10\sqrt{k_v E / F_y}$
= 34.2004

$1.37\sqrt{k_v E / F_y}$
= 42.595

$h / t_w \le 1.10\sqrt{k_v E / F_y}$
$C_v = 1.0$ (G2-3)
= 1

$V_n = 0.6 F_y A_w C_v$ (G2-1)
= 313.632 kips

$\phi_v = 0.90$
$\phi_v V_n$
= 282.269 kips

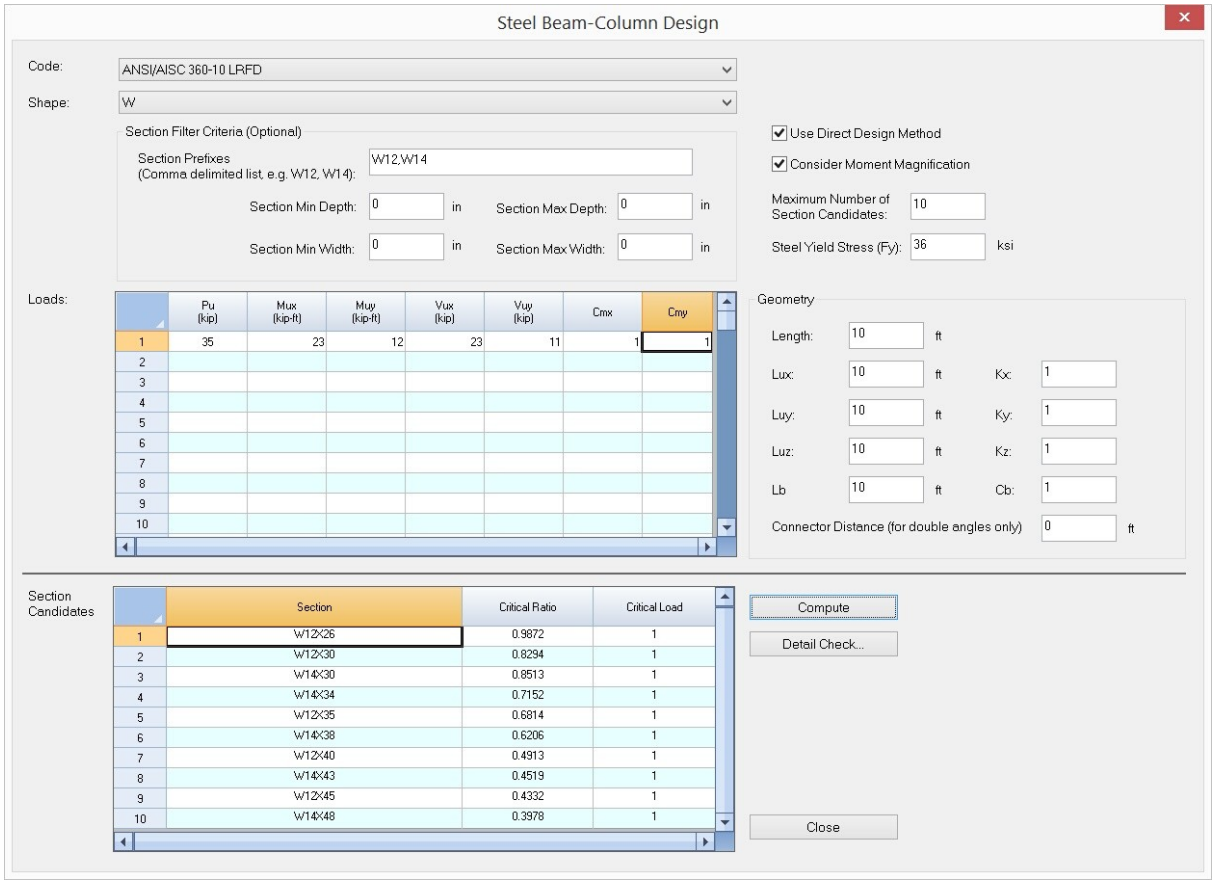
$$\frac{V_u}{\phi_v V_n}$$

= 0.00389699

Shear Strength (Minor Axis): OK

© 2026 ENERCALC, LLC

Steel Design > Steel Design Tools > Steel Section Design allows you to quickly design steel sections against a set of load effects. The Section Design input and output are shown below:



For Section Filter Criteria, you can use either Section Prefixes or section dimension limits (but not both). The section prefixes is a comma delimited list such as W12, W14. If section prefixes is used, the section dimension limits will be ignored. If a section dimension limit is zero, then that limit criteria is ignored.

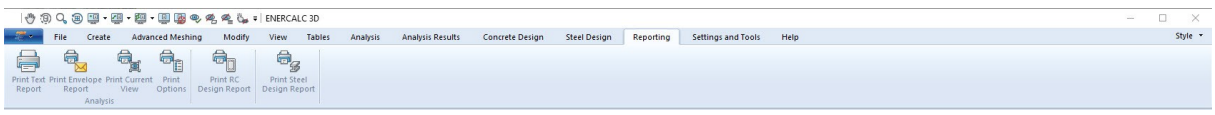
By default, a maximum of ten section candidates will be provided after a successful design. You can then view the detailed check for each of the section candidate.

Unity Check

Steel Design > Unity Check displays a graphical result of the pass/fail status of all designed members. Passing members are highlighted in blue. Failing members are highlighted in red.

Reporting

The Reporting ribbon provides access to tools for creating printed reports.



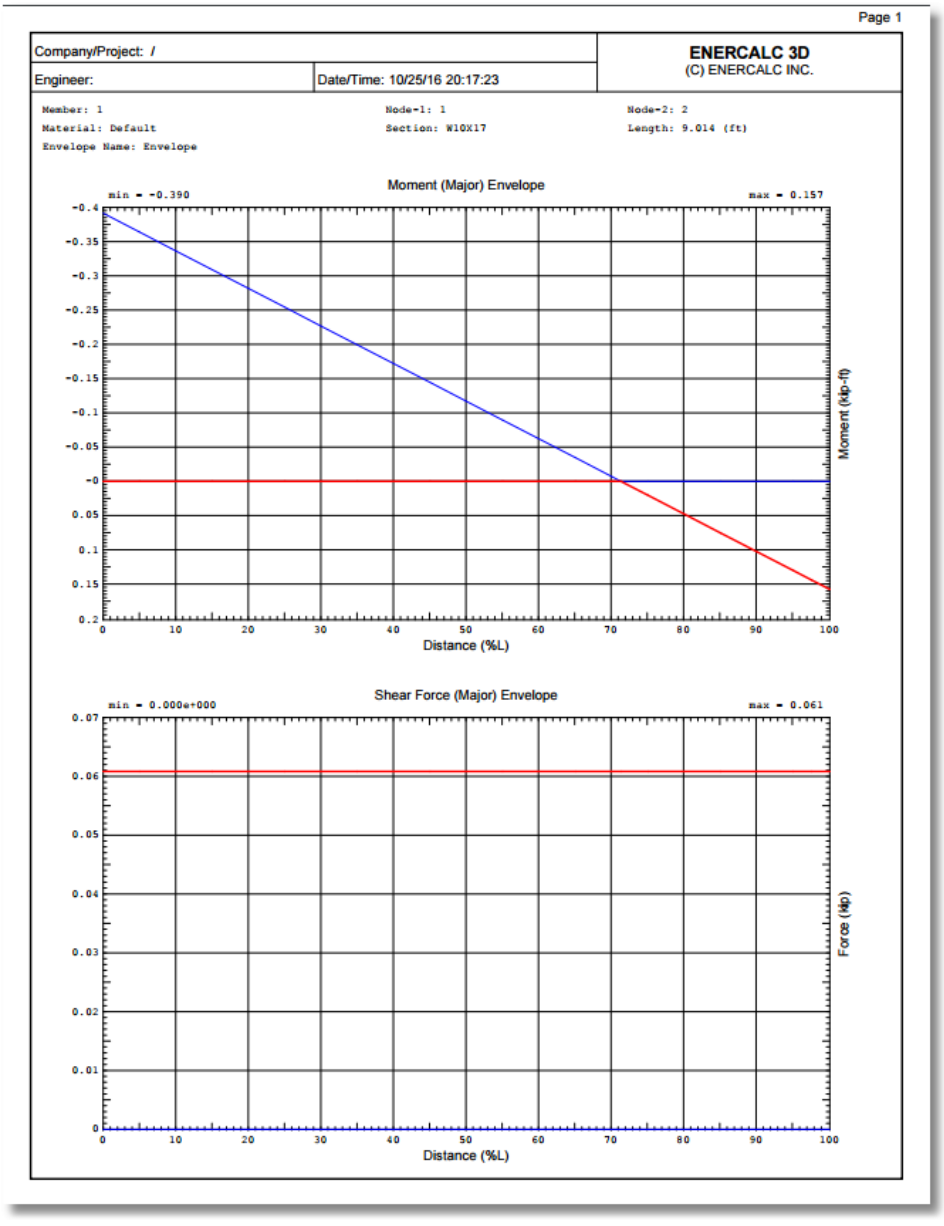
© 2026 ENERCALC, LLC

Print Text Report

Reporting > Print Text Report allows you to preview and print a report for the model using the current report configuration options.

Print Envelope Report

Reporting > Print Envelope Report allows you to print enveloped diagrams of multiple load combinations on members. You may give a name to the type of envelope. You have the option to print an envelope report on selected members only. Two charts are printed on each page. You have the option to select the diagram type for each chart. For example, you may select the major moment Mz diagram for chart one and the major shear Vy diagram for the second chart.



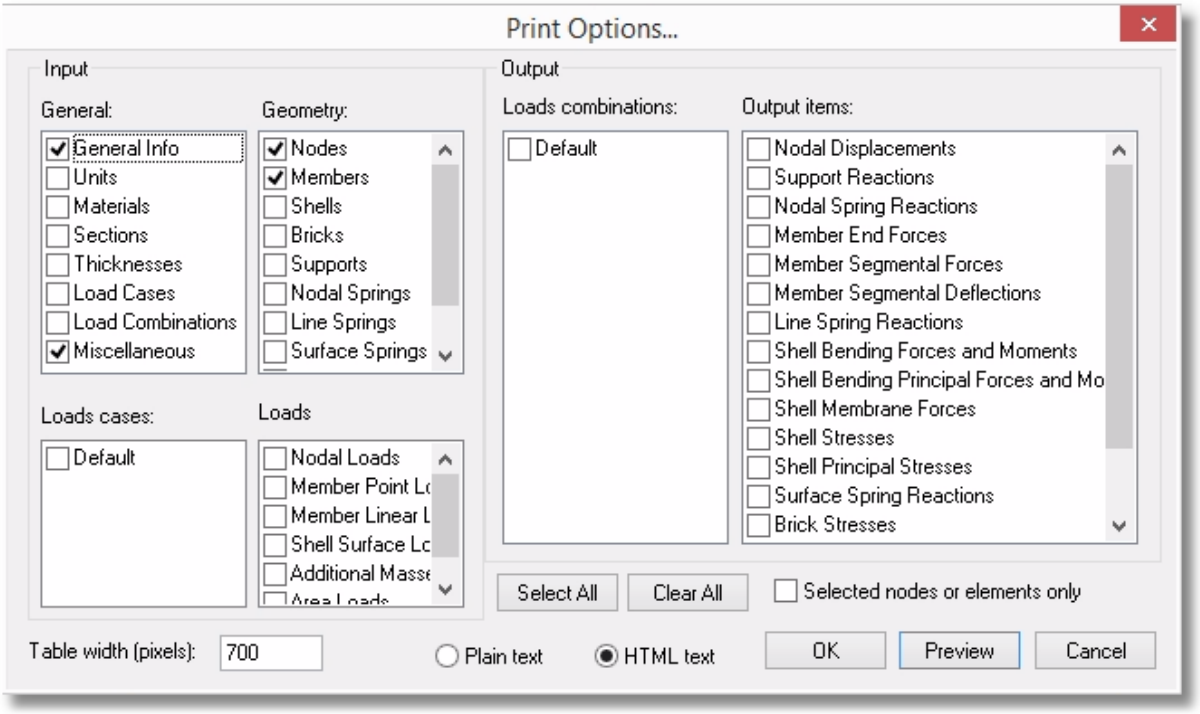
© 2026 ENERCALC, LLC

Print Current View

Reporting > Print Current View allows you to preview and print the current model view.

Print Options

Reporting > Print Options prompts you with the following dialog.



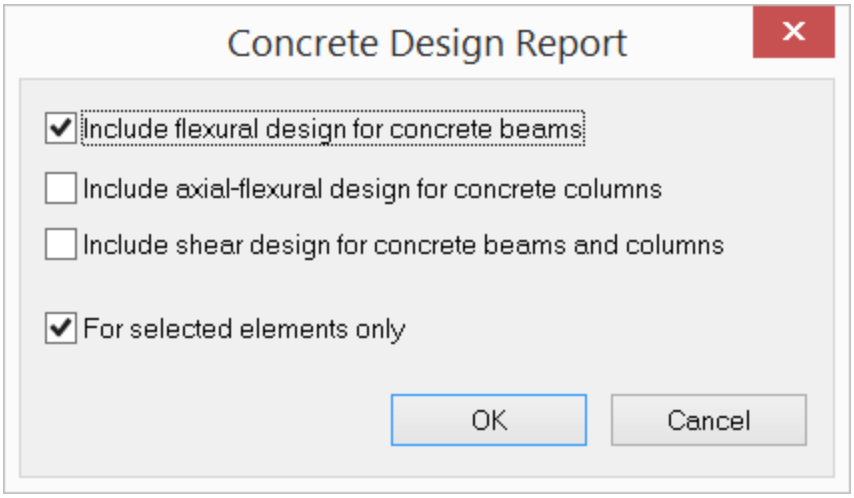
It allows you to generate a report for input and/or output data in html file format.

The command provides different options to control the contents of the report. For example, you may generate a report for selected nodes or elements only. After clicking the OK button, the graphical report will be displayed in a report view within the web browser. You may then print the report to a PDF file.

© 2026 ENERCALC, LLC

Print RC Report

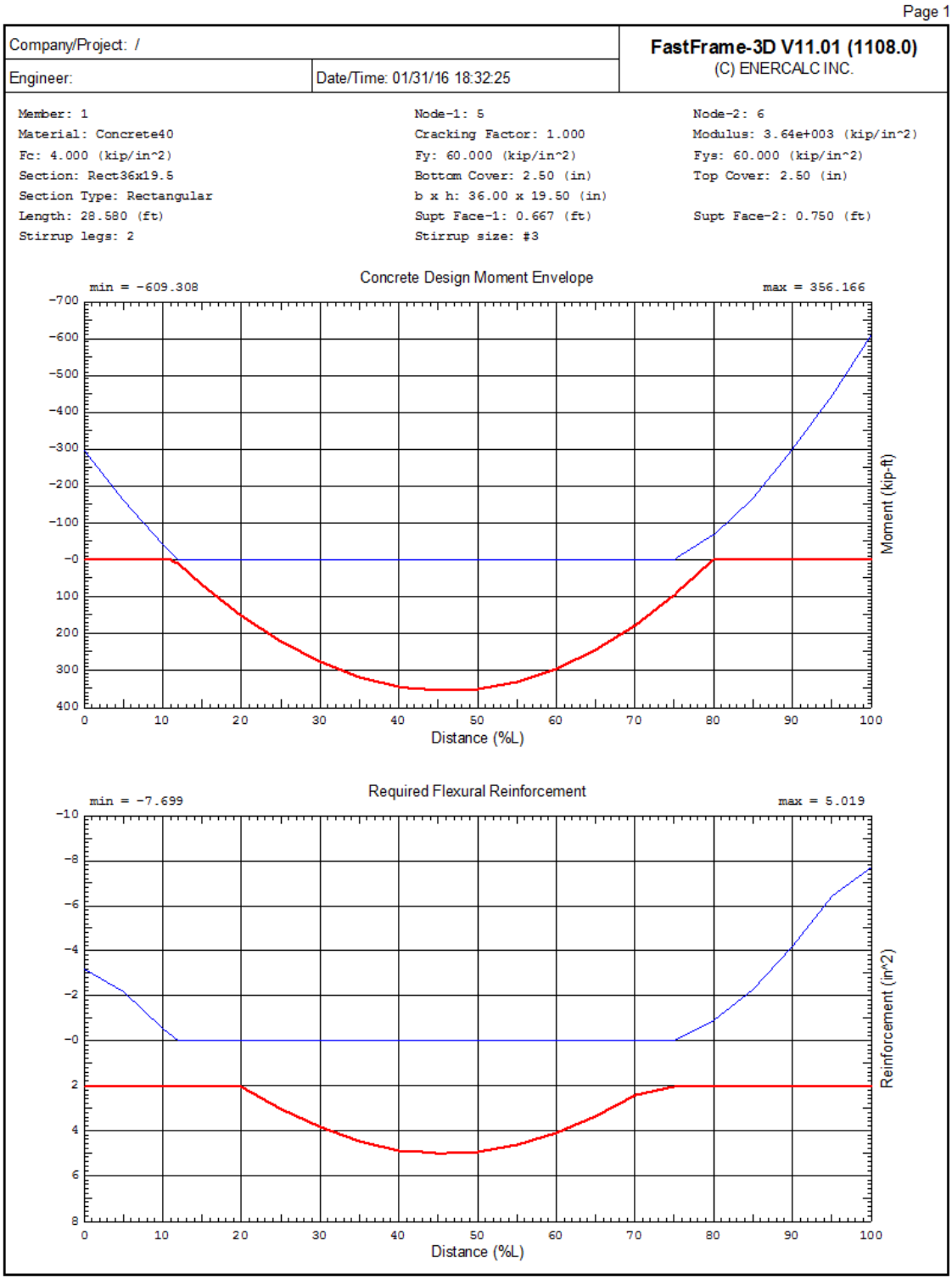
Reporting > Print RC Report displays the following dialog.



It allows you to print concrete design report on beams and columns. You have the options to include flexural design for concrete beams, axial-flexural design for concrete columns as well as shear design for concrete beams and columns. You also have the option to print the report on selected members only.

© 2026 ENERCALC, LLC

The following figure shows the print preview for a column axial-flexural design report.



© 2026 ENERCALC, LLC

Print Steel Design Report

Reporting > Print Steel Design Report is the same command as Design > Steel Design Results.

Settings & Tools

The Settings & Tools ribbon provides commands related to settings for model data and graphical entities in model views. Some of these settings may be applied beyond the current model, that is, they may be saved for use in future models.

New Window

Settings & Tools > New Window creates a new window or view based on the current view. You may create different model views with different display settings with respect to zooming, panning, loading diagram, shear or moment diagrams, contours, etc. For example, you may have one model view to display moment diagram, another view to display shear diagram. You may create as many views as you want. However, too many views may clutter the view area and make graphic display sluggish.

Tile Horizontal

Settings & Tools > Tile Horizontal arranges all opened windows horizontally.

Tile Vertical

Settings & Tools > Tile Vertical arranges all opened windows vertically.

Cascade

Settings & Tools > Cascade arranges all opened windows in an overlapped manner.

Clear Undo & Redo

Settings & Tools > Clear Undo & Redo clears the undo/redo buffer, thus frees up computer memory. It is a good idea to use this command before the solution so that more memory may be committed to the solver.

Clear Results

Settings & Tools > Clear Results clears all results from computer memory. You need to re-solve the model to obtain new results.

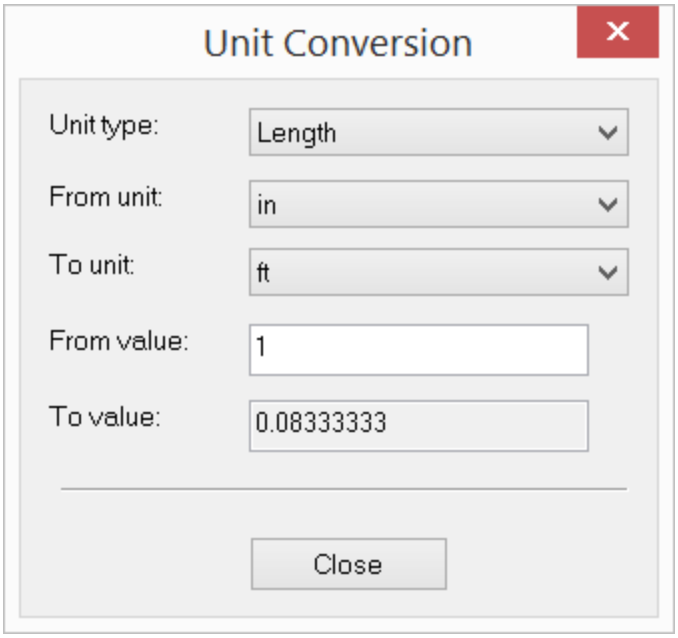
Clear Everything

Settings & Tools > Clear Everything clears all input and output (results) data from computer memory. You should think twice before running this command.

© 2026 ENERCALC, LLC

Unit Conversion

Settings & Tools > Unit Conversion displays a tool for conversion between various units.



Calculator

Settings & Tools > Calculator displays the Windows Calculator.

Text Editor

Settings & Tools > Text Editor displays the Windows Notepad.

Copy Command History

Settings & Tools > Copy Command History copies the history in the command window to the clipboard. You may then paste the command history content to a text editor using Ctrl + V. A command history is associated with each open document.

Clear Command History

Settings & Tools > Clear Command History clears the history in the command window. You may want to copy the command history before running this command.

© 2026 ENERCALC, LLC

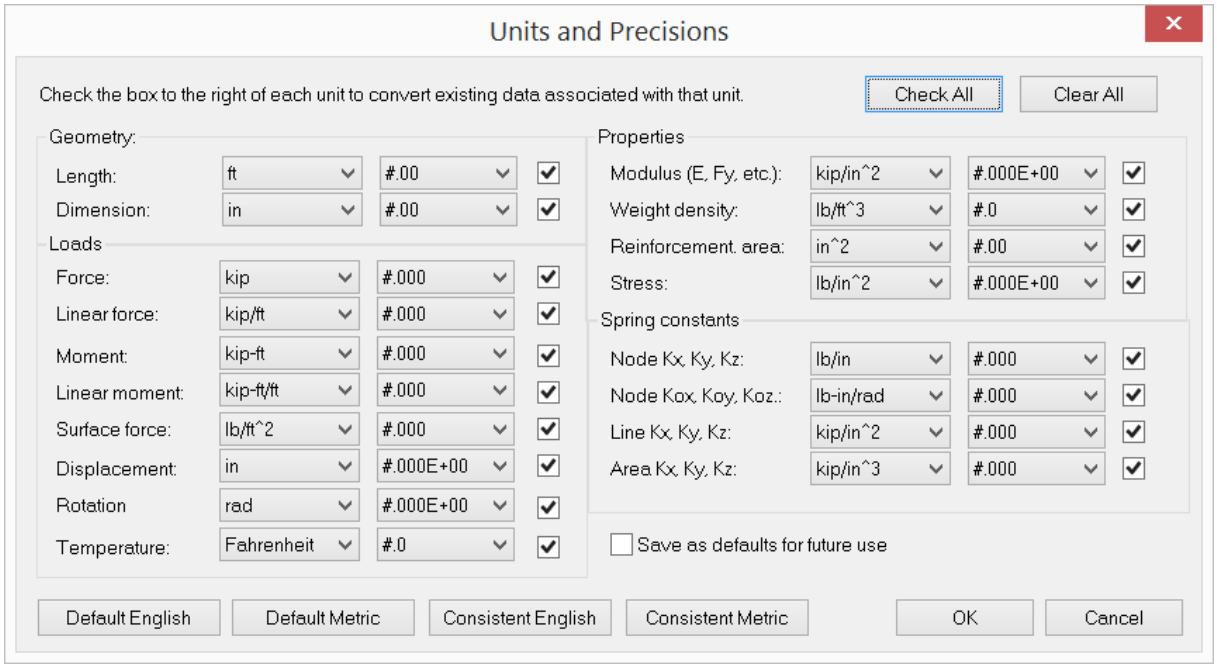
Toolbar

Settings & Tools > Toolbar toggles the display of the Input/Output Data toolbar.

© 2026 ENERCALC, LLC

Units & Precisions

Settings & Tools > Units & Precisions prompts you with the following dialog.



You may select different units and precisions for various physical measurements used in the model. You may use this command as many times as you like. You may convert existing data associated with a unit in the model by checking or unchecking the check box to the right of that unit. For example, if you mistakenly enter all nodal coordinates in a wrong length unit, you may select the correct length unit and uncheck the conversion checkbox to correct nodal coordinate input.

Default English and Default Metric let you quickly set predefined units commonly used for the imperial or metric system. Consistent English and Consistent Metric buttons let you set predefined consistent units for the imperial or metric system. In a consistent unit system, units for the same type of physical measurements are the same. For example, units for both length and dimension are the same, which is inches for imperial system and meters for metric system.

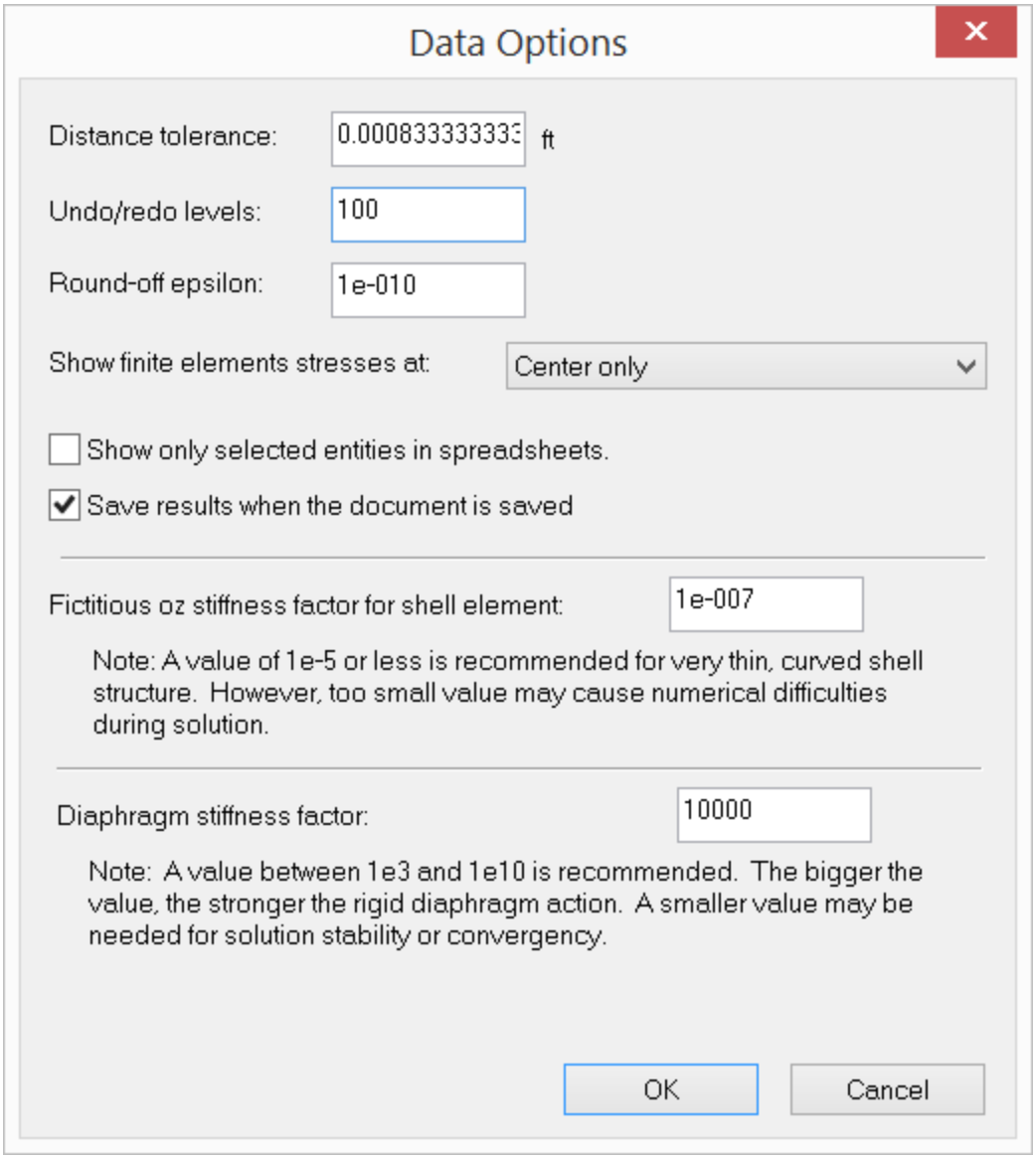
You may set the precision for each unit in either decimal or scientific format. Precision settings are used in displaying data in spreadsheets, diagrams, and reports.

By checking “Save as defaults for future use”, units will be remembered for use in future models. It is a good idea to also save graphic scales at the same time. To do that, just click Settings & Tools > Graphic Scales.

© 2026 ENERCALC, LLC

Data Options

Settings & Tools > Data Options prompts you with the following dialog.



© 2026 ENERCALC, LLC

Distance tolerance is used for distance comparisons in certain commands such as **Modify > Merge All Nodes & Elements** and **Modify > Split Members**. Distances less than distance tolerance are considered zero by the program.

The “Undo/redo levels” sets the maximum undo/redo levels which the program will perform. The program requires extra computer memory for each undo/redo level. The default undo levels setting is 100. Depending on your computer memory and model sizes, you may want to set undo levels to be smaller.

Round-off epsilon is used to truncate floating point numbers such as those found in results. For example, a fixed support may have a displacement of 1.077e-10 when in fact it should zero. A round-off epsilon of 1e-9 will do just that.

Stresses are computed at the center and at the nodes of finite elements such as shells or solids. However, you may request the program to show stresses at the finite element center only, nodes only, or both. The checkbox “Show only selected entities in spreadsheet” determines if all or selected nodes, elements and their dependents will be shown in the spreadsheet. By checking this checkbox, you may easily query selected entities in a large model. It is important to point out that data in some input spreadsheets may not be modified when this option is checked. The checkbox “Save results when the document is saved” gives you the option to save results (when available) to a file when the model input data is saved. The result file is a binary file and has the same file name as model input file, but with an extension of “rst” (static results) or “dyn” (dynamic results). The result file could be much larger than the model input file.

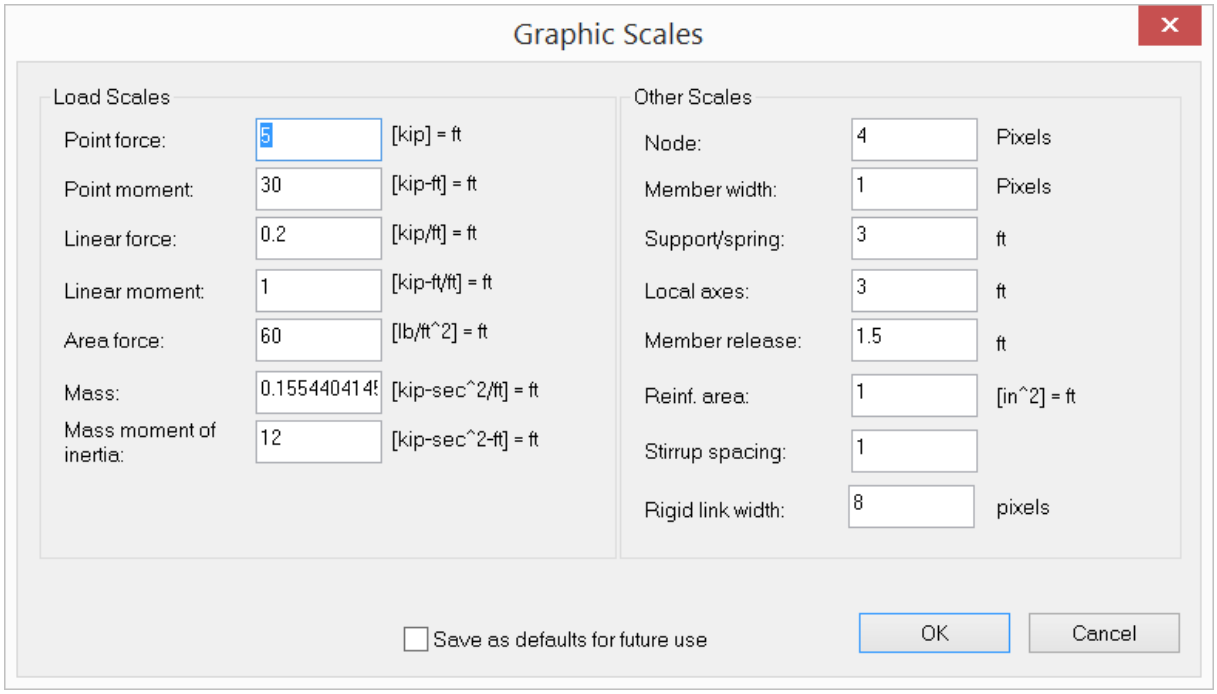
The fictitious oz stiffness factor is used to multiply the minimum of diagonal terms (excluding oz) in the shell stiffness matrix to construct the fictitious oz stiffness terms. The smaller this factor, the more accurate the solution, especially for very thin and doubly curved shells. The valid range for this factor is [1e-12, 1e-3]. You normally do not need to change its default value (1e-7). Numerical difficulties may arise during solution if this value is set too small.

The diaphragm stiffness factor is used to control the diaphragm rigidity. The larger this factor, the more rigid the diaphragm action is. The valid range for this factor is [0, 1e20]. The default value is 1e4. Numerical difficulties may be present during static or frequency analysis if the diaphragm stiffness factor is set too large (say 1e13 for 64-bit floating point solver). It is generally recommended to use 128-bit floating point solver to avoid the aforementioned problem.

© 2026 ENERCALC, LLC

Graphic Scales

Settings & Tools > Graphic Scales prompts you with the following dialog.

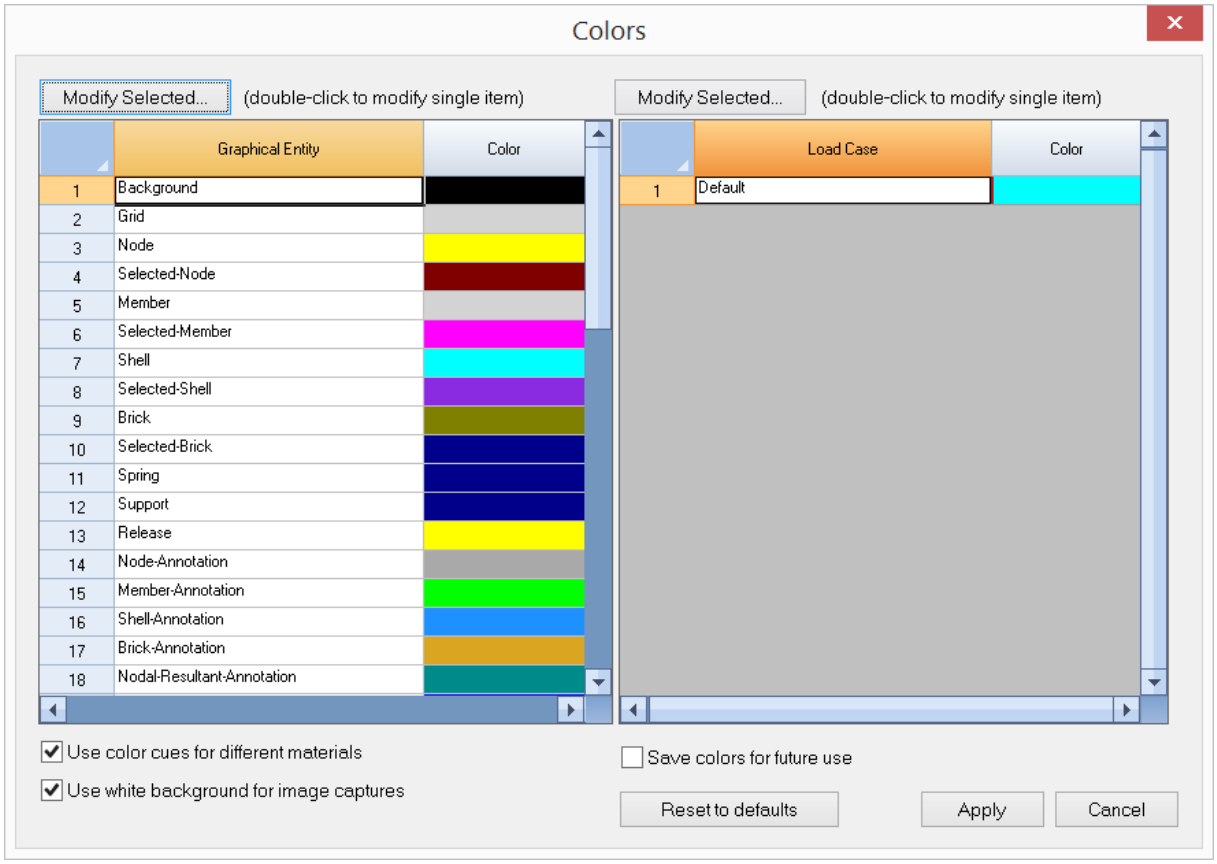


You may set scales for graphical entities such as loads, nodes, supports etc. By “Save as defaults for future use”, these scales will be saved for future use. It is a good idea to save units at the same time. To do that, click Settings & Tools > Units & Precisions.

© 2026 ENERCALC, LLC

Colors

Settings & Tools > Colors prompts you with the following dialog.



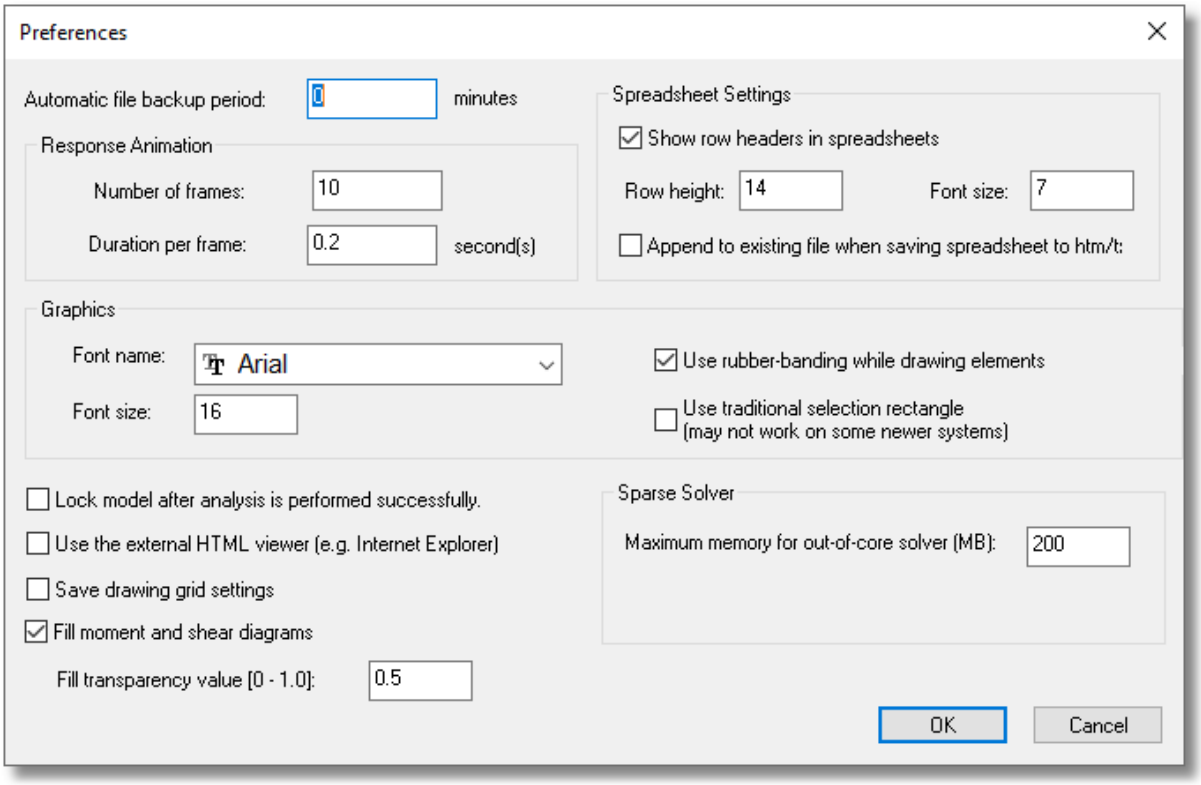
It allows you to set colors of different graphical entities in the model. You may modify the color(s) of one or more items at a time. By checking “Use color cues for different materials”, concrete, steel and wood materials will show different colors in rendering mode.

By checking “Use white background for image captures”, a white background will be used for the captured image even if a different background color is used in the model views. This option will reduce the amount of ink required to print the images. Color settings can be optionally saved for future use.

© 2026 ENERCALC, LLC

Preferences

Settings & Tools > Preferences prompts you with the following dialog.



The “Automatic file backup period” determines how frequently the model files are saved automatically. Enter 0 for no auto-saves.

Settings for “Response Animation” can be set here. You may activate the Response Animation command from the Analysis Results ribbon after an analysis has been performed successfully.

You have the options to lock the model after analysis is performed successfully. By default, an internal HTML viewer is used to view text and graphical reports.

By default, rubber-banding is enabled while drawing beams, shells or bricks. You may want to disable this feature if your computer graphic card is not fully OpenGL compatible.

Additional settings related to the font for graphics and the spreadsheet appearance are available.

When the sparse solver is used for static analysis, you may choose an out-of-core approach so computer memory usage is minimized. You may specify the maximum amount of memory to be used in the out-of-core sparse solver. This value should be smaller than the physical memory available in your system.

Preferences are always saved for future use.

© 2026 ENERCALC, LLC

Help

The Help ribbon provides access to user's manuals, tech support, training materials and reference info.

Online User's Manual

Help > Online User's Manual opens an online version of the ENERCALC 3D User's Manual.

User's Manual PDF

Help > User's Manual PDF opens a PDF version of the ENERCALC 3D User's Manual.

Technical Support

Help > Technical Support opens a form to submit a support request.

Upload Model to Support

Help > Technical Support opens a form to upload a model for a technical support question.

Online Training Manual

Help > Online Training Manual opens an online version of the ENERCALC 3D Training Manual.

Training Manual PDF

Help > Training Manual PDF opens a PDF version of the ENERCALC 3D Training Manual.

Training Videos

Help > Training Manual PDF opens a list of links to training videos.

Verification Manual

Help > Verification Manual provides a link to the verification manual for ENERCALC 3D.

Release Notes

Help > Release Notes provides technical details on the various releases of ENERCALC 3D.

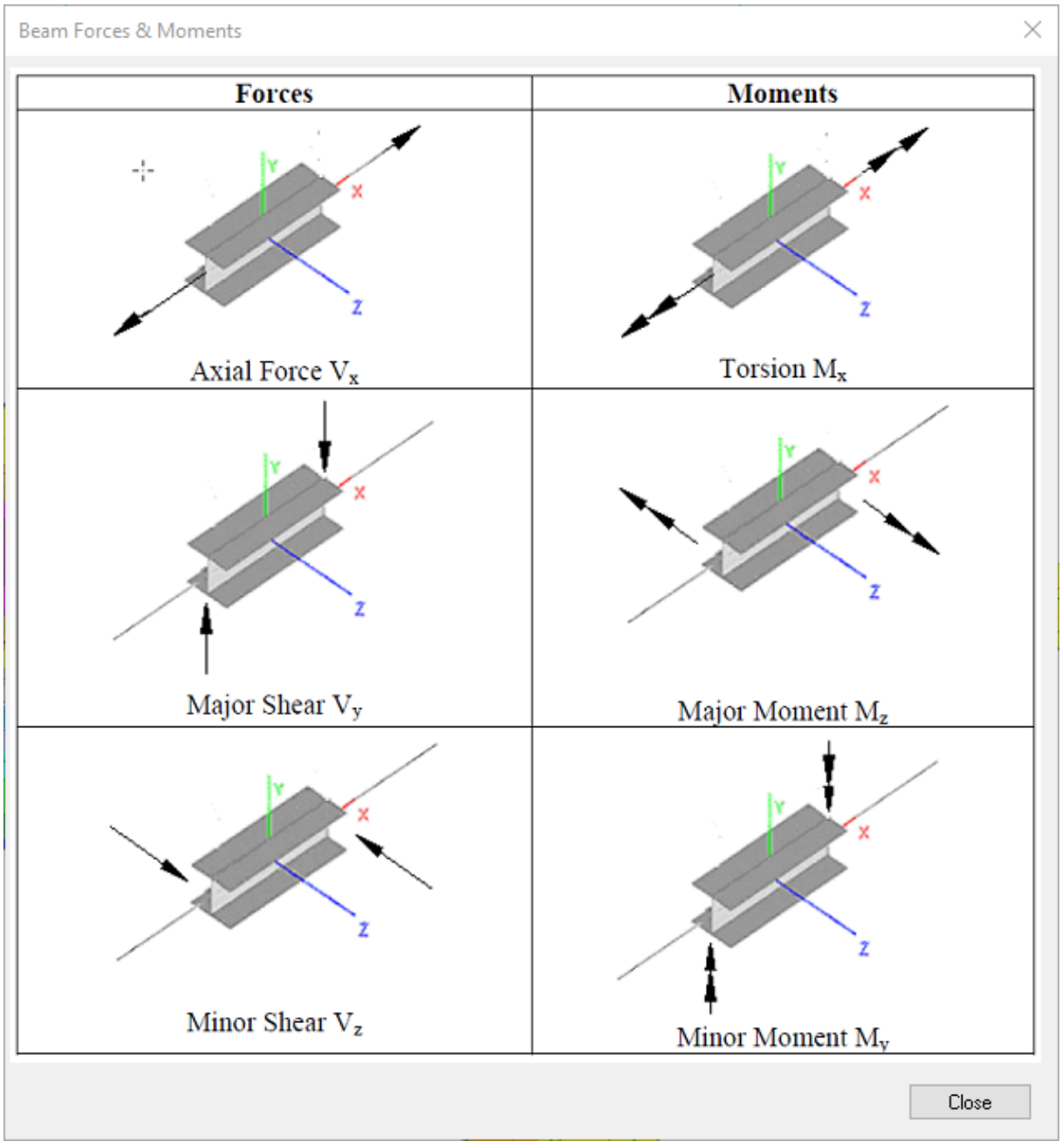
About ENERCALC 3D

Help > About ENERCALC 3D indicates the build number of the current release of ENERCALC 3D.

© 2026 ENERCALC, LLC

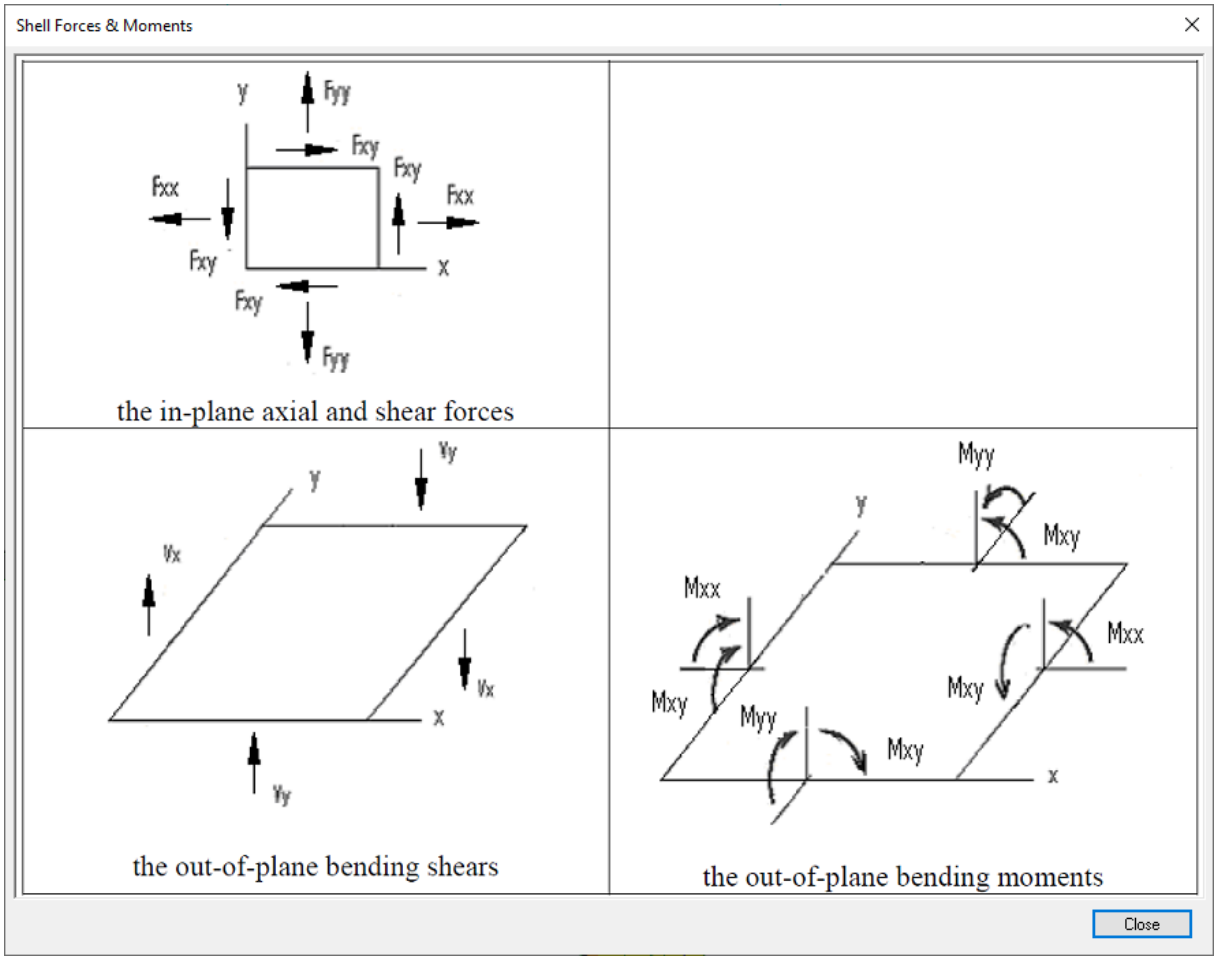
Sign Conventions

Help > Sign Conventions > Beam Forces & Moments displays the beam sign conventions portion of the user's manual.



© 2026 ENERCALC, LLC

Help > Sign Conventions > Shell Forces & Moments displays the shell sign conventions portion of the user's manual.



13.3.1.6 Toolbars

Input/Output Toolbar

Settings > Toolbars > Input/Output Toolbar shows or hides the toolbar that contains input and output buttons. They are an alternate way to open the named tables or forms.

© 2026 ENERCALC, LLC

Status Bar

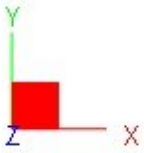
Settings > Toolbars > Status Bar shows or hides the status bar at the bottom of the screen.

1: Currently selected command
2: Type of model
3: Cursor coordinates (when cursor is on a grid point)
4: Current Load Combination
5: Solution status

13.3.1.7 Coordinate Systems

Two kinds of coordinate systems are used in the program, namely, the global coordinate system and the local coordinate system. The global coordinate system is the one and only fixed Cartesian system in a structural model. The local coordinate system applies to each individual member or finite element.

Global Coordinate System



The global coordinate system is a fixed Cartesian system that is used for entire model. The three axes are denoted by capital letters X, Y and Z. They follow the right-hand rule. By default, that is, when a model is not rotated for viewing purpose, the X axis points from left to right (horizontal), the Y axis points from bottom to top (vertical), and the Z axis points from screen to out of screen (perpendicular to screen).

The global coordinate system is used in the following input:
- nodal coordinates, nodal loads
- degrees of freedom related to nodes, supports and springs
- self weights
- point, line, and surface loads on members and finite elements [may also be specified in the element local coordinate system]

The global coordinate system is used in the following output:
- nodal displacements
- support and spring reactions
- brick stresses

Local Coordinate Systems - General

Each member or finite element has a local coordinate system. It is a Cartesian system that has a default orientation (when local angle equals 0) and may be changed at any time. The three axes are denoted by small letters x, y, z. They follow the right-hand rule.

The local coordinate system exists to facilitate input and output for member and finite elements. For example, point or line loads on a member may be most conveniently specified

© 2026 ENERCALC, LLC

in the local coordinate system of the member. The element results such as shears and moments are output in the local coordinate system for design purposes.

Since the local coordinate systems directly affect input and results, it is always prudent to check them for correctness using the commands such as View > Annotate or Render. You may change the local coordinate systems using the commands such as Edit > Element Local Angle or Reverse Node Order for Selected Elements. *It is the directional vectors that matter...the origin of the local coordinate system is insignificant in this program.*

The local coordinate system is used in the following input:

- point, line, and surface loads on members and finite elements [may also be specified in the global coordinate system]
- member moment releases

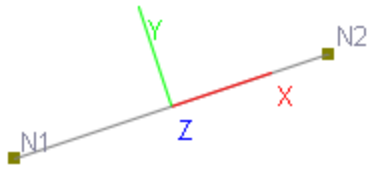
The local coordinate system is used in the following output:

- member forces, moments, and local deflections
- shell forces, moments, and stresses

In the following sections, $\mathbf{V_x}$, $\mathbf{V_y}$ and $\mathbf{V_z}$ (with lowercase subscripts) represent the local x, y, and z vectors respectively. $\mathbf{V_X}$, $\mathbf{V_Y}$ and $\mathbf{V_Z}$ (with uppercase subscripts) represent the global X, Y, and Z vectors respectively. For vector algebra, please refer to relevant math textbooks.

Member Local Coordinate System

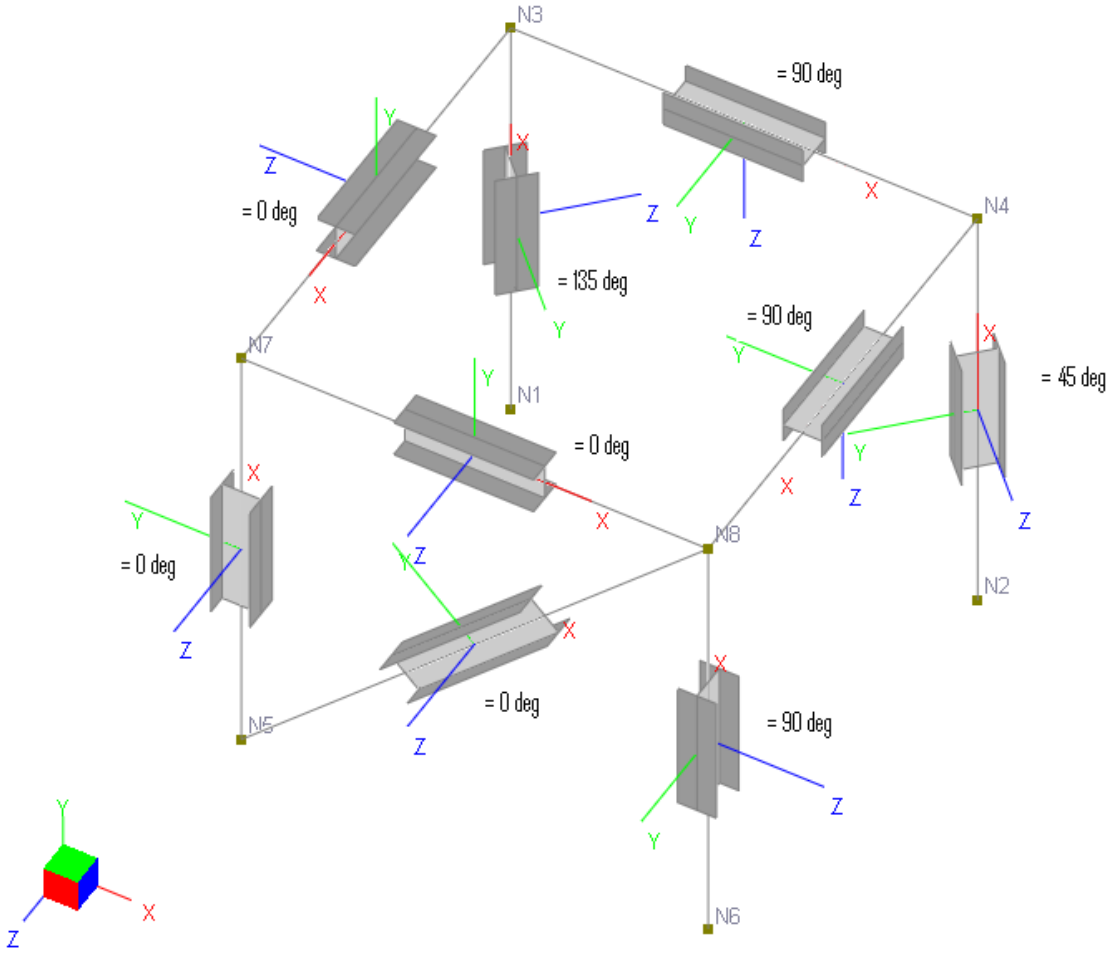
The local coordinate system of a member is determined by the start and end nodes, and an element local angle. The default (local angle equals 0 degrees) local coordinate system of a member is defined using the following procedures:



Steps	Description	Mathematical Notations
A	$\mathbf{V_x}$ points from node 1 (N1) to node 2 (N2)	$\mathbf{V_x}$ = N2 – N1
B1	For vertical members: $\mathbf{V_z}$ is always parallel to $\mathbf{V_Z}$	For vertical members $\mathbf{V_z} = \mathbf{V_Z}$
B2	For non-vertical members: $\mathbf{V_z}$ is perpendicular to a plane formed by $\mathbf{V_x}$ and $\mathbf{V_Y}$	For non-vertical members $\mathbf{V_z} = \mathbf{V_x} \times \mathbf{V_Y}$
C	$\mathbf{V_y}$ is determined based on $\mathbf{V_x}$ and $\mathbf{V_z}$ and the right-hand rule	$\mathbf{V_y} = \mathbf{V_z} \times \mathbf{V_x}$

© 2026 ENERCALC, LLC

For a member with a non-zero local angle (γ), first follow the procedures above that determine the default local coordinate system. Then rotate the default system a γ angle about its local x vector $\mathbf{V_x}$. The rotated $\mathbf{V_x}$, $\mathbf{V_y}$ and $\mathbf{V_z}$ define the local coordinate system. The figure below shows the local coordinate systems of some members with different local angles (γ).

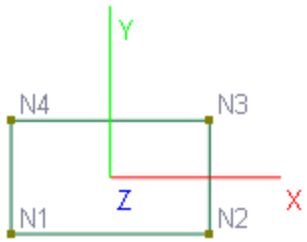
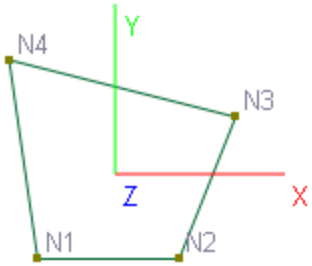


Local coordinate systems for members with different local angles

© 2026 ENERCALC, LLC

Four-Node Shell Local Coordinate System

The local coordinate system of a shell is determined by its four nodes, and an element local angle. The default (local angle equals 0 degrees) local coordinate system of a four-node shell is defined based on the shape of the shell element.

Rectangular Shell	Non-Rectangular Shell
	

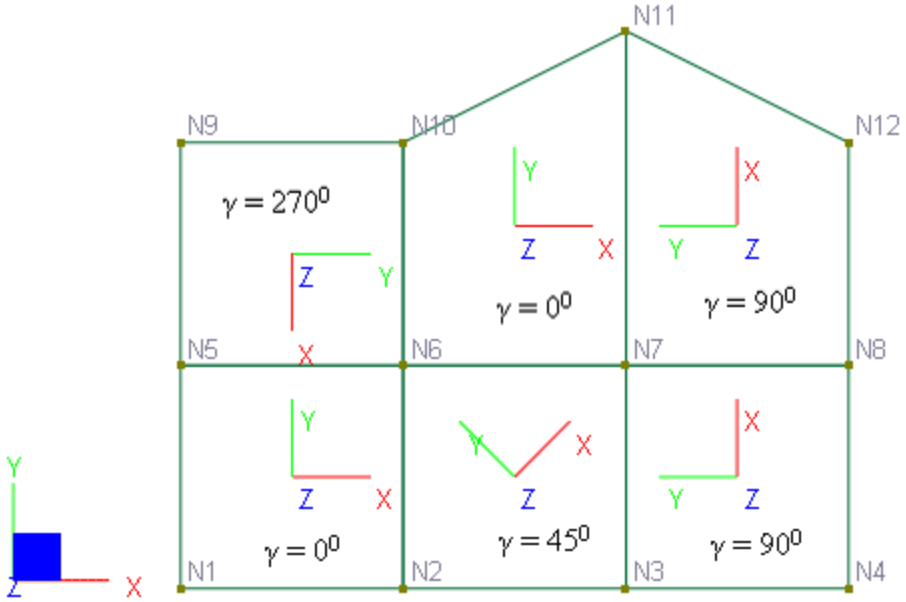
For rectangular shells, the default local coordinate system is easily defined by the following: local x points from N1 to N2, local y points from N1 to N4 and local z is perpendicular to the shell surface.

For non-rectangular shells, the default local coordinate system is defined using the following procedures:

Steps	Description	Mathematical Notations
A	Local z is perpendicular to the shell surface	Let $\mathbf{V_1}$ = N2 – N1, Let $\mathbf{V_2}$ = N4 – N1 $\mathbf{V_z = V_1} \times \mathbf{V_2}$
B1	For horizontal shells that are parallel to global XZ plane, local x is parallel to global X	For horizontal shells $\mathbf{V_x} = \mathbf{V_X}$
B2	For non-horizontal shells, $\mathbf{V_x}$ is perpendicular to a plane formed by $\mathbf{V_Y}$ and $\mathbf{V_z}$	For non-horizontal shells $\mathbf{V_x = V_Y} \times \mathbf{V_z}$
C	$\mathbf{V_y}$ is determined based on $\mathbf{V_x}$ and $\mathbf{V_z}$ and the right-hand rule	$\mathbf{V_y = V_z} \times \mathbf{V_x}$

© 2026 ENERCALC, LLC

For a shell with a non-zero local angle (γ), first follow the procedures above that determine the default local coordinate system. Then rotate the default system a γ angle about is its local z vector $\mathbf{V_z}$. The rotated $\mathbf{V_x}$, $\mathbf{V_y}$ and $\mathbf{V_z}$ define the local coordinate system. The figure below shows the local coordinate systems of some shell elements with different local angles (γ)



Local coordinate systems for shells with different local angles

Eight-Node Brick Local Coordinate System

The local coordinate system for a brick element is always identical to the global coordinate system. It is fixed and cannot be changed.

13.3.1.8 Nodes

Nodes are numbered points in space. They are used to define the geometry and connectivity of all members and finite elements in a model. For members, a node is sometimes referred to as a joint, which has a physical meaning of the intersection of two members such as a beam and a column. However, in this program, the term “node” is generally preferred because it carries a more general meaning.

Nodal Coordinates

The location of a node is defined by the global X, Y, and Z coordinates. Since each member or finite element connects to two or more nodes, nodal coordinates define the geometry of a model. For example, when you move an element, you actually move the locations of the nodes connected to that element.

Degrees of Freedom (DOFs)

Each node may have a maximum of six global degrees of freedom (DOFs) associated with it. They are three translational DOFs along the global X, Y, Z directions (D_x, D_y and D_z) and three rotational DOFs about the global X, Y, Z directions (D_{ox}, D_{oy} and D_{oz}).

© 2026 ENERCALC, LLC

Some of these DOFs may not be available depending upon the type of a model.

For example, the model type “2D Truss” has D_x and D_y available and D_z, D_{ox}, D_{oy}, D_{oz} unavailable or suppressed; while the model type “2D Plate Bending” only has D_z, D_{ox}, D_{oy} available and D_x, D_y, D_{oz} suppressed.

You may always use the model type “3D Frame and Shell” to analyze any structure, however, time and computer memory may be wasted if a simpler model type can be used instead. You may choose the appropriate model type by command Analysis > Analysis Options.

Six nodal displacements associated with 6 DOFs are output for each node. For restrained or unavailable DOFs, the program outputs the corresponding displacements as zero. Nodal displacements should be the first thing to check for when determining result correctness since the solution is displacement-based. If the displacements are wrong, nothing else will be correct.

Node Numbers

A distinct integral number is assigned to each node. Node numbers are used to define the connectivity of member and finite elements. Duplicate numbers in nodes are not permitted. There can be gaps in node numbering sequence. The program will automatically renumber the nodes internally before performing the solution. The order of node numbering in a model is insignificant to the final results, but it may affect the time and computer memory required to solve the model. For a very large model, node renumbering may be important in order to reduce the half band width in the global stiffness matrix and therefore the solution time. You may renumber the nodes sequentially based on nodal coordinates using the command Edit > Renumber Nodes.

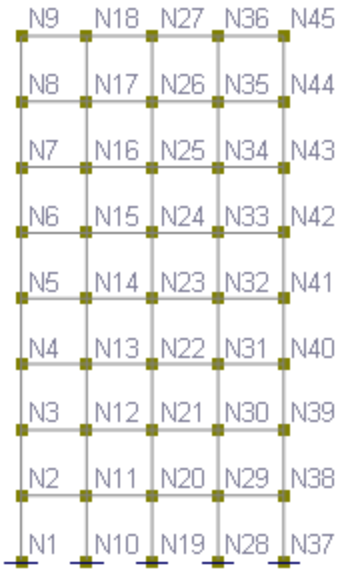
Half Band Width (HBW) is defined as [Ref.7] as follows:

$$HBW = \underbrace{\max}_{1 \le el \le m} (\max dof_{el} - \min dof_{el})$$

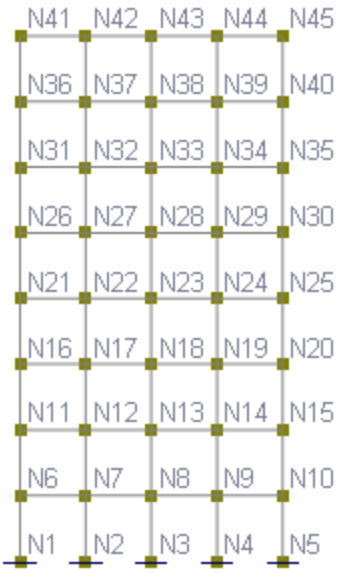
where m is the total number of structural elements, and the max dof_{el} and min dof_{el} are the maximum and minimum global degrees of freedom numbers associated with element el.

© 2026 ENERCALC, LLC

For example, in the screen captures below, models A and B are identical 3D frames (6 DOFs per node) but with different node numbering schemes. Model A has a HBW of 6 * (9 + 1) = 60 while model B has a HBW of 6 * (5 + 1) = 36. Therefore, model B is more economical than model A because of the reduction of half band width.



Model A: HBW=60



Model B: HBW=36

Loads

Forces or moments may be applied to a node. These forces and moments are specified in the global coordinate system. You may regard enforced displacements as special kinds of loads. They are specified in supports.

Supports

By default, a node is unrestrained, that is, it is free to move in any of the six available DOFs. However, for a model to be stable, restraints on one or more DOFs must be imposed on some nodes. Restraints may be rigid or flexible. Rigid restraints are referred to as supports while flexible restraints are referred to as springs. You may regard supports as springs with infinite spring constants.

You may assign a support with one or more DOFs (D_x, D_y, D_z, D_{ox}, D_{oy} and D_{oz}) restrained to a node. The program uses a six-character code to represent restraint conditions of a support in six DOFs. For example, “111111” represents a fixed a support while “111000” represents a pinned support. By default, restrained DOFs have zero enforced displacements. You may specify non-zero enforced displacements to any or all of restrained DOFs. The enforced displacements are discarded if they are assigned to unrestrained DOFs. You may regard these enforced displacements as special kinds of loads. They participate in all load combinations but always with a load factor of 1.0.

The forces or moments required to enforce rigid restraints are called support reactions. They are computed by the program.

© 2026 ENERCALC, LLC

Springs

Springs are flexible restraints. Springs applied to nodes are referred to as nodal springs. You may assign a nodal spring to a node with one or more global DOFs (D_x, D_y, D_z, D_{ox}, D_{oy} and D_{oz}) restrained. To qualify to be a valid flexible restraint, the corresponding spring constant must be specified. A restraint may be designated as linear, compression-only or tension only. A compression-only restraint is active only when the nodal displacement in the restrained direction is negative. A tension-only restraint is active only when the nodal displacement in the restrained direction is positive. If a model contains one or more compression-only or tension-only springs, the whole problem becomes nonlinear and the solution becomes iterative for each load combination.

The forces or moments required to enforce the flexible restraints are called spring reactions. They are computed by the program.

13.3.1.9 Members

A member is a two-node straight frame element with a constant cross section. The term “frame element”, “beam element”, and “member” are used interchangeably in this program. The truss element is a special frame element with moments fully released at both ends. The frame element formulation accounts for axial, torsional, and bending about strong and weak axes, with options to include shear deformations and axial stress stiffening (P-Delta) effects. Moment releases may be applied to either or both ends of the element.

The frame element may be used to model continuous beams, 2D or 3D frames, 2D or 3D trusses or a mixture of two. The program provides powerful commands to generate commonly used framed structures such as continuous beams, 2D or 3D frames, arc beams, and non-prismatic beams. A non-prismatic member is approximated by subdividing the original member into several prismatic members. You may access these commands from the Create ribbon.

Member Sections

Each member must have a section assigned to it. The section properties include:

- A: axial section area
- A_y: shear area along the member local y direction
- A_z: shear area along member the local z direction
- I_{zz}: moment of inertia about strong the local axis z
- I_{yy}: moment of inertia about weak the local axis y
- J: torsional moment of inertia

A_y and A_z may be zero, in which case, the program ignores shear deformations of the element. Mathematically speaking, the program interprets them as being infinite. For rectangular sections, $A_y = A_z = 5/6A$. For solid circular sections, $A_y = A_z = 0.9A$. For thin-walled hollow circular sections, $A_y = A_z = 0.5A$. For wide flange sections, A_y = web area, A_z = area of two flanges [Ref. 6]. To consider member shear deformation, you must choose the proper option from the command Analysis > Analysis Options. *Shear deformation, when considered, applies to both element stiffness and local deflections.*

© 2026 ENERCALC, LLC

Local Coordinate System

Each member has its own local coordinate system. The element local coordinate systems are used in element stiffness formulations. They are also used for inputs such as loads and releases and outputs such as internal shears and moments. For the definition of the member local coordinate system, refer to Coordinate Systems.

Member Numbers

A distinct integral number is assigned to each member. Duplicate numbers in members are not permitted. There can be gaps in the member numbering sequence. The order of member numbering in a model is insignificant to the results or solution time. You may renumber the members sequentially using the command Edit > Renumber Members.

Beams vs. Trusses

By default, a member or frame element is a beam. However, if you choose the model type to be either "2D Truss" or "3D Truss", then the frame element becomes a truss element. The program assigns full moment releases automatically to the ends of all members and suppresses all three rotational DOFs D_{ox}, D_{oy}, D_{oz} for each and every node. For the model type "2D Truss", the program also suppresses translational DOF D_z. Generally speaking, if a model contains only 2D or 3D truss elements, you should choose the model type as "2D Truss" or "3D Truss". If a model contains both trusses and beams, you should choose the model type"2D Frame" or "3D Frame & Shell", and assign appropriate moment releases to individual beams. It may also be necessary to assign appropriate restraints to nodes to ensure stability of the model. You may choose the appropriate model type by running the command Analysis > Analysis Options.

Elastic Stiffness Matrix

Total number of DOFs of a member is the summation of DOFs of the two nodes. Therefore, for a 3D beam, the stiffness matrix is of size 12 x 12. The elastic stiffness matrix in the local coordinate system with shear deformation is given [Ref. 8] as follows:

$$
\begin{bmatrix} F_{x1} \\ F_{y1} \\ F_{z1} \\ M_{x1} \\ M_{y1} \\ M_{z1} \\ F_{x2} \\ F_{y2} \\ F_{z2} \\ M_{x2} \\ M_{y2} \\ M_{z2} \end{bmatrix} =
\begin{bmatrix}
EA/L & 0 & 0 & 0 & 0 & 0 & -EA/L & 0 & 0 & 0 & 0 & 0 \\
 & 12\beta_3/L^2 & 0 & 0 & 0 & 6\beta_3/L & 0 & -12\beta_3/L^2 & 0 & 0 & 0 & 6\beta_3/L \\
 & & 12\beta_2/L^2 & 0 & -6\beta_2/L & 0 & 0 & 0 & -12\beta_2/L^2 & 0 & -6\beta_2/L & 0 \\
 & & & J/(GL) & 0 & 0 & 0 & 0 & 0 & -J/(GL) & 0 & 0 \\
 & & & & (4+\alpha_2)\beta_2 & 0 & 0 & 0 & 6\beta_2/L & 0 & (2-\alpha_2)\beta_2 & 0 \\
 & & & & & (4+\alpha_1)\beta_3 & 0 & -6\beta_3/L & 0 & 0 & 0 & (2-\alpha_1)\beta_3 \\
 & & & & & & EA/L & 0 & 0 & 0 & 0 & 0 \\
 & & & & & & & 12\beta_3/L^2 & 0 & 0 & 0 & -6\beta_3/L \\
 & & & & & & & & 12\beta_2/L^2 & 0 & 6\beta_2/L & 0 \\
 & & & & & & & & & J/(GL) & 0 & 0 \\
 & & & & & & & & & & (4+\alpha_2)\beta_2 & 0 \\
 & & & & & & & & & & & (4+\alpha_1)\beta_3
\end{bmatrix}
\begin{bmatrix} \Delta_{x1} \\ \Delta_{y1} \\ \Delta_{z1} \\ \theta_{x1} \\ \theta_{y1} \\ \theta_{z1} \\ \Delta_{x2} \\ \Delta_{y2} \\ \Delta_{z2} \\ \theta_{x2} \\ \theta_{y2} \\ \theta_{z2} \end{bmatrix}
$$

where: $G = \frac{E}{2(1+\nu)}$; $\alpha_1 = \frac{12EI_z}{GA_yL^2}$; $\alpha_2 = \frac{12EI_y}{GA_zL^2}$; $\beta_1 = \frac{EI_z}{(1+\alpha_1)L}$; $\beta_2 = \frac{EI_y}{(1+\alpha_2)L}$

© 2026 ENERCALC, LLC

Geometric Stiffness Matrix

When a tensile axial force is present in a member, the bending stiffness of that member is increased. Conversely, when a compressive axial force is present in a member, the bending stiffness of that member is reduced. The stiffness matrix that reflects this kind of stress stiffening effect is called the geometric stiffness matrix [Ref. 3, 7]. It is determined by the element geometry and stress conditions, and is independent of the elastic properties. The geometric stiffness matrix is very effective in accounting for the P-Delta effect and is implemented in the program. It may also be used to perform buckling analysis of the structure but is not implemented in the program directly.

Like the elastic stiffness matrix, the geometric stiffness matrix is of size 12 x 12 and is given [Ref. 3, 7] as follows:

$$\begin{bmatrix} F_{x1} \\ F_{y1} \\ F_{z1} \\ M_{x1} \\ M_{y1} \\ M_{z1} \\ F_{x2} \\ F_{y2} \\ F_{z2} \\ M_{x2} \\ M_{y2} \\ M_{z2} \end{bmatrix} = \frac{P}{L} \begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & -1 & 0 & 0 & 0 & 0 & 0 \\ & 6/5 & 0 & 0 & 0 & L/10 & 0 & -6/5 & 0 & 0 & 0 & L/10 \\ & & 6/5 & 0 & -L/10 & 0 & 0 & 0 & -6/5 & 0 & -L/10 & 0 \\ & & & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ & & & & 2L^2/15 & 0 & 0 & 0 & L/10 & 0 & -L^2/30 & 0 \\ & & & & & 2L^2/15 & 0 & -L/10 & 0 & 0 & 0 & -L^2/30 \\ & & & & & & 1 & 0 & 0 & 0 & 0 & 0 \\ & & & & & & & 6/5 & 0 & 0 & 0 & -L/10 \\ & & & & & & & & 6/5 & 0 & L/10 & 0 \\ & & & & & & & & & 0 & 0 & 0 \\ & & & & & & & & & & 2L^2/15 & 0 \\ & & & & & & & & & & & 2L^2/15 \end{bmatrix} \begin{bmatrix} \Delta_{x1} \\ \Delta_{y1} \\ \Delta_{z1} \\ \theta_{x1} \\ \theta_{y1} \\ \theta_{z1} \\ \Delta_{x2} \\ \Delta_{y2} \\ \Delta_{z2} \\ \theta_{x2} \\ \theta_{y2} \\ \theta_{z2} \end{bmatrix}$$

where P is the average of the axial forces (positive in tension, negative in compression) at the member ends.

When the linear static (first order) analysis is chosen, the member stiffness matrix is the elastic stiffness matrix. When the P-Delta (second order) analysis option is chosen, the member stiffness matrix is the summation of the elastic stiffness matrix and the geometric stiffness matrix. You may set the appropriate analysis option with the command Analysis > Analysis Options 414.

Moment Releases

By default, a member is rigidly connected to two end nodes. You may however assign moment releases to either end of the member. *It is important to note that the releases are applied with respect to the member local coordinate system.* The moment releases may be in major bending direction (D_{oz}) or minor bending direction (D_{oy}) or both. The element stiffness matrix is modified to enforce moment releases.

© 2026 ENERCALC, LLC

Tension/Compression-Only

By default, a member is linear. You may assign nonlinearity (tension-only or compression-only) to the selected members. The member stiffness will be ignored if a tension-only member is subjected to compressive forces or if a compression-only member is subjected to tensile forces. The presence of tension- or compression-only members makes the model nonlinear, so an iterative solution is required for each load combination.

Rigid Links

A rigid link is a member that has very large sectional properties (A, Ay, Az, Iz, Iy and J). There can only be one rigid link section defined in the model and it must be named as "RIGID_LINK". The properties for the RIGID_LINK section must be set to 0's on the member section dialog. The program will appropriately calculate A, Ay, Az, Iz, Iy and J during the solution process. **Self weight for rigid links will be ignored by the program.**

Rigid Diaphragms

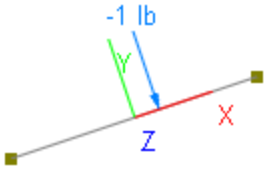
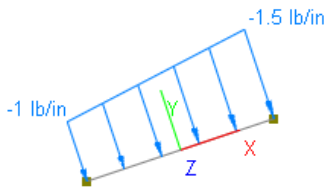
Rigid diaphragms may be used instead of plate finite elements to model stiff in-plane actions such as concrete floors. Internally, the program creates multiple in-plane rigid links for each diaphragm prior to static or frequency analysis. A rigid link is simply a member with very large sectional properties that can be adjusted with the diaphragm stiffness factor (see Settings > Data Options). The larger the diaphragm stiffness factor, the stronger the in-plane rigid diaphragm action is. The presence of rigid links with large diaphragm stiffness factor (say 1E10) could create numerical difficulties during the solution if the 64-bit floating point solver is used. However, the unique 128-bit floating point solver in ENERCALC 3D makes this problem nonexistent in that much larger diaphragm stiffness factor (say 1E20) may be used without creating numerical difficulties during the solution.

The program further provides the option to ignore the rigid diaphragm actions as an analysis option (Analysis > Analysis Options).

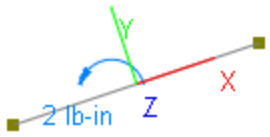
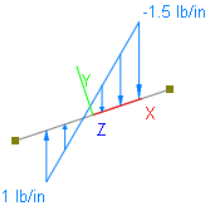
It is important to point out that rigid diaphragm action in the program does <u>not</u> use Equal Displacement Constraints.

Loads

Point loads or line loads may be applied to a member. Point loads may be forces or moments. Line loads must be forces. You may specify loads in either the global or local coordinate system. The locations of loads must be in ratios of the member length, measured from the start of the member. The following figure shows examples of point and line loads.

Point Loads	Line Loads
	
local y force, distance=0.6	local y force, start distance=0,end distance=1

© 2026 ENERCALC, LLC

Point Loads	Line Loads
	
local z moment, distance=0.4	global Y force, stat distance=0.2, end distance=0.8

The self weight of members may be calculated automatically if the material weight densities and self weight multiplier are nonzero. By default, self weight acts in the negative global Y direction. You may however change the direction to positive or negative direction of the global X, Y, or Z. This flexibility is useful in some circumstances. For example, if you model a grillage on the XY plane, the self weight may be either in the positive or negative global Z direction, depending on your preference on load sign convention. To activate automatic self weight calculation, use the command Loads > Assign Self Weights.

An area load may be applied to multiple members on a whole planar area. The area is defined by three or four coplanar nodes. The area load is then distributed as line loads to perimeter members of enclosed sub-areas within the load area prior to static or dynamic solution. Area loads are distributed to perimeter members that form each of the enclosed sub-areas according to the following methods:

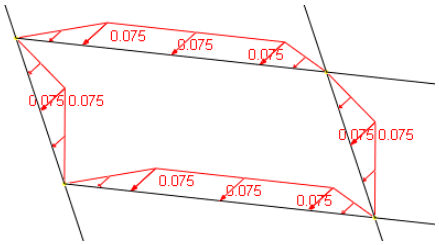
- Two-way (rectangular sub-areas)
- Short-Sides (rectangular sub-areas)
- Long-Sides (rectangular sub-areas)
- AB-CD Sides (rectangular sub-areas)
- BC-AD Sides (rectangular sub-areas)
- Centroid-based
- Circumference-based

The first five distribution methods apply to four-node rectangular sub-areas only. The centroid-based method may be applied to convex sub-areas only. The circumference-based method may be applied to both convex and concave sub-areas. Loads may also be distributed to sides parallel to AB-CD or BC-AD sides of the load area. The program is intelligent enough to determine the most appropriate load distribution if inconsistencies arises. For example, if you select two-way distribution method for a sub-area that is not rectangular, the program will use the centroid-based method if the sub-area is convex or the circumference-based method if the sub-area is concave.

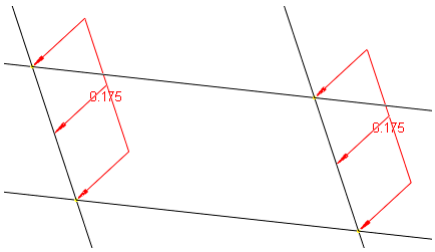
The program allows you to convert area loads to line loads directly and automatically. This feature allows you to see how exactly the program would distribute area loads to members prior to the solution. Of course, you can always undo the conversion if you want to keep the area loads. For more information on the load conversion, please see Loads > Assign Area Loads.

© 2026 ENERCALC, LLC

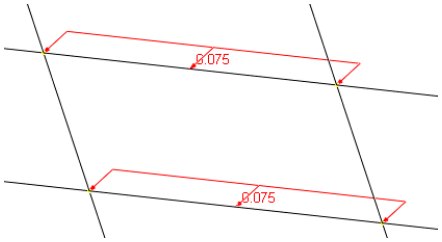
As an example, let's say we have a 3.5 x 1.5 ft rectangular sub-area subjected to 100 lb/ft^2. The following line loads are converted from the same area load based on different distribution methods.



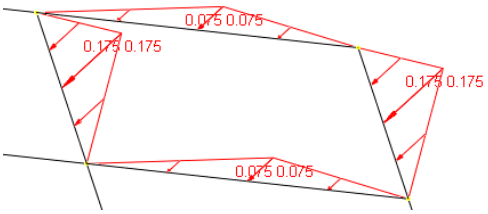
Two-way Distribution



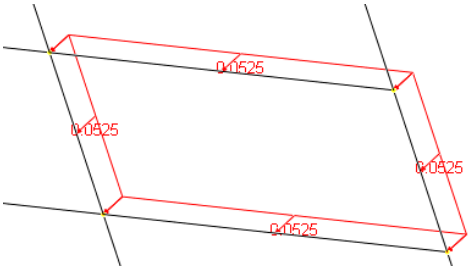
Short-side Distribution



Long-side Distribution



Centroid-based Distribution

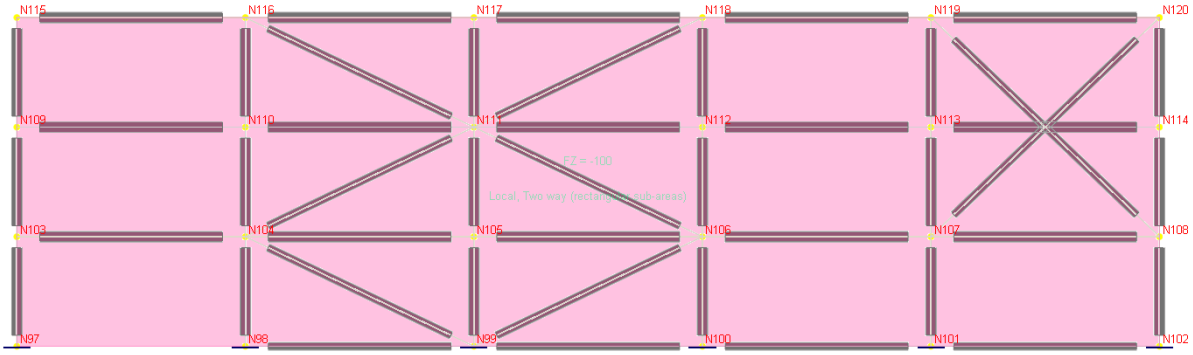


Circumference-based Distribution

© 2026 ENERCALC, LLC

Area loads may be specified in either the local or global coordinate system. Global area loads may be in the global X, Y, or Z direction. Local area loads may only be in the local z direction, which is perpendicular to the load area. It is recommended that area loads be defined in their own load cases. In this way, you will find it easier to identify, edit, and delete area loads later on.

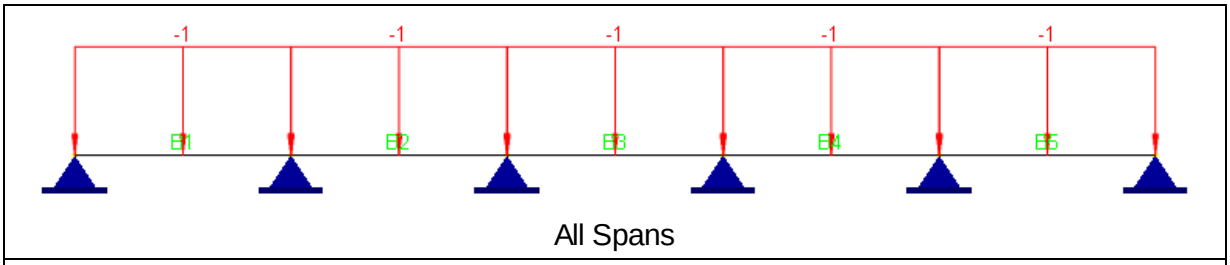
There are a few limitations to the area load concept in the program. The first limitation is that the sub-areas must be close-formed by perimeter members. In the following figure, the sub-area formed by node 97, 98, 104 and 103 is not a closed sub-area because there is no member connecting the node 97 and 98. As a result, no area loading will be distributed to the three perimeter members from the sub-area. The second limitation is that sub-areas must not overlap. In Figure 16.3, the sub-areas in node 101, 102, 120 and 119 are overlapping. This will result more load being distributed to the members in these sub-areas. The program gives a warning when the area load footprint is not equal to the total actual loaded area. The problem may be solved by splitting members 119-108, 107-120 and 113-114 at the intersection point.



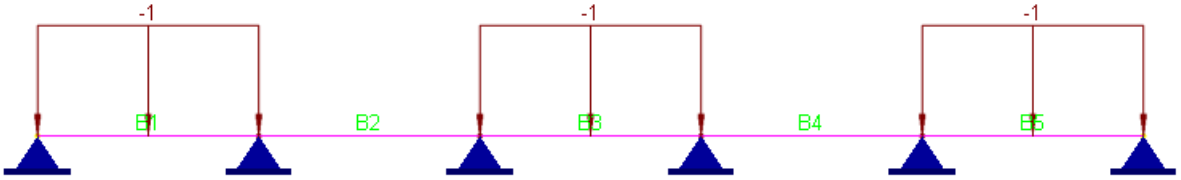
The third limitation is that any sub-area may not contain more than one concave node (with internal angle more than 180 degrees). The fourth limitation is that any sub-area may not contain the same node more than once in forming the perimeter polygon.

The program offers automatic generation of live load patterning (point and line loads only). The following figure shows how the program generates load patterning on a five-span continuous beam. Loads on each generated pattern reside in a separate load case automatically generated. Additional load combinations are generated as needed as well.

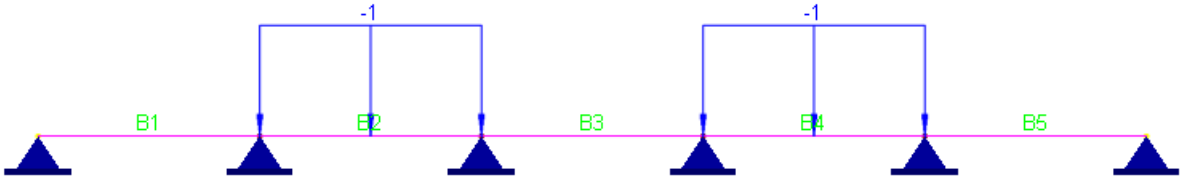
© 2026 ENERCALC, LLC



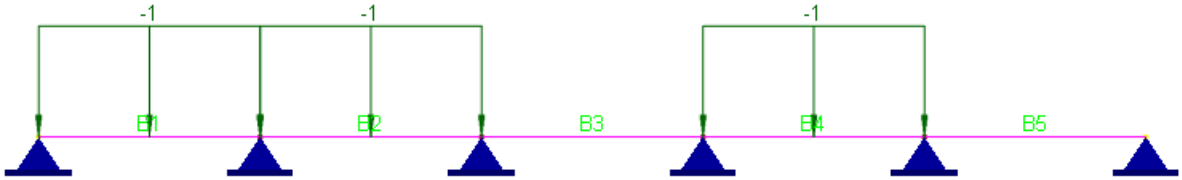
All Spans



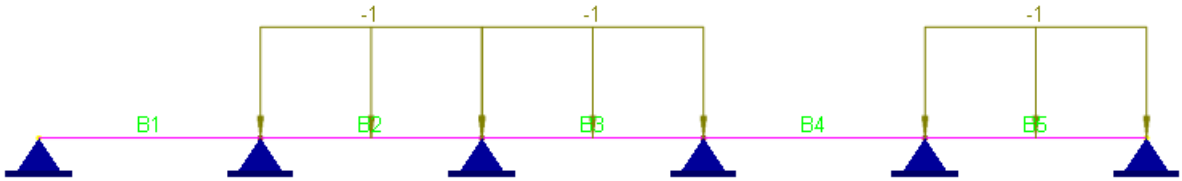
Odd Spans



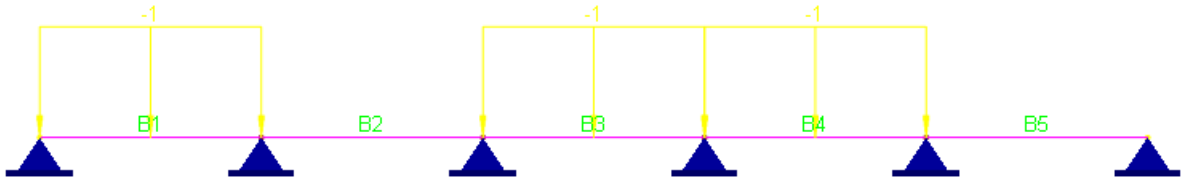
Even Spans



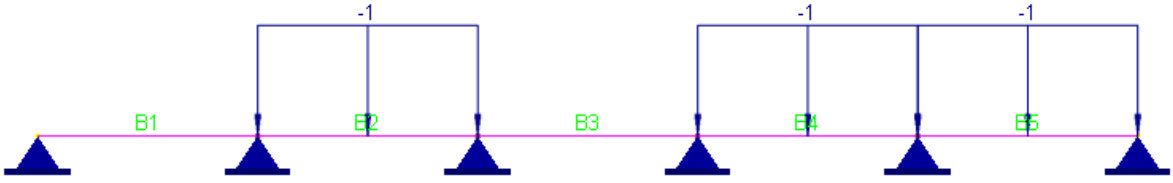
Second Support + Alternate Spans



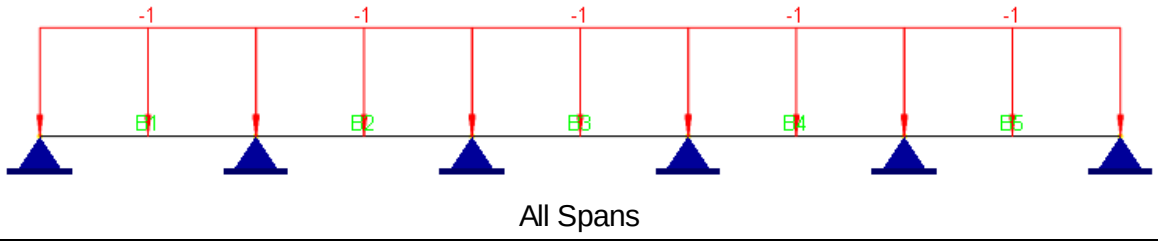
Third Support + Alternate Spans



Fourth Support + Alternate Spans



© 2026 ENERCALC, LLC



All Spans

Fifth Support + Alternate Spans

The program also offers automatic generation of moving loads (point loads only). The mechanism employed by the program is similar to the live load patterning.

Line Springs

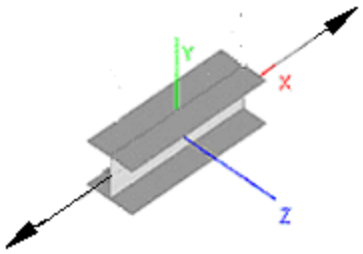
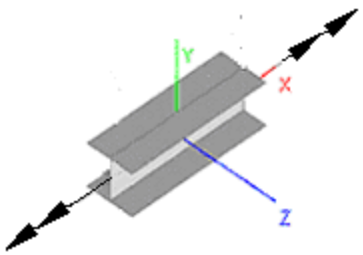
Springs are flexible restraints. Springs applied to members are referred to as line springs. You may assign a line spring to a member with one or more global DOFs (D_x, D_y and D_z) restrained. To qualify as a valid flexible restraint, the corresponding spring constant must be specified. A restraint may be designated as linear, compression-only or tension-only. A compression-only restraint is active only when the nodal displacement in the restrained direction is negative. A tension-only restraint is active only when the nodal displacement in the restrained direction is positive. If a model contains one or more compression-only or tension-only springs, the whole problem becomes nonlinear and the solution becomes iterative for each load combination

The forces or moments required to enforce the flexible restraints are called spring reactions. They are computed by the program.

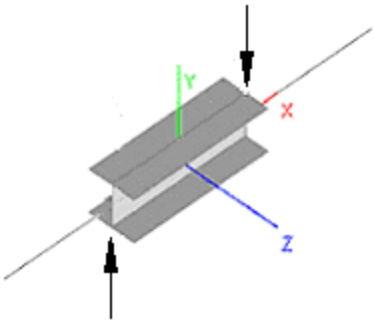
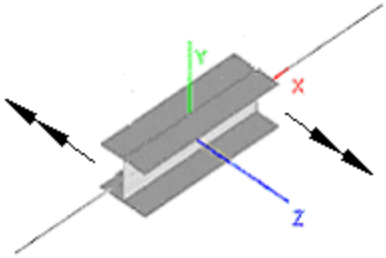
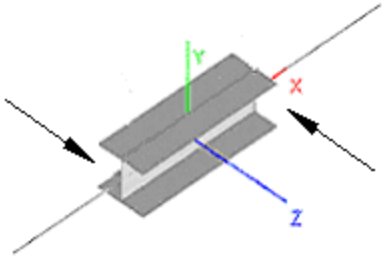
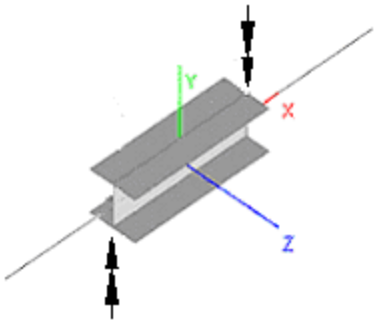
Internal Forces and Moments

The program outputs internal forces and moments at designated stations along the member length. You may specify the number of segments ranging from 1 to 127 for member output by running the command Analysis > Analysis Options. For smooth moment and shear diagrams, the program may add extra segments.

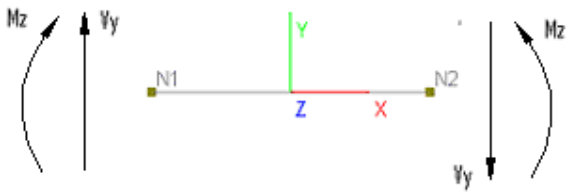
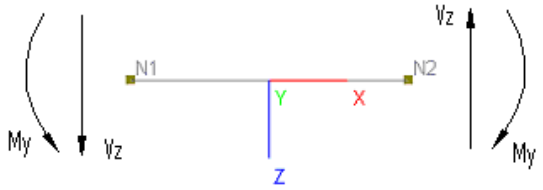
The following figure shows the positive direction of the internal forces and moments of members.

Forces	Moments
	

© 2026 ENERCALC, LLC

Forces	Moments
Axial Force V_x	Torsion M_x
 Major Shear V_y	 Major Moment M_z
 Minor Shear V_z	 Minor Moment M_y

The following figure is an alternative way to show the positive direction of the internal shears and moments on the local xy and xz planes.

Moment & shear on the local xy plane	Moment & shear on the local xz plane
	

13.3.1.10 Shells

A shell is a structure or part of a structure which has a relatively small thickness in comparison with the other two dimensions. A general shell forms a curved surface in space. When it forms a flat surface, it is also called a plate. In this program, when the term “plate” is used, it refers to flat shell in the out-of-plane bending action.

© 2026 ENERCALC, LLC

The shell element in the program is a four-node (quadrilateral) element that combines the in-plane membrane action and the out-of-plane bending action. The in-plane membrane action is a standard isoparametric compatible formulation with the option to add incompatible modes. The out-of-plane bending action is a thick-plate formulation, with the option to use the thin-plate formulation when the element is rectangular. The element can be used to model both flat-surface plates and curved-surface general shells. Applications of shell elements in structures are wide and far-reaching. Examples are concrete floors, mat foundations, shear wall, folded plates, barrel vaults, cooling towers, spherical domes, water tanks, etc. The program provides powerful commands to generate these and other commonly used plate and shell structures. These commands include Geometry > Generate Shells, Edit > Extrude, Revolve, etc.

For many years, a great number of papers have been published on the subject of plate and shell structures. Although the membrane action of a shell element is relatively simple, the (plate) bending action is much more complex. Many plate elements have been proposed, some of which have been implemented in commercial programs. However, most of these proposed plate elements are either ineffective or unreliable. One of the main hurdles is known as transverse “shear locking”, that is, elements behave too stiff with respect to shear deformation especially when elements are thin or geometrically distorted.

One of the few reliable plate elements is a rectangular thin plate element developed by O.C. Zienkiewicz [Ref. 2]. It is based on the Kirchhoff thin plate bending theory in which a line straight and normal to the mid-surface of the plate before loading is assumed to remain straight and normal to the deformed mid-surface after loading. The transverse shear strain is therefore assumed to be zero. This plate element is important in that it is the first plate element that can be applied reliably in engineering practice. Prior to this, plate analysis depended mainly on very few “closed-form” solutions of simple geometry and boundary conditions, and other very approximate methods such as equivalent frame method of ACI [Ref. 12]. The Kirchhoff rectangular thin plate is implemented in the program. It produces results that converge to “closed-form” solutions as finite element meshes are refined. The element, however, has to be rectangular in shape and does not account for shear deformation.

A much more reliable and effective plate bending element is the MITC4 developed by K.J. Bathe and others [Ref. 1]. It is a thick plate that is based on Mindlin plate theory in which a line straight and normal to the mid-surface of the plate before loading is assumed to remain straight but not necessarily normal to the deformed mid-surface after loading. The element considers shear as well as bending deformations and may be used for both thick and thin plates. This plate element differs from earlier Mindlin theory based plate elements in that different (mixed) interpolations are used to account for the bending and transverse shear strains. The MITC4 plate bending element is implemented in the program. It is free from “shear locking” and performs well even when element meshes are distorted. The shape of the element may be any general quadrilateral as long as the aspect ratio is within a reasonable range (say 0.2 to 5.0).

Shell Thicknesses

Each shell must have a thickness assigned to it. Based on the ratio of thickness to span length, you may choose to use the thin or thick plate bending formulation.

© 2026 ENERCALC, LLC

The thick plate formulation is generally recommended over the thin plate formulation because it applies equally well to both thick and thin plates. The program therefore uses the thick plate formulation by default. If thickness to span ratio is less than 1/20 and elements are rectangular, you may use the thin plate formulation. The thickness should be compared to the support distances, not to the sizes of individual plate elements.

It is important to point out that out-of-plane shear forces exist in thin plates even though shear deformations are not considered. You may draw an analogy between a plate and a beam. A thin plate is analogous to a Euler-Bernoulli beam while a thick plate is analogous to a Timoshenko beam. We consider shear deformation for the Timoshenko beam but not for the Euler-Bernoulli beam, while shear forces exist in both the Euler-Bernoulli and Timoshenko beams.

Local Coordinate System

Each shell element has its own local coordinate system. The element local coordinate systems are used in element stiffness formulations. They are also used for inputs such as loads and outputs such as internal shears, moments, and stresses. Local angles for rectangular shells must be 0s if thin plate bending formulation is used in the analysis options. For definition of the shell local coordinate system, refer to Coordinate Systems.

Shell Numbers

A distinct integral number is assigned to each shell. Duplicate numbers in shells are not permitted. There can be gaps in shell numbering sequence. The order of shell numbering in a model is insignificant to the results or solution time. You may renumber the shells sequentially using the command Modify > Renumber > Renumber Selected Shells.

Element In-Plane Stiffness Matrix

The in-plane element formulation accounts for D_x and D_y of the local coordinate system. The in-plane stiffness matrix of the element is based on the standard isoparametric formulation [Ref 1, 2, 3]. However, when the element is rectangular in shape, incompatible modes may be optionally added to the formulation [Ref. 3]. An incompatible element, when applied, yields results of high quality especially when used to model in-plane bending. Full two by two numerical integration is used to calculate the in-plane stiffness matrix of the element.

Element Out-of-Plane Stiffness Matrix

Out-of-plane bending accounts for D_z, D_{ox} and D_{oy} of the local coordinate system. By default, the MITC4 thick plate formulation is used [Ref. 1]. If the thin plate option is chosen, elements with rectangular shapes will be calculated based on the Kirchhoff thin plate formulation [Ref. 1]. Full two by two numerical integration is used in either case to calculate the out-of-plane stiffness matrix of the element.

Combining Element In-Plane and Out-of-Plane Stiffness Matrices

The shell element stiffness matrix is the combination of the in-plane and out-of-plane stiffness matrices. In order to avoid singularity of the stiffness matrix, a very small “fictitious” stiffness is added to the diagonal term associated with the local DOF D_{oz}.

© 2026 ENERCALC, LLC

Loads

Surface loads may be applied to a shell. You may specify loads in either the global or local coordinate system. Surface loads are lumped to element nodes before solution. The self weight of shells may be calculated automatically if the material weight densities and self weight multiplier are nonzero. By default, the self weight acts in the negative global Y direction. You may, however, change the direction to positive or negative direction of the global X, Y or Z. This flexibility is useful under certain circumstances. For example, if you select the model type “2D Plate Bending”, the self weight may be either in positive or negative global Z direction, depending on your preference on the sign convention. To activate automatic self weight calculation, use the command Create > Draw Loads > Self Weights.

Surface Springs

Springs are flexible restraints. Springs applied to shells are referred to as surface springs. You may assign a surface spring to a shell with one or more global DOFs (D_x, D_y and D_z) restrained. To qualify as a valid flexible restraint, the corresponding spring constant must be specified. A restraint may be designated as linear, compression-only or tension-only. A compression-only restraint is active only when the nodal displacement in the restrained direction is negative. A tension-only restraint is active only when the nodal displacement in the restrained direction is positive. If a model contains one or more compression-only or tension-only springs, the whole problem becomes nonlinear and the solution becomes iterative for each load combination.

The forces or moments required to enforce flexible restraints are called spring reactions. They are computed by the program. Surface springs may be used to model Winkler mat foundations. It may be worthwhile to note that in modeling a mat foundation, surface spring constants are the soil subgrade moduli while surface spring reactions are the soil pressures.

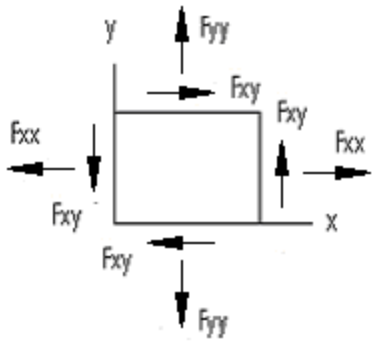
Internal Forces or Moments

The internal forces and moments exist at every point on the middle surface of the shell element. They represent the resultants of different normal and shear stresses over the element thickness. The internal forces have the units of force per unit length and the internal moments have the units of moment per unit length.

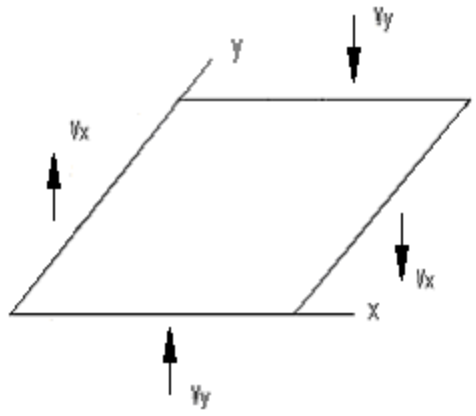
The in-plane or membrane results include the normal forces F_{xx}, F_{yy} and shear force F_{xy}. The out-of-plane results include the shear forces V_x, V_y and bending moments M_{xx}, M_{yy}, M_{xy}. M_{xy} is also called twisting moments. It is important to differentiate these forces and moments

© 2026 ENERCALC, LLC

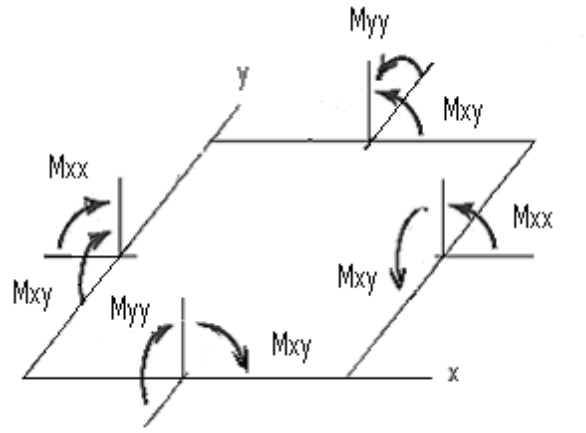
The following figure shows the positive direction of the internal forces and moments of a shell. They represent forces and moments at one point on the middle surface of the element. The program outputs these forces and moments at the four corner nodes and /or at the center of the element. You may use Analysis Results > Results Diagrams > Contour Diagram to see the distribution of these and other resultants. Generally speaking, internal forces or moments (or result in general) are different across element boundaries. You have the option to average forces and moments for adjacent elements at nodes. To do that, click Analysis > Analysis Options.



the in-plane axial and shear forces



the out-of-plane bending shears



the out-of-plane bending moments

© 2026 ENERCALC, LLC

Based on the internal forces and moments, the program computes the internal stresses at the shell bottom (the –z side) and top (the +z side) as follows. The stresses are expressed in the local coordinate systems. The stress directions correspond to the in-plane normal axial forces and shear, and the out-of-plane shears.

$\sigma_{xx} = \frac{F_{xx}}{t} + \frac{6M_{xx}}{t^2}$ (@ bottom) or $\sigma_{xx} = \frac{F_{xx}}{t} - \frac{6M_{xx}}{t^2}$ (@ top)

$\sigma_{yy} = \frac{F_{yy}}{t} + \frac{6M_{yy}}{t^2}$ (@bottom) or $\sigma_{yy} = \frac{F_{yy}}{t} - \frac{6M_{yy}}{t^2}$ (@ top)

$\sigma_{xy} = \frac{F_{xy}}{t} + \frac{6M_{xy}}{t^2}$ (@bottom) or $\sigma_{xy} = \frac{F_{xy}}{t} - \frac{6M_{xy}}{t^2}$ (@ top)

$\sigma_{xz} = \frac{V_x}{t}$

$\sigma_{yz} = \frac{V_y}{t}$

The program also outputs in-plane principal forces and angles, and out-of-plane principal forces, moments, and angles. In addition, principal stresses S_1, S_2, and S_3 are computed based on the stresses $\sigma_{xx}, \sigma_{yy}, \sigma_{xy}$ as follows:

$$S_1 = \frac{\sigma_{xx} + \sigma_{yy}}{2} + \sqrt{(\frac{\sigma_{xx} - \sigma_{yy}}{2})^2 + \sigma_{xy}{}^2}$$

$$S_2 = \frac{\sigma_{xx} + \sigma_{yy}}{2} - \sqrt{(\frac{\sigma_{xx} - \sigma_{yy}}{2})^2 + \sigma_{xy}{}^2}$$

$S_3 = 0$

The Von Mises stress, which is often used to estimate the yield of ductile materials, is then computed as follows:

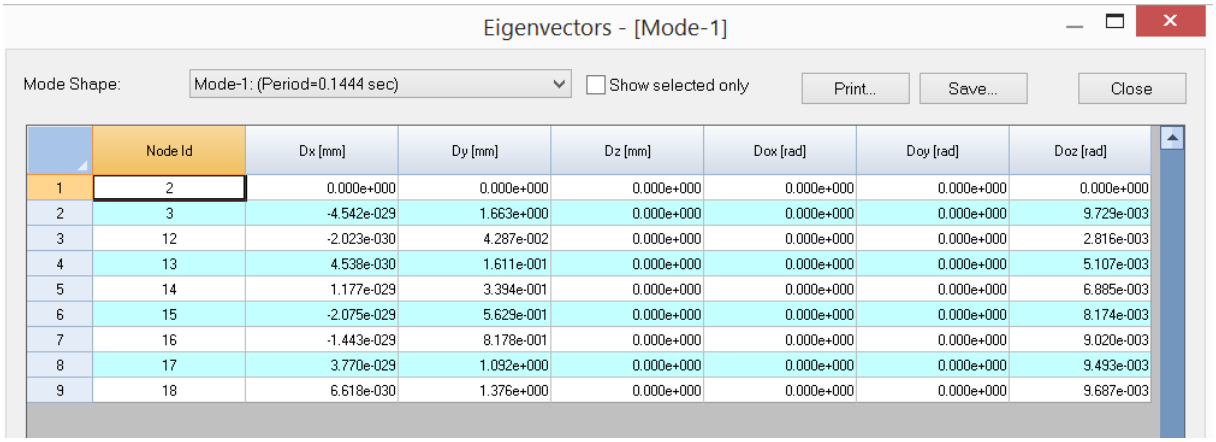
$$\sigma_{VonMises} = \sqrt{\frac{(S_1 - S_2)^2 + (S_1 - S_3)^2 + (S_2 - S_3)^2}{2}}$$

© 2026 ENERCALC, LLC

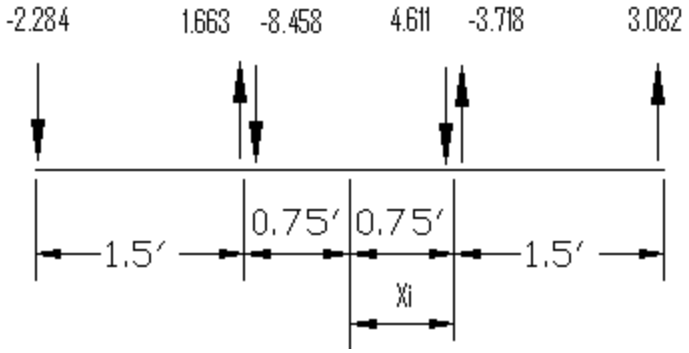
Membrane Nodal Resultants

The membrane nodal resultants of a shell are concentrated forces F_x and F_y (with force units) at the four nodes of each shell element. They in effect keep each individual element in equilibrium (in-plane). They are expressed in the local coordinate system.

You may view nodal forces of selected shell elements by Quick Access Toolbar > Annotate. The membrane nodal forces may be used to compute shears, axial forces, or moments in a shear wall. For example, the following three shells represent a pier in a shear wall. Each shell is 1.5 x 1.5 ft in size. Membrane nodal resultants F_x and F_y are shown in the first and second rows respectively at each corner of the element. The shear, axial force and moment resultants on the top of the pier may be computed as follows:



	Node Id	Dx [mm]	Dy [mm]	Dz [mm]	Dox [rad]	Doy [rad]	Doz [rad]
1	2	0.000e+000	0.000e+000	0.000e+000	0.000e+000	0.000e+000	0.000e+000
2	3	-4.542e-029	1.663e+000	0.000e+000	0.000e+000	0.000e+000	9.729e-003
3	12	-2.023e-030	4.287e-002	0.000e+000	0.000e+000	0.000e+000	2.816e-003
4	13	4.538e-030	1.611e-001	0.000e+000	0.000e+000	0.000e+000	5.107e-003
5	14	1.177e-029	3.394e-001	0.000e+000	0.000e+000	0.000e+000	6.885e-003
6	15	-2.075e-029	5.629e-001	0.000e+000	0.000e+000	0.000e+000	8.174e-003
7	16	-1.443e-029	8.178e-001	0.000e+000	0.000e+000	0.000e+000	9.020e-003
8	17	3.770e-029	1.092e+000	0.000e+000	0.000e+000	0.000e+000	9.493e-003
9	18	6.618e-030	1.376e+000	0.000e+000	0.000e+000	0.000e+000	9.687e-003



F_{xi} (kips)	F_{yi} (kips)	X_i (ft)	$F_{yi} * X_i$ (ft-kips)
1.758	-2.284	-2.25	5.139
3.226	1.663	-0.75	-1.24725
7.226	-8.458	-0.75	6.3435
5.786	4.611	0.75	3.45825
3.144	-3.718	0.75	-2.7885

© 2026 ENERCALC, LLC

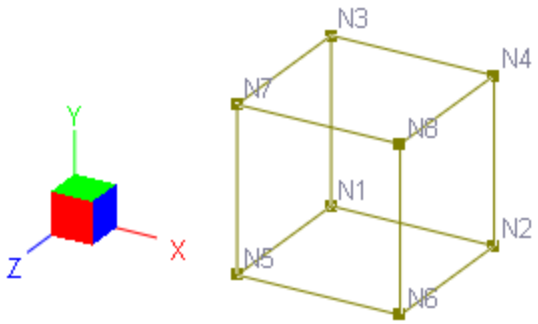
F_{xi} (kips)	F_{yi} (kips)	X_i (ft)	$F_{yi} * X_i$ (ft-kips)
2.722	3.082	2.25	6.9345
Shear ΣF_x = 23.862	Axial Force ΣF_y = -5.104		Moment ΣM = 17.8395

13.3.1.11 Bricks

The Brick element in the program is an eight-node solid element based on isoparametric compatible or incompatible formulation [Ref 1, 2, 3]. It may be used to model structures where actions in all three dimensions are significant.

Local Coordinate System

The local coordinate systems for all Bricks are the same. They are identical to the global coordinate system. The element nodal connectivity must be numbered in such a way so that the normal vector of the surface 1-2-3-4 points to the surface 5-6-7-8 in the following figure.



solid element

This is to avoid negative diagonals in the element stiffness matrix. You may use the command Modify > Reverse Node Order for Selected Elements if the normal vector is not in accordance with the requirement. For more information about the brick local coordinate system, refer to Coordinate Systems.

Brick Numbers

A distinct integral number is assigned to each Brick. Duplicate numbers in Bricks are not permitted. There can be gaps in Brick numbering sequence. The order of Brick numbering in a model is insignificant to the results or solution time. You may renumber the Bricks sequentially using the command Modify > Renumber > Renumber Selected Bricks.

Element Stiffness Matrix

The element formulation accounts for D_x, D_y and D_z of the local coordinate system. The element stiffness matrix is based on isoparametric compatible or incompatible formulation [Ref 1, 2, 3]. Therefore, the stiffness matrix is of size 24 by 24. Full two by two numerical integration is used to calculate the stiffness matrix.

© 2026 ENERCALC, LLC

Loads

Loads on Brick elements must be input as nodal loads.

The self weight of Bricks may be calculated automatically if material weight densities and self weight multiplier are nonzero. By default, the self weight acts in the negative global Y direction. You may however change the direction to positive or negative direction of the global X, Y or Z. To activate automatic self weight calculation, use the command Create > Draw Loads > Self Weights.

Internal Stresses

Three normal stresses σ_{xx}, σ_{yy} , σ_{zz} and three shear stresses σ_{xy}, σ_{xz}, σ_{yz} are computed by the program. They are output at the eight nodes and/or at the center of the element.

The program also outputs principal stresses S_1, S_2, S_3 and the corresponding directional vectors (V_{1x}, V_{1y}, V_{1z}) and (V_{3x}, V_{3y}, V_{3z}). The Von Mises stress, which is often used to estimate the yield of ductile materials, is then computed as follows:

$$\sigma_{VonMises} = \sqrt{\frac{(S_1 - S_2)^2 + (S_1 - S_3)^2 + (S_2 - S_3)^2}{2}}$$

13.3.1.12 Static Analysis

The stiffness (or displacement-based) method is used in the solution of the structural model. The following outlines the major analysis steps:

- The individual element stiffness matrix [k] is computed in the element local coordinate system.
- Based on the element nodal connectivity, [k] is transformed to the global coordinate system and assembled into the global stiffness matrix [K].
- The load vector [R] for each load combination is formed.
- The equation [K] [U] = [R] is solved for the nodal displacements [U].
- Other structural responses such as internal forces and moments are computed based on the nodal displacements.

Load Cases and Load Combinations

Each of the nodal loads, point loads, line loads, surface loads, and self weights must be assigned to a load case. The enforced displacements of supports are special loads and are considered in each load combination. The load cases are used as bases for the load combinations and are not solved directly. If you desire to solve for a particular load case, you may form a load combination with a unit load factor for that load case and 0s for all other load cases.

P-Delta analyses may be performed on one or more load combinations.

© 2026 ENERCALC, LLC

Linear, Non-linear Static Analyses

The program is capable of performing linear and nonlinear static analyses. The linear analysis may be applied to models where structural responses such as the displacements are expected to be linearly related to the applied loads. Otherwise the nonlinear analysis must be applied. The program currently handles two types of nonlinearity: the element nonlinearity when compression-only springs or tension-only springs are present, and the geometric nonlinearity which is commonly known as the P-Delta effect. The P-Delta effect refers to the axial stress influence on the element bending stiffness. Generally, a tensile axial force increases the element bending stiffness while a compressive axial force reduces the element bending stiffness. The P-Delta effect exists in both members and shell elements. However, the program only accounts for the P-Delta effect on members.

The program assigns each load combination to be linear or nonlinear just before analysis is performed. If a model includes one or more nonlinear elements (compression-only springs or tension-only springs), the entire problem becomes nonlinear, that is, all load combinations are assigned to be nonlinear. If there are no nonlinear elements present in the model, only the P-Delta load combinations are set to be nonlinear while the rest of load combinations are linear. The non-linear load combinations must be solved iteratively and therefore are potentially time consuming. Analyses are performed on all linear load combinations first and then on all nonlinear load combinations.

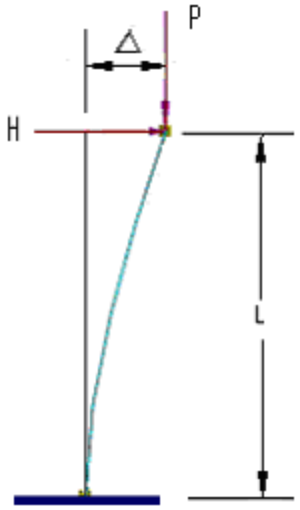
In order to avoid excessive iterations on nonlinear load combinations, you can use the command Analysis > Analysis Options to set “Maximum nonlinear iterations”. For the P-Delta load combinations, you can use the same command to set “Axial force tolerance between P-Delta iterations”. A tolerance of 0.5% is normally acceptable. It is strongly recommended that you perform linear analyses for all load combinations before you attempt P-Delta analyses. In this way, you can identify any modeling problems prior to performing more rigorous and generally more time consuming P-Delta analyses.

It may be interesting to note that the P-Delta analysis may be used to estimate the buckling load of a structure for a P-Delta load combination. To do that, try to apply different scales (λ) uniformly to the load factors of all load cases in the P-Delta load combination, until a zero or negative diagonal term is detected in the global stiffness matrix during the solution process. The lowest scale λ is the buckling load factor.

© 2026 ENERCALC, LLC

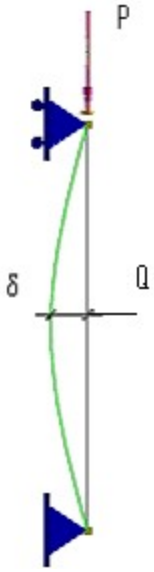
P-Delta (P-Δ) vs. P-delta (P-δ)

The P-Delta (P-Δ) refers to the second order effect associated with the lateral translation of the members [Ref. 10, 11, 12]. Consider the moment M at the bottom of the column in the figure below.



If the effect of the axial force on bending is ignored, M = H * L. However, if the effect of the axial force on bending is considered, M = H * L + P * Δ. The increase in moment in turn increases the deflection Δ, which further increases M, and so on. An equilibrium will eventually be reached unless the axial load P exceeds the column critical buckling load.

P-delta (P-δ) refers to the second order effect associated with the member curvature [Ref. 10, 11, 12]. Consider the moment M at the middle of the column in the figure below.



© 2026 ENERCALC, LLC

A secondary moment P * δ is induced by the axial load acted upon the lateral defection of the column. This additional moment will cause more lateral deflection, which in turn will induce more secondary moment, and so on. An equilibrium will eventually be reached unless the axial load P exceeds the column critical buckling load.

The presence of the axial force in effect reduces the column bending stiffness. The member geometric stiffness accounts for this reduction. The P-Delta analysis in the program is capable of handling both P-Δ and P-δ effects. In order to account for the P-δ component, however, you must split compression members (columns) into several segments. Normally four segments for each column are enough. The program provides the command Edit > Split Members to automatically split members.

As an example [Ref. 13], assume in the figure above, the beam-column is of L = 12 ft in length, and is subjected to an axial compressive load of P = 100 kips and a transverse load of Q = 6 kips at midspan. The member section: 4 x 4 inches, I = 21.33 in^4, A = 16 in^2. The material: E = 30000 ksi, υ = 0.30. Theoretical results are calculated as follows:

Linear (bending only): $M_{mid} = \frac{QL}{4} = 18$ ft-kips; $\delta_{mid} = \frac{QL^3}{48EI} = 0.583$ in

P-δ (bending and axial load): $u = \frac{L}{2}\sqrt{\frac{P}{EI}} = 0.90$ radian (or 51.57^O)

$M_{mid} = \frac{QL}{4}\frac{Tan(u)}{u} = 25.2$ ft-kips; $\delta_{mid} = \frac{QL}{4P}\frac{Tan(u)-u}{u} = 0.864$ in

To solve this problem in the program, we can create one linear load combination and one P-Delta load combination. Since the problem involves the P-δ effects, the beam-column must be modeled with multiple elements (4 beam elements generally sufficient). The results from the program are compared with the theoretical results below:

The moments and deflections at the midspan for linear and P-δ behaviors

Analysis Type	Effects	ENERCALC 3D	Theoretical
Linear	**δ_{mid}** (in)	0.5832	0.583
	M_{mid} (ft-kips)	18	18
P-δ	**δ_{mid}** (in)	0.8643	0.864
	M_{mid} (ft-kips)	25.203	25.2

© 2026 ENERCALC, LLC

Solution Algorithm

Mathematically, the static analysis involves solving the following simultaneous equations:

$$[K]\,[U] = [R]$$

where [K] is the global stiffness matrix, [U] is the displacement vector, and [R] is the load vector for each load combination.

There are two solution algorithms used in ENERCALC 3D: skyline and sparse. The skyline solution algorithm used to solve the equation above was developed by K.J. Bathe [Ref. 1]. It is an active column (also called profile or skyline) solver that involves the factorization of a stiffness matrix and the back-substitution of the load vector. The factorization generally takes most of solution time while the back-substitution is relatively fast. For all linear load combinations, the factorization only needs to be performed once. For nonlinear load combinations, the factorization has to be performed multiple times on each load combination because the global stiffness matrix has to be updated during the solving process. This is the reason why linear and nonlinear load combinations are analyzed separately.

The sparse solver only stores non-zero elements in the global stiffness matrix, thus it is both more memory efficient and much faster than the skyline solver. It also has the option to use an out-of-core approach to minimize the requirement of computer memory. This is useful to solve extremely large structural models. The sparse solver is available for static analysis only. It lacks some of the informative error messages when something goes wrong during the solution process.

Solution Accuracy and Stability

At the very basic level, the solution involves basic arithmetic operations such as addition, subtraction, multiplication and division on floating point numbers. Since all numbers in computers are stored in finite number of bits or digits, round-off errors are introduced by manipulations of these numbers. Round-off errors depend on the precisions of floating point arithmetic and may affect the solution accuracy and stability under certain circumstances. Two types of precisions are generally available on most computers today: single precision and double precision. A single precision (or 32-bit) floating point value has numerical accuracy of about 7 significant digits while a double precision (or 64-bit) floating point value has numerical accuracy of about 15 or 16 significant digits.

Take a look at the following example:

A = 1.00000001; B = -1.0; C = 1.0; D = C / (A + B);

Theoretically, D = 100000000.0. With 64-bit floating point arithmetic, the statement yields D = 100000000.60775 while with 32-bit floating point arithmetic, the statement yields D = +∞. As we can see, D is approximately (not exactly) equal to the theoretical answer with double precision arithmetic. The solution collapsed (division by zero) with single precision arithmetic. The reason for this to happen is during the addition of A and B, the fractional part of A (0.00000001) is rounded off due to lack of enough significant digits. In general, 32-bit floating point arithmetic should never be used in any structural or finite element analysis programs.

© 2026 ENERCALC, LLC

The 64-bit floating point (double precision) has been the predominant solver over the last several decades. For most not-so-large and well-conditioned models, standard 64-bit floating point solvers produce results that are sufficiently accurate for practical uses. However for very large and complex models and especially those under ill-conditioned circumstances, standard 64-bit floating point solvers sometimes produce inaccurate results.

Ill-conditioning occurs when small errors in the coefficients of equations before or during the solution process have large impact on solution results. It may make the solution unstable and results unreliable. Very severe ill-conditioning may even make the coefficient matrix singular and thus a solution non-existent. Some examples where ill-conditioning may occur are: finite elements with severe shape distortion or large aspect ratio; shells with very strong in-plane stiffness and very weak out-of-plane bending stiffness; very flexible elements connected to very stiff elements. *It may be worthwhile to note that when ill-conditioning does happen, finer element meshes tend to make the problem worse.*

During the solution process of a large model, round-off errors tend to accumulate. We can determine the number of significant digits lost based on the diagonal decay ratio [Ref. 3].

$$r_i = K_{ii} / P_{ii}$$

where K_{ii} is the original diagonal coefficient of the global stiffness matrix and P_{ii} is the reduced value of K_{ii} just before it is used for back-substitution. The number of significant digits lost is about $\log_{10}(r_i)$. For example, if r_i is 10^8, then 8 digits are lost. The solution results given by the 64-bit floating point (double precision) are unreliable if 12 or more significant digits are lost during the solution process. The program reports the number of digits lost during the solution process.

Consider the following cantilever beam under a tip load of 10,000 lbs:

L = 100 in; I_{zz} = 200 in^4;

E = 2.9e7 psi; v = 0.3;

P = -10000 lb

© 2026 ENERCALC, LLC

The beam is modeled with 1, 1000, 10000, 20000, 50000 elements and an analysis is performed on each model. Theoretically all models should yield the same tip deflection of -0.5747 inch (shear deformation ignored). The following table shows tip deflections for the all five models using the 64-bit floating point (double precision) in the program.

Effect of number of elements on accuracy (the 64-bit skyline solver) of a cantilever beam

No of elements	1	1,000	10,000	20,000	50,000
Tip deflection (in)	-0.5747	-0.5748	-0.6522	-0.1534	No solution
No of digits lost	0	8	12	12	-

As we can see from the table above, the tip deflections given by the 64-bit skyline solver tend to deteriorate in accuracy as the number of elements increases. For the model with 50,000 elements, some diagonal terms in the global stiffness matrix even become negative. The solver has to abort and the solution is not obtainable anymore.

After identifying a severe ill-conditioning problem, the 64-bit floating point (double precision) generally stops the solution process. **No results are better than wrong results.** To address the problem, a more accurate solver is needed. ENERCALC 3D implements a unique 128-bit floating point (quad precision) solver which offers unparalleled advantages in solution accuracy and most importantly solution stability over the standard 64-bit floating point (double precision) solver. The 128-bit floating point (quad precision) provides numerical accuracy up to 30 significant digits. Many of ill-conditioned problems for the 64-bit floating point solver become well-conditioned problems for the 128-bit floating point solver. The superiority of the 128-bit floating point solver can be demonstrated by running the same cantilever beam above with 50,000 elements, the tip deflection given by the 128-bit floating point (quad precision) is -0.5747 inch, the correct answer.

A more practical application of the 128-bit floating point (quad precision) may be in the modeling of a rigid diaphragm (e.g. floor with much larger in-plane stiffness than out-of-plane stiffness). Most of the other programs model this kind of rigid diaphragm action through equal displacement constraints in order to avoid numerical difficulties. With the 128-bit floating point solver available in ENERCALC 3D, you may model the floor as a flexible diaphragm, yielding much more realistic results.

It should be pointed out that the 128-bit floating point (quad precision) requires twice as much memory as the 64-bit floating point (double precision). It is also significantly slower. However, in situations where the standard 64-bit floating point solver produces unreliable or even wrong results, the 128-bit floating point (quad precision) provides an invaluable alternative. Between faster but wrong results and slower but correct results, the latter is obviously preferable.

13.3.1.13 Frequency Analysis

The frequency analysis solves for frequencies and corresponding mode shapes (eigenvectors) of the structural system. Many concepts discussed in the previous chapter-“Static Analysis”, apply to the frequency analysis as well.

© 2026 ENERCALC, LLC

Solution Algorithm

Mathematically, the frequency analysis involves solving the following Eigen problem:

$$[K]\,[\Phi_i] = \lambda_i\,[M]\,[\Phi_i]$$

where [K] is the global stiffness matrix, [M] is the global mass matrix, $[\Phi_i]$ is the ith mode shape and λ_i is the ith eigenvalues which is equal to the free vibration circular frequency squared $(\omega_i)^2$. Other related values are frequency f_i which is $2\pi\ \omega_i$ and period T_i which is $1 / f_i$. For practical reasons, we are generally interested only in the lowest eigenvalues (and therefore lowest frequencies).

The solution of eigenvalue problems must be iterative in nature because it is equivalent to finding the roots of the polynomial p(λ). The solution algorithm to solve the equation above is given by K.J. Bathe [Ref. 1]. It uses the subspace iteration method to iteratively find the lowest p eigenvalues $\lambda_1, \lambda_2, \ldots \lambda_p$ and corresponding vectors $[\Phi_1], [\Phi_2], \ldots [\Phi_p]$. Eigenvalues are extracted in ascending order. Each eigenvector is then normalized such that $[\Phi_i]^T [M]\,[\Phi_i] = [I]$ where [I] is the identity matrix, a diagonal matrix with unit values along the main diagonal.

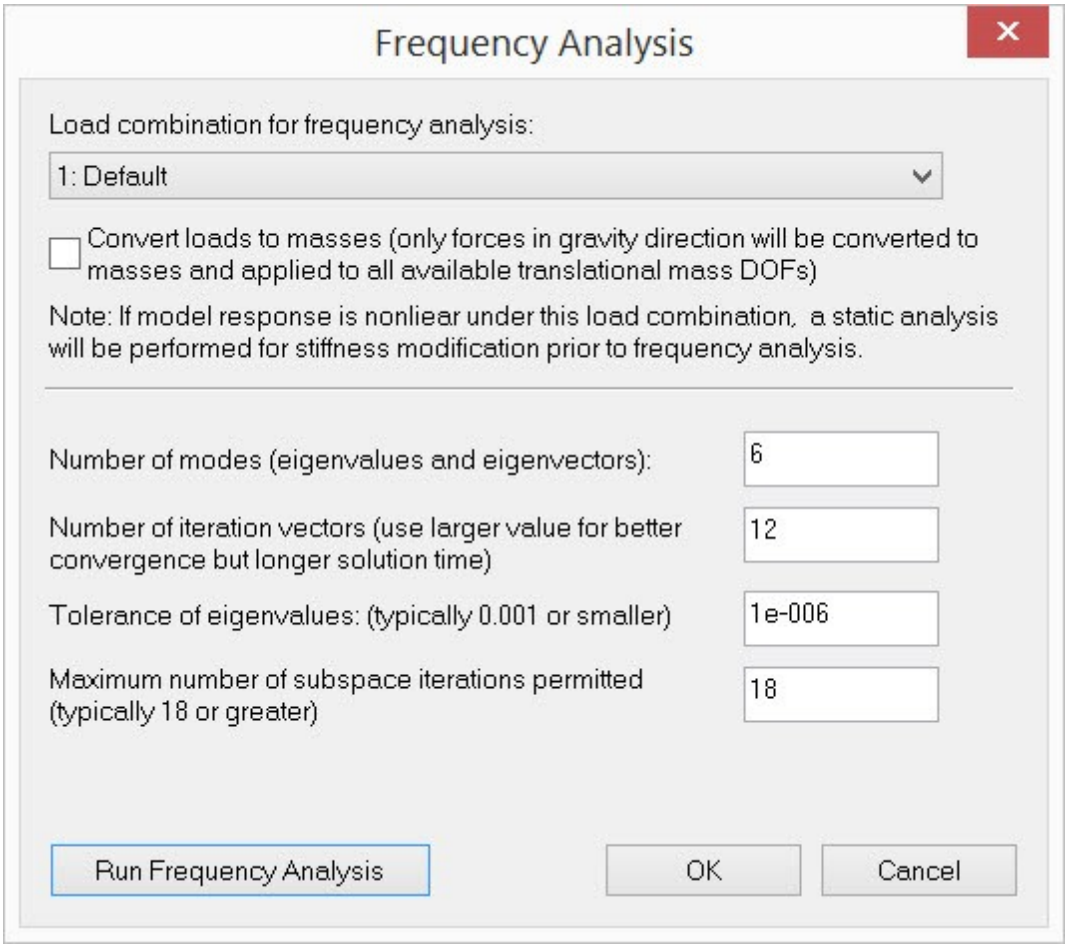
A tolerance may be set before the solution to control the convergence of eigenvalues during each successive solver iteration. It is expressed as the following:

$$\frac{\lambda_i^{(k+1)} - \lambda_i^{(k)}}{\lambda_i^{(k+1)}} \le tolerance \qquad (i = 1, 2, \ldots \text{number of requested modes})$$

where k is the subspace iteration counter.

© 2026 ENERCALC, LLC

To prevent excessive computing time, a maximum number of subspace iterations may be set before the solution. If the solver reaches this limit without convergence, the eigen results should not be trusted.



Mass and Stiffness

The global mass matrix [M] is diagonal and is computed based on the load combination for frequency analysis and/or additional nodal masses/mass moments of inertia. The load combination for frequency analysis may be specified in Analysis > Frequency Analysis. The program will automatically convert all forces (not moments) in the positive or negative gravity direction to nodal masses and apply them in all available mass degrees of freedom. Additional nodal masses and mass moments of inertia may be input from Loads > Additional Masses or Input > Additional Masses. Zero terms in the global mass matrix [M] are allowed. The number of eigenvalues requested must be fewer than the mass DOFs which is the number of nonzero diagonal terms in [M]. Due to the lumped mass modeling, the elements should be properly divided or submeshed for a continuous vibration model. For example, a beam with uniformly distributed mass should be divided into at least eight elements in order to find accurate vibration results.

© 2026 ENERCALC, LLC

The load combination for frequency analysis is also used to compute the global stiffness matrix [K] if the model response is not linear. This may be the case if 1). The load combination for frequency analysis is of P-Delta type; or 2). The model contains nonlinear elements such as compression-only springs. In the first case (geometric nonlinearity), the compressive forces decrease the model stiffness (and therefore lengthen the vibration periods of the model) while tensile forces increase the model stiffness. The influence of the axial loads is greater on the lower frequencies than on the higher ones. The effect of nonlinearity on the stiffness matrix of the structure is incorporated as follows:

- An iterative (nonlinear) static analysis is first performed with the loads in the load combination for frequency analysis.
- The stiffness matrix at the end of the static analysis will be used in the frequency analysis. The stiffness therefore includes geometric and element nonlinearities corresponding to the end of the nonlinear static analysis.

Forced displacements at supports are ignored in frequency analysis.

Solution Convergence

Due to the iterative nature in the eigen solution, much more computational effort is required (in order to achieve satisfactory convergence) in frequency analysis than in static analysis. Another important difference is solution stability, which is more difficult to achieve in frequency analysis. To ensure that the smallest required eigenvalues and the corresponding eigenvectors have been computed, the program performs a *Sturm* sequence check after the subspace iterations [Ref 1]. A warning message is given in the solver dialog if some eigen values are missing after the Sturm sequence check. Under some rare circumstances, the solution may become unstable and the solver has to abort the solution process

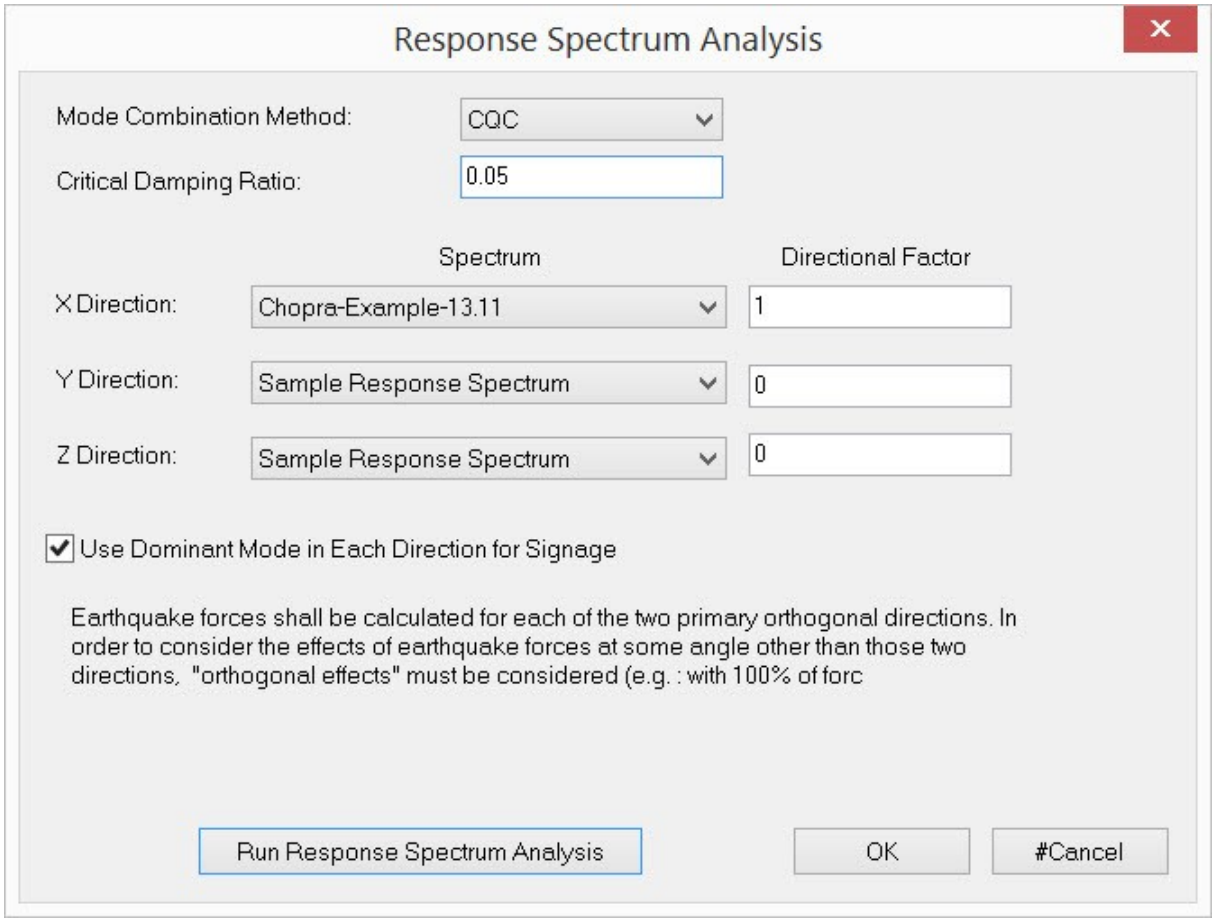
Several remedies can be used to address the solution instability and solution divergence.
1). Solve for fewer number of modes.
2). Use larger number of iteration vectors.
3). Use 128-bit floating point arithmetics instead of 64-bit floating point arithmetics.
These remedies may be used in tandem. Once again, the 128-bit floating point arithematics is especially effective for solution stability.

The maximum number of subspace iterations is set to 18 by default. If no convergence is achieved at this limit, you should rerun the frequency analysis with a larger maximum number of subspace iterations.

© 2026 ENERCALC, LLC

13.3.1.14 Response Spectrum Analysis

Analysis > Response Spectrum Analysis prompts you with the dialog shown in the screen capture below.



It allows you to perform response spectrum analysis in global X, Y and/or Z directions. There are three mode combination methods available in the program: CQC (complete quadratic combination), SRSS (Square root of sum of squares) and ABSSUM (absolute sum). CQC method for modal combination is applicable to a wider class of structures and is therefore recommended method. Critical damping ratio (0 <= damp < 1.0) affects CQC results. When critical damping ratio is 0, CQC method is the same as SRSS method.

You must first run from Analysis | Frequency Analysis prior to running this command. Response spectrums can be defined from Loads | Response Spectra Library or Input Data | Response Spectra Library. Inertia forces in global direction X, Y or/and Z from response spectrum analysis will be calculated and then converted to nodal loads. These nodal forces will be placed in respective load cases such as INERTIA_LOADCASE_X_MODE_1, INERTIA_LOADCASE_X_MODE_2 etc. Existing loads in these load cases will be deleted prior to the load conversion. In addition, response spectrum load combinations INERTIA_LOADCOMB_X_MODE_1, INERTIA_LOADCOMB_X_MODE_2 etc. will be created or recreated. Static analysis will be performed on spectrum load combinations (as well as normal user-defined load combinations) automatically. Modal combinations will be

© 2026 ENERCALC, LLC

subsequently calculated for results such as displacements, forces and stresses etc. using CQC, SRSS or ABSSUM on the response spectrum load combinations. Normally, modal combination results are all positive due to the sign lost during SRSS, CQC and ABSSUM procedures. However, you can choose to use signage for modal combination results based on the dominant mode (with maximum participation factor) in each global direction.

Modal combination results is done in each global direction first. Using directional factors, these directional modal results will be combined into final modal combination results, which can be added to any user-defined load combination results if non-zero response spectrum load factor is specified in the load combination definition (see Create > Loads > Load Combinations).

13.3.1.15 Concrete Design – ACI 318-02/05/08/11/14

The concrete design module performs concrete design for beams, columns and plates (bending only) according ACI 318-02, 05, 08, 11 and 14 [Ref. 19]. Static analysis must be performed successfully before concrete design can be performed. Sound engineering judgment is especially important to interpret and apply the design results given by the program.

Note: The provisions that apply for ACI 318-02 and 05 shall apply to ACI 318-08, 11 and 14 unless explicitly stated otherwise.

Running Concrete Design and Viewing Results

© 2026 ENERCALC, LLC

To run the Concrete Design and obtain results, the following minimum conditions must be met. Note that concrete design is a complex process, and this list is not comprehensive. For a more detailed description of the Concrete Design feature, please refer to the Concrete Design 444 section of the ENERCALC 3D User Manual.

1. Load Combinations must be set for Concrete Design:
   a. Navigate to Create > Loads > Load Combinations.
   b. For each load combination to be considered, double-click the combination.
   c. Enable the option "Perform Concrete Design using this Load Combination".
   d. When concrete columns are to be designed, a sustained load combination for computing βd must be defined. See Load Combinations 337 for more information regarding setting up the sustained load combination.
   e. If the concrete design includes sway column criteria requiring P-Delta analysis, also enable "Perform P-Delta Analysis on this Load Combination".
   f. Click OK in the Load Combination window.
   g. Click OK in the Load Combinations window.

2. Sustained load combination must be designated in RC Model Design Criteria:
   a. Go to Concrete Design > RC Model Design Criteria.
   b. Under "Sustained load combination for computing Beta-d in columns", select the sustained load combination described above.
   c. Click OK to confirm.

3. Members in the model must be assigned a concrete material:
   a. Go to Tables > Properties > Materials and click the Std Material... button.
   b. Choose the desired concrete material from the drop down menu, then click OK.
   c. Click Apply to close the window.
   d. Next, go to Tables > Entities > Members.
   e. Update the Material column to assign the concrete material previously added.
      i. Tip: To assign the same material to all members quickly, right-click the section in the table after setting it for one member. You will be prompted to auto-fill the column with this value. Click Yes to apply it to all members.
   f. Click OK to confirm.

4. Members in the model must be assigned a regular section:
   a. Go to Tables > Properties > Sections and add a section by clicking the Regular Section... button.
   b. Choose the desired section shape and enter the section dimensions, then click OK.
   c. Click Apply to close the window.
   d. Next, go to Tables > Entities > Members.
   e. Update the Section column to assign the regular shape previously added.
      i. Tip: To assign the same section to all members quickly, right-click the section in the table after setting it for one member. You will be prompted to auto-fill the column with this value. Click Yes to apply it to all members.
   f. Click OK to confirm.

5. Static analysis has been run:
   a. Go to Analysis > Static Analysis.

© 2026 ENERCALC, LLC

b. After running the static analysis, check the Log File to ensure there are no model instabilities or other issues. Address any issues before proceeding with the design.

Once those conditions are met, execute the steel design by navigating to Concrete Design > Perform Concrete Design. You can then review the results via Concrete Design > Concrete Design Output.

Concrete Column Design

General

The concrete column module designs concrete rectangular or circular columns against axial, uniaxial or biaxial bending as well as shear based on ACI 318-02/05/08/11/14 Code Provisions. The program generates **EXACT** (not approximate or empirical) P-Mx-My interaction surfaces for all sections according to user-specified design criteria. The capacity ratio is computed for each column based on capacity interaction surfaces and axial force-biaxial bending in each load combination. Slenderness effects are considered for both non-sway (braced) and sway (unbraced) frames. Shear design in columns is based on the shear force envelope with the option to include or exclude axial force influence on concrete shear capacity.

Axial Load and Moment Convention

For concrete design, compressive and tensile axial loads have positive and negative signs respectively. The major moment is designated as Mx in design as opposed to Mz used in analysis output. The minor moment is designated as My in both analysis and design.

Solution Assumptions

- The strain in reinforcement and concrete is directly proportional to the distance from the neutral axis (ACI 318-02/05 10.2.2).
- The maximum usable strain at the extreme concrete compression fiber is equal to 0.003 (ACI 318-02/05 10.2.3).
- The stress of steel is $f_s = E_s * \varepsilon_s$ but $f_s <= f_y$ where E_s = 29000 ksi, ε_s is steel strain and f_y is the yield strength of steel (ACI 318-02/05 10.2.4).
- The tensile strength of the concrete is neglected in flexural calculation (ACI 318-02/05 10.2.5).
- A uniformly distributed stress of 0.85fc is assumed over an equivalent compression zone bounded by the edge of the cross section and a line parallel to the neutral axis at a distance $a = \beta_1 * c$ where c is the distance from extreme compression fiber to neutral axis (ACI 318-02/05 10.2.7.1).
- $\beta_1 = 0.85 - 0.05 * (f'_c - 4)$ and $0.65 <= \beta_1 <= 0.85$ and f'_c unit is ksi
- Reinforcement ratio ρ should be 1% <= ρ <= 8% for column sections (ACI 318-02/05 10.2.7.3).

Solution Algorithms

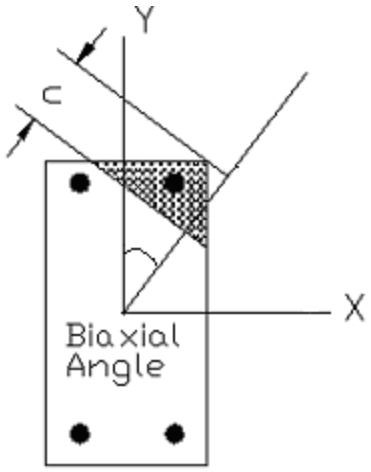
1. All sections are EXACTLY solved biaxially based on the solution assumptions above. Each section is solved based on the following steps.

© 2026 ENERCALC, LLC

2. Nominal Strength Calculation (P_n, M_{nx}, M_{ny})

2a. The nominal capacity of a section is computed at successive choices of biaxial angles. The choices of angles are based on the user input for biaxial angle steps found in the command Concrete > Design Options. Biaxial angle steps affect the solution accuracy and speed. For biaxial problems, steps must be multiples of 4. A value of 16 ~ 32 is sufficiently accurate for most sections. The adequacy of biaxial angle steps can be determined by smoothness of the M_x-M_y interaction diagram. For uniaxial problems, biaxial angle steps should be set to 4. This will give P-M_x (+) at 0 degree angle, P-M_x (-) at 180 degrees angle, P-M_y (+) at 90 degree angle, P-M_y (-) at 270 degrees angle.

The number of biaxial angle steps is analogous to the number of sides of a polygon used to approximate a circle or ellipse. A uniaxial solution in the program is therefore analogous to using a square to approximate a circle or a rectangle to approximate an ellipse. A biaxial solution with 16 angle steps is analogous to using a 16-sided polygon to approximate a circle or an ellipse. Obviously, the 16-sided polygon is closer or more accurate to approximate a circle than a square. The moral of this comparison is that a low value of biaxial angle steps tends to give more conservative biaxial capacity for the section.



2b). For each biaxial angle, P_n, M_{nx}, M_{ny} and maximum tensile steel strain ε_t are computed at successive choices of neutral axis distance c using strain compatibility and stress-strain relations to establish bar forces and the concrete compressive results. The choices of c are based on the neutral axial steps found in the command Concrete > Design Options. Neutral axial steps affect the solution accuracy and speed. A value of 250 ~ 500 for neutral axis steps is sufficiently accurate for most sections. The adequacy of neutral axis steps can be determined by smoothness of the P-M_x and/or P-M_y interaction diagrams. In addition, the program always computes several control points. They are maximum P_n (compression), minimum P_n (tension), f_s = 0; 0.25f_y; 0.5f_y and 1.0f_y (balanced condition). Concrete displaced by steel may be optionally included or excluded (by default).

2c). M_{nx}-M_{ny} contour curves are computed for successive choices of axial forces. This is achieved through interpolation on the P_n, M_{nx} and M_{ny} already calculated for each biaxial

© 2026 ENERCALC, LLC

angle in the procedure above. The choices of axial forces are based on the neutral axial steps found in the command Concrete > Design Options.

3. Design Strength Calculation (φP_n, φM_{nx}, φM_{ny})

Design strength according to ACI 318-02,05/08/11/14 is obtained by multiplying P_n, M_{nx} and M_{ny} of each biaxial angle by applying strength reduction factor φ as determined in the following (ACI 318-02/05 9.3.2):

Φ_c = 0.65, α = 0.80 for tied confinement

Φ_c = 0.70, α = 0.85 for spiral confinement for ACI 318-02/05

Φ_c = 0.75, α = 0.85 for spiral confinement for ACI 318-08/11/14

For ($\varepsilon_t <= \varepsilon_y$, compression-controlled sections)

$\varphi = \Phi_c$

For (ε_t > 0.005, tension-controlled sections)

φ = 0.90

For ($\varepsilon_y < \varepsilon_t < 0.005$)

$\varphi = \Phi_c + (0.9 - \Phi_c) * (\varepsilon_t - \varepsilon_y) / (0.005 - \varepsilon_y)$

where ε_t is maximum tensile steel strain for the biaxial angle and ε_y is steel yield strain (at balanced condition)

In addition, φP_n must be always less than the following (ACI 318-02/05 10.3.6.1)

$\Phi_c * \alpha * [0.85 * f_c' * (A_g - A_s) + f_y * A_s]$ if concrete displaced by steel is excluded or

$\Phi_c * \alpha * [0.85 * f_c' * A_g + f_y * A_s]$ if concrete displaced by steel is not excluded.

4. **Capacity Ratio**

Capacity ratio is computed for each section based on the loads and the capacity of the section. It is defined as the following:

For a given load set (P_u, M_{ux}, M_{uy}), find the section capacity M_x-M_y contour at $\varphi P_n = P_u$. The capacity ratio for the load set is the larger of:

$$\left(\frac{\sqrt{(M_{ux})^2 + (M_{uy})^2}}{\sqrt{(\varphi M_{nx,max})^2 + (\varphi M_{ny,max})^2}}, \frac{P_u}{(\varphi P_{n,max})} \right)$$

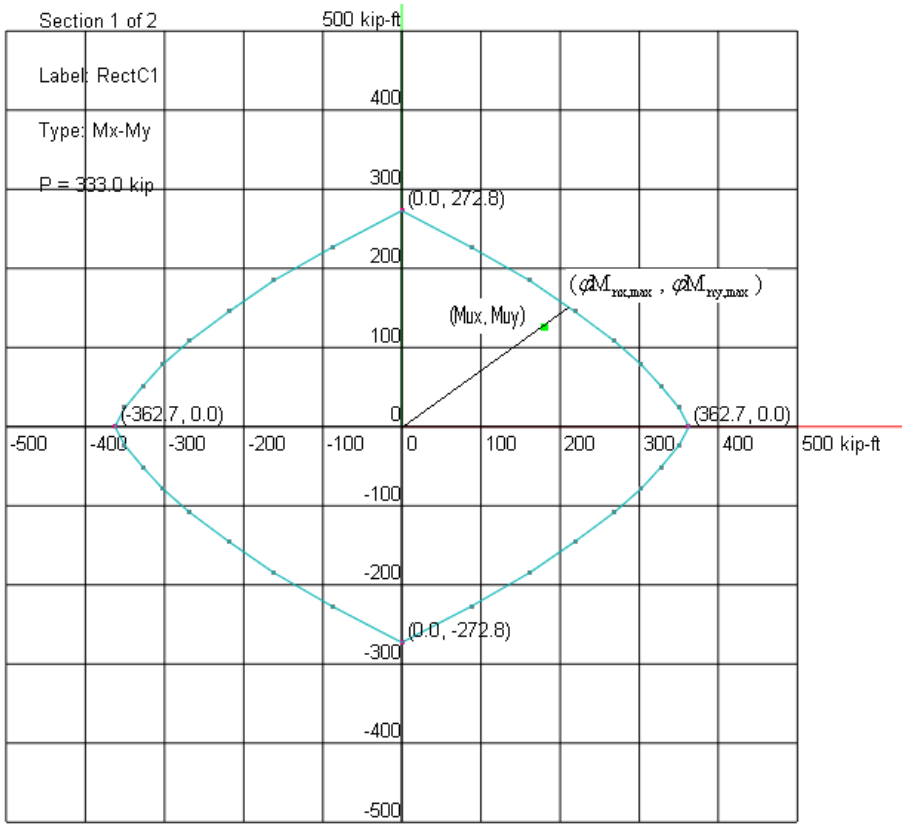
Where ($\varphi M_{nx,max}$, $\varphi M_{ny,max}$) is the interaction point between the line from point (M_{ux}, M_{uy}) to point (0, 0) and the M_x-M_y contour line. is the maximum compression or tension capacity of the section, depending on the positive or negative sign of P_u. If P_u is outside the maximum compression or tension capacity, a capacity ratio of 99.9 is assigned.

© 2026 ENERCALC, LLC

A capacity ratio equal or less than 1.0 means the design strength is greater than the required strength; and the section is adequate to resist all input loads. A capacity ratio greater than 1.0 means the design strength is less than the required strength and the section is inadequate to resist all input loads. *It is important to realize that capacity ratio defined in the program is just a measure of section adequacy against loads. It should not be equated to a factor of safety.*

Capacity Ratio Calculation Example

To illustrate the calculation of capacity ratio in the program, see the following example. For a given load set (P_u, M_{ux}, M_{uy}) = (333 kip, 180 ft-kip, 125 ft-kip), a M_x-M_y capacity contour at φP_n = 333 kip is obtained as shown below. In addition, the maximum compression capacity = 1050.2 kip.



© 2026 ENERCALC, LLC

The interaction point between the line from point (180 ft-kip, 125 ft-kip) to point (0, 0) and the contour line is obtained as (214.1 ft-kip, 148.7 ft-kip)

$$\frac{\sqrt{(M_{ux})^2+(M_{uy})^2}}{\sqrt{(\phi M_{nx,max})^2+(\phi M_{ny,max})^2}} = \frac{\sqrt{(180)^2+(125)^2}}{\sqrt{(214.1)^2+(148.7)^2}} = 0.841$$

$$\frac{P_u}{(\phi P_{n,max})} = \frac{333}{1050.2} = 0.317$$

Therefore, the capacity ratio corresponds to the load set is 0.841. The section is adequate to resist the load.

P-δ and P-Δ Effects 337

Two types of second-order moment effects may develop in a frame.
1). P-δ effect is associated with individual member curvature. Additional second-order moment may develop by member (usually a column) axial force (P) acting upon the lateral deflection (δ) of the column axis away from the chord connecting the column ends. It is possible to account for P-δ effects on columns independently.

2). P-Δ effect is associated with the lateral drifts of the frame members. Additional second-order moment may develop by axial force (P) acting upon the lateral translation (Δ) of the frame nodes relative to their original position. It is NOT possible to account for P-Δ effects on columns independently.

Slenderness Effects 449

For a non-sway frame, P-Δ effect may be safely ignored and the first-order structural analysis is therefore sufficient. The program then accounts for P-δ effect by magnifying the first-order moments using ACI moment magnification method.

For a sway frame, the second-order structural analysis must be performed to account for P-Δ effect. In addition, the program accounts for P-δ effect by magnifying the second-order moments using ACI moment magnification method. In fact, all columns in sway frames must first be considered as braced columns under gravity loads acting alone.

Braced or Unbraced Column 449

The column is considered braced if one of the following two criteria is met:
Criterion 1: Increase in column end moment due to second-order effects is less than 5% of the first-order moment

Criterion 2: Stability index Q for the column story under consideration from the first-order analysis

$$Q = \frac{(\sum P_u)\Delta_0}{V_u l_c} < 0.05$$

(ACI 318-02/05 Eq10-6)

© 2026 ENERCALC, LLC

Section Properties for Structural Analysis and Computing K

It is important to point out that in both first- and second-order analyses; appropriate member stiffness must be used to account for the effects of axial loads, cracking, and creep.

$E_c = 57000\sqrt{f_c}$ for normal weight concrete

$E_c = w_c^{1.5} 33\sqrt{f_c}$ for w_c between 90 and 155 lb/ft^3

Moment of inertia (ACI 318-02/05 10.11.1)
= 0.35 I_g for beams and cracked walls
= 0.70 I_g for columns and uncracked walls
= 0.25 I_g for flat plates and flat slabs

Area
A = 1.0 A_g

Note:
a). Ig and A_g are based on the gross concrete cross section, neglecting reinforcement.
a). Ig for Tee beams can be closely approximated as 2 times I_g for the web.
b). 0.70 I_g should be used for walls first. If the factored moments and shears indicate that a portion of the wall will crack due to stresses reaching the concrete modulus of rupture, the analysis should be repeated with 0.35 I_g for the cracked portions of the wall.
[Ref. 16 pp577]

The program allows a user to modify the moments of inertia through Element Cracking Factor from Concrete > Cracking Factors. In order to use stiffness reduction, you also need to check “Use cracked section properties (Icr) for members and finite elements” in Analysis Options. This allows you to consider or ignore cracking in the analysis without re-entering element cracking information.

Radius of Gyration

$r = \sqrt{I_g / A_g}$

© 2026 ENERCALC, LLC

Effective Length Factor K 449

Find relative stiffness ratios ($\Psi_1 \text{ and } \Psi_2$) of columns and beams at the top and bottom joints of the column

$$\Psi = \frac{\sum \frac{EI}{L} \text{ for column members}}{\sum \frac{EI}{L} \text{ for beam members}}$$

Section properties are the same as used in the first-order analysis (step 1)
For practical reasons, $\Psi = 0.2$ for fixed end and $\Psi = 20$ for hinged end.

The effective length factor K is solved from the from the following equations
For braced frames:

$$\frac{\Psi_1 \Psi_2 \pi^2}{4K^2} + \frac{\Psi_1 + \Psi_2}{2}\left[1 - \frac{\pi / K}{\tan(\pi / K)}\right] + \frac{2}{\pi / K}\tan\left(\frac{\pi}{2K}\right) - 1 = 0$$

For unbraced frames:

$$\frac{\Psi_1 \Psi_2 (\pi / K)^2 - 36}{6(\Psi_1 + \Psi_2)} - \frac{\pi / K}{\tan(\pi / K)} = 0$$

The program provides a tool to calculate the effective length factor K based on the input $\Psi_1 \text{ and } \Psi_2$. The equations above provide a more accurate K calculation than what is given by (ACI 318-02/05 10.12.1)

Unsupported Length Lu 449

The unsupported lengths L_{uy}, L_{uz} of a column are the clear distances between lateral supports in column local y and z directions (ACI 318-02/05 10.11.3.1). A zero value of Lu means that it is equal to the member length between the end nodes.

For non-sway frames, an optional check is made kLu / r <= 34 – 12(M_1/M_2) (ACI 318-02/05 Eq10-7). Braced frame k is used here. Lu is unbraced length in local x and y directions. M_1 and M_2 are the smaller and larger factored end moments on the compression member respectively. (M_1/M_2) is positive if the member is bent in single curvature and negative otherwise.

For sway frames, an optional check is made kLu / r <= 22 (ACI 318-02/05 10.13.2). Sway frame k is used here.

© 2026 ENERCALC, LLC

Minimum Moments
The program calculates minimum moments for both braced and unbraced frames, $M_{min} = P_u(0.6 + 0.03h)$, where h is in inches (ACI 318-02/05 10.12.3.2). The program conservatively applies the minimum eccentricity about both axes simultaneously.

Equivalent Moment Factor Cm (ACI 318-02/05 10.12.3.1)

$C_m = 1.0$ if M1 = 0 or M2 = 0

$C_m = 1.0$ if transverse load exists

$C_m = 0.6 + 0.4\frac{M_1}{M_2} \geq 0.4$ if end moments only. (ACI 318-02/05 Eq10-13).

Although not required, the program also conservatively applies $C_m \geq 0.4$ *for ACI 318-08 /11/14.*

The sign of $\frac{M_1}{M_2}$ is: positive if the column is bent in single curvature, negative otherwise.

Note, Cm is only applicable to non-sway frames. You have the conservative option to always use Cm = 1.0 from Model Design Criteria under Concrete > Design Criteria.

Section Properties for Critical Loads Computation
The EI used in the frame analysis above is an average value. In designing individual columns, the following reduced EI should be used to reflect the greater chance of cracking:

$EI = \frac{0.4E_cI_g}{1+\beta_d}$ (ACI 318-02/05 Eq10-12)

The Infamous 446 β_d
The ratio of maximum factored axial sustained load to maximum factored axial total load. The factor β_d accounts for the effects of creep.

Generally: $\beta_d = \frac{Factored\ Dead\ Load}{Factored\ Dead\ Load + Factored\ Live\ Load}$

Critical Load Pc

$P_c = \frac{\pi^2 EI}{(kl_u)^2}$ where $k \leq 1.0$ (ACI 318-02/05 Eq10-10)

© 2026 ENERCALC, LLC

Moment Magnification Factor

$$\delta_{ns} = \frac{C_m}{1 - \frac{P_u}{0.75 P_c}} \geq 1.0$$

(ACI 318-02/05 Eq10-9)

P_u is the average of axial force at both ends. If $1 - \frac{P_u}{0.75 P_c} < 0$, the design fails and a capacity ratio of 999.9 is assigned.

Other Requirements
Reinforcement ratio for columns:

Bar requirements: minimum 4 bars for tied columns, 6 bars for spiral columns.

Tie requirements: >= #3 for No. 10 longitudinal bars and smaller; >= #4 for No. 11 14, 18 longitudinal bars.

Column Trial Size

The ACI code requires that the reinforcement ratio for columns be within $0.01 \leq \rho_t \leq 0.08$. It is usually economical to have $\rho_t = 0.01 \sim 0.02$.
For tied columns

$$A_s \geq \frac{P_u}{0.40\left(f_c' + f_y \rho_t\right)}$$

For spiral columns

$$A_s \geq \frac{P_u}{0.45\left(f_c' + f_y \rho_t\right)}$$

Based on rectangular or circular sections used for analysis, the program will generate column sections with different reinforcement configurations.

Column Shear Reinforcement

The column shear design is based on

$$\phi(V_c + V_s) \geq V_u$$

(ACI 318-02/05 Eq11-1)

where $\phi = 0.75$.

Given b_w, d, f_c, f_y, number of stirrup legs n, and stirrup (tie) area A_V, the required stirrup spacing is computed at every analysis station.

© 2026 ENERCALC, LLC

Concrete shear strength

1. For P_u < 0 (column subjected to tension)

$$\phi V_c = \phi 2\left(1+\frac{N_u}{500A_g}\right)\sqrt{f_c'}\,b_w d \geq 0$$

(ACI 318-02/05 Eq11-8)

2. For P_u >= 0 (column subjected to compression)

$$\phi V_c = \phi 2\left(1+\frac{N_u}{2000A_g}\right)\sqrt{f_c'}\,b_w d$$

(ACI 318-02/05 Eq11-4)

Note:

- For circular section, $b_w = 2R$ and d = 0.8(2R) where R is the radius of the circular section. (ACI 318 11.3.3 and 11.5.7.3)
- Nu = 0 if the influence of compression on concrete shear strength is ignored.
- $\sqrt{f_c'} \leq 100 psi$ (ACI 318 11.1.2)
- $f_c' \leq 60$ ksi in design of shear reinforcement. (ACI 318-02/05 11.5.2)
- Light-weight concrete is not considered.
- Torsional forces are not considered.

The following is the algorithm used to compute the stirrup (tie) spacing(s) in the program.

If $V_u - \phi V_c > \phi 8\sqrt{f_c'}\,b_w d$, the design fails (ACI 318-02/05 11.5.7.9).

If $V_u < \frac{\phi V_c}{2}$, no stirrup required (ACI 318-02/05 11.5.6.1). *The program does not check member depths when applying minimum shear reinforcement for ACI 318-08/11/14.*

If $V_u - \phi V_c \leq \phi 4\sqrt{f_c'}\,b_w d$, smax <= min(d/2, 24 in) (ACI 318-02/05 11.5.5.1)

If $V_u - \phi V_c > \phi 4\sqrt{f_c'}\,b_w d$, smax <= min(d/4, 12 in) (ACI 318-02/05 11.5.5.3)

If $\phi V_c < V_u < \frac{\phi V_c}{2}$, s = min $\left(\frac{A_v f_y}{0.75\sqrt{f_c'}\,b_w}, \frac{A_v f_y}{50 b_w}\right)$ <= smax (ACI 318-02/05 11.5.6.3)

Otherwise, s = $\frac{\phi A_v f_y d}{V_u - \phi V_c}$ <= smax (ACI 318-02/05 Eq11-15)

According to ACI 318-02/05, column confinement spacing shall not exceed 16 longitudinal bar diameters, 48 tie bar or wire diameters, or the least dimension of the compression members.

© 2026 ENERCALC, LLC

The following additional requirements are needed for column spirals:

- The maximum center-to-center spacing:

$$s < \frac{\pi d_{sp}^2 f_y}{0.45 D_c f_c [A_g / A_c - 1]}$$ (Derived from ACI 318-02/05 Eq10-5)

- The clear spacing between successive turns shall not exceed 3 inches, nor be less than 1 inch. (ACI 318-02/05 7.10.4.3)

Concrete Beam Design

General

The concrete beam module designs concrete rectangular or Tee beams against enveloped bending about strong axis (local z) and enveloped shear along local y. Axial force, bending about weak axis (local y), and torsion are not considered. Furthermore, no deep beam action is considered. If axial force or biaxial bending actions cannot be neglected, the use of column design module is recommended.

Beam Flexural Reinforcement

The beam top and bottom flexural reinforcement is computed at each analysis station along the beam length. Minimum reinforcement is computed for the bottom steel. The program designs each beam against positive or negative moment with single layer of tension steel with tension-controlled condition. For flexural design, the critical section at a support may be taken at the face of the support (but not greater than 0.175 * span length from the support center). The program offers an option to account for these conditions by automatically computing beam support widths from Model Design Criteria under Concrete > Design Criteria.

The following algorithm assumes one layer of tension steel, that is, d_t = d, the depth of the tension steel centroid. This assumption is made due to its simplicity and conservative nature and is reasonable unless the tension steel strain is very close to 0.005 – the tension-controlled limit strain.

The design result is reflected in top and bottom reinforcement diagrams.

© 2026 ENERCALC, LLC

Rectangular Beam Flexural Design Algorithm

Given b, d = d_t, d', f_c, f_y and M_u find required A_S (and A_S' if needed)
Step 1: Determine maximum moment without compression steel, using the tension-controlled limit $\varepsilon_t = 0.005$ and $\phi = 0.9$.

$$c_0 = 0.375d$$
$$a_0 = \beta_1 c_0$$
$$C_{f0} = 0.85 f_{c1}^{'} b a_0$$
$$\phi M_{n0} = \phi C_{f0}(d - \frac{a_0}{2})$$
$$A_{s0} = C_{f0} / f_y$$

Step 2: If $M_u \le \phi M_{n0}$, design the section as singly-reinforced as follows:

$$R_n = \frac{M_u}{\phi(bd^2)}$$

$$\rho = \frac{0.85 f_c^{'}}{f_y}\left(1 - \sqrt{1 - \frac{2R_n}{0.85 f_c^{'}}}\right) \ge \rho_{\min} = \max(\frac{3\sqrt{f_c^{'}}}{f_y}, \frac{200}{f_y})$$ (ACI 318-02/05 10.5.1)

$$a = \frac{\rho d f_y}{0.85 f_c^{'}}$$
$$c = \frac{a}{\beta_1}$$
$$\varepsilon_s = \left(\frac{d_t - c}{c}\right) 0.003$$
$$A_s = \rho b d$$

© 2026 ENERCALC, LLC

Step 3: If $M_u > \phi M_{n0}$, design the section as doubly-reinforced as follows (still assuming the tension-controlled limit $\varepsilon_t = 0.005$ and $\phi = 0.9$):

$$f_s' = \left(1 - \frac{d'}{c_0}\right) 0.003(E_s) \le f_y$$

$$A_s' = \left(\frac{M_u - \phi M_{n0}}{f_s'(d - d')\phi}\right)$$

$$A_s = A_{s0} + A_s'\left(\frac{f_s'}{f_y}\right)$$

Note, the tensile steel required to balance the compressive steel is $A_s'\left(\frac{f_s'}{f_y}\right)$

The design fails if $f_S' < 0$. For practical reasons, the design also fails if $A_s' > \frac{1}{2}A_{s0}$.

Tee Beam Flexural Design Algorithm

Given b, b_w, h_f, d = d_t, f_c, f_y and Mu, find required A_S

Step a). Assuming a <= h_f and tension-controlled section with $\phi = 0.9$

$$R_n = \frac{M_u}{\phi(bd^2)}$$

$$\rho = \frac{0.85 f_c'}{f_y}\left(1 - \sqrt{1 - \frac{2R_n}{0.85 f_c'}}\right)$$

$$\rho_{min} = \max\left(\frac{3\sqrt{f_c'}}{f_y}, \frac{200}{f_y}\right)$$

© 2026 ENERCALC, LLC

$$A_s = \max(\rho bd, \rho_{min} b_w d)$$

$$a = \frac{A_s f_y}{0.85 f_c' b}$$

If a > h_f, go to Step b)

$$c = \frac{a}{\beta_1}$$

$$\varepsilon_s = \left(\frac{d_t - c}{c} \right) 0.003$$

If $\varepsilon_s < 0.005$, the design fails.

© 2026 ENERCALC, LLC

Step b). a > h_f and tension-controlled section with $\phi = 0.9$

$$M_{uw} = M_u - \phi(0.85) f_c'(b - b_w) h_f \left(d - \frac{h_f}{2} \right)$$

$$R_n = \frac{M_{uw}}{\phi(b_w d^2)}$$

$$\rho_w = \frac{0.85 f_c'}{f_y} \left(1 - \sqrt{1 - \frac{2R_n}{0.85 f_c'}} \right)$$

$$a_w = \frac{\rho_w d f_y}{0.85 f_c'}$$

$$c = \frac{a_w}{\beta_1}$$

$$\varepsilon_s = \left(\frac{d_t - c}{c} \right) 0.003$$

If $\varepsilon_s < 0.005$, the design fails.

Beam Shear Reinforcement 448

The member shear reinforcement (stirrup spacing) is computed at each analysis station along the member length. Stirrup size and number of legs are assumed uniform along the length of a member as part of input. For shear design, sections located less than a distance d from the face of the support may be permitted to be designed for Vu computed at a distance d from the support (ACI 318-02/05 11.1.3.1). The program offers an option to account for these conditions by automatically computing beam support widths from Model Design Criteria under Concrete > Design Criteria.

The design result is reflected in a stirrup spacing diagram.

Beam Shear Design Algorithm

The shear design for concrete beam is the same as that of concrete column except the axial force is always treated as zero.

© 2026 ENERCALC, LLC

Concrete Slab/Wall Design

General

The concrete slab/wall module designs concrete slabs or walls against enveloped positive and negative Wood-Armer bending moments in slab local x and y directions. Axial force action is ignored. The program produces contours of required areas of steel which can be averaged with some commonsense to finish the design.

Wood-Armer Moments

Wood-Armer Formula [Ref 18, pp198] is the most popular approach to convert Mx, My and Mxy to orthogonal plate design moments Mux and Muy

The procedure to obtain Mux and Muy for designing plate bottom reinforcement is as follows:

- Mux = Mxx + |Mxy|
  Muy = Myy + |Mxy|
- If Mux < 0 and Muy < 0
  Mux = 0
  Muy = 0
  If Mux < 0 and Muy > 0
  Mux = 0
  Muy = Myy + |Mxy * Mxy / Mxx|
  If Mux > 0 and Muy < 0
  Muy = 0
  Mux = Mxx + |Mxy * Mxy / Myy|
- Mux >= 0
  Muy >= 0

The procedure to obtain Mux and Muy for designing plate top reinforcement is as follows:

- Mux = Mxx - |Mxy|
  Muy = Myy - |Mxy|
- If Mux > 0 and Muy > 0
  Mux = 0
  Muy = 0
  If Mux > 0 and Muy < 0
  Mux = 0
  Muy = Myy - |Mxy * Mxy / Mxx|
  If Mux < 0 and Muy > 0
  Muy = 0
  Mux = Mxx - |Mxy * Mxy / Myy|
- Mux <= 0
  Muy <= 0

© 2026 ENERCALC, LLC

Wood-Armer Formula is a lower bound solution method which satisfies the following conditions for a given external load:

- The equilibrium conditions are satisfied at all points in the plate.
- The yield strength of the plate elements is not exceeded anywhere in the plate.
- The boundary conditions are complied with.

A lower bound solution is conservative in nature.

Stress Singularity

The stresses and bending moments at the point of a concentrated load on the slab are theoretically infinite. This *theoretically* means that if we used all the steel in the world, we still did not have enough steel to resist the stress at that point. This is of course ridiculous. The reason is of course because we prescribe an impossible loading ("concentrated load"). If we distribute the load over a small area (circle), the stresses become finite.

In finite element analysis, the program will never give you a stress of infinite magnitude. Still, at a point of concentrated force such as a column acting on a flat plate, stresses may have rather significant spikes. According to Ugural [Ref 15, pp116], the actual stress caused by a load on a very small area of radius r_c can be obtained by replacing the actual r_c with an equivalent radius r_e.

$$r_e = \sqrt{1.6r_c^2 + t^2} - 0.675t \qquad (r_c > 0.5t)$$

$$r_e = r_c \qquad (r_c <= 0.5t)$$

where t is the plate thickness.

By applying this method, we can use stresses at half of the slab thickness instead of those at the concentrated loading point. By excluding the finite elements (usually finely meshed) near the concentrated loading points, we can provide practical and reasonable design results.

Flexural Reinforcement 451

The plate top and bottom flexural reinforcement in local x and y direction is computed at each nodal point as well as the center. No minimum reinforcement is considered. The program only designs each plate with tension-controlled condition. The procedure is similar to that of concrete beams except no double reinforcement is considered.

13.3.1.16 Steel Design

The steel design module performs steel design for beams and columns according AISC 15th edition LRFD [Ref. 21]. Static analysis must be performed successfully before steel design can be performed. Sound engineering judgment is especially important to interpret and apply the design results given by the program.

Due to the fact that the software provides step-by-step calculation procedures, the technical treatment of the design process is kept minimal here.

Running Steel Design and Viewing Results

© 2026 ENERCALC, LLC

To run the Steel Design and obtain results, the following minimum conditions must be met. Note that steel design is a complex process, and this list is not comprehensive. For a more detailed description of the Steel Design feature, please refer to the Steel Design 470 section of the ENERCALC 3D User Manual.

1. Load Combinations must be set for Steel Design:
   a. Navigate to Create > Loads > Load Combinations.
   b. For each load combination to be considered, double-click the combination.
   c. Enable the option "Perform Steel Design using this Load Combination".
   d. If the steel design includes sway column criteria requiring P-Delta analysis, also enable "Perform P-Delta Analysis on this Load Combination".
   e. Click OK in the Load Combination window.
   f. Click OK in the Load Combinations window.

2. Members in the model must be assigned a steel material:
   a. Go to Tables > Properties > Materials and click the Std Material... button.
   b. Choose the desired steel material from the drop down menu, then click OK.
   c. Click Apply to close the window.
   d. Next, go to Tables > Entities > Members.
   e. Update the Material column to assign the steel material previously added.
      i. Tip: To assign the same material to all members quickly, right-click the section in the table after setting it for one member. You will be prompted to auto-fill the column with this value. Click Yes to apply it to all members.
   f. Click OK to confirm.

3. Members in the model must be assigned a steel section:
   a. Go to Tables > Properties > Sections and add a steel section by clicking the AISC Table... button.
   b. Choose the desired section, then click OK.
   c. Click Apply to close the window.
   d. Next, go to Tables > Entities > Members.
   e. Update the Section column to assign the steel shape previously added.
      i. Tip: To assign the same section to all members quickly, right-click the section in the table after setting it for one member. You will be prompted to auto-fill the column with this value. Click Yes to apply it to all members.
   f. Click OK to confirm.

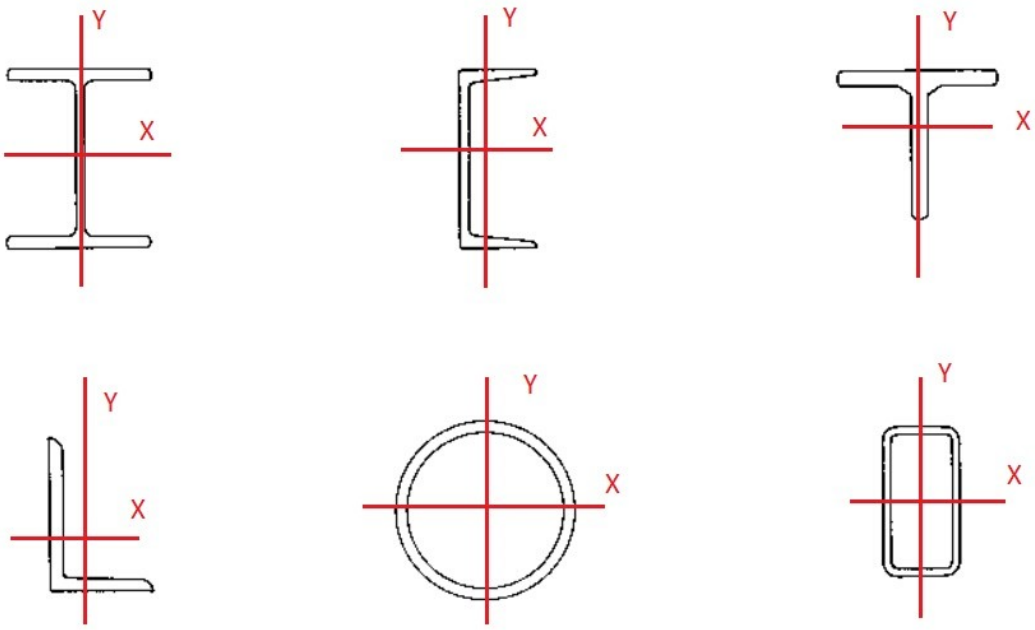
4. Static analysis has been run:
   a. Go to Analysis > Static Analysis.
   b. After running the static analysis, check the Log File to ensure there are no model instabilities or other issues. Address any issues before proceeding with the design.

Once those conditions are met, execute the steel design by navigating to Steel Design > Perform Steel Design. You can then review the results via Steel Design > Steel Design Results.

© 2026 ENERCALC, LLC

Section Orientation

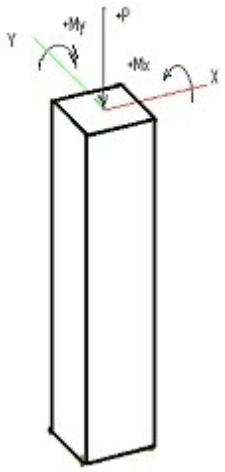
The orientations of section local X and Y axes of various AISC shapes are shown in the figure below.



© 2026 ENERCALC, LLC

Member Internal Forces and Moments

1. Axial force P acts perpendicular to the section. Moments Mx and My act about section local X and Y axes respectively. They have the following sign conventions.
   Axial Force P: positive for compression; negative for tension
   Moment Mx: Positive when section top most fiber is under compression.
   Moment My: Positive when section rightmost fiber is under compression.



2. All moments are referenced about the geometric centroid of the gross section.

3. Loads are the required strength computed by the code-specified factored load combinations using either hands or analysis program. It is assumed that an overall 2nd order P-Delta (P-Δ) analysis has been performed on a sway structure. If desired, the program uses moment magnification procedure to calculate the P-delta (P-δ) effect, which accounts for slenderness of columns in non-sway structure or for slenderness along the lengths of columns in sway structure.

4. Critical ratio (also called capacity ratio) is computed for each section based on the magnified factored loads and the capacity of the section. Critical ratio equal or less than 1.0 means the design strength is greater than the required strength and the section is adequate. Critical ratio greater than 1.0 means the design strength is less than the required strength and the section is inadequate.

Solution Algorithms

Because detailed step-by-step calculation procedure is available for each member on Design > Steel Design Results, we will not list the algorithm here.

13.3.1.17 References

1. K.J. Bathe, “Finite Element Procedures” Prentice-Hall, Inc., 1996
2. O.C. Zinenkiewicz, “The Finite Element Method”, 3rd ed., McGraw-Hill Book Company (UK) Limited, 1983

© 2026 ENERCALC, LLC

3. R.C. Cook & D.S. Malkus & M.E. Plesha, “Concepts and Applications of Finite Element Analysis”, 3rd ed., John Wiley & Sons, Inc., 1989
4. R.H. Macneal & R.L. Harder, “A proposed Standard Set of Problems to Test Finite Element Accuracy”, pp3-20 of “Finite Elements in Analysis and Design”, North-Holland, 1985
5. J.P. Hartog, “Advanced Strength of Materials”, McGraw-Hill Book Company, 1952.
6. R.J. Roark & W.C. Young, “Formulas for Stress and Strain”, 5th ed., McGraw-Hill Book Company, 1975.
7. W. McGuire & R.H. Gallagher & R.D. Ziemian, “Matrix Structural Analysis”, 2nd ed., John Wiley & Sons, Inc., 2000
8. J.S. Przemieniecki, “Theory of Matrix Structural Analysis”, McGraw-Hill, 1968
9. D. Breyer, “Design of Wood Structures”, 3rd ed, McGraw-Hill, 1993
10. J. MaCormac, “Structural Steel Design LRFD Method”, 2nd ed, HarperCollins College Publishers, 1995
11. A. Tamboli, “Steel Design Handbook LRFD Method”, McGraw-Hill, 1997
12. ACI, “Building Code Requirements for Reinforced Concrete (ACI 318-89) (Revised 1992)”, American Concrete Institute, Detroit, Michigan.
13. K. Leet & D. Bernal, “Reinforced Concrete Design”, 3rd ed, McGraw-Hill Book Company, 1997.
14. A. K. Chopra, “Dynamic of Structures – Theory and Applications to Earthquake Engineering”, 2nd ed, Prentice-Hall, 2001
15. Ansel Ugural, “Stresses in Plates and Shells” 2nd Edition, The GcGraw-Hill Companies, Inc., 1999
16. James G. MacGregor & James K. Wight, “Reinforced Concrete – Mechanics and Design” 4th Edition, Pearson Prentice Hall, 2005
17. “Notes on ACI 318-02 Building Code Requirements for Structural Concrete”, 8th Edition, Portland Cement Association, 2002
18. R. Park and W.L. Gamble “Reinforced Concrete Slabs”, John Wiley & Sons, 1980
19. ACI, “Building Code Requirements for Structural Concrete (ACI 318-05) and Commentary (ACI 318R-05)”, American Concrete Institute, Detroit, Michigan, 2004
20. Arthur H. Nilson, David Darwin, Charles W. Dolan, “Design of Concrete Structures”, 13th Edition, McGraw-Hill Higher Education, 2004
21. AISC “Steel Construction Manual”, 14th Edition
22. Charles Salmon, John Johnson and Faris Malhas, “Steel Structures” 5th Edition, Pearson Prentice Hall, 2009

© 2026 ENERCALC, LLC

The following works are not referenced directly in this documentation, but are the primary programming references in the development of this program. Like the great engineering works cited above, these great programming works deserve the respect of being listed here.

a. B. Stroustrup, "The C++ Programming Language" 3rd ed., Addison-Wesley Publishing Company, 1997

b. R.Wright & M.Sweet, "OpenGL SuperBible", 2nd ed., Waite Group Press, 2000

c. D.J. Kruglinski, "Inside Visual C++", Microsoft Press, 1996

d. M. Woo & J. Neider & T. Davis & D. Shreiner, "OpenGL Programming Guide", Addison-Wesley Publishing Company, 1999

e. C. Petzold, "Programming Windows 95", Microsoft Press, 1996

f. C. Musciano & B. Kennedy, "HTML The Definitive Guide" 3rd ed., O'Reilly & Associates, Inc., 1998

13.3.1.18 Appendix

Unit Conversions

From English to Metric			From Metric to English		
1 ft	=>	0.3048 m	1 m	=>	3.28084 ft
1 in	=>	25.4 mm	1 mm	=>	0.03937 in
1 kip	=>	4.44822 kN	1 kN	=>	0.22481 kip
1 lb	=>	4.44822 N	1 N	=>	0.22481 lb
1 kip-ft	=>	1.35582 kN-m	1 kN-m	=>	0.73756 kip-ft
1 kip-in	=>	0.112985 kN-m	1 kN-m	=>	8.85073 kip-in
1 lb-ft	=>	1.35582 N-m	1 N-m	=>	0.73756 lb-ft
1 lb-in	=>	0.112985 N-m	1 N-m	=>	8.85073 lb-in
1 kip/in^2	=>	0.00689476 kN/mm^2	1 kN/mm^2	=>	145.04 kip/in^2
1 lb/in^2	=>	0.00689476 N/mm^2	1 N/mm^2	=>	145.04 lb/in^2

Designations, diameters and areas of standard bars

© 2026 ENERCALC, LLC

ASTM 615 (English)			ASTM 615 96a (Metric)		
Bar No	Diameter (in)	Area (in^2)	Bar No	Diameter (mm)	Area (mm^2)
#3	0.375	0.11	#10	9.5	71
#4	0.500	0.20	#13	12.7	129
#5	0.625	0.31	#16	15.9	199
#6	0.750	0.44	#19	19.1	284
#7	0.875	0.60	#22	22.2	387
#8	1.000	0.79	#25	25.4	510
#9	1.128	1.00	#29	28.7	645
#10	1.270	1.27	#32	32.3	819
#11	1.410	1.56	#36	35.8	1006
#14	1.693	2.25	#43	43.0	1452
#18	2.257	4.00	#57	57.3	2581

13.3.2 2D Frame

This module provides force & deflection analysis, steel member stress analysis and wood stress analysis for two dimensional frames.

The frame you create can be general in nature...the module is not limited to trusses or frames with rigid connections. You have the ability to specify the connection between all members in terms of axial, shear and bending restraints or releases.

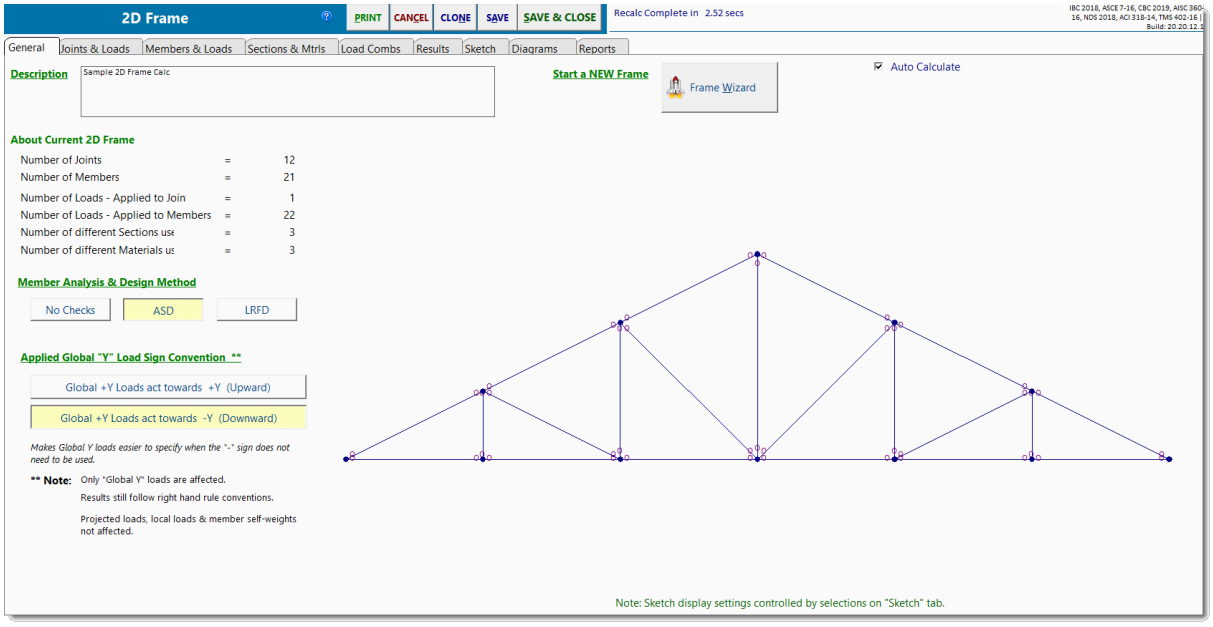
Here are the highlights of this module:

- Use the Frame Wizard to automatically create common trusses and frames.
- Generic joint & member method of specifying the geometry of the frame.
- Completely flexible way to specify how the X, Y & Z degrees of freedom for joints are connected to the boundary and how the members are connected to the joints (three degrees of freedom are available at each end of each member).
- Apply forces and moments directly to joints. You can also specify joint start temperatures.
- All loads can have dead, live, roof live, snow, wind, seismic and earth types.
- Apply concentrated and distributed loads to beams. These loads can be Global or local in direction and can be applied as forces, moments or temperature changes.
- Members are easily linked to section properties. For stress analysis you can set unbraced length, slenderness factors, Cb & Cm for members.

© 2026 ENERCALC, LLC

- Loads and members can be deactivated for quick modeling of alternatives.
- Complete AISC and NDS section databases are available.
- Specify material properties used by sections.
- Specify an extensive set of load combinations.
- Complete graphics of frame with deflected shapes.
- Complete stress & deflection diagrams of members individually.
- Extensive reporting capability and control.
- Module uses beam elements and a very fast matrix solver. With each change of the input data, the frame can be completely recalculated in an instant. (For convenience with large models or when running many load combinations, this module has been enhanced with the ability to turn the Automatic Recalculation feature off, and to trigger manual recalculations only when desired.)

Note: This module is intended to be a simple alternative to complex frame programs. The intent is to provide a simple and fast tool to perform simple indeterminate analysis & member design. No P-Delta effects, exotic elements, or non-linear analysis will ever be added to this module.



Program Limitations

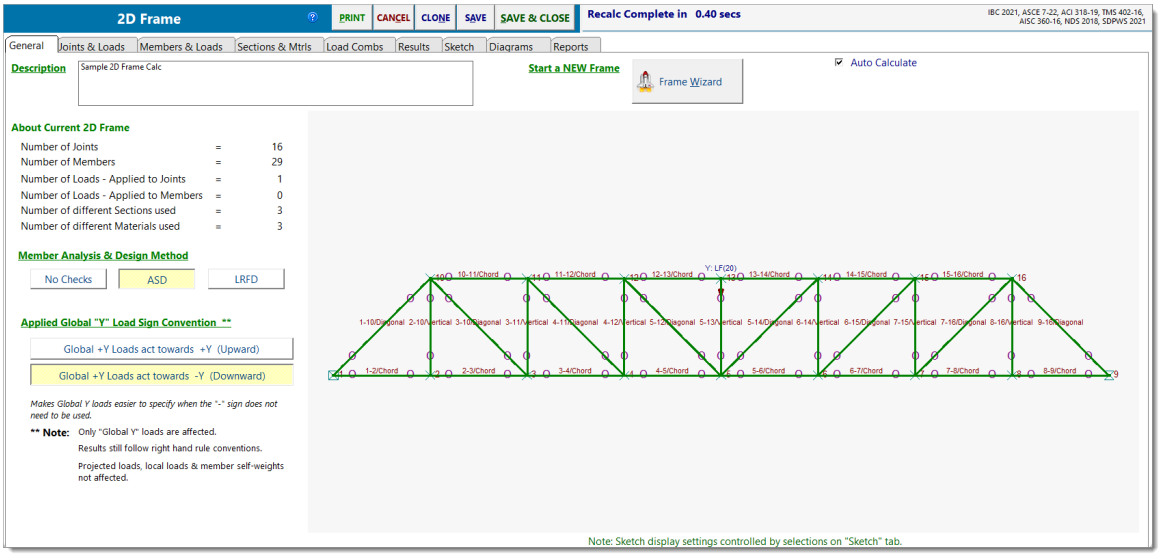
Maximum Number of Allowed:

Joints =	300
Joint Loads =	600
Beams =	600
Beam Point Loads =	600
Beam Distributed Loads =	600
Sections =	100
Materials =	50
Load Combinations =	100

© 2026 ENERCALC, LLC

13.3.2.1 General

The **General** tab is where you can define the model description, analysis method, global Y load sign behavior, and gives a live schematic of the frame.



Description: Free-text label for the calculation. This name appears in the Project Manager and on printed reports.

About Current 2D Frame: A read-only snapshot of the current model, including:

- Number of joints
- Number of members
- Number of joint loads
- Number of member loads
- Number of unique sections
- Number of unique materials

Member Analysis & Design Method: Select the design method for how members are checked. The selected method applies to all applicable members in the frame.

- *No Checks:* Analysis only. No code checks.
- *ASD:* Allowable Stress Design.
- *LRFD:* Load and Resistance Factor Design.

Applied Global "Y" Load Sign Convention: Controls how Global Y loads are interpreted when entered:

- *Global +Y loads act towards +Y (Upward):* Global +Y loads are applied in the upward direction, consistent with the global positive Y-axis.
- *Global +Y loads act towards -Y (Downward):* Global +Y loads are applied in the downward direction, aligned with gravity. This option can simplify the entry of gravity loads by removing the need to manually apply a negative sign for downward forces.

Note:

© 2026 ENERCALC, LLC

- This setting only affects applied Global Y loads. Local loads, projected loads, and self-weight are not affected.
- Analysis results continue to follow standard right-hand rule sign conventions.
- Reported reactions still align with the global +Y convention, meaning a downward load produces a positive +Y reaction, while an upward load produces a -Y reaction.

Frame Wizard: The Frame Wizard 570 can create complete 2D Frame models for many typical framing configurations and truss layouts with just a few simple inputs.

Auto Calculate: When enabled, the frame recalculates automatically after changes. When disabled, recalculation must be triggered manually.

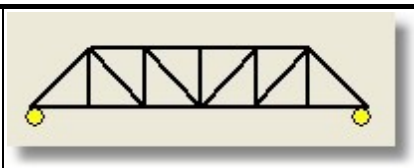
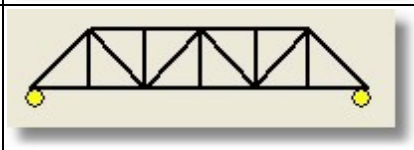
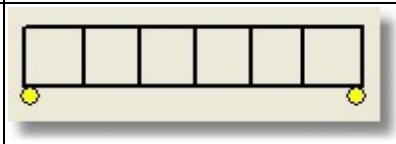
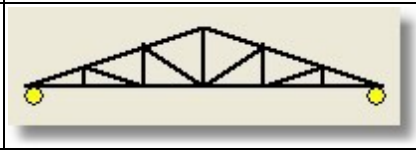
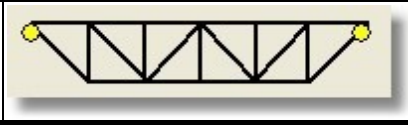
Frame Wizard

Overview

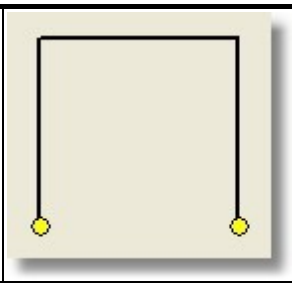
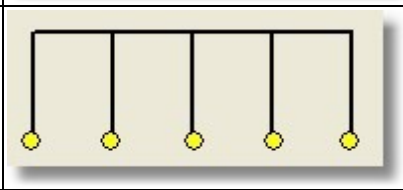
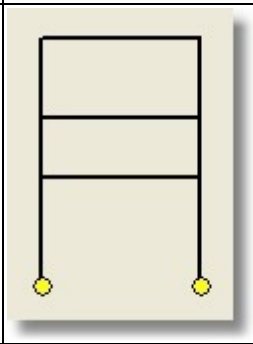
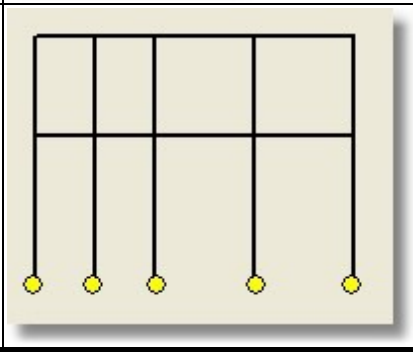
The **Frame Wizard** can create complete 2D Frame models for many typical framing configurations and truss layouts with just a few simple inputs.

The Frame Wizard is also especially helpful when you are just beginning to use this module and want to learn about how to specify joint end restraints and member end releases.

Types of Models the Frame Wizard Can Generate

Pratt Truss	
Warren Truss	
Vierendeel Truss	
Howe Truss	
Inverted Warren Truss	

© 2026 ENERCALC, LLC

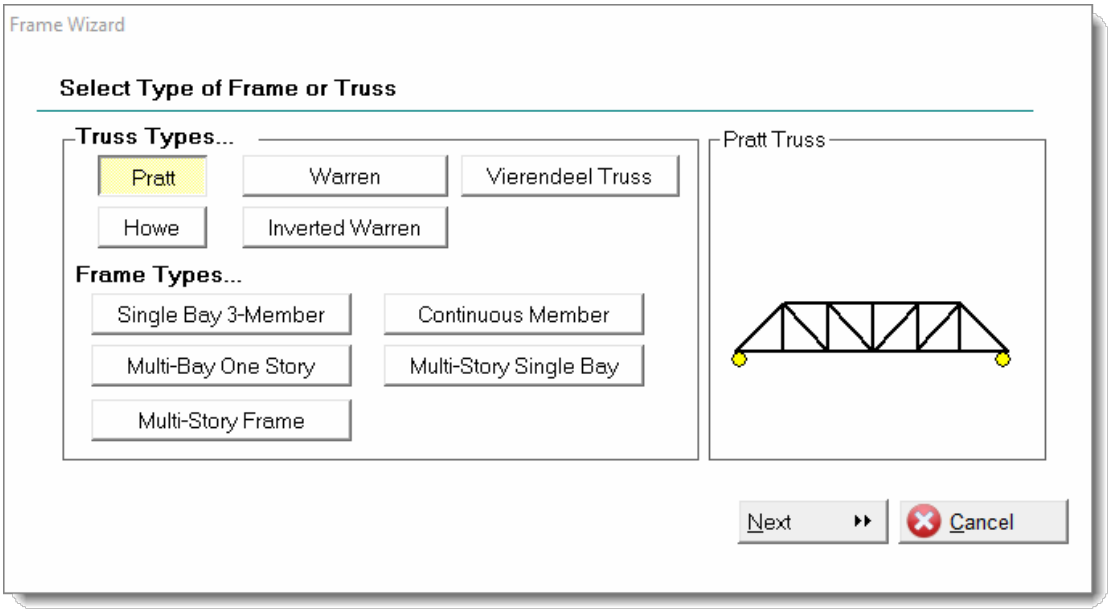
Single Bay 3-Member Frame	
Continuous Member	
Multi-Bay One Story Frame	
Multi-Story Single Bay Frame	
Multi-Story Frame	

Please send additional requests to support@enercalc.com.

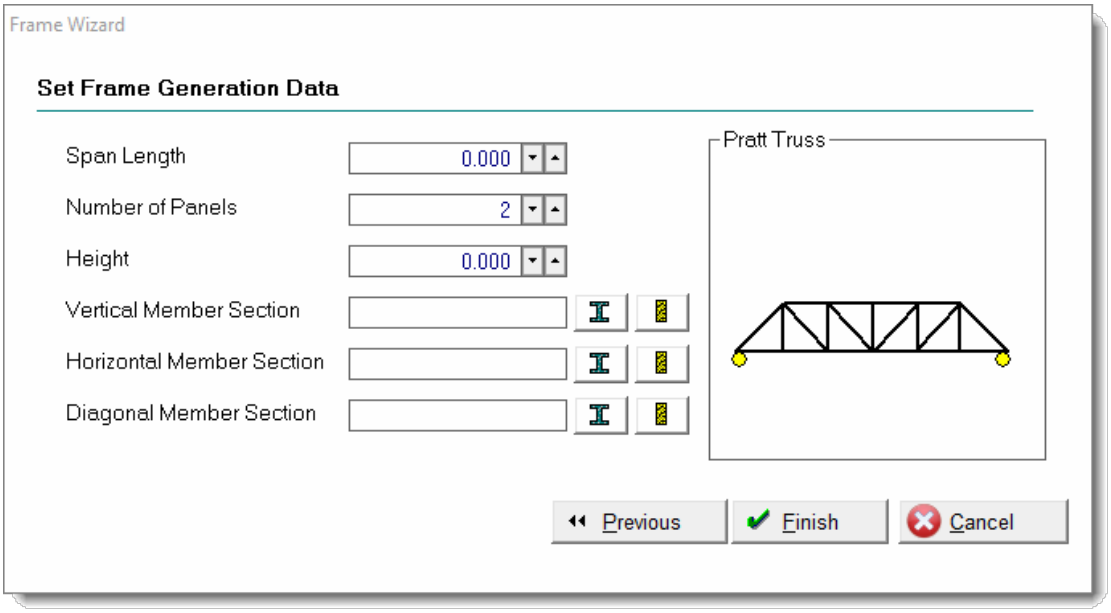
© 2026 ENERCALC, LLC

Example: Using the Frame Wizard to create a Pratt Truss

Creating a truss with the Frame Wizard is extremely simple. Just start by selecting [**Pratt**] and clicking [**Next**].



The screen you see below will be displayed.



Simply enter the overall length and height of the truss and the number of panels the truss will have. The sample image shown has 6 panels.

© 2026 ENERCALC, LLC

The next entries allow you to easily specify a member size for the three main members that are typically the same on a truss like this (Verticals, Horizontals, and Diagonals).

The button to the right of each data entry area provides access to the built-in section database in **ENERCALC SEL**.

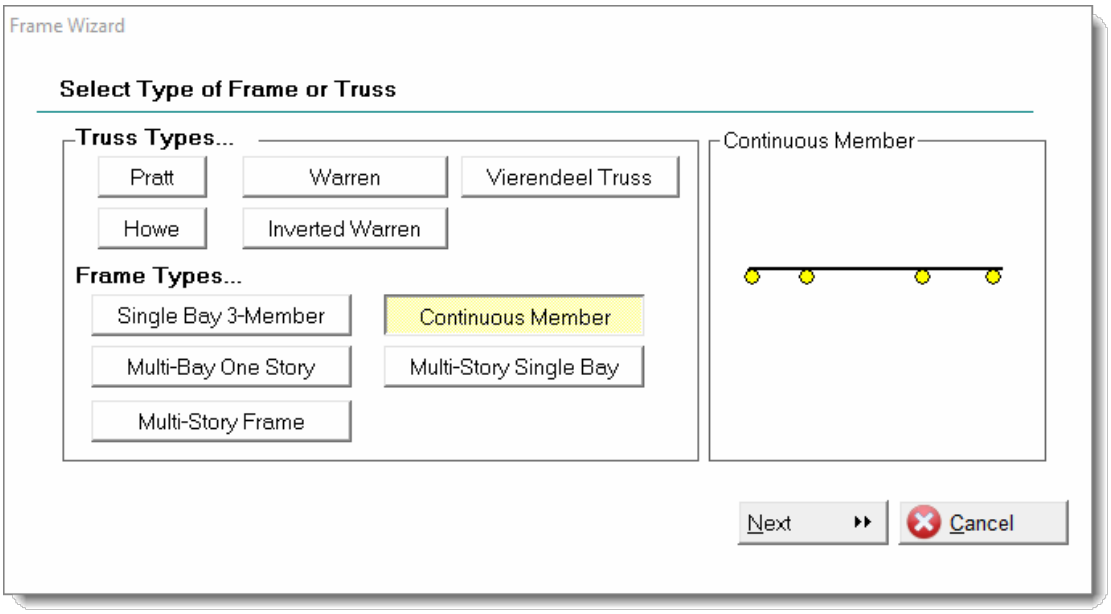
If you **DO** specify a database section, all of the associated section properties will be brought into the frame model for you.

If you **DO NOT** specify a database section, then the members will be assigned a name of "Vertical", "Diagonal" or "Chord" as a section group. Then all you need to do is enter property values for those three sections.

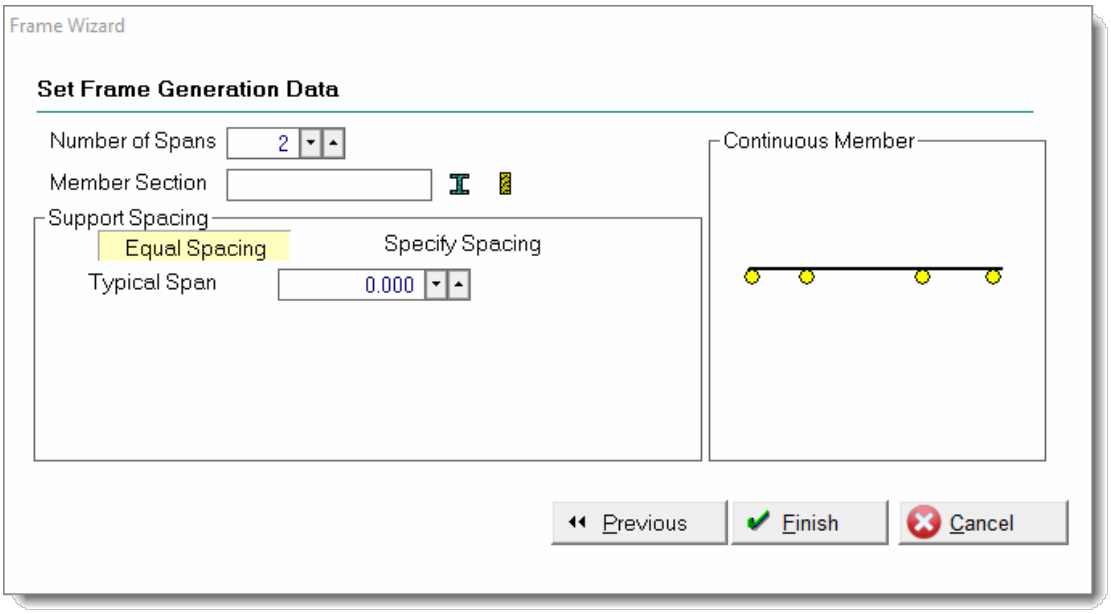
To finish just click [**Finish**]. The model will be instantly created.

Example: Using the Frame Wizard to create a Continuous Beam

The Frame Wizard is an extremely efficient way to model a continuous beam. Just start by selecting [**Continuous Member**] and then click [**Next**].



The screen you see below will be displayed:



Note: This wizard allows just one section to be entered BUT you can easily change that beam specification for each span after the model is built.

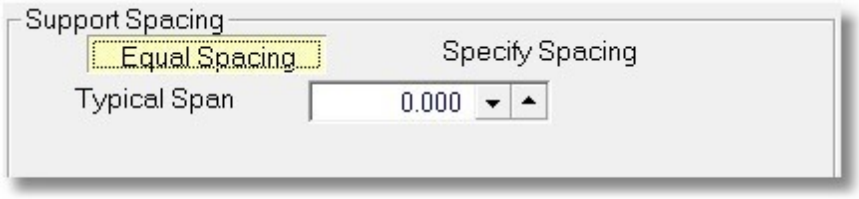
Enter the Number of Spans first.

The next entry allows you to easily specify a member section name. The button to the right of each data entry area provides access to the built-in section database in **ENERCALC SEL**.

If you **DO** specify a database section, all of the associated section properties will be brought into the continuous beam model for you.

If you **DO NOT** specify a database section then the members will be assigned a name of "Beam" as a section group. Then all you need to do is enter property values for that section.

For specification of span lengths you can either use [**Equal Spacing**]:



-or- [**Specify Spacing**] which allows you to enter the span lengths for each span.

Then just click [**Finish**], and the model will be instantly created.

© 2026 ENERCALC, LLC

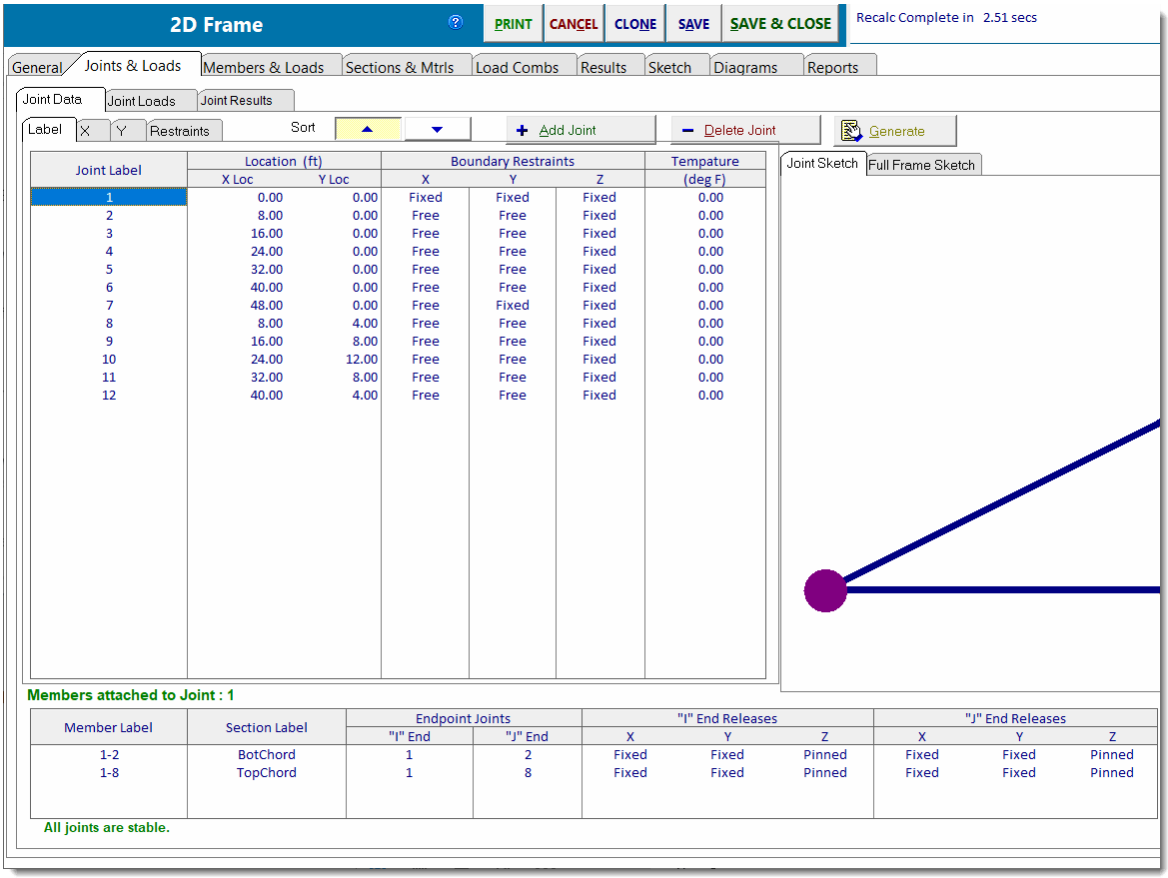
13.3.2.2 Joints & Joint Loads

Joints are the points in space where members are interconnected. Members can ONLY connect to another member or the boundary by first connecting to a joint.

IMPORTANT: See **JOINT RESTRAINTS** below.

Joints & Loads

This tab contains two sub-tabs that allow you to specify joints and loads applied to the joints, and one sub-tab dedicated to the display of results that are relevant to joints.

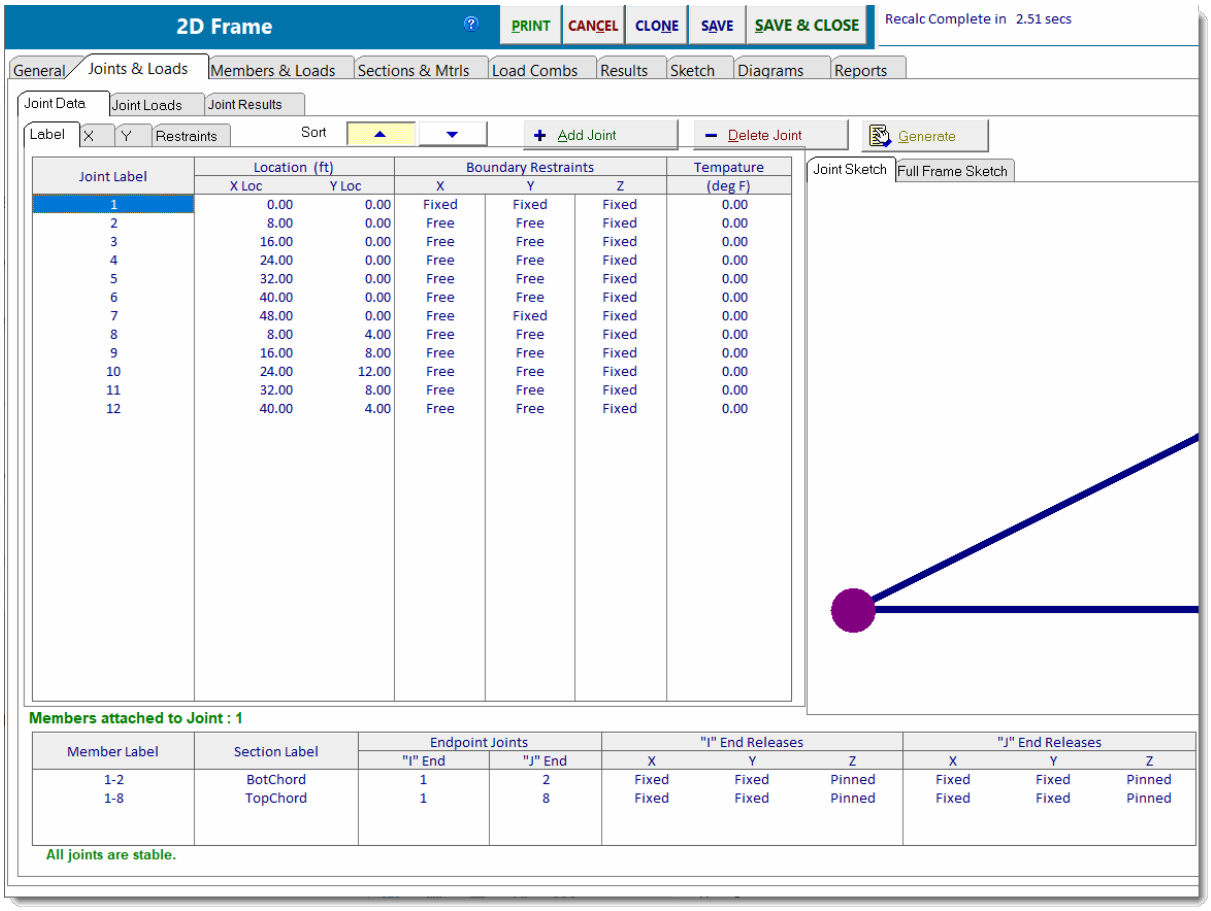


Joint Label	Location (ft)		Boundary Restraints			Tempature
	X Loc	Y Loc	X	Y	Z	(deg F)
1	0.00	0.00	Fixed	Fixed	Fixed	0.00
2	8.00	0.00	Free	Free	Fixed	0.00
3	16.00	0.00	Free	Free	Fixed	0.00
4	24.00	0.00	Free	Free	Fixed	0.00
5	32.00	0.00	Free	Free	Fixed	0.00
6	40.00	0.00	Free	Free	Fixed	0.00
7	48.00	0.00	Free	Fixed	Fixed	0.00
8	8.00	4.00	Free	Free	Fixed	0.00
9	16.00	8.00	Free	Free	Fixed	0.00
10	24.00	12.00	Free	Free	Fixed	0.00
11	32.00	8.00	Free	Free	Fixed	0.00
12	40.00	4.00	Free	Free	Fixed	0.00

Member Label	Section Label	Endpoint Joints		"I" End Releases			"J" End Releases		
		"I" End	"J" End	X	Y	Z	X	Y	Z
1-2	BotChord	1	2	Fixed	Fixed	Pinned	Fixed	Fixed	Pinned
1-8	TopChord	1	8	Fixed	Fixed	Pinned	Fixed	Fixed	Pinned

Note: The data entered into this module is linked by a database system. The effects of this linking can be seen in the screen capture seen below. Notice that the table in the upper half of the view lists available joints for which Joint Data can be viewed. As a result of the data linking, selecting Joint #1 in the upper table automatically causes the lower table to display a list of members connected to Joint #1.

© 2026 ENERCALC, LLC

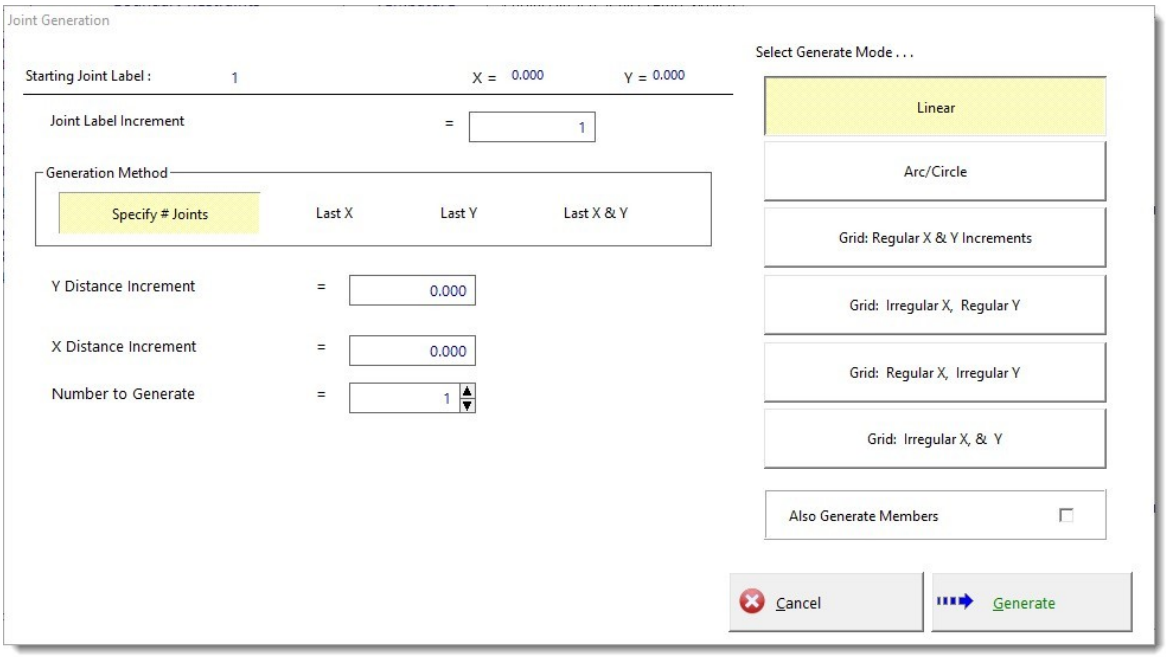


Automatic Joint Generation

You can use the [**Generate**] button to have joints automatically generated. The joint generation popup is self explanatory....just select from the mode of generation, specify a label increment, the number of joints to generate, and the x and y distance increments to use. The "Grid Irregular X & Y" can generate some very complex joint layouts.

© 2026 ENERCALC, LLC

Another powerful tool is the Also Generate Members option, which will generate members to interconnect between the joints for the generation grid or arc/circle you have specified.



Joint Restraints

These interconnections are extremely important, and when set incorrectly, they lead to the majority of the instabilities and other unexpected results when using this 2D Frame module. The connections between members and the connections to the boundary have a significant effect on structural behavior, and as such, they warrant a thorough discussion and complete understanding.

Always keep in mind the following concepts related to joint boundary restraints and the end releases of the member(s) connecting to the joint:

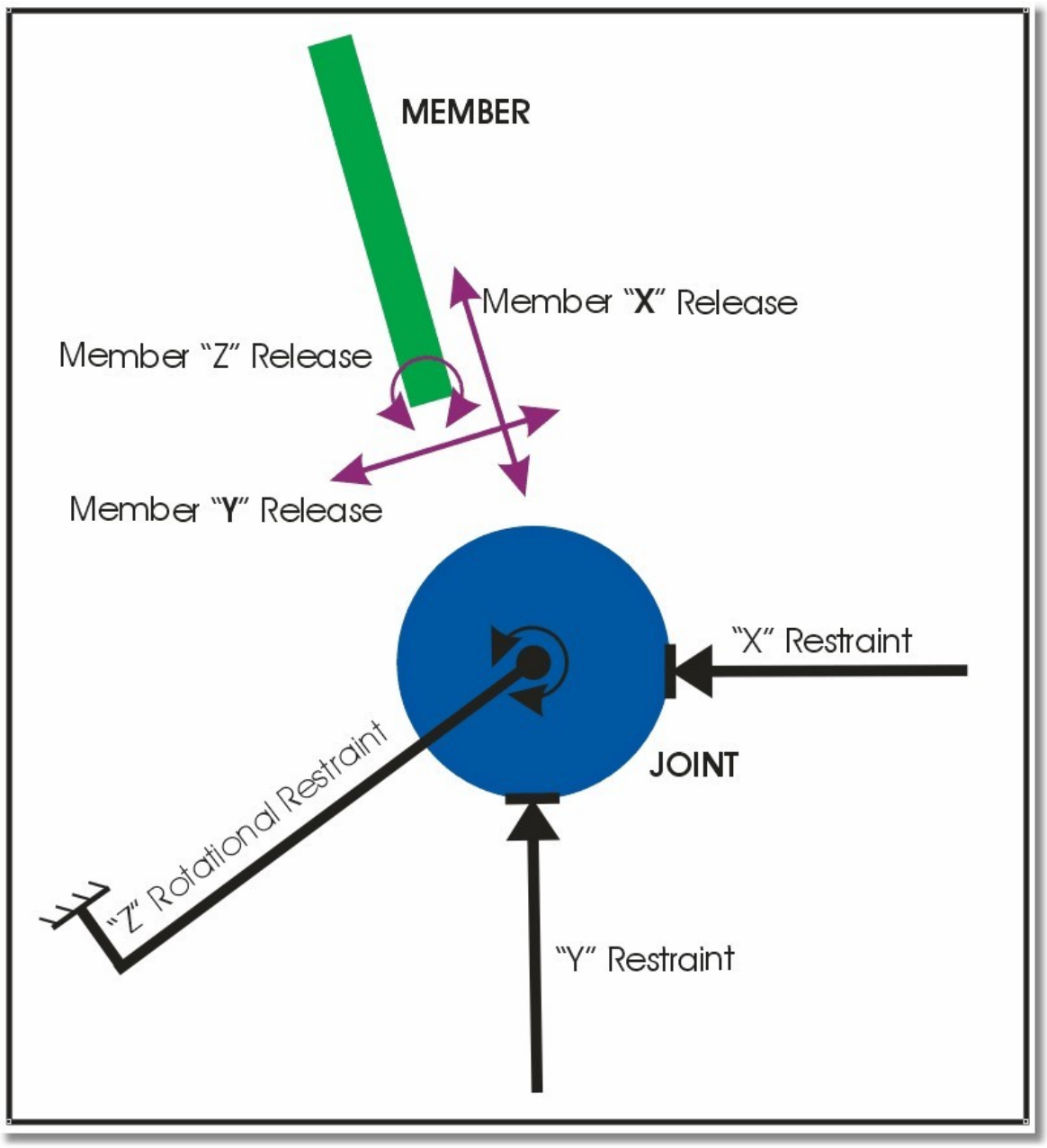
- Joints have Degrees of Freedom (DOF) associated with them.
- In the 2D Frame program, each joint has two translational and one rotational DOF.
- An individual DOF may be restrained or released.
- DOF can be defined as Boundary Restraints or as Member End Releases.
- When the degrees of freedom are defined as Boundary Restraints, they are defined with respect to the global coordinate axis system.
- When the degrees of freedom are defined as Member End Releases, they are defined with respect to the member local coordinate axis system.

The term Degrees of Freedom refers to the capability of a joint to move in the X and Y direction and to rotate about the Z axis.

© 2026 ENERCALC, LLC

An X Restraint prevents a joint from moving in the Global X direction.
A Y Restraint prevents a joint from moving in the Global Y direction.
A Z Restraint prevents a joint from rotating about the Global Z axis.

Please see the diagram below for a brief primer on joint restraints and member releases.



The **dark black** items define the joint restraints. **Joint restraints attach the joint to the boundary.** The boundary is an infinitely stiff support and is most commonly a foundation. When a joint is restrained against translation in the X and Y directions and against rotation about the Z axis, it is held firmly in place. Any forces or moments applied to it will result in

© 2026 ENERCALC, LLC

a reaction which is the boundary applying a force to the joint to keep it from moving in a particular direction.

THE MOST IMPORTANT PIECE OF INFORMATION: Each joint must be prevented from moving in/rotating about each of its three degrees of freedom, either by a joint restraint or by a positive connection to member. If a joint can translate or rotate in an unrestrained manner, then it is unstable.

NEXT MOST IMPORTANT PIECE OF INFORMATION: When a joint restraint is specified, then any connected members that are not released in that particular direction will not deflect in that direction. If a joint has an "X" restraint and a member framing into that joint does not have an "X" release, then the member is actually rigidly connected to the boundary through the joint.

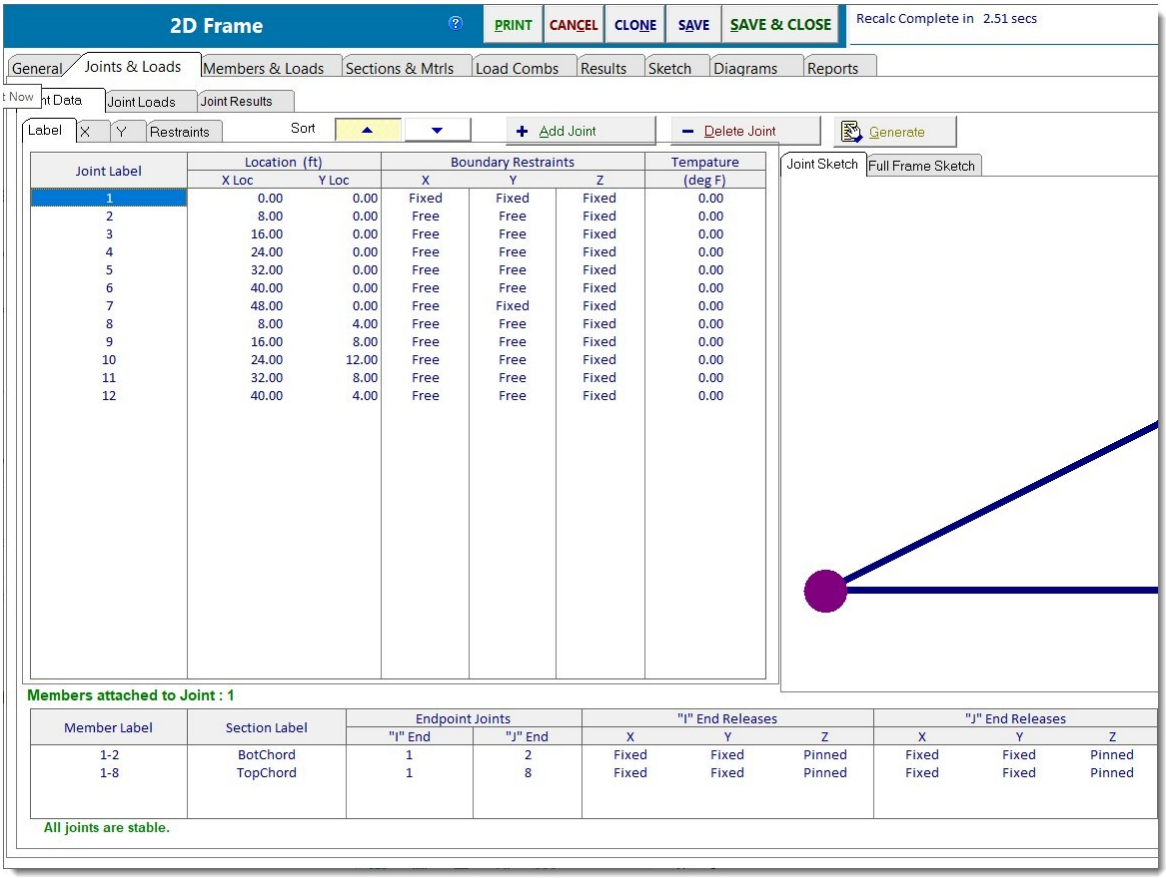
ALSO IMPORTANT: When you are modeling a frame, nearly all joints will be free from the boundary. This means that those joints will need to have their X, Y & Z degrees of freedom restrained by a connection to one or more members. Because members can only attach to each other through a joint, this is usually handled by virtue of the member having all three degrees of freedom "fixed" at its ends. BUT FOR TRUSS CONNECTIONS, THIS CAN CAUSE PROBLEMS WITH THE "Z" rotational degree of freedom. For truss models (where moments are not applied to nodes), it may be easier to achieve stability for all joints by declaring all joints as "fixed" against rotation about the Z-axis, and then "pinning" both ends of all members to prevent the transfer of moment from one member to another.

© 2026 ENERCALC, LLC

Joint Data

Joint Data Tab

This tab is where you add and edit the joints that define the member interconnections for the frame.



Joint Label

The label for a joint can be up to 25 characters long. They are CASE INSENSITIVE meaning the module internally converts all joint names to all lower case for internal usage.

Location

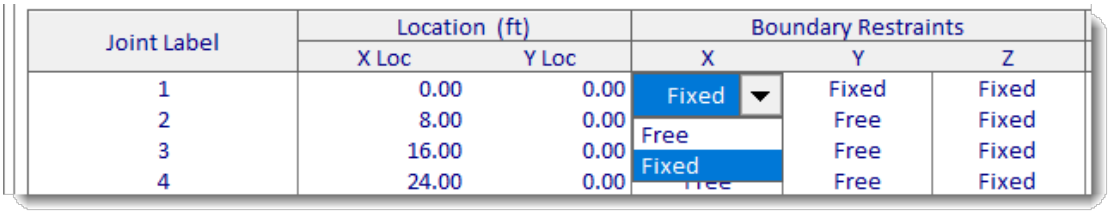
This is the X and Y location of the joint in a Cartesian coordinate system.

© 2026 ENERCALC, LLC

Boundary Restraints

These specify how the joint is connected to the boundary. The boundary is an infinitely rigid item and is typically a footing. See the Joints & Joint Loads 575 topic for more information.

When you click on these items (or use [Tab] to move between them) the entry turns into a drop-down list box with the available selections.



Temperature

This specifies the base temperature for the joint. It is only used when you are defining temperature loads for the members.

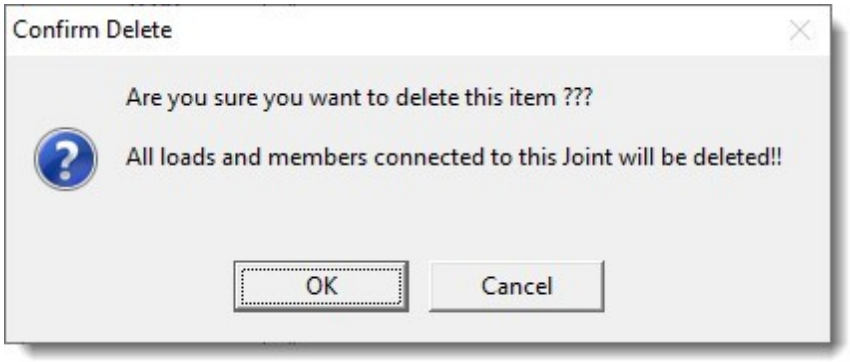
[+Add Joint]

Prompts you for the label to assign to the newly added joint. Joint labels can be up to 25 characters long.

© 2026 ENERCALC, LLC

[- Delete Joint]

Deletes the highlighted joint after your confirmation. Keep in mind that members depend upon joints for connectivity. So if there are any members connected to a joint, the members will be deleted if the joint is deleted. Any joint loads that were assigned to a joint will also be deleted if the joint is deleted.



Generate

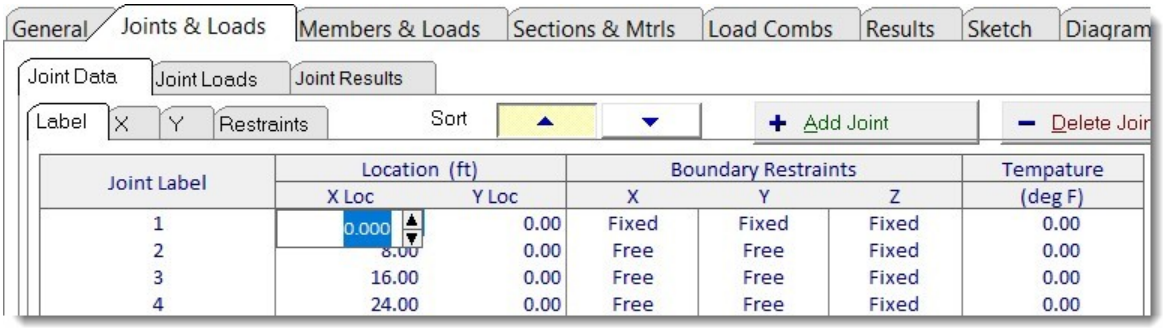
See bottom of this section.

Joint Sketch/Full Frame Sketch

The Joint Sketch displays a graphic representation of the individual joint that is currently selected in the list of joints along with the member(s) that frame into it. The Full Frame Sketch displays a graphic representation of the entire model with the currently selected joint highlighted for easy recognition.

Editing Joint Information

Click on any data item in the joint list, and the list will switch to an editing mode. See the image below where we have clicked on the "X Loc" for joint 1.



Joint Label	Location (ft)		Boundary Restraints			Tempature
	X Loc	Y Loc	X	Y	Z	(deg F)
1	0.000	0.00	Fixed	Fixed	Fixed	0.00
2	8.00	0.00	Free	Free	Fixed	0.00
3	16.00	0.00	Free	Free	Fixed	0.00
4	24.00	0.00	Free	Free	Fixed	0.00

You can type in a numeric value or use the "spin" buttons to change the value by a fixed decimal amount.

To finish the entry either press [**Tab**], [**Enter**] or click on another data item in the list.

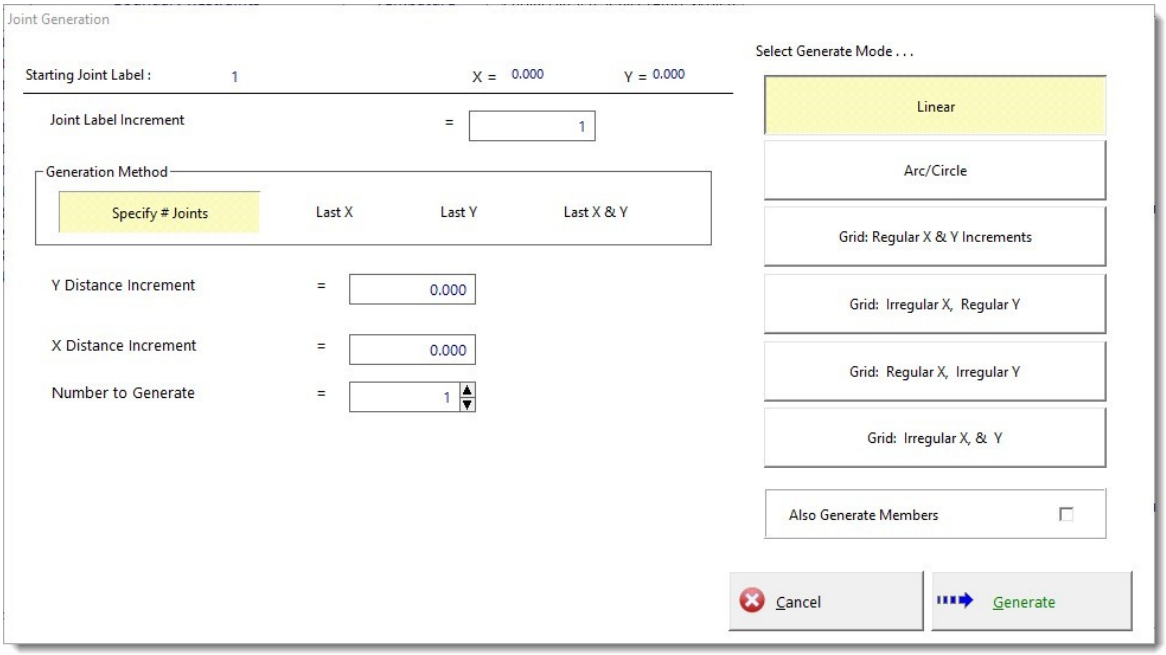
© 2026 ENERCALC, LLC

When your entry is completed the entire frame will be recalculated if the Auto Calculate option is selected.

Automatic Joint Generation

You can use the [**Generate**] button to initiate the automatic generation of joints. Just select from the mode of generation, and you can specify the label and the x and y distance increments to use. The "Grid Irregular X & Y" can generate some very complex joint layouts.

Another powerful tool is the Also Generate Members option, which will generate members to interconnect the joints for the generation grid or arc/circle you have specified.

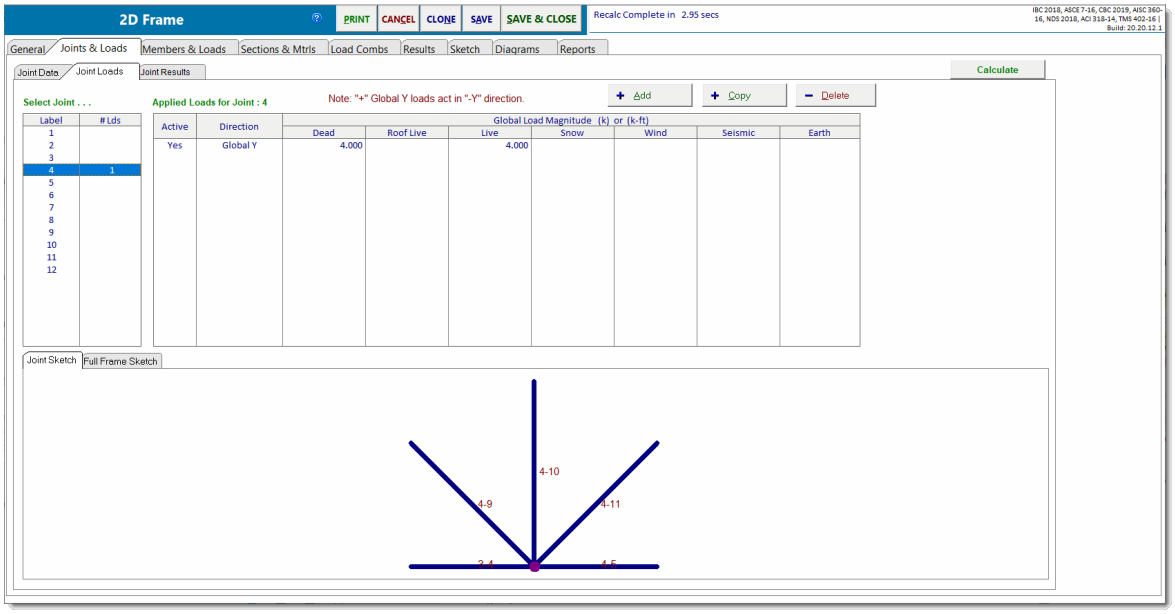


© 2026 ENERCALC, LLC

Joint Loads

Joint Loads

This tab provides the input locations for all loads applied to the joints in the frame.



Select Joint: list is where you select the joint for which you wish to view/modify loads. This selection controls what is visible in the other two sections.

Applied Loads for Joint: is where you assign and edit the loads applied to that joint.

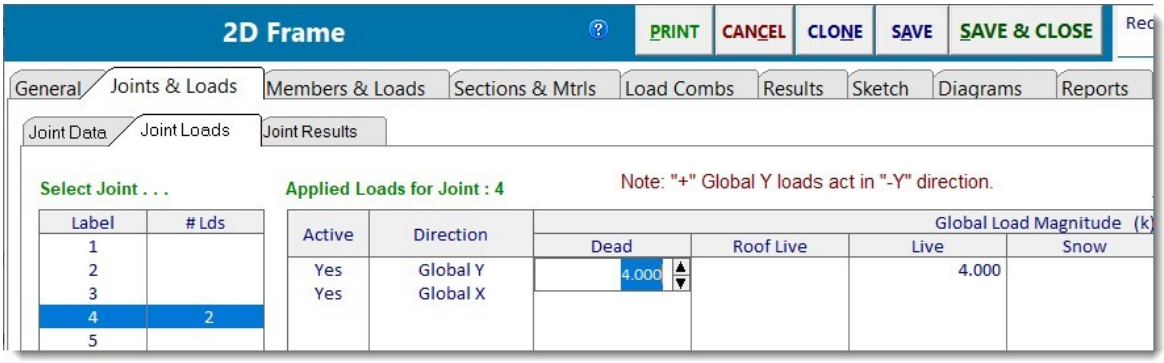
Adding & Deleting Loads

To add a joint load just click the [**Add**] button and a new load entry line will be added at the bottom of the list for the currently selected joint.
To delete a load just click on the load line you wish to delete and click the [**Delete**] button.

© 2026 ENERCALC, LLC

Editing Joint Load Information

Click on any data item in the joint load list, and it will switch into an editing mode. See the image below where we have clicked on the Dead load column in the first line in the list of loads associated with joint 4.



You can type in a numeric value or use the "spin" buttons to change the value by a fixed decimal amount.

To finish the entry either press [**Tab**], [**Enter**] or click on another data item in the list.

When your entry is completed the entire frame will be recalculated if the Auto Calculate option is selected.

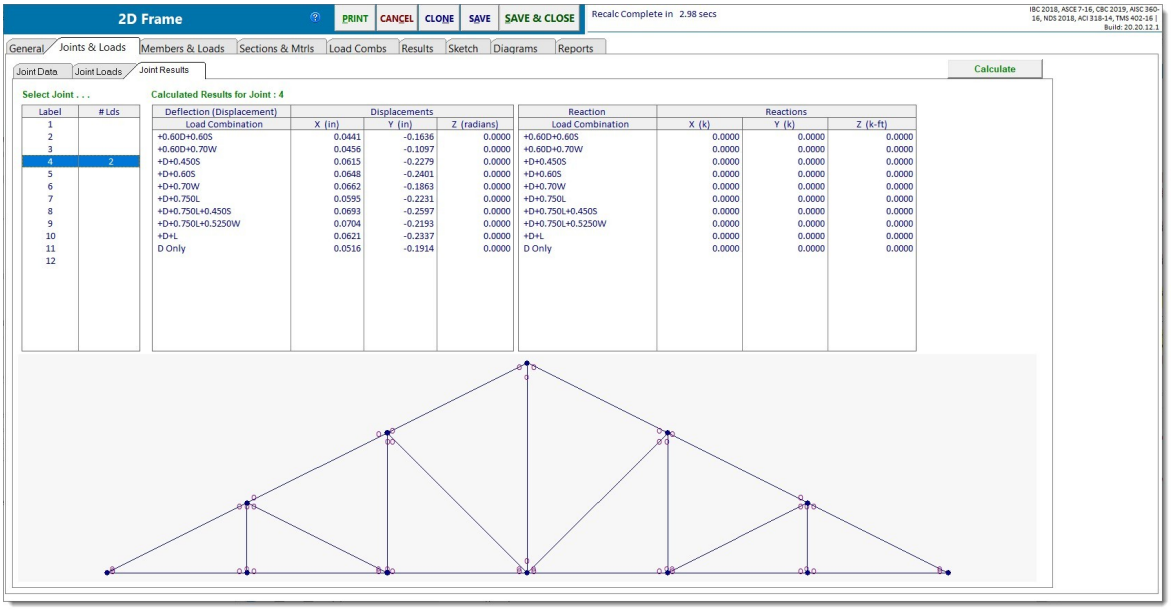
Note: If a load is applied in the same direction that a Joint Restraint is specified, that load will immediately be "absorbed" by the joint restraint, so it will not have any effect on the frame.

© 2026 ENERCALC, LLC

Joint Results

Joint Results

This tab displays the calculated displacements/rotations and reactions for the joint selected in the Select Joint list.



Label	# Lds
1	
2	
3	
4	2
5	
6	
7	
8	
9	
10	
11	
12	

Deflection (Displacement)	Displacements		
Load Combination	X (in)	Y (in)	Z (radians)
+0.60D+0.60S	0.0441	-0.1636	0.0000
+0.60D+0.70W	0.0456	-0.1097	0.0000
+D+0.450S	0.0615	-0.2279	0.0000
+D+0.60S	0.0648	-0.2401	0.0000
+D+0.70W	0.0662	-0.1863	0.0000
+D+0.750L	0.0595	-0.2231	0.0000
+D+0.750L+0.450S	0.0693	-0.2597	0.0000
+D+0.750L+0.5250W	0.0704	-0.2193	0.0000
+D+L	0.0621	-0.2337	0.0000
D Only	0.0516	-0.1914	0.0000

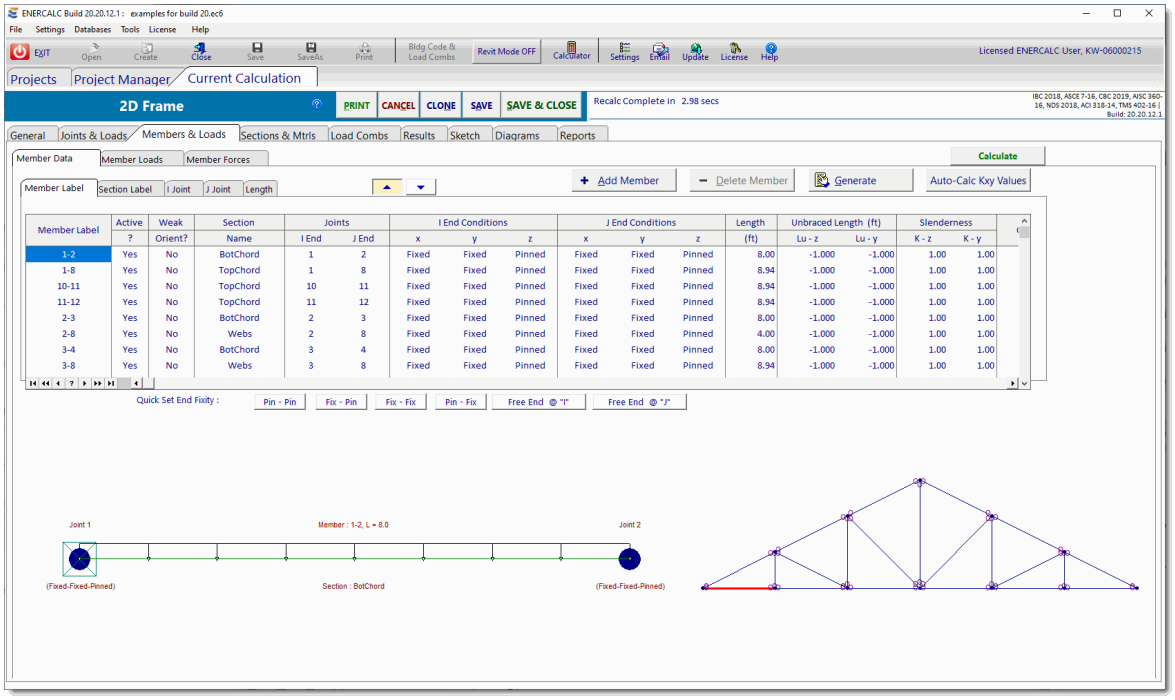
Reaction	Reactions		
Load Combination	X (k)	Y (k)	Z (k-ft)
+0.60D+0.60S	0.0000	0.0000	0.0000
+0.60D+0.70W	0.0000	0.0000	0.0000
+D+0.450S	0.0000	0.0000	0.0000
+D+0.60S	0.0000	0.0000	0.0000
+D+0.70W	0.0000	0.0000	0.0000
+D+0.750L	0.0000	0.0000	0.0000
+D+0.750L+0.450S	0.0000	0.0000	0.0000
+D+0.750L+0.5250W	0.0000	0.0000	0.0000
+D+L	0.0000	0.0000	0.0000
D Only	0.0000	0.0000	0.0000

13.3.2.3 Members & Member Loads

"Member" is the generic term that describes the beams, columns, struts, braces, diagonals, hangers, and other structural entities that make up the truss or frame. This module uses a general stiffness matrix approach to solve for forces and deflections. Members are modeled by one-dimensional (length-only) entities that offer resistance to axial loads, bending moment about an axis perpendicular to the plane of the frame, and shear forces acting in the plane of the frame.

© 2026 ENERCALC, LLC

Members & Loads



Member End Releases

These interconnections lead to the greatest potential for instabilities and other unexpected effects from the use of the 2D module. The connections between members and the connections to the boundary have a significant effect on structural behavior, and as such, they warrant a thorough discussion and complete understanding. Always keep in mind the following concepts related to joint boundary restraints and the end releases of the member(s) connecting to the joint:

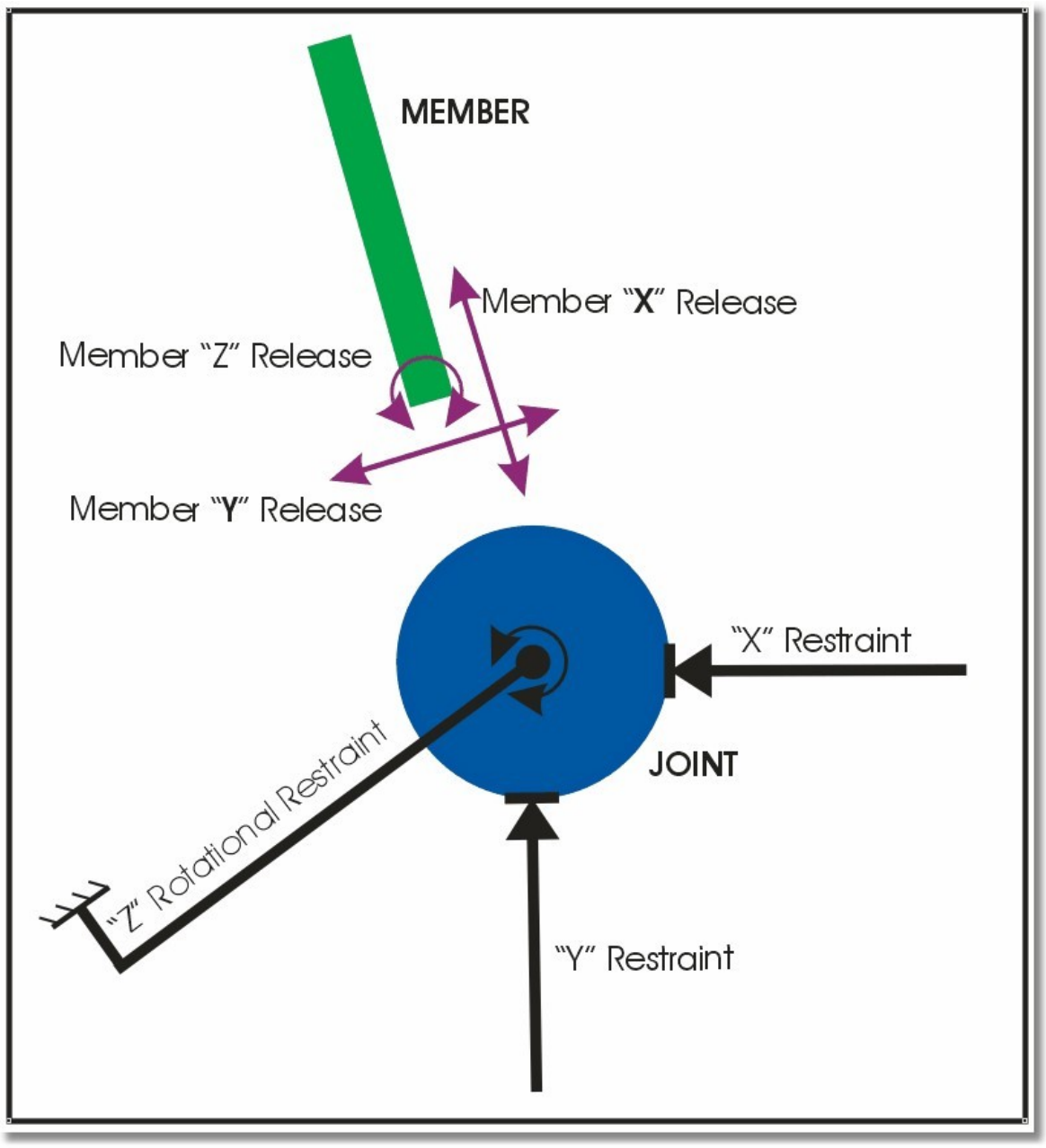
- Joints have Degrees of Freedom (DOF) associated with them.
- In the 2D Frame Analysis program, each joint has two translational and one rotational DOF.
- An individual DOF may be restrained or released.
- DOF can be defined as Boundary Restraints or as Member End Releases.
- When the degrees of freedom are defined as Boundary Restraints, they are defined with respect to the global coordinate axis system.
- When the degrees of freedom are defined as Member End Releases, they are defined with respect to the member local coordinate axis system.

The term Degrees of Freedom refers to the capability of a joint to move in the X and Y direction and to rotate about the Z axis.

An X Restraint prevents a joint from moving in the Global X direction.
A Y Restraint prevents a joint from moving in the Global Y direction.
A Z Restraint prevents a joint from rotating about the Global Z axis.

© 2026 ENERCALC, LLC

Please see the diagram below for a brief primer on joint restraints and member releases.



The **dark black** items define the joint restraints. **Joint restraints attach the joint to the boundary.** The boundary is an infinitely stiff support and is most commonly a foundation. When a joint is restrained against translation in the X and Y directions and against rotation about the Z axis, it is held firmly in place. Any forces or moments applied to it will result in a reaction which is the boundary applying a force to the joint to keep it from moving in a particular direction.

THE MOST IMPORTANT PIECE OF INFORMATION: Each joint must be prevented from moving in/rotating about each of its three degrees of freedom, either by a joint restraint or

© 2026 ENERCALC, LLC

by a positive connection to member. If a joint can translate or rotate in an unrestrained manner, then it is unstable.

NEXT MOST IMPORTANT PIECE OF INFORMATION: When a joint restraint is specified, then any connected members that are not released in that particular direction will not deflect in that direction. If a joint has an "X" restraint and a member framing into that joint does not have an "X" release, then the member is actually rigidly connected to the boundary through the joint.

ALSO IMPORTANT: When you are modeling a frame, nearly all joints will be free from the boundary. This means that those joints will need to have their X, Y & Z degrees of freedom restrained by a connection to one or more members. Because members can only attach to each other through a joint, this is usually handled by virtue of the member having all three degrees of freedom "fixed" at its ends. BUT FOR TRUSS CONNECTIONS, THIS CAN CAUSE PROBLEMS WITH THE "Z" rotational degree of freedom. For truss models (where moments are not applied to nodes), it may be easier to achieve stability for all joints by declaring all joints as "fixed" against rotation about the Z-axis, and then "pinning" both ends of all members to prevent the transfer of moment from one member to another.

Member Data

Member Data

Members always span between two joints. The Member End Conditions tell the module how each of the three degrees of freedom at each end of the member are attached to the joint. To connect two members rigidly together you must specify the two connecting ends as "Unreleased" conditions for all three degrees of freedom. That locks each beam end to the joint.

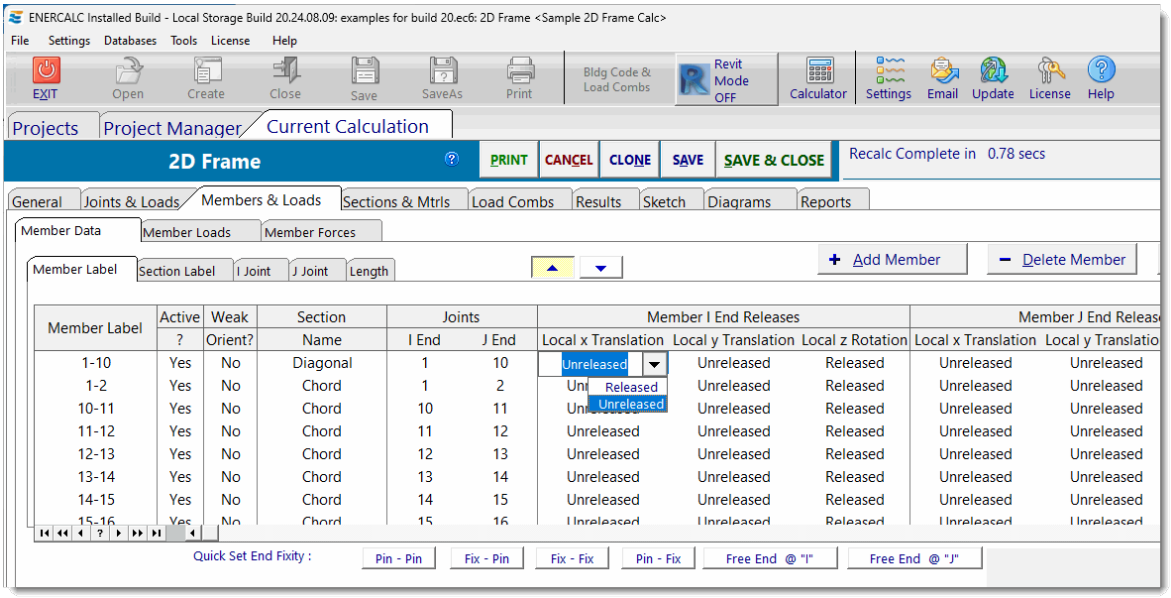
For each member you can specify the section to use for its properties, and values to be used when stress analysis is performed (unbraced length, slenderness factor, Cm and Cb).

You can also set the member to be inactive to test force distributions and stresses for alternate framing conditions.

© 2026 ENERCALC, LLC

Editing Member Information

Click on any data item in the member list, and the list will switch into an editing mode. See the image below where we have clicked on the "X" degree of freedom for the I: end of member 1-2.



You can type in a numeric value or use the "spin" buttons to change the value by a fixed decimal amount.

To finish the entry, press [**Tab**] or click on another data item in the list.

Note: Each column in the Member Data table can have its own type of editing mode. The Active column turns into a checkbox for yes/no selections. The Section Name column turns into a button so you can select or add a section from the Section list. See descriptions for each item below.

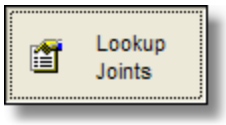
Adding & Deleting Members

To add a member click the [**Add Member**] button. You will be prompted for three pieces of information:

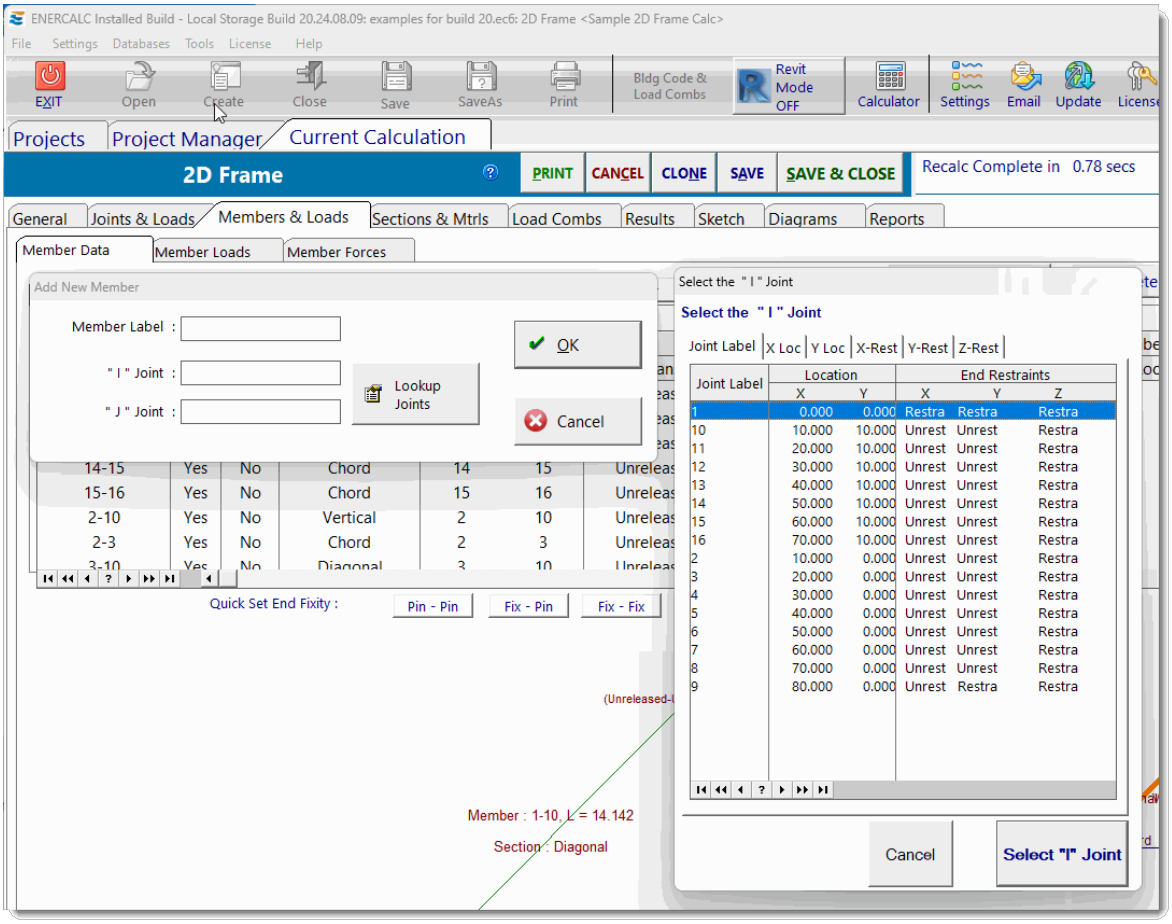
1) A label for the member.
2) The member's "I" joint number.
3) The member's "J" joint number.

© 2026 ENERCALC, LLC

In the screen capture below you will notice a button to the right of the joint label entries.



Just click in the "I" or "J" joint entry and click the [**Lookup Joints**] button . This will display a window where you can scroll through the list of created joints and select one. Simply click on one of the listed joints and then click [**Select**].



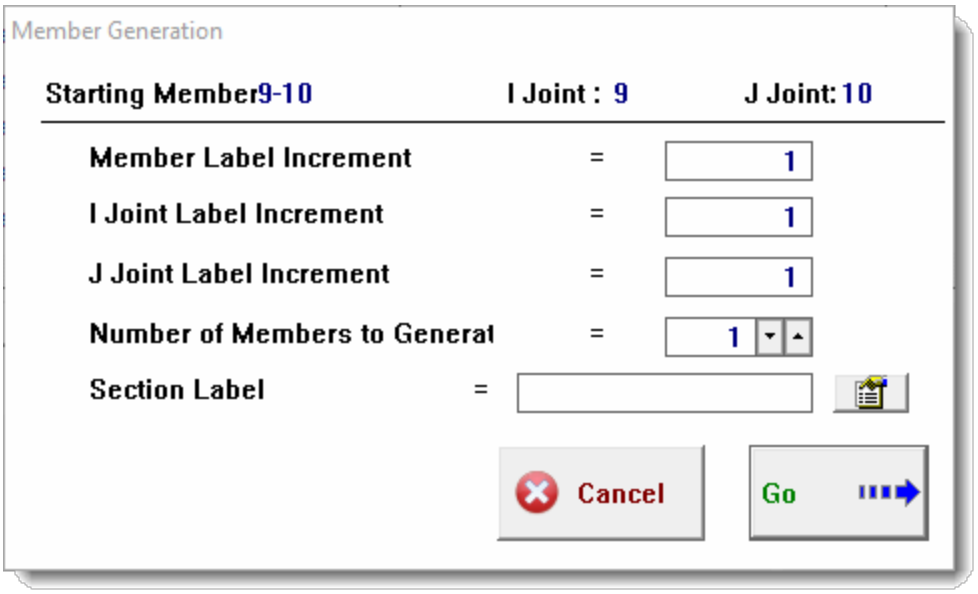
To delete a member, select the member and click [**Delete Member**]. After you approve the deletion, that member and all loads applied to that member will be deleted.

© 2026 ENERCALC, LLC

Generating Members

You can use the [**Generate**] button to initiate the automatic generation of members. This is a simple process that automates member creation by generating members between existing nodes in the specified order.

Note: This tool will generate new members, but it will not generate new nodes.



Detailed Item Descriptions (*see 2 screen captures below*)

Member Label

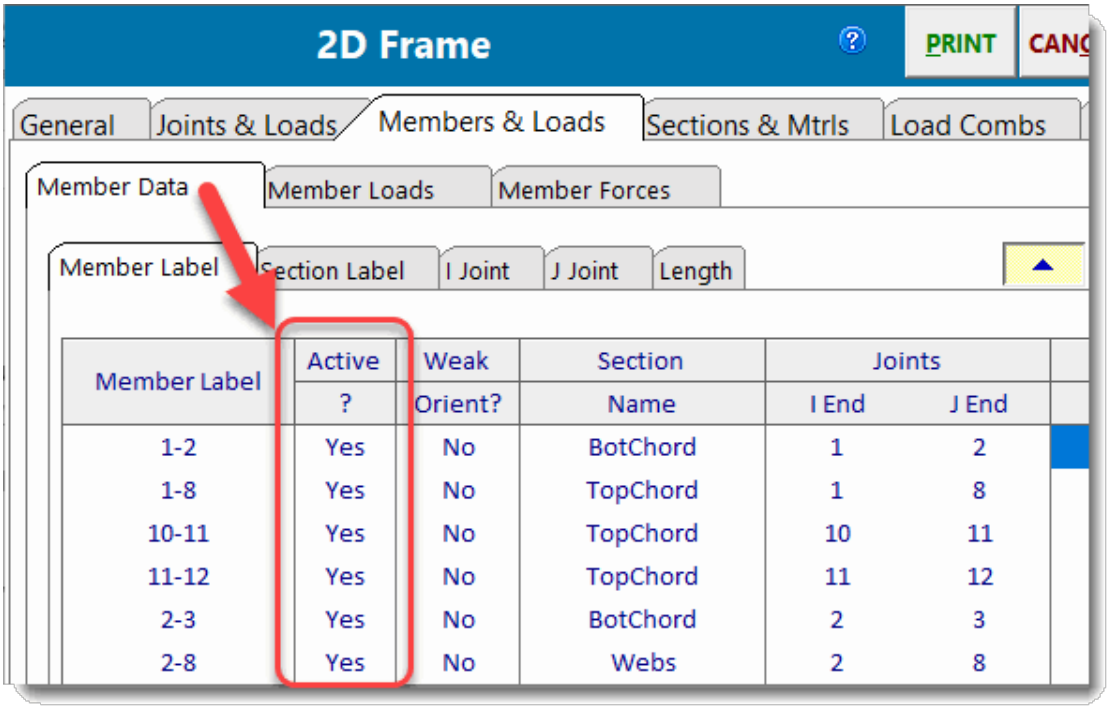
This label can be any combination of letters and numbers. It is common to use numbers for joints and letters for members, but a very convenient member labeling convention is to use the I and J joint numbers separated by a dash. So the member between joint 5 and 12 would be labeled "**5-12**".

The member label can ONLY be specified when Adding the member. After that point, the name can't be changed unless the member is deleted and added again.

© 2026 ENERCALC, LLC

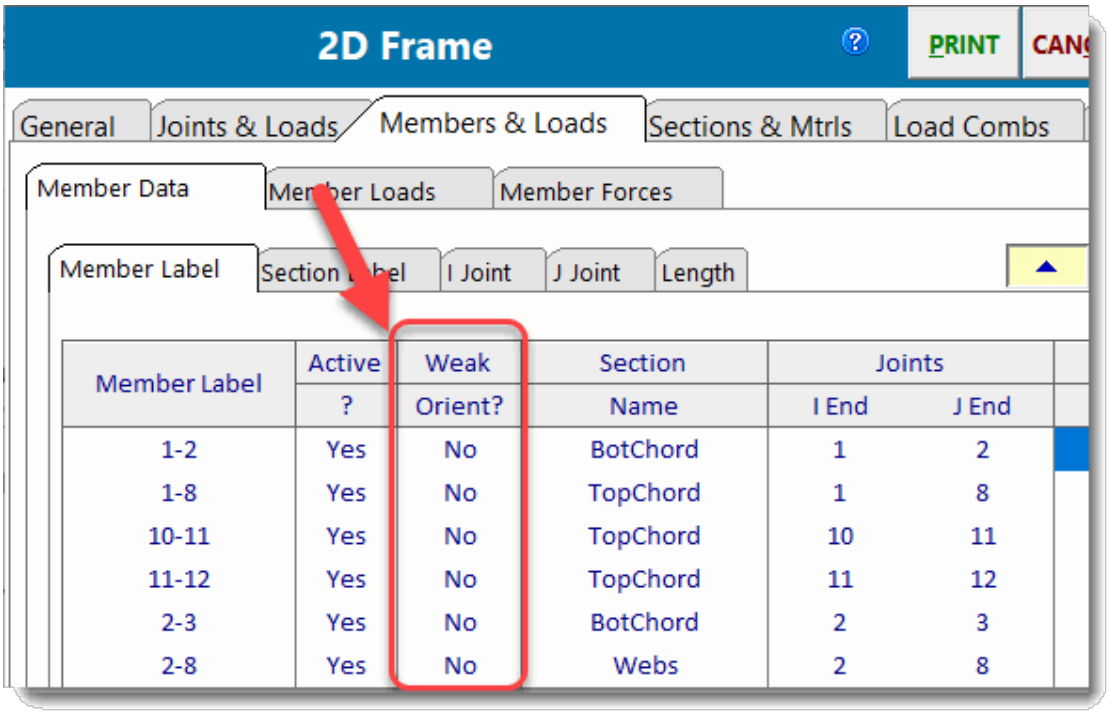
Active

This item is a yes/no selection. When you click on it or [**Tab**] to it, the entry changes to a checkbox. When the box is checked, the member will be considered active, meaning that it will contribute stiffness to the model. When the box is not checked, the member will be considered inactive, meaning that it will be ignored in the analysis and will not contribute stiffness to the model.



Weak Orientation

This item allows the member to be rotated so that its strong axis is **in** the plane of the truss or frame. By default the member is assumed to be oriented so that its strong axis is **perpendicular to** the plane of the truss or frame.

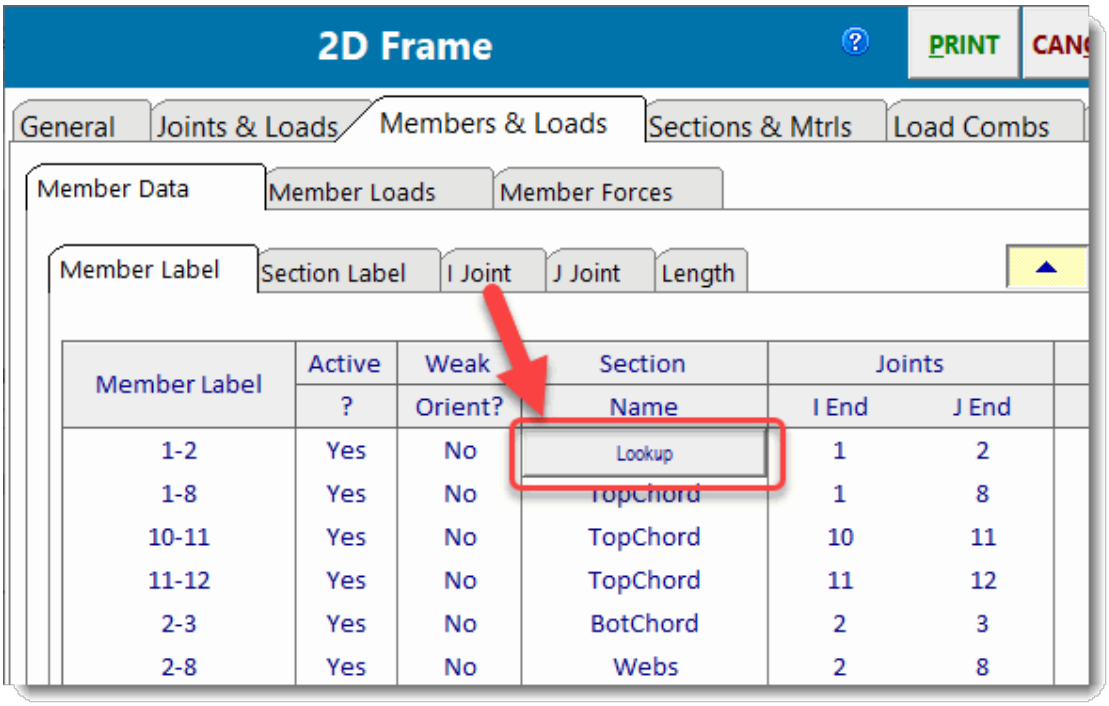


Member Label	Active ?	Weak Orient?	Section Name	Joints I End	Joints J End
1-2	Yes	No	BotChord	1	2
1-8	Yes	No	TopChord	1	8
10-11	Yes	No	TopChord	10	11
11-12	Yes	No	TopChord	11	12
2-3	Yes	No	BotChord	2	3
2-8	Yes	No	Webs	2	8

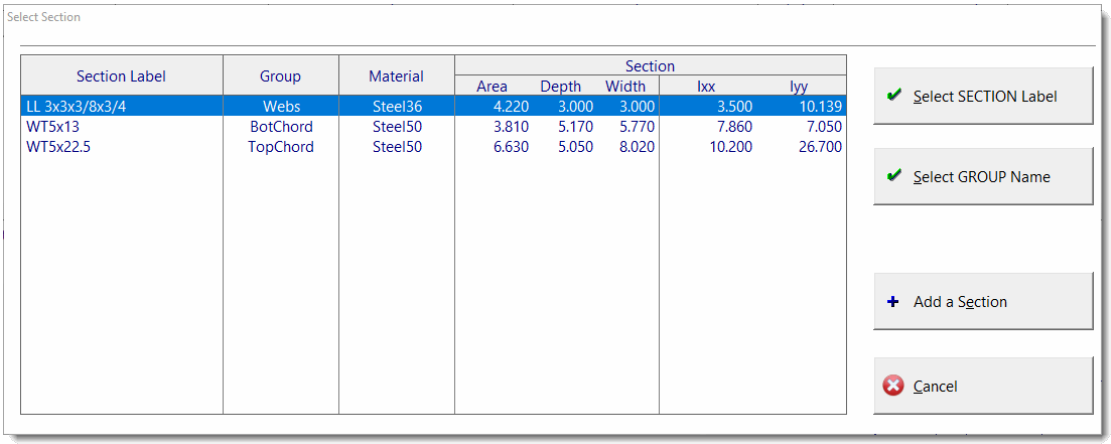
© 2026 ENERCALC, LLC

Section Name

This establishes what section is to be used for that member. When you click on it or [**Tab**] to it, the entry changes to a button labeled Lookup.



When you click the [Lookup] button, a selection window appears as shown below:



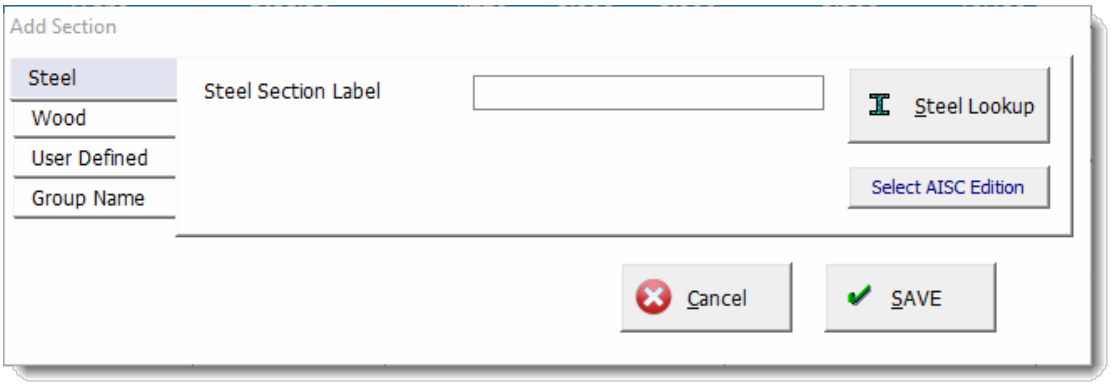
On this window you can scroll through the sections you have already added to the section list.

© 2026 ENERCALC, LLC

Note that the sections are listed with both Section labels and Group labels. A Group label is used when you want to have more than one member have the same section properties. In the screen capture above, notice that there are groups named "Chord", "Diagonal", and "Vertical", all of which have the same section assigned to them. This offers two convenient benefits:

- First, it provides a handy way to manage multiple sets of members, some of which may have the same section.
- Second, by assigning a group name to the appropriate members in the model, it is possible to change the AISC or NDS section assigned to all members of that group by simply assigning a new section to the group, rather than having to assign the new section to many individual members.

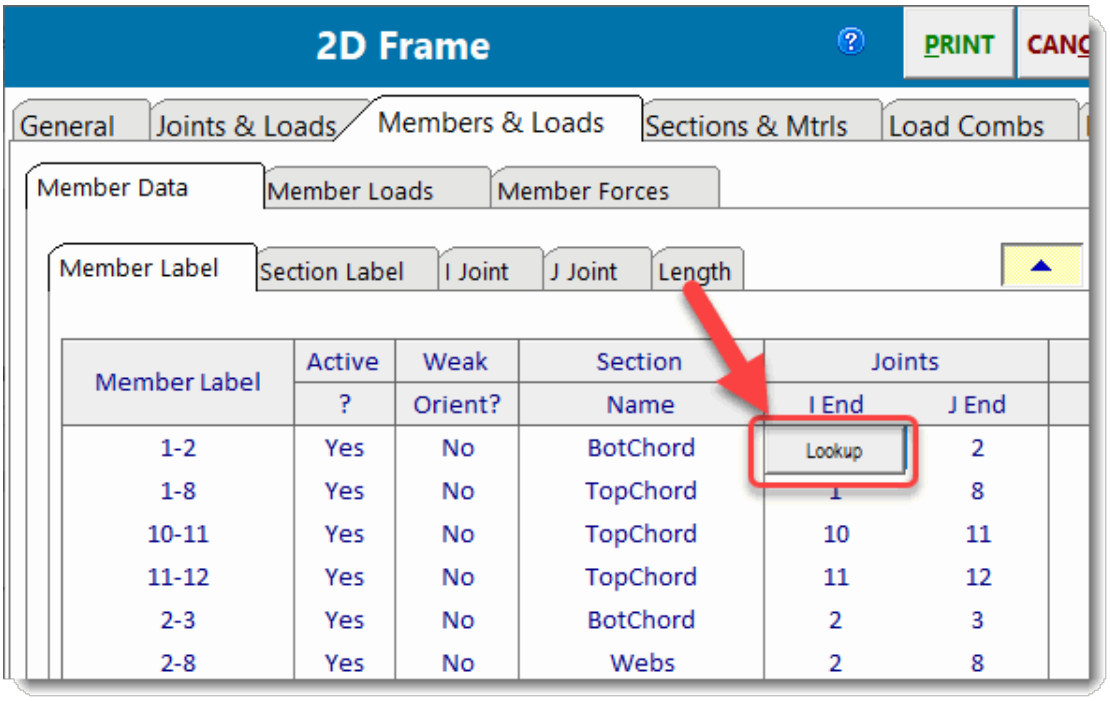
The [**+Add Section**] button provides access to the built-in section property databases. It displays the screen below, which allows you to either type in a typical section name or click a button and display the database to select the desired section from the database.



© 2026 ENERCALC, LLC

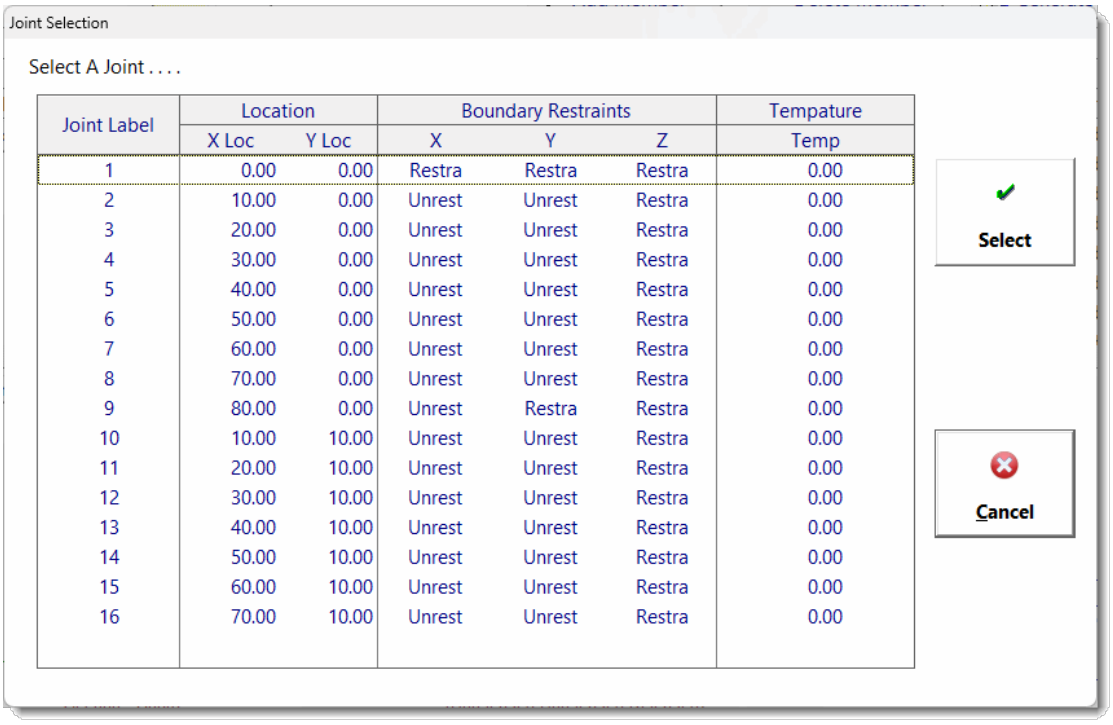
Joints

These two columns let you specify and change the I and J end joints of a member. When you click on these columns, the entry will change to a [**Lookup**] button as shown below:



© 2026 ENERCALC, LLC

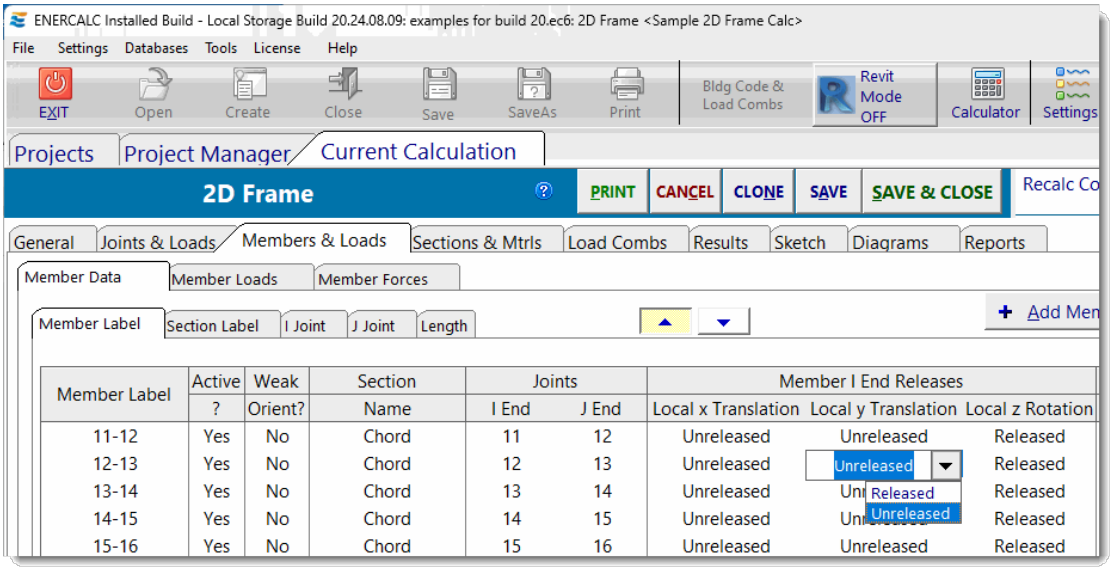
Clicking that button displays a window where you can scroll through the joint list and click to identify the desired joint.



Joint Label	Location		Boundary Restraints			Tempature
	X Loc	Y Loc	X	Y	Z	Temp
1	0.00	0.00	Restra	Restra	Restra	0.00
2	10.00	0.00	Unrest	Unrest	Restra	0.00
3	20.00	0.00	Unrest	Unrest	Restra	0.00
4	30.00	0.00	Unrest	Unrest	Restra	0.00
5	40.00	0.00	Unrest	Unrest	Restra	0.00
6	50.00	0.00	Unrest	Unrest	Restra	0.00
7	60.00	0.00	Unrest	Unrest	Restra	0.00
8	70.00	0.00	Unrest	Unrest	Restra	0.00
9	80.00	0.00	Unrest	Restra	Restra	0.00
10	10.00	10.00	Unrest	Unrest	Restra	0.00
11	20.00	10.00	Unrest	Unrest	Restra	0.00
12	30.00	10.00	Unrest	Unrest	Restra	0.00
13	40.00	10.00	Unrest	Unrest	Restra	0.00
14	50.00	10.00	Unrest	Unrest	Restra	0.00
15	60.00	10.00	Unrest	Unrest	Restra	0.00
16	70.00	10.00	Unrest	Unrest	Restra	0.00

I & J End Conditions

These six columns allow you to specify how the ends of the member are attached to the joints. When you click on or [**Tab**] to one of these columns, the entry will change to a drop-down list box offering appropriate fixity options:



Member Label	Active	Weak	Section	Joints		Member I End Releases		
	?	Orient?	Name	I End	J End	Local x Translation	Local y Translation	Local z Rotation
11-12	Yes	No	Chord	11	12	Unreleased	Unreleased	Released
12-13	Yes	No	Chord	12	13	Unreleased	Unreleased	Released
13-14	Yes	No	Chord	13	14	Unreleased	Unreleased	Released
14-15	Yes	No	Chord	14	15	Unreleased	Unreleased	Released
15-16	Yes	No	Chord	15	16	Unreleased	Unreleased	Released

© 2026 ENERCALC, LLC

In order to most efficiently describe the I & J End Conditions, it helps to introduce the concept of the member local axis system. Each member can be thought of as having its own x, y, and z coordinate axes that are mutually perpendicular and follow the right-hand rule. The orientation of the member local axes can be determined as follows:

1. The local x axis is always parallel to a vector from the "I" node to the "J" node. (This axis is parallel to the longitudinal axis of the member.)
2. The local z axis is always parallel to the Global Z axis and points out of the plane of the screen.
3. The local y axis can be found by taking the vector cross product of local z cross local x. (Envision using the right hand to rotate the local z axis into the local x axis, and the right thumb will automatically indicate the positive direction of the local y axis.)

Note: It is important to understand that I & J End Conditions are defined with respect to the member local axes, not the Global axis system.

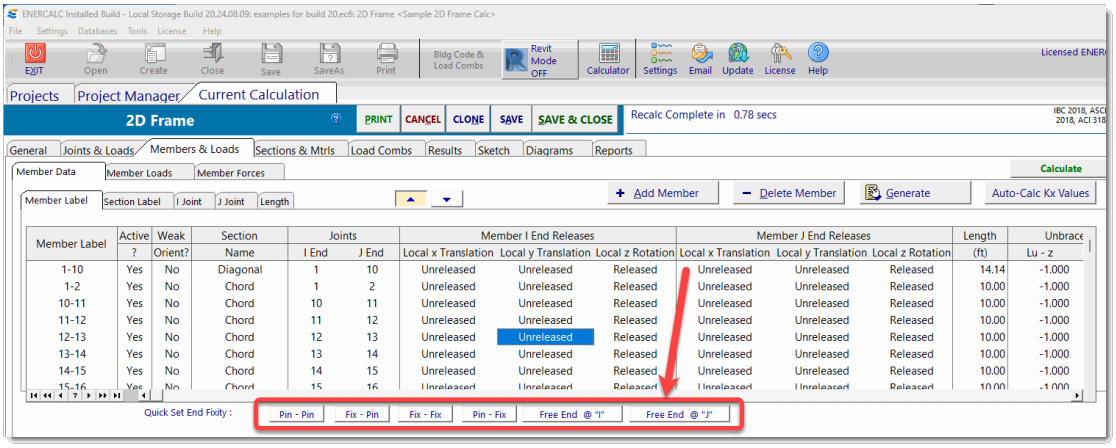
© 2026 ENERCALC, LLC

An "Unreleased" status for a particular end of a member, for a particular degree of freedom (X, Y or Z) means that end of the member is locked to the joint for that degree of freedom. A "Released" status means that end of the member is disconnected from the joint for that degree of freedom. Here are some examples:

X, Y, Z Setting	How it will work
Unreleased, Unreleased, Unreleased	The member is locked to the joint. If this joint was completely restrained to the boundary, then this end of the member would have an X and Y force reaction and a Z moment reaction.
Unreleased, Unreleased, Released	The member is locked to the joint in its local X and Y directions, meaning that this end of the member cannot translate with respect to the joint, however, this end of the member is free to rotate independently of the joint about the member's local Z axis (which is always parallel to the Global Z axis).
Unreleased, Released, Unreleased	The member is locked to the joint with respect to translation in the local X axis direction. It is disconnected from the joint with respect to translation in the local Y axis direction. It is locked to the joint with respect to rotation about the member's local Z axis. If this member is oriented horizontally, then this combination of end conditions could be thought of as a vertical roller. When a member is in the horizontal orientation, its member local axes are parallel to the corresponding Global axes. Therefore, the fixed condition in the local X axis direction means that this end of this horizontal member cannot move left or right in the Global X direction. The free condition in the local Y direction means that this end of this horizontal member is free to move up and down in the Global Y direction. Finally, the fixed condition about the local Z axis means that this end of the member is fixed against rotation about the local Z axis, which is parallel to the Global Z axis.

© 2026 ENERCALC, LLC

Along the bottom of the list is a set of buttons that allows you to quickly set the Z-axis rotation end conditions. Clicking one of these buttons will set the end releases for both ends of the member that is currently selected in the list.



NOTE! A truss connection is unique. You must look at the joint where the truss members intersect. If they are all free to rotate, you can set the "X" and "Y" conditions of all member ends at that joint to "Unreleased". And for all "Z" conditions you can set them to "Released" so they can rotate freely. Then for each joint you can set its "X" and "Y" restraint to "Released" (assuming it is not a support location) and set its "Z" restraint to "Unreleased", so that the joint will be stable.

Length

This value is automatically calculated for you from the distance between the I and J joints.

© 2026 ENERCALC, LLC

Unbraced Length

Enter the unbraced lengths that should be used for determining the capacity of this member. Entering a "-1" instructs the program to consider the full length of the member to be unbraced. This is a convenience, so the actual member length does not need to be determined and entered for unbraced members.

Any other number (0.00 or greater) is used as the unbraced length. Note that the unbraced length can be assigned a value that is greater than the node-to-node length of a member.

Lu-z is used in Kz * Lu-z to define the unbraced length for column buckling about the z axis of the member.

Lu-y in Ky * Lu-y to define the unbraced length for column buckling about the y axis of the member.

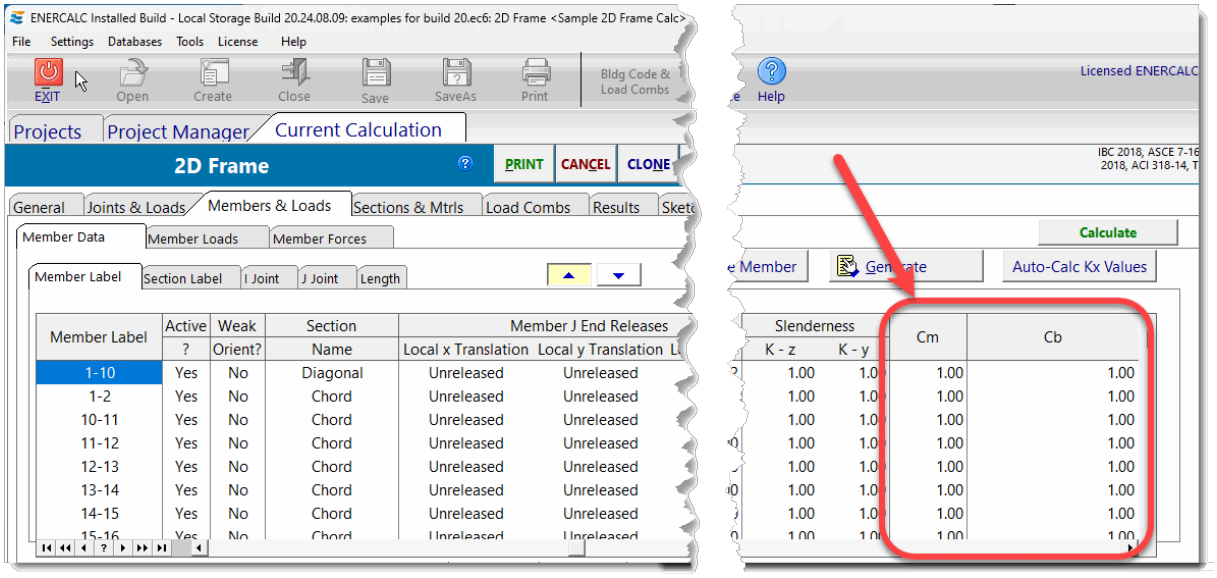
As of July 2023 the program collects an Lb value for each member. Lb defines the unbraced length to use in determination of the flexural capacity of the member. (Previously the module was interpreting the value of Lu-z to serve double-duty as the unbraced length for column buckling about the Z axis as well as unbraced length for flexure. Decoupling this forced relationship allows greater flexibility in modeling design conditions.)

Slenderness

This entry is a simple multiplier to be applied to the Unbraced Length you have entered.

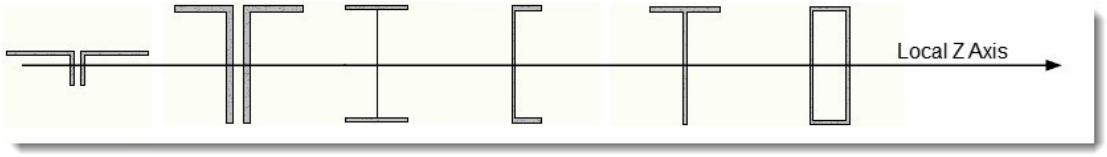
Cm & Cb

You can specify these values for use in allowable stress calculations.



Default Member Orientation

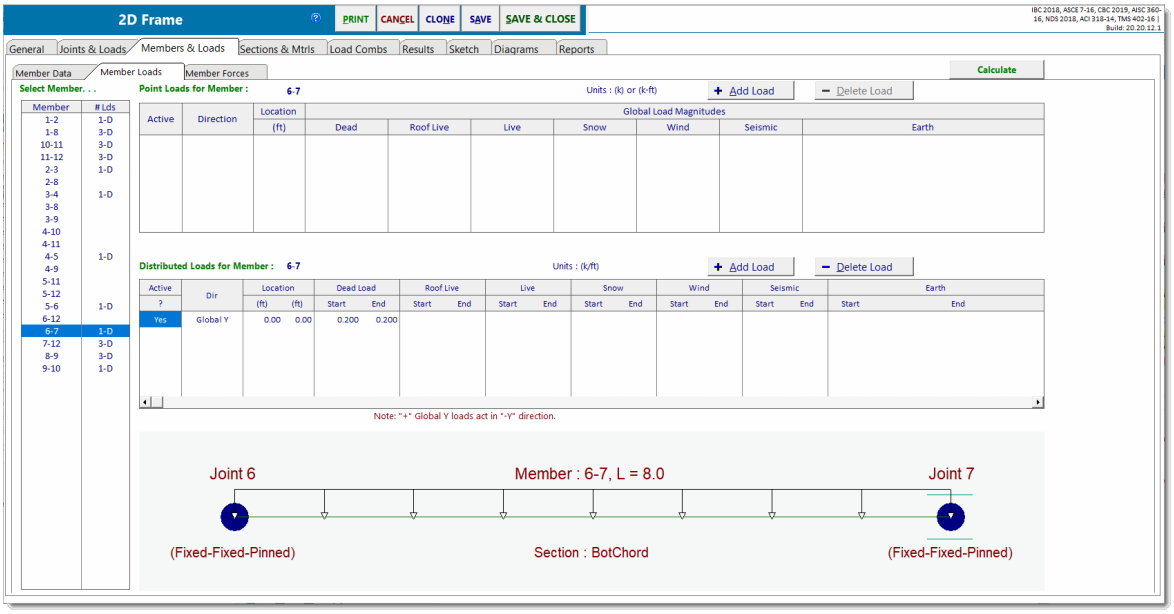
Based on the concept of the member local axis system introduced above, it is now meaningful to describe the default orientation of the various sections that can be assigned to a member in a 2D Frame Analysis model. Of most interest is the default orientation of steel sections. When a steel section is assigned to a member, it will be assumed to be oriented as shown in the following diagram, which references the member local Z axis, the axis that is perpendicular to the plane of the model by default:



Member Loads

Member Loads

This tab provides the input locations for all member loads applied to the frame EXCEPT member self weights. Member self weight loads can be automatically generated using the controls on the Load Combinations tab.



Select Member: lists all members in the model. This is where you click to select a member for which to add/delete/modify loads. This selection controls what is visible on the other two lists and in the graphic image at the bottom. The column labeled # Lds indicates the number and type of loads that have been applied to each member. "P" represents a point load, and "D" is for distributed loads. "1-P, 2-D" means that the member has one point load and two distributed loads applied to it.

Point Loads for Member X: shows the point loads that are applied to the selected member.

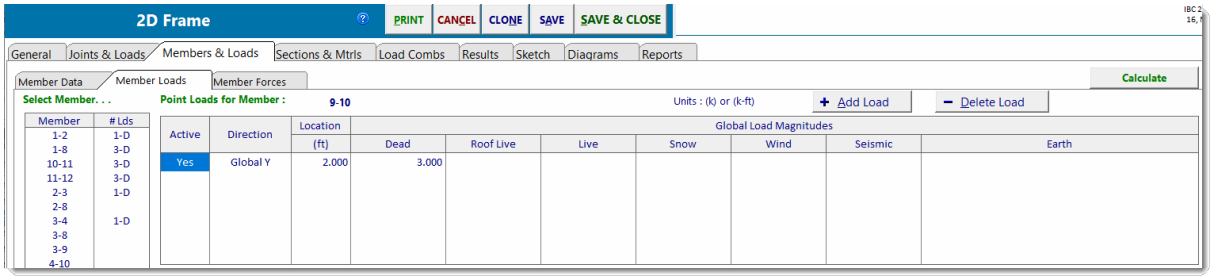
© 2026 ENERCALC, LLC

Distributed Loads for Member X: shows the distributed loads that are applied to the selected member.

At the bottom is a sketch showing the member and applied loads for reference.

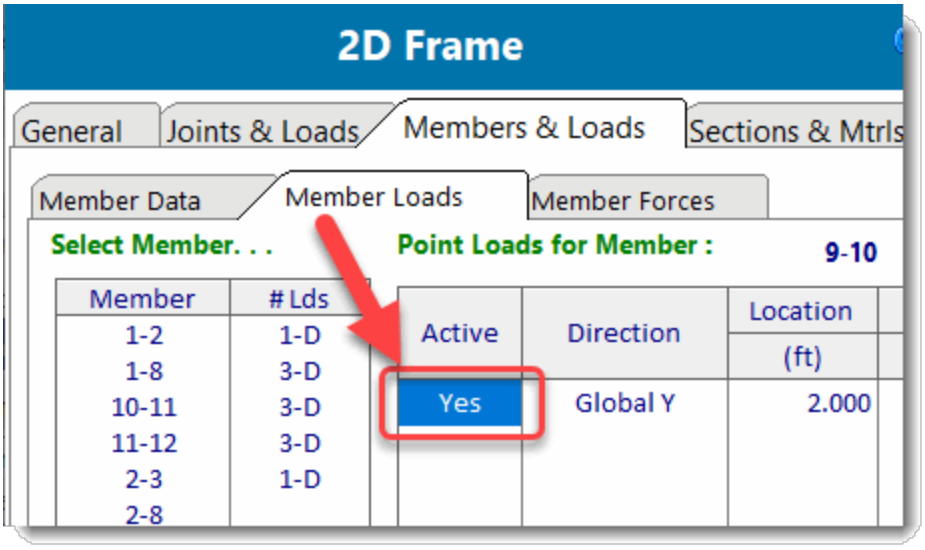
When you click on any item in the two load entry lists, that location in the list will change into a data entry item. The columns in the list have their own specific entry types that will be described below.

Point Loads Entry List



Active

This is a Yes/No checkbox that allows you to turn the load on and off. This is useful when you want to quickly see the effect of removing the load without actually having to delete the load and then potentially have to redefine the load at a later time.



© 2026 ENERCALC, LLC

Direction

This specifies the direction of application of the load. Here is a description of each direction:

Global X: This point load acts in a direction parallel to the Global X Axis. Entering a positive value will apply the load to the right (in the positive X direction).

Global Y: This point load acts in a direction parallel to the Global Y Axis. The algebraic sign on the magnitude will affect the direction of application based on the Applied Global Y Load Sign Convention setting on the General tab as follows:

When the Applied Global Y Load Sign Convention is set to "Global +Y Loads act towards +Y (Upward)", then loads applied in the Global Y direction with a positive algebraic sign act upward, and loads applied in the Global Y direction with a negative algebraic sign act downward.

When the Applied Global Y Load Sign Convention is set to "Global +Y Loads act towards -Y (Downward)", then loads applied in the Global Y direction with a positive algebraic sign act downward, and loads applied in the Global Y direction with a negative algebraic sign act upward.

In order to most efficiently describe the direction of application of the "Local" load types, it helps to refer to the member local axis system. Each member can be thought of as having its own x, y, and z coordinate axes that are mutually perpendicular and follow the right-hand rule. The orientation of the member local axes can be determined as follows:

1. The local x axis is always parallel to a vector from the "I" node to the "J" node. (This axis is parallel to the longitudinal axis of the member.)
2. The local z axis is always parallel to the Global Z axis.
3. The local y axis can be found by taking the vector cross product of local z cross local x. (Envision using the right hand to rotate the local z axis into the local x axis, and the right thumb will automatically indicate the positive direction of the local y axis.)

Local x: This point load acts parallel to the member's local x axis. If this load is specified with a positive magnitude, the load will act in the positive direction of the local x axis. If this load is specified with a negative magnitude, the load will act in the negative direction of the local x axis.

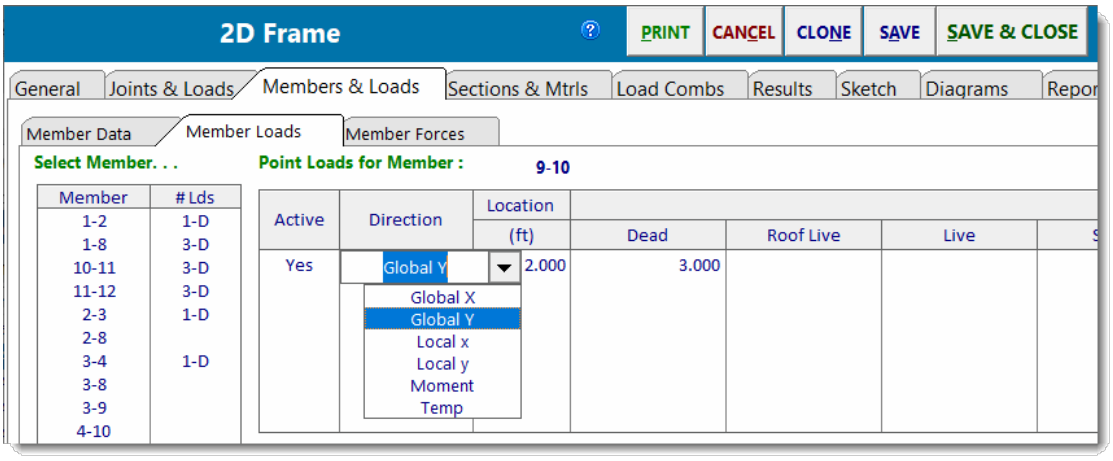
Local y: This point load acts parallel to the member's local y axis. If this load is specified with a positive magnitude, the load will act in the positive direction of the local y axis. If this load is specified with a negative magnitude, the load will act in the negative direction of the local y axis.

It should now be obvious that it is VERY important to have a thorough understanding of member orientation when using "Local" load types.

© 2026 ENERCALC, LLC

Moment: This specifies that the load is a concentrated moment. Positive moments follow the right-hand rule and apply a counter-clockwise rotational force to the member (when viewing the model from the positive Z direction).

Temperature: This is used to specify temperatures at particular locations along the length of a member. The module then uses these spot temperatures, in conjunction with the joint temperatures specified in the Joint Data list, to establish temperature gradients along the member(s). The module will calculate the effects of the specified temperature gradient from the end joint to the location along the member at which the temperature was specified. If you apply more temperature loads, the gradients are developed between each adjacent point of temperature load.



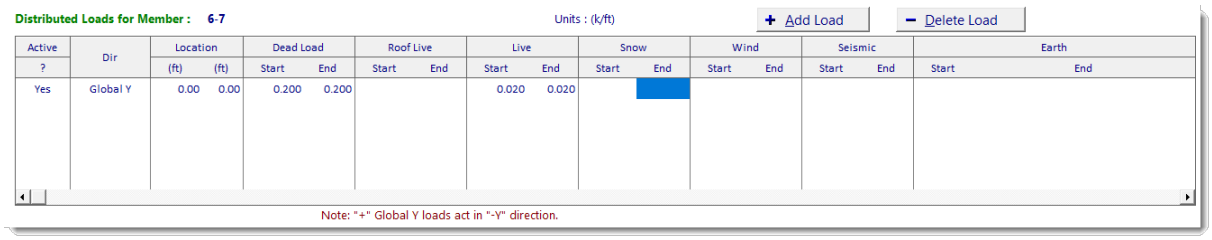
Location

This specifies the distance from the "I" joint where the point load is located.

Load Type & Magnitude

You can enter seven different types of loads on the frame and combine them using the factors on the Load Combinations tab.

Distributed Load Entry List



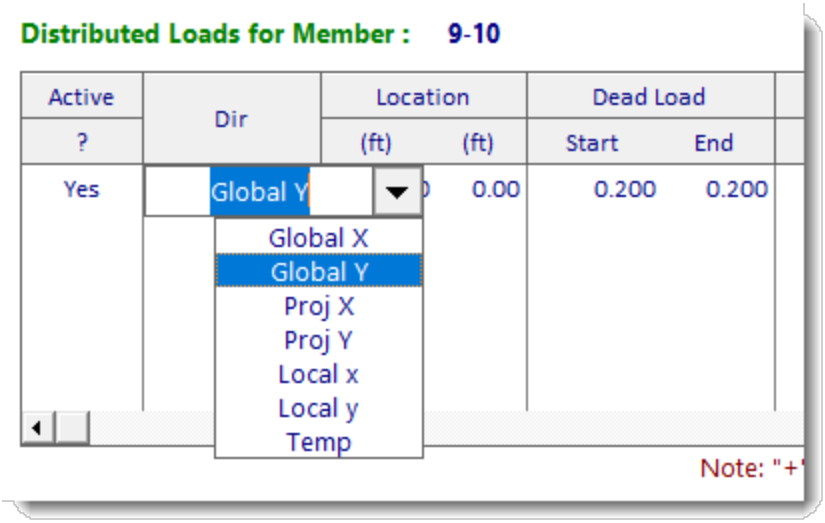
Active

This is a Yes/No checkbox that allows you to turn the load on and off. This is useful when you want to quickly see the effect of removing the load.

© 2026 ENERCALC, LLC

Direction

This specifies the direction of application of the load.



Here is a description of the different directions:

Global X: This load acts in a direction parallel to the Global X Axis and is distributed along the full length of the member. If the member is sloped, the load will be applied for the full length of the member. So if a member has a 10-foot rise and a 10-foot run, the length of the load will equal 14.14 feet. Entering a positive value will apply the load to the right (in the positive X direction).

Global Y: This load acts in a direction parallel to the Global Y Axis and is distributed along the full length of the member. If the member is sloped, the load will be applied for the full length of the member. So if a member has a 10-foot rise and a 10-foot run, the length of the load will equal 14.14 feet. The algebraic sign on the magnitude will affect the direction of application based on the Applied Global Y Load Sign Convention setting on the General tab as follows:

> When the Applied Global Y Load Sign Convention is set to "Global +Y Loads act towards +Y (Upward)", then loads applied in the Global Y direction with a positive algebraic sign act upward, and loads applied in the Global Y direction with a negative algebraic sign act downward.
>
> When the Applied Global Y Load Sign Convention is set to "Global +Y Loads act towards -Y (Downward)", then loads applied in the Global Y direction with a positive algebraic sign act downward, and loads applied in the Global Y direction with a negative algebraic sign act upward.

Projected X: This load acts in a direction parallel to the Global X Axis, but it is applied only to the length of the member projected onto the Global Y Axis. So if a member has a 10-foot rise and a 20-foot run, the length of the load will equal 10.00 feet (the rise of the member).

© 2026 ENERCALC, LLC

Projected Y: This load acts in a direction parallel to the Global Y Axis, but it is applied only to the length of the member projected onto the Global X axis. So if a member has a 10-foot rise and a 20-foot run, the length of the load will equal 20.00 feet (the run of the member).

In order to most efficiently describe the direction of application of the "Local" load types, it helps to refer to the member local axis system. Each member can be thought of as having its own x, y, and z coordinate axes that are mutually perpendicular and follow the right-hand rule. The orientation of the member local axes can be determined as follows:

1. The local x axis is always parallel to a vector from the "I" node to the "J" node. (This axis is parallel to the longitudinal axis of the member.)
2. The local z axis is always parallel to the Global Z axis.
3. The local y axis can be found by taking the vector cross product of local z cross local x. (Envision using the right hand to rotate the local z axis into the local x axis, and the right thumb will automatically indicate the positive direction of the local y axis.)

Local x: This distributed load acts parallel to the member's local x axis. If this load is specified with a positive magnitude, the load will act in the positive direction of the local x axis. If this load is specified with a negative magnitude, the load will act in the negative direction of the local x axis.

Local y: This distributed load acts parallel to the member's local y axis. If this load is specified with a positive magnitude, the load will act in the positive direction of the local y axis. If this load is specified with a negative magnitude, the load will act in the negative direction of the local y axis.

It should now be obvious that it is VERY important to have a thorough understanding of member orientation when using "Local" load types.

Temperature: This is used to specify temperatures at particular locations along the length of a member. The module then uses these spot temperatures, in conjunction with the joint temperatures specified in the Joint Data list, to establish temperature gradients along the member(s). The module will calculate the effects of the specified temperature gradient from the end joint to the location along the member at which the temperature was specified. If you apply more temperature loads, the gradients are developed between each adjacent point of temperature load.

Location - Start, End

This specifies the distance from the "I" joint to the beginning and end of the load. Leaving BOTH values as zero (0.0) will cause the load to be applied to the full length of the member.

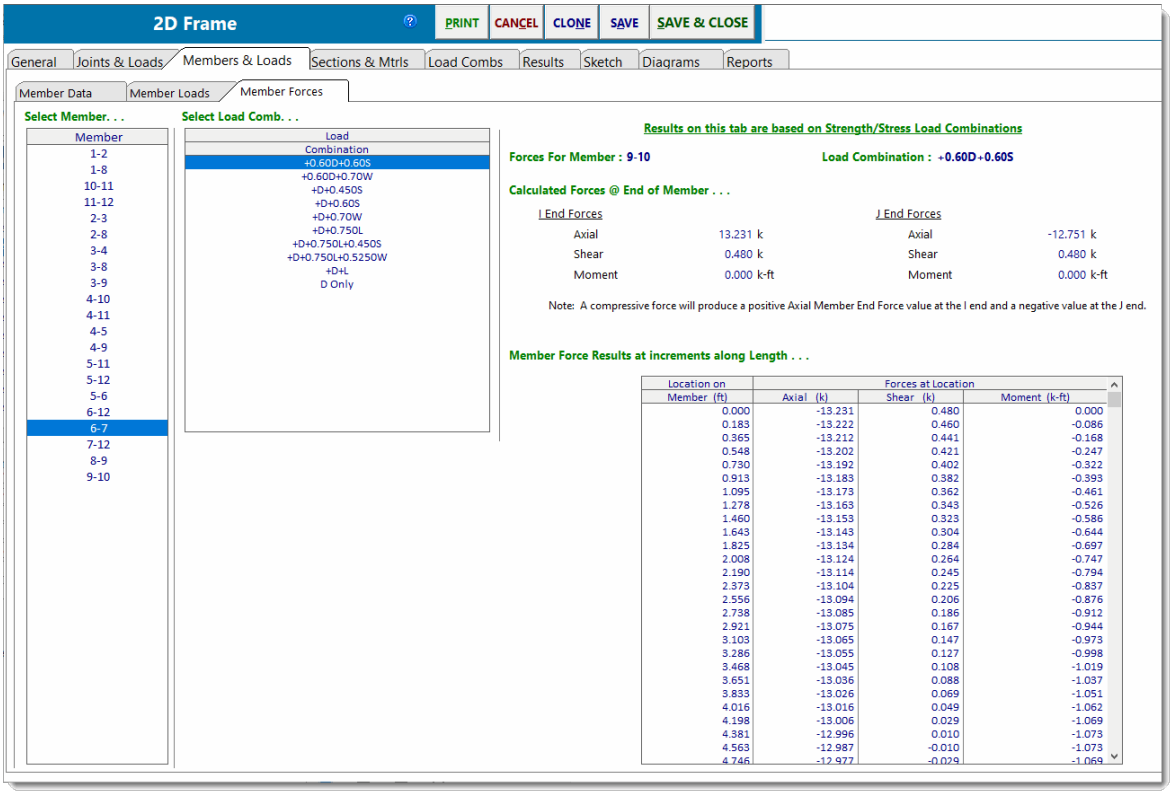
Load Type & Magnitude

You can enter seven different types of loads on the frame and combine them using the values on the Load Combinations tab.

© 2026 ENERCALC, LLC

Member Forces

This tab allows you to review the final calculated forces for a member.



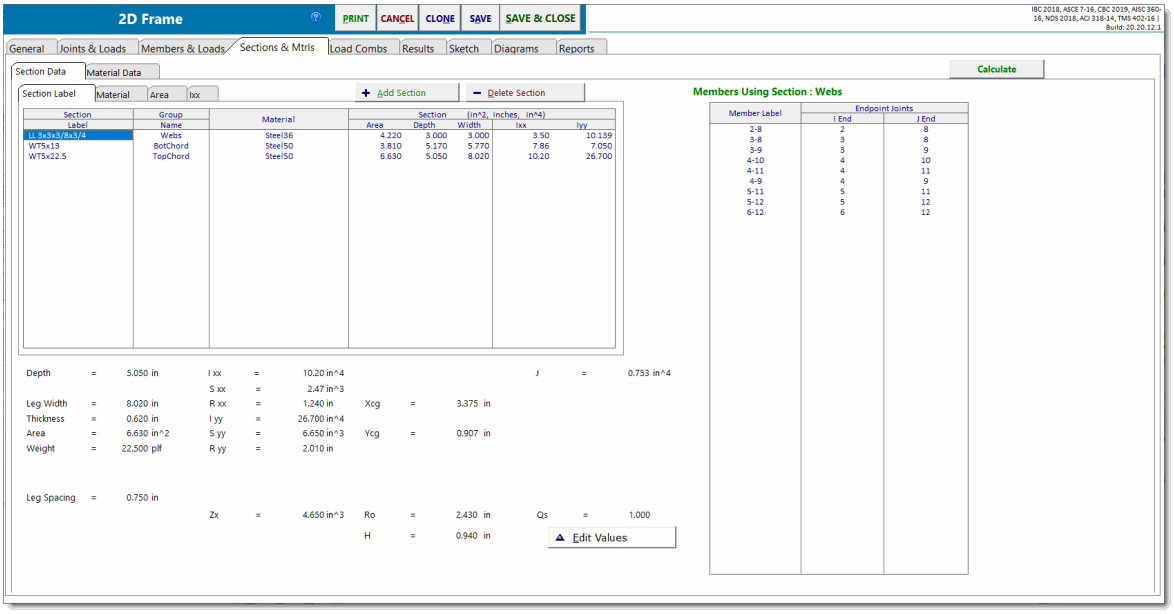
To review member forces, first click the member of interest in the Select Member list. Next, click on a load combination in the Select Load Comb. list. The member end forces will be displayed for the chosen load combination, and a table of member forces at increments along the length of the member will be shown.

This tab provides a more simplified view of member forces than the large tables displayed on the Results tab.

© 2026 ENERCALC, LLC

13.3.2.4 Sections & Materials

The two tabs under Sections & Materials provide the ability to define the section properties and material properties for the members used in the frame.



Section Data

The Section Data tab allows you to specify sections to be used for the frame.

Section Label vs. Group Label

These two ways of labeling a section are very helpful and should be understood.

The Section Label is the actual name of the section, whether it is an AISC section name like W14x22 or a wood section like 4x10. The Section Label always represents something that can be retrieved from the internal databases.

It can also be the name of a section that you create by entering a name and some properties.

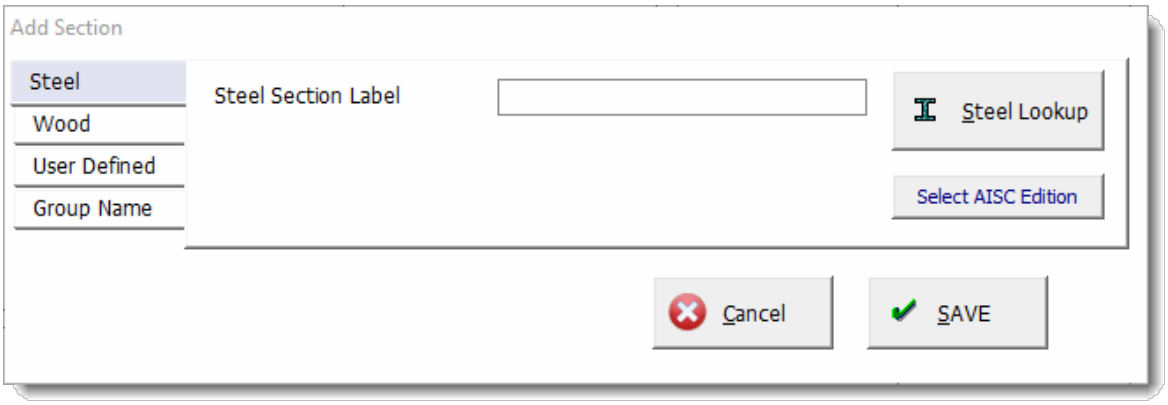
The Group Label lets you associate a section name with a label that is meaningful to you and that can be used on multiple members. This makes it so you can easily revise the section that is assigned to a set of members without the need to change the section name individually for all the members where it is used.

For example, say that members 1,3,5,7,9,11,13,15 and 17 will all use the same section. You don't know which AISC section will be selected, but you DO know it will be the same section. You can assign a preliminary section name of HSS 3x3x1/4 and a group label of "diagonal_1". If the frame analysis shows that the HSS section fails, you can simply change it to a different section (with new properties). Because that section is linked to those members with the group label, it simplifies the section assignment and modification for the entire group.

© 2026 ENERCALC, LLC

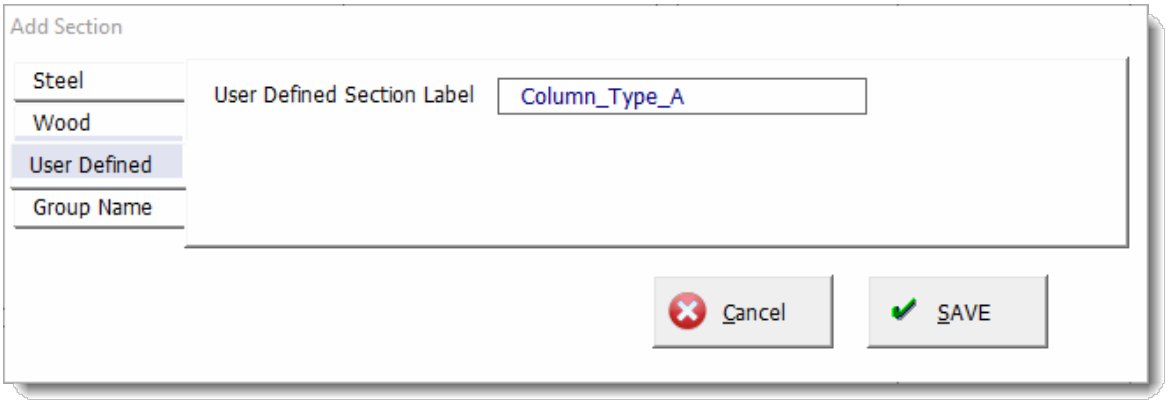
Adding and Deleting Sections

When you click the [**Add Section**] button, the following window appears:



To select a section from the built-in AISC or NDS databases, just click the appropriate tab and either type in the section name or click the [**Lookup**] button.

If you want to add your own section name and type in the properties, click the User Defined tab, type in the desired section name, and click [**OK**].



[**Delete Section**] will delete the section you've highlighted in the list. Any members using that section will be changed to reference the Default section.

Editing the Table

Section Label

When you click on the entries in this column the entry position will change to a button labeled [**Lookup**]. This button provides access to the built-in section databases.

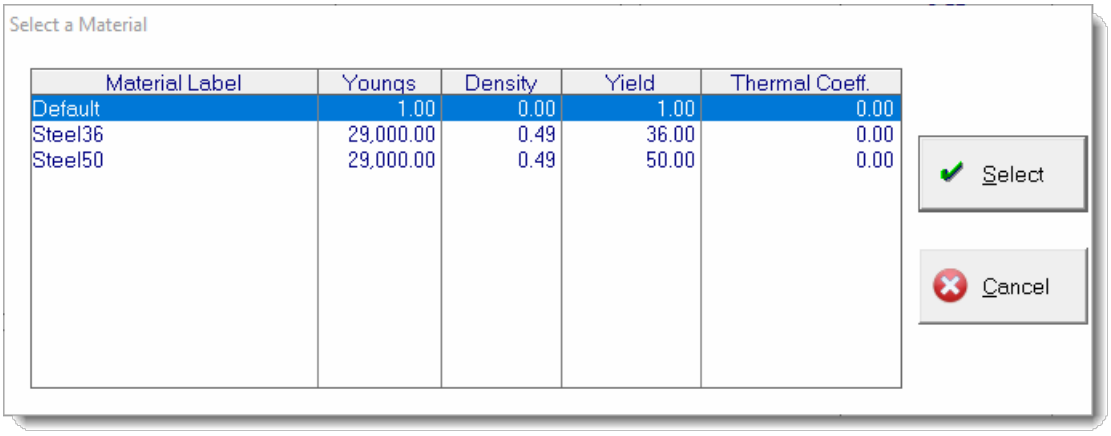
© 2026 ENERCALC, LLC

Group Label

When you click on the entries in this column the entry position changes to a text editing box.

Material

When you click on the entries in this column the entry position changes to a [**Lookup**] button. This button provides access to a list of the material properties that you have already defined on the Materials tab.



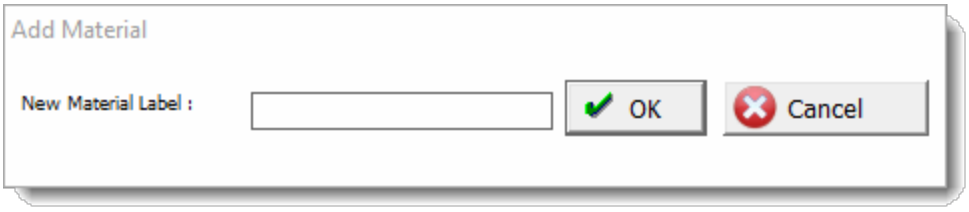
Section property values

These entry columns change to numeric entries when selected to allow entry of the values.

Material Data

This screen provides the ability to define the material values used by the sections you define.

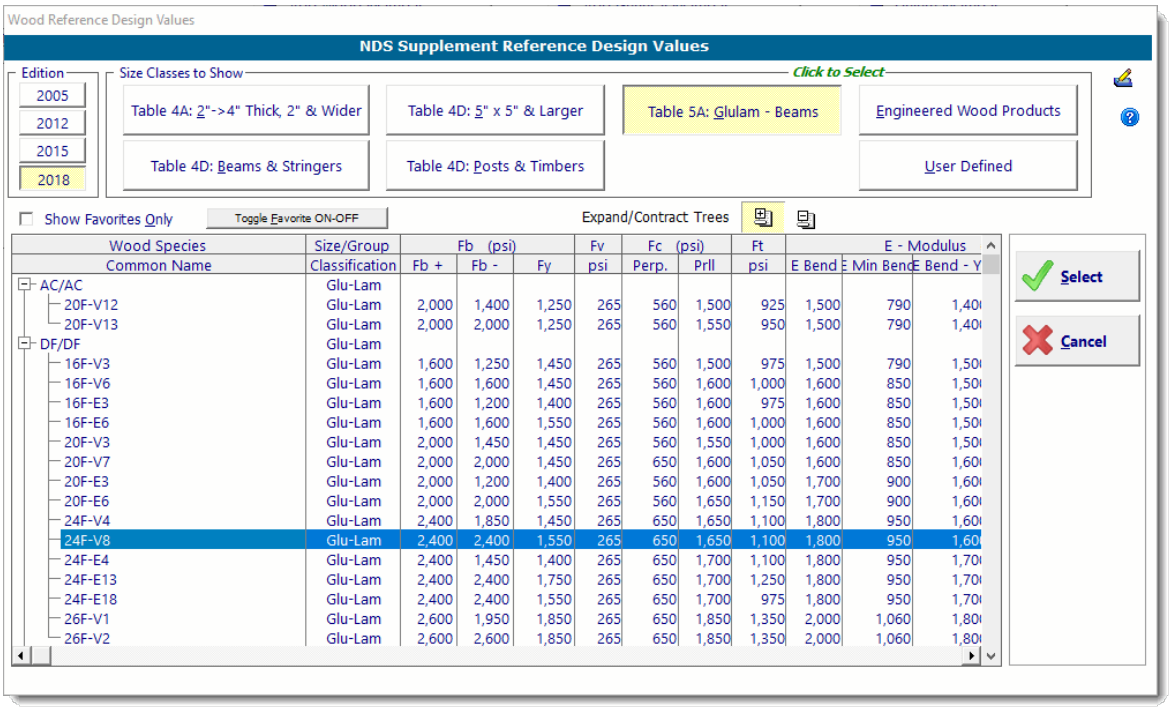
Clicking [**Add General Material**] displays the following input box for you to enter the material name:



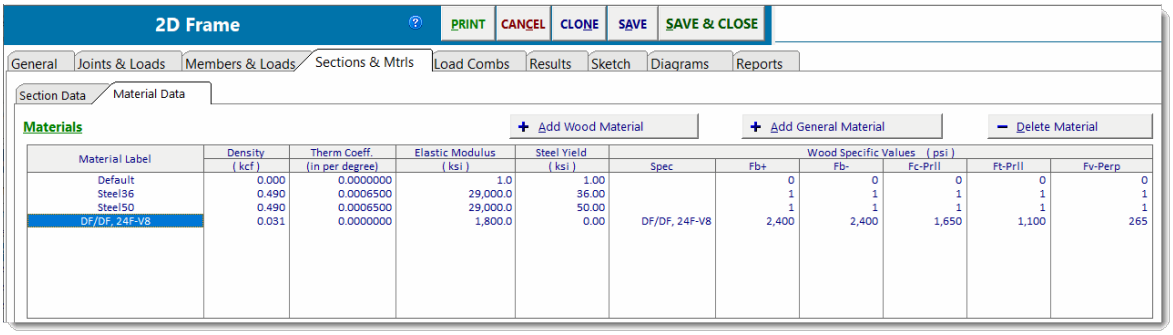
To add a new material, enter the name of the material and then click [**OK**]. The new material will appear in the Materials list on the Material Data tab.

© 2026 ENERCALC, LLC

Clicking [**Add Wood Material**] displays the Wood Reference Design Values database where you can select a wood species and grade combination:



To select a wood material to use in the current calculation, click on the desired material in the Wood Reference Design Values database and then click [**Select**]. The new wood material will appear in the Materials list on the Material Data tab as shown below:



Clicking on the Elastic Modulus, Density, Yield Stress and Thermal Coefficient entry items will change them into numeric entry boxes where you can type in a value and press [**Tab**] or click off of the value to complete the data entry.

Elastic Modulus

The elastic modulus defines how the members will react to forces by defining the relationship between stress and strain.

© 2026 ENERCALC, LLC

Density

The density entry is used only when the module calculates and applies member self weights using the **Member Self Weight** entry column on the Load Combination tab.

Yield Stress

The use of this column and additional allowable stress information will be enhanced as the module matures. For steel members this property is used to perform the AISC member allowable stress evaluation. When wood stress evaluation is added, the module will store other pertinent values from the built-in database.

Thermal Coefficient

This defines the rate of thermal expansion per degree of temperature change. This value is only used when temperature loads are defined for a member.

Wood-Specific Values

The Wood-Specific Values are only populated when the selected material comes from the Wood Reference Design Values database.

13.3.2.5 Load Combinations

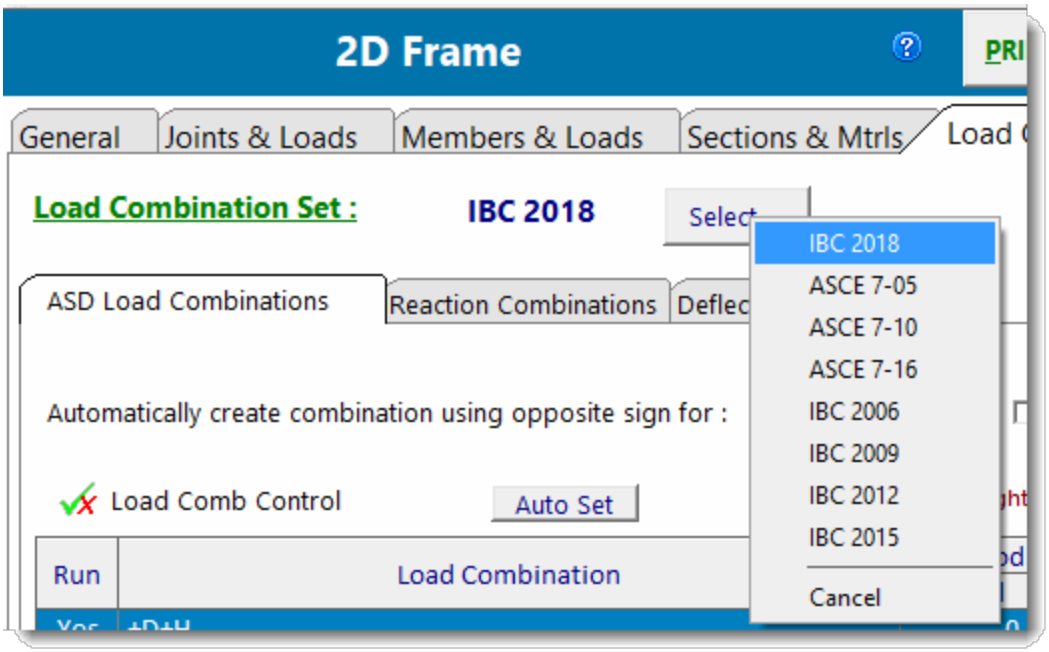
This data table controls how all of the loads are combined, and also enables you to have member self weight loads automatically considered in specified directions. Note that there is are three Load Combination tabs:

- ASD Load Combinations or LRFD Load Combinations (Based on your selection of Design Method on the General tab. Used for member design code checks.)
- Reaction Combinations (Always service level combinations. Used to calculate reactions.)
- Deflection Combinations (Always service level combinations. Used to calculate deflections.)

© 2026 ENERCALC, LLC

Load Combination Set

The Select button is used to retrieve load combination sets from the built-in load combination database and place them in the list. Clicking it will display a dropdown menu listing the load combination sets currently included in the database.



Add (plus button)

This button adds a load combination line to the list. You can then edit the factors.

Delete (minus button)

This button deletes the currently selected load combination from the list.

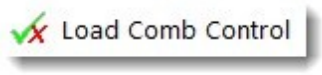
This button allow you to edit the currently selected load combination.

© 2026 ENERCALC, LLC

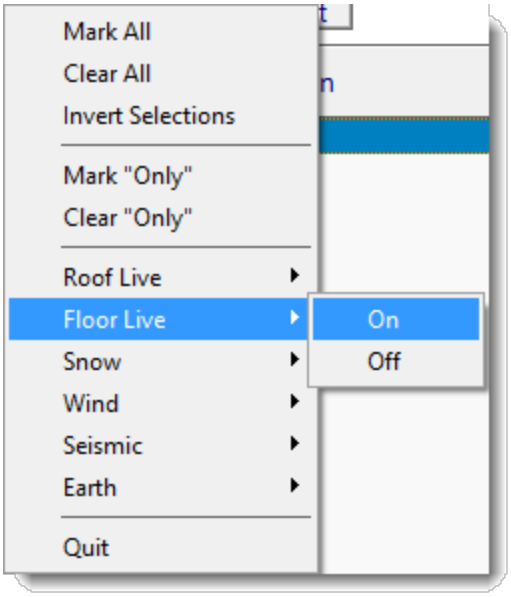
Description of columns in table

Run

This column is a Yes/No toggle to control whether that load combination is used or ignored.



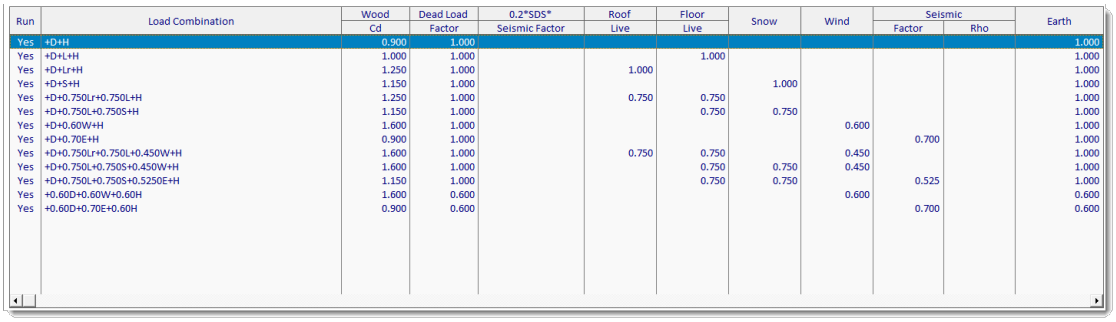
This button allows you to change the Run setting for all the load combinations at once using several options.



Load Combination

This is column represents the load combination. It consists of abbreviations for each included load type along with the associated numeric values that represent the respective load factors. You cannot directly edit this string. It is constructed automatically based on the entries you make in the following columns.

Note in the following image how the Load Combination Name is constructed from the specified values:



Run	Load Combination	Wood	Dead Load	0.2*SDS*	Roof	Floor	Snow	Wind	Seismic		Earth
		Cd	Factor	Seismic Factor	Live	Live			Factor	Rho	
Yes	+D+H	0.900	1.000								1.000
Yes	+D+L+H	1.000	1.000			1.000					1.000
Yes	+D+Lr+H	1.250	1.000		1.000						1.000
Yes	+D+S+H	1.150	1.000				1.000				1.000
Yes	+D+0.750Lr+0.750L+H	1.250	1.000		0.750	0.750					1.000
Yes	+D+0.750L+0.750S+H	1.150	1.000			0.750	0.750				1.000
Yes	+D+0.60W+H	1.600	1.000					0.600			1.000
Yes	+D+0.70E+H	0.900	1.000						0.700		1.000
Yes	+D+0.750Lr+0.750L+0.450W+H	1.600	1.000		0.750	0.750		0.450			1.000
Yes	+D+0.750L+0.750S+0.450W+H	1.600	1.000			0.750	0.750	0.450			1.000
Yes	+D+0.750L+0.750S+0.5250E+H	1.150	1.000			0.750	0.750		0.525		1.000
Yes	+0.60D+0.60W+0.60H	1.600	0.600					0.600			0.600
Yes	+0.60D+0.70E+0.60H	0.900	0.600						0.700		0.600

© 2026 ENERCALC, LLC

Load Duration Factor (Not a load factor. Applies to wood designs only.)

This value acts as a multiplier on the allowable stress for the corresponding load combination.

Individual Load Factors

Enter the numeric value to be applied to each type of load.

0.2*SDS*Seismic Factor

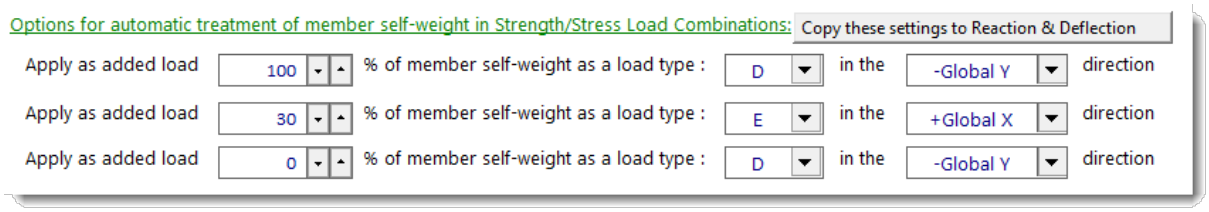
When a value of Sds is specified, this column will report the additional component of vertical load that is to be included due to seismic loading.

Member Self Weight

The controls at the bottom of the Load Combinations tabs indicate whether the program should automatically apply member self weight. Note that automatic application of self-weight can be set up differently on the three Load Combination sub-tabs, if desired. Alternatively, there is a convenience button to copy the self-weight settings from the first tab to the other two tabs, if the goal is to make them all the same.

When self-weight is non-zero, the module will calculate the weight of each member (as cross sectional area * density * length) and consider that weight during the analysis.

Note that the controls allow for the self-weight to be assigned to the Dead Load or the Seismic Load case, and to be assigned a direction and a percentage if desired. This makes it possible to model, say, 30% of the self-weight as an applied lateral load to be considered a Seismic Load.



13.3.2.6 Wood Design

The 2D Frame module now incorporates wood stress checking according based on ASD or LRFD methods according to NDS methods.

Adjustment Factors for Sawn Lumber

The module only collects C_D or Lambda from the user.

The module assumes a value of 1.0 for C_M, C_t, C_{fu}, C_i, C_r and C_T.

© 2026 ENERCALC, LLC

The module calculates values for C_L, C_F, and C_P.

The bearing area factor C_b never comes into play in the functions performed within this module.

Adjustment Factors for Glued Laminated Timber

The module only collects C_D or Lambda from the user.

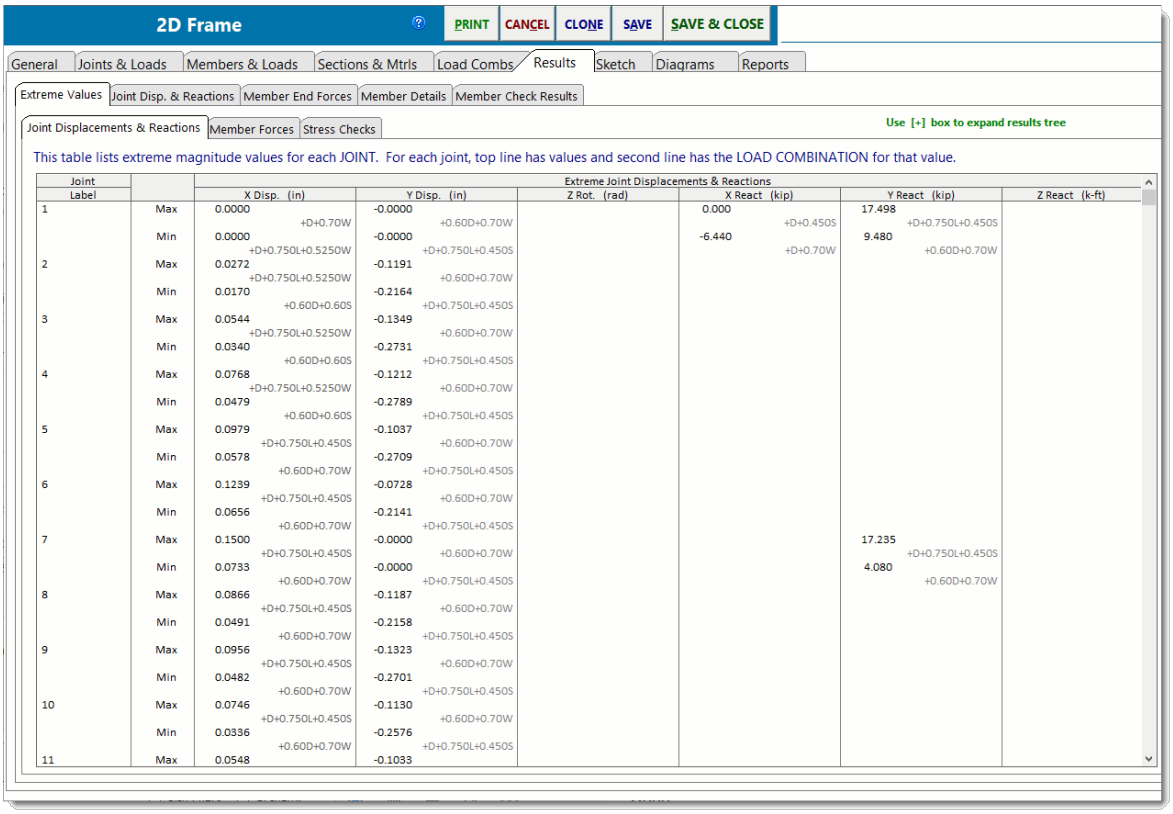
The module assumes a value of 1.0 for C_M, C_t, C_{fu}, C_c, C_i, and C_{vr}.

The module calculates values for C_L, C_V, and C_P.

The bearing area factor C_b never comes into play in the functions performed within this module.

13.3.2.7 Results

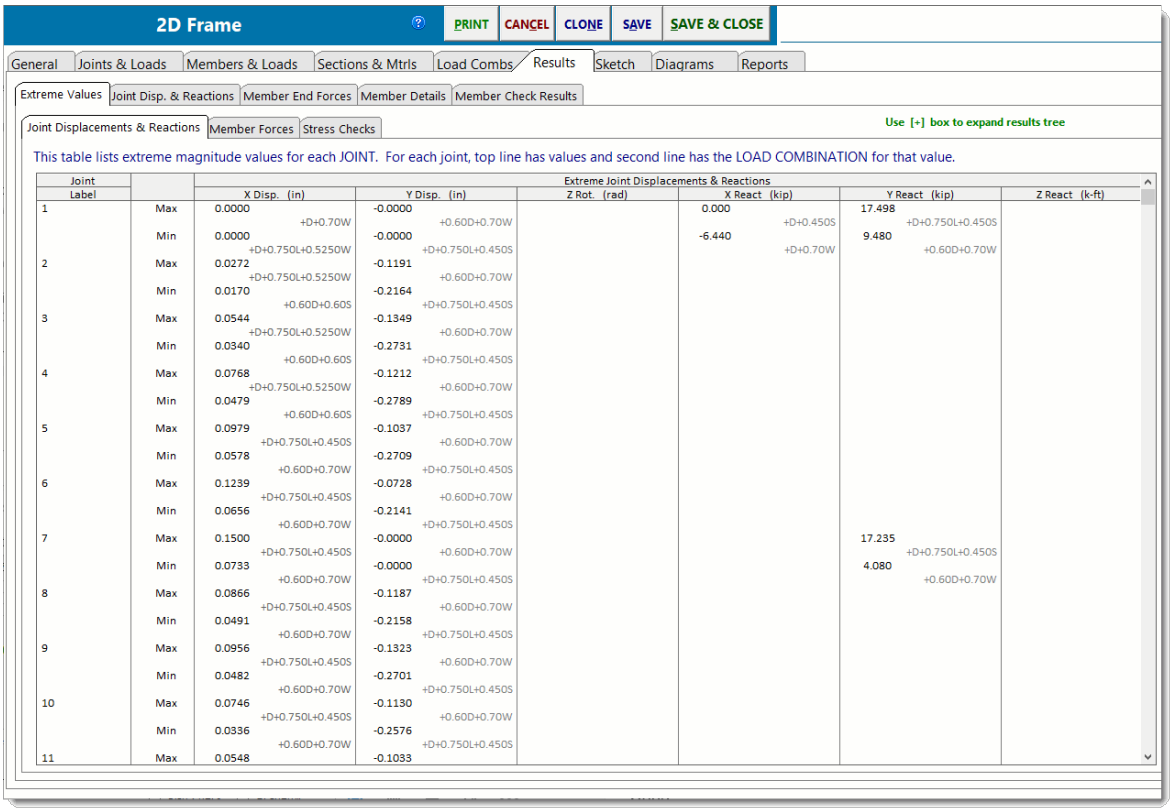
The Results tab provides extensive lists of detailed and summarized results from the frame analysis and the design checks.



Extreme Values

Joint Displacements & Reactions

This list summarizes the extreme joint displacements and support reactions.



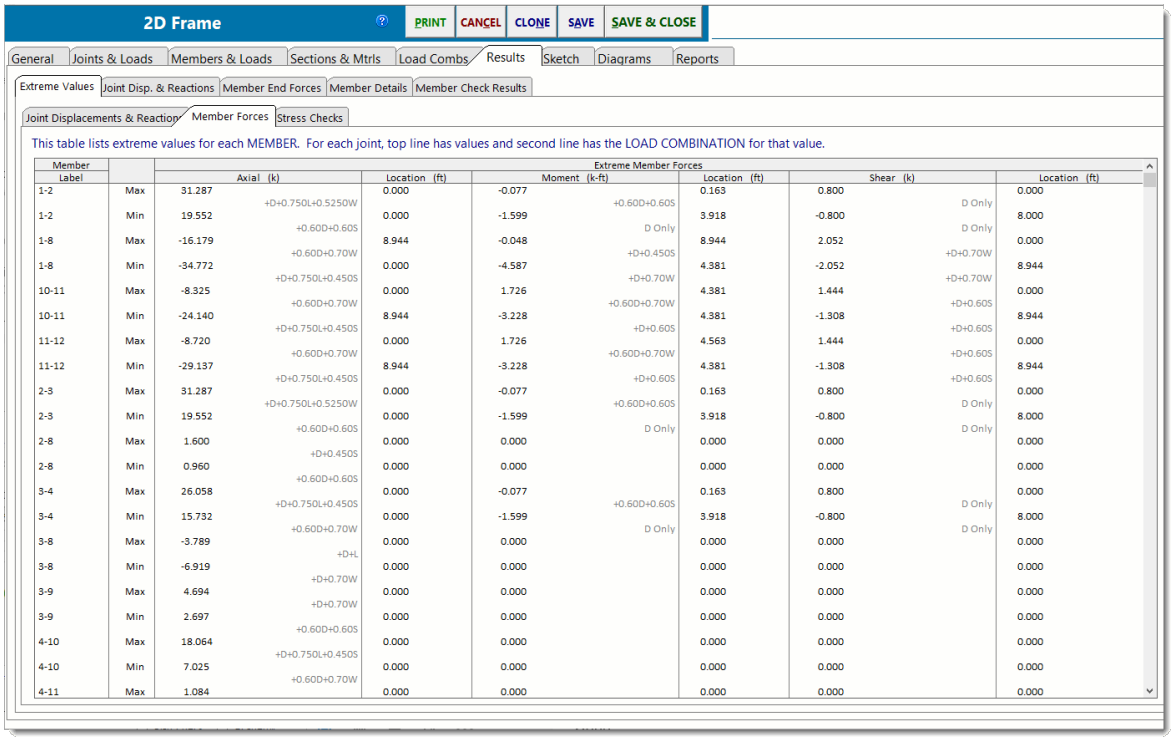
Joint Label		Extreme Joint Displacements & Reactions					
		X Disp. (in)	Y Disp. (in)	Z Rot. (rad)	X React (kip)	Y React (kip)	Z React (k-ft)
1	Max	0.0000	-0.0000		0.000	17.498	
		+D+0.70W	+0.60D+0.70W		+D+0.450S	+D+0.750L+0.450S	
	Min	0.0000	-0.0000		-6.440	9.480	
		+D+0.750L+0.5250W	+D+0.750L+0.450S		+D+0.70W	+0.60D+0.70W	
2	Max	0.0272	-0.1191				
		+D+0.750L+0.5250W	+0.60D+0.70W				
	Min	0.0170	-0.2164				
		+0.60D+0.60S	+D+0.750L+0.450S				
3	Max	0.0544	-0.1349				
		+D+0.750L+0.5250W	+0.60D+0.70W				
	Min	0.0340	-0.2731				
		+0.60D+0.60S	+D+0.750L+0.450S				
4	Max	0.0768	-0.1212				
		+D+0.750L+0.5250W	+0.60D+0.70W				
	Min	0.0479	-0.2789				
		+0.60D+0.60S	+D+0.750L+0.450S				
5	Max	0.0979	-0.1037				
		+D+0.750L+0.450S	+0.60D+0.70W				
	Min	0.0578	-0.2709				
		+0.60D+0.70W	+D+0.750L+0.450S				
6	Max	0.1239	-0.0728				
		+D+0.750L+0.450S	+0.60D+0.70W				
	Min	0.0656	-0.2141				
		+0.60D+0.70W	+D+0.750L+0.450S				
7	Max	0.1500	-0.0000			17.235	
		+D+0.750L+0.450S	+0.60D+0.70W			+D+0.750L+0.450S	
	Min	0.0733	-0.0000			4.080	
		+0.60D+0.70W	+D+0.750L+0.450S			+0.60D+0.70W	
8	Max	0.0866	-0.1187				
		+D+0.750L+0.450S	+0.60D+0.70W				
	Min	0.0491	-0.2158				
		+0.60D+0.70W	+D+0.750L+0.450S				
9	Max	0.0956	-0.1323				
		+D+0.750L+0.450S	+0.60D+0.70W				
	Min	0.0482	-0.2701				
		+0.60D+0.70W	+D+0.750L+0.450S				
10	Max	0.0746	-0.1130				
		+D+0.750L+0.450S	+0.60D+0.70W				
	Min	0.0336	-0.2576				
		+0.60D+0.70W	+D+0.750L+0.450S				
11	Max	0.0548	-0.1033				

For each joint there is a four-line result display. The first line shows the joint label and the maximum values for each displacement and reaction direction. The second line indicates the load combinations that create the maximum values. The third line indicates the minimum (most negative) values for each displacement and reaction direction. The fourth line indicates the load combinations that create the minimum values.

© 2026 ENERCALC, LLC

Member Forces

This table summarizes the extreme forces that occur anywhere along the length of each member. (The detailed forces ALONG the length of the member are given in another list within this Results section.)



Member Label		Extreme Member Forces					
		Axial (k)	Location (ft)	Moment (k-ft)	Location (ft)	Shear (k)	Location (ft)
1-2	Max	31.287	0.000	-0.077	0.163	0.800	0.000
		+D+0.750L+0.525W		+0.60D+0.60S		D Only	
1-2	Min	19.552	0.000	-1.599	3.918	-0.800	8.000
		+0.60D+0.60S		D Only		D Only	
1-8	Max	-16.179	8.944	-0.048	8.944	2.052	0.000
		+0.60D+0.70W		+D+0.450S		+D+0.70W	
1-8	Min	-34.772	0.000	-4.587	4.381	-2.052	8.944
		+D+0.750L+0.450S		+D+0.70W		+D+0.70W	
10-11	Max	-8.325	0.000	1.726	4.381	1.444	0.000
		+0.60D+0.70W		+0.60D+0.70W		+D+0.60S	
10-11	Min	-24.140	8.944	-3.228	4.381	-1.308	8.944
		+D+0.750L+0.450S		+D+0.60S		+D+0.60S	
11-12	Max	-8.720	0.000	1.726	4.563	1.444	0.000
		+0.60D+0.70W		+0.60D+0.70W		+D+0.60S	
11-12	Min	-29.137	8.944	-3.228	4.381	-1.308	8.944
		+D+0.750L+0.450S		+D+0.60S		+D+0.60S	
2-3	Max	31.287	0.000	-0.077	0.163	0.800	0.000
		+D+0.750L+0.525W		+0.60D+0.60S		D Only	
2-3	Min	19.552	0.000	-1.599	3.918	-0.800	8.000
		+0.60D+0.60S		D Only		D Only	
2-8	Max	1.600	0.000	0.000	0.000	0.000	0.000
		+D+0.450S					
2-8	Min	0.960	0.000	0.000	0.000	0.000	0.000
		+0.60D+0.60S					
3-4	Max	26.058	0.000	-0.077	0.163	0.800	0.000
		+D+0.750L+0.450S		+0.60D+0.60S		D Only	
3-4	Min	15.732	0.000	-1.599	3.918	-0.800	8.000
		+0.60D+0.70W		D Only		D Only	
3-8	Max	-3.789	0.000	0.000	0.000	0.000	0.000
		+D+L					
3-8	Min	-6.919	0.000	0.000	0.000	0.000	0.000
		+D+0.70W					
3-9	Max	4.694	0.000	0.000	0.000	0.000	0.000
		+D+0.70W					
3-9	Min	2.697	0.000	0.000	0.000	0.000	0.000
		+0.60D+0.60S					
4-10	Max	18.064	0.000	0.000	0.000	0.000	0.000
		+D+0.750L+0.450S					
4-10	Min	7.025	0.000	0.000	0.000	0.000	0.000
		+0.60D+0.70W					
4-11	Max	1.084	0.000	0.000	0.000	0.000	0.000

For each member there is a four-line result display. The first line shows the member label and the maximum member force values that occur anywhere along the length of the member. The second line indicates the load combinations that create the maximum values. The third line indicates the minimum (most negative) member force values that occur anywhere along the length of the member. The fourth line indicates the load combinations that create the minimum values.

Stress Checks

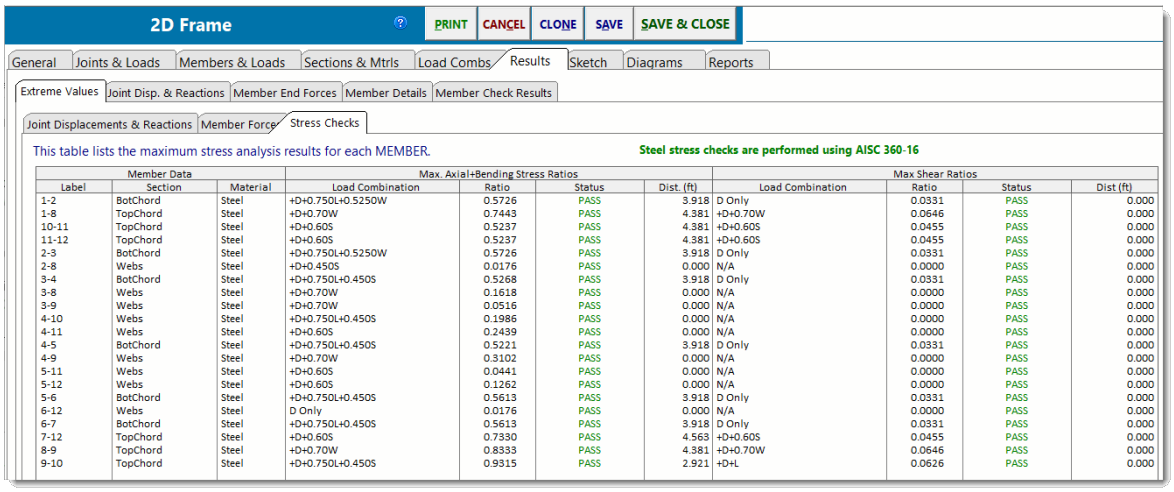
The Stress Checks tab is only displayed when the Member Stress Check Status item on the General tab has been set to ASD or LRFD stress checks.

© 2026 ENERCALC, LLC

For each member this list shows the following information:

- member label
- section or group label
- material
- governing load combination that results in the maximum Axial plus Bending stress ratio
- maximum stress ratio for the Axial plus Bending check
- pass/fail status for the Axial plus Bending check
- location along the member length where the critical Axial plus Bending result was found to occur
- governing load combination that results in the maximum Shear stress ratio
- maximum stress ratio for the Shear check
- pass/fail status for the Shear check
- location along the member length where the critical Shear result was found to occur.

The image below shows the results for the steel members used in this frame:

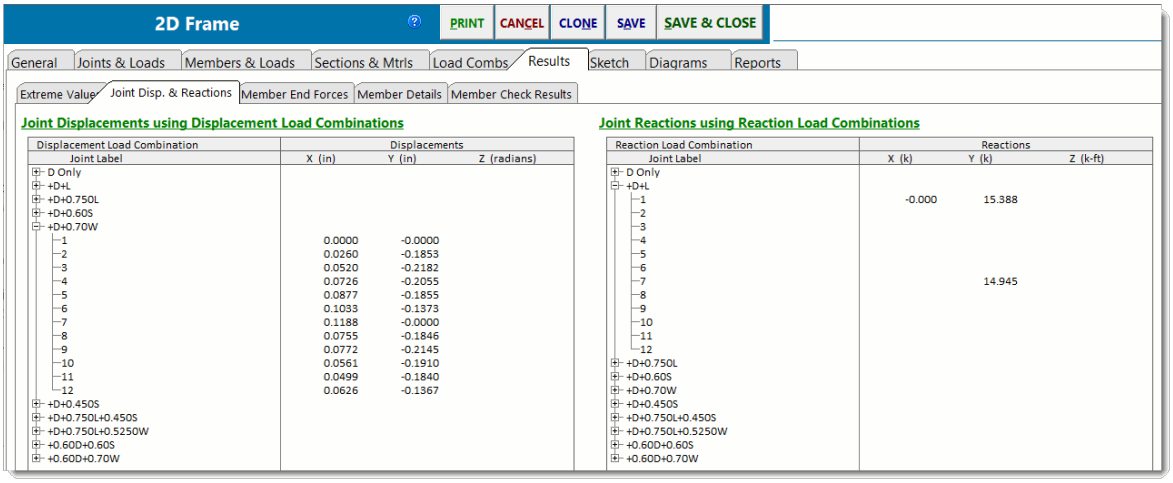


Member Data			Max. Axial+Bending Stress Ratios				Max Shear Ratios			
Label	Section	Material	Load Combination	Ratio	Status	Dist (ft)	Load Combination	Ratio	Status	Dist (ft)
1-2	BotChord	Steel	+D+0.750L+0.5250W	0.5726	PASS	3.918	D Only	0.0331	PASS	0.000
1-8	TopChord	Steel	+D+0.70W	0.7443	PASS	4.381	+D+0.70W	0.0646	PASS	0.000
10-11	TopChord	Steel	+D+0.60S	0.5237	PASS	4.381	+D+0.60S	0.0455	PASS	0.000
11-12	TopChord	Steel	+D+0.60S	0.5237	PASS	4.381	+D+0.60S	0.0455	PASS	0.000
2-3	BotChord	Steel	+D+0.750L+0.5250W	0.5726	PASS	3.918	D Only	0.0331	PASS	0.000
2-8	Webs	Steel	+D+0.450S	0.0176	PASS	0.000	N/A	0.0000	PASS	0.000
3-4	BotChord	Steel	+D+0.750L+0.450S	0.5268	PASS	3.918	D Only	0.0331	PASS	0.000
3-8	Webs	Steel	+D+0.70W	0.1618	PASS	0.000	N/A	0.0000	PASS	0.000
3-9	Webs	Steel	+D+0.70W	0.0516	PASS	0.000	N/A	0.0000	PASS	0.000
4-10	Webs	Steel	+D+0.750L+0.450S	0.1986	PASS	0.000	N/A	0.0000	PASS	0.000
4-11	Webs	Steel	+D+0.60S	0.2439	PASS	0.000	N/A	0.0000	PASS	0.000
4-5	BotChord	Steel	+D+0.750L+0.450S	0.5221	PASS	3.918	D Only	0.0331	PASS	0.000
4-9	Webs	Steel	+D+0.70W	0.3102	PASS	0.000	N/A	0.0000	PASS	0.000
5-11	Webs	Steel	+D+0.60S	0.0441	PASS	0.000	N/A	0.0000	PASS	0.000
5-12	Webs	Steel	+D+0.60S	0.1262	PASS	0.000	N/A	0.0000	PASS	0.000
5-6	BotChord	Steel	+D+0.750L+0.450S	0.5613	PASS	3.918	D Only	0.0331	PASS	0.000
6-12	Webs	Steel	D Only	0.0176	PASS	0.000	N/A	0.0000	PASS	0.000
6-7	BotChord	Steel	+D+0.750L+0.450S	0.5613	PASS	3.918	D Only	0.0331	PASS	0.000
7-12	TopChord	Steel	+D+0.60S	0.7330	PASS	4.563	+D+0.60S	0.0455	PASS	0.000
8-9	TopChord	Steel	+D+0.70W	0.8333	PASS	4.381	+D+0.70W	0.0646	PASS	0.000
9-10	TopChord	Steel	+D+0.750L+0.450S	0.9315	PASS	2.921	+D+L	0.0626	PASS	0.000

© 2026 ENERCALC, LLC

Joint Displacements & Reactions

This list summarizes the joint displacements resulting from each load combination.



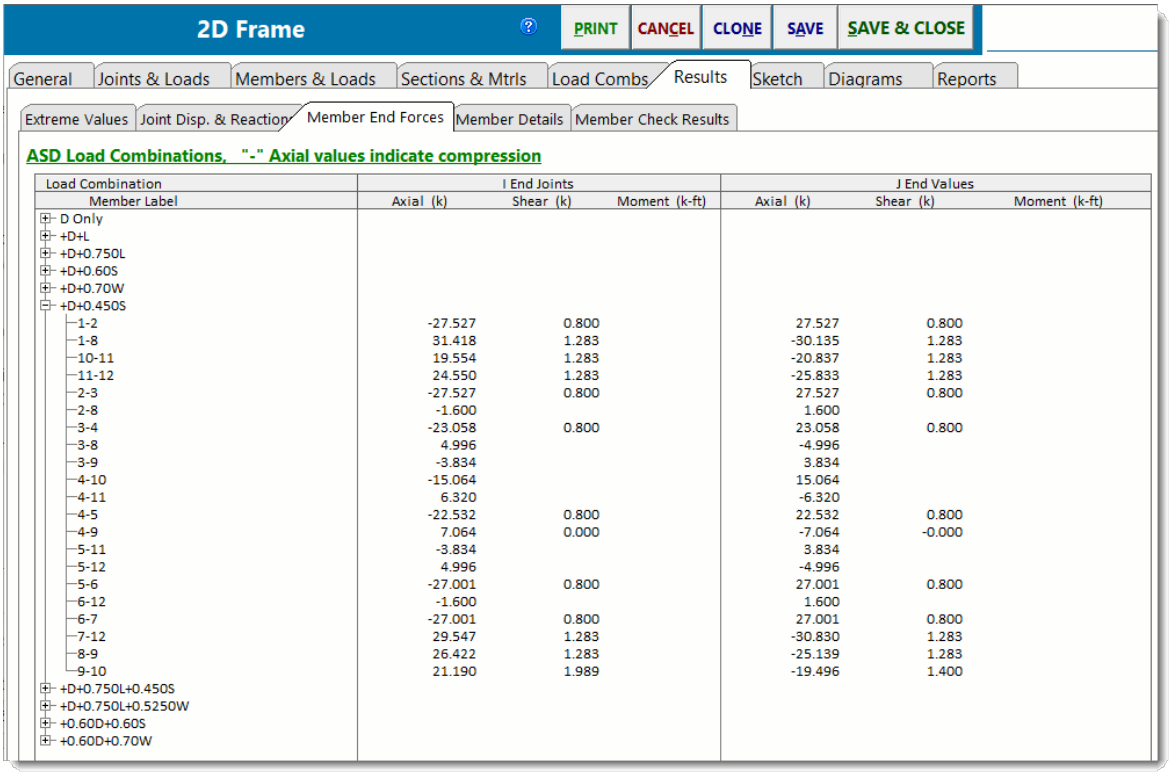
Joint displacements are reported with respect to the global coordinate system.

Clicking on the [**+**] icon to the left of each combination will expand the sub-list to show the detailed values.

© 2026 ENERCALC, LLC

Member End Forces

This list summarizes the member end forces resulting from each load combination.



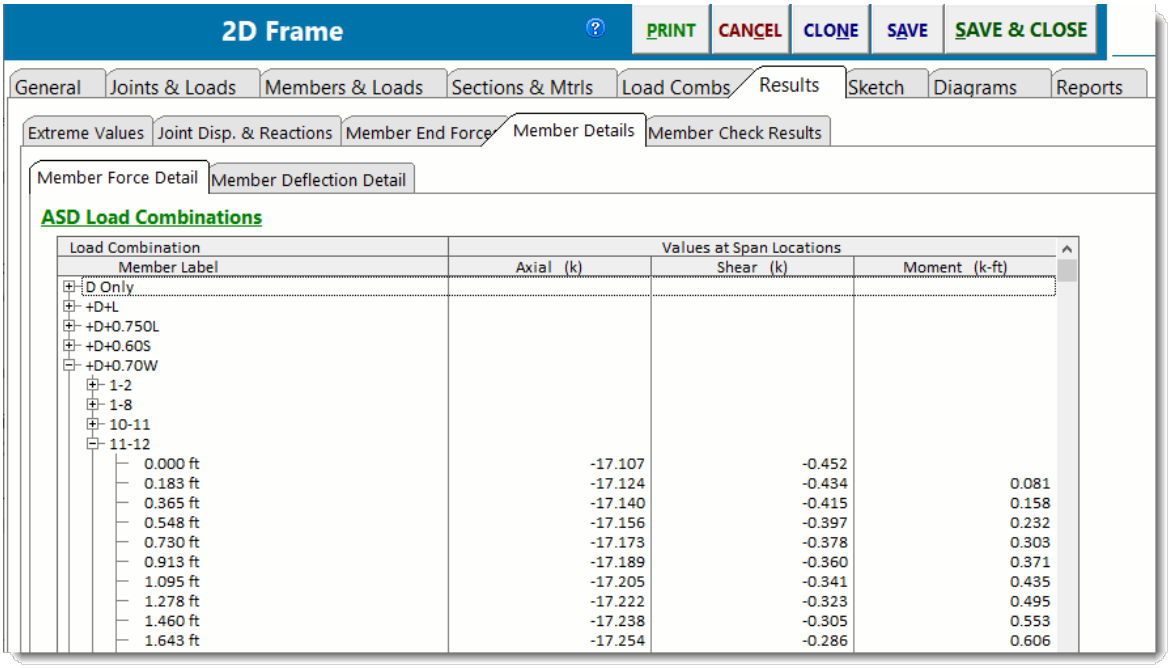
Load Combination / Member Label	I End Joints Axial (k)	I End Joints Shear (k)	I End Joints Moment (k-ft)	J End Values Axial (k)	J End Values Shear (k)	J End Values Moment (k-ft)
D Only						
+D+L						
+D+0.750L						
+D+0.60S						
+D+0.70W						
+D+0.450S						
1-2	-27.527	0.800		27.527	0.800	
1-8	31.418	1.283		-30.135	1.283	
10-11	19.554	1.283		-20.837	1.283	
11-12	24.550	1.283		-25.833	1.283	
2-3	-27.527	0.800		27.527	0.800	
2-8	-1.600			1.600		
3-4	-23.058	0.800		23.058	0.800	
3-8	4.996			-4.996		
3-9	-3.834			3.834		
4-10	-15.064			15.064		
4-11	6.320			-6.320		
4-5	-22.532	0.800		22.532	0.800	
4-9	7.064	0.000		-7.064	-0.000	
5-11	-3.834			3.834		
5-12	4.996			-4.996		
5-6	-27.001	0.800		27.001	0.800	
6-12	-1.600			1.600		
6-7	-27.001	0.800		27.001	0.800	
7-12	29.547	1.283		-30.830	1.283	
8-9	26.422	1.283		-25.139	1.283	
9-10	21.190	1.989		-19.496	1.400	
+D+0.750L+0.450S						
+D+0.750L+0.5250W						
+0.60D+0.60S						
+0.60D+0.70W						

Clicking on the [**+**] icon to the left of each combination will expand the sub-list to show the detailed values.

© 2026 ENERCALC, LLC

Member Details

This list gives a very detailed presentation of the member forces and member deflections at small increments along the member length.



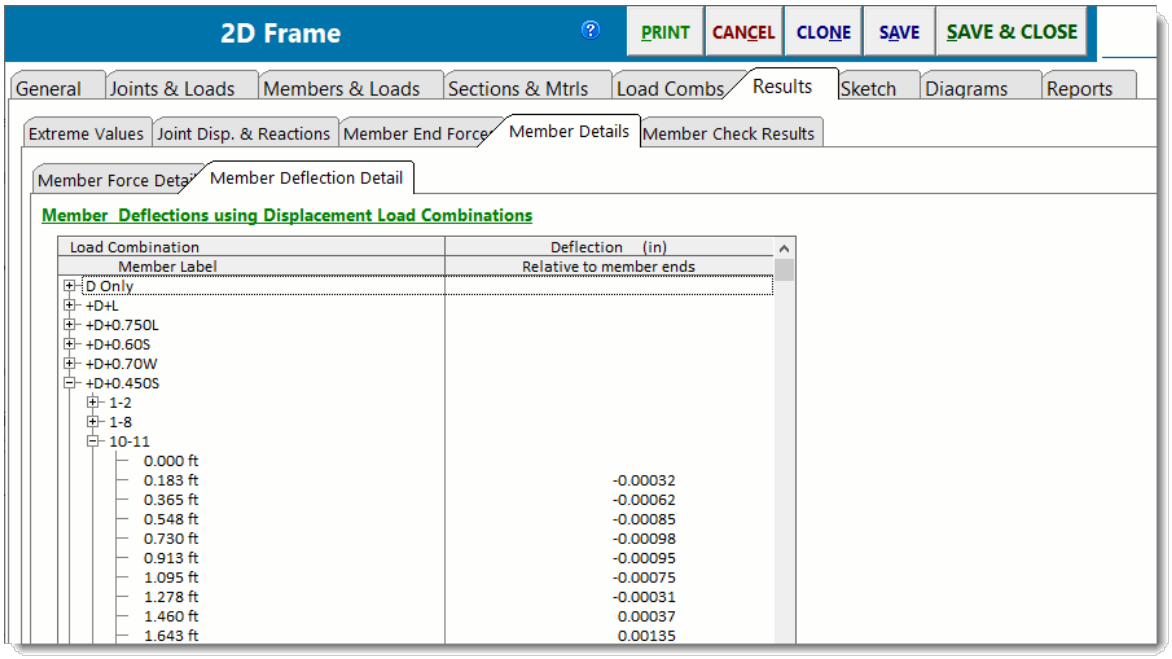
The list is a tree with two sub-levels:

- The main level allows a choice of the load combination.
- The next level down offers the choice of which member to observe.

Click on the [**+**] icons to expand the list to show more details. Click on the [**-**] icon will compress the level.

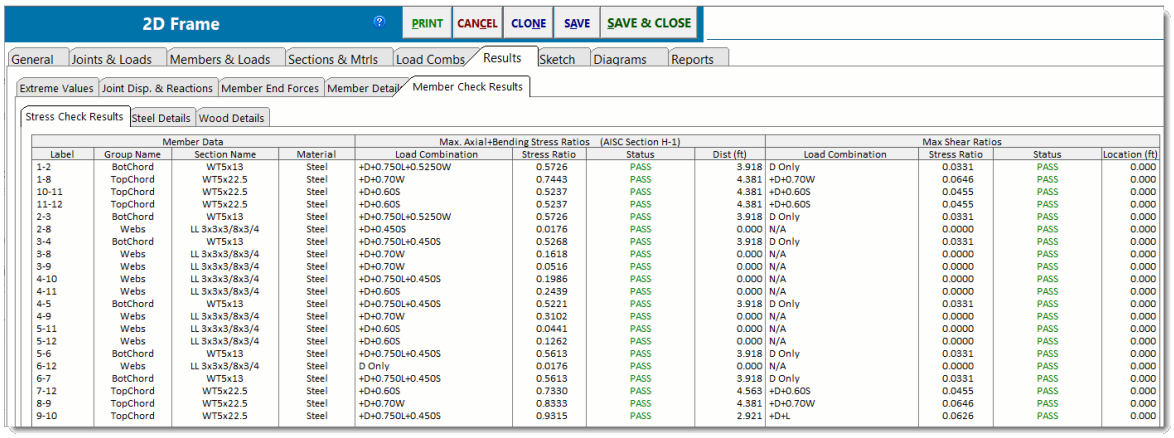
© 2026 ENERCALC, LLC

Note that the Member Deflections provided in this list are reported relative to the straight-line chord drawn between the *deflected* position of the two end nodes of the member. In other words, these Member Deflections will ALWAYS report a value of zero at both ends of all members.



Member Check Results

The Member Check Results tab is only shown when the Member Stress Check Status item on the General tab has been set to ASD or LRFD stress checks. The image below shows the results for the steel members used in this frame:



© 2026 ENERCALC, LLC

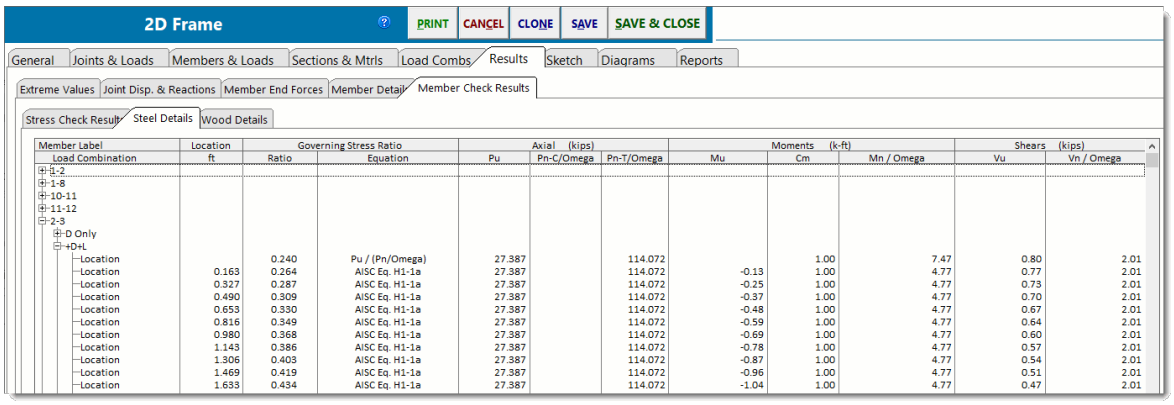
Stress Check Results

For each member this list shows the following information:

- member label
- section or group label
- material
- governing load combination that results in the maximum stress ratios
- maximum stress ratio for the Axial plus Bending check
- pass/fail status for the Axial plus Bending check
- location along the member length where the critical Axial plus Bending result was found to occur
- maximum stress ratio for the Shear check
- pass/fail status for the Shear check
- location along the member length where the critical Shear result was found to occur.

Steel Details

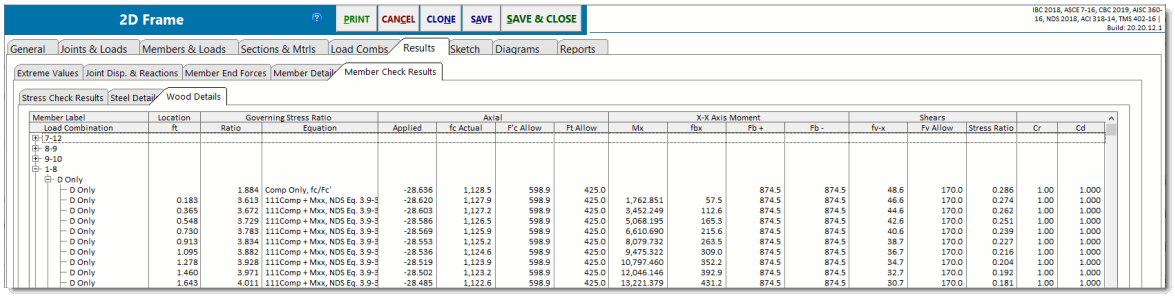
This list pertains only to the stress checks for steel members, and it provides detailed information about the stress checks at small increments along the length of each member.



Member Label / Load Combination	Location	Governing Stress Ratio		Axial (kips)			Moments (k-ft)			Shears (kips)	
	ft	Ratio	Equation	Pu	Pn-C/Omega	Pn-T/Omega	Mu	Cm	Mn / Omega	Vu	Vn / Omega
1-2											
1-8											
10-11											
11-12											
2-3											
D Only											
+D+L											
Location		0.240	Pu / (Pn/Omega)	27.387		114.072		1.00	7.47	0.80	2.01
Location	0.163	0.264	AISC Eq. H1-1a	27.387		114.072	-0.13	1.00	4.77	0.77	2.01
Location	0.327	0.287	AISC Eq. H1-1a	27.387		114.072	-0.25	1.00	4.77	0.73	2.01
Location	0.490	0.309	AISC Eq. H1-1a	27.387		114.072	-0.37	1.00	4.77	0.70	2.01
Location	0.653	0.330	AISC Eq. H1-1a	27.387		114.072	-0.48	1.00	4.77	0.67	2.01
Location	0.816	0.349	AISC Eq. H1-1a	27.387		114.072	-0.59	1.00	4.77	0.64	2.01
Location	0.980	0.368	AISC Eq. H1-1a	27.387		114.072	-0.69	1.00	4.77	0.60	2.01
Location	1.143	0.386	AISC Eq. H1-1a	27.387		114.072	-0.78	1.00	4.77	0.57	2.01
Location	1.306	0.403	AISC Eq. H1-1a	27.387		114.072	-0.87	1.00	4.77	0.54	2.01
Location	1.469	0.419	AISC Eq. H1-1a	27.387		114.072	-0.96	1.00	4.77	0.51	2.01
Location	1.633	0.434	AISC Eq. H1-1a	27.387		114.072	-1.04	1.00	4.77	0.47	2.01

Wood Details

This list pertains only to the stress checks for wood members, and it provides detailed information about the stress checks at small increments along the length of each member.



Member Label / Load Combination	Location	Governing Stress Ratio		Axial				X-X Axis Moment				Shears				
	ft	Ratio	Equation	Applied	fc Actual	F'c Allow	Ft Allow	Mx	fbx	Fb +	Fb -	fv-x	Fv Allow	Stress Ratio	Cr	Cd
7-12																
8-9																
9-10																
1-8																
D Only																
D Only		1.884	Comp Only, fc/Fc'	-28.636	1,128.5	598.9	425.0			874.5	874.5	48.6	170.0	0.286	1.00	1.000
D Only	0.183	3.613	111Comp + Mxx, NDS Eq. 3.9-3	-28.620	1,127.9	598.9	425.0	1,762.851	57.5	874.5	874.5	46.6	170.0	0.274	1.00	1.000
D Only	0.365	3.672	111Comp + Mxx, NDS Eq. 3.9-3	-28.603	1,127.2	598.9	425.0	3,452.249	112.6	874.5	874.5	44.6	170.0	0.262	1.00	1.000
D Only	0.548	3.729	111Comp + Mxx, NDS Eq. 3.9-3	-28.586	1,126.5	598.9	425.0	5,068.195	165.3	874.5	874.5	42.6	170.0	0.251	1.00	1.000
D Only	0.730	3.783	111Comp + Mxx, NDS Eq. 3.9-3	-28.569	1,125.9	598.9	425.0	6,610.690	215.6	874.5	874.5	40.6	170.0	0.239	1.00	1.000
D Only	0.913	3.834	111Comp + Mxx, NDS Eq. 3.9-3	-28.553	1,125.2	598.9	425.0	8,079.732	263.5	874.5	874.5	38.7	170.0	0.227	1.00	1.000
D Only	1.095	3.882	111Comp + Mxx, NDS Eq. 3.9-3	-28.536	1,124.6	598.9	425.0	9,475.322	309.0	874.5	874.5	36.7	170.0	0.216	1.00	1.000
D Only	1.278	3.928	111Comp + Mxx, NDS Eq. 3.9-3	-28.519	1,123.9	598.9	425.0	10,797.460	352.2	874.5	874.5	34.7	170.0	0.204	1.00	1.000
D Only	1.460	3.971	111Comp + Mxx, NDS Eq. 3.9-3	-28.502	1,123.2	598.9	425.0	12,046.146	392.9	874.5	874.5	32.7	170.0	0.192	1.00	1.000
D Only	1.643	4.011	111Comp + Mxx, NDS Eq. 3.9-3	-28.485	1,122.6	598.9	425.0	13,221.379	431.2	874.5	874.5	30.7	170.0	0.181	1.00	1.000

© 2026 ENERCALC, LLC

Sign Convention

Coordinate Axis Convention

The right-hand rule states that if you take the vector cross product of X cross Y, the result is in the Z direction. This is what is used to establish the positive Z direction if you know the positive X and Y directions. It applies to the Global Coordinate Axis system and the member local coordinate axis system in the 2-D Frame Analysis module, in the 2-D Frame Analysis module.

The Global Coordinate Axis system is oriented such that X points to the right, Y points upward, and Z is perpendicular to the screen in the 2-D Frame Analysis module.

The member local coordinate axis system is established as follows. A vector from the I node to the J node establishes the member local x axis. The vector cross product of local x cross Global Y produces the member local z direction. This works for all member orientations except for vertically oriented members, because it is not possible to take the vector cross product of two parallel vectors. So in those cases the module adopts the convention that the member local z axis will be parallel to the Global Z axis. These are just mathematical rules that establish that member local z will be perpendicular to the screen (unless you have specified that the member is working in weak axis bending). The local z axis is perpendicular to the web of a wide flange section. It typically represents the “strong” axis of a member. Finally, we need to establish the orientation of the local y axis. Vector cross product rules for a right-hand coordinate system also state that local z cross local x produces local y. So this pins down the orientation of the local y axis. The local y axis of a wide flange member lies in the plane of the web and is mutually perpendicular to the local x and local z axes. For a horizontally oriented beam member, the local y axis points straight up, parallel to the Global Y axis.

Result Sign Convention

Now for the sign conventions of the various results that are available.

Joint Displacements

Joint Displacements are reported with respect to the Global Coordinate Axis system. A positive displacement indicates a displacement in the direction of the positive corresponding axis. A positive rotation indicates a positive rotation about the Global Z axis. (Using the thumb of the right hand, point the thumb in the direction of the positive Global Z axis, and the natural curl of the right fingers will indicate the direction of a positive rotation.)

© 2026 ENERCALC, LLC

Reactions

Reactions are also reported with respect to the Global Coordinate Axis system. A positive force reaction indicates a force in the direction of the positive corresponding axis. A positive moment reaction indicates a positive moment about the Global Z axis. (Using the thumb of the right hand, point the thumb in the direction of the positive Global Z axis, and the natural curl of the right fingers will indicate the direction of a positive moment.)

Member End Forces

Member End Forces are reported with respect to the member local coordinate axis system. A positive value of axial load at the I end of the member means that the force acts in the member local x direction, so it is pushing into the starting end of the member, so it represents a compressive force. Likewise, a negative value of axial load at the J end of the member means that the force acts in the member local -x direction, so it is pushing into the ending end of the member, so again this represents a compressive force.

Shears are also reported with respect to the member local coordinate axis system. A positive value of shear at the I end of the member means that the force acts in the member local y direction. Likewise, a negative value of shear at the J end of the member means that the force acts in the member local -y direction.

A positive value of moment at either end of the member means that the moment acts in the member local z direction. (Using the thumb of the right hand, point the thumb in the direction of the positive Global Z axis, and the natural curl of the right fingers will indicate the direction of a positive moment.)

Member Forces at Sections

The physical sense of the member forces at sections can best be established by relating to the physical sense of the member end forces at the starting end of the member as described above.

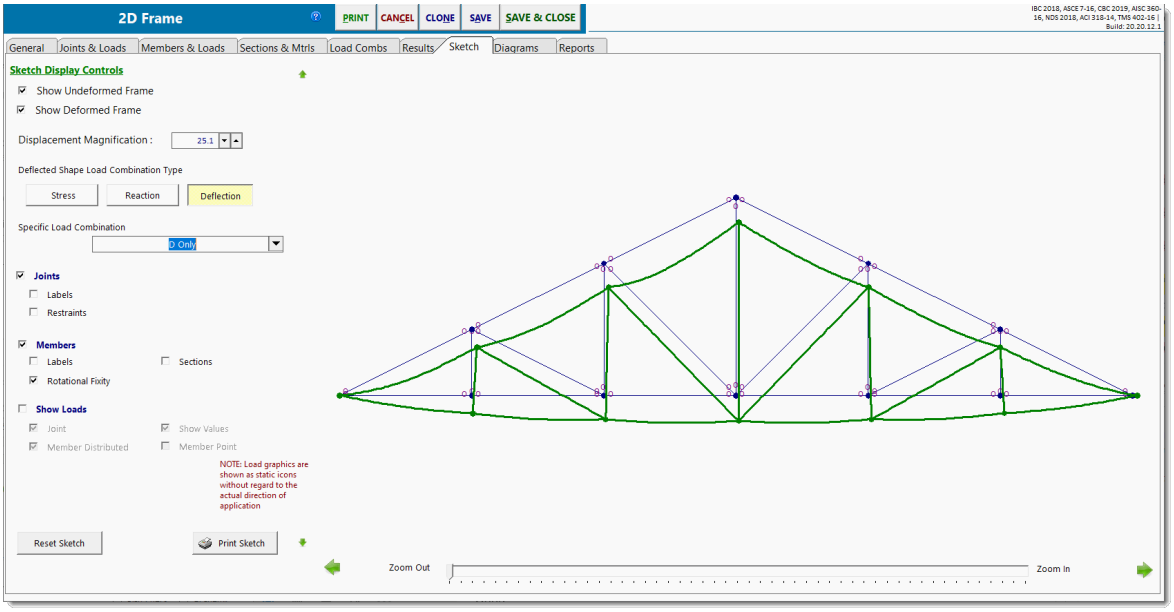
Deflections (Relative to member ends)

The deflections relative to member ends are measured parallel to the member local y axis and are referenced from the straight-line chord connecting the undeflected end node locations. Positive values represent deflections in the positive local y axis direction from that straight-line chord.

© 2026 ENERCALC, LLC

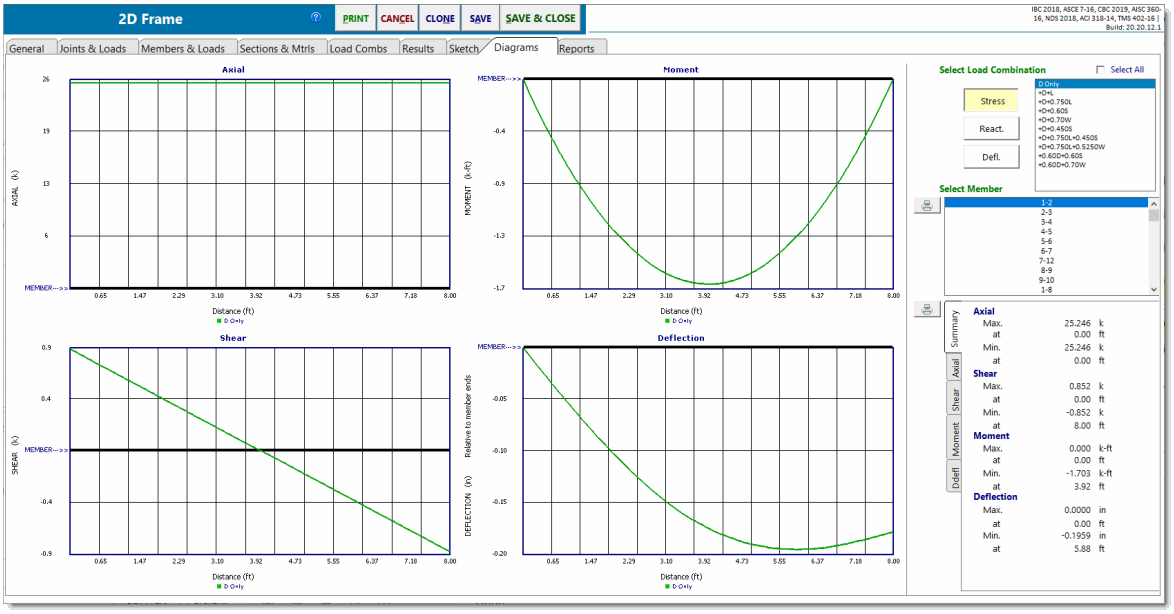
13.3.2.8 Frame Sketch

This tab provides a graphical display of the frame. You can use various check boxes, displacement magnification and load combination selection items to control the display.



13.3.2.9 Diagrams

This tab allows you to display axial load, moment, shear and deflection diagrams for each individual member in the frame.



© 2026 ENERCALC, LLC

Select Load Combination

This box lists all of the load combinations that are being run. Click on any of the available load combinations in the list to view diagrams based on that load combination.

Select Member

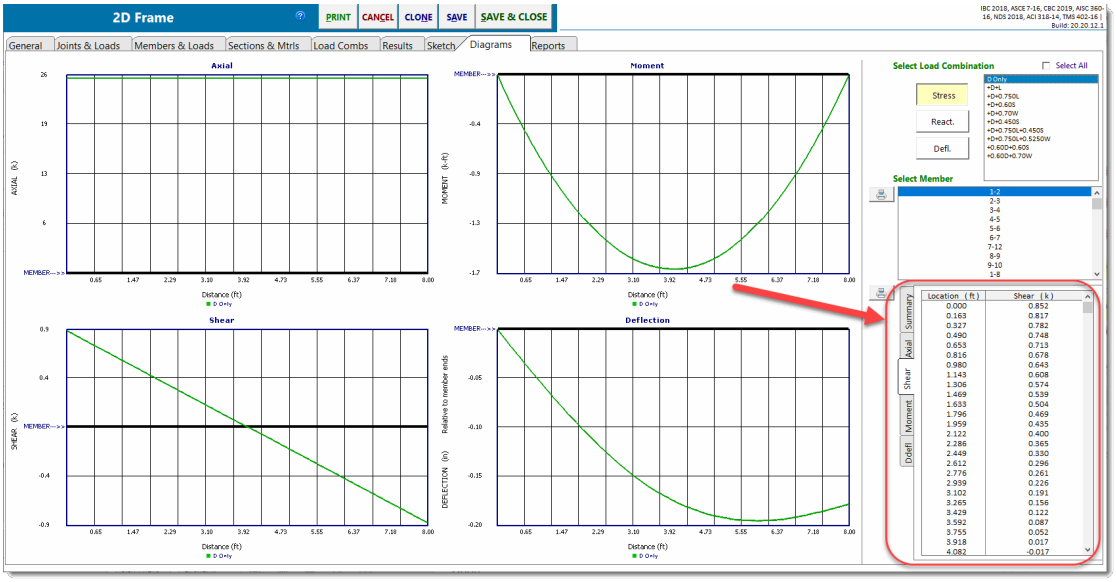
This box lists all of the members in the frame. Click on any of the members in the list to view diagrams for that member.

Tabs Summary, Axial, Shear, Moment, Deflection

These tabs let you explore the force details for the selected member and load combination in different ways.

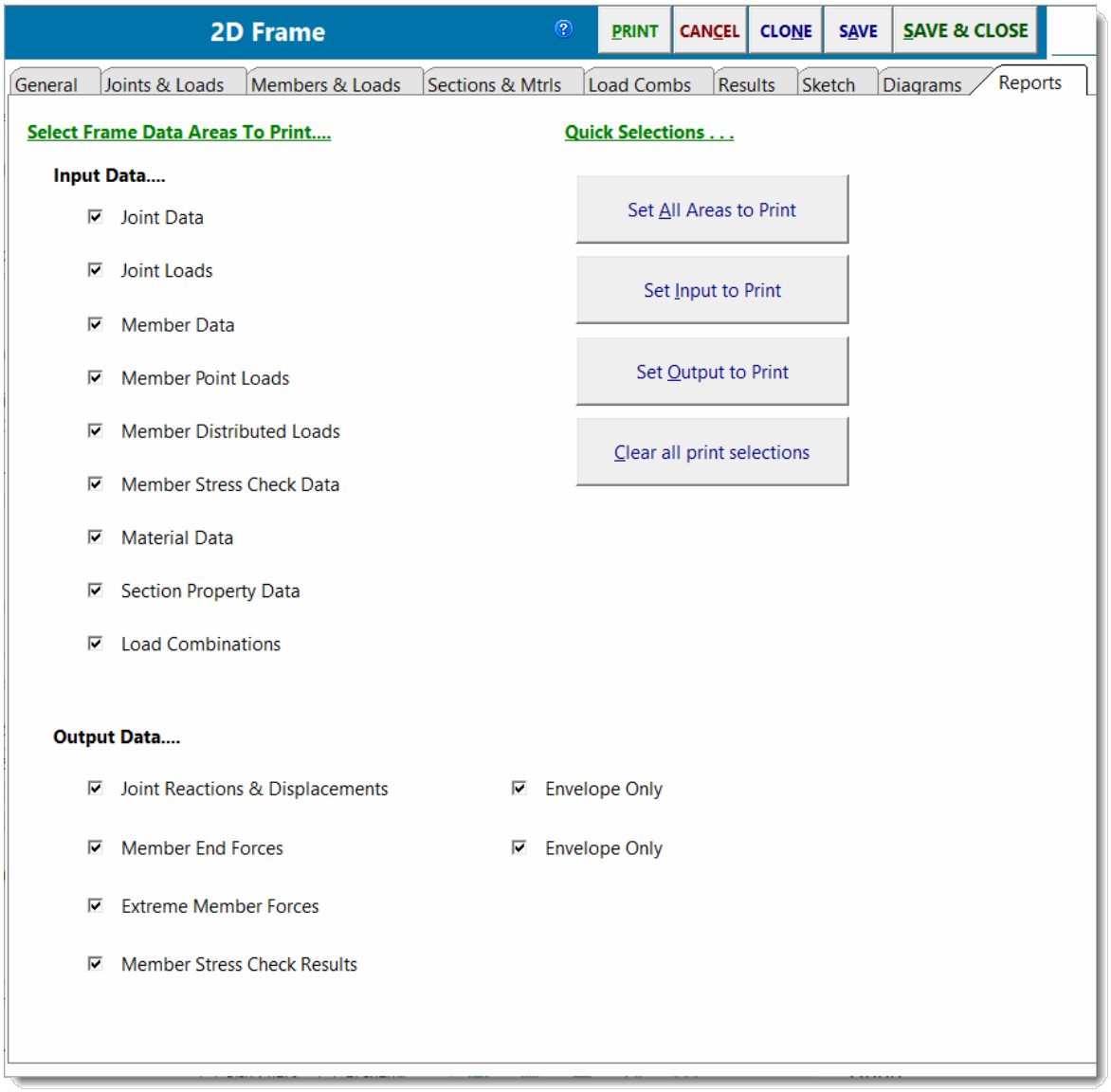
When the All tab is selected, the extreme values of axial load, moment, shear, and deflection are presented for the currently selected member.

The other tabs provide detailed list of the values for the currently selected which are used to create the respective diagrams.



13.3.2.10 Reports

This tab lets you select what data to print from the frame analysis.



Under Output Data you have the choice of compressing the output to an Envelope Only status. If Envelope Only is not checked the full details will be printed which can results in many pages of output. It is advisable to look at a print preview to see what it generates.

Using Envelope Only will examine each table and print only the extreme values for each load combination.

13.3.3 Torsional Analysis of Rigid Diphragm

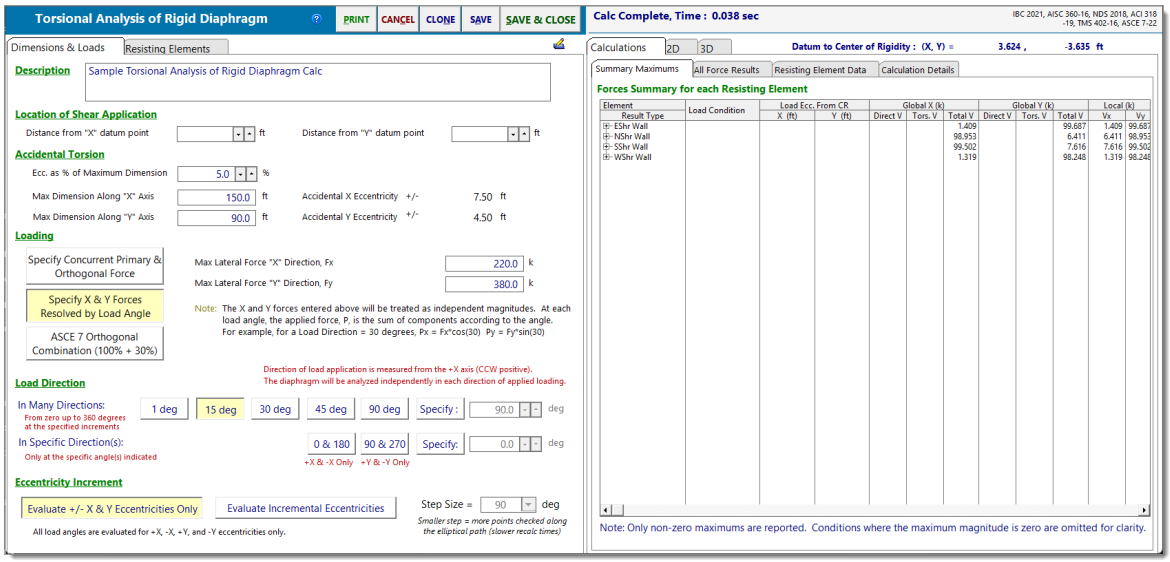
Verification Example

© 2026 ENERCALC, LLC

This module analyzes the distribution of lateral forces across a rigid diaphragm, accounting for both direct shear and torsional effects. It uses a stiffness-based approach to determine how lateral loads, typically from wind or seismic forces, are transferred to each resisting element based on its position, orientation, and relative stiffness, while also accounting for accidental eccentricity between the center of mass and center of rigidity.

Limitations:

- Assumes a rigid diaphragm; flexible and semi-rigid diaphragm effects are not included.
- Supports up to 160 different force resisting elements (walls, bending elements, or generic).
- Analysis is for one diaphragm level only. Multi-level buildings require separate calculations per level.
- Results depend on accurate element locations, stiffnesses, and orientations entered by the user.
- Does not model vertical alignment (i.e. when elements are offset between levels). The center of mass must be recalculated manually for each level.
- Neglects individual resisting element local torsional stiffness (J).



Unique Features

- *Automated Directional and Eccentricity Load Conditions* – The module automatically evaluates all combinations of lateral load directions and accidental eccentricity positions. Lateral forces can be applied at custom angular increments (e.g., 15°, 30°), and eccentricities are resolved along an elliptical path centered at the shear application point. This produces a full set of load conditions capturing both translational and torsional effects without requiring users to manually define multiple scenarios.

- *Rigorous Stiffness Analysis* – This module uses a numerical, stiffness-based approach to calculate how lateral forces are distributed and how torsional effects

© 2026 ENERCALC, LLC

develop. Unlike simplified methods that assume elements act only in orthogonal directions, this module supports elements oriented at any angle. Each element's stiffness is calculated about its local principal axes, transformed into the global coordinate system, and combined with all other elements to determine the overall stiffness matrix and center of rigidity. This enables a more complete and realistic evaluation of force distribution and torsional behavior.

- *Support for Multiple Element Types* – Walls, bending members, and generic resisting elements can be modeled together, each with independent stiffness properties, orientations, and boundary conditions, enabling realistic simulation of complex lateral force-resisting systems.

Coordinate System

Please ensure a strict X-Y coordinate system is used for accurate analysis. When setting up the model, keep in mind that Global +X increases to the right and Global +Y increases upward on the screen.

Analysis Procedure

For more details on how the lateral force-resisting system stiffness is calculated and how forces are resolved for each resisting element, please refer to the topic: Torsional Analysis of Rigid Diaphragm Theory 656

Basic Usage

1. **Define the Location of Shear Application:** Enter the X-Y coordinates where lateral forces act on the diaphragm. These typically represent seismic or wind loads at that level. If loads from other levels contribute, combine them into a single adjusted load location before input.

2. **Configure Accidental Eccentricity:** To account for seismic or wind-induced torsional effects, specify the accidental eccentricity as a percentage of the building's plan dimensions. You can also define an angular increment to control how many offset load locations (around the eccentricity ellipse) are evaluated. This enables the program to automatically generate all load conditions combining direction and eccentricity.

3. **Specify Lateral Loads:** Define the magnitude and direction of the diaphragm loads by choosing one of the following methods:
   a. Apply a primary force alone or with an orthogonal component;
   b. Resolve independent X and Y forces based on a specified load angle; or
   c. Use a 100% + 30% orthogonal combination method per ASCE 7

4. **Add Resisting Elements:** Use the Walls, Bending Members, or Generic subtab to define each resisting element. For each:

   a. **Enter the X and Y coordinates of the element's centroid**, which serves as its center of rigidity.

© 2026 ENERCALC, LLC

b. **Specify the orientation angle (θ)** of the element's local +y axis (strong axis), measured counterclockwise from the global +X axis.
   i. Positive angles rotate counterclockwise from the global +X axis.
   ii. Default orientation (θ = 0 degrees):
      1. Local +y (strong axis): Parallel to global +X (points right on screen)
      2. Local +x (weak axis): 90° clockwise from +y (points downward on screen)

c. **Enter geometric or section properties** appropriate to the element type (e.g., dimensions for walls, moments of inertia for bending members, or flexibilities for generic elements).

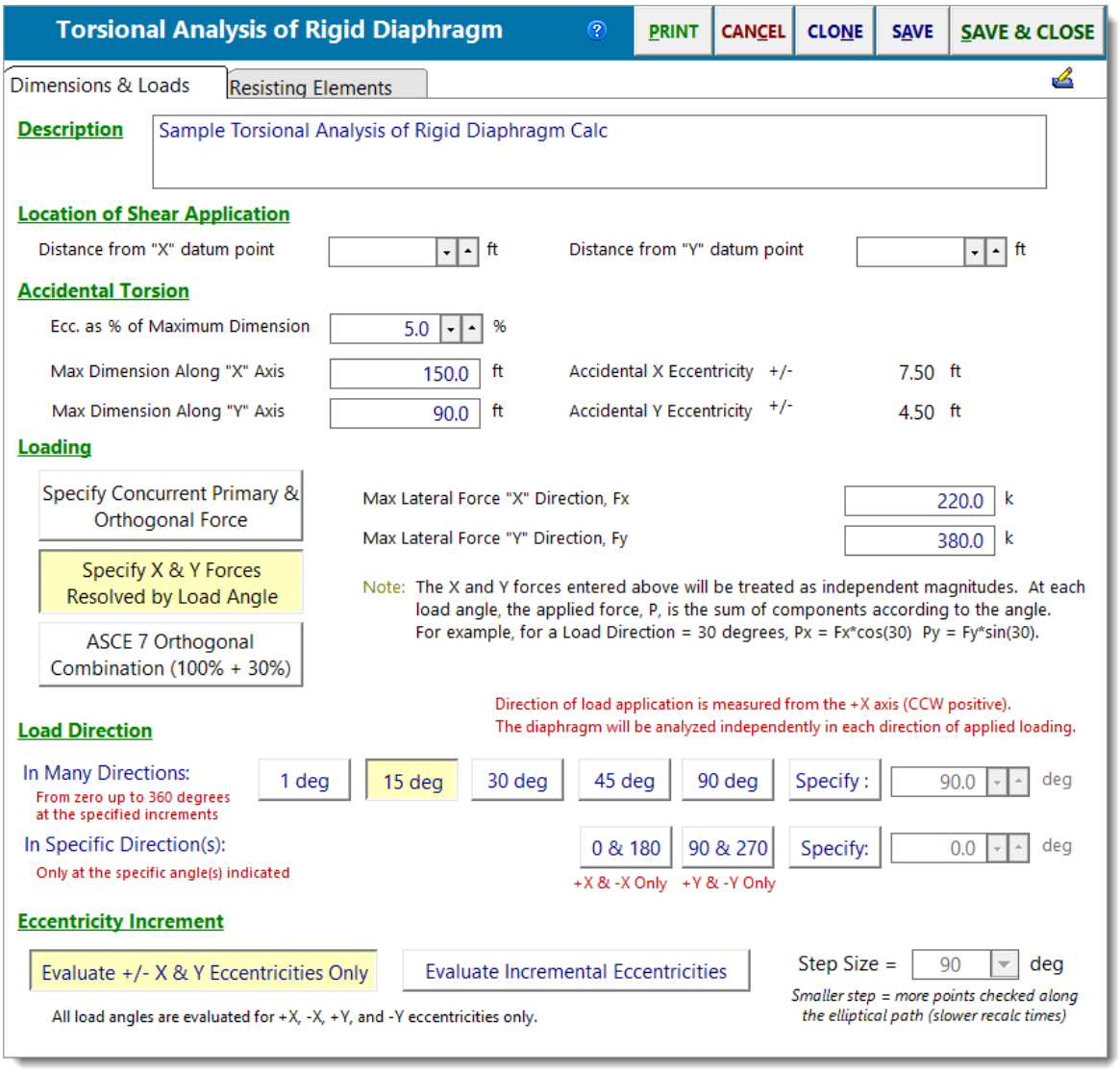
d. **Set Elastic Modulus:** If all elements are made of the same material, you may simplify input by using a relative modulus (e.g., Eb = 1). Only the relative stiffness between elements affects the analysis.

e. **Define Fixity Conditions:** Select the fixity condition that best represents each element's top and bottom restraint against rotation about both the local +x and +y axes:
   i. Use Fix-Pin for inverted pendulum conditions (e.g., cantilevered walls or columns with fixed bases and free tops), or for moment frames with pinned column bases.
   ii. Use Fix-Fix when both ends are restrained, resulting in double curvature behavior.

5. **Review Results:** The module automatically generates all load conditions based on load directions and eccentricity increments. Use the result tabs to view maximum forces per element, detailed forces by load case, and stiffness or displacement matrices for verification.

Dimensions & Loads

© 2026 ENERCALC, LLC



Location of Shear Application

This specifies the X-Y coordinates of the center of the load ellipse where the lateral shears act (e.g. center of mass for seismic loads, center of exposure for wind loads).

Accidental Torsion

Accidental torsion accounts for uncertainty in the exact location of a structure's center of rigidity and center of mass, which can cause additional torsion under seismic or wind loads. ASCE 7 accounts for this this by adding an accidental eccentricity; a small offset between the center of mass (for seismic loads) or center of exposure (for wind loads) and the center of rigidity, on top of any actual offset already present. This offset is typically a percentage of the overall building dimensions perpendicular to each direction.

© 2026 ENERCALC, LLC

Ecc as % of Maximum Dimension: Percent of building dimension to use for eccentricity (e.g., 5%).

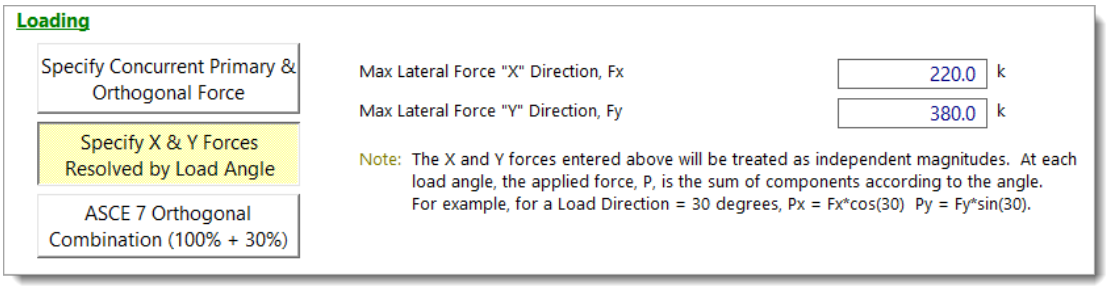
Max Dimension Along "X" Axis: Plan dimension used to calculate the accidental eccentricity in the X direction.

Max Dimension Along "Y" Axis: Plan dimension used to calculate the accidental eccentricity in the Y direction.

For more information regarding how this module accounts for the accidental eccentricity in the analysis, see the sections regarding the Eccentricity Increment 640 and How Load Direction and Eccentricity Increment Work Together 643.

Loading

These controls define how lateral forces are specified and combined for diaphragm loading.



Specify Concurrent Primary & Orthogonal Force

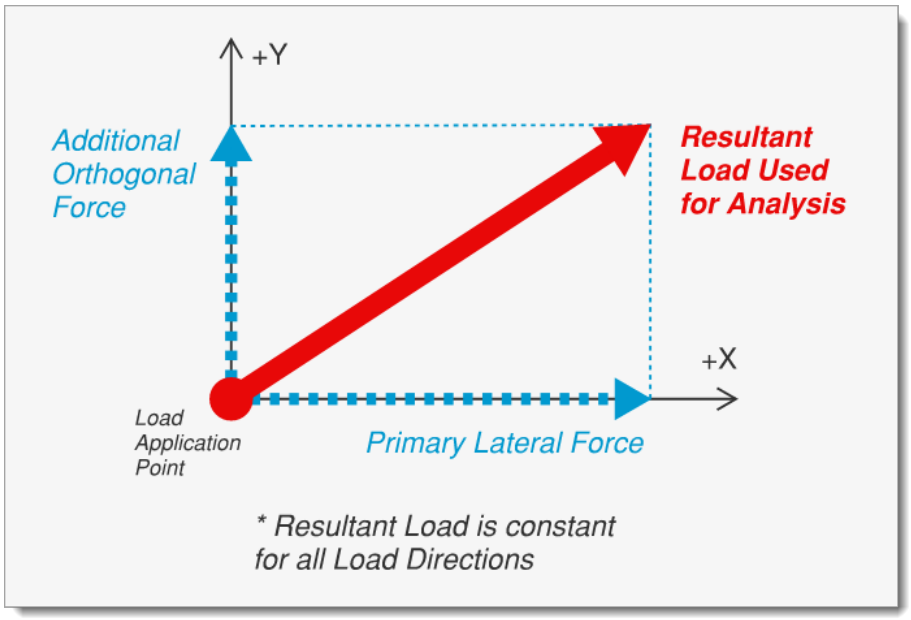
When this option is selected, the resultant of the primary and orthogonal forces is applied to the force resisting system at the angle or angular increments defined below under Load Direction 639. The Resultant Load is applied to the rigid diaphragm at the various load directions 639 and eccentricities 640 described below.

Primary Lateral Force: This is the primary lateral force to be applied to the rigid diaphragm.

Additional Orthogonal Force: This is an optional orthogonal force that is to be applied at a 90-degree angle to the Primary Lateral Force.

Resultant Load Used for Analysis: This is the resultant force which is actually applied to the diaphragm. It is calculated as the square root of the sum of the squares of the Primary Lateral Force and the Additional Orthogonal Force.

© 2026 ENERCALC, LLC



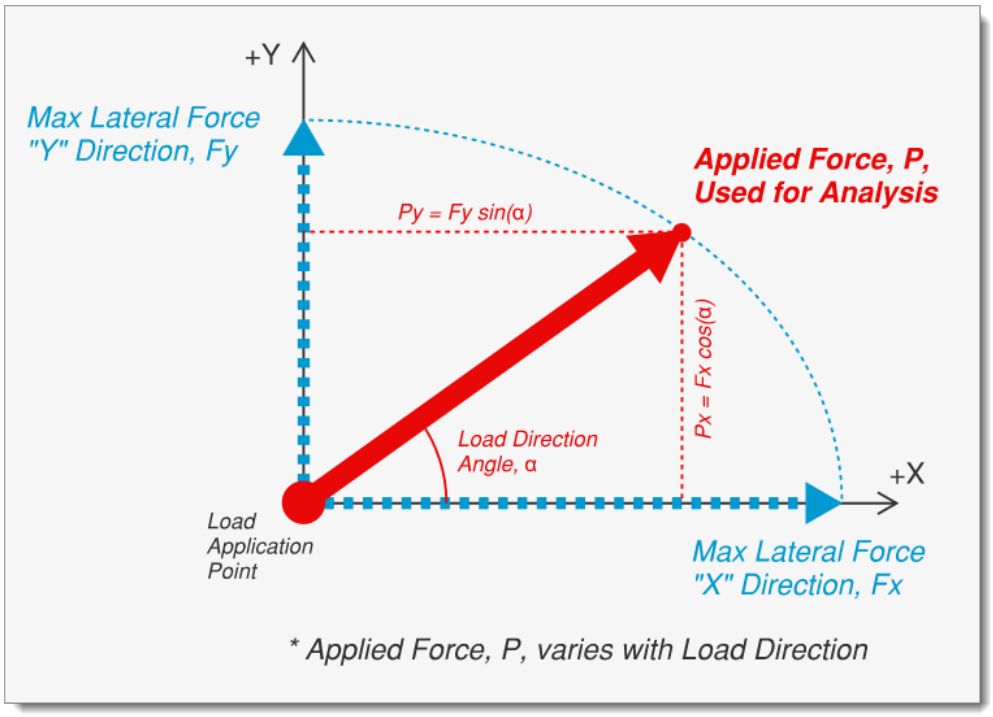
Specify X & Y Forces Resolved by Load Angle

When this option is selected, the X and Y forces will be treated as independent magnitudes. At each load angle, the applied force, P, is the vector sum of components according to the load angle. For example, for a Load Direction = 30 degrees, Px = Fx * cos(30°) and Py = Fy * sin(30°).

Max Lateral Force "X" Direction, Fx: This is the maximum lateral force in the +X & -X directions (i.e. when the load angle is exactly 0 or 180 degrees).

Max Lateral Force "Y" Direction, Fy: This is the maximum lateral force in the +Y & -Y directions (i.e. when the load angle is exactly 90 or 270 degrees).

© 2026 ENERCALC, LLC



ASCE 7 Orthogonal Combination (100% + 30%)

When this option is selected, the diaphragm is analyzed for a series of conditions where it is loaded simultaneously with 100% of lateral force in one direction and 30% of lateral force in the perpendicular direction.

Max Lateral Force "X" Direction, Fx: This is the maximum lateral force in the +X & -X directions.

Max Lateral Force "Y" Direction, Fy: This is the maximum lateral force in the +Y & -Y directions.

Conditions are permuted so that each axis experiences both +/- 100% and +/- 30% as described in the orthogonal combination procedure of ASCE 7. This results in the following 8 load conditions:

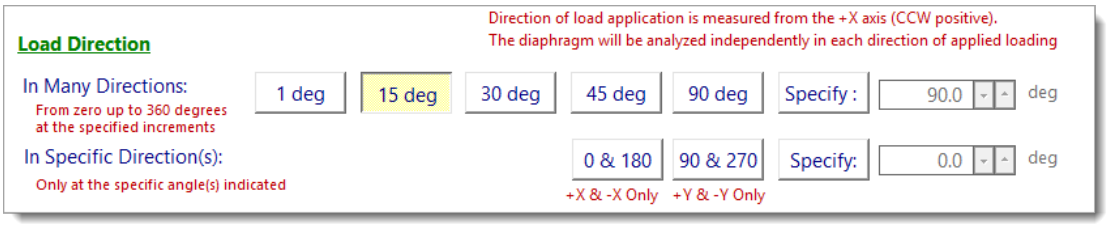
- +100% Fx + 30% Fy
- +100% Fx - 30% Fy
- -100% Fx + 30% Fy
- -100% Fx - 30% Fy
- +30% Fx + 100% Fy
- +30% Fx - 100% Fy
- -30% Fx + 100% Fy
- -30% Fx - 100% Fy

NOTE: *Load Direction inputs are not applicable when the ASCE 7 Orthogonal Combination option is selected. In this case, the Load Direction controls will be disabled, and all loads will be applied parallel to the global X and Y axes of the diaphragm.*

© 2026 ENERCALC, LLC

Load Direction

These controls determine the direction in which lateral forces are applied to the diaphragm.



In Many Directions

Per ASCE 7, seismic forces must be applied in the directions that produce the most critical load effects. This module can satisfy that requirement by applying the specified lateral loads at multiple angles, helping to identify the true critical direction for each resisting element.

When applying loads *In Many Directions*, the module will apply the specified lateral forces to the diaphragm at the chosen angular increment:

- 1 deg: Loads applied at 0°, 1°, 2°, etc.
- 15° deg: Loads applied at 0°, 15°, 30°, etc.
- 30° deg: Loads applied at 0°, 30°, 60°, etc.
- 45° deg: Loads applied at 0°, 45°, 90°, etc.
- 90° deg: Loads applied at 0°, 90°, 180°, etc.
- Specify: Loads at a custom increment (e.g., entering 20° applies loads at 0°, 20°, 40°, etc.).

TIP: *If you would like to utilize ASCE 7's Independent Directional Procedure and only apply the loads independently in two orthogonal directions (X & Y), this can be accomplished by using the In Many Directions > 90 deg option. This applies loads to the diaphragm at 0°, 90°, 180°, and 270°, corresponding to +X, +Y, -X, and -Y directions.*

In Specific Directions

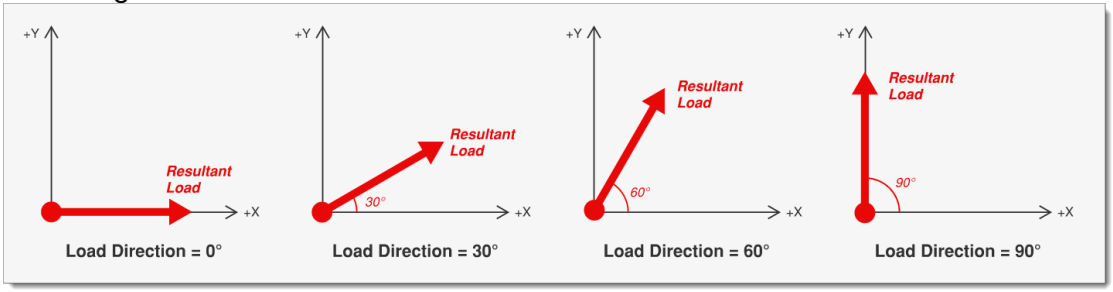
Instead of testing multiple angles, you can also apply loads at specific directions only:

- 0 & 180: Loads applied in +X and -X directions only.
- 90 & 270: Loads applied in +Y and -Y directions only.
- Specify: Loads applied at a single user-defined angle.

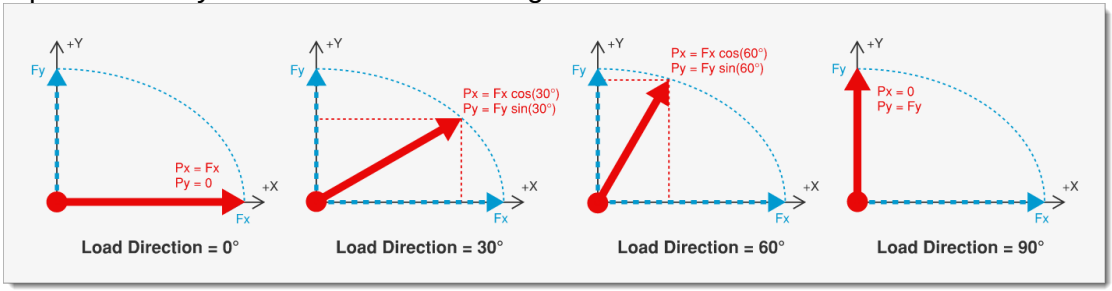
NOTE: *Direction of load application is measured from the +X axis (counterclockwise positive). The diaphragm will be analyzed independently in each direction of applied loading.*

© 2026 ENERCALC, LLC

The magnitude of the lateral force applied to the diaphragm is determined by the previously selected Loading 636 method. For example, if the *Specify Concurrent Primary & Orthogonal Force* mode is used with a 30° angular increment, the Resultant Load will be applied at 0°, 30°, 60°, and so on, continuing around the full 360° range.



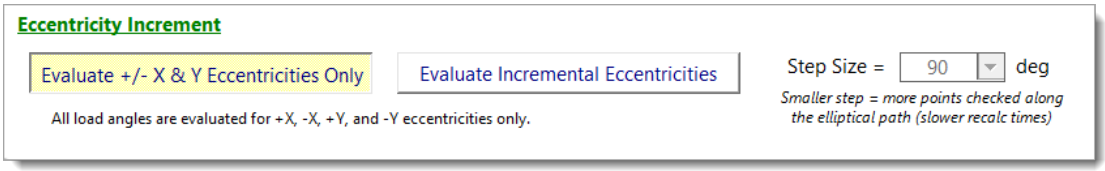
When using the *Specify X & Y Forces Resolved by Load Angle* Loading 636 method with a 30° angular increment, the applied force P is computed as the vector sum of the X and Y components corresponding to each load direction angle. This process repeats at every increment from 0° through 360°.



NOTE: *At 0° and 180°, the applied force, P, equals the maximum lateral force in the X direction (Fx). At 90° and 270°, it equals the maximum lateral force in the Y direction (Fy).*

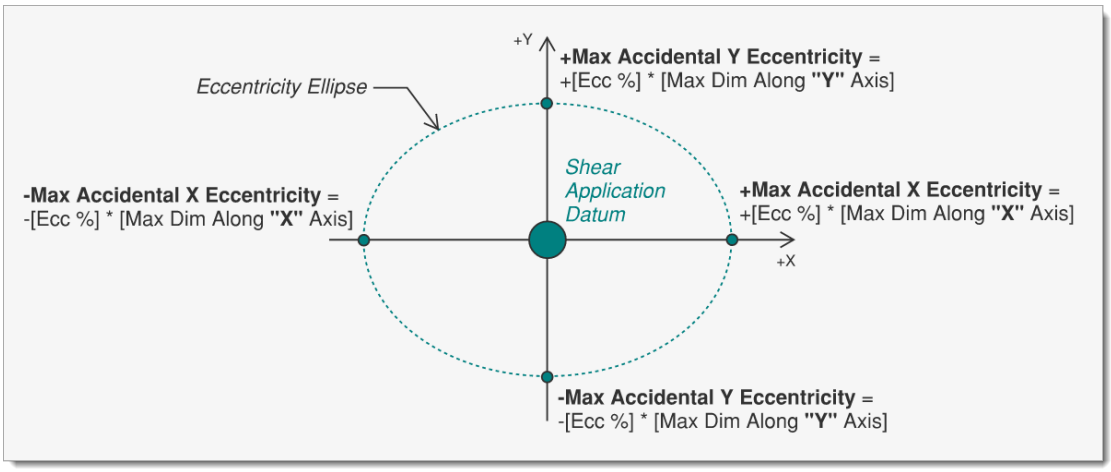
Eccentricity Increment

These settings control how the program accounts for accidental eccentricity when applying the lateral forces.

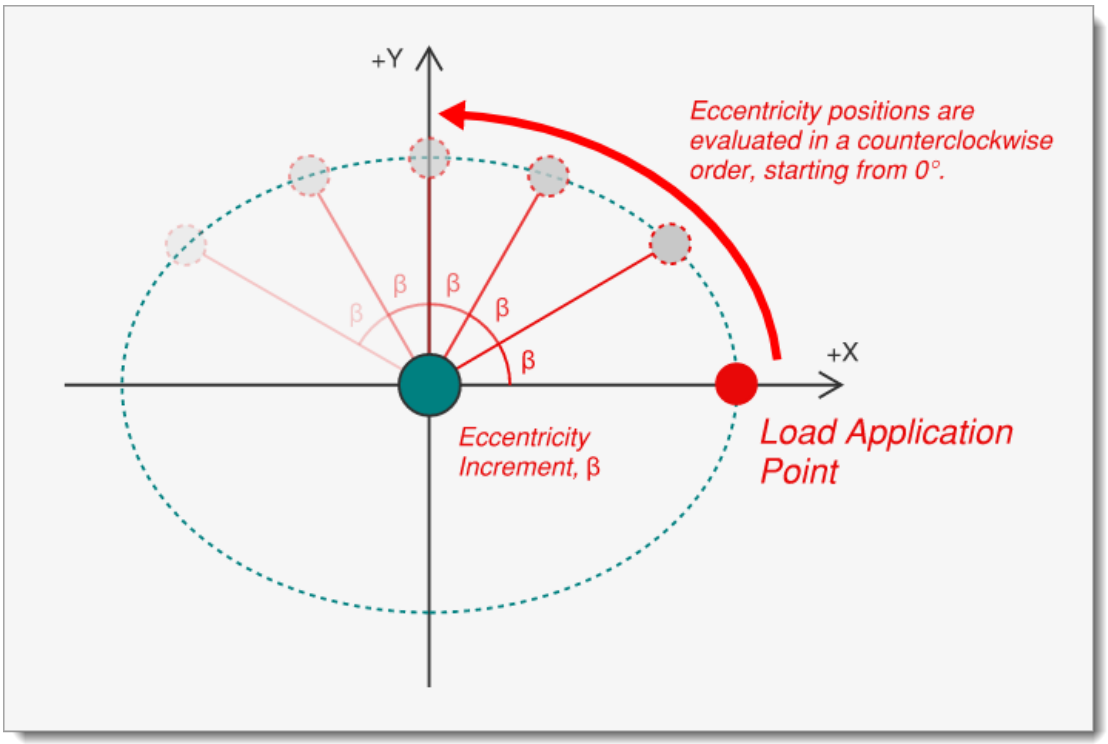


The X- and Y-direction accidental eccentricities from the Accidental Torsion 635 section define the axes of an ellipse centered at the shear application point. This ellipse represents all possible load locations that account for the full range of accidental eccentricity.

© 2026 ENERCALC, LLC



The *Eccentricity Increment* sets the angular step size (in degrees) used to subdivide the ellipse into discrete load application points. Smaller steps create more load cases and a finer resolution; larger steps reduce the number of cases.



Evaluate +/- X & Y Eccentricities Only

Applies loads at four discrete positions: +Max Accidental X Eccentricity, -Max Accidental X Eccentricity, +Max Accidental Y Eccentricity, and -Max Accidental Y Eccentricity (i.e., the poles of the eccentricity ellipse).

This matches a traditional code-based approach of checking maximum offsets in each principal direction.

© 2026 ENERCALC, LLC

Example:

Location of Shear Application: X = 50 ft, Y = 25 ft
Max Accidental X Eccentricity = 5 ft, Max Accidental Y Eccentricity = 2.5 ft

This gives the following load application points:

+Max Accidental X Eccentricity: $e_{acc,x}$ = 5.00, $e_{acc,y}$ = 0.00, x = 55.00, y = 25.00
+Max Accidental Y Eccentricity: $e_{acc,x}$ = 0.00, $e_{acc,y}$ = 2.50, x = 50.00, y = 27.50
-Max Accidental X Eccentricity: $e_{acc,x}$ = -5.00, $e_{acc,y}$ = 0.00, x = 45.00, y = 25.00
-Max Accidental Y Eccentricity: $e_{acc,x}$ = 0.00, $e_{acc,y}$ = -2.50, x = 50.00, y = 22.50

Evaluate Incremental Eccentricities

Applies loads at multiple points along the eccentricity ellipse, using the specified angular increment. This method evaluates a continuous range of eccentricity positions for a more detailed torsional assessment.

The accidental eccentricity and point of load application used for each step at angle θ can be generically described as:

$e_{acc,x} = (Max\ Accidental\ X\ Eccentricity) * \cos\theta$
$e_{acc,y} = (Max\ Accidental\ Y\ Eccentricity) * \sin\theta$
$x = (X\ Location\ of\ Shear\ Application) + e_{acc,x}$
$y = (Y\ Location\ of\ Shear\ Application) + e_{acc,y}$

Example:

Location of Shear Application: X = 50 ft, Y = 25 ft
Max Accidental X Eccentricity = 5 ft, Max Accidental Y Eccentricity = 2.5 ft
Eccentricity Increment: 45 degrees

This produces the following load application points:

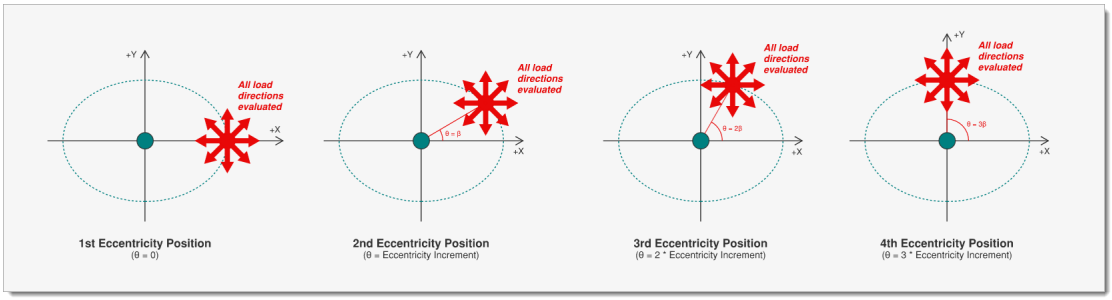
Eccentricity Increment Step = 0°: $e_{acc,x}$ = 5.00, $e_{acc,y}$ = 0.00, x = 55.00, y = 25.00
Eccentricity Increment Step = 45°: $e_{acc,x}$ = 3.54, $e_{acc,y}$ = 1.77, x = 53.54, y = 26.77
Eccentricity Increment Step = 90°: $e_{acc,x}$ = 0.00, $e_{acc,y}$ = 2.50, x = 50.00, y = 27.50
Eccentricity Increment Step = 135°: $e_{acc,x}$ = -3.54, $e_{acc,y}$ = 1.77, x = 46.46, y = 26.77
Eccentricity Increment Step = 180°: $e_{acc,x}$ = -5.00, $e_{acc,y}$ = 0.00, x = 45.00, y = 25.00
Eccentricity Increment Step = 225°: $e_{acc,x}$ = -3.54, $e_{acc,y}$ = -1.77, x = 46.46, y = 23.23
Eccentricity Increment Step = 270°: $e_{acc,x}$ = 0.00, $e_{acc,y}$ = -2.50, x = 50.00, y = 22.50

© 2026 ENERCALC, LLC

Eccentricity Increment Step = 315°: $e_{acc,x}$ = 3.54, $e_{acc,y}$ = -1.77, x = 53.54, y = 23.23

How Load Direction and Eccentricity Increment Work Together

The Load Direction 639 settings controls the direction of the applied lateral force, while the Eccentricity Increment 640 controls the position of that force around the eccentricity ellipse. The module evaluates every combination of these two settings, producing a broad set of load cases from which the maximum forces on each resisting element can be identified.



Example:

If the Load Direction is specified at 30 degree increments and the Eccentricity Increment is 15 degrees, the program evaluates:

- 360° / 30° = 12 load directions per eccentricity position
- 360° / 15° = 24 eccentricity positions

This results in 12 * 24 = 288 total rigid diaphragm analyses of force distribution to the resisting elements.

Resisting Elements

This tab is used to define the elements that make up the lateral force resisting system. There are three types of resisting elements: **Walls**, **Bending Members**, and **Generic Resisting Elements**. Each type has its own subtab where the specific properties and stiffness parameters are entered.

All subtabs share a common layout consisting of:

Copy, Add, & Delete Buttons

Use these buttons to create new resisting elements, duplicate an existing element, or remove an element from the list.

- **Add** creates a blank entry for a new element.
- **Copy** duplicates the currently selected element's data, which can then be edited.
- **Delete** removes the selected element from the list.

Element Table

© 2026 ENERCALC, LLC

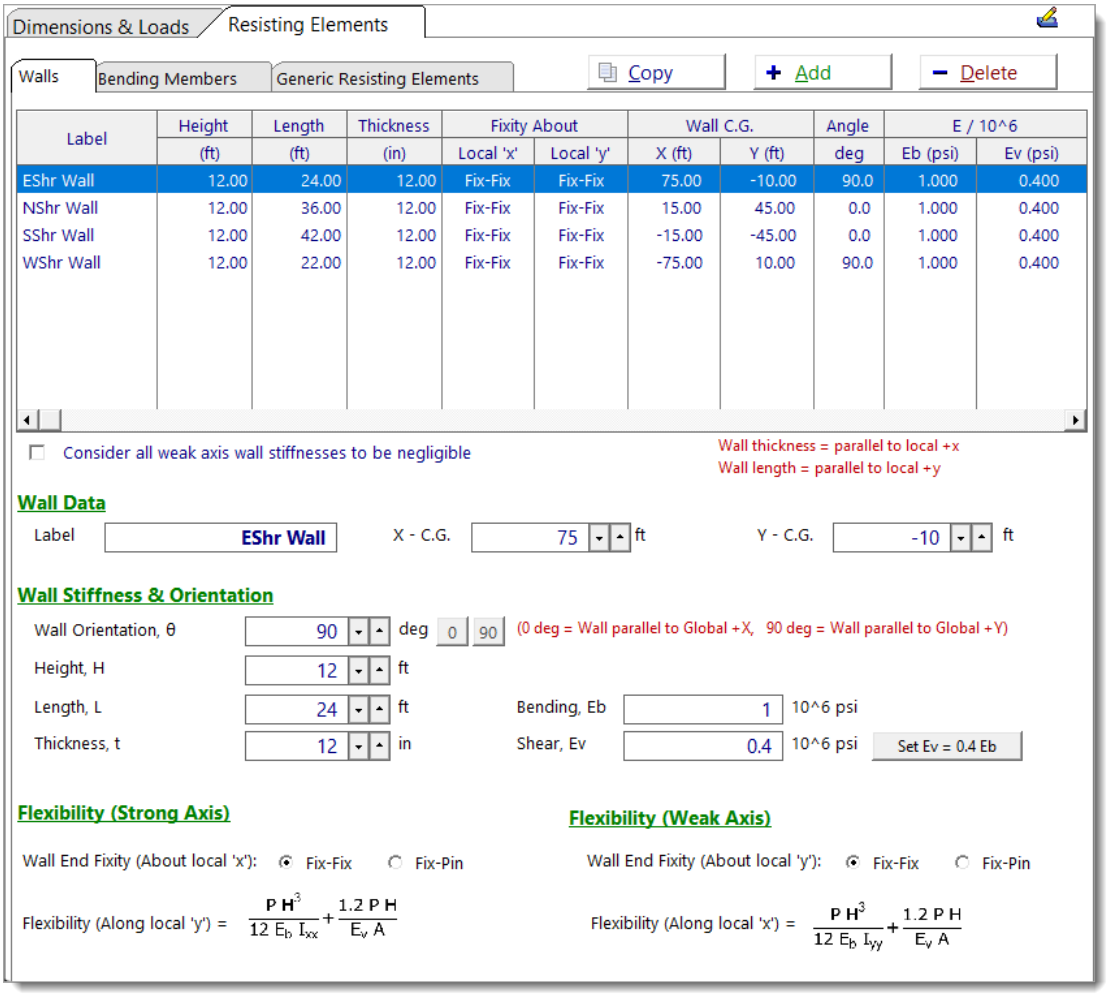
This table lists all resisting elements defined in the current subtab and provides a summary of their input data.

Element Input Data

This area contains the detailed input fields for the selected element, including geometry, material properties, orientation, and end fixity. The specific fields available depend on the element type:

Walls

Use the *Walls* tab to define a wall as a resisting element. Walls must be rectangular in plan and have a non-zero height.



Wall Data

Label: User-defined name for the wall.

X – C.G.: X-coordinate of the wall's centroid in the global coordinate system (ft).

Y – C.G.: Y-coordinate of the wall's centroid in the global coordinate system (ft).

© 2026 ENERCALC, LLC

Wall Stiffness & Orientation

Wall Orientation, θ: Angle of the wall's local +y axis (i.e. strong axis) relative to the global +X-axis (0° = wall parallel to global X, 90° = wall parallel to global Y).

> ***NOTE:*** *The wall's local y axis is parallel to its length dimension; the local x axis is parallel to its thickness dimension.*

Height, H: Vertical height of the wall (ft).

Length, L: Horizontal length of the wall in plan (ft), parallel to the wall's local y axis.

Thickness, t: Thickness of the wall (in), parallel to the wall's local x axis.

Bending, Eb: Modulus of elasticity for bending stiffness (*10^6 psi). For example, a value of '1' means Eb = 1,000,000 psi.

Shear, Ev: Modulus of elasticity for shear stiffness (*10^6 psi). For example, a value of '0.4' means Eb = 400,000 psi.

> ***NOTE:*** *The shear modulus Ev (sometimes also represented as G) of cementitious materials is often estimated as Ev = 0.4 * Eb. This estimate is provided as a convenience in the module but may be overridden with a user-defined value.*
>
> *For concrete, ACI 318 does not formally define Ev, but it does present this as a reasonable approximation in Commentary R6.6.3.1*
>
> *For masonry, TMS 402 includes the same approximation in Table 4.2.2.*

Flexibility

Wall End Fixity (About local 'x'): End boundary condition for in-plane (strong axis) loading (i.e. loading along local +y / about local +x).

Wall End Fixity (About local 'y'): End boundary condition for out-of-plane (weak axis) loading (i.e. loading along local +x / about local +y).

Fix-Fix: Both ends restrained. The flexibility for a 1-kip unit load is calculated using the following equations:

$$Flexibility\ (Strong\ Axis) = \frac{H^3}{12E_b I_{xx}} + \frac{1.2H}{E_v A}$$

$$Flexibility\ (Weak\ Axis) = \frac{H^3}{12E_b I_{yy}} + \frac{1.2H}{E_v A}$$

© 2026 ENERCALC, LLC

Fix-Pin: One end fixed, one pinned. The flexibility for a 1-kip unit load is calculated using the following equations:

$$Flexibility\ (Strong\ Axis) = \frac{H^3}{3E_b I_{xx}} + \frac{1.2H}{E_v A}$$

$$Flexibility\ (Weak\ Axis) = \frac{H^3}{3E_b I_{yy}} + \frac{1.2H}{E_v A}$$

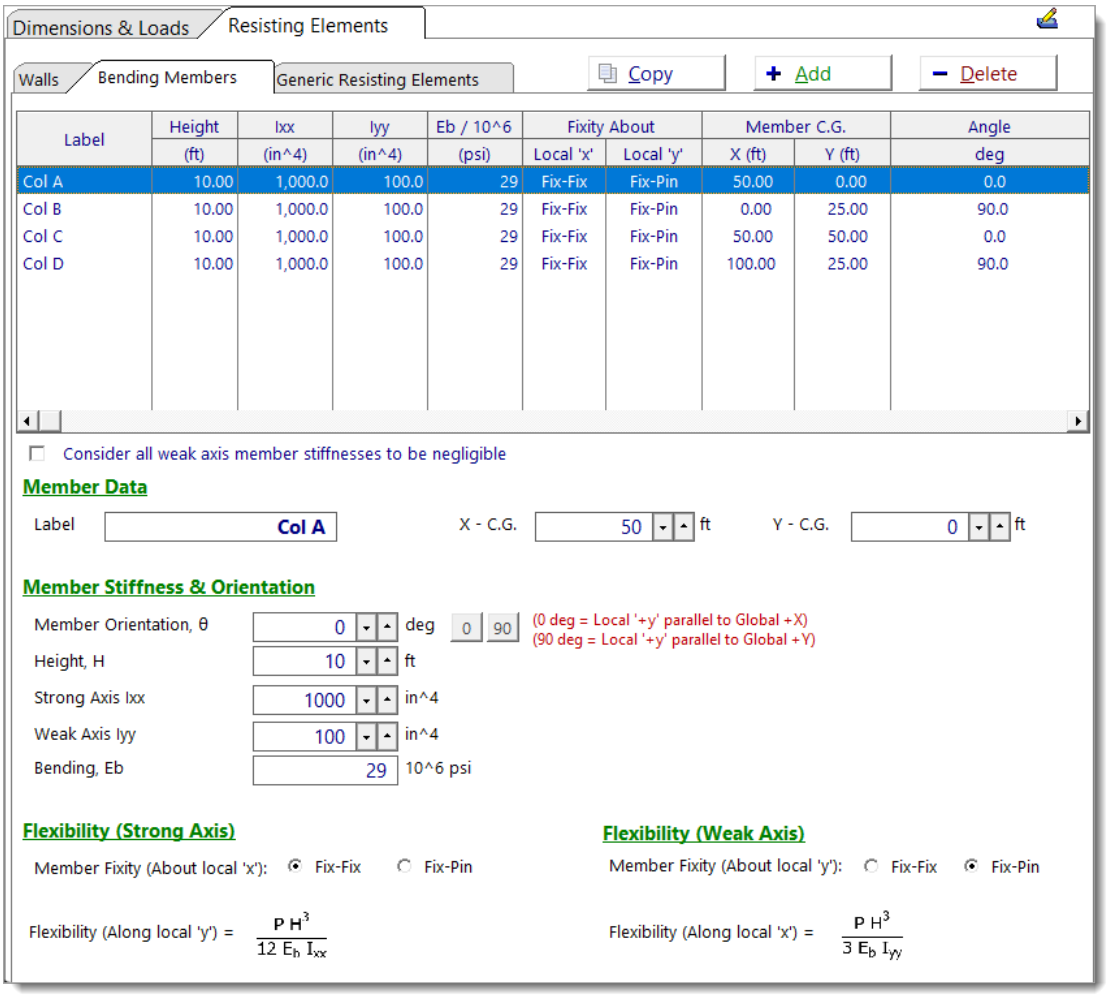
NOTE: *For more information on these flexibility equations, see Torsional Analysis of Rigid Diaphragm Theory - Displacement Behavior of Wall Elements* 657.

Consider all weak axis wall stiffnesses to be negligible: When enabled, the wall is modeled with infinite flexibility in the weak direction (out-of-plane). The flexibility and stiffness will display as "Neglect" on the *Resisting Element Data* tab.

Bending Members

Use the *Bending Members* tab to define a bending member (such as a column) as a resisting element. This will be a linear member whose stiffness is specified simply by its X and Y axis moments of inertia.

© 2026 ENERCALC, LLC



Member Data

Label: User-defined name for the bending member.

X – C.G.: X-coordinate of the bending member's centroid in the global coordinate system (ft).

Y – C.G.: Y-coordinate of the bending member's centroid in the global coordinate system (ft).

Member Stiffness & Orientation

Member Orientation, θ: Angle of the member's local +y axis (i.e. strong axis) relative to the global +X-axis (0° = local +y parallel to global +X, 90° = local +y parallel to global +Y).

Height, H: Vertical height of the member (ft).

Strong Axis Ixx: Moment of inertia about the members local +x axis (in^4).

© 2026 ENERCALC, LLC

Weal Axis Iyy: Moment of inertia about the members local +y axis (in^4).

Bending, Eb: Modulus of elasticity for bending stiffness ($*10^6$ psi). For example, a value of '29' means Eb = 29,000,000 psi.

Flexibility

Member Fixity (About local 'x'): End boundary condition for loading in the member's strong direction (i.e. along local +y / about local +x).

Member Fixity (About local 'y'): End boundary condition for loading in the member's weak direction (i.e. along local +x / about local +y).

Fix-Fix: Both ends restrained. The flexibility for a 1-kip unit load is calculated using the following equations:

$$Flexibility\ (Strong\ Axis) = \frac{H^3}{12E_bI_{xx}}$$

$$Flexibility\ (Weak\ Axis) = \frac{H^3}{12E_bI_{yy}}$$

Fix-Pin: One end fixed, one pinned. The flexibility for a 1-kip unit load is calculated using the following equations:

$$Flexibility\ (Strong\ Axis) = \frac{H^3}{3E_bI_{xx}}$$

$$Flexibility\ (Weak\ Axis) = \frac{H^3}{3E_bI_{yy}}$$

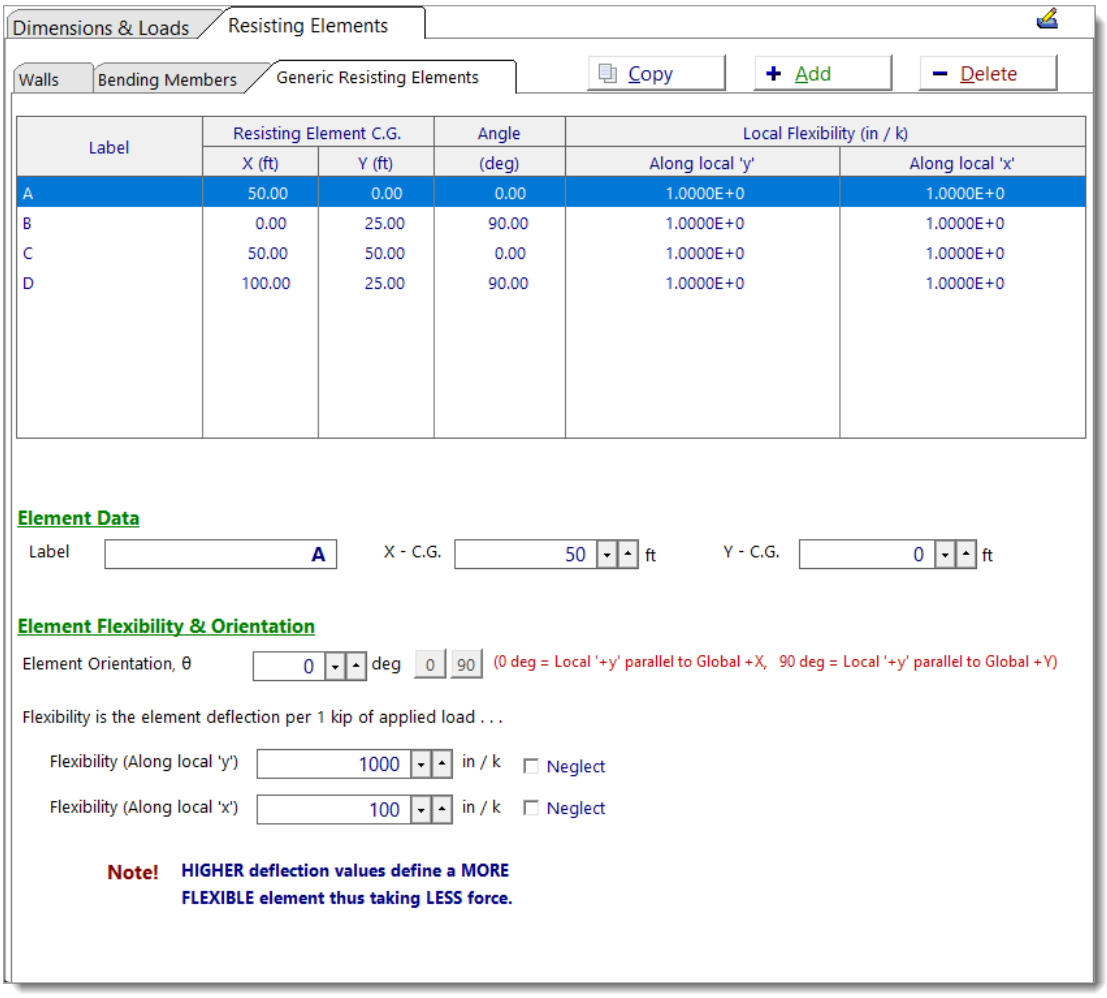
NOTE: *For bending members, flexural deformation dominates the overall behavior and shear deformation is neglected. For more information on these flexibility equations, see* *Torsional Analysis of Rigid Diaphragm Theory - Displacement Behavior of Wall Elements* 657.

Consider all weak axis member stiffnesses to be negligible: When enabled, the bending member is modeled with infinite flexibility in the weak direction. The flexibility and stiffness will display as “Neglect” on the *Resisting Element Data* tab.

Generic Resisting Elements

Use the *Generic Resisting Elements* tab to define a resisting element with user-specified stiffness. This option is intended for complex systems, such as braced or moment frames, where flexibility/stiffness has been determined from another analysis.

© 2026 ENERCALC, LLC



Element Data

Label: User-defined name for the generic resisting element.

X – C.G.: X-coordinate of the element's centroid in the global coordinate system (ft).

Y – C.G.: Y-coordinate of the element's centroid in the global coordinate system (ft).

Element Flexibility & Orientation

Element Orientation, θ: Angle of the element's local +y axis (i.e. strong axis) relative to the global +X-axis (0° = local +y parallel to global +X, 90° = local +y parallel to global +Y).

Flexibility (Along local 'y'): Flexibility of the element along its local +y axis, expressed as the deflection produced by a unit 1-kip load as determined from another analysis.

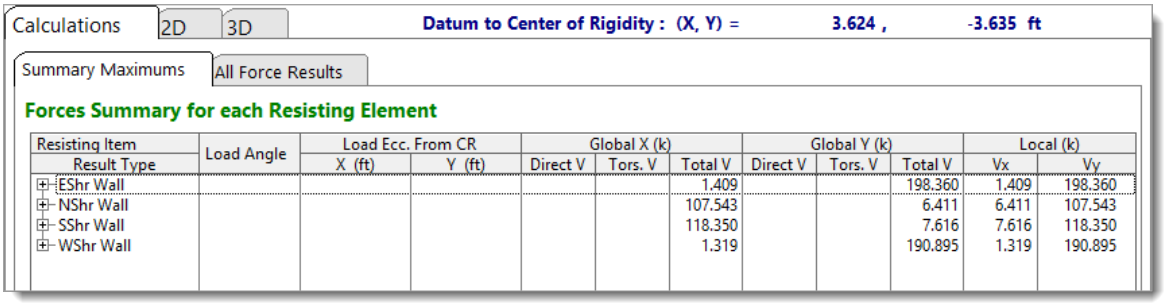
© 2026 ENERCALC, LLC

Flexibility (Along local 'x'): Flexibility of the element along its local +x axis, expressed as the deflection produced by a unit 1-kip load as determined from another analysis.

Neglect: When enabled, the element is modeled with infinite flexibility in the associated direction. The respective flexibility and stiffness will display as "Neglect" on the *Resisting Element Data* tab.

Summary Maximums

The *Summary Maximums* tab lists the peak global X and Y forces for each resisting element, along with the maximum shear forces in the element's local major (Vy) and minor (Vx) directions.



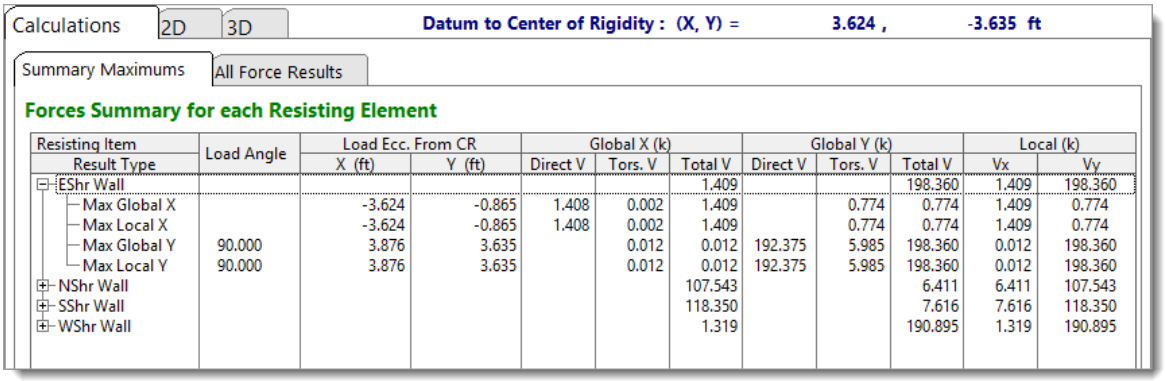
Calculations | 2D | 3D — Datum to Center of Rigidity : (X, Y) = 3.624 , -3.635 ft

Summary Maximums | All Force Results

Forces Summary for each Resisting Element

Resisting Item / Result Type	Load Angle	Load Ecc. From CR X (ft)	Load Ecc. From CR Y (ft)	Global X (k) Direct V	Global X (k) Tors. V	Global X (k) Total V	Global Y (k) Direct V	Global Y (k) Tors. V	Global Y (k) Total V	Local (k) Vx	Local (k) Vy
⊞ EShr Wall						1.409			198.360	1.409	198.360
⊞ NShr Wall						107.543			6.411	6.411	107.543
⊞ SShr Wall						118.350			7.616	7.616	118.350
⊞ WShr Wall						1.319			190.895	1.319	190.895

Clicking the **[+]** icon next to an element name expands the entry to show additional details. Since the maximum global and local forces along each axis may occur under different loading conditions, the expanded view also reports the load direction and eccentricity at which each maximum value occurs.



Calculations | 2D | 3D — Datum to Center of Rigidity : (X, Y) = 3.624 , -3.635 ft

Summary Maximums | All Force Results

Forces Summary for each Resisting Element

Resisting Item / Result Type	Load Angle	Load Ecc. From CR X (ft)	Load Ecc. From CR Y (ft)	Global X (k) Direct V	Global X (k) Tors. V	Global X (k) Total V	Global Y (k) Direct V	Global Y (k) Tors. V	Global Y (k) Total V	Local (k) Vx	Local (k) Vy
⊟ EShr Wall						1.409			198.360	1.409	198.360
Max Global X		-3.624	-0.865	1.408	0.002	1.409		0.774	0.774	1.409	0.774
Max Local X		-3.624	-0.865	1.408	0.002	1.409		0.774	0.774	1.409	0.774
Max Global Y	90.000	3.876	3.635		0.012	0.012	192.375	5.985	198.360	0.012	198.360
Max Local Y	90.000	3.876	3.635		0.012	0.012	192.375	5.985	198.360	0.012	198.360
⊞ NShr Wall						107.543			6.411	6.411	107.543
⊞ SShr Wall						118.350			7.616	7.616	118.350
⊞ WShr Wall						1.319			190.895	1.319	190.895

All Force Results

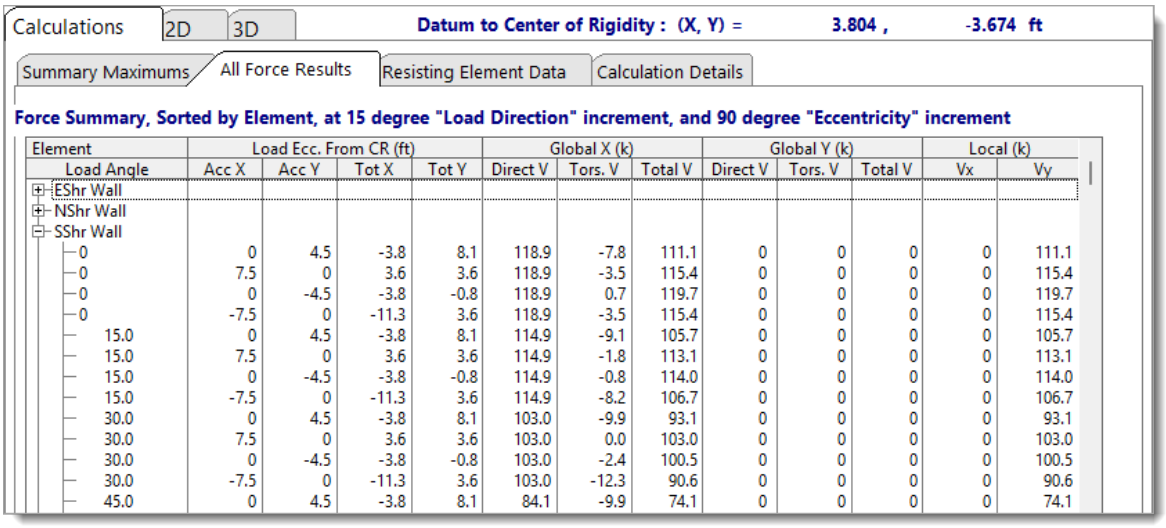
The *All Force Results* tab displays the full set of calculated forces for each resisting element. Results are shown in a tree structure, and clicking the **[+]** icon next to an element name expands its result set.

© 2026 ENERCALC, LLC

For example, in the image below, expanding the wall labeled "SShr Wall" reveals multiple lines labeled 0 deg, 15 deg, 30 deg, etc. Each line corresponds to a lateral load applied at a specific load angle as described under Load Direction 639.

Within each load-angle group, the accidental eccentricity values (*Acc X* and *Acc Y*) change to reflect different load application points along the accidental eccentricity ellipse as described under Eccentricity Increment 640.

A note at the top of the table indicates that the analysis uses 90-degree increments for the Eccentricity Increment, meaning there will be 4 entries for load application point (360 degrees / 90 degrees = 4). Additionally, the load direction varies in 15-degree increments, so scrolling further reveals results for other applied load angles.



Calculations | 2D | 3D — Datum to Center of Rigidity : (X, Y) = 3.804 , -3.674 ft

Summary Maximums | All Force Results | Resisting Element Data | Calculation Details

Force Summary, Sorted by Element, at 15 degree "Load Direction" increment, and 90 degree "Eccentricity" increment

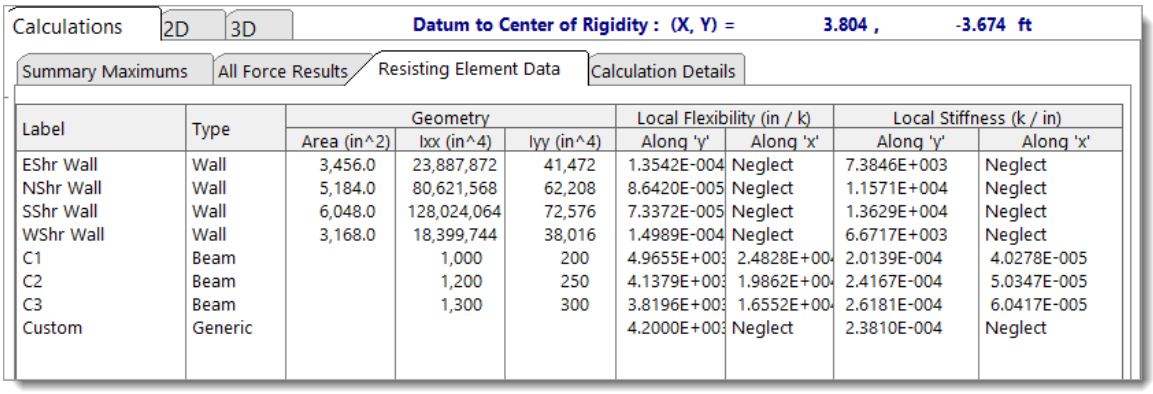
Element	Load Ecc. From CR (ft)				Global X (k)			Global Y (k)			Local (k)	
Load Angle	Acc X	Acc Y	Tot X	Tot Y	Direct V	Tors. V	Total V	Direct V	Tors. V	Total V	Vx	Vy
⊞ EShr Wall												
⊞ NShr Wall												
⊟ SShr Wall												
0	0	4.5	-3.8	8.1	118.9	-7.8	111.1	0	0	0	0	111.1
0	7.5	0	3.6	3.6	118.9	-3.5	115.4	0	0	0	0	115.4
0	0	-4.5	-3.8	-0.8	118.9	0.7	119.7	0	0	0	0	119.7
0	-7.5	0	-11.3	3.6	118.9	-3.5	115.4	0	0	0	0	115.4
15.0	0	4.5	-3.8	8.1	114.9	-9.1	105.7	0	0	0	0	105.7
15.0	7.5	0	3.6	3.6	114.9	-1.8	113.1	0	0	0	0	113.1
15.0	0	-4.5	-3.8	-0.8	114.9	-0.8	114.0	0	0	0	0	114.0
15.0	-7.5	0	-11.3	3.6	114.9	-8.2	106.7	0	0	0	0	106.7
30.0	0	4.5	-3.8	8.1	103.0	-9.9	93.1	0	0	0	0	93.1
30.0	7.5	0	3.6	3.6	103.0	0.0	103.0	0	0	0	0	103.0
30.0	0	-4.5	-3.8	-0.8	103.0	-2.4	100.5	0	0	0	0	100.5
30.0	-7.5	0	-11.3	3.6	103.0	-12.3	90.6	0	0	0	0	90.6
45.0	0	4.5	-3.8	8.1	84.1	-9.9	74.1	0	0	0	0	74.1

The algebraic signs of the direct and torsional shear components are relative to the global Cartesian coordinate system: Global +X acts to the right on screen, and Global +Y acts upward. These values represent diaphragm forces applied at the top of each resisting element. To determine base reactions, invert the sign of each value.

Resisting Element Data

The *Resisting Element Data* tab consolidates all resisting elements, including Walls, Bending Members, and Generic Resisting Elements, into a single table. It displays the calculated cross-sectional properties (such as area and moments of inertia), along with the corresponding local flexibility and stiffness values in both the X and Y directions. If stiffness along a local axis has been set to be neglected, the word “Neglect” will appear in the relevant flexibility and stiffness columns.

© 2026 ENERCALC, LLC


Calculations | 2D | 3D — Datum to Center of Rigidity : (X, Y) = 3.804 , -3.674 ft

Summary Maximums | All Force Results | Resisting Element Data | Calculation Details

Label	Type	Geometry			Local Flexibility (in / k)		Local Stiffness (k / in)	
		Area (in^2)	Ixx (in^4)	Iyy (in^4)	Along 'y'	Along 'x'	Along 'y'	Along 'x'
EShr Wall	Wall	3,456.0	23,887,872	41,472	1.3542E-004	Neglect	7.3846E+003	Neglect
NShr Wall	Wall	5,184.0	80,621,568	62,208	8.6420E-005	Neglect	1.1571E+004	Neglect
SShr Wall	Wall	6,048.0	128,024,064	72,576	7.3372E-005	Neglect	1.3629E+004	Neglect
WShr Wall	Wall	3,168.0	18,399,744	38,016	1.4989E-004	Neglect	6.6717E+003	Neglect
C1	Beam		1,000	200	4.9655E+00	2.4828E+00	2.0139E-004	4.0278E-005
C2	Beam		1,200	250	4.1379E+00	1.9862E+00	2.4167E-004	5.0347E-005
C3	Beam		1,300	300	3.8196E+00	1.6552E+00	2.6181E-004	6.0417E-005
Custom	Generic				4.2000E+00	Neglect	2.3810E-004	Neglect

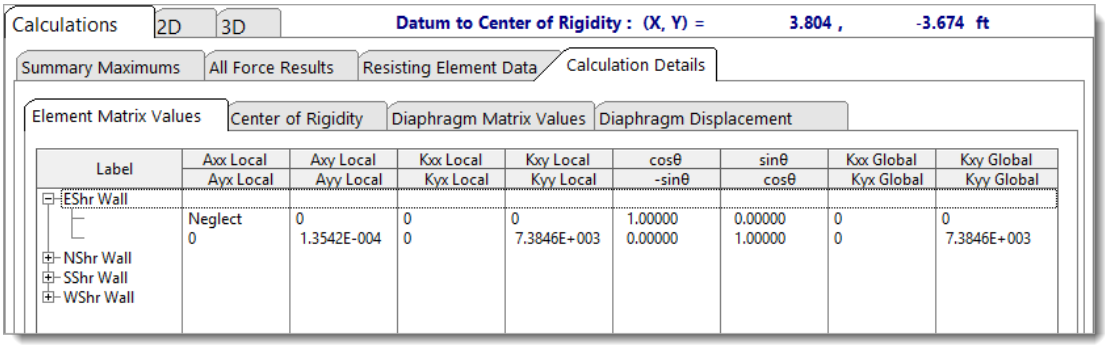
Calculation Details

The *Calculation Details* tab provides detailed calculations and intermediate results that are used to generate the outputs shown in the *All Force Results* and *Summary Maximums* tabs.

Element Matrix Values

The *Element Matrix Values* tab displays the local 2x2 flexibility and stiffness matrices, the transformation matrices, and the resulting global stiffness matrices for each resisting element.

For additional background on these values, refer to the Torsional Analysis of Rigid Diaphragm Theory 656 documentation, specifically the Resisting Element Local Stiffness 657 and Resisting Element Global Stiffness and Force Transformation 658 topics.


Calculations | 2D | 3D — Datum to Center of Rigidity : (X, Y) = 3.804 , -3.674 ft

Summary Maximums | All Force Results | Resisting Element Data | Calculation Details

Element Matrix Values | Center of Rigidity | Diaphragm Matrix Values | Diaphragm Displacement

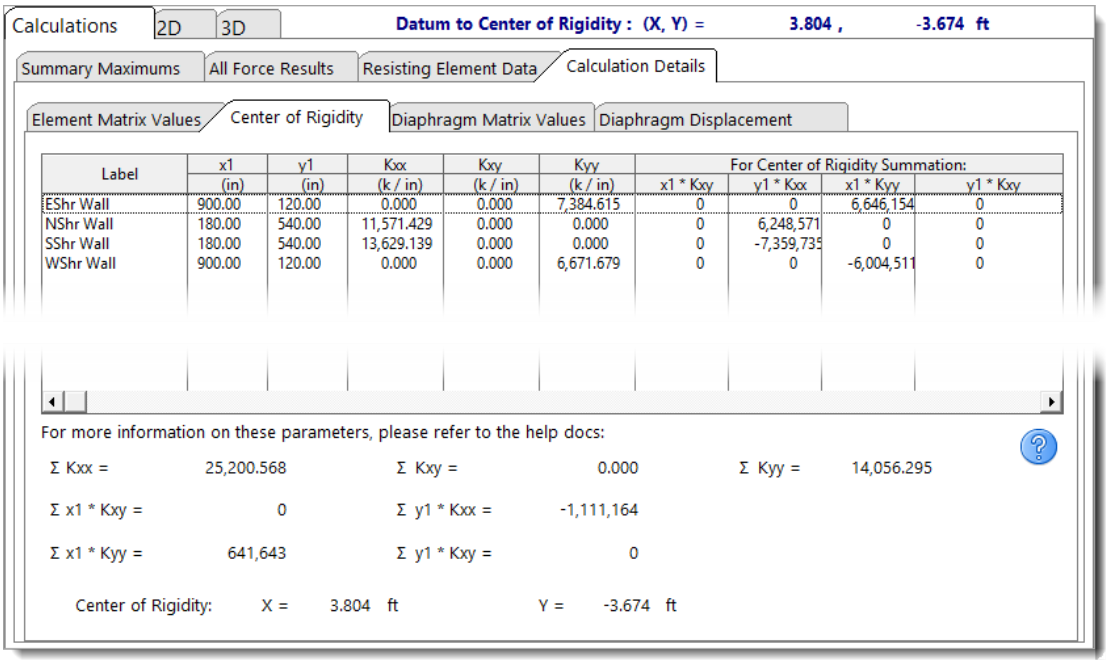
Label	Axx Local Ayx Local	Axy Local Ayy Local	Kxx Local Kyx Local	Kxy Local Kyy Local	cosθ -sinθ	sinθ cosθ	Kxx Global Kyx Global	Kxy Global Kyy Global
EShr Wall								
	Neglect 0	0 1.3542E-004	0 0	0 7.3846E+003	1.00000 0.00000	0.00000 1.00000	0 0	0 7.3846E+003
NShr Wall								
SShr Wall								
WShr Wall								

Center of Rigidity

The *Center of Rigidity* tab displays the individual components and summation terms used to calculate the center of rigidity, including each element's location and stiffness contributions.

© 2026 ENERCALC, LLC

For additional background on these values, refer to the Torsional Analysis of Rigid Diaphragm Theory 656 documentation, specifically the Center of Rigidity 663 and topic.

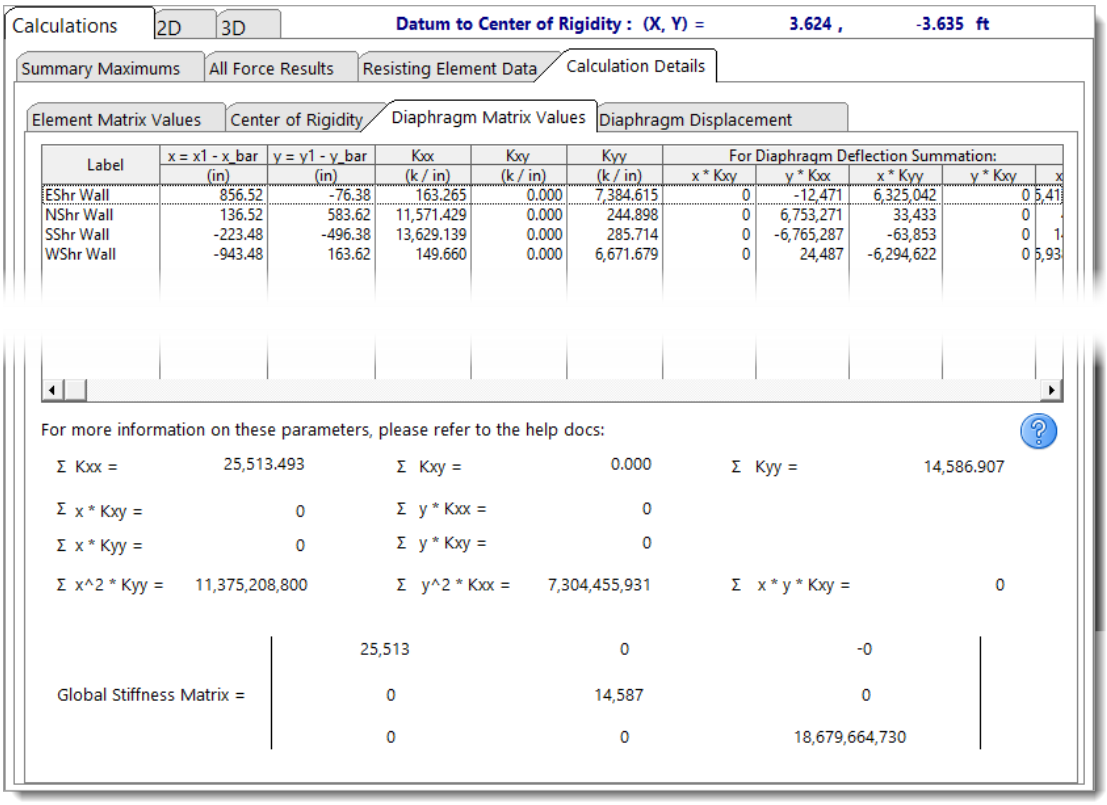


Diaphragm Matrix Values

The *Diaphragm Matrix Values* tab presents the individual components and summation terms used to calculate the global stiffness matrix for the rigid diaphragm.

For additional background on these values, refer to the Torsional Analysis of Rigid Diaphragm Theory 656 documentation, specifically the Rigid Diaphragm Global Stiffness Matrix 659 and topic.

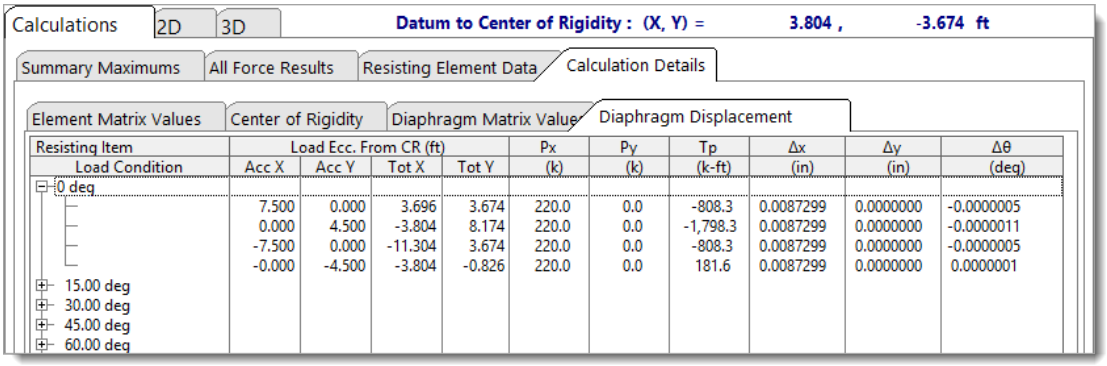
© 2026 ENERCALC, LLC



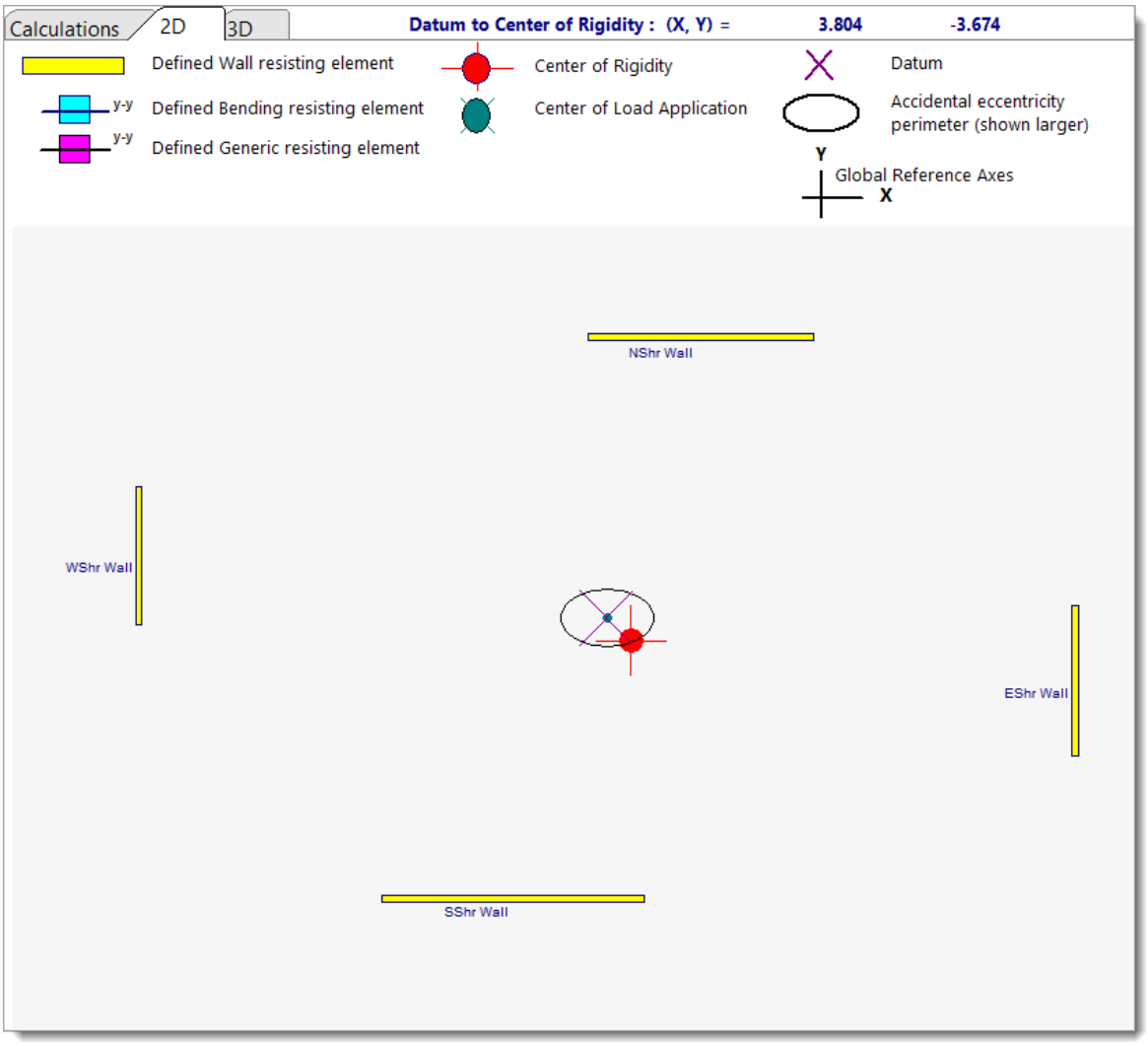
Diaphragm Displacement

The *Diaphragm Displacement* tab reports the load eccentricities relative to the center of rigidity, the applied lateral forces and torsional moments for each load condition or load angle, and the resulting diaphragm displacements and rotations.

For additional background on these values, refer to theTorsional Analysis of Rigid Diaphragm Theory 656 documentation, specifically the Rigid Diaphragm Global Stiffness Matrix 659 and topic.

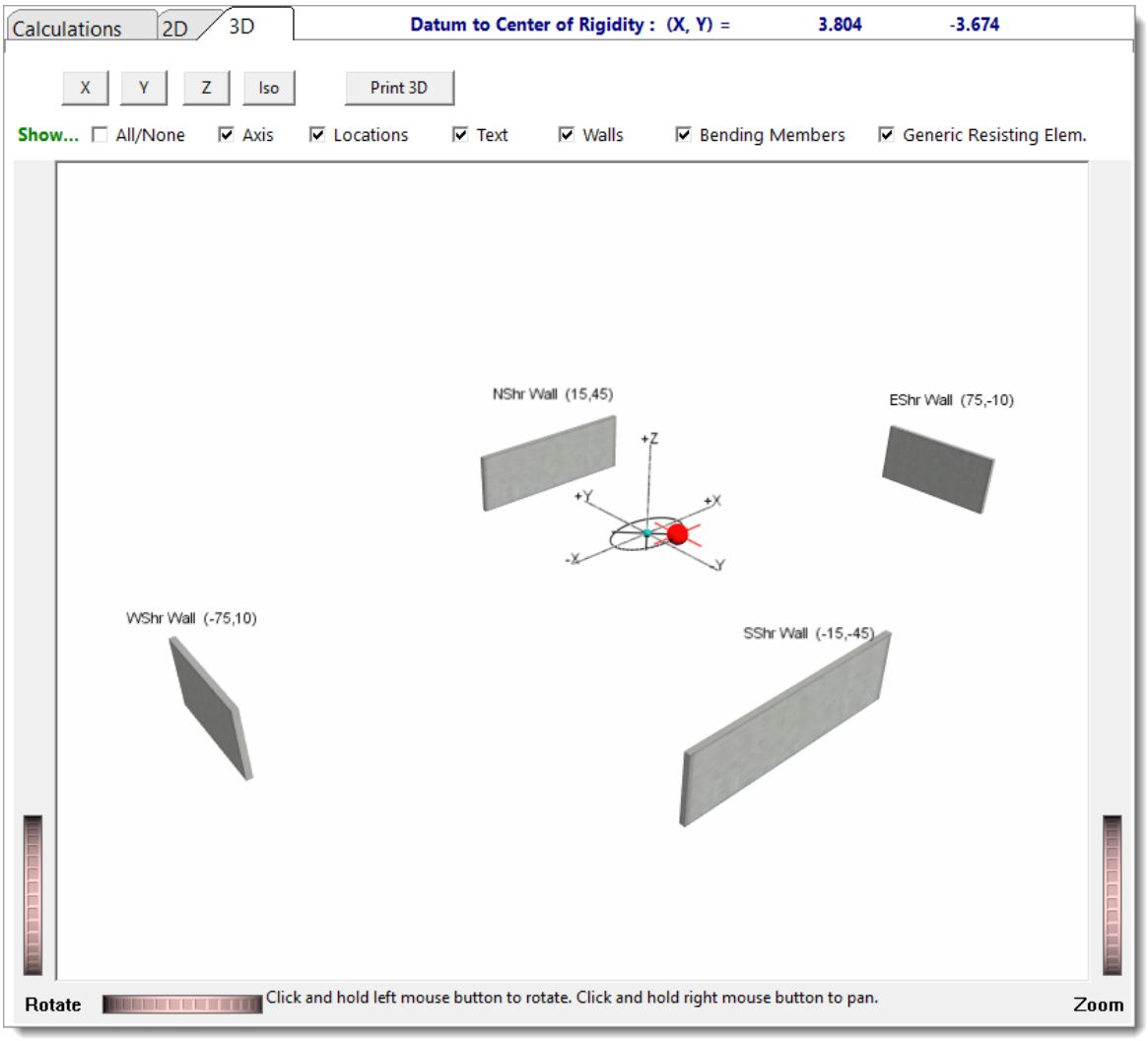


2D Sketch



3D Rendering

© 2026 ENERCALC, LLC



13.3.3.1 Torsional Analysis of Rigid Diaphragm Theory

Introduction

The distribution of lateral loads in structures with rigid diaphragms and orthogonal crosswalls or frame systems is a well-established topic in structural engineering literature. In such systems, the total lateral displacement of a resisting element results from the combined effects of flexural and shear deformations. This document presents a unified design method that captures these behaviors using energy-based principles and stiffness matrix computations.

This document also demonstrates a computational approach to distributing lateral forces in multi-resisting element systems, considering diaphragm rotation and translation, using stiffness matrices and energy principles to determine element forces and the overall center of rigidity.

© 2026 ENERCALC, LLC

For definitions of variables and symbols used throughout, refer to the Notations 667 section at the end of this document.

Resisting Element Local Stiffness

We begin by characterizing how each resisting element displaces under load. These displacements are used to build the element's flexibility matrix, which can then be inverted to produce the element's local stiffness matrix.

The lateral displacement of a resisting element results from both bending and shear deformation. The total displacement δ at the top of the element under a lateral load P is expressed as $\delta = \delta_f + \delta_v$, where δ_f is the displacement due to flexure, and δ_v is the displacement due to shear. These lateral displacements also depend on the end fixity conditions.

Displacement Behavior of Wall Elements

For both ends fixed (i.e., restrained against rotation):

Fixed-Fixed: $$\delta = \delta_f + \delta_v = \frac{Ph^3}{12E_bI} + \frac{\beta Ph}{E_vA} \qquad (1)$$

For one end fixed and the other end pinned:

Fixed-Pinned: $$\delta = \delta_f + \delta_v = \frac{Ph^3}{3E_bI} + \frac{\beta Ph}{E_vA} \qquad (2)$$

Where β, the shear correction factor, is given by:

$$\beta = \frac{A}{I^2}\int_A \left(\frac{Q}{b}\right)^2 dA \qquad (3)$$

For walls with rectangular cross sections, β is equal to 1.2.

NOTE: *The shear modulus E_v (sometimes also represented as G) of cementitious materials is often estimated as $E_v = 0.4E_b$. This estimate is provided as a convenience in the module but may be overridden with a user-defined value.*

For concrete, ACI 318 does not formally define E_v, but it does present this as a reasonable approximation in Commentary R6.6.3.1

For masonry, TMS 402 includes the same approximation in Table 4.2.2.

Displacement Behavior of Bending Elements

For bending members, flexural deformation dominates the overall behavior, and shear effects can often be neglected. In these cases, displacement simplifies to the flexural component only:

© 2026 ENERCALC, LLC

Fixed-Fixed: $$\delta = \delta_f = \frac{Ph^3}{12E_bI} \quad (4)$$

Fixed-Pinned: $$\delta = \delta_f = \frac{Ph^3}{3E_bI} \quad (5)$$

Flexibility and Stiffness Matrices

Once the displacement δ is known for a unit load along each of the elements primary local axes, the flexibility matrix $[a]_{local}$ is formed:

$$[a]_{local} = \begin{bmatrix} \delta_{xx} & \delta_{xy} \\ \delta_{yx} & \delta_{yy} \end{bmatrix} = \begin{bmatrix} \delta_{xx} & 0 \\ 0 & \delta_{yy} \end{bmatrix} \quad (6)$$

Where:
δ_{xx} = displacement along local 'x' in response to a unit force along local 'x'
δ_{yy} = displacement along local 'y' in response to a unit force along local 'y'
$\delta_{xy} = \delta_{yx} = 0$ due to decoupled behavior in orthogonal directions

To compute the local stiffness matrix, $[k]_{local}$, we can simply invert $[a]_{local}$. Since $[a]_{local}$ is diagonal, this inversion is straightforward:

$$[k]_{local} = [a]^{-1}_{local} = \begin{bmatrix} 1/\delta_{xx} & 0 \\ 0 & 1/\delta_{yy} \end{bmatrix} \quad (7)$$

NOTE: *In some cases, users may choose to neglect stiffness along the element's weak axis, typically when its contribution to lateral resistance is insignificant or unconservative to include. When doing so, the corresponding direction in the local stiffness matrix is assigned a value of zero, effectively removing its influence from the overall system stiffness.*

Resisting Element Global Stiffness and Force Transformation

Resisting elements are often oriented at angles relative to the global axes. While each element's local stiffness matrix accurately captures its resistance along its own principal axes, these stiffness properties must be expressed in the global coordinate system before they can be combined with other elements contributing to the overall lateral stiffness.

To accomplish this, a transformation matrix is applied to rotate stiffness values from the element's local coordinate system into the global coordinate system. For a resisting element rotated by an angle θ (measured from the global x-axis to the element's local x-axis), the rotation matrix $[T]$ is defined as:

© 2026 ENERCALC, LLC

$$[T] = \begin{bmatrix} \cos\theta & \sin\theta \\ -\sin\theta & \cos\theta \end{bmatrix} \quad (8)$$

NOTE: *In this program, the rotation angle is typically defined as the angle from the global x-axis to the element's local y-axis, since the local y-axis is generally aligned with the element's strong axis. For example, a wall with a rotation of 0 degrees is parallel to the global x-axis, and a rotation of 90 degrees is parallel to the global y-axis. However, the rotation matrix used for stiffness transformation requires the angle to be measured from the global x-axis to the element's local x-axis. Therefore, when using the program-defined angle, it is necessary to subtract 90 degrees to obtain the correct orientation for the transformation matrix.*

The element global stiffness matrix $[K]$ is then obtained from the element local stiffness matrix $[k]_{local}$ through the following transformation:

$$\boxed{[K] = \begin{bmatrix} K_{xx} & K_{xy} \\ K_{yx} & K_{yy} \end{bmatrix} = [T]^T [k]_{local} [T]} \quad (9)$$

After the rigid diaphragm analysis is complete and elemental forces in the global system have been determined, the same transformation matrix can also used to convert these forces back into the element's local coordinate system to find the strong and weak axis applied shear. For example, given global forces $[F]$, the corresponding local forces $[f]_{local}$ can be determined as:

$$\boxed{[f]_{local} = [T][F]} \quad (10)$$

Where:

$$[f]_{local} = \begin{bmatrix} V_x \\ V_y \end{bmatrix}$$

$$[F] = \begin{bmatrix} F_x \\ F_y \end{bmatrix}$$

Rigid Diaphragm Global Stiffness Matrix

We now turn to assembling the rigid diaphragm global stiffness matrix, which is formed by integrating the element global stiffness matrices into a single global stiffness matrix for the entire system.

Consider the displacements of the single wall assembly shown in Figure 1:

© 2026 ENERCALC, LLC

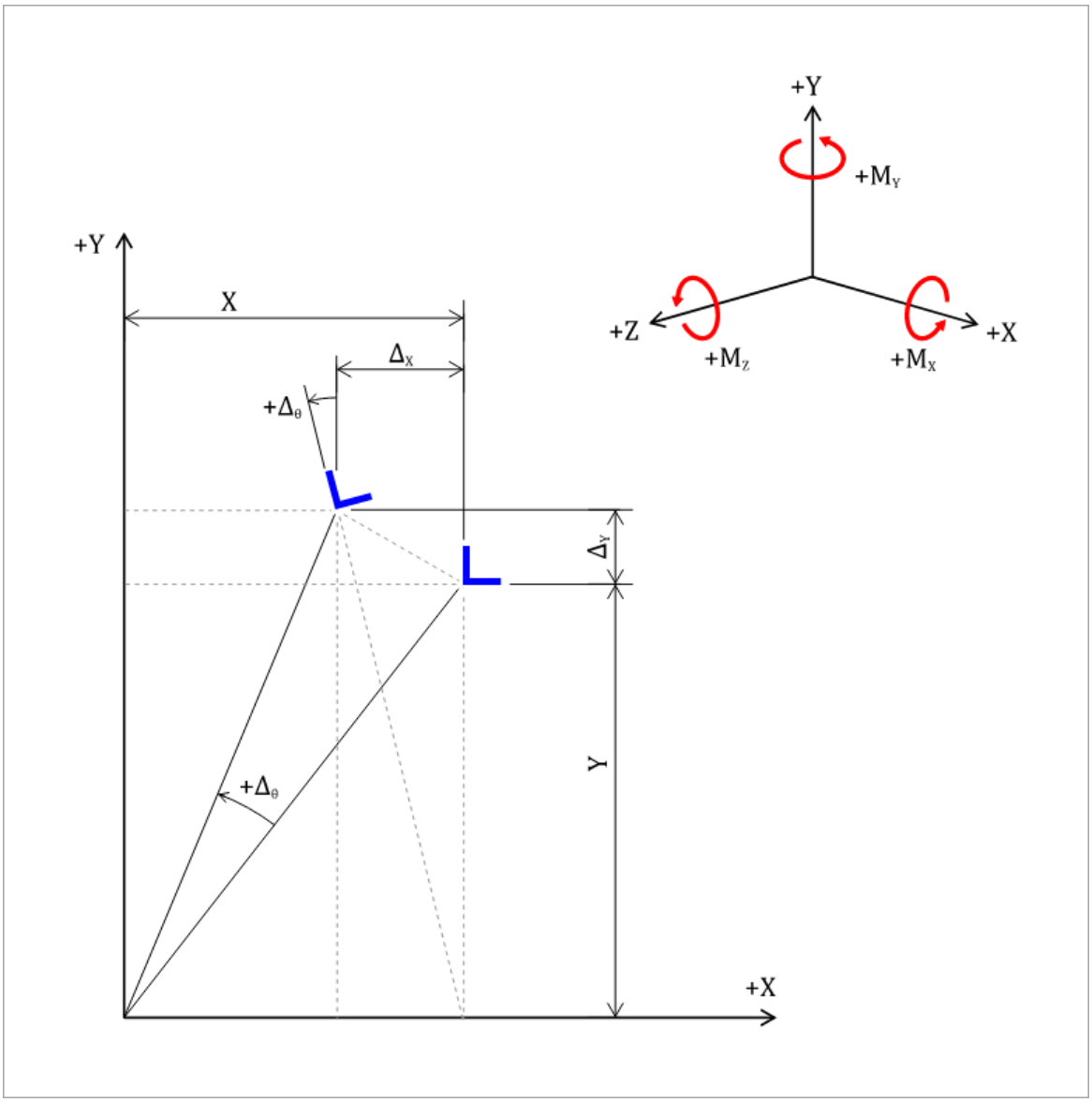


Figure 1: Element Rotation

The magnitude of the displacements are assumed to be very small. The assembly rotates through an angle Δ_θ (counter-clockwise positive) about the origin of coordinates X and Y, and translates Δ_x and Δ_y. The diaphragm is assumed to be very rigid compared to the walls. When the diaphragm rotates without translation, displacements occur which generate the following forces on the element:

$$F_x = K_{xx}\Delta_x + K_{xy}\Delta_y \quad (11)$$

$$F_y = K_{yx}\Delta_x + K_{yy}\Delta_y \quad (12)$$

$$M_z = J\Delta_\theta \quad (13)$$

© 2026 ENERCALC, LLC

$$M_p = xF_y - yF_x + J\Delta_\theta \qquad (14)$$

Where:

K_{xy} is the force in the X direction due to a unit displacement in the Y direction.
J is the rotational stiffness calculated as the moment about the Z axis due to a unit rotation of the element about the Z axis.
All values are plotted using the right-hand Cartesian coordinate axis system; thus a negative sign indicates a displacement in the negative direction or a clockwise moment. F_z, M_x, and M_y are zero for this type of system.

Using small-angle approximations, it can be shown that for the displacement in Figure 1:

$$\Delta_x = -y\Delta_\theta \quad , \quad \Delta_y = x\Delta_\theta$$

Thus, Equations 11, 12, and 14 can be revised as follows. It should also be noted that $K_{xy} = K_{yx}$. Therefore, all subsequent formulations will adopt K_{xy} for consistency.

$$F_x = K_{xx}(-y\Delta_\theta) + K_{xy}(x\Delta_\theta) \qquad (15)$$

$$F_y = K_{xy}(-y\Delta_\theta) + K_{yy}(x\Delta_\theta) \qquad (16)$$

$$M_p = \left(x^2K_{yy} + y^2K_{xx} - 2xyK_{xy} + J\right)\Delta_\theta \qquad (17)$$

When the diaphragm translates without rotation (see Figure 2), displacements occur which generate the following forces on the element:

$$F_x = K_{xx}\Delta_x + K_{xy}\Delta_y \qquad (18)$$

$$F_y = K_{xy}\Delta_x + K_{yy}\Delta_y \qquad (19)$$

© 2026 ENERCALC, LLC

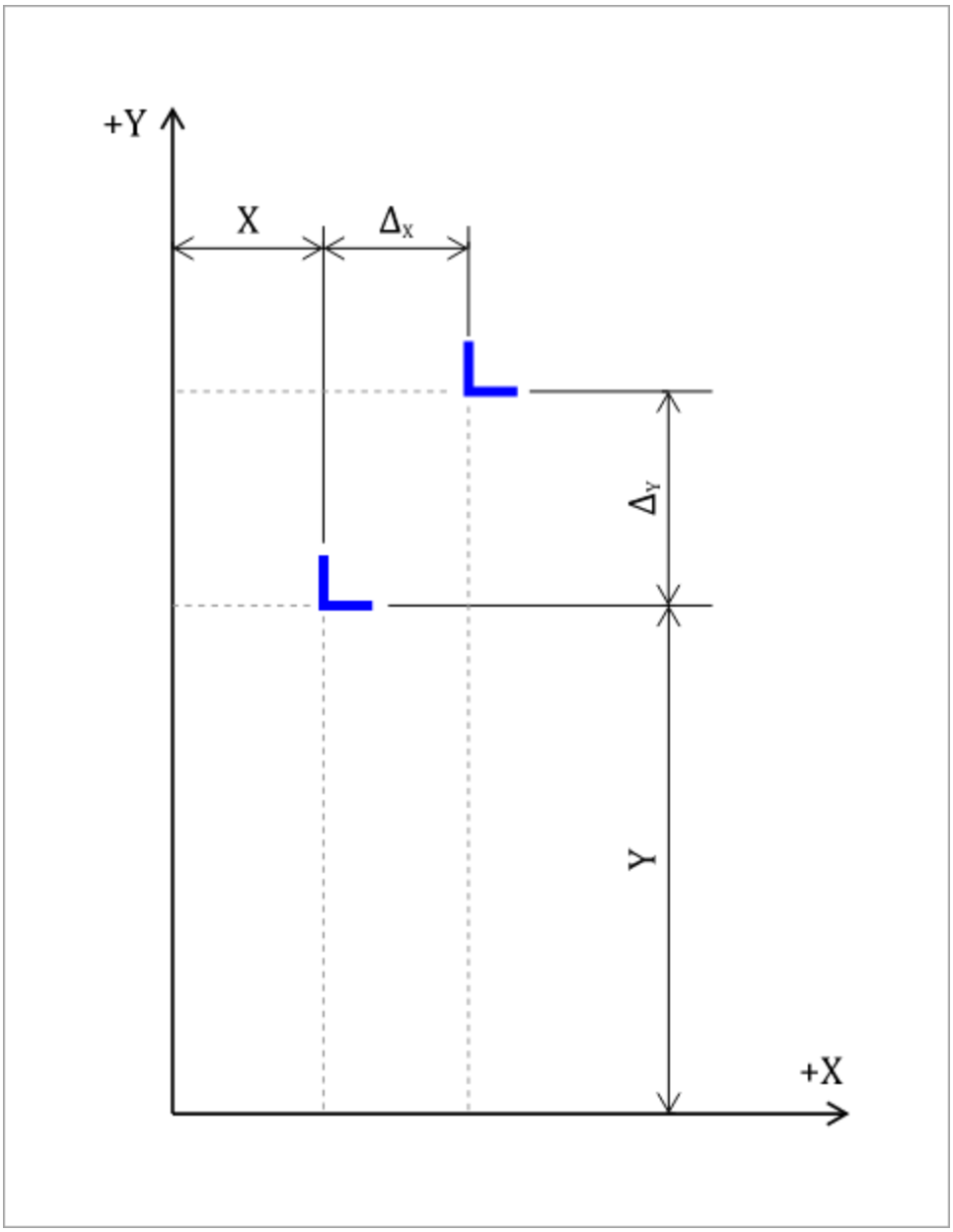


Figure 2: Element Translations

Equations 15 through 19 can then be combined to find the total element forces acting on the diaphragm when the diaphragm translates and rotates:

$$F_x = K_{xx}\Delta_x + K_{xy}\Delta_y + (xK_{xy} - yK_{xx})\Delta_\theta \quad (20)$$

$$F_y = K_{xy}\Delta_x + K_{yy}\Delta_y + (xK_{yy} - yK_{xy})\Delta_\theta \quad (21)$$

$$M_p = \left(x^2K_{yy} + y^2K_{xx} - 2xyK_{xy} + J\right)\Delta_\theta \quad (22)$$

These equations may now be written in matrix notation:

© 2026 ENERCALC, LLC

$$\begin{bmatrix} F_x \\ F_y \\ M_p \end{bmatrix} = \begin{bmatrix} K_{xx} & K_{xy} & (xK_{xy} - yK_{xx}) \\ K_{xy} & K_{yy} & (xK_{yy} - yK_{xy}) \\ 0 & 0 & (x^2K_{yy} + y^2K_{xx} - 2xyK_{xy} + J) \end{bmatrix} \begin{bmatrix} \Delta_x \\ \Delta_y \\ \Delta_\theta \end{bmatrix} \quad (23)$$

If the applied loads on the diaphragm are P_x and P_y in the X and Y directions, and a torsional moment T_p is also applied to the diaphragm in the positive, counter-clockwise direction:

$$\Sigma F_x = 0: \quad P_x = \Sigma F_x \quad (24)$$

$$\Sigma F_y = 0: \quad P_y = \Sigma F_y \quad (25)$$

$$\Sigma M_z = 0: \quad T_p = \Sigma M_p \quad (26)$$

Where:

ΣF_x is the sum of all resisting element forces acting in the global X direction.
ΣF_y is the sum of all resisting element forces acting in the global Y direction.
ΣM_p is the sum of the torsional moments from all resisting element forces acting about the diaphragm center of rigidity.

Substituting the values from Equations 24, 25, and 26 into Equations 20, 21, and 22 yields:

$$P_x = \Delta_x \Sigma K_{xx} + \Delta_y \Sigma K_{xy} + \Delta_\theta (\Sigma x K_{xy} - \Sigma y K_{xx}) \quad (27)$$

$$P_y = \Delta_x \Sigma K_{xy} + \Delta_y \Sigma K_{yy} + \Delta_\theta (\Sigma x K_{yy} - \Sigma y K_{xy}) \quad (28)$$

$$T_p = \Delta_\theta (\Sigma x^2 K_{yy} + \Sigma y^2 K_{xx} - 2\Sigma xy K_{xy} + \Sigma J) \quad (29)$$

Again, these equations may now be written in matrix notation to describe the diaphragm's global force-displacement behavior:

$$\begin{bmatrix} P_x \\ P_y \\ T_p \end{bmatrix} = \begin{bmatrix} \Sigma K_{xx} & \Sigma K_{xy} & (\Sigma x K_{xy} - \Sigma y K_{xx}) \\ \Sigma K_{xy} & \Sigma K_{yy} & (\Sigma x K_{yy} - \Sigma y K_{xy}) \\ 0 & 0 & (\Sigma x^2 K_{yy} + \Sigma y^2 K_{xx} - 2\Sigma xy K_{xy} + \Sigma J) \end{bmatrix} \begin{bmatrix} \Delta_x \\ \Delta_y \\ \Delta_\theta \end{bmatrix} \quad (30)$$

Center of Rigidity

To find the center of rigidity, the following relationships must be true:

$$\Sigma x K_{xy} - \Sigma y K_{xx} = 0 \quad (31)$$

$$\Sigma x K_{yy} - \Sigma y K_{xy} = 0 \quad (32)$$

© 2026 ENERCALC, LLC

Then, Equations 27, 28, and 29 become:

$$P_x = \Delta_x \Sigma K_{xx} + \Delta_y \Sigma K_{xy} \quad (33)$$

$$P_y = \Delta_x \Sigma K_{xy} + \Delta_y \Sigma K_{yy} \quad (34)$$

$$T_p = \Delta_\theta J_p \quad (35)$$

Where $J_p = \Sigma x^2 K_{yy} + \Sigma y^2 K_{xx} - 2\Sigma xy K_{xy} + \Sigma J$ (36)

The solution for the center of rigidity as defined in Equation 31 and 32 can be found if any set of trial coordinates are taken (x_1, y_1) with distances $\bar{x}$ and $\bar{y}$ to the center of rigidity. From Figure 3,

$$x = x_1 - \bar{x} \quad (37)$$

$$y = y_1 - \bar{y} \quad (38)$$

© 2026 ENERCALC, LLC

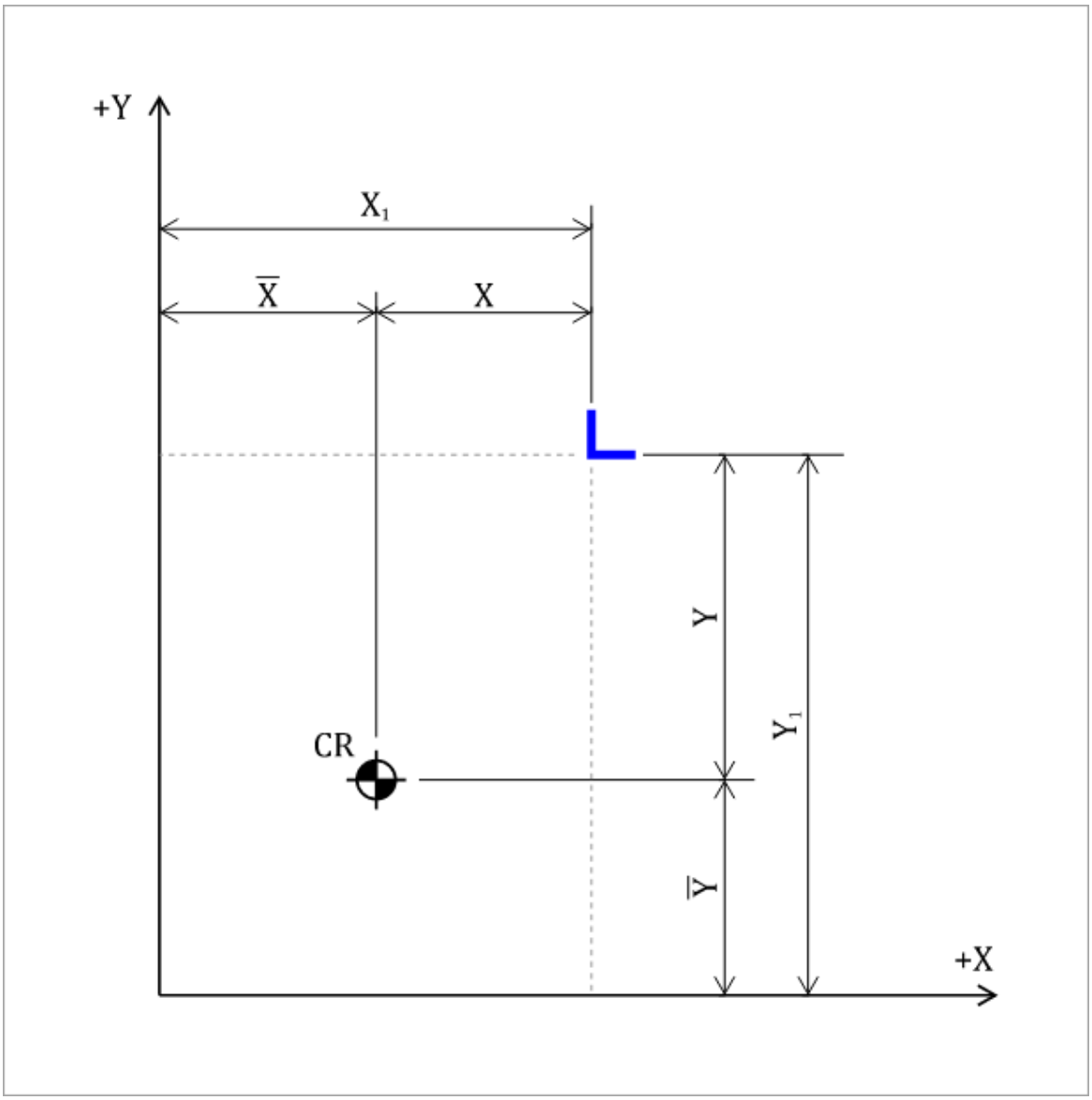


Figure 3: Center of Rigidity Trial Coordinates

Substituting Equations 37 and 38 into Equations 31 and 32:

$$\Sigma(x_1 - \bar{x})K_{xy} - \Sigma(y_1 - \bar{y})K_{xx} = 0 \quad (39)$$

$$\Sigma(x_1 - \bar{x})K_{yy} - \Sigma(y_1 - \bar{y})K_{xy} = 0 \quad (40)$$

By expanding and rearranging Equations 39 and 40, and noting that $\bar{x}$ and $\bar{y}$ (the coordinates of the center of rigidity) are constants, we obtain:

$$\bar{y}\Sigma K_{xx} - \bar{x}\Sigma K_{xy} = \Sigma y_1 K_{xx} - \Sigma x_1 K_{xy} \quad (41)$$

$$\bar{y}\Sigma K_{xy} - \bar{x}\Sigma K_{yy} = \Sigma y_1 K_{xy} - \Sigma x_1 K_{yy} \quad (42)$$

© 2026 ENERCALC, LLC

To solve this system of simultaneous equations for $\bar{x}$ and $\bar{y}$, several approaches are possible. One common method is substitution, where one equation is rearranged to isolate a variable and then substituted into the other.

Equation 42 can be rearranged to express $\bar{x}$ explicitly:

$$\bar{x} = \frac{\bar{y}\sum K_{xy} - \sum y_1 K_{xy} + \sum x_1 K_{yy}}{\sum K_{yy}} \qquad (43)$$

The resulting expression for $\bar{x}$ can then be substituted into Equation 41, allowing the system to be solved explicitly for $\bar{y}$, as follows:

$$\bar{y} = \frac{\sum y_1 K_{xx} - \sum x_1 K_{xy} + \frac{\sum x_1 K_{yy} \cdot \sum K_{xy} - \sum y_1 K_{xy} \cdot \sum K_{xy}}{\sum K_{yy}}}{\sum K_{xx} - \frac{(\sum K_{xy})^2}{\sum K_{yy}}} \qquad (44)$$

Final Solution

The solution for element global forces F_x and F_y and local shears V_x and V_y acting on any resisting element is then found as follows:

1. Compute the local and global stiffness matrices $[k]_{local}$ and $[K]$ for each element from Equations 7 and 9.
2. Locate the center of rigidity from Equations 43 and 44.
3. Assemble the rigid diaphragm global stiffness matrix and compute displacements $\Delta_x, \Delta_y, \Delta_\theta$ from Equation 30.
4. Solve for element forces in the global system from Equation 23.
5. Transform the element global forces to local shears V_x and V_y from Equation 10.

Orthogonal Lateral Force Resisting Systems

For orthogonal lateral force resisting systems, the problem simplifies significantly. $K_{xy} = 0$, and the preceding equations reduce to a standard set of equations which may be familiar from manual or spreadsheet-based rigid diaphragm analyses.

Equations 43 and 44 for the coordinates of the center of rigidity reduce to:

$$\bar{x} = \frac{\sum x_1 K_{yy}}{\sum K_{yy}} \qquad (45)$$

$$\bar{y} = \frac{\sum y_1 K_{xx}}{\sum K_{xx}} \qquad (46)$$

© 2026 ENERCALC, LLC

Similarly, Equation 30 for the diaphragm displacement simplifies to:

$$\Delta_x = \frac{P_x}{\Sigma K_{xx}} \quad (47)$$

$$\Delta_y = \frac{P_y}{\Sigma K_{yy}} \quad (48)$$

$$\Delta_\theta = \frac{T_p}{J_p} \quad (49)$$

Finally, Equation 23 describing individual element forces acting on the diaphragm simplifies further, and substituting the results from Equations 47, 48, and 49 into this expression yields the final form of the solution.

$$F_x = \Delta_x K_{xx} + \Delta_\theta(-yK_{xx}) = \frac{P_x K_{xx}}{\Sigma K_{xx}} - \frac{T_p(yK_{xx})}{J_p} \quad (50)$$

$$F_y = \Delta_y K_{yy} + \Delta_\theta(xK_{yy}) = \frac{P_y K_{yy}}{\Sigma K_{yy}} + \frac{T_p(xK_{yy})}{J_p} \quad (51)$$

Review

A design approach was presented which allows for the distribution of lateral loads in structures with rigid diaphragms and crosswalls and/or frame systems of any orientation. Required stiffness coefficients were found by inverting the flexibility matrix of the lateral force resisting system. The analysis presented herein for a one-story structure can be easily extended to multi-level structure where the center of gravity and center of rigidity are the resultants of the centers of all the levels above the level of analysis.

For a detailed worked example using this approach, please see our Verification Example.

Notations

A = Cross-sectional area
$[a]_{local}$ = Element local flexibility matrix
b = Width of wall element
E_b = Modulus of elasticity
E_v = Shear modulus
F = Force
F_x, F_y, F_z = Force in the global 'x', 'y', or 'z' direction
$[F]$ = Element global force matrix
$[f]_{local}$ = Element local force matrix
h = Element height
I = Moment of inertia

© 2026 ENERCALC, LLC

I_1, I_2 = Minimum and maximum moment of inertia
I_{xx}, I_{yy} = Moment of inertia about local 'x' or 'y' axis ($\int Ax^2 \cdot da$)
I_{xy} = Moment of inertia about x-y axis ($\int Axy \cdot da$)
J = Rotational stiffness
K = Stiffness
K_{ij} = Element stiffness along local 'i' in response to a unit displacement along local 'j'
$[K]$ = Element global stiffness matrix
$[k]_{local}$ = Element local stiffness matrix
M = Moment
M_p = Torsional moment acting on resisting element
M_x, M_y, M_z = Moment about the global 'x', 'y', or 'z' axis
P = Applied lateral load
P_x, P_y, P_z = Applied lateral load in the global 'x', 'y', or 'z' direction
Q = Statical moment of area above plane of desired shear flow
T_p = Diaphragm torsional moment
$[T]$ = Rotation matrix
X, Y, Z = Coordinate axes
x, y, z = Distance from resisting element to center of rigidity
x_1, y_1, z_1 = Distance from resisting element to origin
$\bar{x}, \bar{y}$ = Distance from the center of rigidity to origin

β = Shear correction factor for cross-sectional shape of the element
δ = Element total displacement
δ_f = Element displacement due to flexure
δ_v = Element displacement due to shear
δ_{ij} = Element displacement along local 'i' in response to a unit force along local 'j'
$\Delta_x, \Delta_y, \Delta_z$ = Displacement of the rigid diaphragm in the global 'x', 'y', or 'z' direction
Δ_θ = Angle of rotation of the rigid diaphragm about the center of rigidity
θ = Angle of rotation measured from the global x-axis to the element's local x-axis

13.3.4 General Section Property Calculator

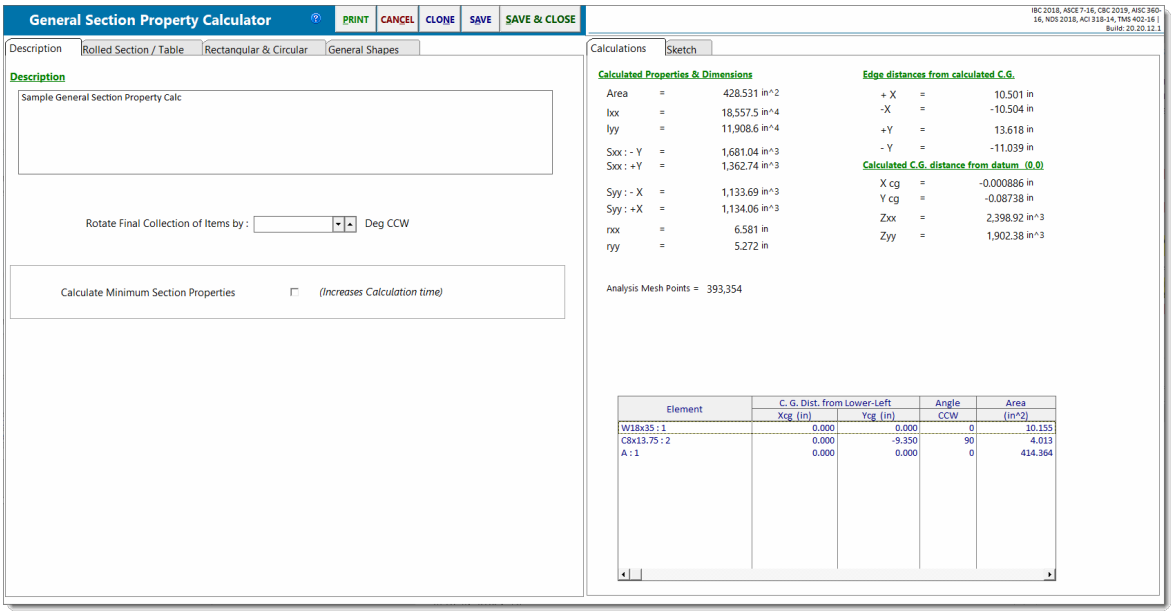
Overview

This module determines section properties for built-up sections with rectangles, hollow circles, solid circles, standard AISC steel sections and general multi-sided solid shapes.

AISC sections can be recalled from the database files and can be included in the built-up section. All sections from the 13th Edition AISC Steel Construction Manual are available, and can be reoriented as necessary.

© 2026 ENERCALC, LLC

The calculated section property values include: area, moments of inertia, center of gravity location, extreme fiber distances, section moduli, and radius of gyration.



Basic Usage

- Before starting data entry, be sure you have set up an X-Y coordinate system to consistently reference all component locations.
- For each rectangular shape, enter the height, width, and center of area measured from the datum.
- Hollow circular sections are entered by specifying the outside radius and thickness. Solid circular sections are entered by specifying the outside radius and setting the thickness to zero.
- For AISC sections, you can use the Xcg and Ycg input fields to locate the section's centroid position with respect to the datum. The module knows the centroid location of AISC members with respect to their own extreme fiber locations. However, you need to enter the location of the member's centroid in relation to the other members in the built-up section. Be careful, as this can be tricky when entering channels, angles, and tee sections that are rotated.
- A unique feature allows the user to specify that AISC sections can be rotated in one-degree increments, steel angle sections can also be mirrored about their Y axis, and the entire built-up section can be rotated to any desired angle.

© 2026 ENERCALC, LLC

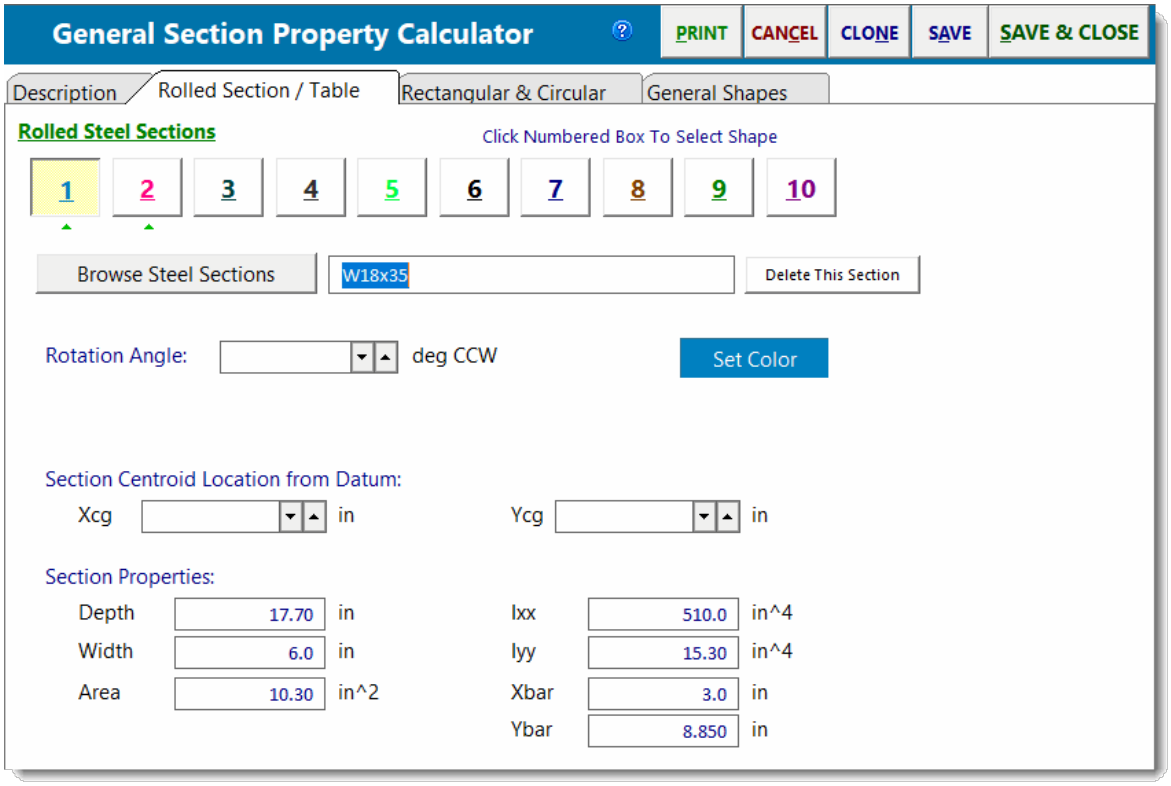
Assumptions & Limitations

The module operates on a simple calculation procedure:

- Calculate the moment of inertia of each shape,
- Calculate the neutral axis of the group of shapes, and
- Calculate the moment of inertia of the group using I + A*D^2 equations.

More complex analysis such as polar moment of inertia, plastic moduli, and buckling constants are beyond the scope of the module at this time but continuing development will add these items in future updates.

Rolled Section / Table



This tab enables you to specify up to 10 sections from the AISC Edition database to use in a built-up member.

The square buttons across the top of the tab are used to represent the component sections that comprise your built-up shape. When a section has been specified for a particular button, a small green upward facing triangle will be shown under the corresponding button. Click on any button to add a section or view and modify the section that has already been assigned to that button.

Note: It is important to understand that the numbered buttons on the various tabs DO NOT represent different built-up shapes. Instead, each instance of this module only creates ONE built-up shape, and the overall built-up shape consists of a composite of ALL

© 2026 ENERCALC, LLC

sections that currently exist on ANY buttons in the Rolled Section / Table tab, the Rectangular & Circular tab, and the General Shapes tab.

To insert an AISC section you can:

- Type in the section name and press [**Tab**]. The module will search the database and retrieve the information.
- Use the [**Browse Steel Sections**] button to display the steel database where you can navigate and select the desired section.

Rotation Angle: Counter-Clockwise

If you need to rotate a section, click one of the four angular rotation buttons.

Rotate Section 180 degrees about its own y-y Axis

This checkbox option will only be displayed for single-angle sections. It offers the ability to mirror a single-angle section if needed.

Xcg & Ycg

Enter the location of the section's centroidal axis measured from the datum (the origin of your assumed X-Y Cartesian coordinate system).

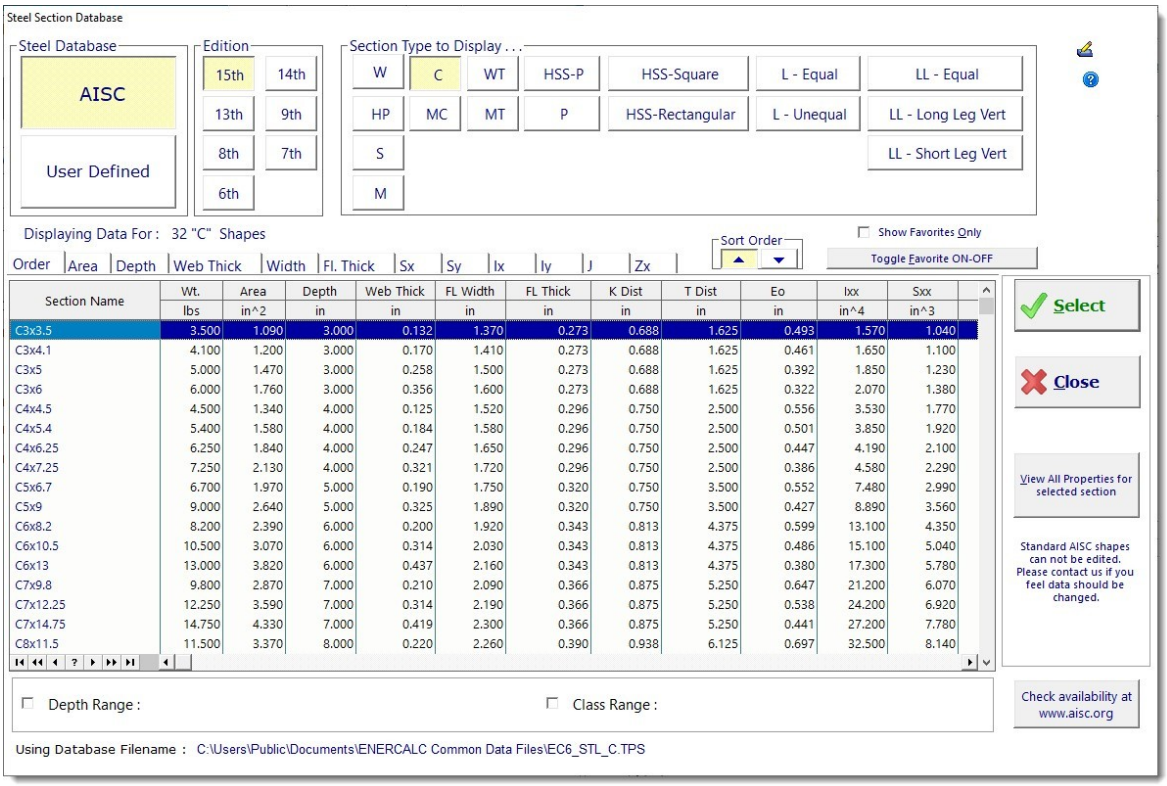
Section Properties

These values will be filled in after you make your choice from the AISC database. HOWEVER you can alter these values yourself. Of particular importance for unsymmetrical sections is entering the correct "Xbar" and "Ybar" location. This is the distance from the lower-left edge of the section, measured upwards and to the right, to the centroidal axis position of the section.

© 2026 ENERCALC, LLC

Steel Section Database

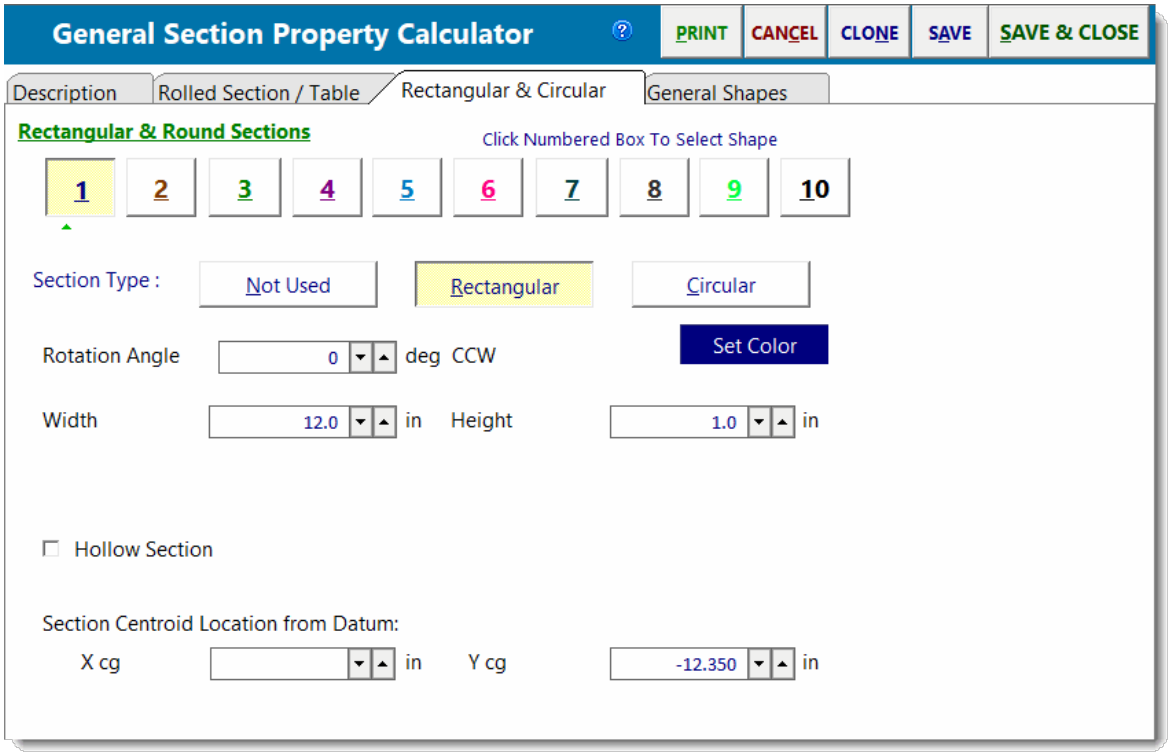
Click the [**Browse Steel Sections**] button to display the AISC database window:



Section Name	Wt.	Area	Depth	Web Thick	FL Width	FL Thick	K Dist	T Dist	Eo	Ixx	Sxx
	lbs	in^2	in	in	in	in	in	in	in	in^4	in^3
C3x3.5	3.500	1.090	3.000	0.132	1.370	0.273	0.688	1.625	0.493	1.570	1.040
C3x4.1	4.100	1.200	3.000	0.170	1.410	0.273	0.688	1.625	0.461	1.650	1.100
C3x5	5.000	1.470	3.000	0.258	1.500	0.273	0.688	1.625	0.392	1.850	1.230
C3x6	6.000	1.760	3.000	0.356	1.600	0.273	0.688	1.625	0.322	2.070	1.380
C4x4.5	4.500	1.340	4.000	0.125	1.520	0.296	0.750	2.500	0.556	3.530	1.770
C4x5.4	5.400	1.580	4.000	0.184	1.580	0.296	0.750	2.500	0.501	3.850	1.920
C4x6.25	6.250	1.840	4.000	0.247	1.650	0.296	0.750	2.500	0.447	4.190	2.100
C4x7.25	7.250	2.130	4.000	0.321	1.720	0.296	0.750	2.500	0.386	4.580	2.290
C5x6.7	6.700	1.970	5.000	0.190	1.750	0.320	0.750	3.500	0.552	7.480	2.990
C5x9	9.000	2.640	5.000	0.325	1.890	0.320	0.750	3.500	0.427	8.890	3.560
C6x8.2	8.200	2.390	6.000	0.200	1.920	0.343	0.813	4.375	0.599	13.100	4.350
C6x10.5	10.500	3.070	6.000	0.314	2.030	0.343	0.813	4.375	0.486	15.100	5.040
C6x13	13.000	3.820	6.000	0.437	2.160	0.343	0.813	4.375	0.380	17.300	5.780
C7x9.8	9.800	2.870	7.000	0.210	2.090	0.366	0.875	5.250	0.647	21.200	6.070
C7x12.25	12.250	3.590	7.000	0.314	2.190	0.366	0.875	5.250	0.538	24.200	6.920
C7x14.75	14.750	4.330	7.000	0.419	2.300	0.366	0.875	5.250	0.441	27.200	7.780
C8x11.5	11.500	3.370	8.000	0.220	2.260	0.390	0.938	6.125	0.697	32.500	8.140

© 2026 ENERCALC, LLC

Rectangular & Circular



This tab allows you to specify simple rectangular and circular shapes.

The square buttons across the top of the tab are used to represent the component sections that comprise your built-up shape. When a section has been specified for a particular button, a small green upward facing triangle will be shown under the corresponding button. Click on any button to add a section or view and modify the section that has already been assigned to that button.

Not Used / Rectangular / Circular

Select the shape you wish to use for this item.

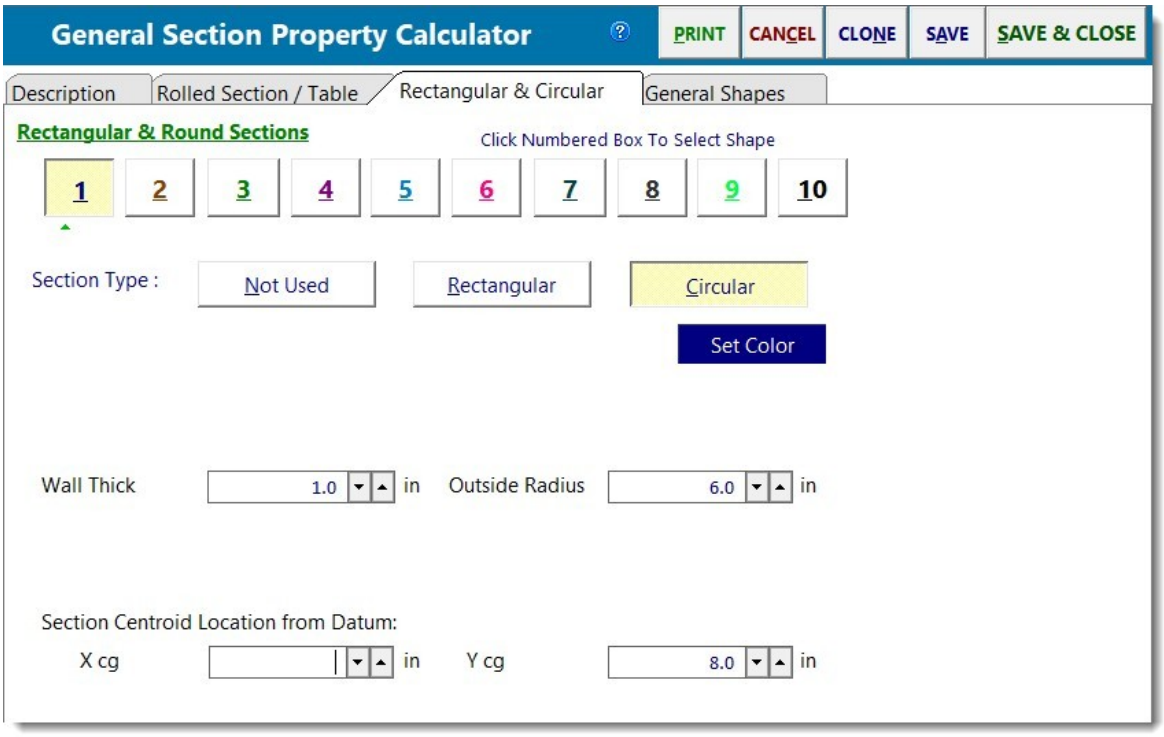
Rectangular Data Entry

When a rectangular shape is chosen the data entry consists of height and width.

© 2026 ENERCALC, LLC

Circular Data Entry

When a circular shape is selected the data entry consists of Outside Radius and Wall Thickness (not inside radius). To model a solid circular section, enter the appropriate Outside Radius and set the Wall Thickness to zero.

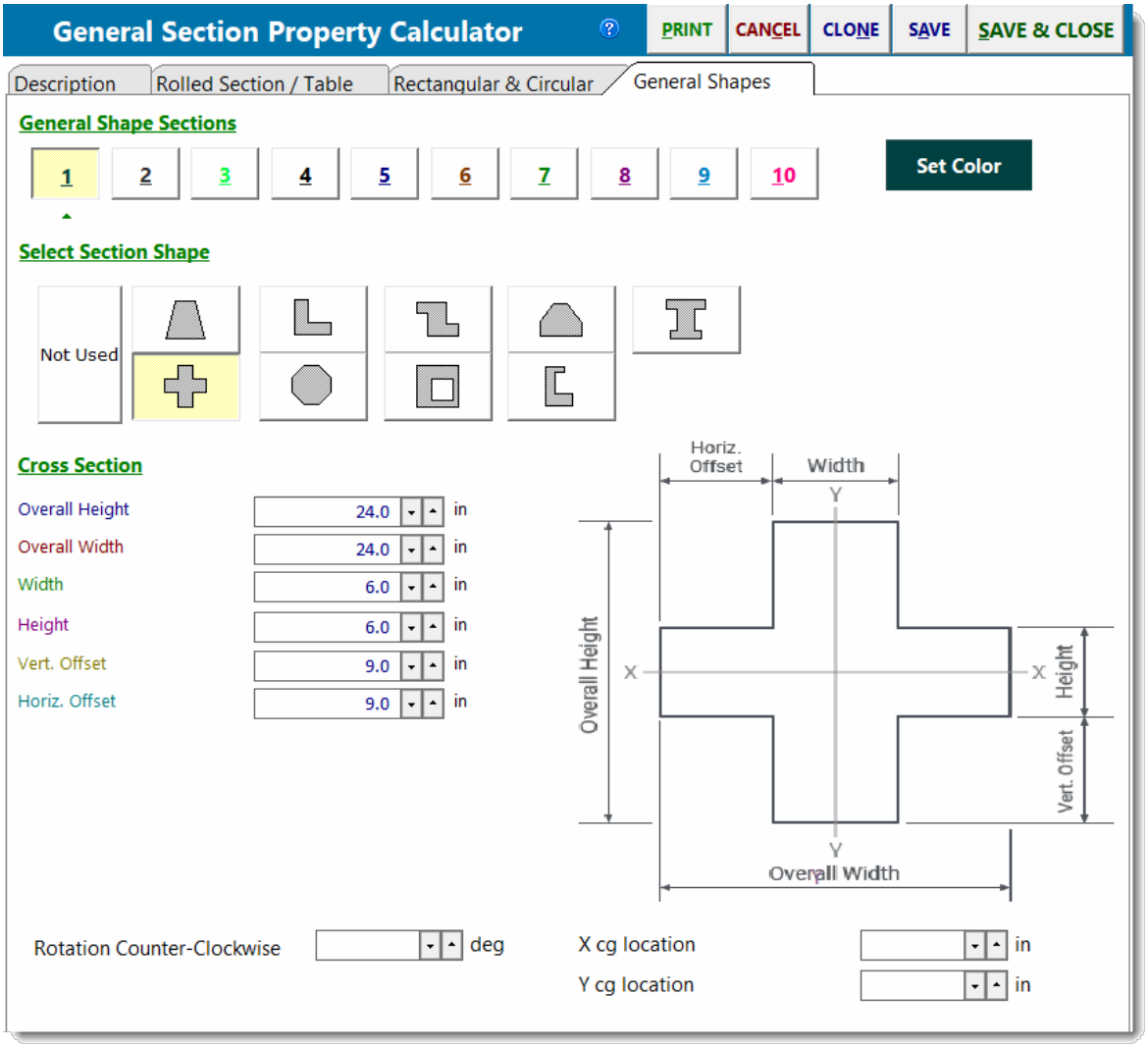


Xcg & Ycg

Enter the location of the section's centroidal axis measured from the datum (the origin of your assumed X-Y Cartesian coordinate system).

© 2026 ENERCALC, LLC

General Shapes



This tab allows you to select from a number of common polygonal shapes. With each selection the reference drawing and data entry prompts will change.

The square buttons across the top of the tab are used to represent the component sections that comprise your built-up shape. When a section has been specified for a particular button, a small green upward facing triangle will be shown under the corresponding button. Click on any button to add a section or view and modify the section that has already been assigned to that button.

Xcg & Ycg

Enter the location of the section's centroidal axis measured from the datum (the origin of your assumed X-Y Cartesian coordinate system).

© 2026 ENERCALC, LLC

Rotation Angle: Counter-Clockwise

For these shapes you can rotate the section in one-degree increments. Positive angles represent counter-clockwise rotation.

Calculations

Calculations | Sketch

Calculated Properties & Dimensions

Area	=	300.725 in^2
Ixx	=	10,642.3 in^4
Iyy	=	7,814.24 in^4
Sxx : - Y	=	831.84 in^3
Sxx : +Y	=	806.47 in^3
Syy : - X	=	651.19 in^3
Syy : +X	=	651.19 in^3
rxx	=	5.949 in
ryy	=	5.098 in

Edge distances from calculated C.G.

+ X	=	12.0 in
-X	=	-12.0 in
+Y	=	13.196 in
- Y	=	-12.794 in

Calculated C.G. distance from datum (0,0)

X cg	=	0.0 in
Y cg	=	0.7937 in
Zxx	=	1,395.96 in^3
Zyy	=	1,166.26 in^3

Analysis Mesh Points = 174,860

Element	C. G. Dist. from Lower-Left		Angle	Area
	Xcg (in)	Ycg (in)	CCW	(in^2)
W18x35 : 1	0.000	0.000	0	10.155
C8x13.75 : 2	0.000	-9.350	90	4.013
Circ : 1	0.000	8.000	0	34.558
Cross : 1	0.000	0.000	0	252.000

Detailed Properties Table

This table summarizes each of the component items you have added to the section. It reports their individual locations, properties and maximum distance from CG for each of the four edges.

Note: This table scrolls to the right. Just use the scroll bar along the bottom of the table.

© 2026 ENERCALC, LLC

Total Area

The total area of all defined shapes, including the area of any AISC sections which have been included in the built-up shape.

Inertia: Ixx & Iyy

The overall moment of inertia of the composite section is determined by applying the following equation to all the defined shapes:

$$Ixx = Iox + (A * d_y^2) \text{ and } Iyy = Ioy + (A * d_x^2)$$

where d = Distance from the shape's C.G. to the overall C.G. of the composite section, measured in the direction indicated by the subscript.

Section Modulus: Sxx and Syy

These values are the calculated section moduli of the composite section. The values are determined by dividing Ixx or Iyy by the extreme fiber distances above, below, right, and left of the center of gravity of the section.

Radius of Gyration

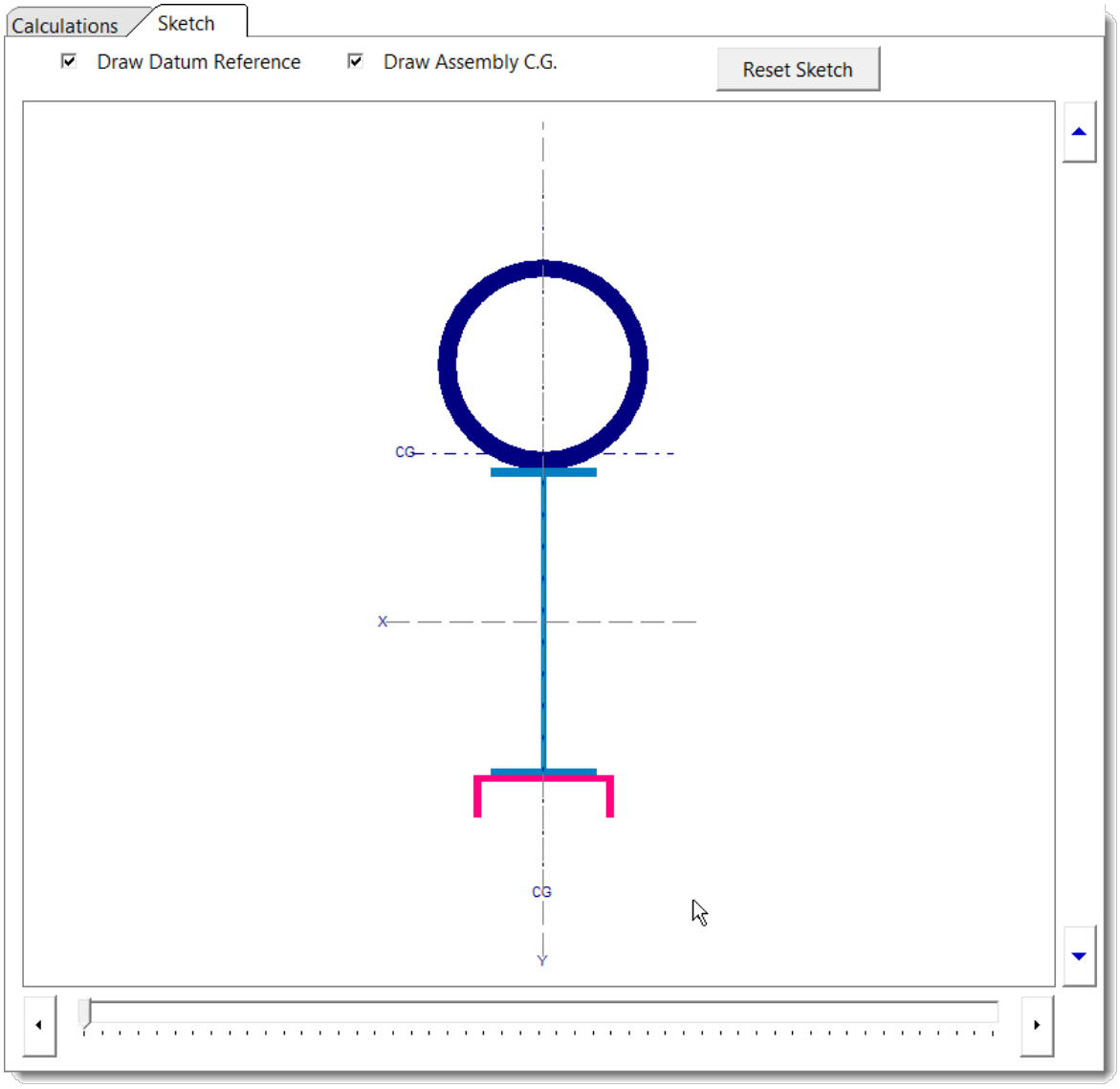
The radius of gyration of the composite section is determined using the typical equation: $rxx = (Ixx/A)^{½}$ and $ryy = (Iyy/A)^{½}$.

Max Distance from CG

For each of the sections in the built-up shape, these columns report the distance from the extreme fibers of that section to the C.G. of the composite section.

© 2026 ENERCALC, LLC

Sketch



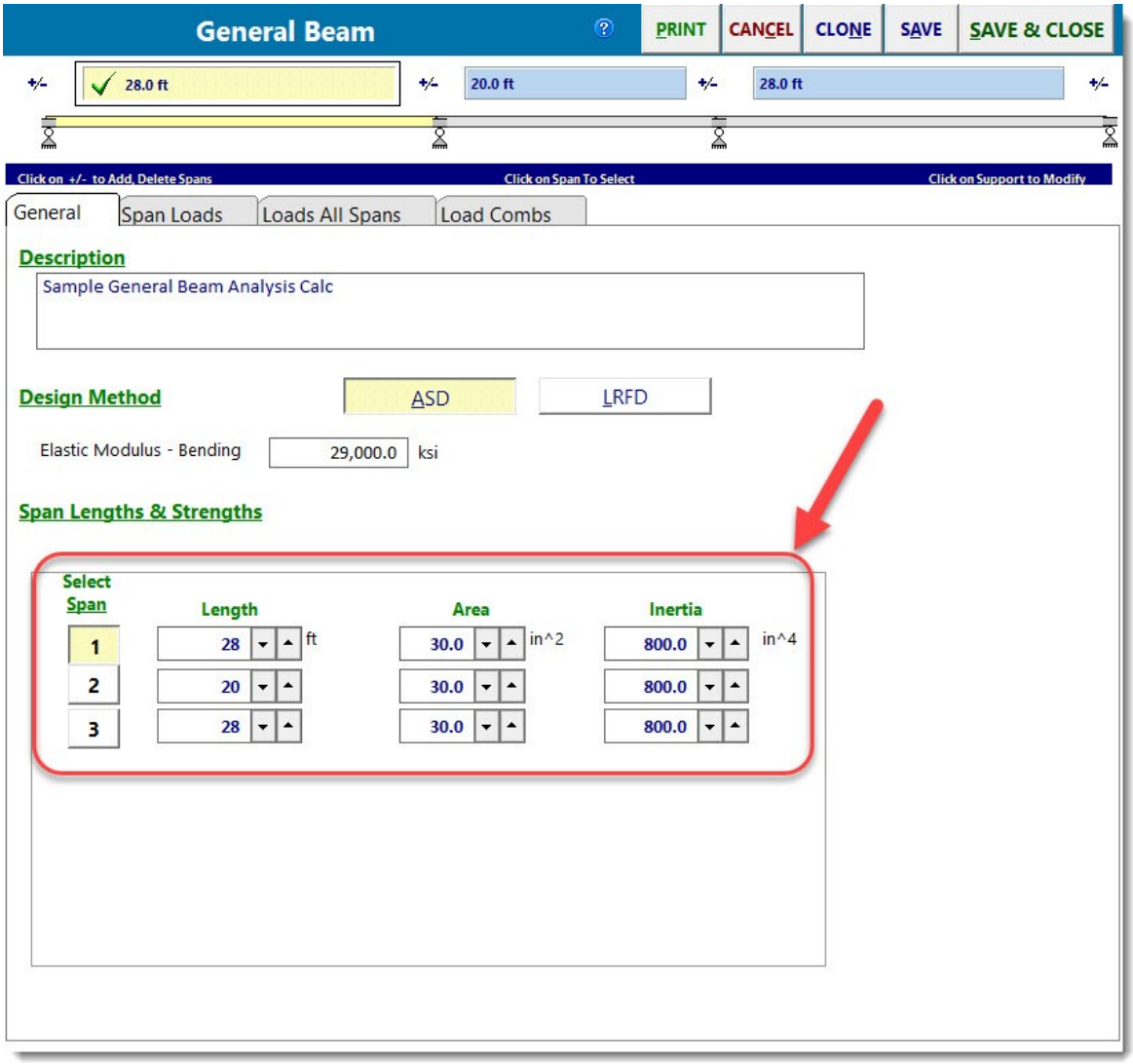
13.3.5 General Beam Analysis

The General Beam Analysis module offers beam analysis functionality but does not incorporate any design processes. In this way, it can be a useful tool for situations where only analysis results are desired, such as shear, moment, reactions, and deflections.

© 2026 ENERCALC, LLC

General tab:

The General Data tab allows you to set the span conditions, span lengths, and support conditions in much the same way that this information is provided in the other beam modules. Refer to the Beams 684 topic for additional explanation. In addition to these pieces of data, the General Data tab also provides input fields for the elastic modulus for bending, and the cross sectional area and moment of inertia of each span of the beam, as shown below:



Span Loads tab:

The Span Loads tab allows you to specify loads on one span at a time. The behavior of the tools on this tab is identical to the tools described for use in the other beam modules. Refer to the Beams topic for additional explanation.

© 2026 ENERCALC, LLC

Loads All Spans tab:

The Loads All Spans tab allows you to specify loads on all spans at the same time. The behavior of the tools on this tab is identical to the tools described for use in the other beam modules. Refer to the Beams topic for additional explanation.

Load Combinations tab:

The Load Combinations tab provides a view of the load combinations that will be analyzed. It also offers the ability to:

- Select a different set of load combinations,
- Modify the values used as load factors, and
- Turn certain combinations on and off.

Refer to the Beams topic for additional explanation.

The General Beam Analysis module offers output options that are analogous to the output options provided by the other beam modules, with the exception that no design results are provided.

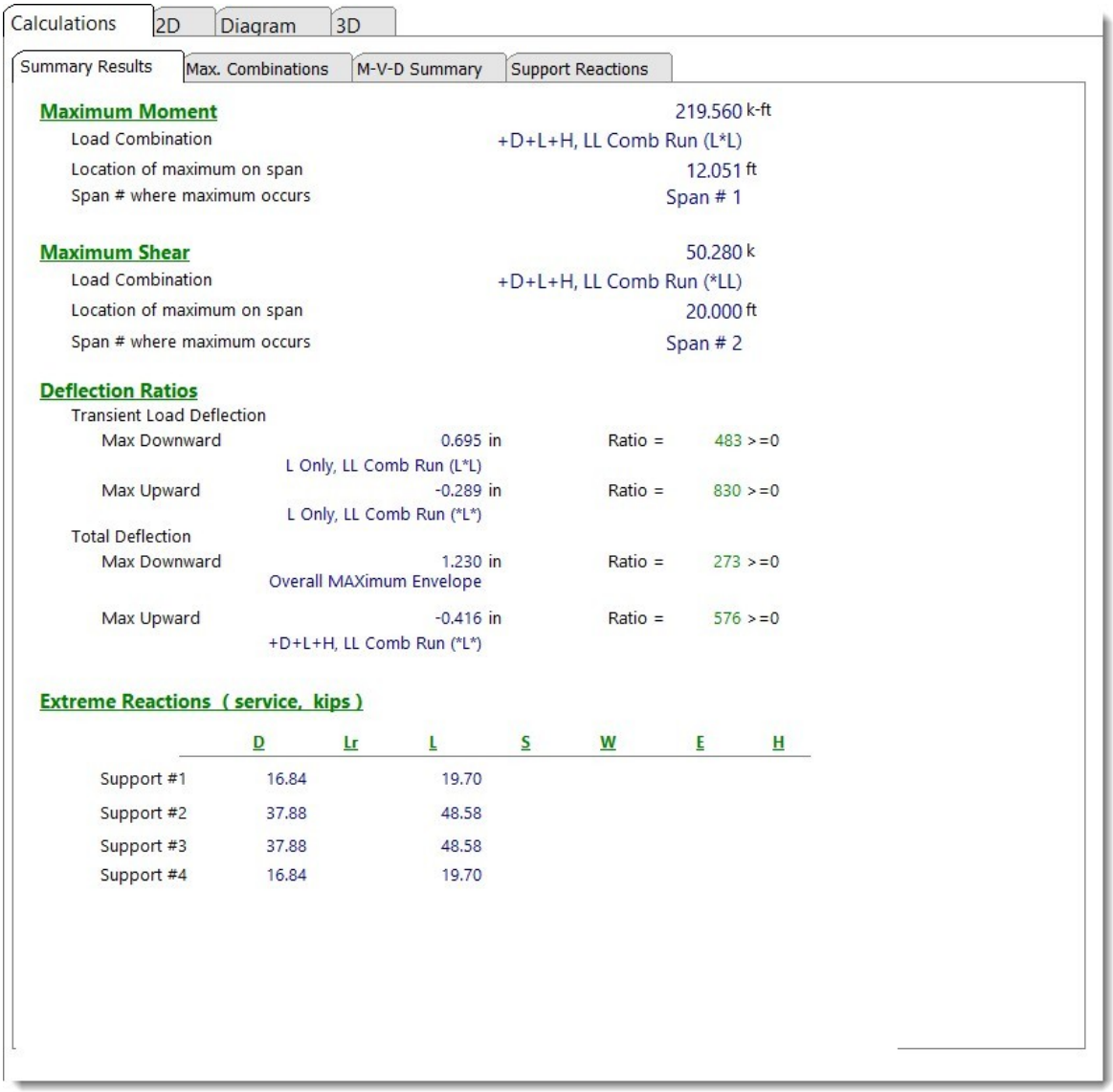
The right half of the screen is dedicated to the display of results. The horizontal strip of tabs in the upper right-hand corner of the display allows you to choose between Calculations, 2D Sketch, Diagram and 3D Rendering as explained below.

© 2026 ENERCALC, LLC

Calculations:

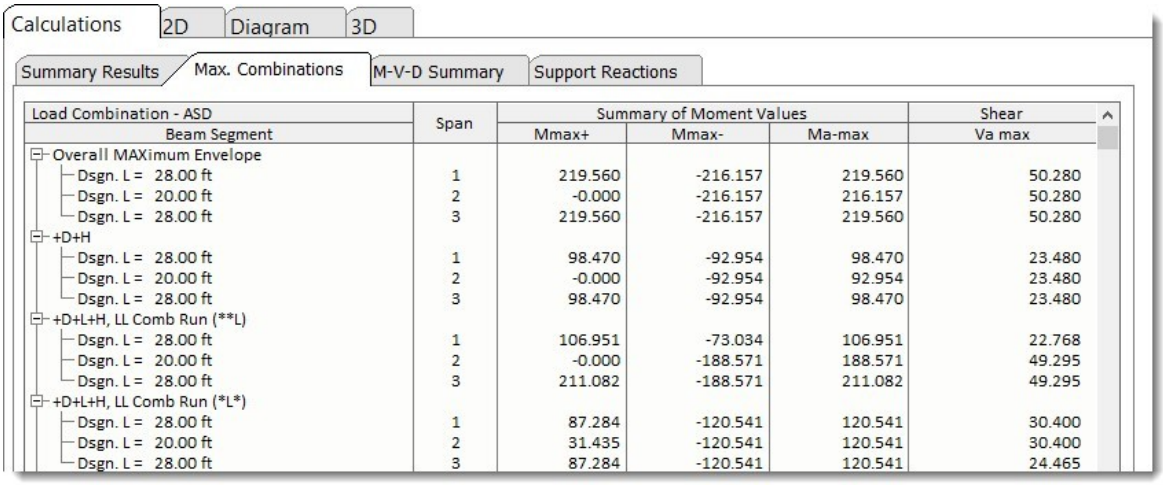
The Calculations tab offers four sub-tabs:

Summary Results: Displays extreme moments, maximum shear, extreme deflections and extreme reactions.



© 2026 ENERCALC, LLC

Maximum Combinations: Displays extreme moments and shears, on a span-by-span basis, for all load combinations.

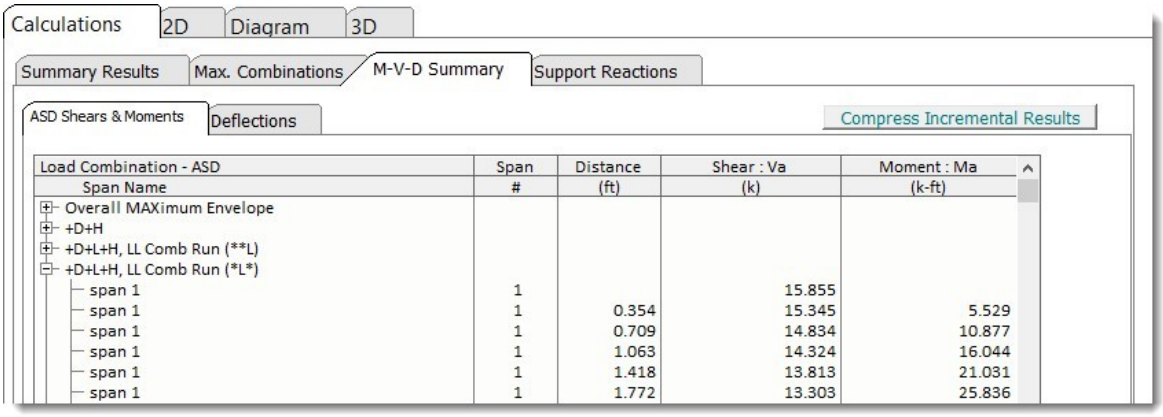


Calculations | 2D | Diagram | 3D

Summary Results | Max. Combinations | M-V-D Summary | Support Reactions

Load Combination - ASD / Beam Segment	Span	Summary of Moment Values Mmax+	Mmax-	Ma-max	Shear Va max
Overall MAXimum Envelope					
Dsgn. L = 28.00 ft	1	219.560	-216.157	219.560	50.280
Dsgn. L = 20.00 ft	2	-0.000	-216.157	216.157	50.280
Dsgn. L = 28.00 ft	3	219.560	-216.157	219.560	50.280
+D+H					
Dsgn. L = 28.00 ft	1	98.470	-92.954	98.470	23.480
Dsgn. L = 20.00 ft	2	-0.000	-92.954	92.954	23.480
Dsgn. L = 28.00 ft	3	98.470	-92.954	98.470	23.480
+D+L+H, LL Comb Run (**L)					
Dsgn. L = 28.00 ft	1	106.951	-73.034	106.951	22.768
Dsgn. L = 20.00 ft	2	-0.000	-188.571	188.571	49.295
Dsgn. L = 28.00 ft	3	211.082	-188.571	211.082	49.295
+D+L+H, LL Comb Run (*L*)					
Dsgn. L = 28.00 ft	1	87.284	-120.541	120.541	30.400
Dsgn. L = 20.00 ft	2	31.435	-120.541	120.541	30.400
Dsgn. L = 28.00 ft	3	87.284	-120.541	120.541	24.465

M-V-D: Summary: Displays moment, shear, and deflections at small increments along all spans. Moment and shear are displayed for all load combinations. Deflection is displayed for service load combinations only.



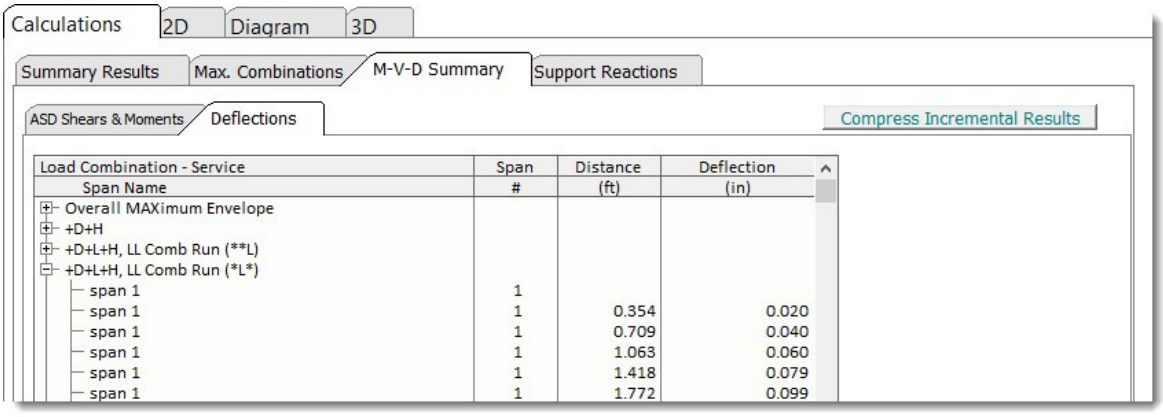
Calculations | 2D | Diagram | 3D

Summary Results | Max. Combinations | M-V-D Summary | Support Reactions

ASD Shears & Moments | Deflections

Compress Incremental Results

Load Combination - ASD / Span Name	Span #	Distance (ft)	Shear : Va (k)	Moment : Ma (k-ft)
Overall MAXimum Envelope				
+D+H				
+D+L+H, LL Comb Run (**L)				
+D+L+H, LL Comb Run (*L*)				
span 1	1		15.855	
span 1	1	0.354	15.345	5.529
span 1	1	0.709	14.834	10.877
span 1	1	1.063	14.324	16.044
span 1	1	1.418	13.813	21.031
span 1	1	1.772	13.303	25.836



Calculations | 2D | Diagram | 3D

Summary Results | Max. Combinations | M-V-D Summary | Support Reactions

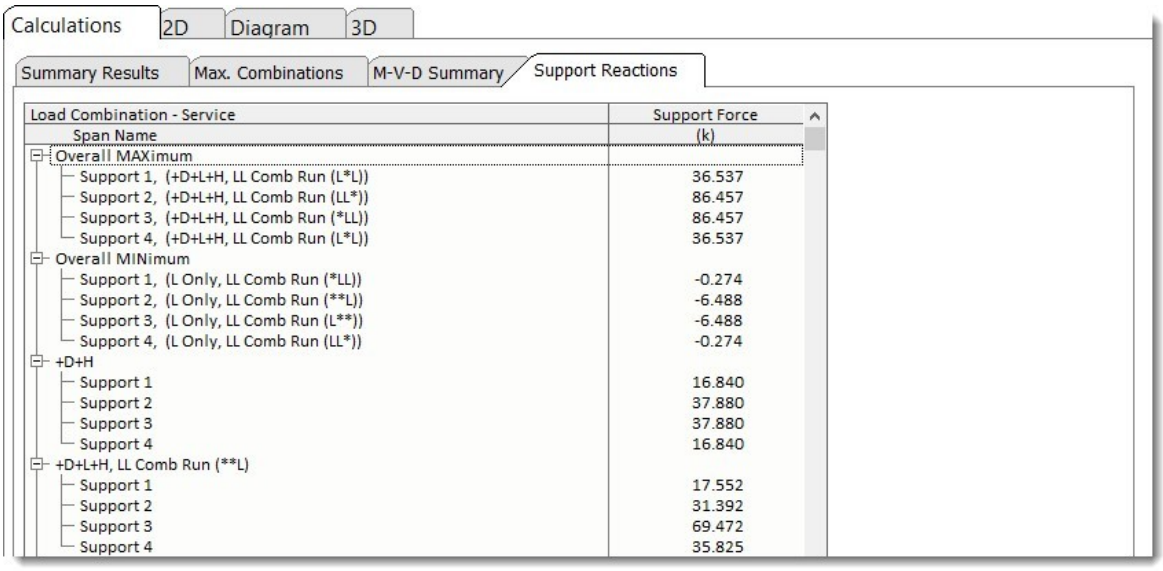
ASD Shears & Moments | Deflections

Compress Incremental Results

Load Combination - Service / Span Name	Span #	Distance (ft)	Deflection (in)
Overall MAXimum Envelope			
+D+H			
+D+L+H, LL Comb Run (**L)			
+D+L+H, LL Comb Run (*L*)			
span 1	1		
span 1	1	0.354	0.020
span 1	1	0.709	0.040
span 1	1	1.063	0.060
span 1	1	1.418	0.079
span 1	1	1.772	0.099

© 2026 ENERCALC, LLC

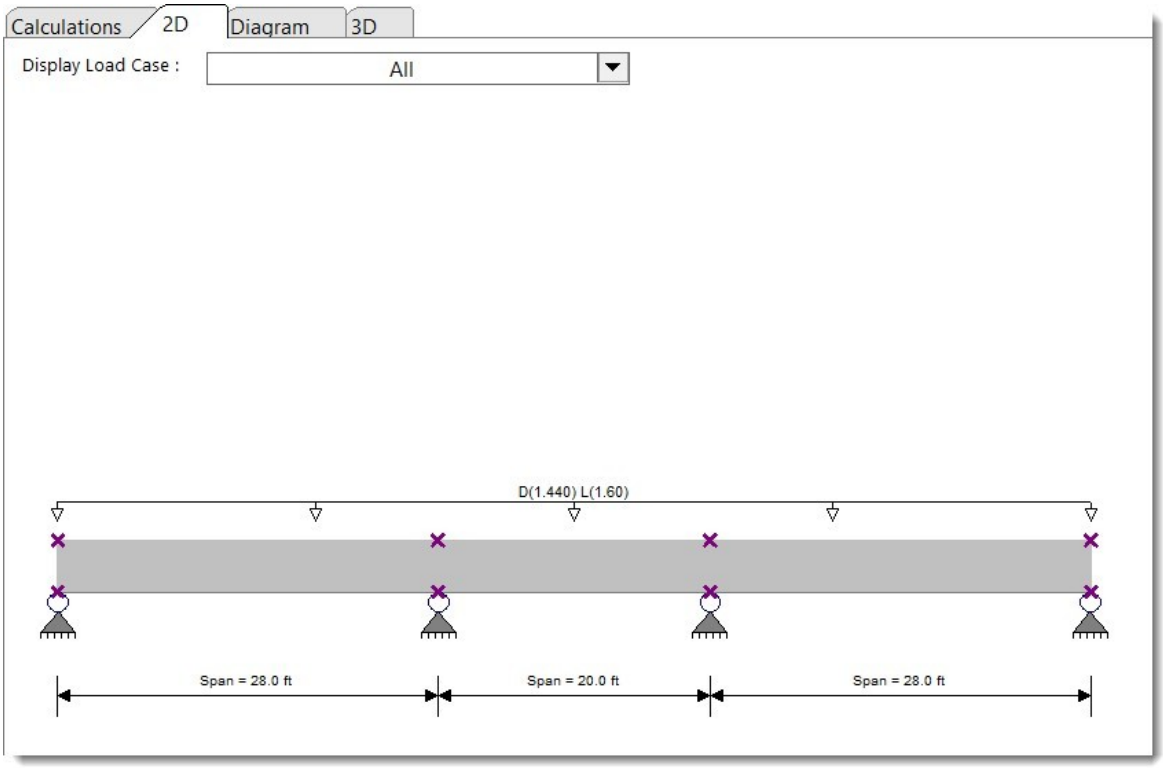
Support Reactions: Displays support reactions for all supports, for all load combinations.



Load Combination - Service Span Name	Support Force (k)
Overall MAXimum	
Support 1, (+D+L+H, LL Comb Run (L*L))	36.537
Support 2, (+D+L+H, LL Comb Run (LL*))	86.457
Support 3, (+D+L+H, LL Comb Run (*LL))	86.457
Support 4, (+D+L+H, LL Comb Run (L*L))	36.537
Overall MINimum	
Support 1, (L Only, LL Comb Run (*LL))	-0.274
Support 2, (L Only, LL Comb Run (**L))	-6.488
Support 3, (L Only, LL Comb Run (L**))	-6.488
Support 4, (L Only, LL Comb Run (LL*))	-0.274
+D+H	
Support 1	16.840
Support 2	37.880
Support 3	37.880
Support 4	16.840
+D+L+H, LL Comb Run (**L)	
Support 1	17.552
Support 2	31.392
Support 3	69.472
Support 4	35.825

2D:

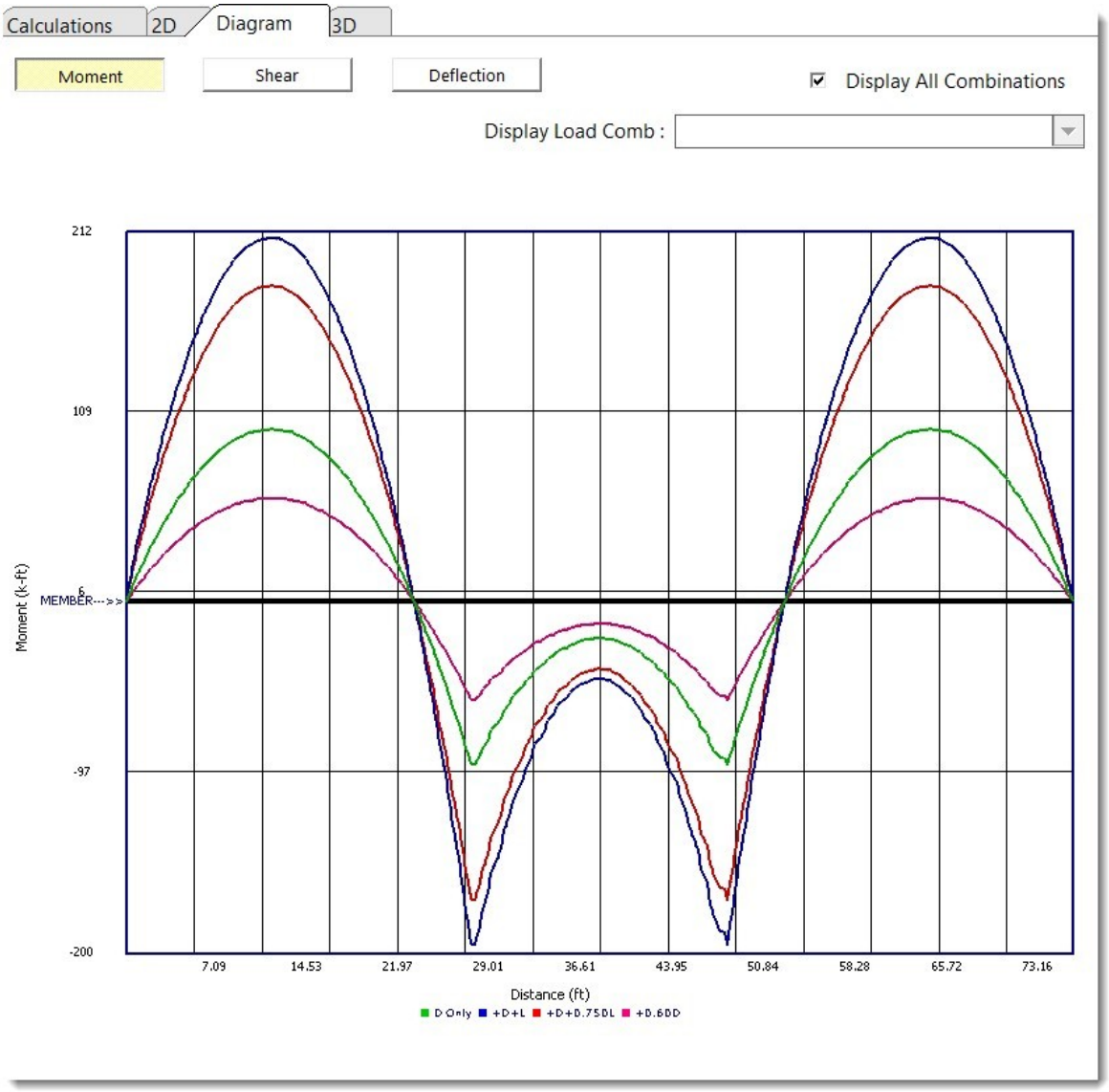
Displays a sketch of the beam, indicating span lengths, support conditions, and applied loads.



© 2026 ENERCALC, LLC

Diagram:

Displays a graphic depiction of the beam with superimposed graphs of Moment, Shear, or Deflection for a selected load combination, or for an envelope of all load combinations.



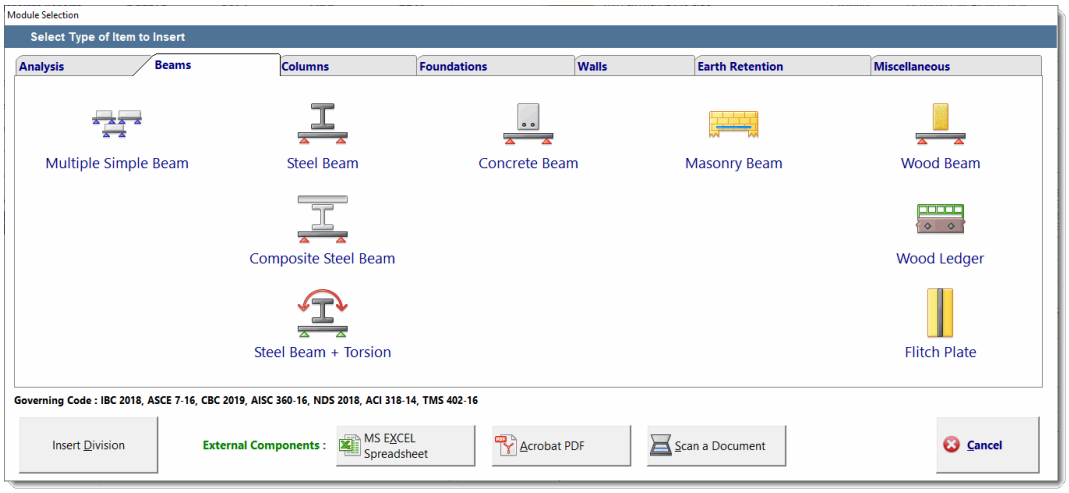
13.4 Beams

Last revised: Tuesday, February 3, 2026 at 06:12 PM

Overview

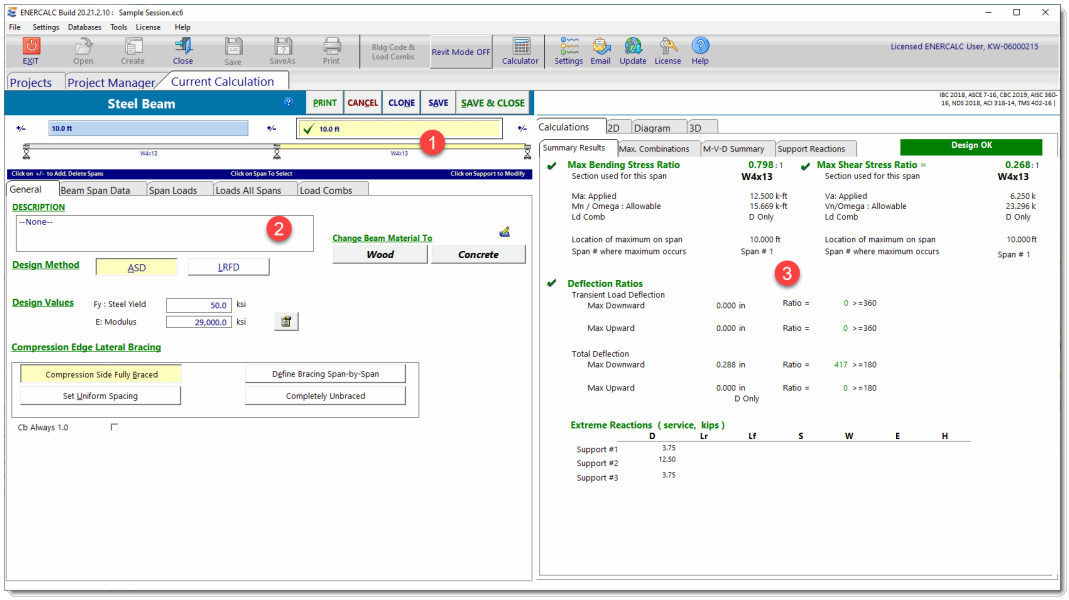
There is one main module for designing Steel, Concrete and Wood beams. There are separate modules for Composite Steel Beams, Steel Beams with Torsional Loads, and one for Masonry Lintels.

© 2026 ENERCALC, LLC



This section deals ONLY with typical single- or multi-span Steel, Concrete & Wood beams.

The presentation screen is divided into three areas: **Beam representation & modification**, **Data Entry** & **Calculation Results** as illustrated in the screen capture below:

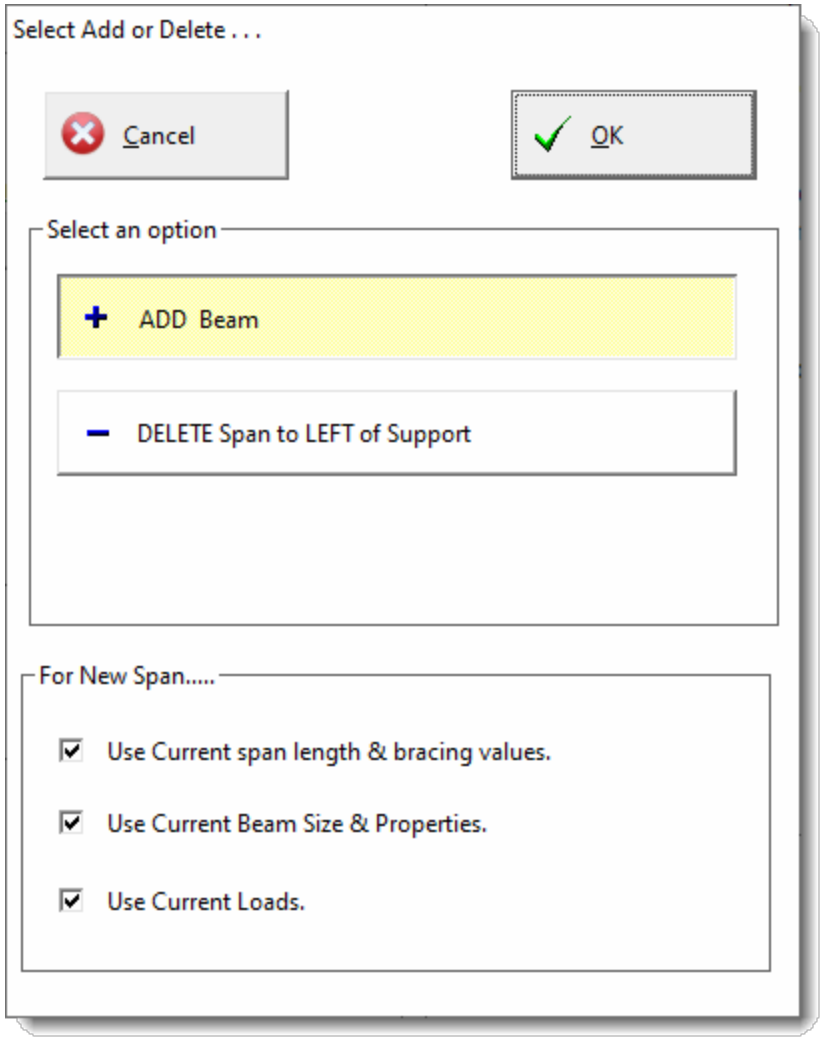


(1) Beam representation & modification: This area allows you to create and modify the beam layout.

Click on the support icons to select the type of support fixity: Fixed, Pinned, or Free.

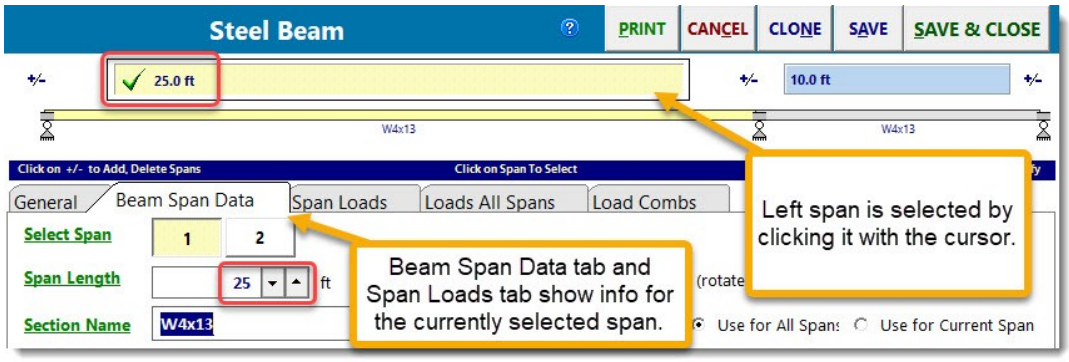
© 2026 ENERCALC, LLC

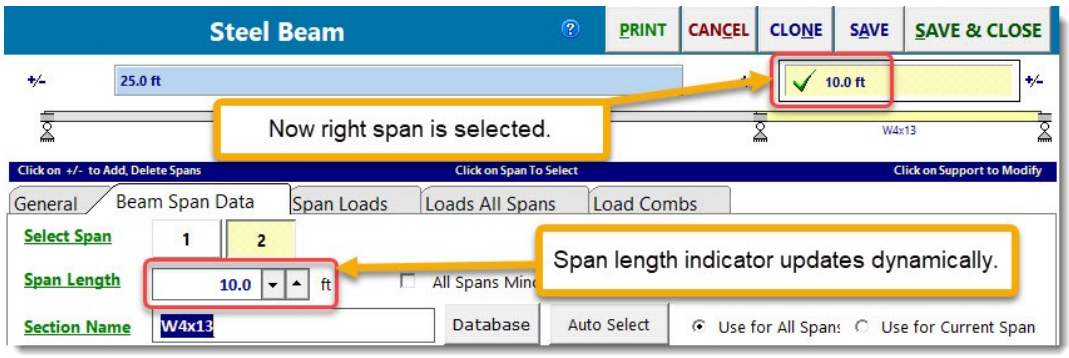
Click on the [**+/-**] icon to display a window to add or delete beam spans:



© 2026 ENERCALC, LLC

Click on the beam representation to change the beam specific data on the **Beam Span Data** and **Span Loads** tabs.



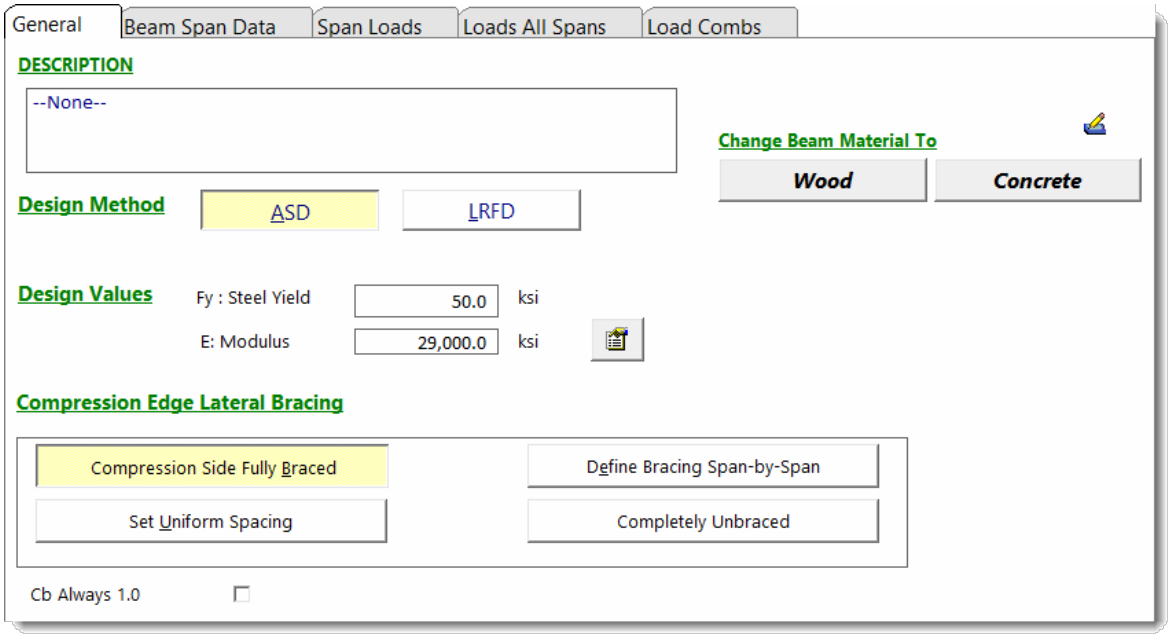


(2) Data Entry: This set of tabs is where you enter all information for the beam. These tabs will show different information according to material type selected. See the specific chapters for the items provided for each material. Click one of the following to jump: Wood 779 Steel 718 Concrete 752

© 2026 ENERCALC, LLC

Here is a summary of the purpose of each tab:

General



Beam Material

Clicking one of these buttons changes the material type used for the beam.

Compression Edge Lateral Bracing

These selections control how the module will evaluate the lateral compression edge bracing for the design. When **Define Bracing Span-by-Span** is selected, the unbraced length is defined on a span-by-span basis on the Beam Span Data tab. When any of the other options are selected, the unbraced length is defined here on the General tab, and that definition is applied to all spans of the beam.

Design Method

For wood & steel you can select ASD or LRFD design methods. Concrete design is always ultimate strength (LRFD).

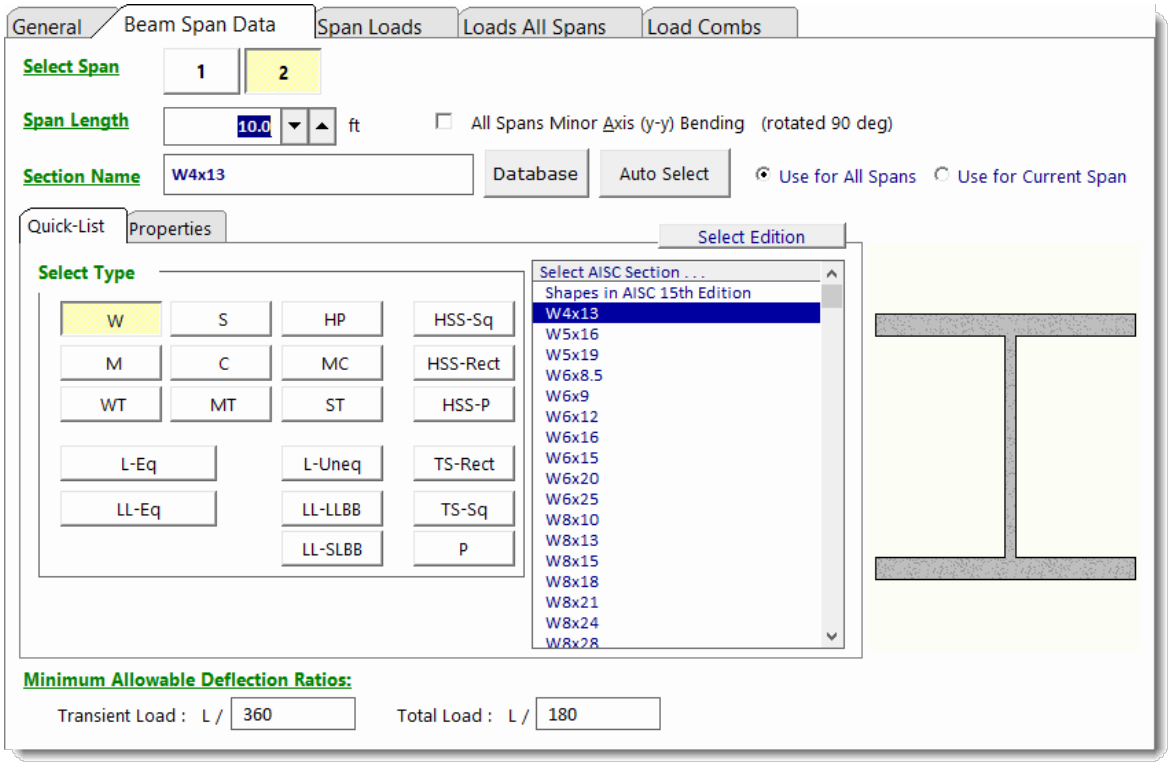
Design Values

Specify material-specific design values.

© 2026 ENERCALC, LLC

Beam Span Data

This tab is used to define span length and beam section information:



Select Span

These buttons allow you to select which span the values in the data entry area apply to.

Span Length

This is where you define the length of the currently selected span.

Deflection Ratios

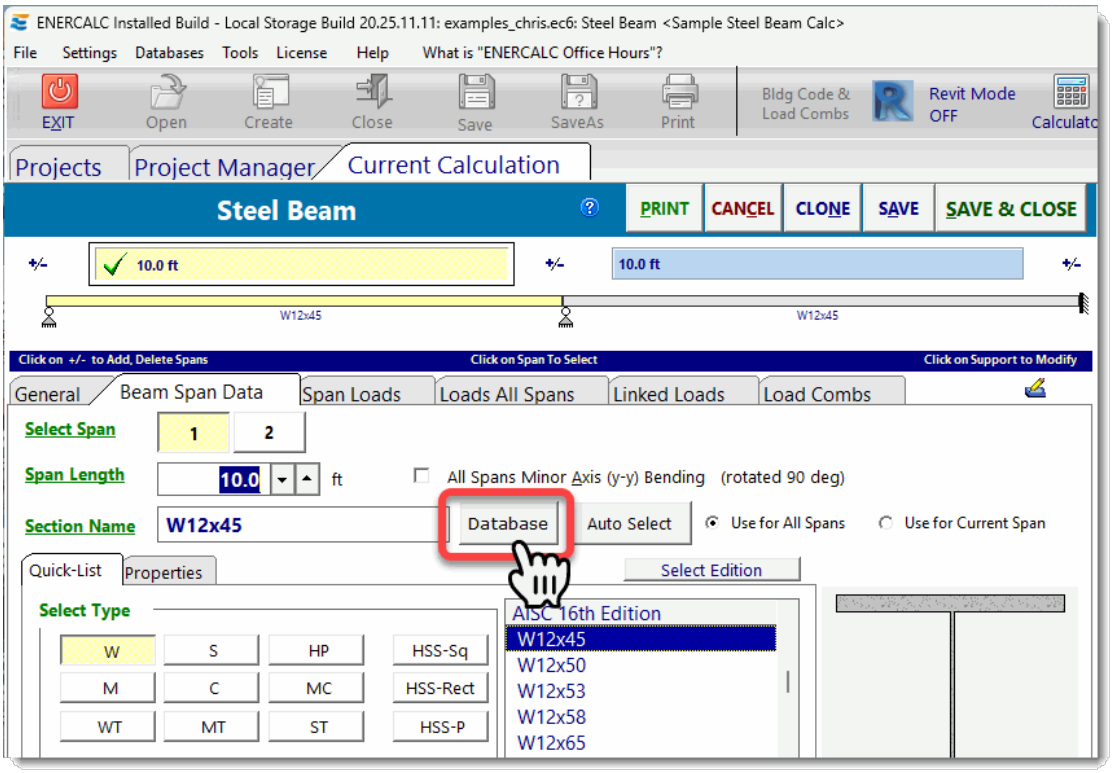
These are used as the basis for the deflection design check and also as the starting point for automatic member selection.

Section Name & Buttons

For steel and wood you can enter the standard beam section designation. You can also type in the section name and the module will search the built-in database for a match. If a match is found, the section properties will be loaded from the database and they will appear on the Properties tab.

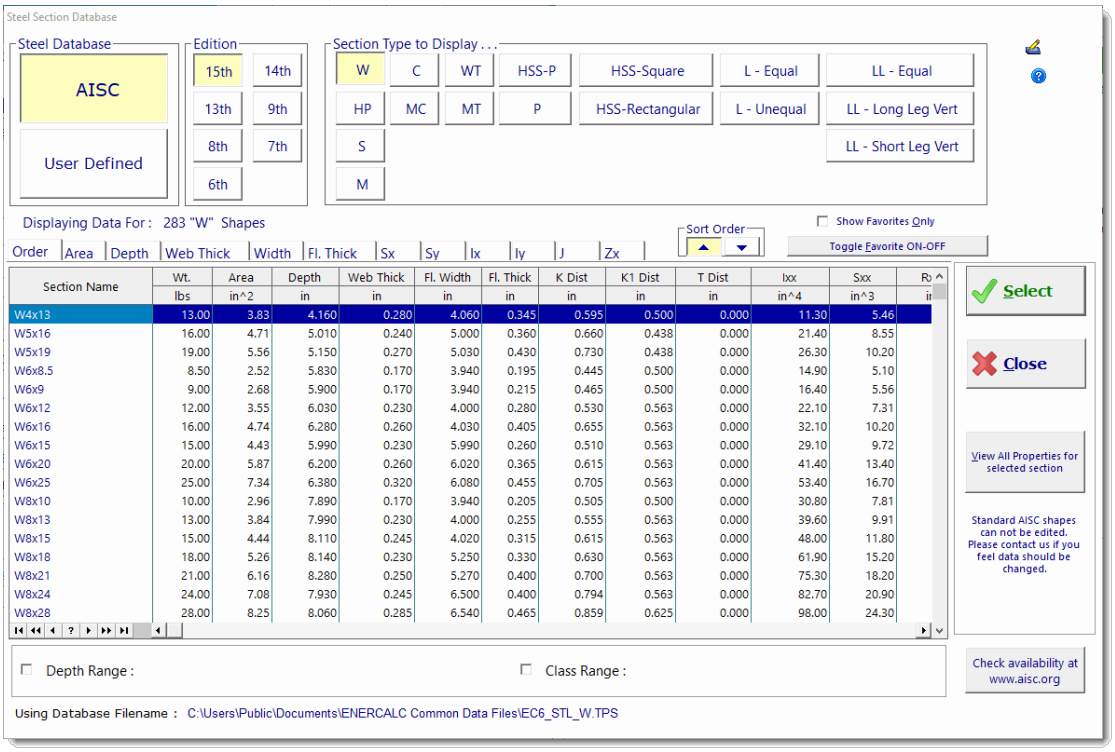
© 2026 ENERCALC, LLC

Click the button indicated below to open the section database for rolled steel shapes:



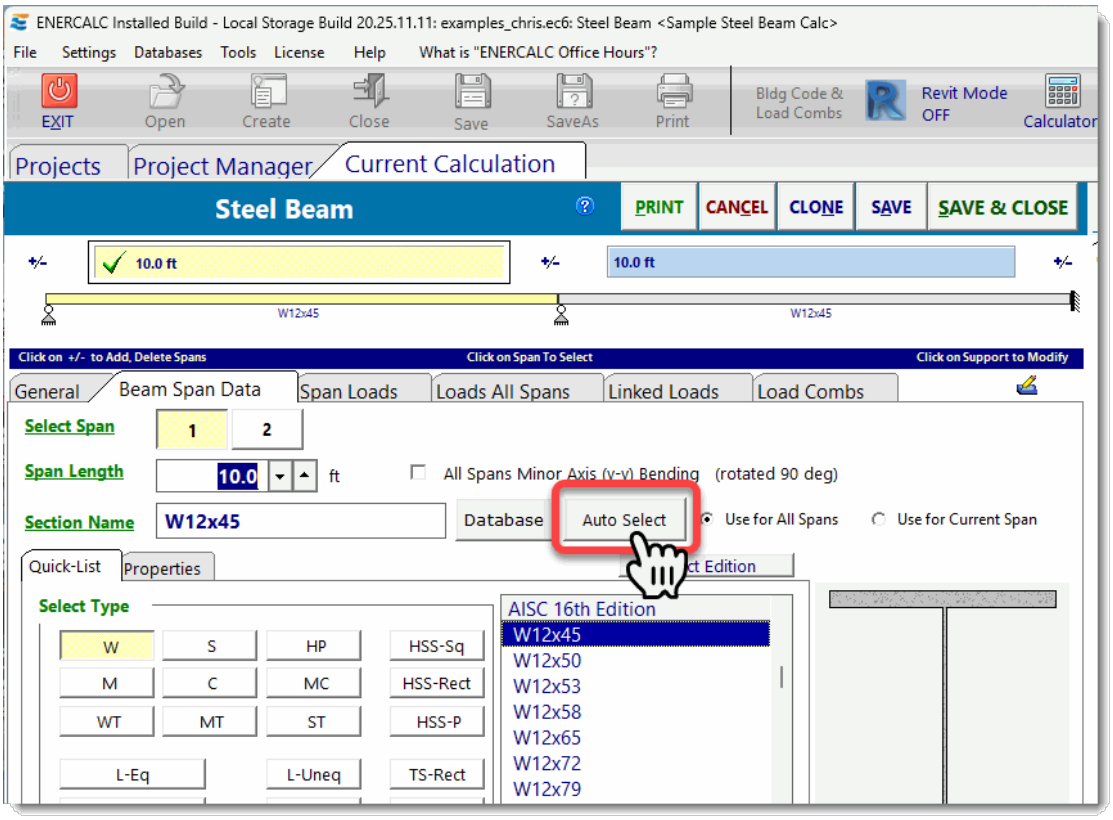
© 2026 ENERCALC, LLC

The database contains an extensive number of standard shapes commonly used in the USA.



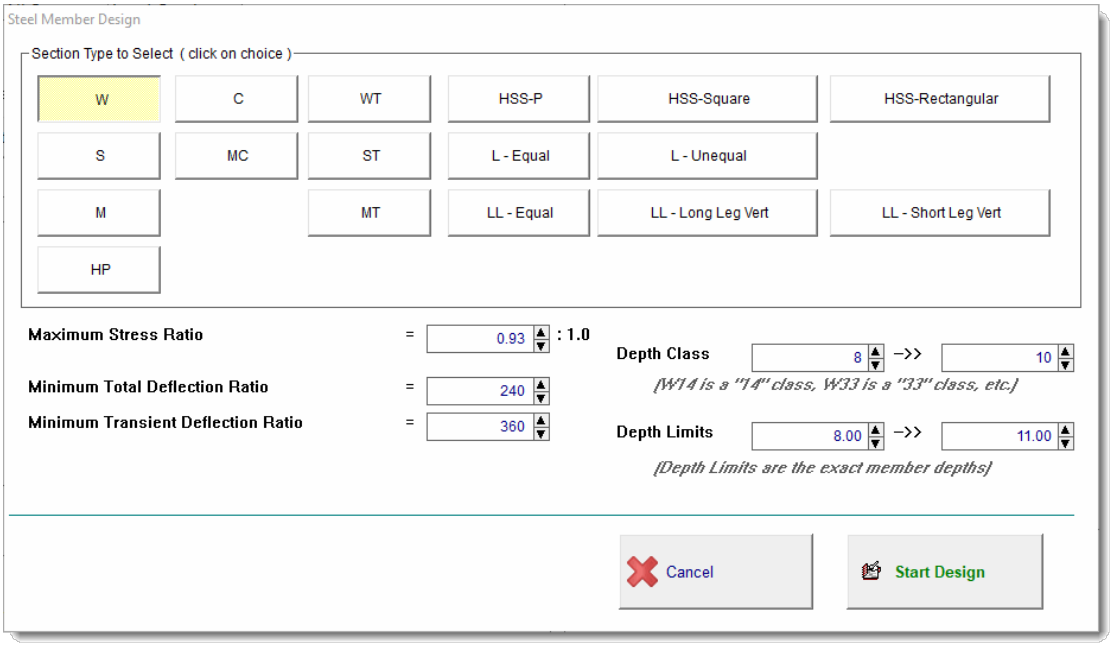
© 2026 ENERCALC, LLC

Click the button indicated below to display the Steel Member Design dialog:

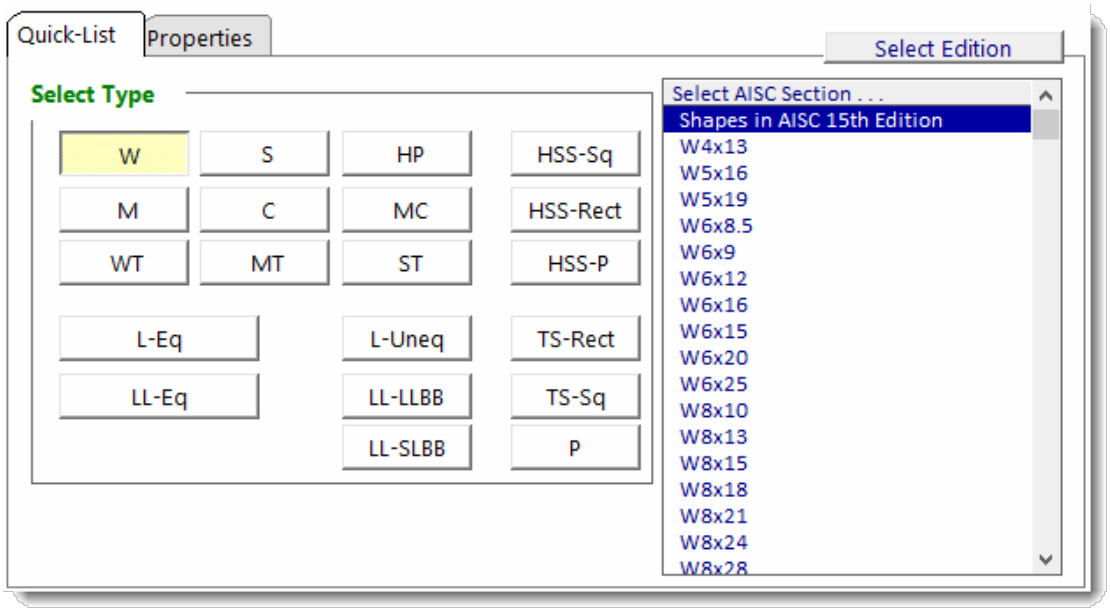


© 2026 ENERCALC, LLC

This provides control over the type of member to be selected and various stress ratio, deflection ratio and size limits to respect during the automatic member selection process.



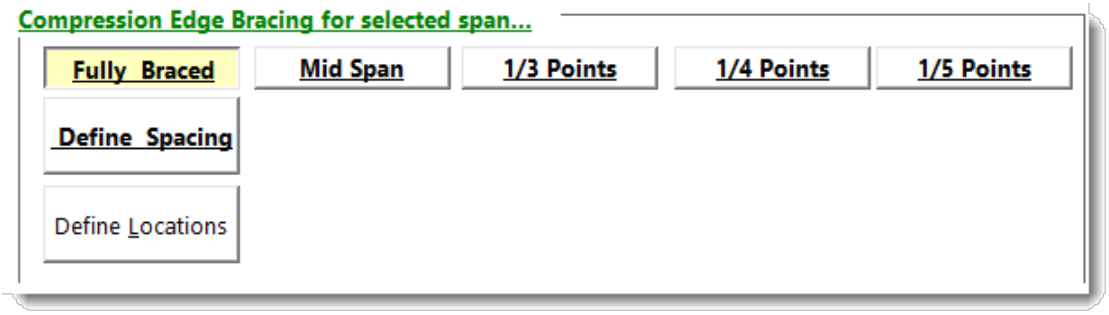
The Quick-List provides a fast way to select a member section from the database. Just select the desired edition of the database, click the type of member, scroll through the list and click on your selection.



© 2026 ENERCALC, LLC

Compression Edge Lateral Bracing

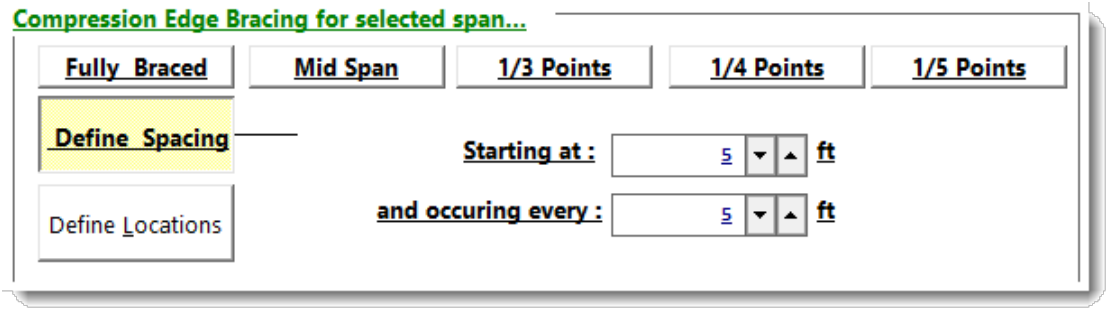
When you have selected Define Bracing Span-by-Span on the General tab you will see the following bracing options on the Beam Span Data tab:



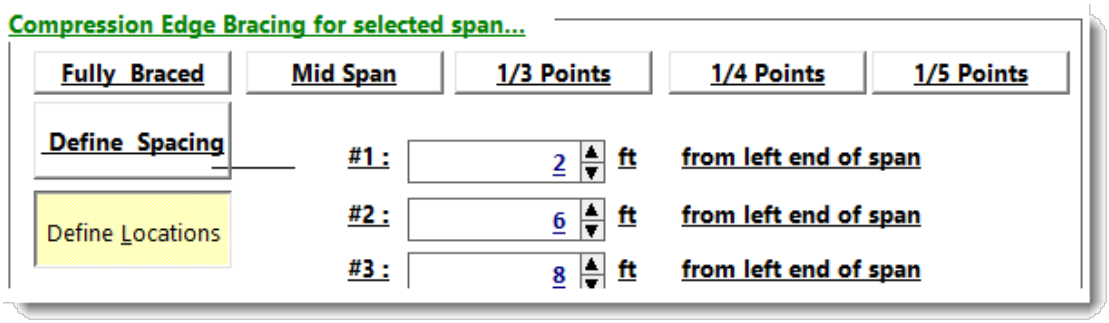
These selections control how the module will evaluate the lateral compression edge bracing for the beam span selected above.

The top row has common selections for fully-braced conditions as well as options to divide the selected span into segments of equal braced length.

The Define Spacing option lets you set a starting point and subsequent spacing for the bracing within the selected span:



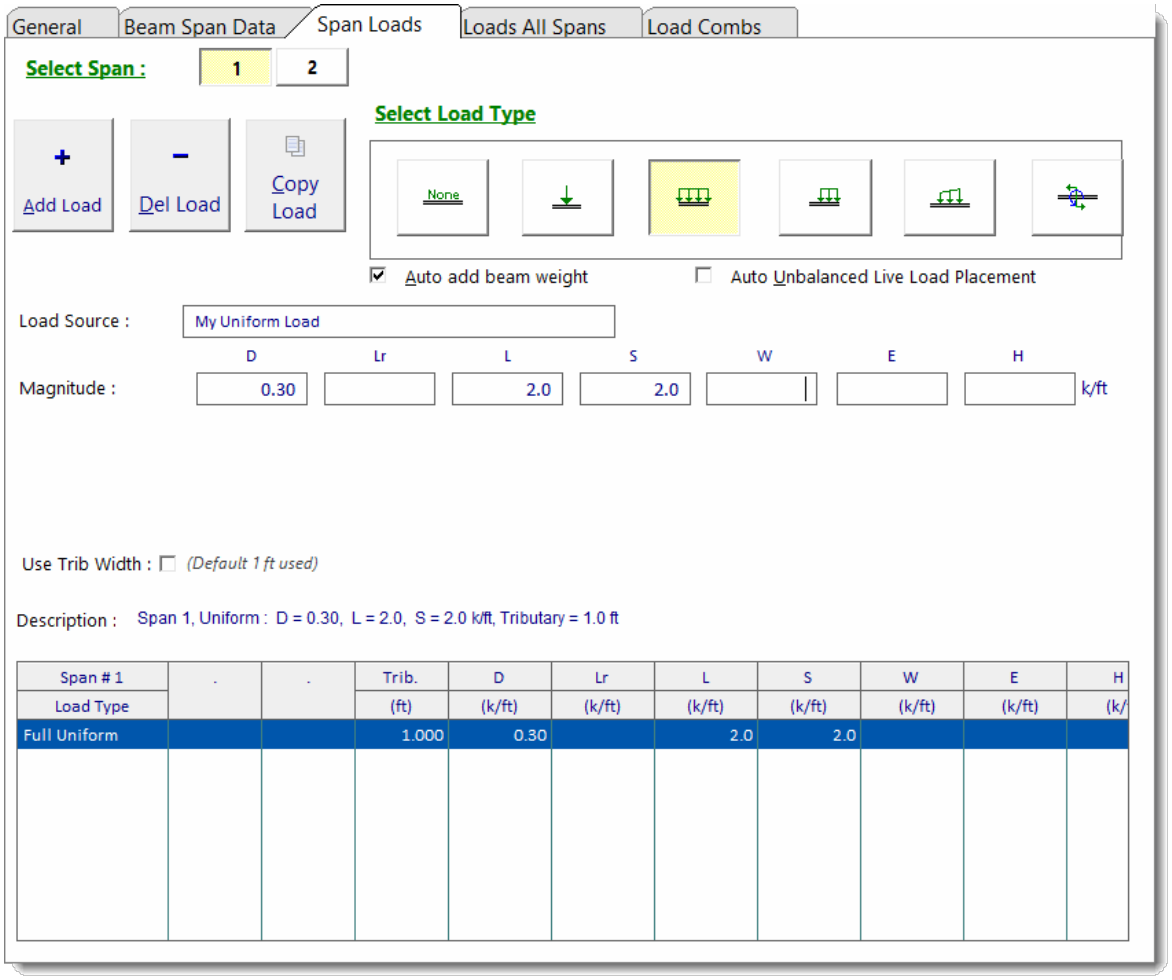
The Define Locations option lets you set up to three specific bracing locations referenced from the left end of the selected span:



© 2026 ENERCALC, LLC

Span Loads

This tab is used to specify loads **FOR THE SELECTED SPAN ONLY** (except as noted below) using the tools shown on the following screen capture:



Use the [**Add Load**], [**Copy Load**] and [**Delete Load**] buttons to add, copy, or delete loads on the selected span.

Use the **Load Type** selections to specify the type of load that will be added. This selection affects the **currently highlighted** item in the table of loads. The data entry areas below will change based on the type of load selected.

The **Auto Add Beam Weight** option will calculate the weight of the beam and add it to your applied loads as a uniform dead load on the beam. Note that this option applies to the FULL LENGTH of the beam...it is NOT a setting that can be set on a span-by-span basis.

The **Auto Unbalanced Live Load Placement** option is a very powerful selection. When you have two or more spans, you can select this item and the module will automatically generate load combinations for all possible permutations of patterned live load being placed on alternate spans. For instance, on a two-span beam, it would create conditions

© 2026 ENERCALC, LLC

that place live load on both spans, live load on the left span only, and live load on the right span only. That is a total of three permutations of live load, and it will do this for ALL of the load combinations that are selected to run.

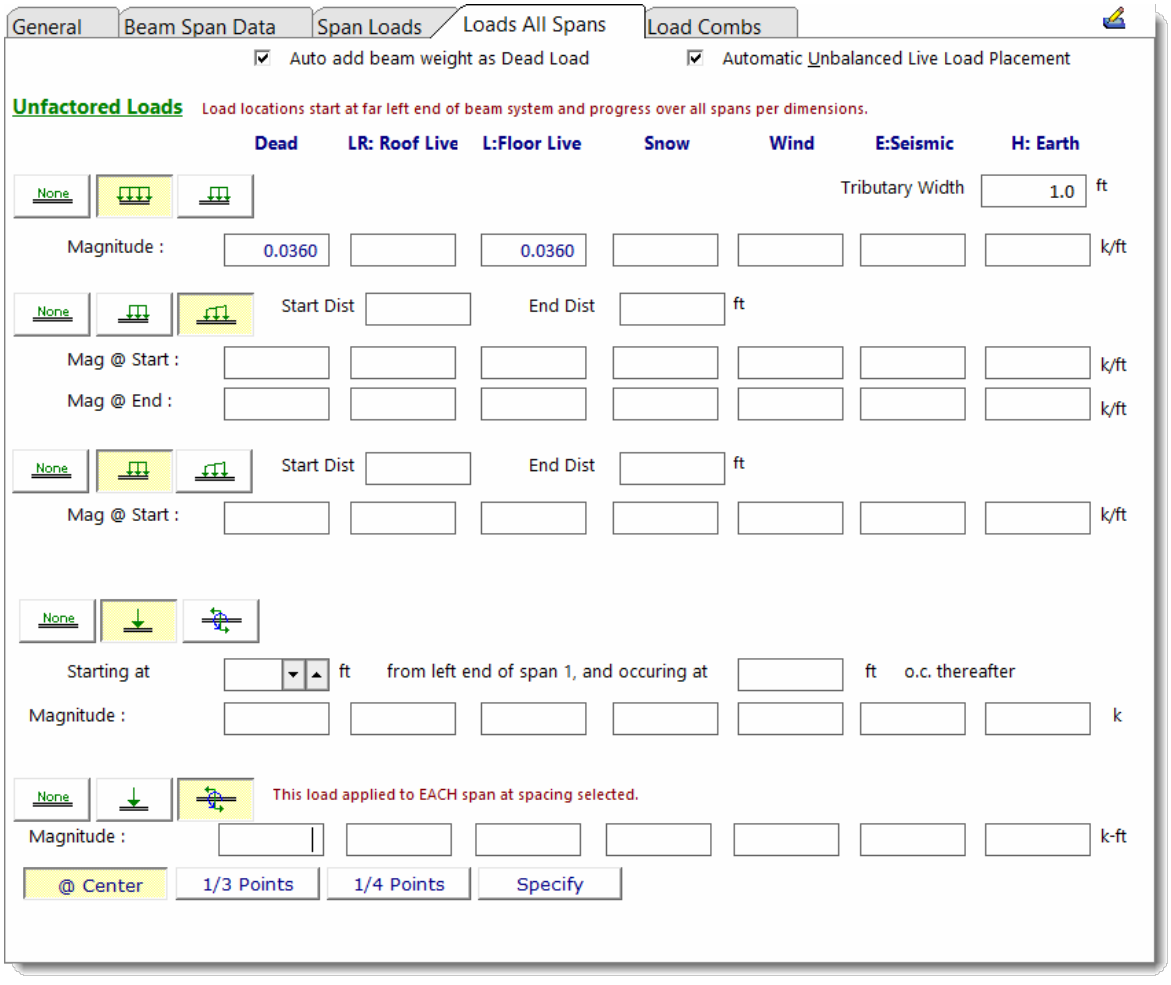
NOTE: This can significantly increase recalculation time needed for beams with many spans.

Load classifications are: D: Dead, Lr: Roof Live, L: Live, S: Snow, W: Wind, E: Seismic, H: Earth Pressure

Loads that get patterned with this feature: Lr: Roof Live, L: Live

Loads All Spans

This tab offers tools that are used to specify loads with spacings or lengths that might cause them to overlap multiple spans as shown in the screen capture below:



Click the load type button on the left and the appropriate load entry items will appear to allow you to define the magnitude, location and extent of the load. With all load items set to [**None**], the tab will be almost entirely blank.

Start Dist and **End Dist** defines the application distance from the FAR LEFT end of the beam. For a beam with two 20' spans where you want to apply a uniform load 5' in from each end use Start Dist = 5.00 and End Dist = 35.00.

The third item allows you to define repeating point loads or moments. You define the position of the first load and the spacing increment thereafter.

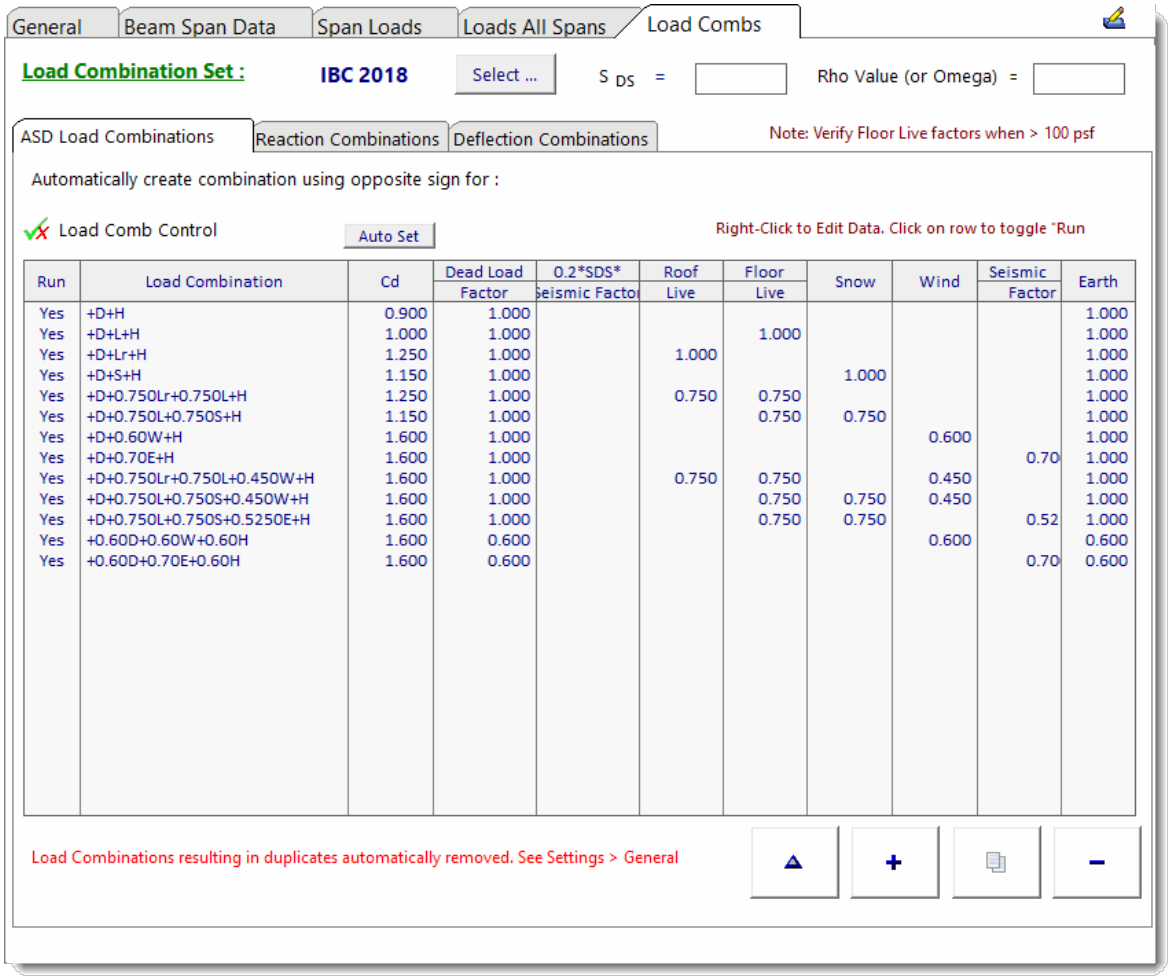
The last item allows you to define repeating loads on EACH span. Load specification is for the load in EACH span. For example, consider a two-span beam where the first span is 25' and the second span is 45'. Selecting [**1/3 Points**] will place loads at 8.33', 16.66' from the left support of the first span and at 15' and 30' from the left support of the second span. The "Specify" option allows you to provide a unique spacing measured from the left end of each span.

Although this tab is not typically used for single-span beams, the tools are perfectly applicable to single-span beams.

© 2026 ENERCALC, LLC

Load Combinations

This tab is used to specify the load combinations that will be run for the analysis of this beam, as shown in the screen capture below:



Run	Load Combination	Cd	Dead Load Factor	0.2*SDS* Seismic Factor	Roof Live	Floor Live	Snow	Wind	Seismic Factor	Earth
Yes	+D+H	0.900	1.000							1.000
Yes	+D+L+H	1.000	1.000			1.000				1.000
Yes	+D+Lr+H	1.250	1.000		1.000					1.000
Yes	+D+S+H	1.150	1.000				1.000			1.000
Yes	+D+0.750Lr+0.750L+H	1.250	1.000		0.750	0.750				1.000
Yes	+D+0.750L+0.750S+H	1.150	1.000			0.750	0.750			1.000
Yes	+D+0.60W+H	1.600	1.000					0.600		1.000
Yes	+D+0.70E+H	1.600	1.000						0.70	1.000
Yes	+D+0.750Lr+0.750L+0.450W+H	1.600	1.000		0.750	0.750		0.450		1.000
Yes	+D+0.750L+0.750S+0.450W+H	1.600	1.000			0.750	0.750	0.450		1.000
Yes	+D+0.750L+0.750S+0.5250E+H	1.600	1.000			0.750	0.750		0.52	1.000
Yes	+0.60D+0.60W+0.60H	1.600	0.600					0.600		0.600
Yes	+0.60D+0.70E+0.60H	1.600	0.600						0.70	0.600

The **[Select]** button is used to retrieve load combinations sets from the Load Combination 78 database.

The icon with the delta symbol is used to edit the load combination values. Click on the delta symbol, and the multipliers for the currently selected load combination can be revised. After you change an item's values, click OK to save the change and view the revised load combination in the list.

Specify the magnitude of S_{DS} and/or Rho to automatically integrate the associated additional internal factor in the load combinations.

The option labeled [**Automatically create combination using opposite sign for**] triggers the module to create additional load combinations with the "**W**" and/or "**E**" factors set to

© 2026 ENERCALC, LLC

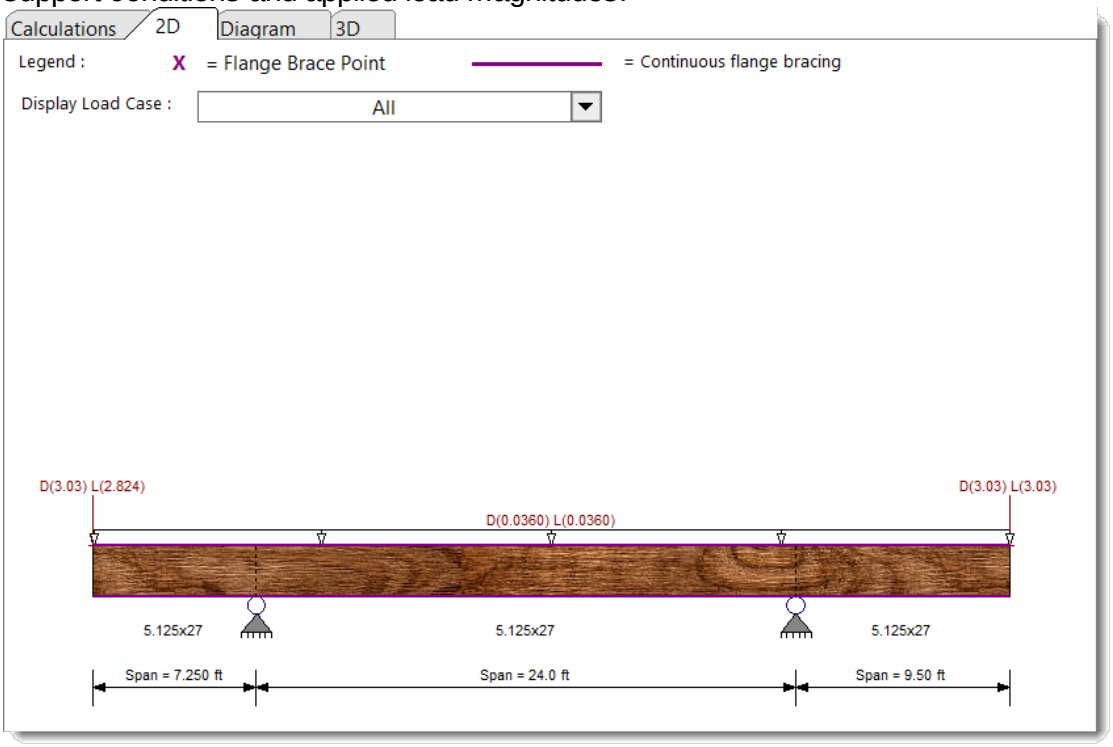
negative values. This has the effect of reversing the direction of application of the wind and seismic loads that have been applied to the member.

(3) Calculation Result: This set of tabs at the top of the right side of the screen allows you to select the desired results for review:

Calculations provides several tabs that let you review the numeric detail of the calculation. The left-most tab is always the Summary Results where the concise design is given.

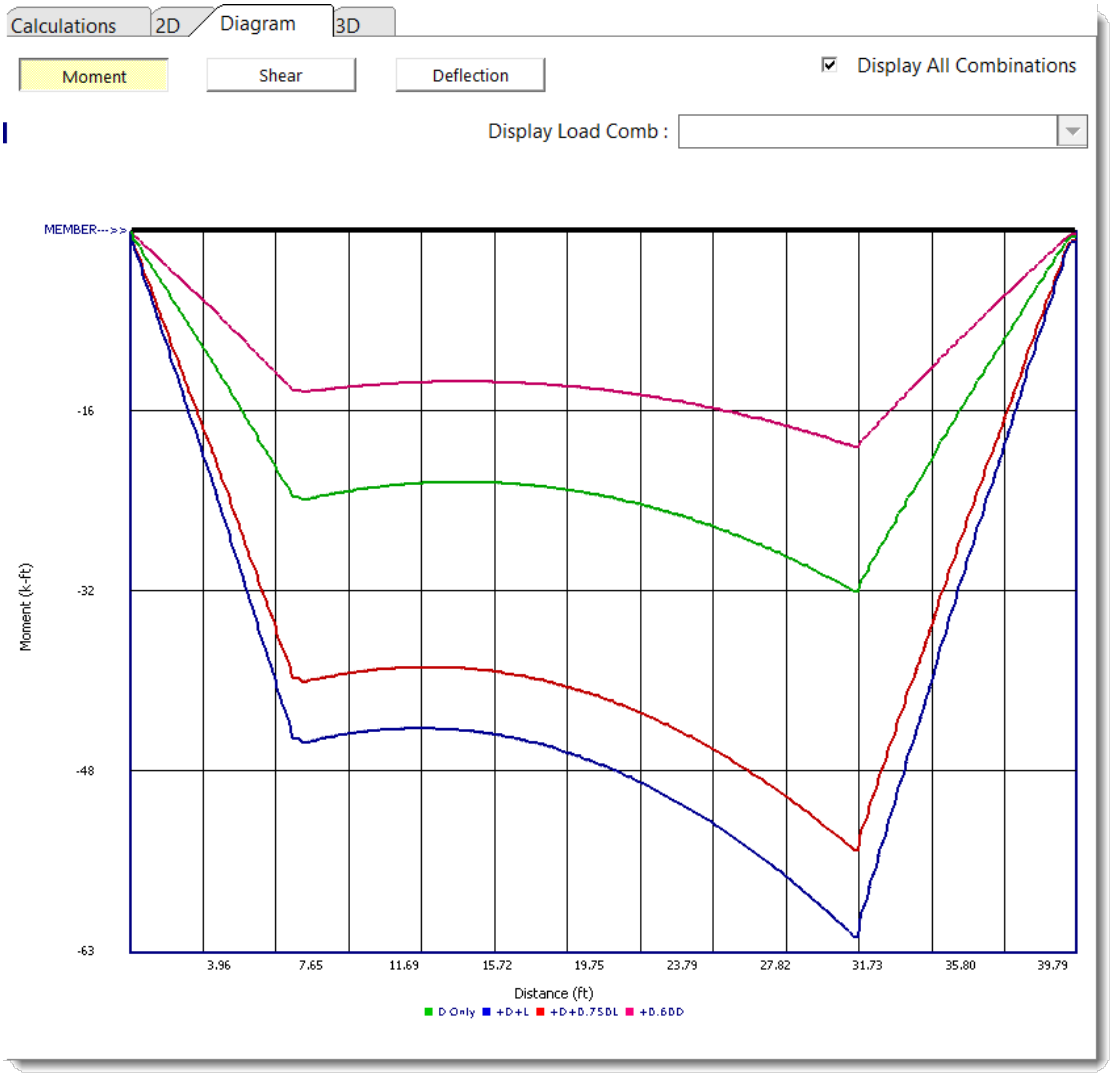
See the individual sections for each material type for specific discussions of these result sections.

2D provides a scale illustration of the item you are designing, including an indication of support conditions and applied load magnitudes.

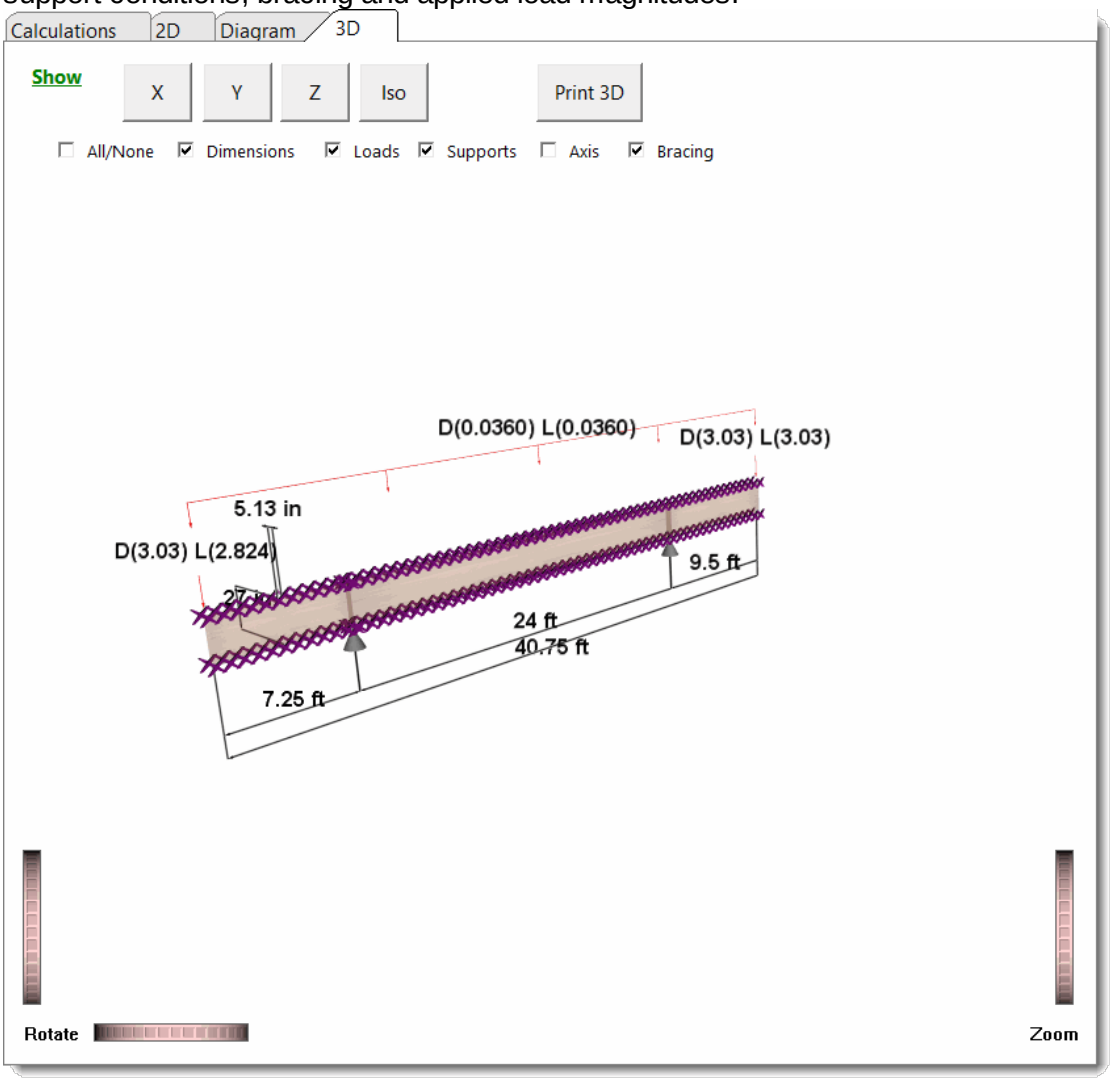


© 2026 ENERCALC, LLC

Diagram provides a moment, shear, or deflection diagram for the item you are designing.



3D displays a 3D rendering of the item you are designing, including an indication of support conditions, bracing and applied load magnitudes.



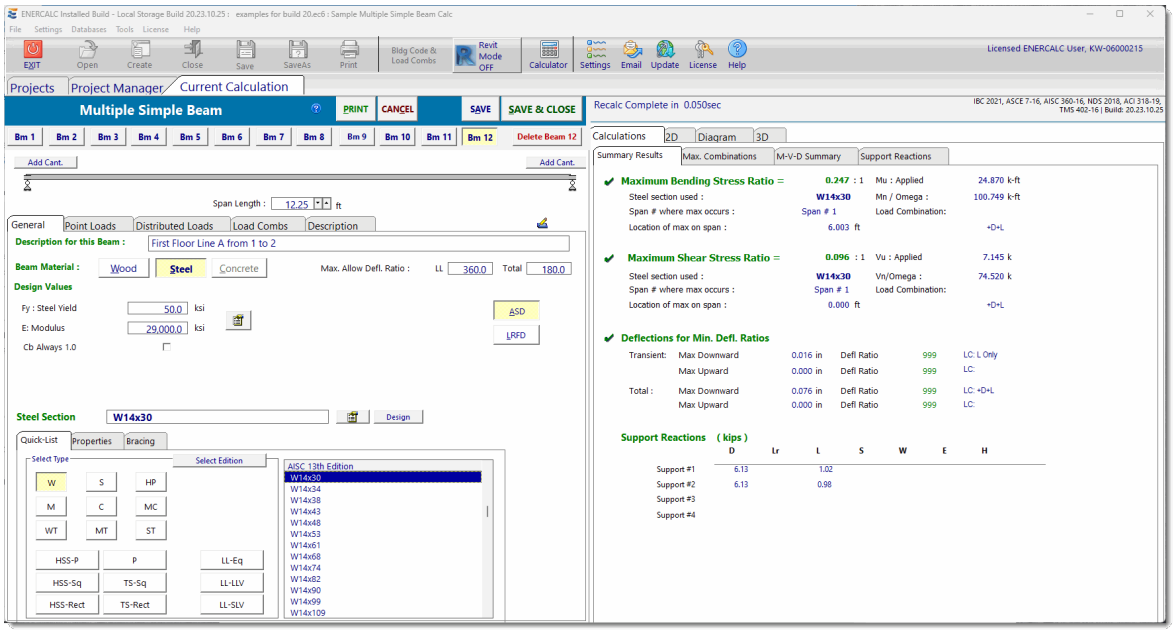
13.4.1 Multiple Simple Beam

Last revised: Tuesday, February 3, 2026 at 06:12 PM

This module is specifically designed to provide rapid analysis and design of simple beams. For complex, multi-span beams please use the other beam modules for wood, steel and concrete beams.

© 2026 ENERCALC, LLC

This module has a row of buttons above the beam graphical representation that allows you to add and select up to 12 beams to design. This can be seen in the screen capture below:



Looking at the screen capture above you can see:

(1) The highlighted beam button indicates the currently selected beam for which data is displayed.
(2) The button labeled [**Beam 2**] represents the second beam that was defined. Clicking on that button will save all of the data for [**Beam 1**] and display all the data for [**Beam 2**].
(3) [**Add**] is used to add another beam to this calculation. When two beams have already been defined, clicking [**Add**] will add a button labeled [**Beam 3**].
(4) [**Delete Bm X**] is used to delete the noted beam. It will delete the beam that is currently selected.

Directly below the band of beam selection buttons is the graphic that shows the basic layout of the currently selected beam. See below for a more detailed description.

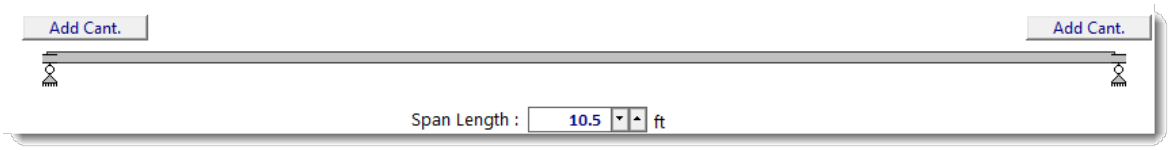
Note: In this module, the beam span length is always specified on the beam layout graphic.

Beam Layout Graphic

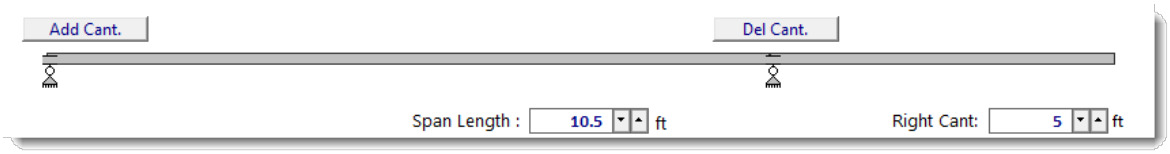
This area shows a graphic representation of the selected beam. There are several variations of beams that you can specify using the [**Add Cant**] buttons and also by clicking on the end support graphics.

© 2026 ENERCALC, LLC

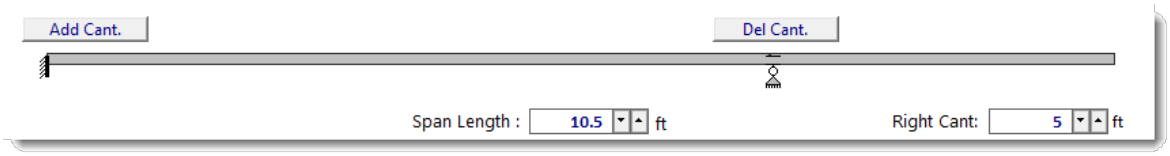
Here is a basic simple span beam:



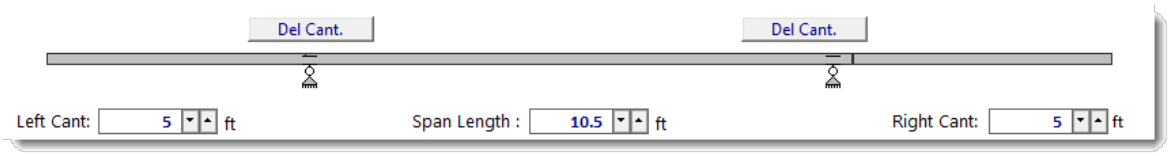
Clicking on the right [**Add Cant**] button in the image above adds a single cantilever at the right support as shown below:



Clicking on the left support icon in the image above displays a selection box so you can select a fixed end. Doing this sets the left support to fixed as shown below:



Clicking on the left [**Add Cant**] button in the image above adds a single cantilever at the left support, resulting in the double cantilever as shown below:



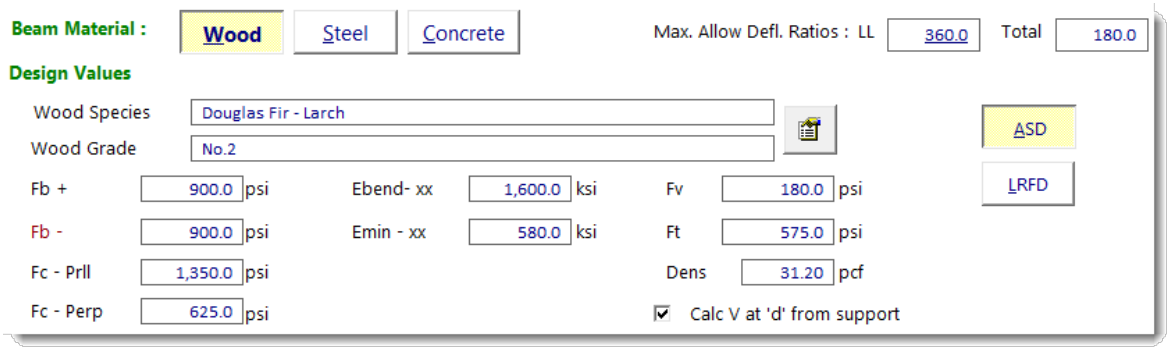
Note that adding the cantilever on the left also had the effect of automatically revising the left support back to a pinned condition.

General Tab

This tab is where you select the beam material, allowable stresses, beam size, and set the beam bracing layout.

© 2026 ENERCALC, LLC

For steel and wood you can select ASD or LRFD design methods. For concrete, only strength design is available.

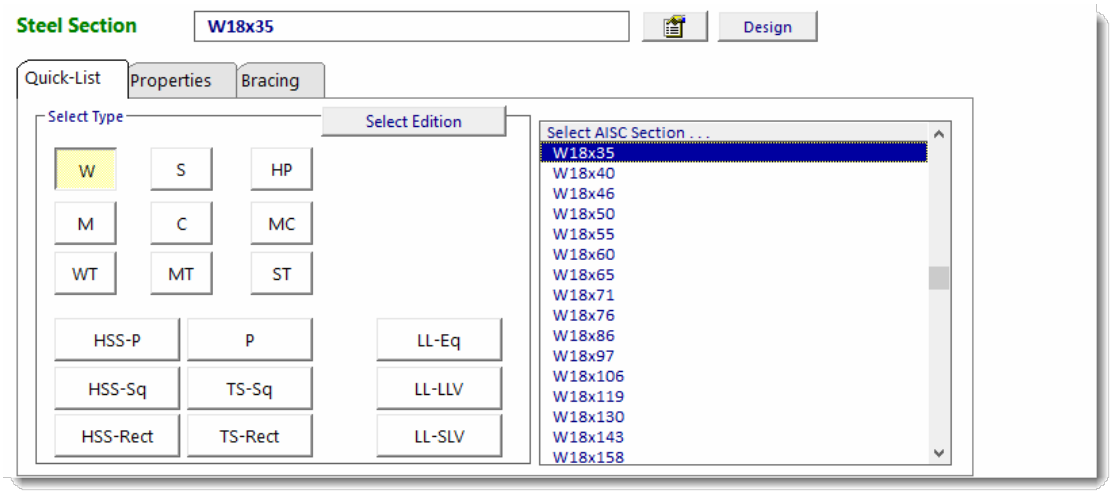


Steel Specific Tab Items

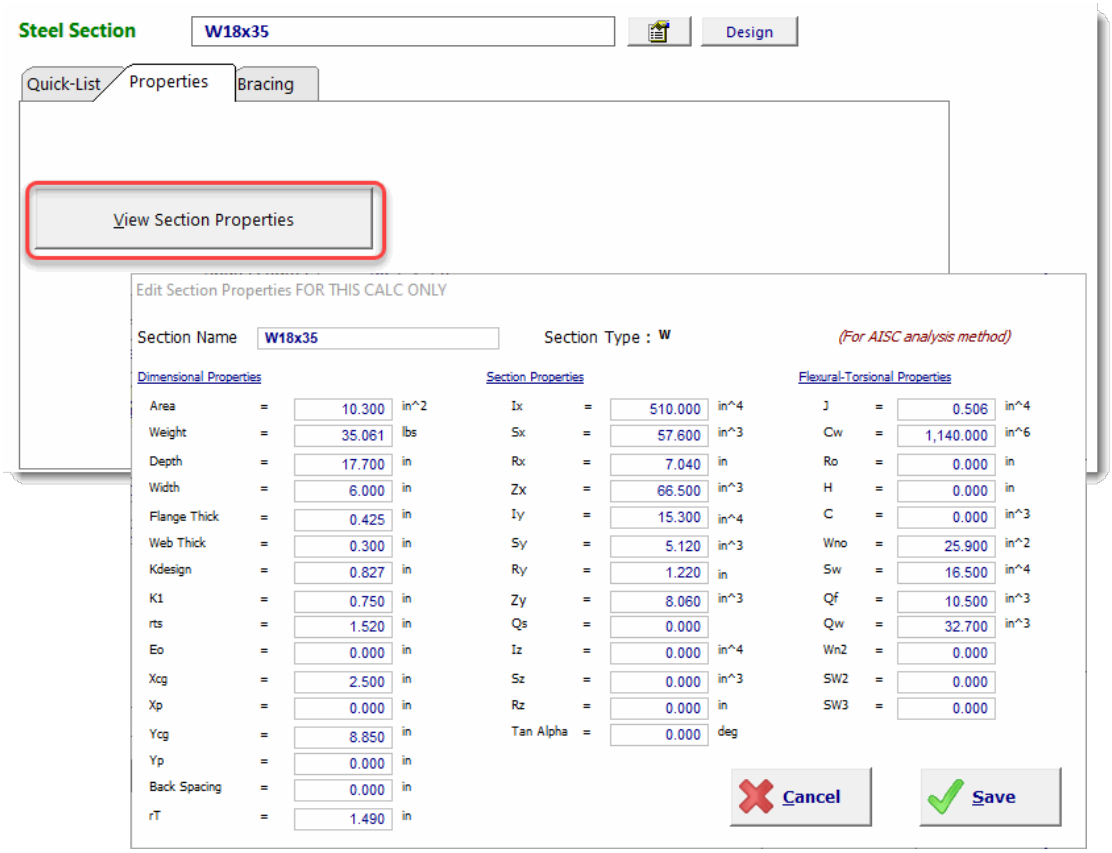
Design Values: This area enables you to enter the yield strength and elastic modulus of the steel. The paper-with-hand icon gives you access to the available AISC steel stress grades.

© 2026 ENERCALC, LLC

Quick-List Tab: This tab provides quick access to the built-in 13th Edition AISC steel section list. Clicking a section name like [**W**] or [**HP**] in the Select AISC Type category will display all the steel sections of that type in the list to the right.

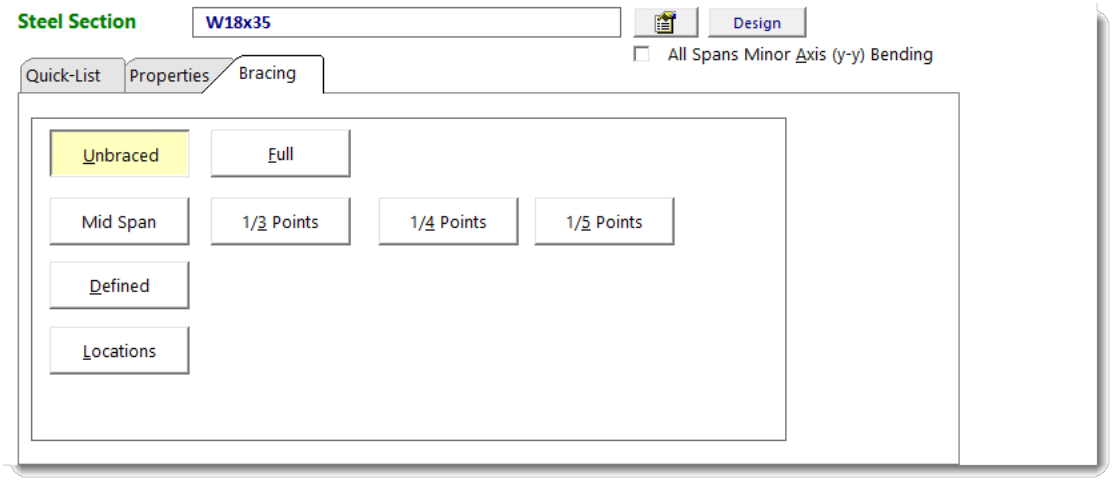


Properties Tab: This tab provides a [**View Section Properties**] button that when clicked displays all of the design values for the AISC section you have selected.



© 2026 ENERCALC, LLC

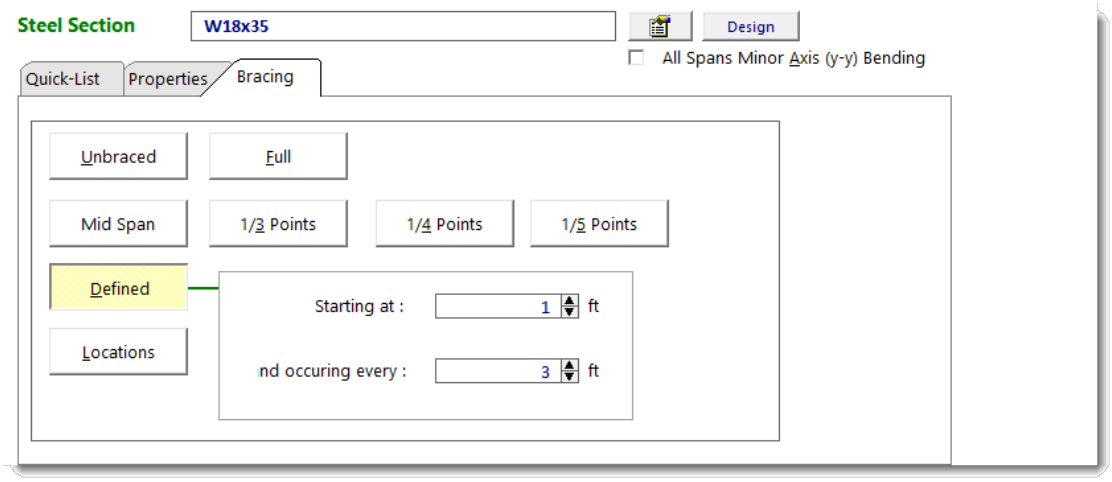
Bracing Tab: This tab allows you to select how your beam is braced against lateral-torsional buckling. Defined brace points are automatically assumed to brace both the top and the bottom flanges.



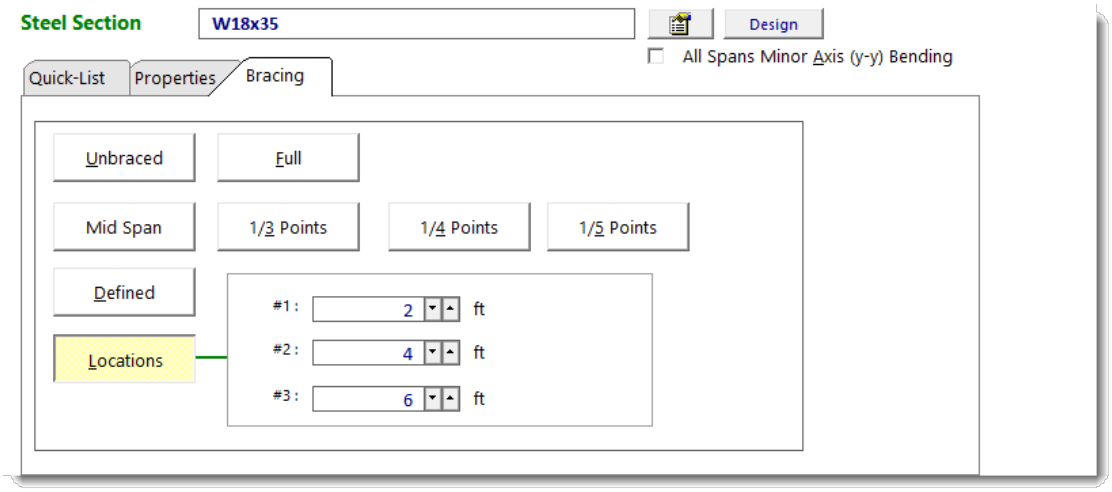
The first line offers the most basic options.....fully unbraced or fully braced.

The next line offers uniform brace spacing options.

The last two buttons offer the ability to specify brace spacings from a starting point or enable you to specify selected brace points, as shown below:

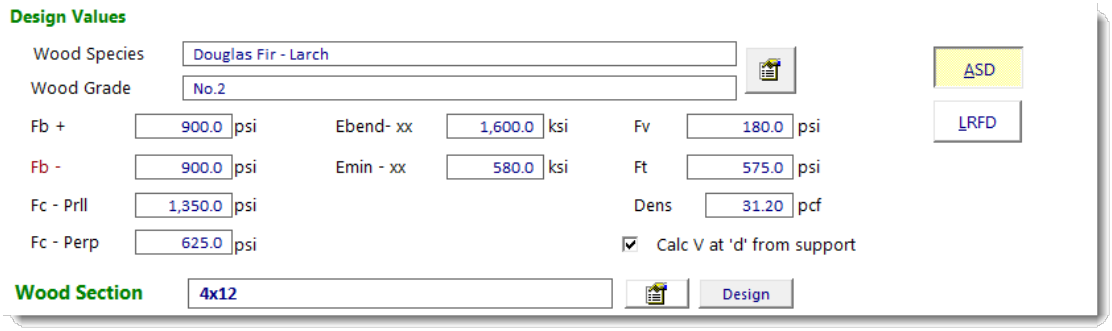


© 2026 ENERCALC, LLC



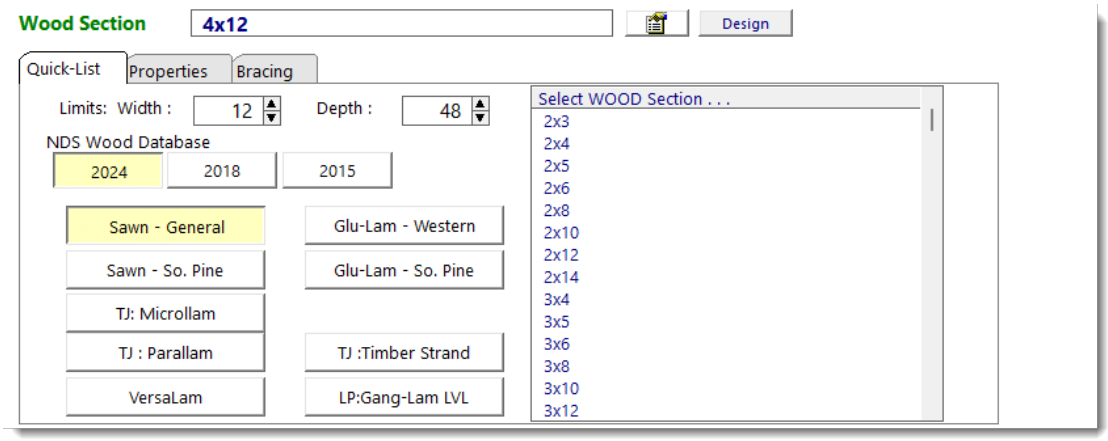
Wood Specific Tab Items

Design Values: This area enables you to specify the design values for the wood species & grade you want to use. Click the button to access the built-in NDS allowable stresses database.

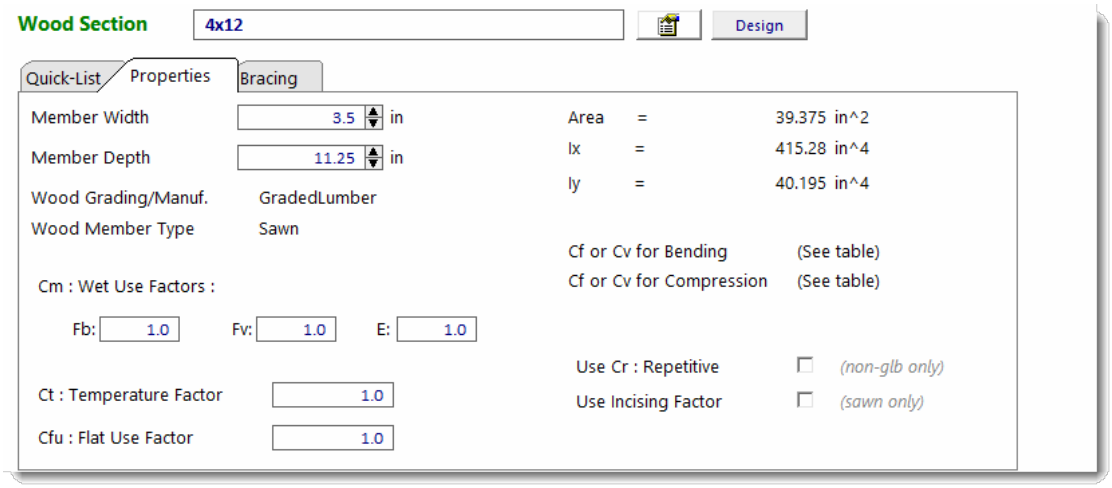


© 2026 ENERCALC, LLC

Quick-List Tab: This tab provides quick access to the database of wood sections. Clicking a section type button like [**Sawn-General**] or [**TJ:Microllam**] in the Select Type category will display all the sections of that type in the list to the right.



Properties Tab: This tab provides the values for the wood section you have chosen. You can enter different numbers here to modify the section.



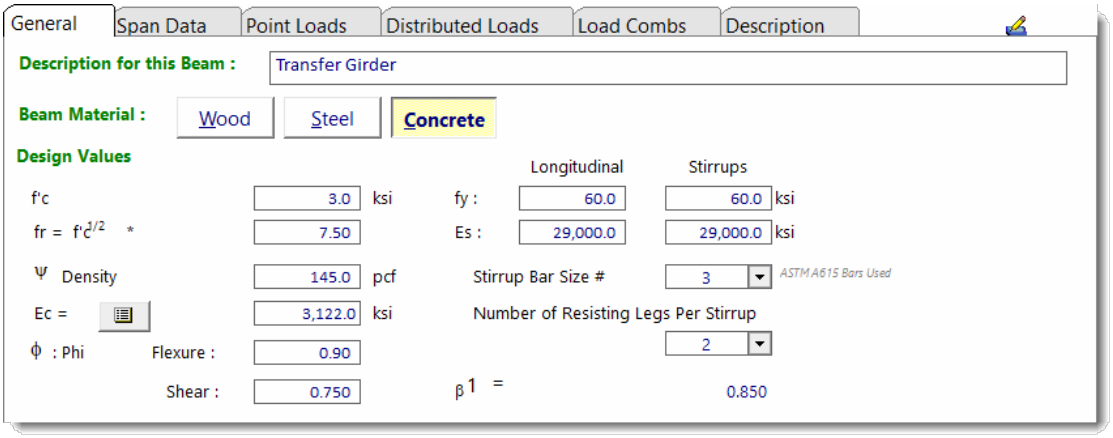
Bracing Tab: See information provided above in the Steel section for a summary of bracing options.

© 2026 ENERCALC, LLC

Concrete Specific Tab Items

Design Values: This area enables you to enter concrete and reinforcing strengths for the beam. In addition you can specify the stirrup size and Phi values.

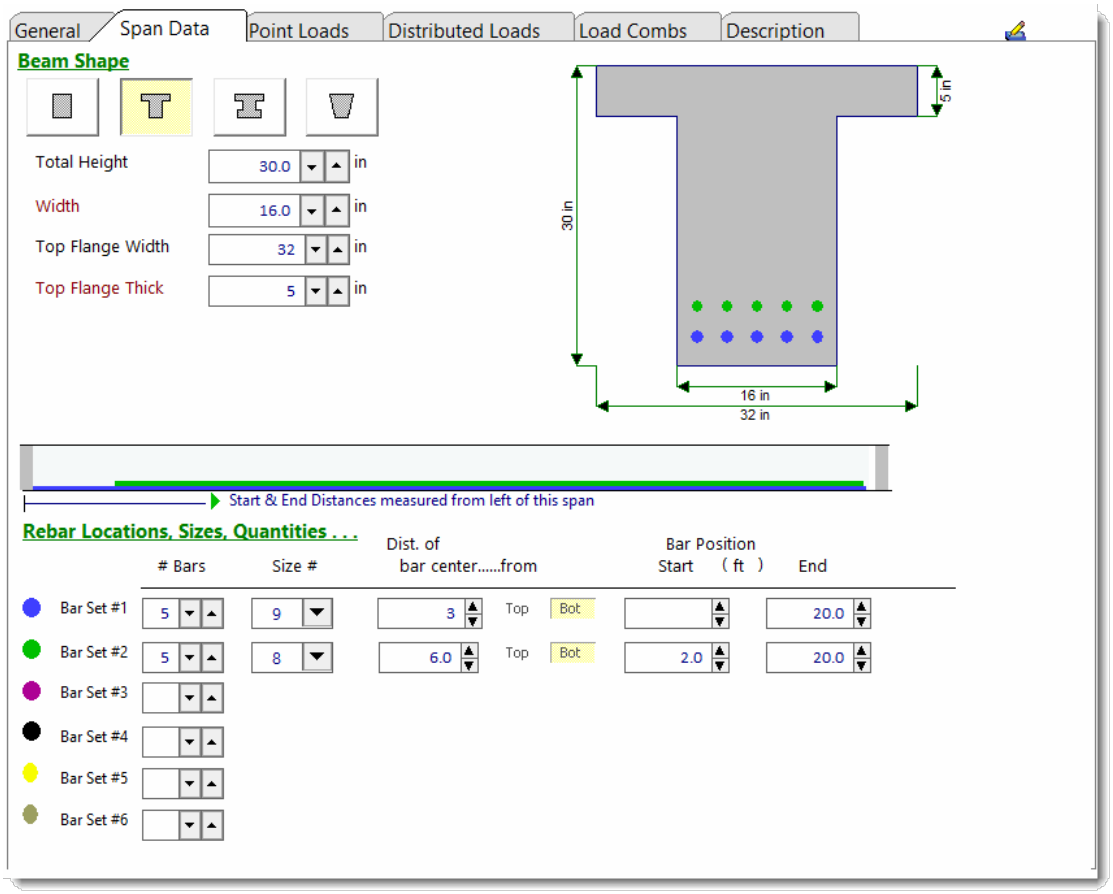
> **NOTE:** Support for concrete beams has been removed from the Multiple Simple Beam module when using IBC 2021 or later. To view any existing concrete calcs in Multiple Simple Beam, temporarily revise the governing code to an edition prior to IBC 2021. Also consider transcribing such calculation to the Concrete Beam module.



© 2026 ENERCALC, LLC

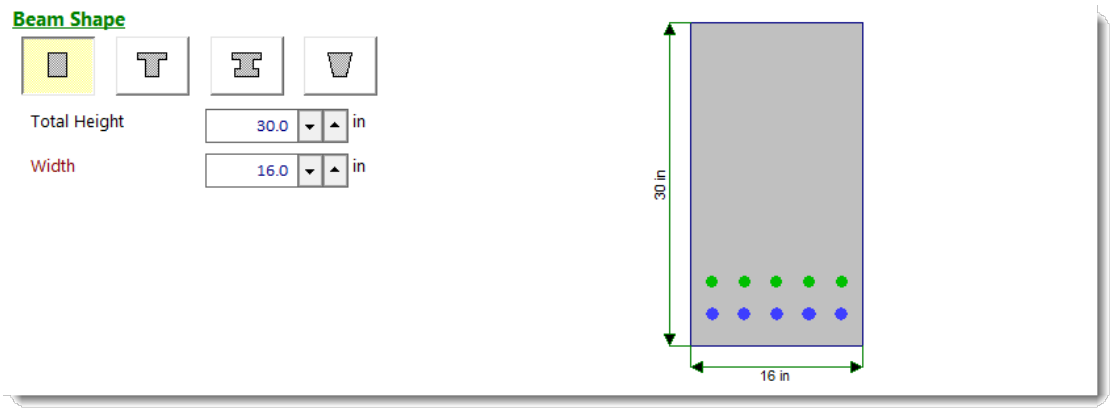
Span Data

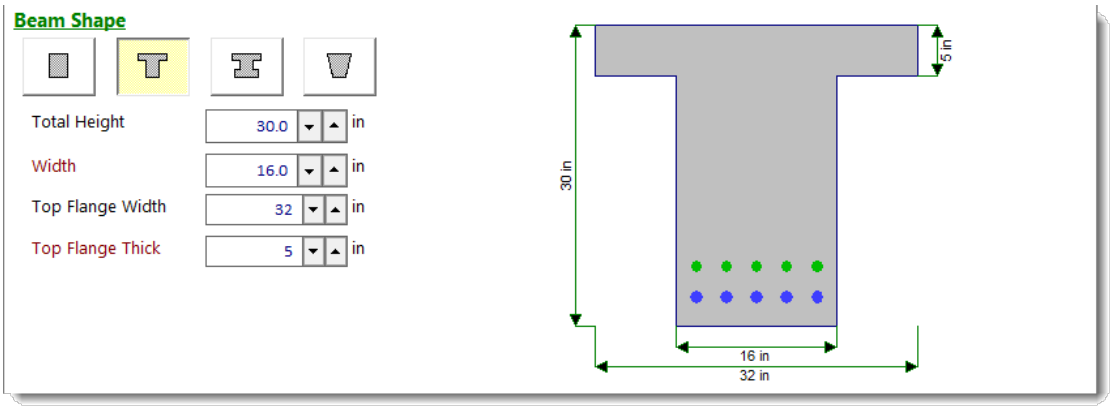
When a concrete beam is chosen, a new tab named **Span Data** will be added to the user interface (see below). This tab is where you specify the beam cross section and reinforcing. This tab will only be shown when the material is set to concrete.

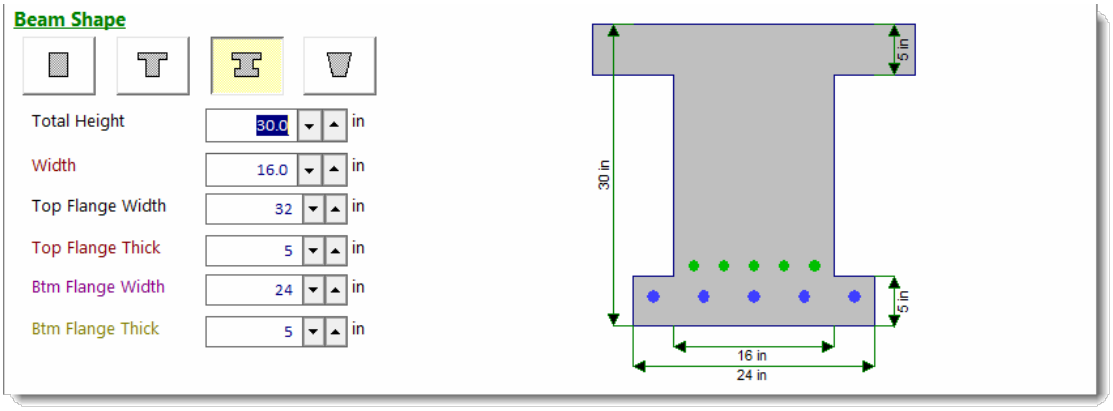


© 2026 ENERCALC, LLC

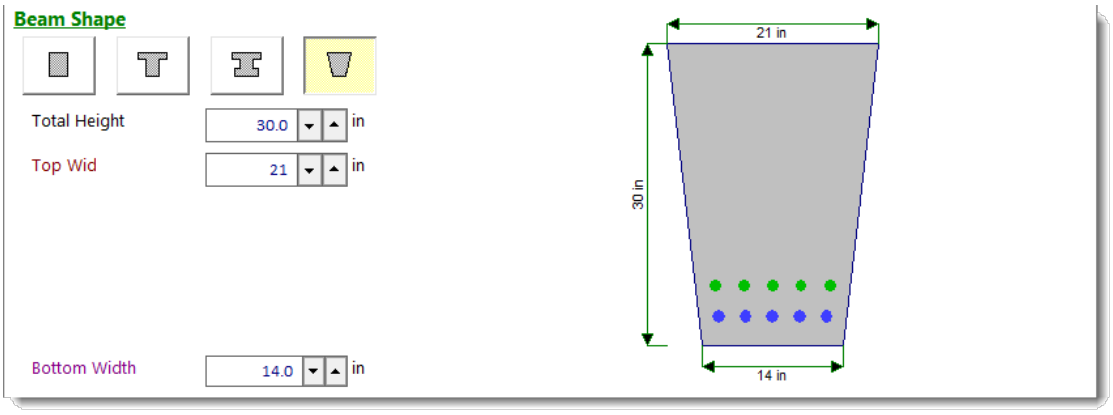
Beam Shape: You can select between 4 beam shapes:







© 2026 ENERCALC, LLC



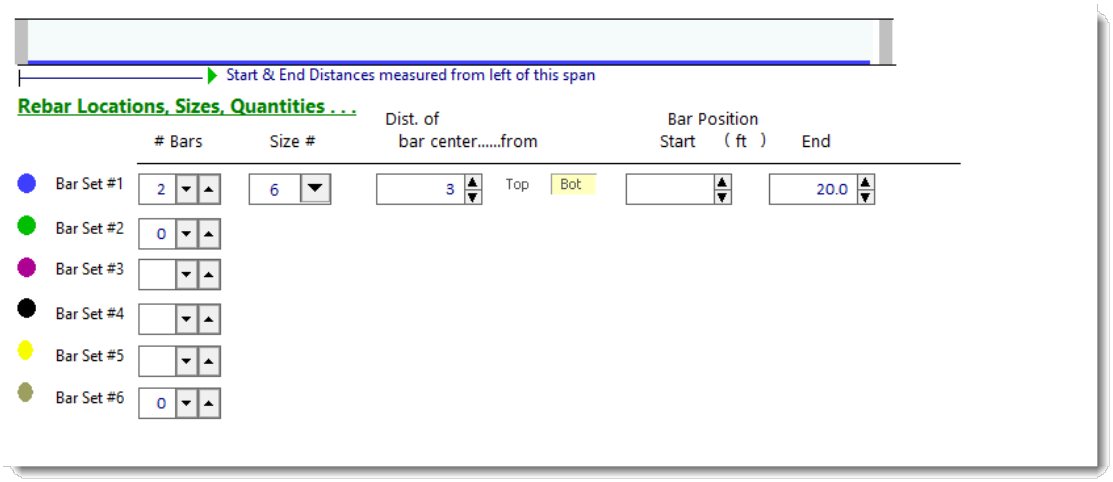
On the right side of the tab you can specify up to 6 bars set (quantity, size, vertical location and start/stop endpoints).
Each bar set is referenced on the sketch with a color shown as a dot to the left of the set description.

The column highlighted in light blue titled "Dist of bar center....from...." is how you set the vertical position of the bars set in the beam.

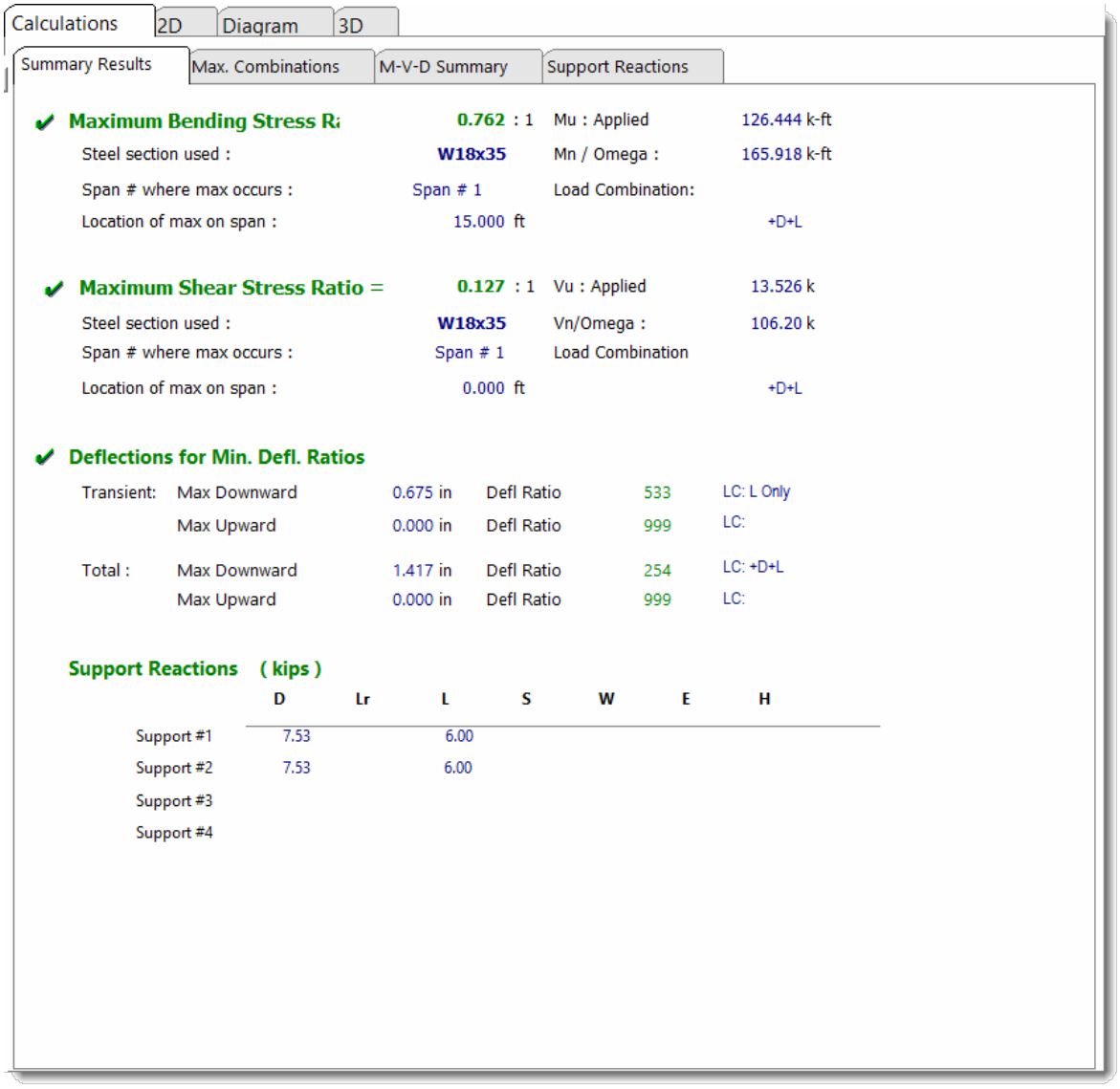
When you look at the top one you can read it as "The top bar set is 3 inches from the bottom of the beam". Note that the module will know whether the bars are in tension or compression and handle the calculations properly.

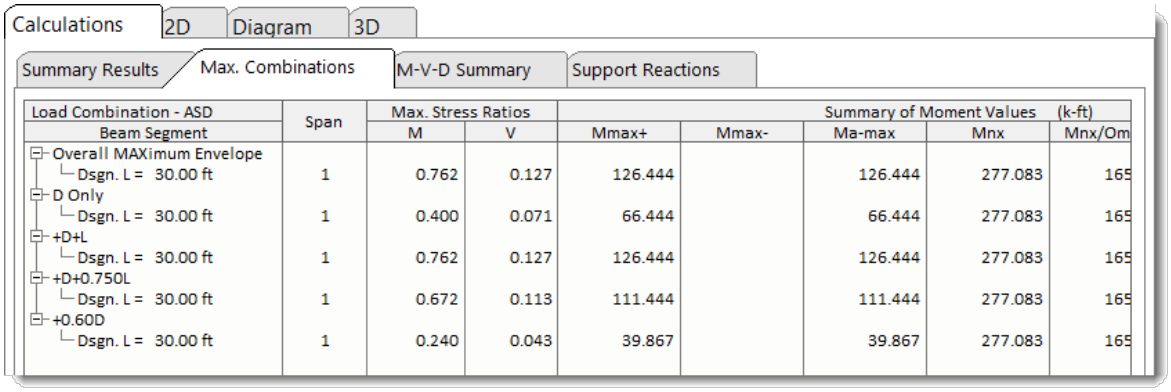
The item named Bar Position This Span defines the starting and ending location of the bar's ends with respect to this span's left support. Using these starting and ending locations you can fine tune the bar layout and end cutoffs.

The data in the screen capture below shows that bar set #1 runs from the left end (0.0 ft) to 20.0 ft from the left support.



Results Tabs





© 2026 ENERCALC, LLC

Calculations | 2D | Diagram | 3D

Summary Results | Max. Combinations | M-V-D Summary | Support Reactions | Compress Incremental Results

ASD Shears & Moments | Service Load Deflections

Load Combination - ASD Span Name	Span #	Distance (ft)	Shear : Va (k)	Moment : Ma (k-ft)	Lu (ft)
Overall MAXimum Envelope					
D Only					
+D+L					
span 1	1		13.526		upport Brac
span 1	1	0.100	13.502	1.351	Lu = 30.00
span 1	1	0.200	13.479	2.700	Lu = 30.00
span 1	1	0.300	13.455	4.047	Lu = 30.00
span 1	1	0.400	13.432	5.392	Lu = 30.00
span 1	1	0.500	13.408	6.734	Lu = 30.00
span 1	1	0.600	13.385	8.073	Lu = 30.00
span 1	1	0.700	13.361	9.411	Lu = 30.00
span 1	1	0.800	13.338	10.746	Lu = 30.00
span 1	1	0.900	13.314	12.078	Lu = 30.00
span 1	1	1.000	13.291	13.408	Lu = 30.00

Calculations | 2D | Diagram | 3D

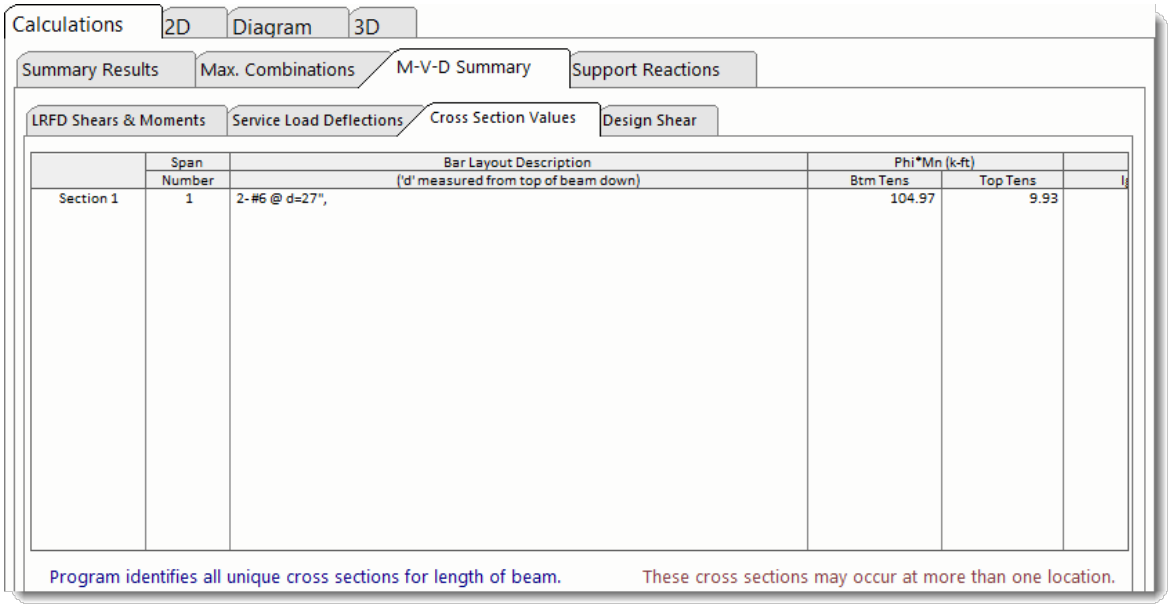
Summary Results | Max. Combinations | M-V-D Summary | Support Reactions | Compress Incremental Results

ASD Shears & Moments | Service Load Deflections

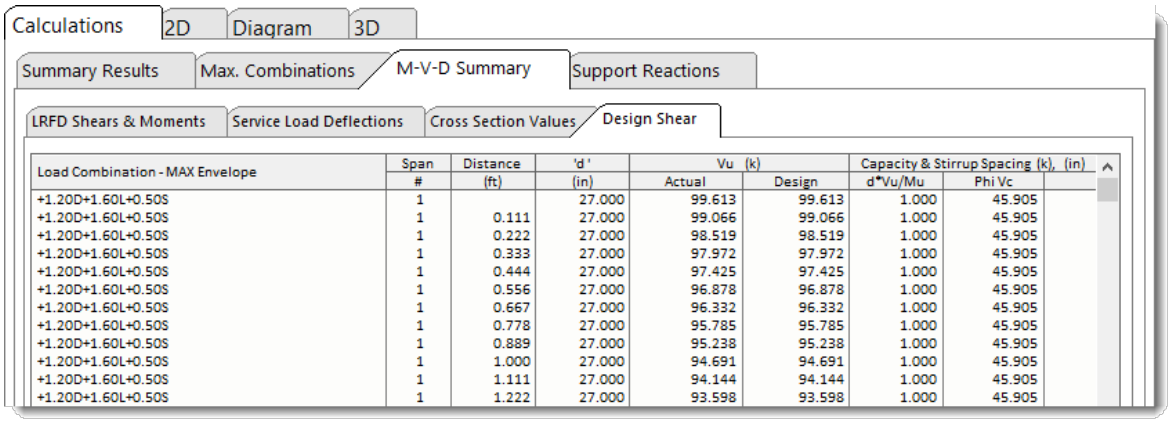
Load Combination - Service Span Name	Span #	Distance (ft)	Deflection (in)	 Ratio	I-effective (in^4)	eflCheckTyp
Overall MAXimum Envelope						
D Only	1					Total
+D+L	1					Total
span 1	1				510.000	Total
span 1	1	0.100	0.015	24,368	510.000	Total
span 1	1	0.200	0.030	12,185	510.000	Total
span 1	1	0.300	0.044	8,124	510.000	Total
span 1	1	0.400	0.059	6,093	510.000	Total
span 1	1	0.500	0.074	4,875	510.000	Total
span 1	1	0.600	0.089	4,063	510.000	Total
span 1	1	0.700	0.103	3,484	510.000	Total
span 1	1	0.800	0.118	3,049	510.000	Total

© 2026 ENERCALC, LLC

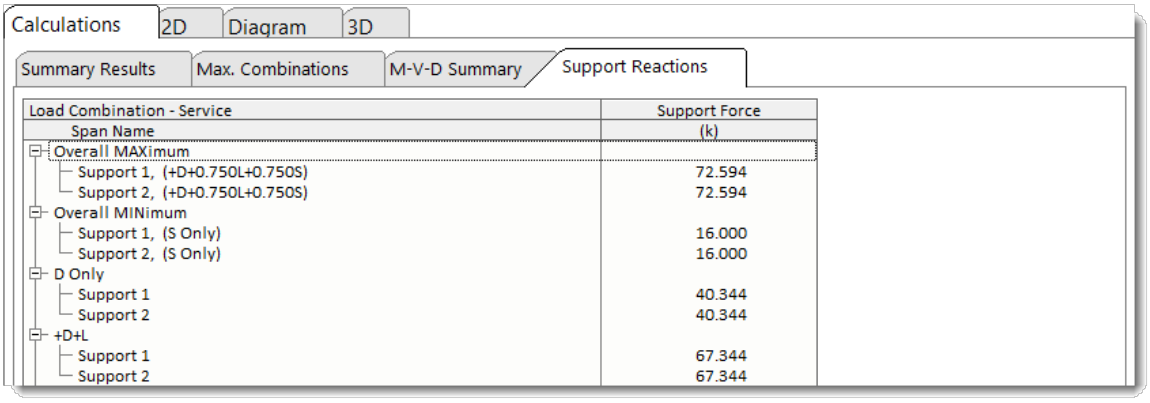
And for concrete beams these two additional sub-tabs are also visible:



	Span Number	Bar Layout Description ('d' measured from top of beam down)	Phi*Mn (k-ft) Btm Tens	Top Tens
Section 1	1	2-#6 @ d=27",	104.97	9.93



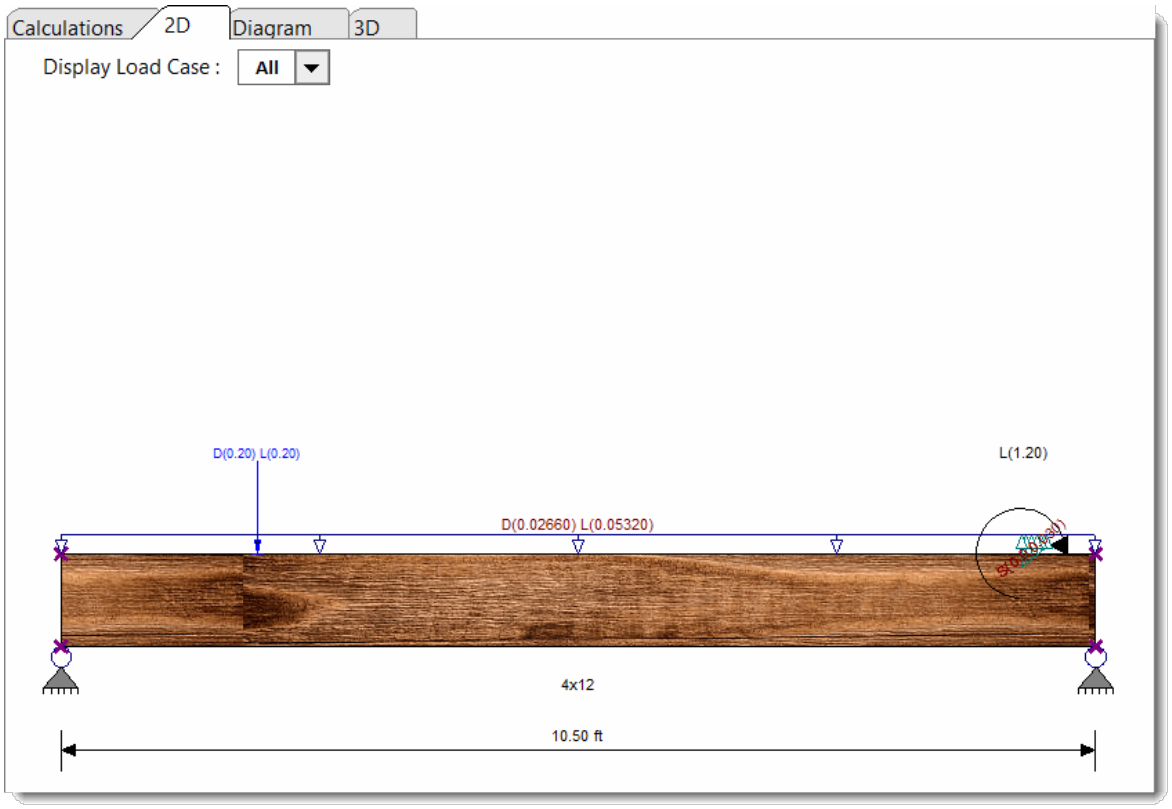
Load Combination - MAX Envelope	Span #	Distance (ft)	'd' (in)	Vu (k) Actual	Vu (k) Design	Capacity & Stirrup Spacing (k), (in) d*Vu/Mu	Capacity & Stirrup Spacing (k), (in) Phi Vc
+1.20D+1.60L+0.50S	1		27.000	99.613	99.613	1.000	45.905
+1.20D+1.60L+0.50S	1	0.111	27.000	99.066	99.066	1.000	45.905
+1.20D+1.60L+0.50S	1	0.222	27.000	98.519	98.519	1.000	45.905
+1.20D+1.60L+0.50S	1	0.333	27.000	97.972	97.972	1.000	45.905
+1.20D+1.60L+0.50S	1	0.444	27.000	97.425	97.425	1.000	45.905
+1.20D+1.60L+0.50S	1	0.556	27.000	96.878	96.878	1.000	45.905
+1.20D+1.60L+0.50S	1	0.667	27.000	96.332	96.332	1.000	45.905
+1.20D+1.60L+0.50S	1	0.778	27.000	95.785	95.785	1.000	45.905
+1.20D+1.60L+0.50S	1	0.889	27.000	95.238	95.238	1.000	45.905
+1.20D+1.60L+0.50S	1	1.000	27.000	94.691	94.691	1.000	45.905
+1.20D+1.60L+0.50S	1	1.111	27.000	94.144	94.144	1.000	45.905
+1.20D+1.60L+0.50S	1	1.222	27.000	93.598	93.598	1.000	45.905



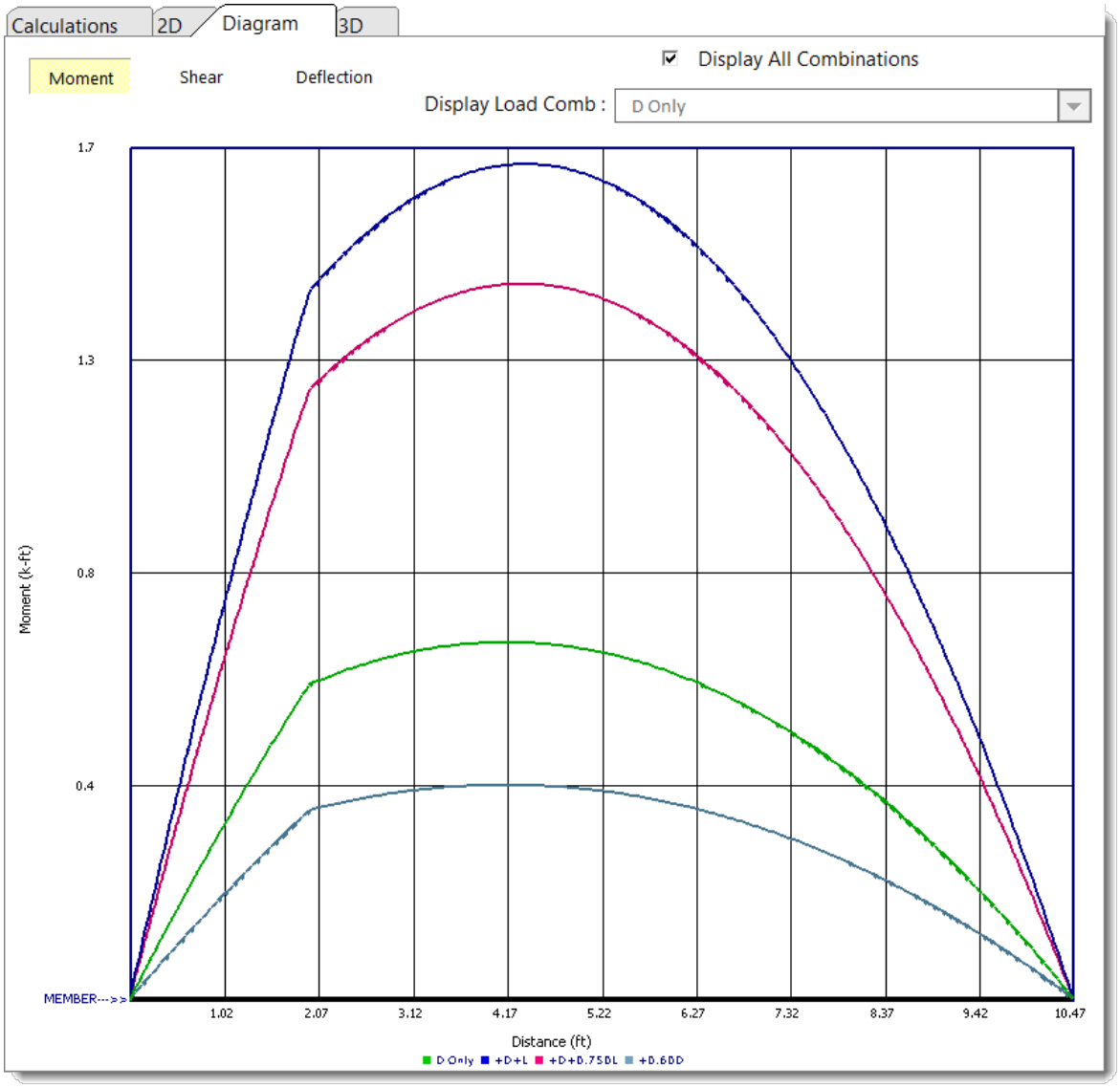
Load Combination - Service / Span Name	Support Force (k)
Overall MAXimum	
Support 1, (+D+0.750L+0.750S)	72.594
Support 2, (+D+0.750L+0.750S)	72.594
Overall MINimum	
Support 1, (S Only)	16.000
Support 2, (S Only)	16.000
D Only	
Support 1	40.344
Support 2	40.344
+D+L	
Support 1	67.344
Support 2	67.344

© 2026 ENERCALC, LLC

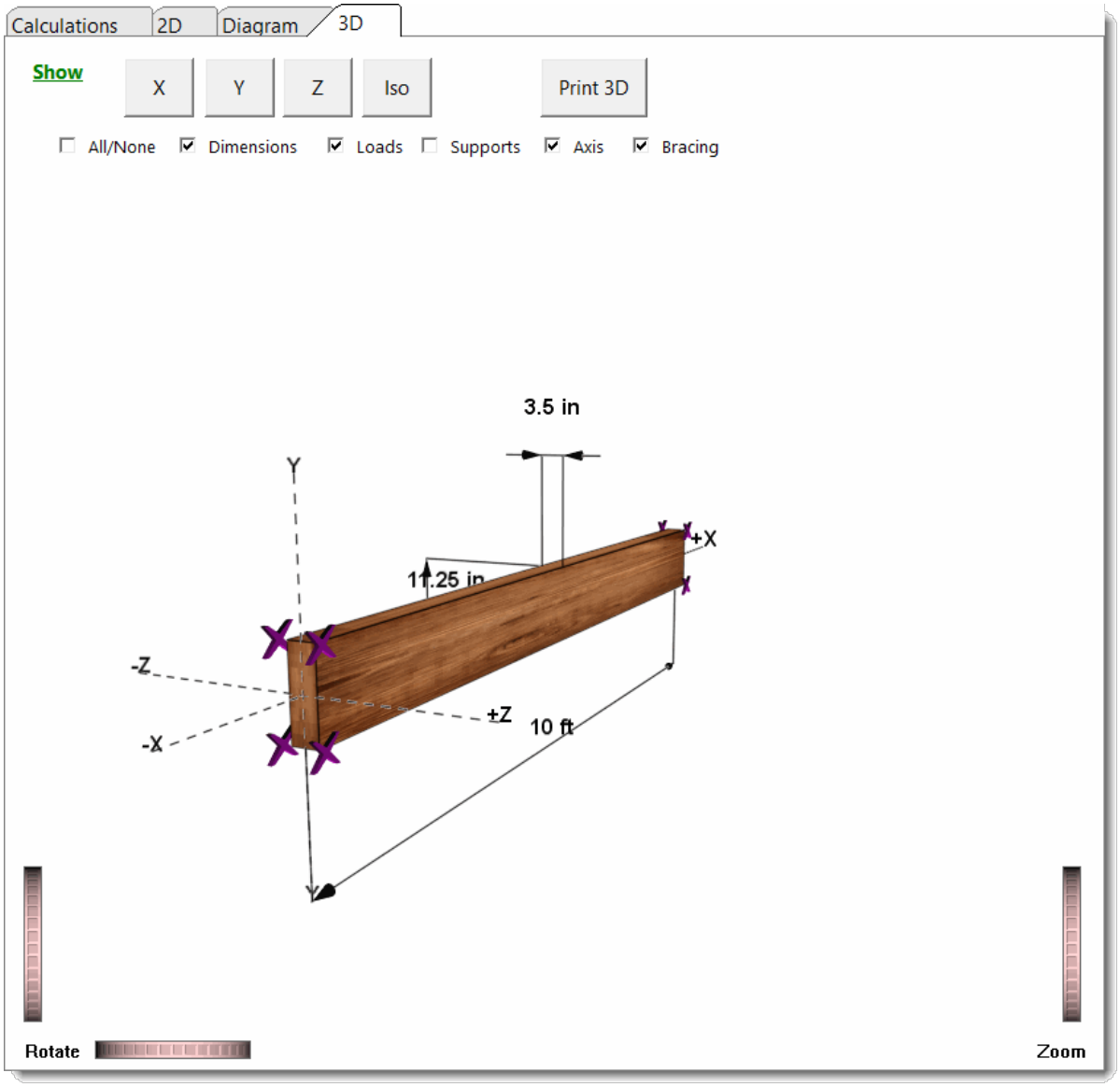
2D


Calculations
2D
Diagram
3D
Display Load Case :
All
D(0.20) L(0.20)
L(1.20)
D(0.02660) L(0.05320)
4x12
10.50 ft

Diagram


Calculations
2D
Diagram
3D
Moment
Shear
Deflection
Display All Combinations
Display Load Comb :
D Only
1.7
1.3
0.8
0.4
Moment (k-ft)
MEMBER--->>
1.02
2.07
3.12
4.17
5.22
6.27
7.32
8.37
9.42
10.47
Distance (ft)
D Only
+D+L
+D+0.75DL
+0.6DD

3D Rendering



13.4.2 Steel Beam

Last revised: Tuesday, February 3, 2026 at 06:12 PM

Verification Example

In this section, for each input tab we will review only the items that are unique to the STEEL material type.

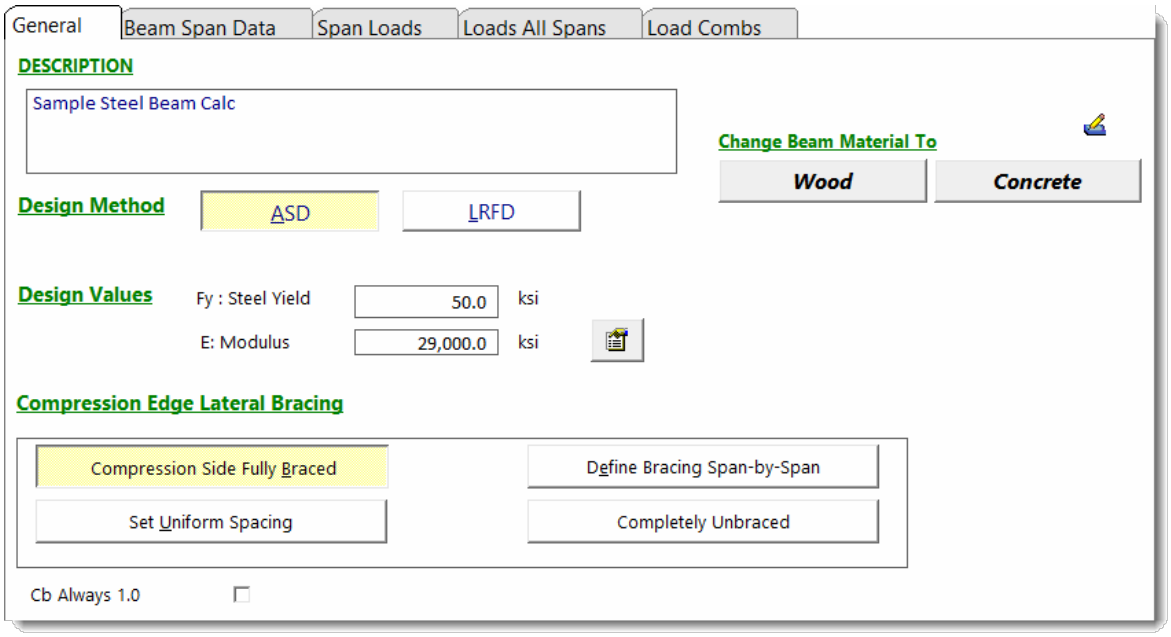
For general information on the typical data input for all beams see the Beams 684 topic.

This module offers complete design of single and multi-span steel members. Among its capabilities are:

- Single or multi-span (up to 8-span) beams.
- End fixity can be pinned, fixed, free or a combination.
- Steel member analysis and design are according to AISC 360.
- ASD or LRFD design methods can be selected.
- A complete steel section database is provided.
- Unbraced compression edge lengths can be specified in a variety of ways.
- Automatic member selection is provided.

General

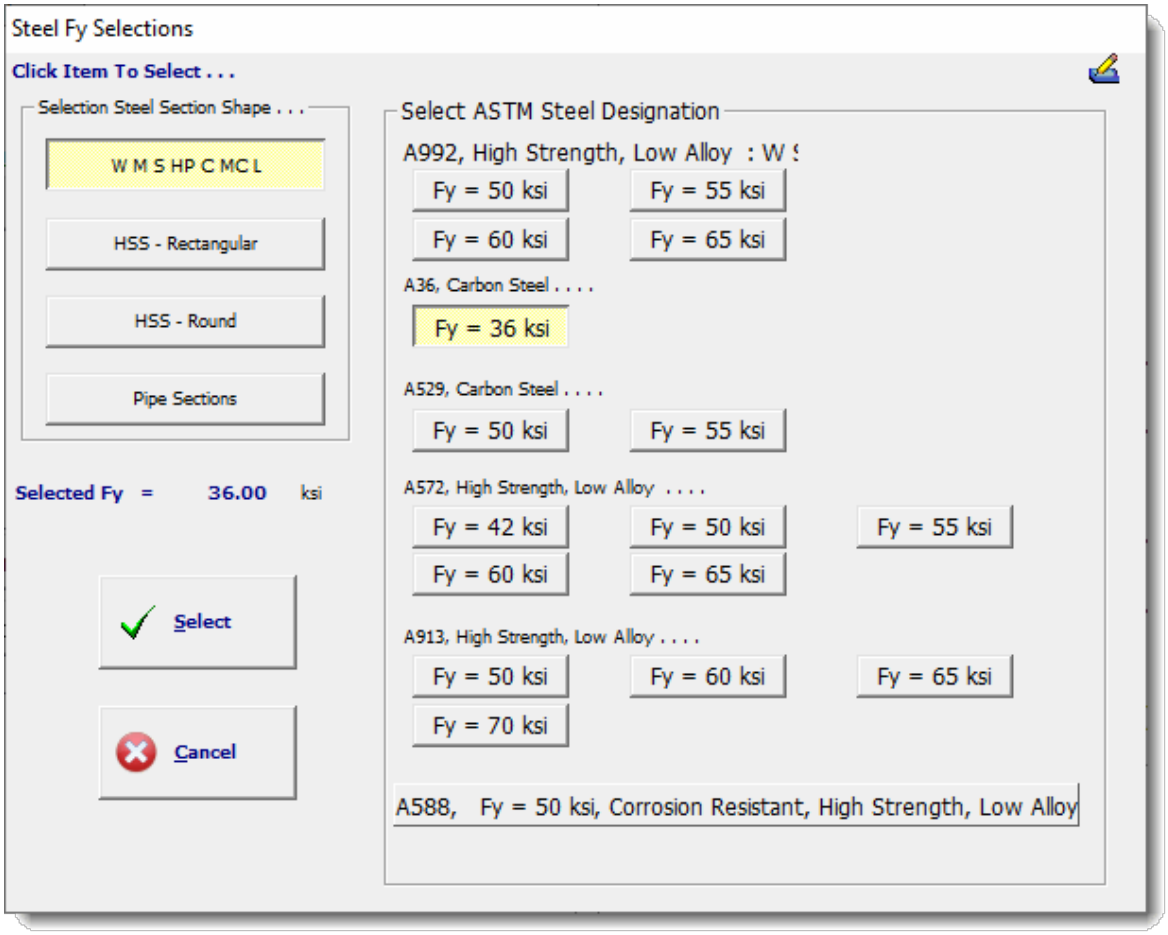
When steel is used, this tab includes input fields to set the values for yield stress and elastic modulus, as well as an option to set the Cb factor equal to 1, as shown in the screen capture below:



If Cb is not forced to 1, the program automatically calculates Cb based on the moment and direction of curvature at various locations along the beam.

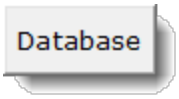
© 2026 ENERCALC, LLC

For convenience the module includes a built-in steel database. Click the button to the right of the Fy entry and you will see the following table, which offers a number of commonly used steel grades:

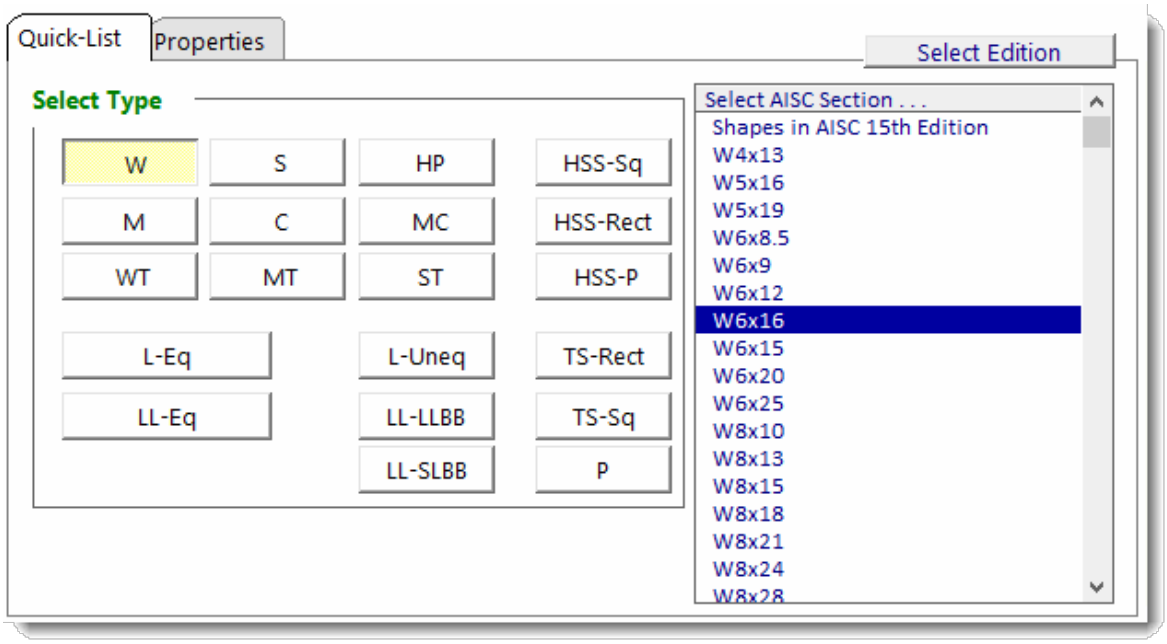


© 2026 ENERCALC, LLC

Beam Span Data

When steel is selected, the beam size selection is specifically for steel. You can select a section in 4 ways:
1 - Simply type the AISC name into the Steel Section Name field and press [**Tab**].
2 - Click the 

button and select from the built-in AISC section database.
3 - Click the [**Auto Select**] button to have the module evaluate steel sections from the database according to your criteria.
4 - Select a steel section from the Quick-List tab as shown below:



Span Loads

No differences from other materials.

Loads All Spans

No differences from other materials.

Load Combinations

No differences from other materials.

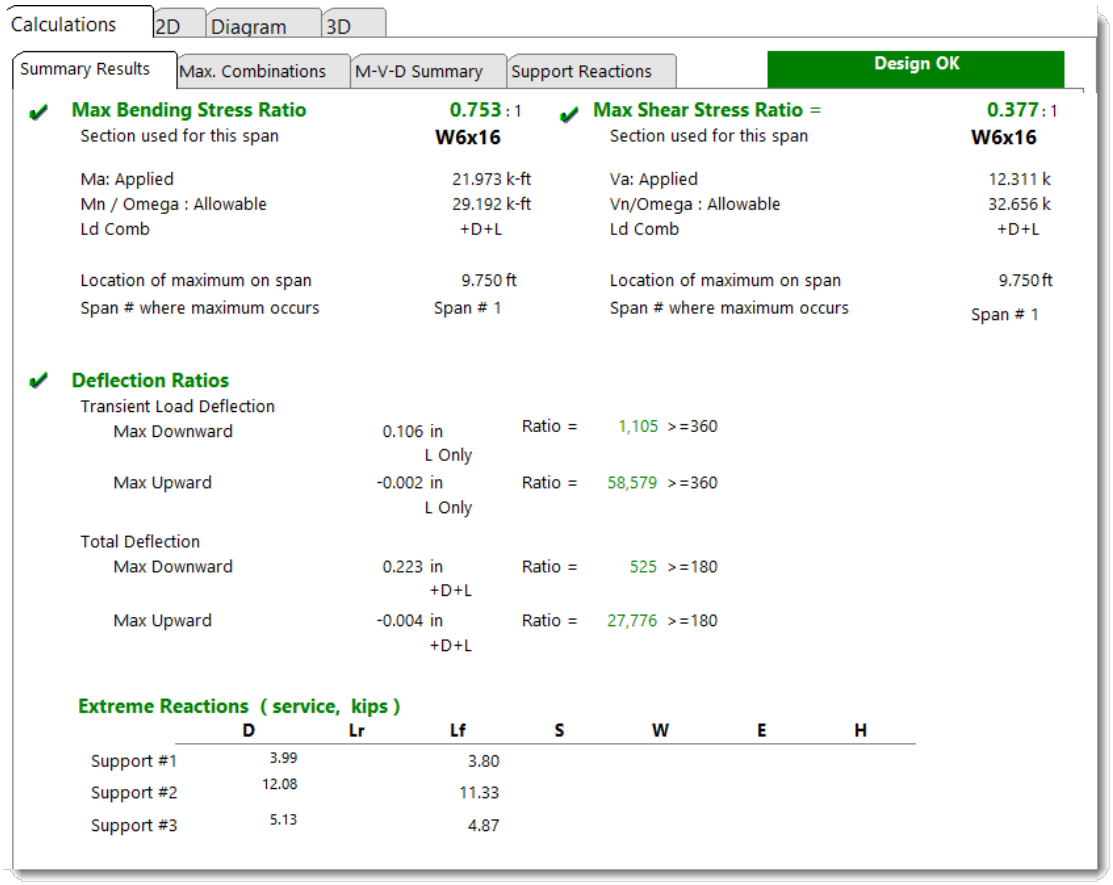
Results Tab

This set of tabs provides detailed results for the current calculation. The tabs in the upper right-hand corner of the screen allow you to select the major areas available for review: Calculations, 2D Sketch, Diagram and 3D Rendering.

© 2026 ENERCALC, LLC

The Calculations tab offers the following results options:

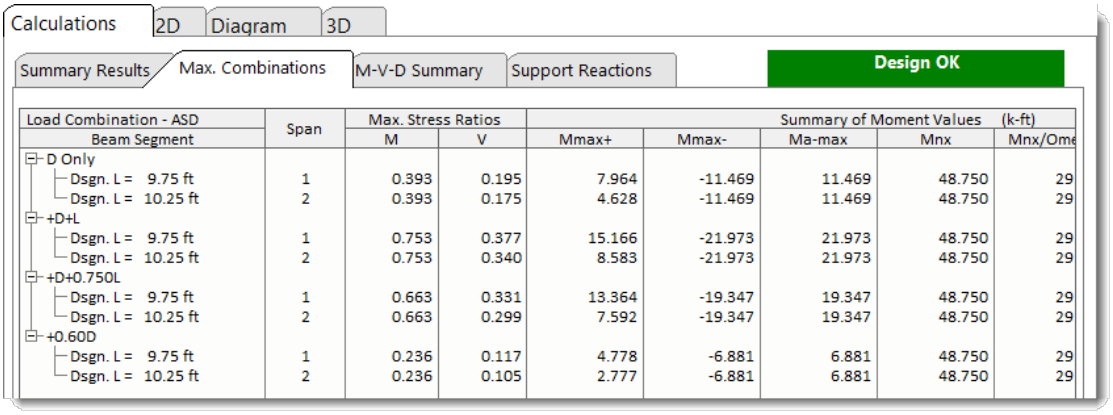
Summary Results provides details for shear, moment and deflection for the governing load combinations.



© 2026 ENERCALC, LLC

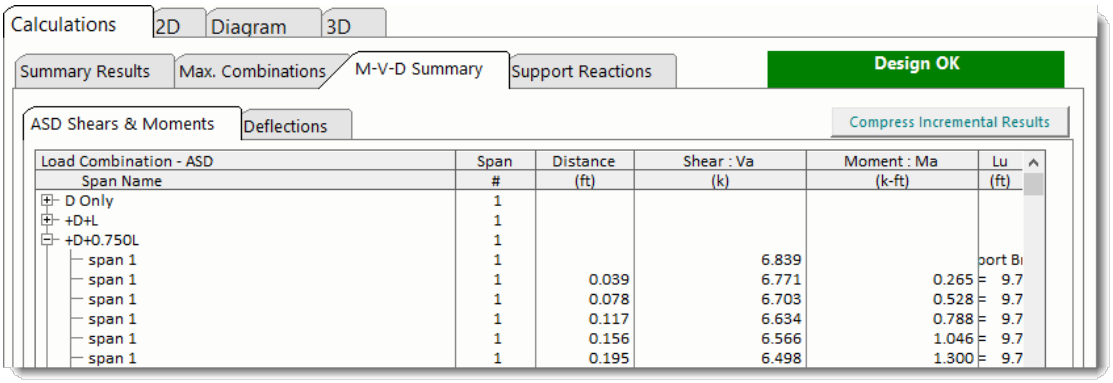
Max. Combinations provides detailed results for each beam segment for each load combination. The leftmost column lists the load combinations and the unbraced length being considered.

These results are a consolidation of the highly detailed incremental results on the M-V-D Summary tab.



Load Combination - ASD / Beam Segment	Span	Max. Stress Ratios M	Max. Stress Ratios V	Summary of Moment Values (k-ft) Mmax+	Mmax-	Ma-max	Mnx	Mnx/Ome
D Only								
Dsgn. L = 9.75 ft	1	0.393	0.195	7.964	-11.469	11.469	48.750	29
Dsgn. L = 10.25 ft	2	0.393	0.175	4.628	-11.469	11.469	48.750	29
+D+L								
Dsgn. L = 9.75 ft	1	0.753	0.377	15.166	-21.973	21.973	48.750	29
Dsgn. L = 10.25 ft	2	0.753	0.340	8.583	-21.973	21.973	48.750	29
+D+0.750L								
Dsgn. L = 9.75 ft	1	0.663	0.331	13.364	-19.347	19.347	48.750	29
Dsgn. L = 10.25 ft	2	0.663	0.299	7.592	-19.347	19.347	48.750	29
+0.60D								
Dsgn. L = 9.75 ft	1	0.236	0.117	4.778	-6.881	6.881	48.750	29
Dsgn. L = 10.25 ft	2	0.236	0.105	2.777	-6.881	6.881	48.750	29

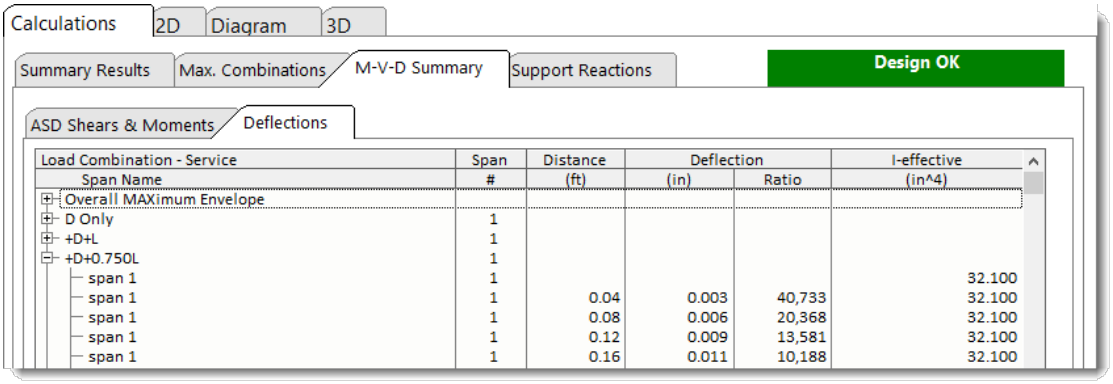
M-V-D Summary - Shears & Moments shows highly detailed moment and shear information for each beam and for each load combination. For multi-span beams using Automatic Unbalanced Live Load Placement there may be thousands of lines of results.



Load Combination - ASD / Span Name	Span #	Distance (ft)	Shear : Va (k)	Moment : Ma (k-ft)	Lu (ft)
D Only	1				
+D+L	1				
+D+0.750L	1				
span 1	1		6.839		port B
span 1	1	0.039	6.771	0.265	= 9.7
span 1	1	0.078	6.703	0.528	= 9.7
span 1	1	0.117	6.634	0.788	= 9.7
span 1	1	0.156	6.566	1.046	= 9.7
span 1	1	0.195	6.498	1.300	= 9.7

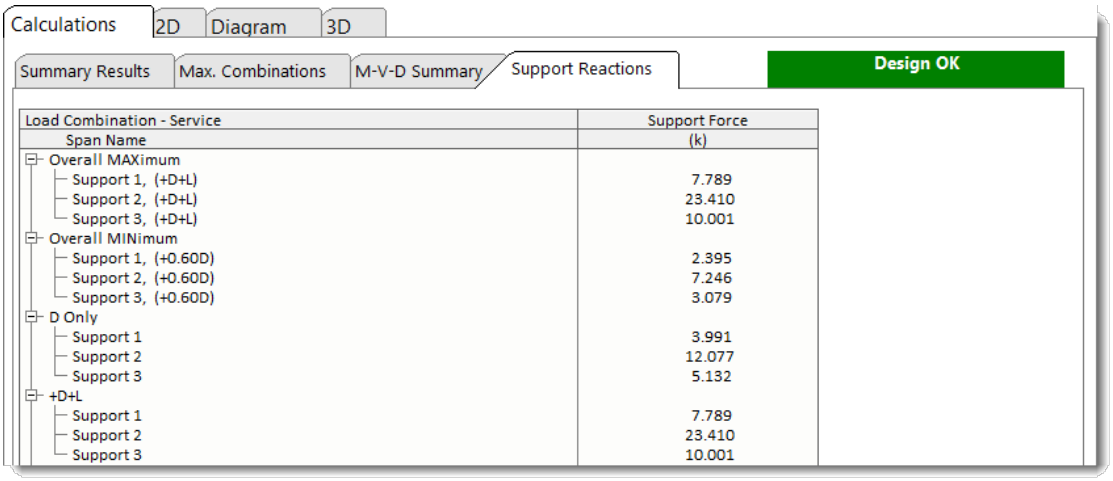
© 2026 ENERCALC, LLC

M-V-D Summary - Deflections shows highly detailed deflection results for all load combinations.



Load Combination - Service Span Name	Span #	Distance (ft)	Deflection (in)	 Ratio	I-effective (in^4)
⊞ Overall MAXimum Envelope					
⊞ D Only	1				
⊞ +D+L	1				
⊟ +D+0.750L	1				
span 1	1				32.100
span 1	1	0.04	0.003	40,733	32.100
span 1	1	0.08	0.006	20,368	32.100
span 1	1	0.12	0.009	13,581	32.100
span 1	1	0.16	0.011	10,188	32.100

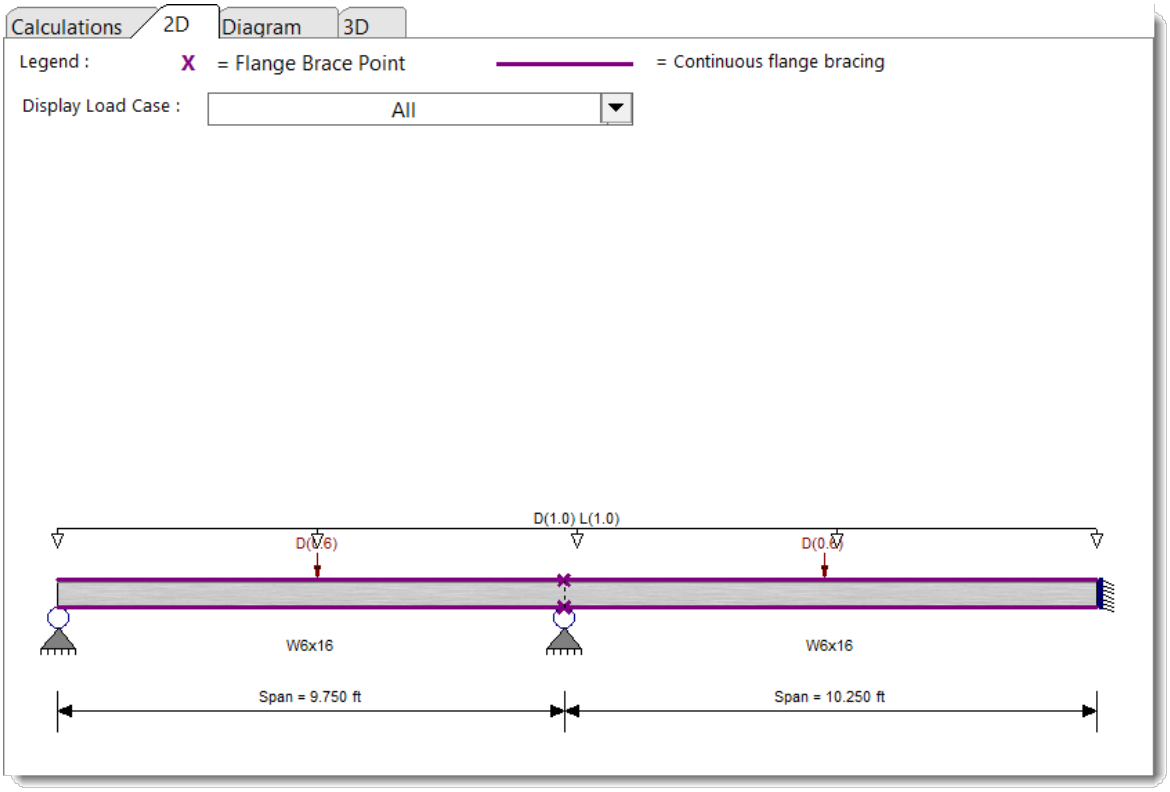
Support Reactions shows reactions for each support for each load condition.



Load Combination - Service Span Name	Support Force (k)
⊟ Overall MAXimum	
Support 1, (+D+L)	7.789
Support 2, (+D+L)	23.410
Support 3, (+D+L)	10.001
⊟ Overall MINimum	
Support 1, (+0.60D)	2.395
Support 2, (+0.60D)	7.246
Support 3, (+0.60D)	3.079
⊟ D Only	
Support 1	3.991
Support 2	12.077
Support 3	5.132
⊟ +D+L	
Support 1	7.789
Support 2	23.410
Support 3	10.001

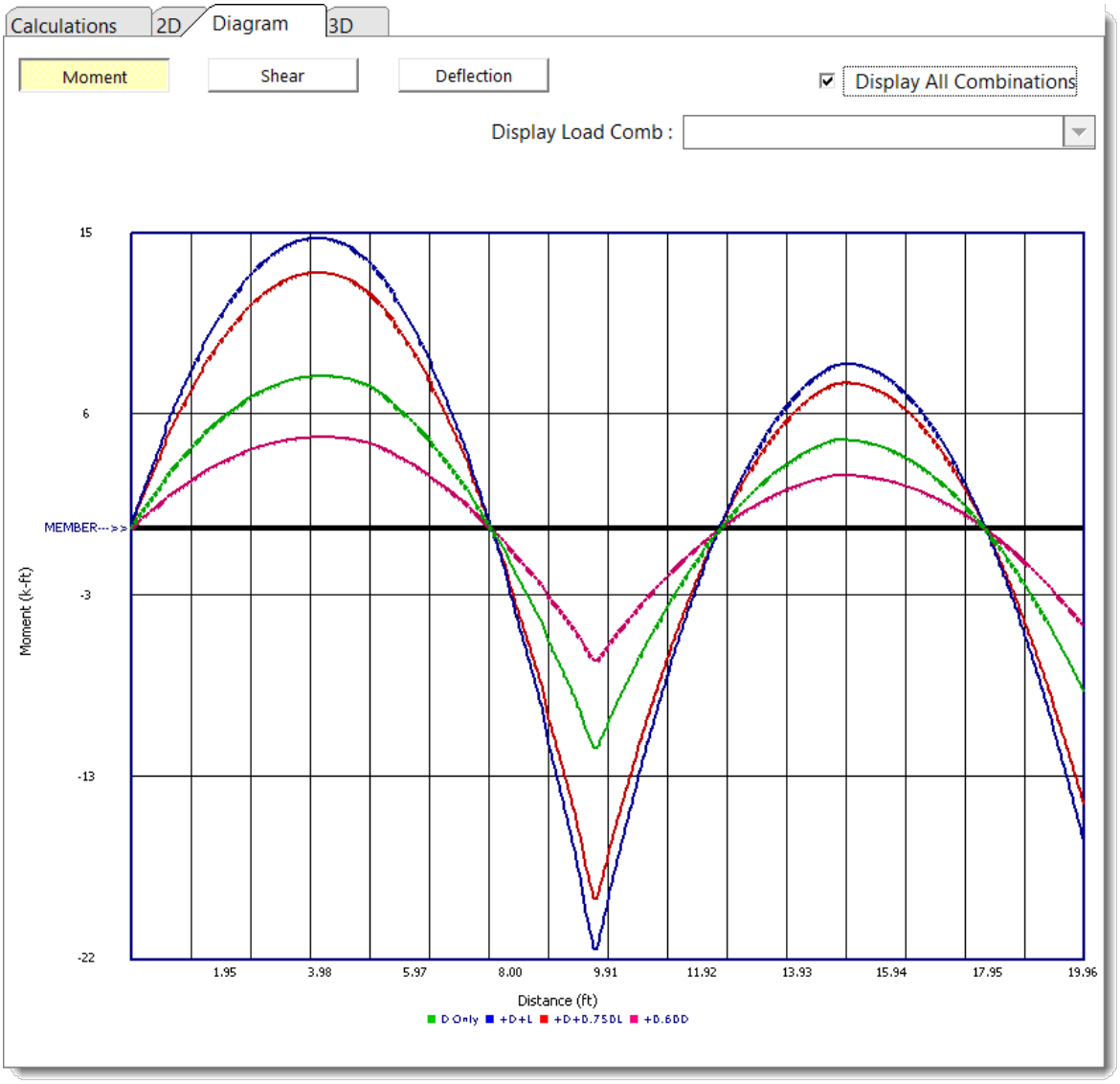
© 2026 ENERCALC, LLC

The 2D Sketch tab provides a graphic representation of the beam currently being designed:

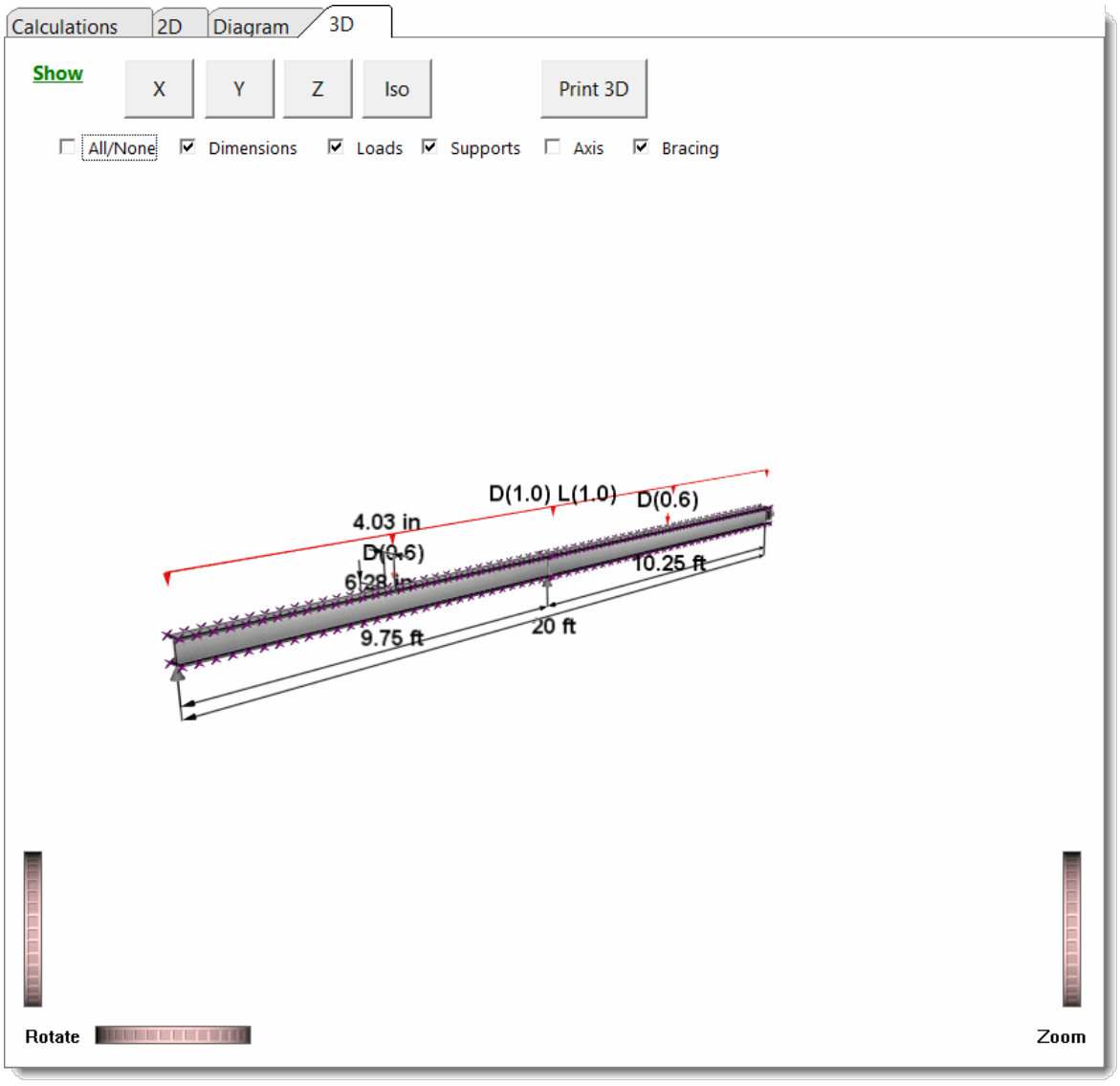


© 2026 ENERCALC, LLC

The Diagram tab offers the ability to view shear, moment, and deflection diagrams for selected load combinations:



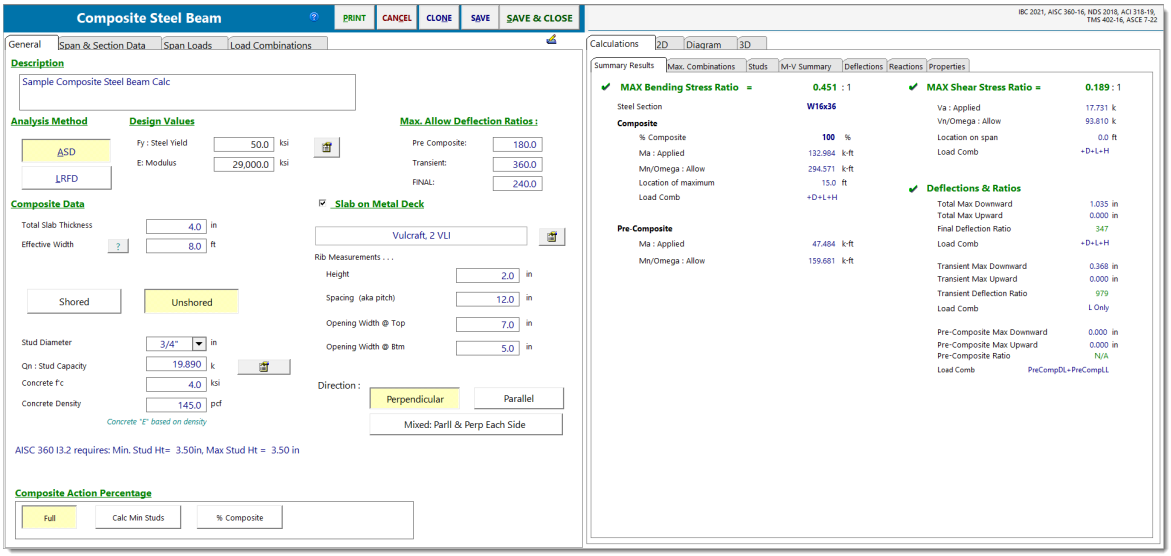
The 3D Rendering offers a 3D view of the beam with many display controls:



13.4.3 Composite Steel Beam

Last revised: Tuesday, February 3, 2026 at 06:12 PM

This module provides analysis and design of AISC steel sections acting compositely with a concrete slab that is continuously connected to the compression flange of the beam with suitable shear connectors.



Features of the module include:

- The concrete slab can be either full depth or cast over formed steel decking, with rib orientation perpendicular or parallel to the beam.
- Stud capacity can be calculated by the module using standard AISC procedure.
- Normal or lightweight concrete may be used for both strength and deflection calculations.
- Both shored and unshored construction techniques can be analyzed by the module.
- ASD or LRFD design methods can be chosen.
- Flexible specification of shear studs is available.
- Very flexible loading specification, including the ability to specify construction loads (applied to pre-composite checks only), loads that apply to both the pre-composite and post-composite checks (always applied), and loads that are only applicable to the post-composite checks (applied after curing).
- Extensive load combination capability.
- Can use many sections from the AISC databases.

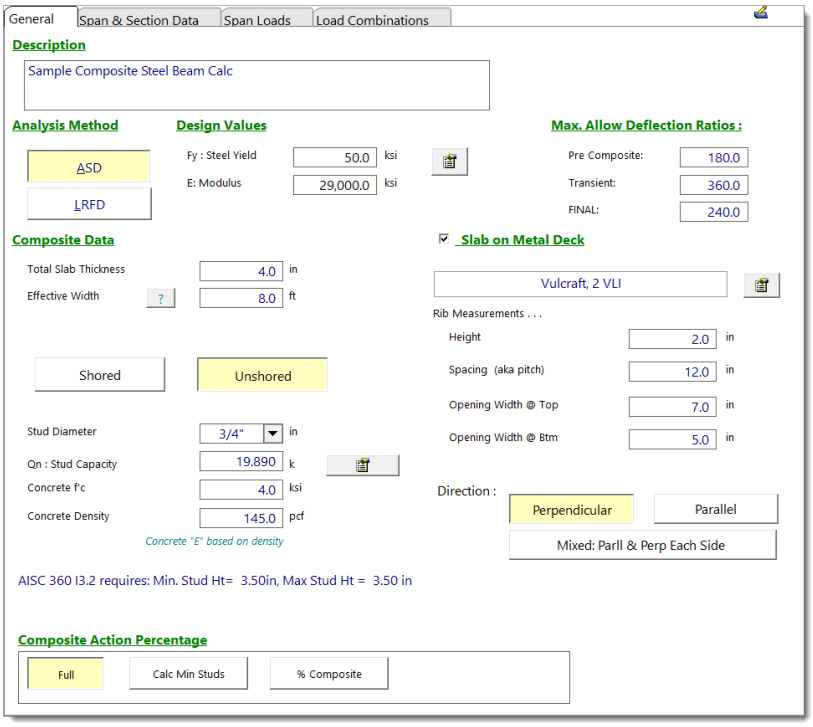
General

This tab gathers all input except for the beam section size and loads.

The Analysis Method category offers an option of ASD or LRFD methods.

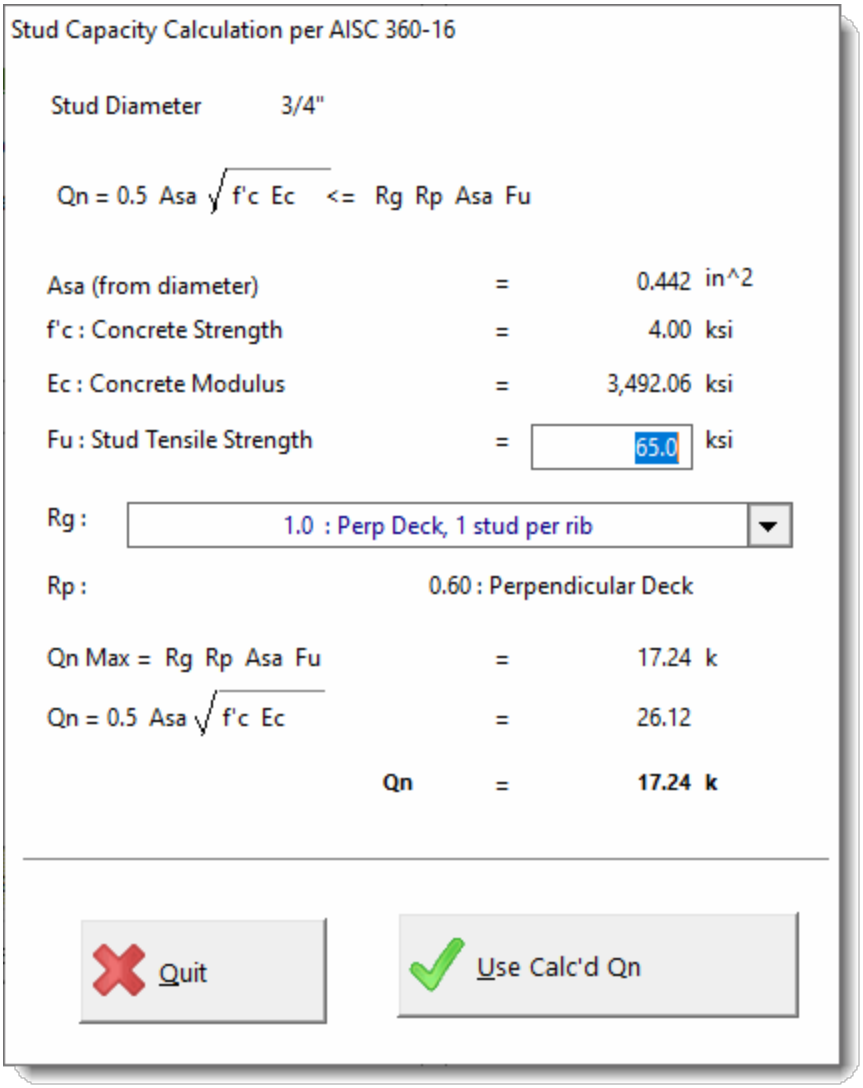
© 2026 ENERCALC, LLC

The Composite Data category offers all of the necessary input fields to completely specify the slab, its effective width, stud information and metal deck specification (if applicable).



© 2026 ENERCALC, LLC

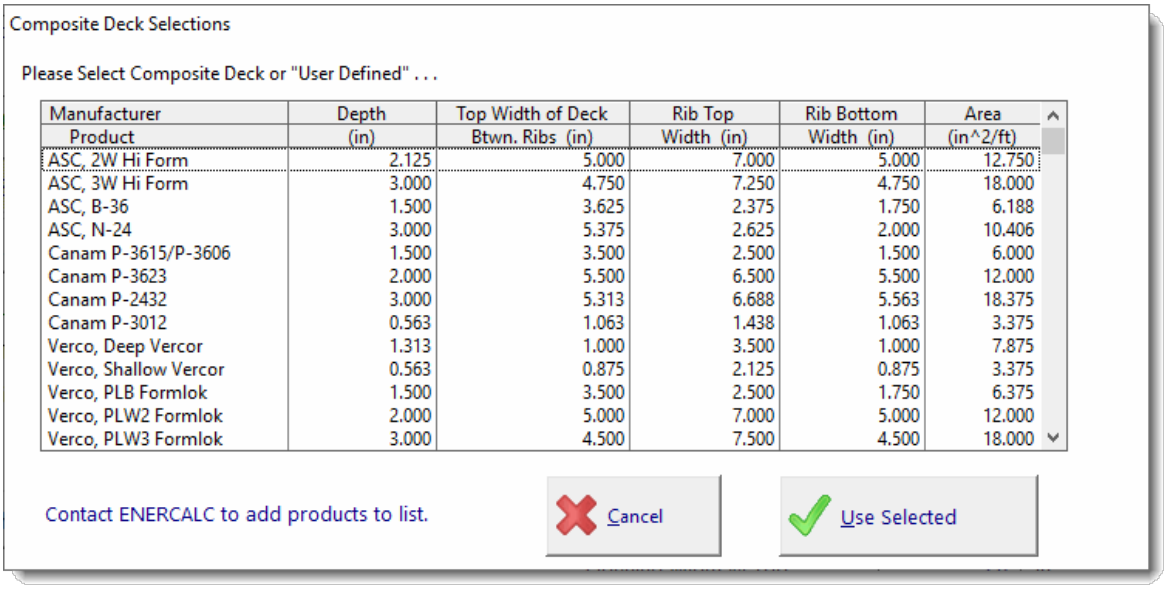
For Qn you can enter the value directly or click the button to display the stud capacity calculation dialog:



When you click the checkbox for Slab on Metal Deck, the associated input fields appear to allow the selection of a manufactured deck product, or the manual specification of the required deck properties.

© 2026 ENERCALC, LLC

Click the button to display the metal deck selection window as shown below, or simply enter the deck cross section data in the dimension input fields:



Partial Composite Action

This category provides three ways to have the module calculate stud requirements.

Full Composite tells the module to use the number of studs necessary to provide full Vh shear resistance for the slab to beam connection.

Specify % Composite allows the user to enter a percentage of maximum composite action. The module will then determine the number of studs needed for that Vh' and complete the bending capacity calculations.

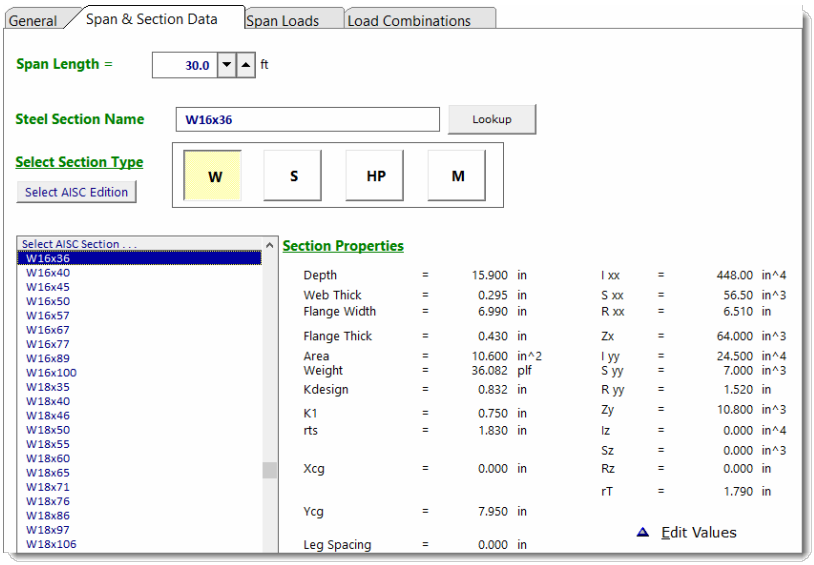
Calc Min Studs tells the module to calculate the minimum number of studs (greater than 25% composite per code recommendations) that will adequately supply the required moment capacity based on the applied moment.

© 2026 ENERCALC, LLC

Span & Section Data

This is basically the same tab used for a normal steel beam except that there is no ability to specify unbraced compression flange lengths. For regions of positive moment, the top flange is considered to be continuously braced by its composite connection to the concrete slab.

Note: Use caution in situations where the applied loading may result in regions of negative moment, because this module considers the bottom flange to be continuously braced.



© 2026 ENERCALC, LLC

Span Loads

This tab is basically the same as the normal load entry for other beam design modules except for two differences:

1 - There are two checkboxes that allow the user to indicate whether the module should automatically calculate and apply the beam self weight and slab self weight.

2 - There are three "Load Application" options that allow the user to specify the sequencing of the specified loads as follows:

- **Applied to Pre-Composite Section & REMOVED before curing:** Dead Load and/or Live Load that will be considered in the pre-composite checks, but not in the post-composite checks.

  Use this load application type to specify temporary Construction Live Load or Construction Dead Load that will be removed before the concrete cures. Do NOT use this load application type to specify permanent loads like the weight of the concrete fill on the metal deck. That type of load should be specified using the next option.

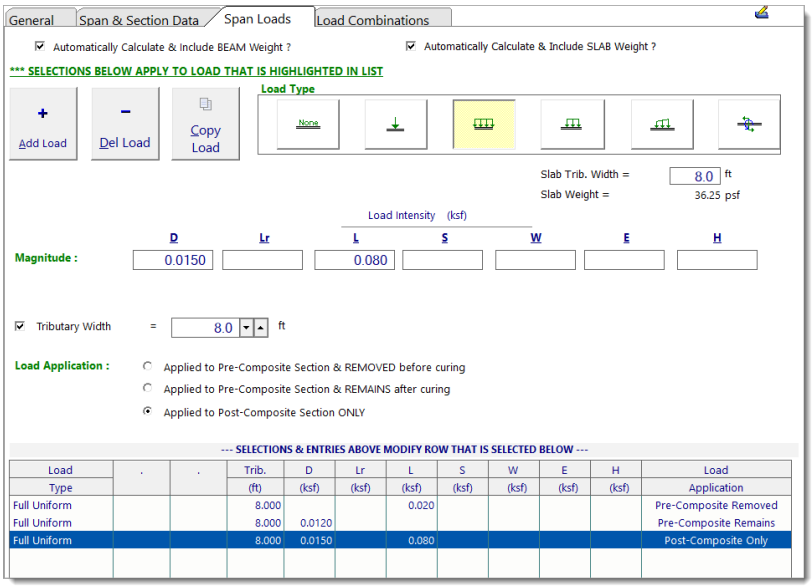
- **Applied to Pre-Composite Section & REMAINS after curing:** Dead Load that will be considered in the pre-composite AND the post-composite checks.

  Use this option for loads that will be in place while the section is pre-composite and will REMAIN in place after composite action is achieved. Examples might include the weight of the concrete fill on the metal deck or the self-weight of the steel beam.

- **Applied to Post-Composite Section ONLY:** Any type of load that will be superimposed on the cured composite section. These will only be considered in the post-composite checks.

  Use this option for loads that will be placed on the structure only after composite action has developed. Some examples might include the weight of architectural floor finishes and occupancy live load.

© 2026 ENERCALC, LLC



Load Combinations

The function of the Load Combinations tab in the Composite Steel Beam module is the same as in other beam modules.

Results Tabs: This set of tabs provides detailed results for the current calculation. The tabs in the upper right-hand corner of the screen allow the user to select the four major areas available for review: Calculations, 2D, Diagram, and 3D.

The Calculations tab offers the following results options:

Summary Results tab presents the maximum/governing ratio values from all of the results presented on the Max. Combinations tab. The module looks for the maximum bending stress ratio and presents the components that are calculated to create that ratio.

For the bending stress item the module reports the Pre-Composite bending ratio, which considers all loads specified to be applied to the beam before curing. The resulting moment is compared to the capacity of the steel section acting alone (non-compositely).

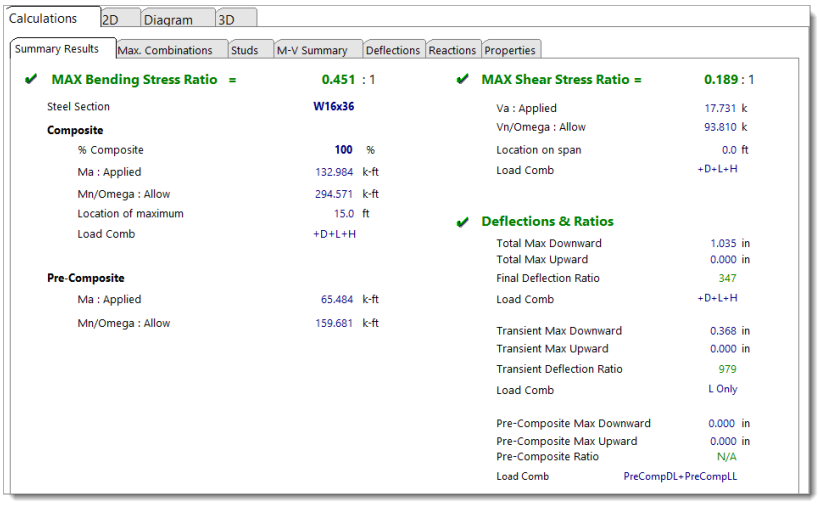
For the item reported as Composite, the module considers all loads specified to be applied to the composite section. That moment is then compared to the full composite section moment capacity for the percentage of shear connection specified.

The results indicate the load combination that creates the governing values along with the span id and location at which the governing ratio was found to occur.

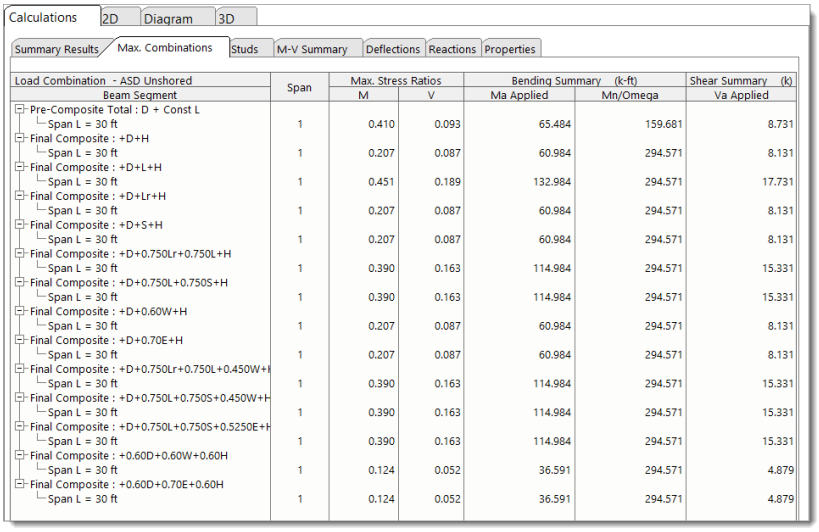
The module calculates the maximum total deflection as the sum of the deflection of the pre-composite steel section resisting all of the loads that are specified as "Pre-Composite Remains" (always applied) plus the deflection of the post-composite section resisting all of the loads that are specified as "Post-Composite Only" (applied after curing).

© 2026 ENERCALC, LLC

Note! The screen shown below, which gives information for Pre-Composite loads (i.e. construction loads), is only for unshored beams. For shored beams, the Pre-Composite portion of moments is not applicable, so it is not shown.



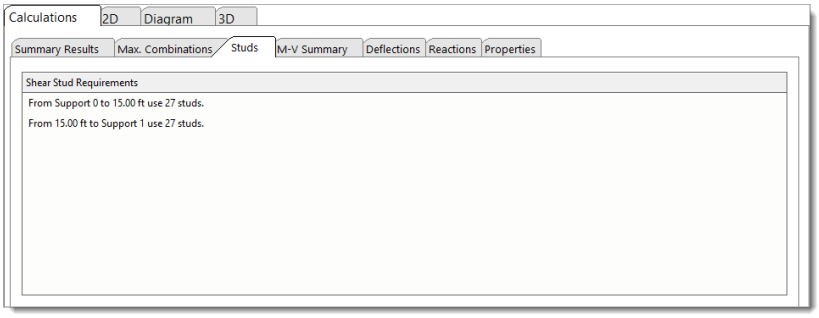
Max Combinations tab presents in more concise detail the bending and shear values for all load combinations.



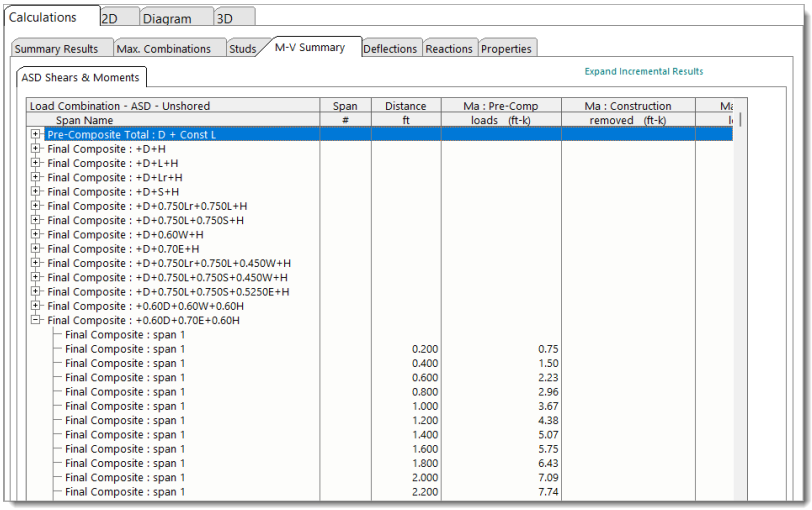
Load Combination - ASD Unshored / Beam Segment	Span	Max. Stress Ratios M	Max. Stress Ratios V	Bending Summary (k-ft) Ma Applied	Bending Summary (k-ft) Mn/Omega	Shear Summary (k) Va Applied
Pre-Composite Total : D + Const L						
Span L = 30 ft	1	0.410	0.093	65.484	159.681	8.731
Final Composite : +D+H						
Span L = 30 ft	1	0.207	0.087	60.984	294.571	8.131
Final Composite : +D+L+H						
Span L = 30 ft	1	0.451	0.189	132.984	294.571	17.731
Final Composite : +D+Lr+H						
Span L = 30 ft	1	0.207	0.087	60.984	294.571	8.131
Final Composite : +D+S+H						
Span L = 30 ft	1	0.207	0.087	60.984	294.571	8.131
Final Composite : +D+0.750Lr+0.750L+H						
Span L = 30 ft	1	0.390	0.163	114.984	294.571	15.331
Final Composite : +D+0.750L+0.750S+H						
Span L = 30 ft	1	0.390	0.163	114.984	294.571	15.331
Final Composite : +D+0.60W+H						
Span L = 30 ft	1	0.207	0.087	60.984	294.571	8.131
Final Composite : +D+0.70E+H						
Span L = 30 ft	1	0.207	0.087	60.984	294.571	8.131
Final Composite : +D+0.750Lr+0.750L+0.450W+H						
Span L = 30 ft	1	0.390	0.163	114.984	294.571	15.331
Final Composite : +D+0.750L+0.750S+0.450W+H						
Span L = 30 ft	1	0.390	0.163	114.984	294.571	15.331
Final Composite : +D+0.750L+0.750S+0.5250E+H						
Span L = 30 ft	1	0.390	0.163	114.984	294.571	15.331
Final Composite : +0.60D+0.60W+0.60H						
Span L = 30 ft	1	0.124	0.052	36.591	294.571	4.879
Final Composite : +0.60D+0.70E+0.60H						
Span L = 30 ft	1	0.124	0.052	36.591	294.571	4.879

© 2026 ENERCALC, LLC

Studs tab presents the shear connector requirements for all span sections. When point loads are present, this chart may list more detailed spacing requirements because of the shear change between applied point loads.



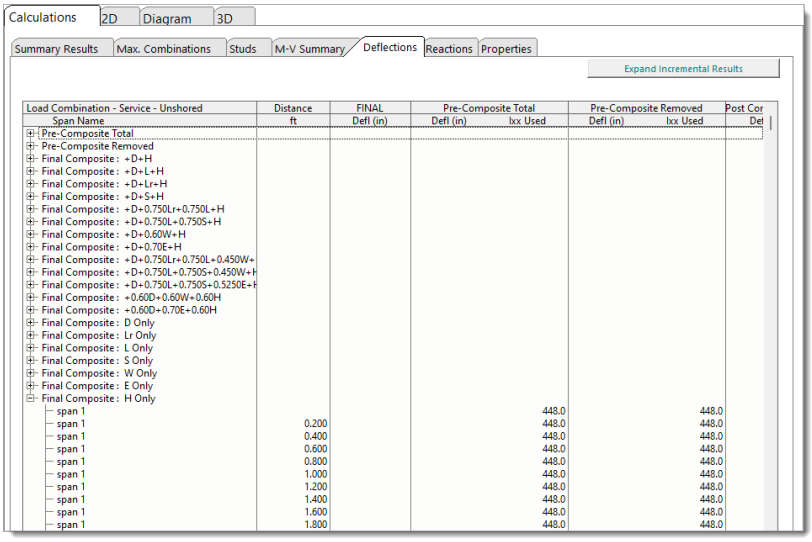
M-V Summary presents shear and moment results. This screen will take on four slightly different forms depending upon the selection of Analysis Method (ASD or LRFD) and shoring (shored or unshored). The example shown below is for an ASD unshored design.



Load Combination - ASD - Unshored Span Name	Span #	Distance ft	Ma : Pre-Comp loads (ft-k)	Ma : Construction removed (ft-k)	Ma
Pre-Composite Total : D + Const L					
Final Composite : +D+H					
Final Composite : +D+L+H					
Final Composite : +D+Lr+H					
Final Composite : +D+S+H					
Final Composite : +D+0.750Lr+0.750L+H					
Final Composite : +D+0.750L+0.750S+H					
Final Composite : +D+0.60W+H					
Final Composite : +D+0.70E+H					
Final Composite : +D+0.750Lr+0.750L+0.450W+H					
Final Composite : +D+0.750L+0.750S+0.450W+H					
Final Composite : +D+0.750L+0.750S+0.5250E+H					
Final Composite : +0.60D+0.60W+0.60H					
Final Composite : +0.60D+0.70E+0.60H					
Final Composite : span 1					
Final Composite : span 1		0.200	0.75		
Final Composite : span 1		0.400	1.50		
Final Composite : span 1		0.600	2.23		
Final Composite : span 1		0.800	2.96		
Final Composite : span 1		1.000	3.67		
Final Composite : span 1		1.200	4.38		
Final Composite : span 1		1.400	5.07		
Final Composite : span 1		1.600	5.75		
Final Composite : span 1		1.800	6.43		
Final Composite : span 1		2.000	7.09		
Final Composite : span 1		2.200	7.74		

© 2026 ENERCALC, LLC

Deflections tab has two versions, one for shored and one for unshored construction. The only differences are the load combinations and the explanations of load applications listed. The unshored version is shown below.



Load Combination - Service - Unshored	Distance	FINAL	Pre-Composite Total		Pre-Composite Removed		Post Cor
Span Name	ft	Defl (in)	Defl (in)	Ixx Used	Defl (in)	Ixx Used	Def
Pre-Composite Total							
Pre-Composite Removed							
Final Composite : +D+H							
Final Composite : +D+L+H							
Final Composite : +D+Lr+H							
Final Composite : +D+S+H							
Final Composite : +D+0.750Lr+0.750L+H							
Final Composite : +D+0.750L+0.750S+H							
Final Composite : +D+0.60W+H							
Final Composite : +D+0.70E+H							
Final Composite : +D+0.750Lr+0.750L+0.450W+							
Final Composite : +D+0.750L+0.750S+0.450W+H							
Final Composite : +D+0.750L+0.750S+0.5250E+H							
Final Composite : +0.60D+0.60W+0.60H							
Final Composite : +0.60D+0.70E+0.60H							
Final Composite : D Only							
Final Composite : Lr Only							
Final Composite : L Only							
Final Composite : S Only							
Final Composite : W Only							
Final Composite : E Only							
Final Composite : H Only							
span 1				448.0		448.0	
span 1	0.200			448.0		448.0	
span 1	0.400			448.0		448.0	
span 1	0.600			448.0		448.0	
span 1	0.800			448.0		448.0	
span 1	1.000			448.0		448.0	
span 1	1.200			448.0		448.0	
span 1	1.400			448.0		448.0	
span 1	1.600			448.0		448.0	
span 1	1.800			448.0		448.0	

In the Unshored Service Deflections table, the columns warrant some detailed explanation:

The column labeled "Pre-Composite" shows the deflection of the bare steel beam subjected to all loads that are specified to act on the bare steel beam. This would include all loads defined using the "Pre-Composite Removed" option and all loads defined using the "Pre-Composite Remains" option on the Span Loads tab.

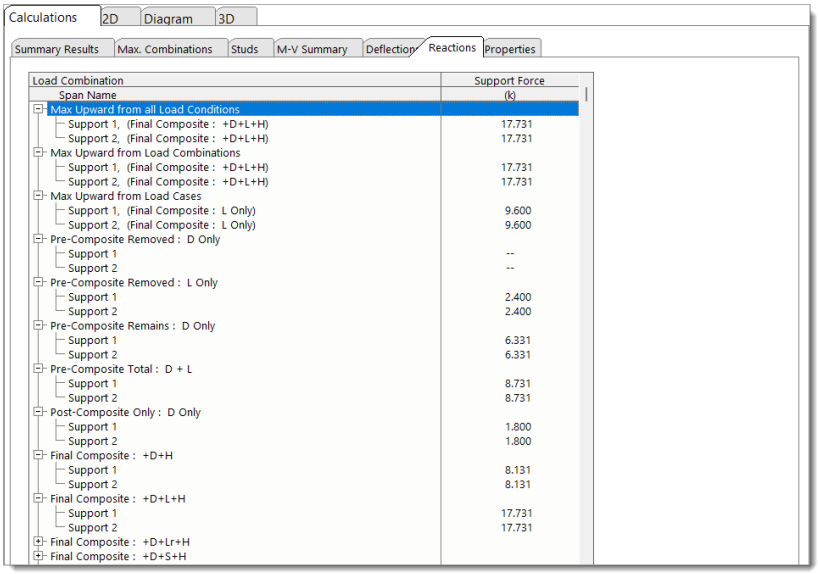
The column labeled "Pre-Composite Removed" shows the deflection of the bare steel beam due all loads defined using the "Pre-Composite Removed" option on the Span Loads tab. Since these loads are removed before the beam reaches its service condition, these construction load deflections are removed from the total deflection, so that the Total deflection represents a correct net total deflection in the service condition.

The column labeled "Added Post Composite" shows the deflection of the composite section due to the loads that are superimposed on the section after it has achieved composite action. This would include all loads defined using the "Post-Composite Only" option on the Span Loads tab.

The column labeled "Final" is calculated by adding the value in the "Pre-Composite" column to the value in the "Added Post Composite" column and then netting out the value in the "Pre-Composite Removed" column. This way, the "Total" deflection represents the full anticipated in-service deflection considering all permanent loads and properly accounting for their sequence of application.

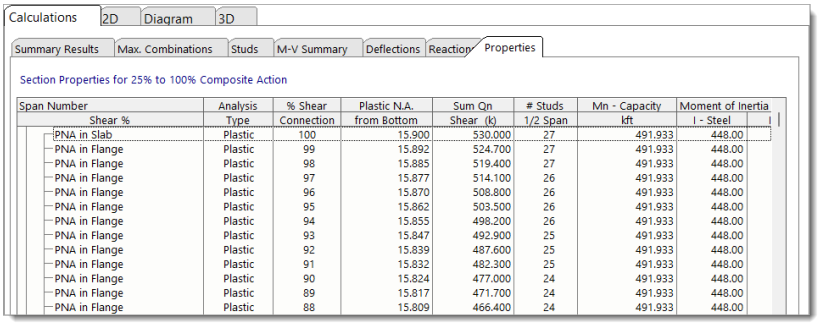
© 2026 ENERCALC, LLC

Reactions tab has two versions, one for shored and one for unshored construction. The only differences are the load combinations and the explanations of load applications listed. The unshored version is shown below.



Load Combination / Span Name	Support Force (k)
Max Upward from all Load Conditions	
Support 1, (Final Composite : +D+L+H)	17.731
Support 2, (Final Composite : +D+L+H)	17.731
Max Upward from Load Combinations	
Support 1, (Final Composite : +D+L+H)	17.731
Support 2, (Final Composite : +D+L+H)	17.731
Max Upward from Load Cases	
Support 1, (Final Composite : L Only)	9.600
Support 2, (Final Composite : L Only)	9.600
Pre-Composite Removed : D Only	
Support 1	--
Support 2	--
Pre-Composite Removed : L Only	
Support 1	2.400
Support 2	2.400
Pre-Composite Remains : D Only	
Support 1	6.331
Support 2	6.331
Pre-Composite Total : D + L	
Support 1	8.731
Support 2	8.731
Post-Composite Only : D Only	
Support 1	1.800
Support 2	1.800
Final Composite : +D+H	
Support 1	8.131
Support 2	8.131
Final Composite : +D+L+H	
Support 1	17.731
Support 2	17.731
Final Composite : +D+Lr+H	
Final Composite : +D+S+H	

Properties tab shows the calculations for transformed section properties calculated in increments of 1% shear connection, from 100% down to the code minimum 25%. "I Lower Bound" and "I Constant Stiffness" are terms described in AISC 360.

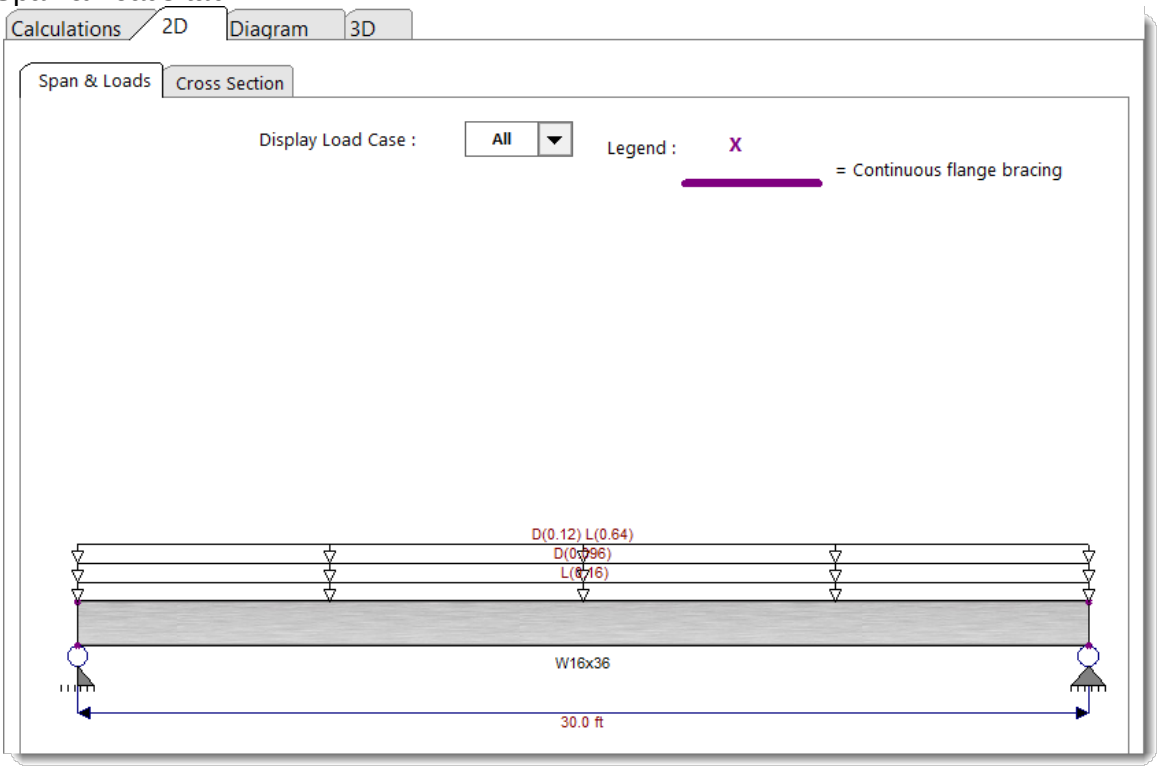


Span Number / Shear %	Analysis Type	% Shear Connection	Plastic N.A. from Bottom	Sum Qn Shear (k)	# Studs 1/2 Span	Mn - Capacity kft	Moment of Inertia I - Steel
PNA in Slab	Plastic	100	15.900	530.000	27	491.933	448.00
PNA in Flange	Plastic	99	15.892	524.700	27	491.933	448.00
PNA in Flange	Plastic	98	15.885	519.400	27	491.933	448.00
PNA in Flange	Plastic	97	15.877	514.100	26	491.933	448.00
PNA in Flange	Plastic	96	15.870	508.800	26	491.933	448.00
PNA in Flange	Plastic	95	15.862	503.500	26	491.933	448.00
PNA in Flange	Plastic	94	15.855	498.200	26	491.933	448.00
PNA in Flange	Plastic	93	15.847	492.900	25	491.933	448.00
PNA in Flange	Plastic	92	15.839	487.600	25	491.933	448.00
PNA in Flange	Plastic	91	15.832	482.300	25	491.933	448.00
PNA in Flange	Plastic	90	15.824	477.000	24	491.933	448.00
PNA in Flange	Plastic	89	15.817	471.700	24	491.933	448.00
PNA in Flange	Plastic	88	15.809	466.400	24	491.933	448.00

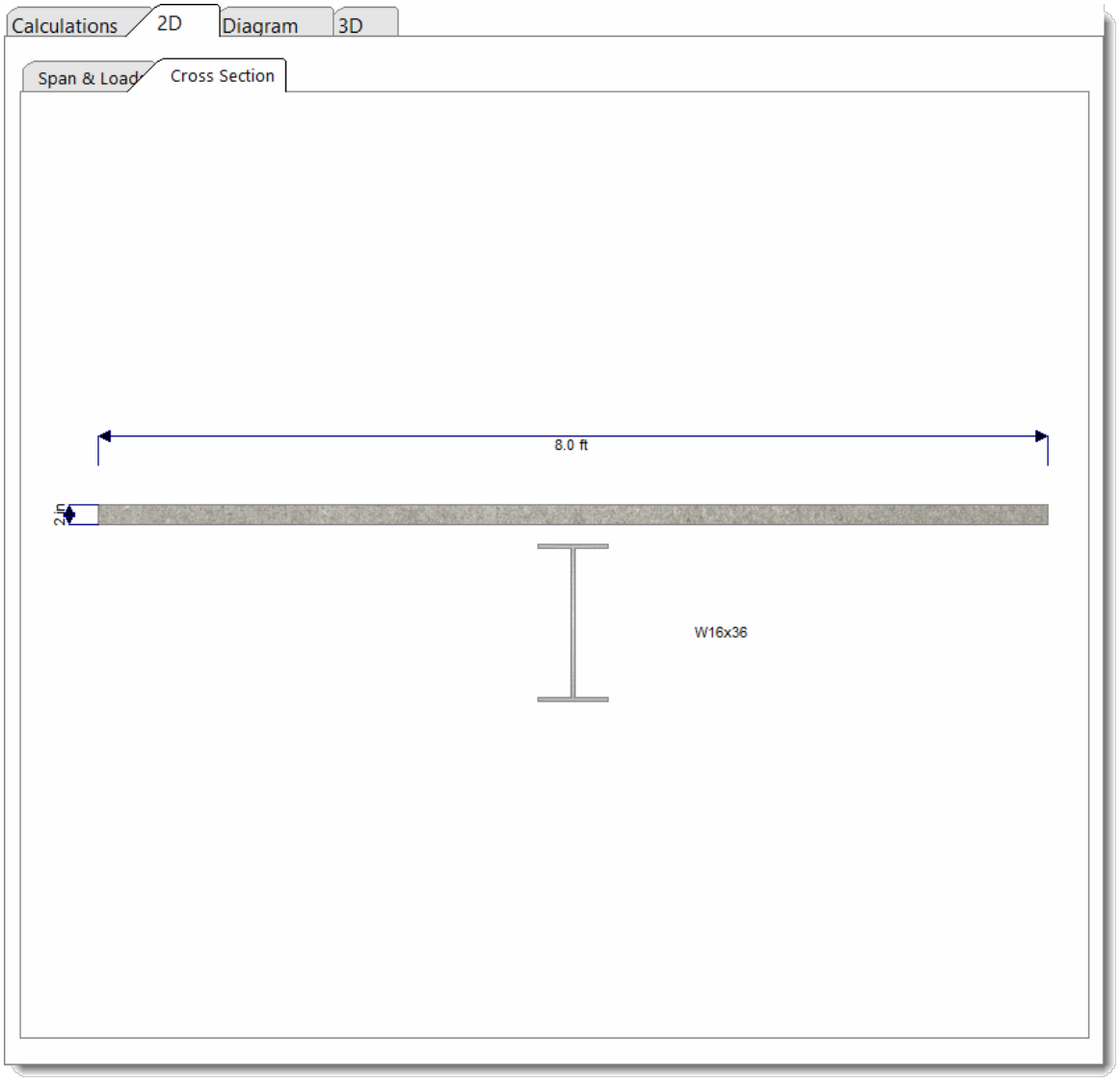
© 2026 ENERCALC, LLC

The 2D Sketch tab provides two ways to view a graphic representation of the beam currently being designed:

Span & Loads tab:

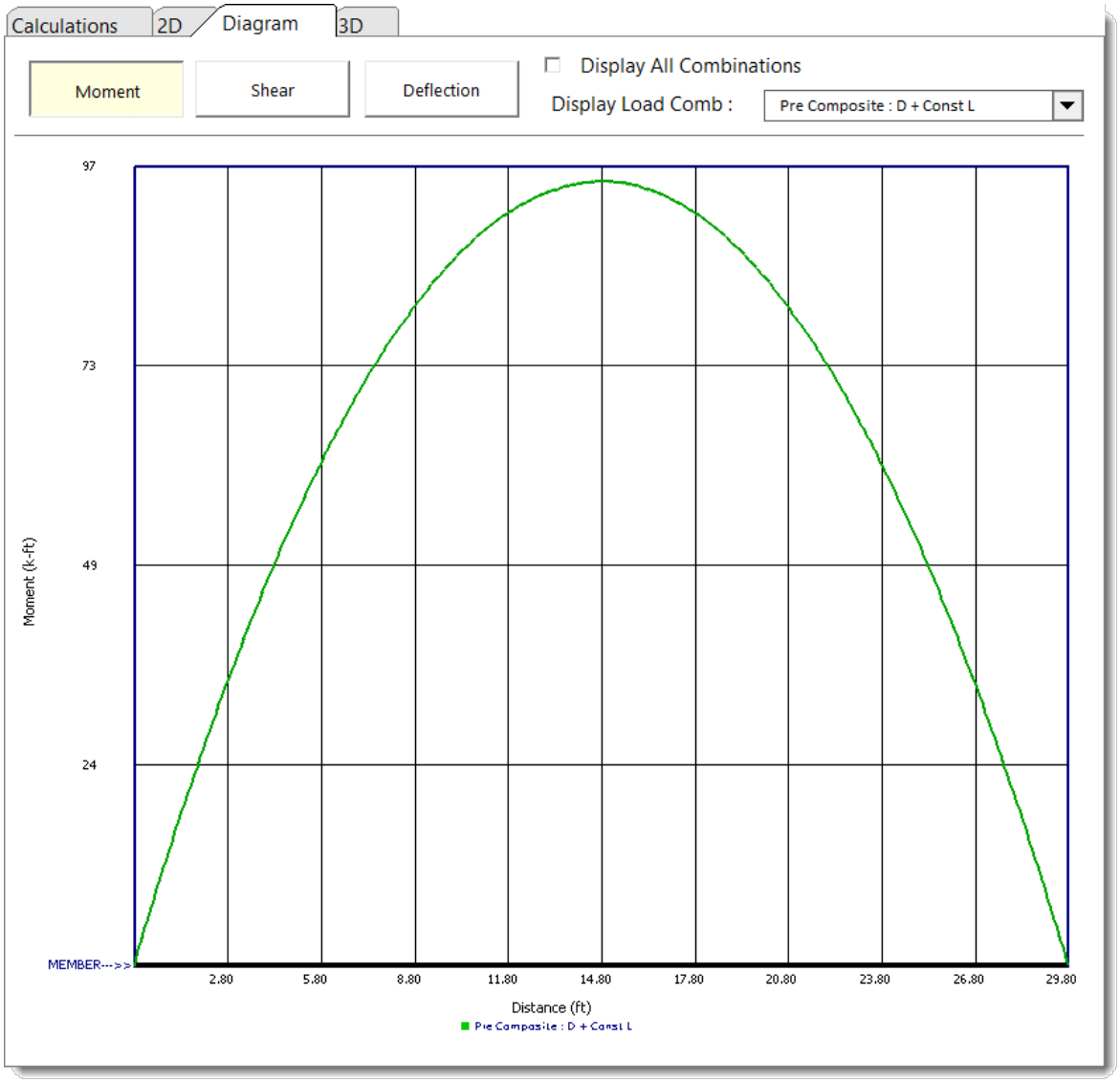


Cross Section tab:

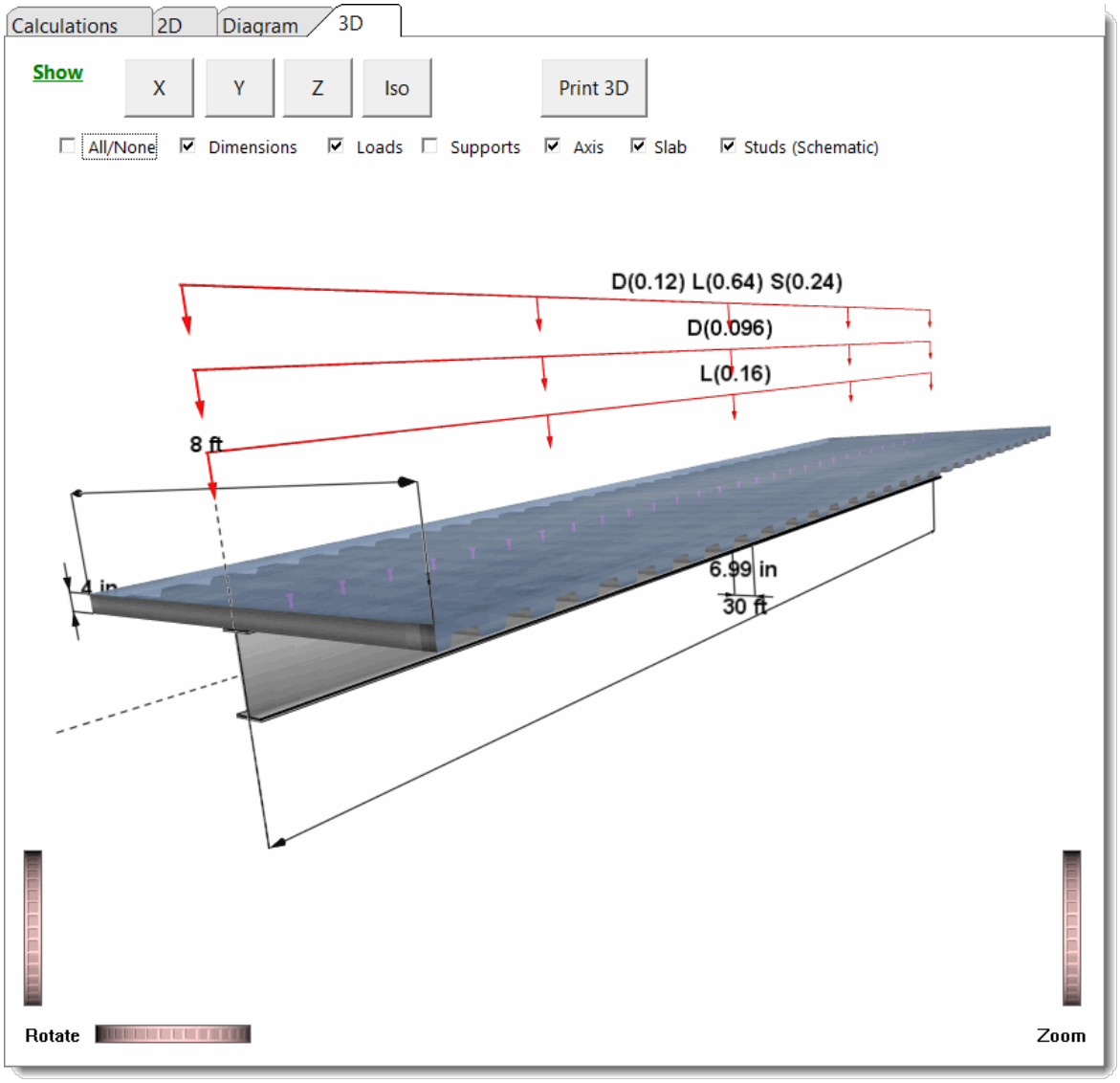


© 2026 ENERCALC, LLC

The Diagram tab offers the ability to view shear, moment, and deflection diagrams for selected load combinations:



The 3D Rendering tab displays a 3D view of the beam and offers display options:



13.4.4 Steel Beam with Torsional Loads

Last revised: Tuesday, February 3, 2026 at 06:12 PM

The Steel Beam with Torsional Loads module offers the ability to analyze and design a single-span steel beam for applied loads that create shear, bending, and torsion per the methodology of AISC Design Guide 9. It can be a useful tool for situations where beams have concentrated or distributed loads that are applied eccentrically, or where beams are subjected to torsional moments.

© 2026 ENERCALC, LLC

General:

The General tab allows you to set the span length, and support conditions in much the same way that this information is provided in the other beam modules. Refer to the Beams 684 topic for additional explanation, but remember that this particular module is limited to single-span conditions. In addition to these pieces of data, the General Data tab also provides input fields for the compression edge lateral bracing condition, the design method, material properties, and an option to force the Cb value to 1.0.

Torsional End Restraints

The support buttons labeled "Warp" at each end of the beam diagram allow you to control the restraint of cross-sectional warping at each end of the beam. This impacts the way in which torsional stresses are handled in the analysis. The following options are available when defining torsional end restraints:

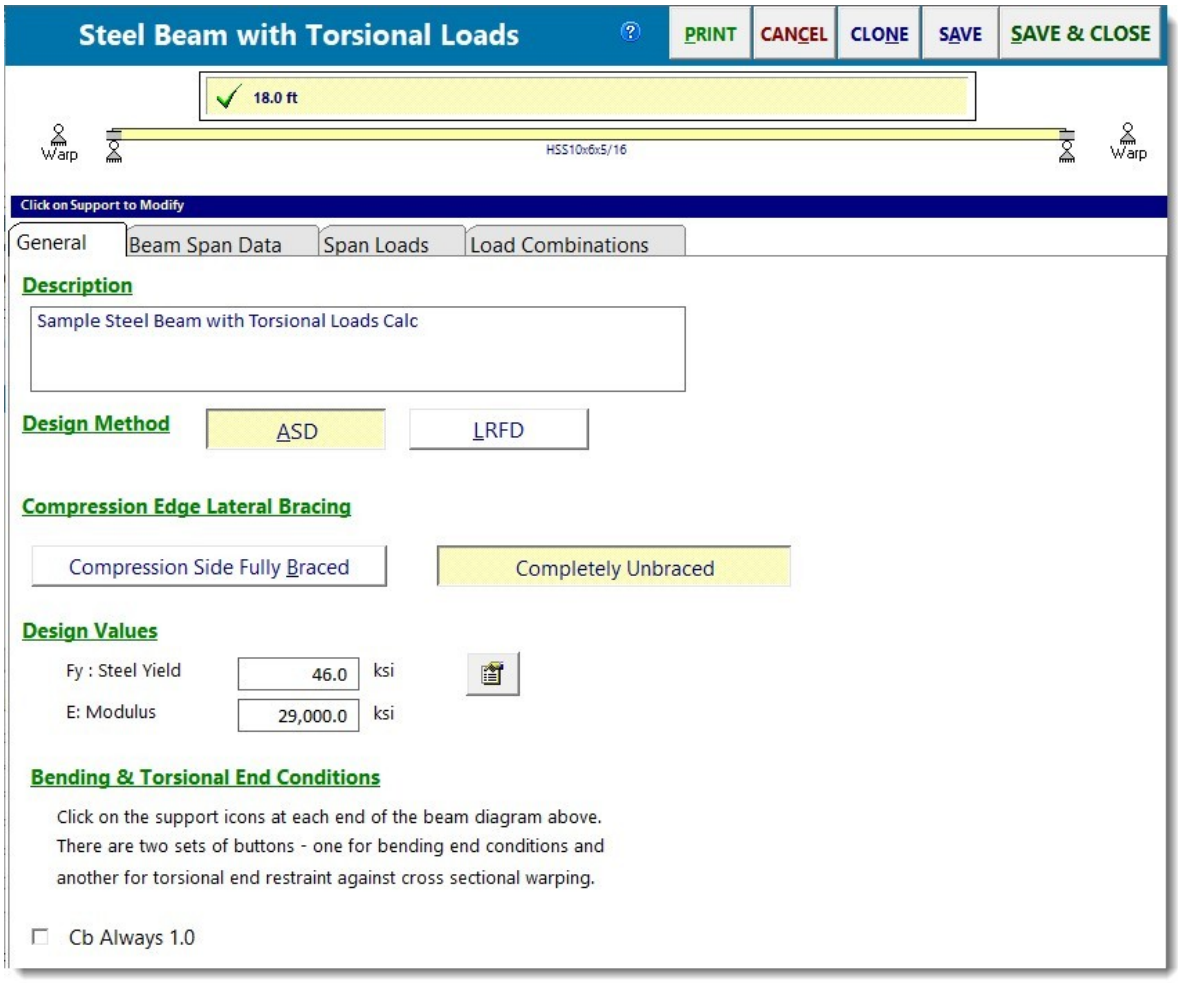
- **Left Pinned, Right Pinned:** Cross-sectional warping is unrestrained at both ends.
- **Left Fixed, Right Fixed:** Cross-sectional warping is restrained at both ends.

NOTE: Regardless of the chosen restraint options, both supports are always assumed to be fixed from a statics and stability standpoint, meaning they will exhibit some torsional reaction. The restraint settings only control whether cross-sectional warping is restrained or not.

Shapes of open cross-sections (like W and C) tend to warp under torsional loading. If this warping is unrestrained, only pure torsional stresses are present per AISC Design Guide 9 §4.1.2 and §4.1.3. However, when warping is restrained, additional direct shear stresses as well as normal flange stresses due to warping are considered.

For closed cross-sections (like pipe and HSS), the effect of warping is negligible whether restrained or not per AISC Design Guide 9 §4.6.2. Therefore, these stresses are neglected from the analysis.

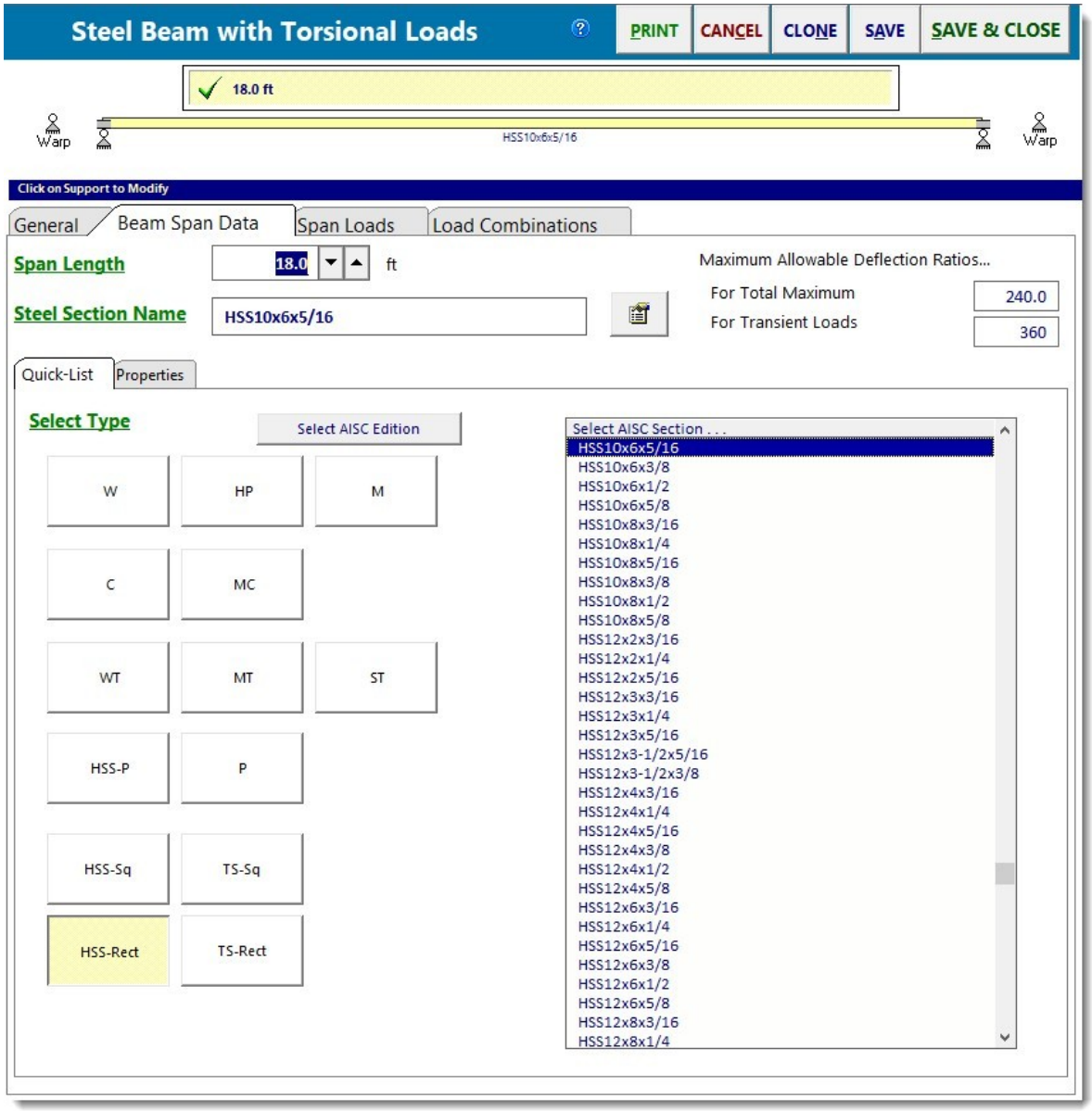
© 2026 ENERCALC, LLC



© 2026 ENERCALC, LLC

Beam Span Data:

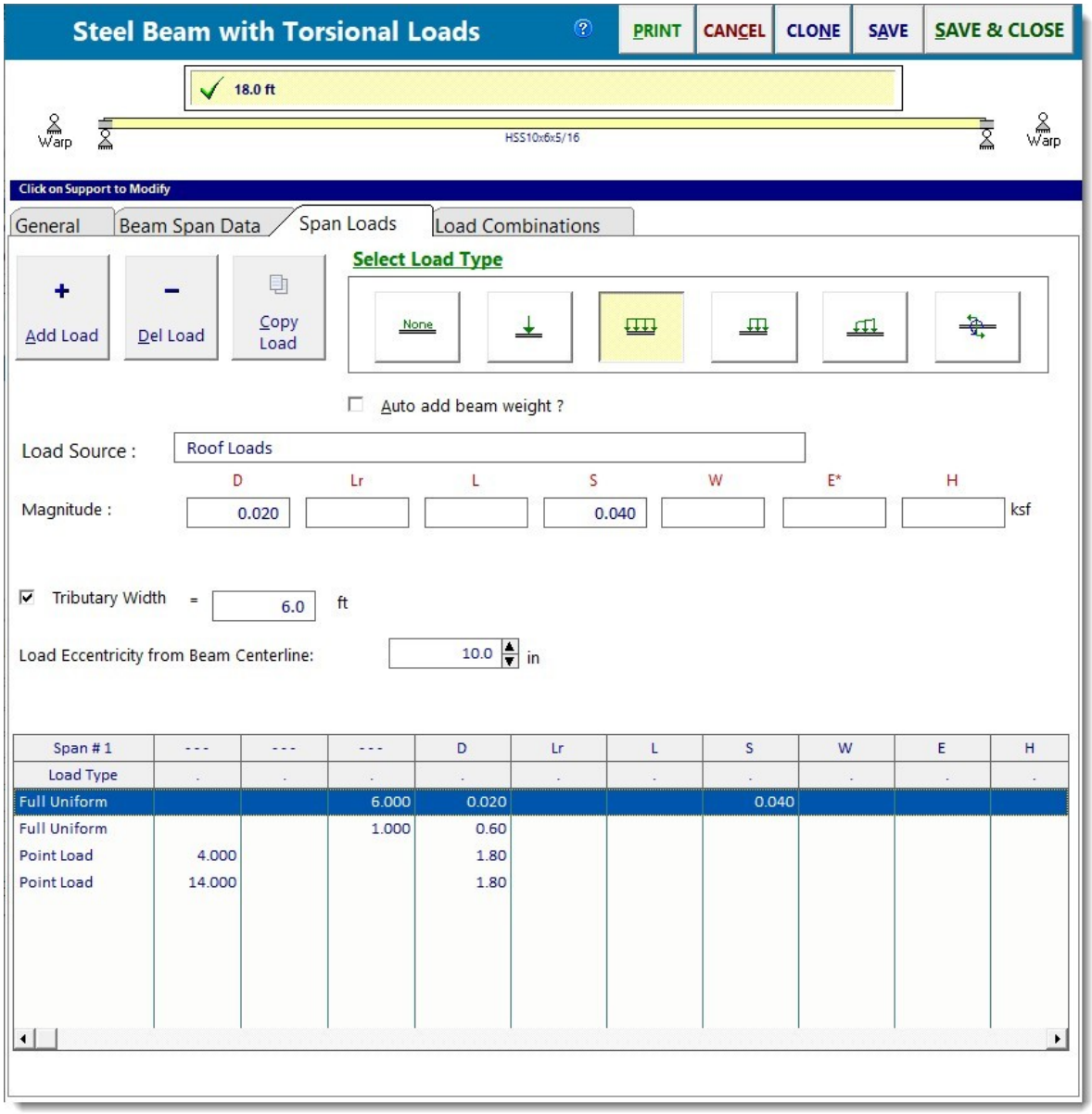
The Beam Span Data tab is used to specify the span length, the allowable deflection ratios, and to select the steel section under consideration.



© 2026 ENERCALC, LLC

Span Loads:

The Span Loads tab allows you to specify loads on one span at a time. The behavior of the tools on this tab is very similar to the tools described for use in the other beam modules, except that it introduces the ability to specify load eccentricities and the ability to indicate that concentrated moments should be considered as torsional moments. Refer to the Beams 684 topic for additional explanation.



© 2026 ENERCALC, LLC

Load Combinations tab:

The Load Combinations tab provides a view of the load combinations that will be analyzed. It also offers the ability to:

- Select a different set of load combinations,
- Modify the values used as load factors, and
- Turn certain combinations on and off.

Refer to the Beams 684 topic for additional explanation.

The Steel Beam with Torsional Loads module offers output options that are analogous to the output options provided by the other beam modules, with the exception that the results include torsional design considerations.

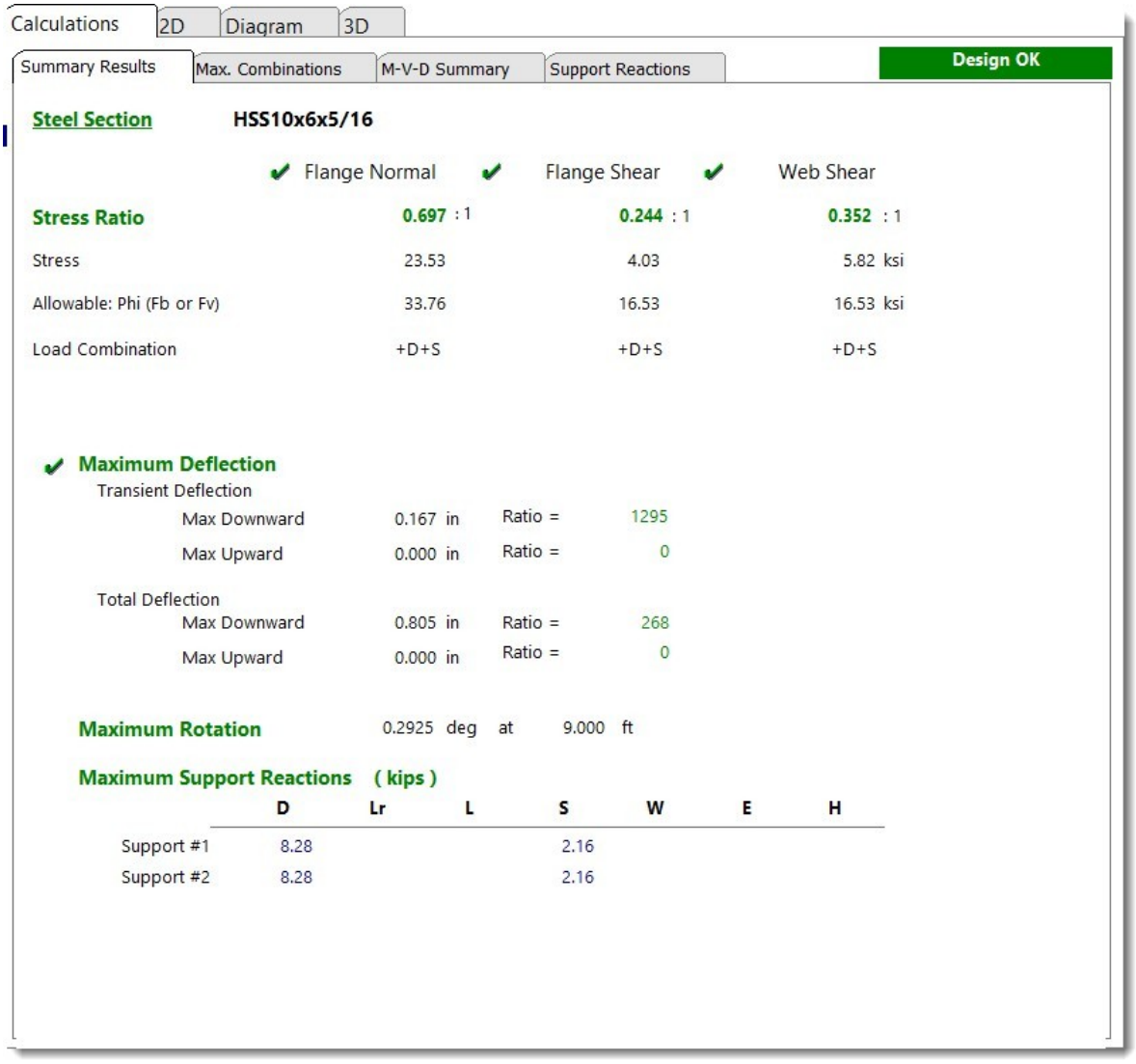
The right half of the screen is dedicated to the display of results. The strip of tabs in the top right-hand corner of the display allows you to choose between Calculations, 2D Sketch, Diagram and 3D Rendering as explained below:

© 2026 ENERCALC, LLC

Calculations:

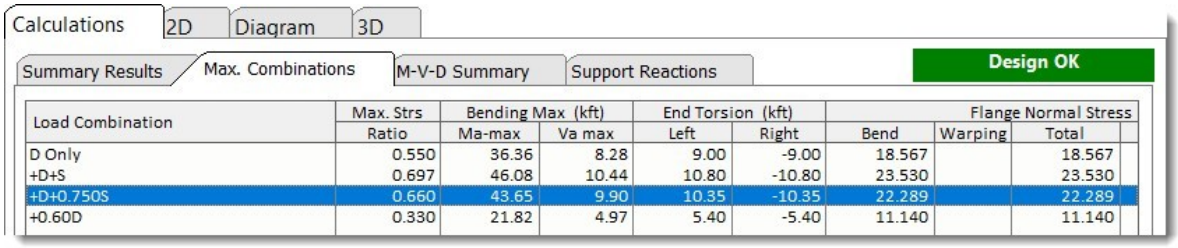
The Calculations tab offers four sub-tabs:

Summary Results: Displays extreme flange normal stress, extreme flange and web shear stress, extreme deflections, extreme rotations, and extreme reactions.



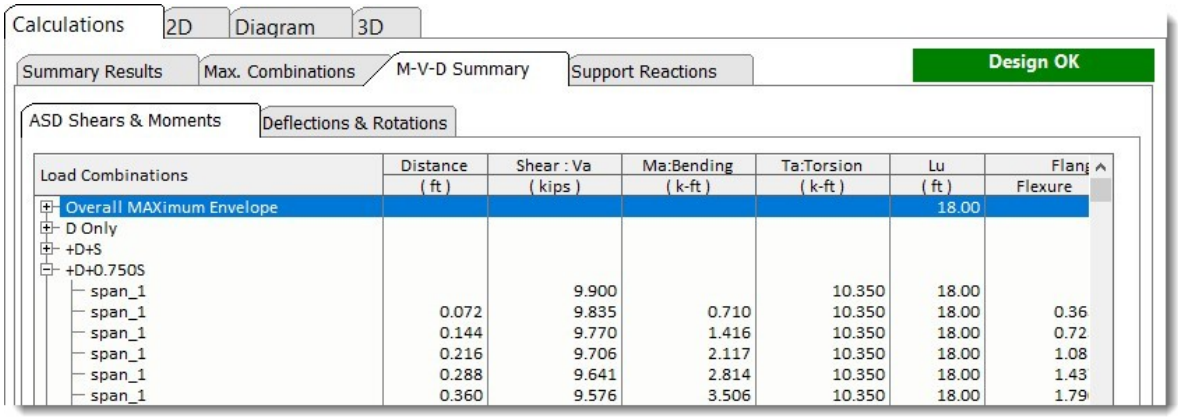
© 2026 ENERCALC, LLC

Maximum Combinations: Displays maximum stress ratio, extreme moments and shears, flange normal stresses due to bending and due to torsion, flange shear stress due to torsion, web shear stress due to bending and due to torsion, for all load combinations.



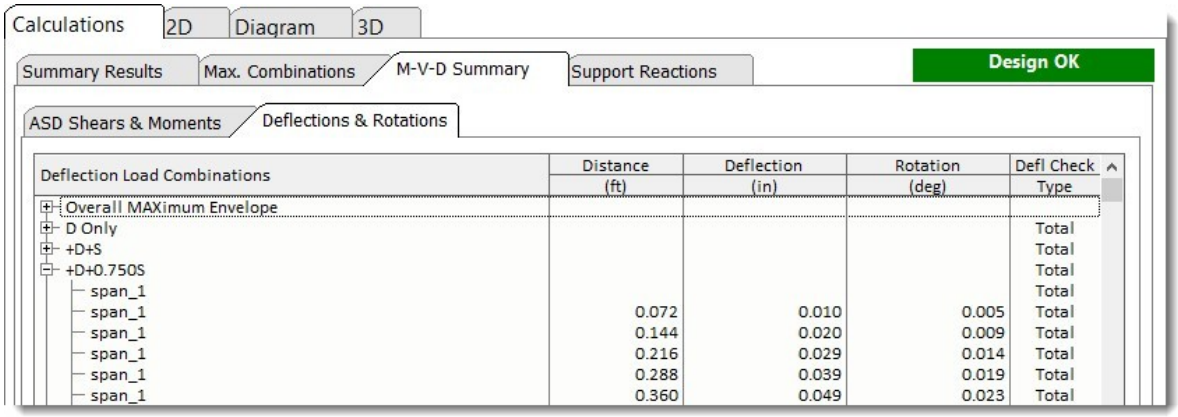
Load Combination	Max. Strs Ratio	Bending Max (kft) Ma-max	Va max	End Torsion (kft) Left	Right	Flange Normal Stress Bend	Warping	Total
D Only	0.550	36.36	8.28	9.00	-9.00	18.567		18.567
+D+S	0.697	46.08	10.44	10.80	-10.80	23.530		23.530
+D+0.750S	0.660	43.65	9.90	10.35	-10.35	22.289		22.289
+0.60D	0.330	21.82	4.97	5.40	-5.40	11.140		11.140

M-V-D: Summary > Shears & Moments: Displays moment, shear, unbraced length, flange normal stress, flange shear stress, and web shear stress at small increments along all spans. Moment, shear, and stresses are displayed for all load combinations.



Load Combinations	Distance (ft)	Shear : Va (kips)	Ma:Bending (k-ft)	Ta:Torsion (k-ft)	Lu (ft)	Flang Flexure
Overall MAXimum Envelope					18.00	
D Only						
+D+S						
+D+0.750S						
span_1		9.900		10.350	18.00	
span_1	0.072	9.835	0.710	10.350	18.00	0.36
span_1	0.144	9.770	1.416	10.350	18.00	0.72
span_1	0.216	9.706	2.117	10.350	18.00	1.08
span_1	0.288	9.641	2.814	10.350	18.00	1.43
span_1	0.360	9.576	3.506	10.350	18.00	1.79

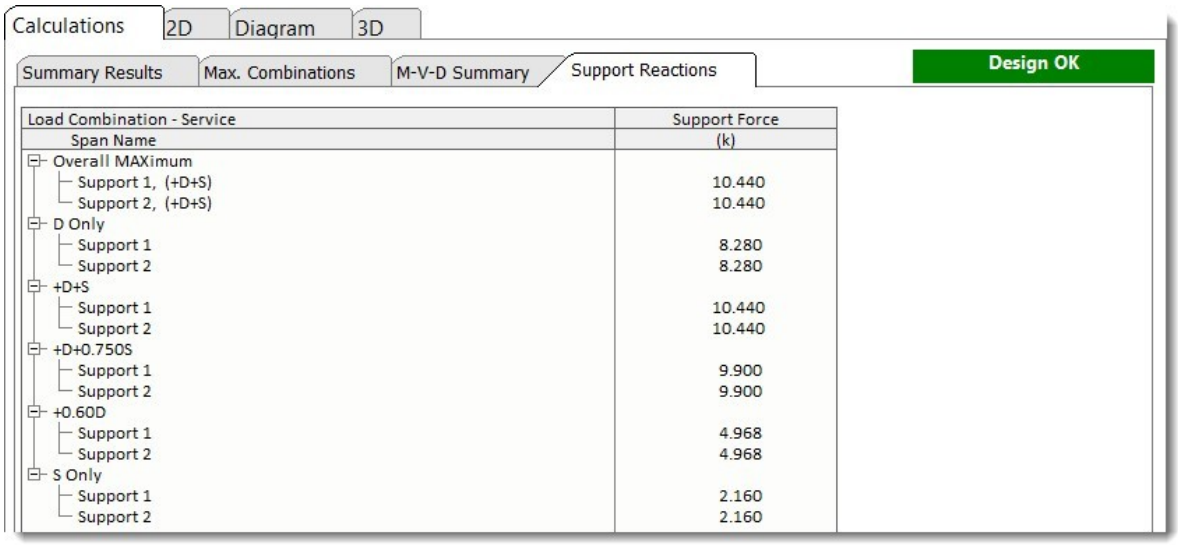
M-V-D: Summary > Deflections & Rotations: Displays deflections and rotations at small increments along all spans. Deflection is displayed for service load combinations only.



Deflection Load Combinations	Distance (ft)	Deflection (in)	Rotation (deg)	Defl Check Type
Overall MAXimum Envelope				
D Only				Total
+D+S				Total
+D+0.750S				Total
span_1				Total
span_1	0.072	0.010	0.005	Total
span_1	0.144	0.020	0.009	Total
span_1	0.216	0.029	0.014	Total
span_1	0.288	0.039	0.019	Total
span_1	0.360	0.049	0.023	Total

© 2026 ENERCALC, LLC

Support Reactions: Displays support reactions for all supports, for all load combinations.



Load Combination - Service Span Name	Support Force (k)
Overall MAXimum	
Support 1, (+D+S)	10.440
Support 2, (+D+S)	10.440
D Only	
Support 1	8.280
Support 2	8.280
+D+S	
Support 1	10.440
Support 2	10.440
+D+0.750S	
Support 1	9.900
Support 2	9.900
+0.60D	
Support 1	4.968
Support 2	4.968
S Only	
Support 1	2.160
Support 2	2.160

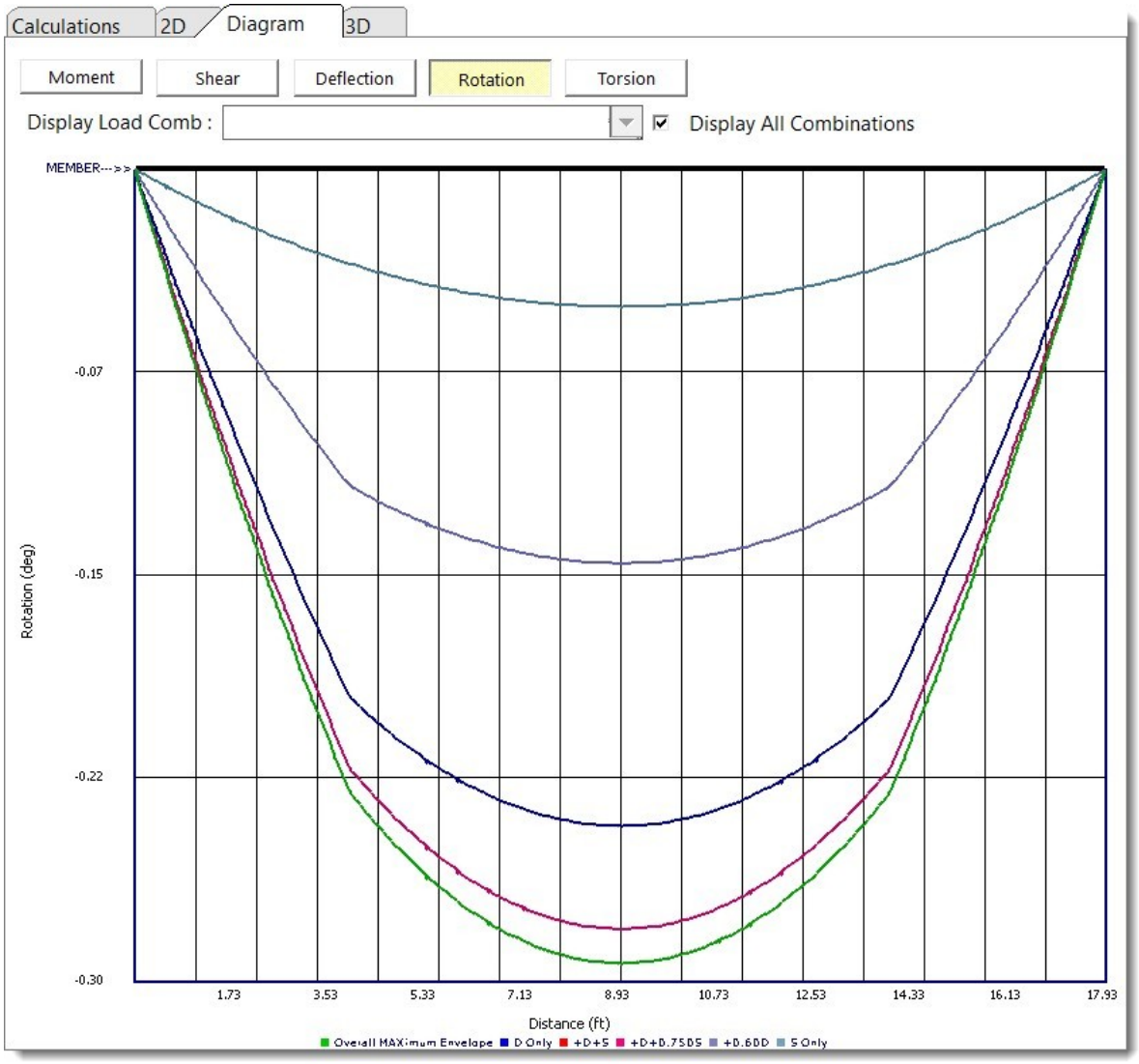
2D Sketch:

Displays a sketch of the beam, indicating span lengths, support conditions, and applied loads.

© 2026 ENERCALC, LLC

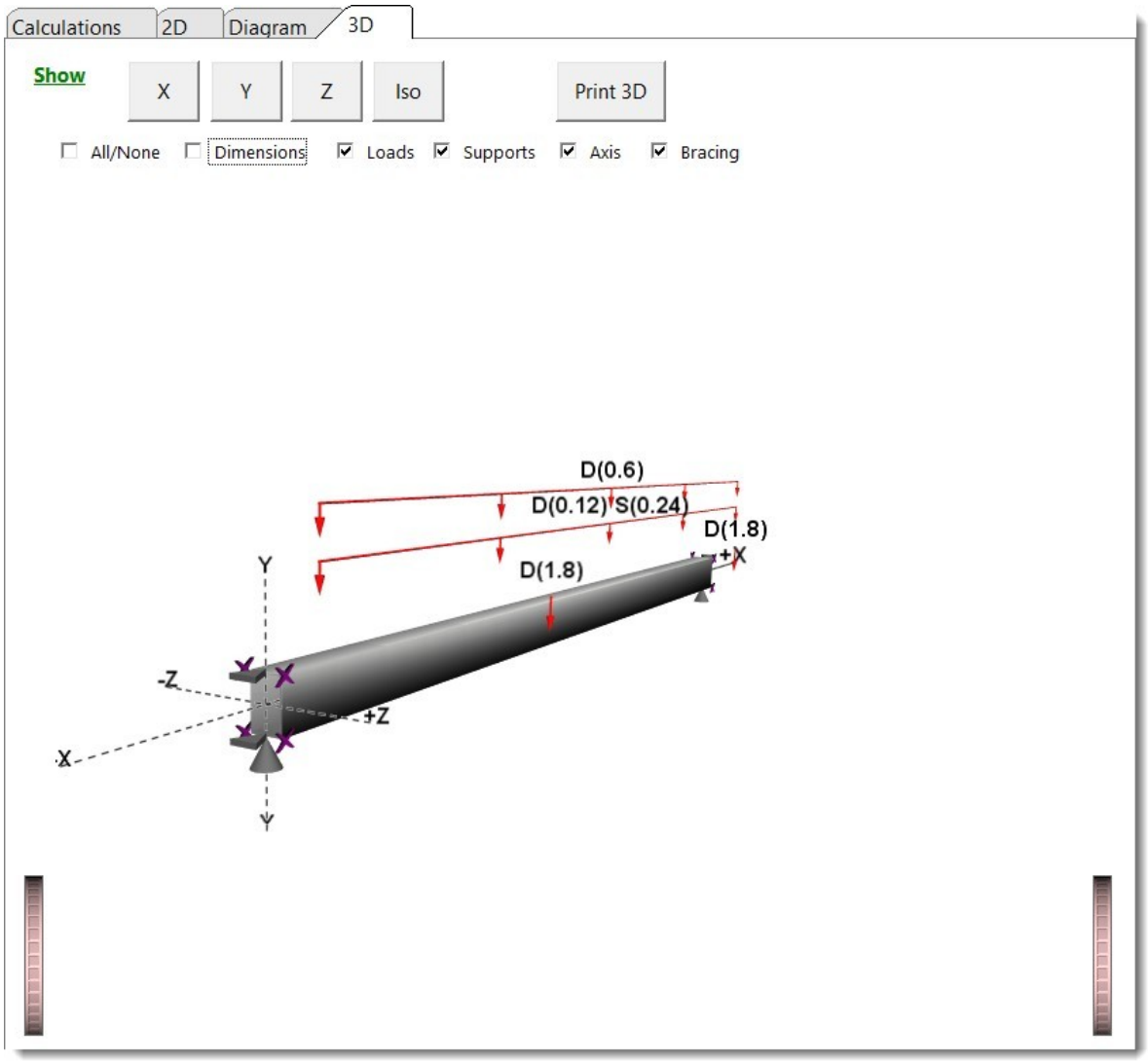
Diagram:

Displays a graphic depiction of the beam with superimposed graphs of Moment, Shear, Deflection or Rotation for a selected load combination, or for an envelope of all load combinations.



3D Rendering:

Displays a 3D graphic depiction of the beam many display options:



13.4.5 Concrete Beam

Last revised: Tuesday, February 3, 2026 at 06:12 PM

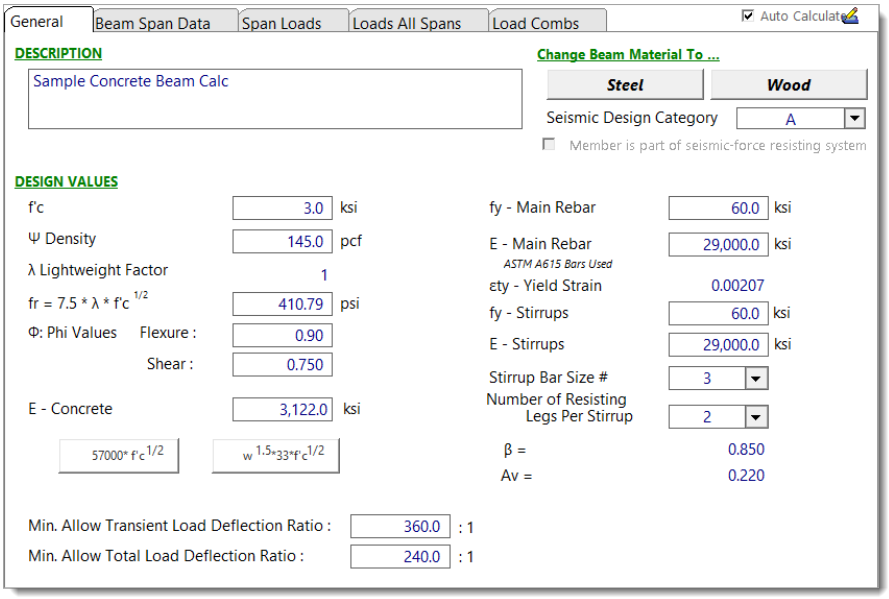
In this section, for each input tab we will review only the items that are unique to the CONCRETE material type.

General

The concrete beam module handles single- and multiple-span (up to 8-span) beams using ONE cross section shape. That shape can have up to six groups of reinforcing per span, and the reinforcing can vary on a span-by-span basis.

© 2026 ENERCALC, LLC

This tab collects all the required rebar, concrete strength and elastic modulus entries, shear stirrup data, strength reduction factors and deflection criteria to check.



NOTE: It is important to know how this module operates with regard to beam stiffness along the span, and how this affects multi-span beams. This module divides each span into a series of segments. The effective moment of inertia for each segment (for each load combination) is calculated using the actual unfactored moment on that segment. Thus the module creates a very accurate variable stiffness model of the beam based on actual moments. For multiple-span beams, this will affect the relative stiffness of each beam span. Thus the moment distribution across multiple spans will be properly performed. This will affect factored load moments and shears and service load level deflections and reactions.

Seismic Design Categories and Concrete Beam Limitations

Concrete beams are subject to different requirements depending on the seismic design category (SDC) and whether they are part of the Seismic Force-Resisting System (SFRS). Below is a detailed explanation of how the Concrete Beam module handles each scenario:

Beam is part of the Seismic Force-Resisting System:

SDC	Support	ACI 318-19 Reference
A	No limitations.	-
B	Results displayed with warning to refer to additional ACI requirements.	Section 18.3
C	Results displayed with warning to refer to additional ACI requirements.	Section 18.4

© 2026 ENERCALC, LLC

D, E, F	Not supported; beyond the scope of this module.	Section 18.6

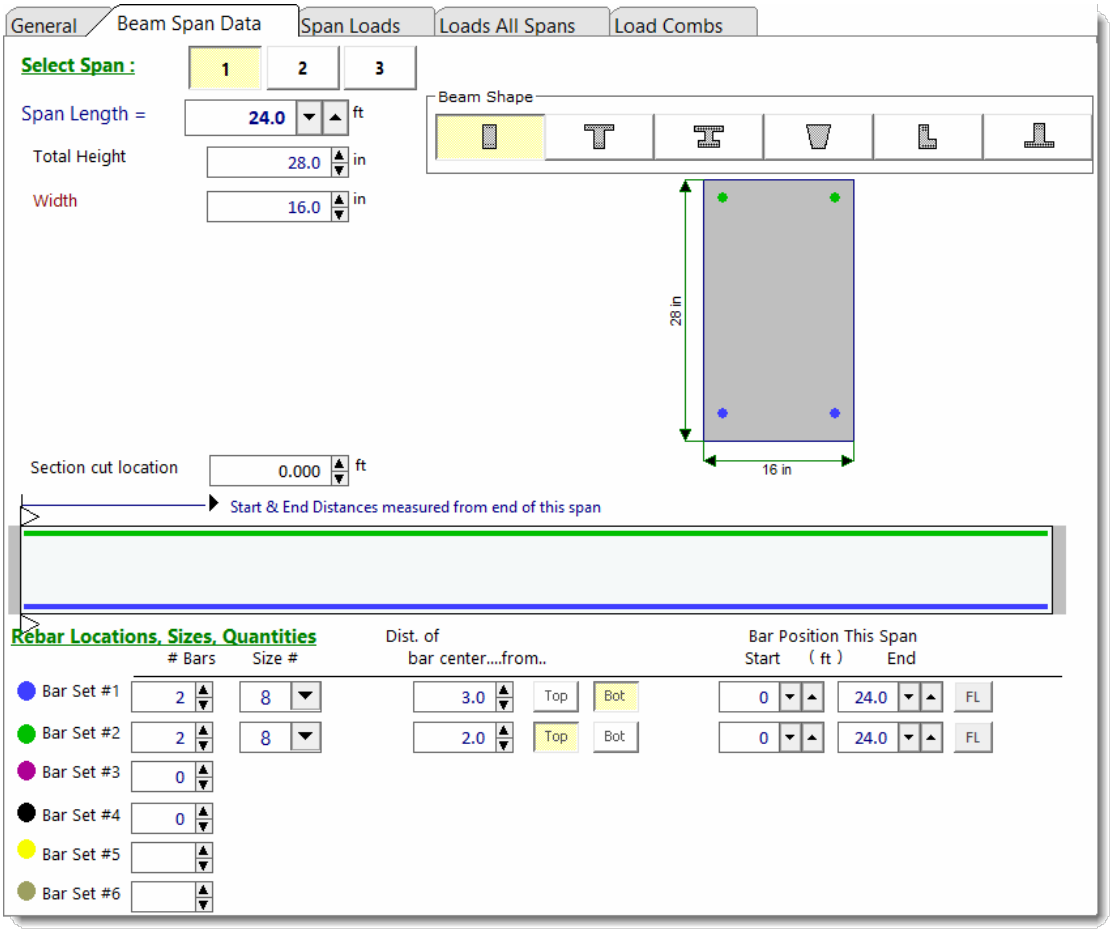
Beam is NOT part of the Seismic Force-Resisting System:

SDC	Support	ACI 318-19 Reference
A, B, C	No limitations.	-
D, E, F	Results displayed with warning to refer to additional ACI requirements.	Section 18.14

For scenarios where warnings are displayed, users must refer to the relevant ACI sections and ensure compliance separately as those additional requirements are outside the scope of this module.

Beam Span Data

This tab has some pieces of input that are constant for all spans and some that can vary on a span-by-span basis.



The cross section shape and dimensions are the same for all spans.

When you click on a span (for multi-span beams), the span length and rebar layout will update to display the arrangement that is specific to that span.

On the bottom of this tab you can specify up to 6 bar sets (quantity, size, vertical location and start/stop endpoints). Each bar set is referenced on the sketch with a color shown as a dot to the left of the set description.

The column titled "Dist of bar center....from...." is how you set the vertical position of the bar sets in the beam. When you look at the top one, you can read it as "The top bar set is 3 inches from the bottom of the beam". Note that the module will know whether the bars are in tension or compression and will handle the calculations properly.

The item labeled Bar Position This Span defines the starting and ending location of the bar ends with respect to the left end of each respective span. The data in the screen capture below shows that bar set #1 and #2 run from the left end (0.0 ft) to 24.0 ft from the left end of Span 1. Using these starting and ending locations you can fine tune the bar layout and end cutoffs.

NOTE: The module will report an error message if any beam segments are found to be completely unreinforced. Therefore, it is imperative that the rebar be defined in such a way as to prevent completely unreinforced segments. This includes the short segments at the extreme ends of a beam, where rebar is typically terminated. Remember that this module is an analysis tool, not a detailing tool, so don't be tempted to define rebar as starting or ending short of the physical end of the beam. The button labeled "FL" adjacent to each rebar definition has been provided as a convenient way to indicate to the program that the selected rebar runs the Full Length of the span.

IMPORTANT: In this module, rebar sets are assumed to be fully effective at the locations where they are defined. This means the reinforcement is considered fully developed at their defined start and end points, as reflected in the capacity diagram. Users must account for this behavior and ensure that any required additional rebar lengths are included in the construction drawings to achieve full development at the intended locations.

Span Loads

No differences from other materials.

All Span Loads

No differences from other materials.

Load Combinations

No differences from other materials.

© 2026 ENERCALC, LLC

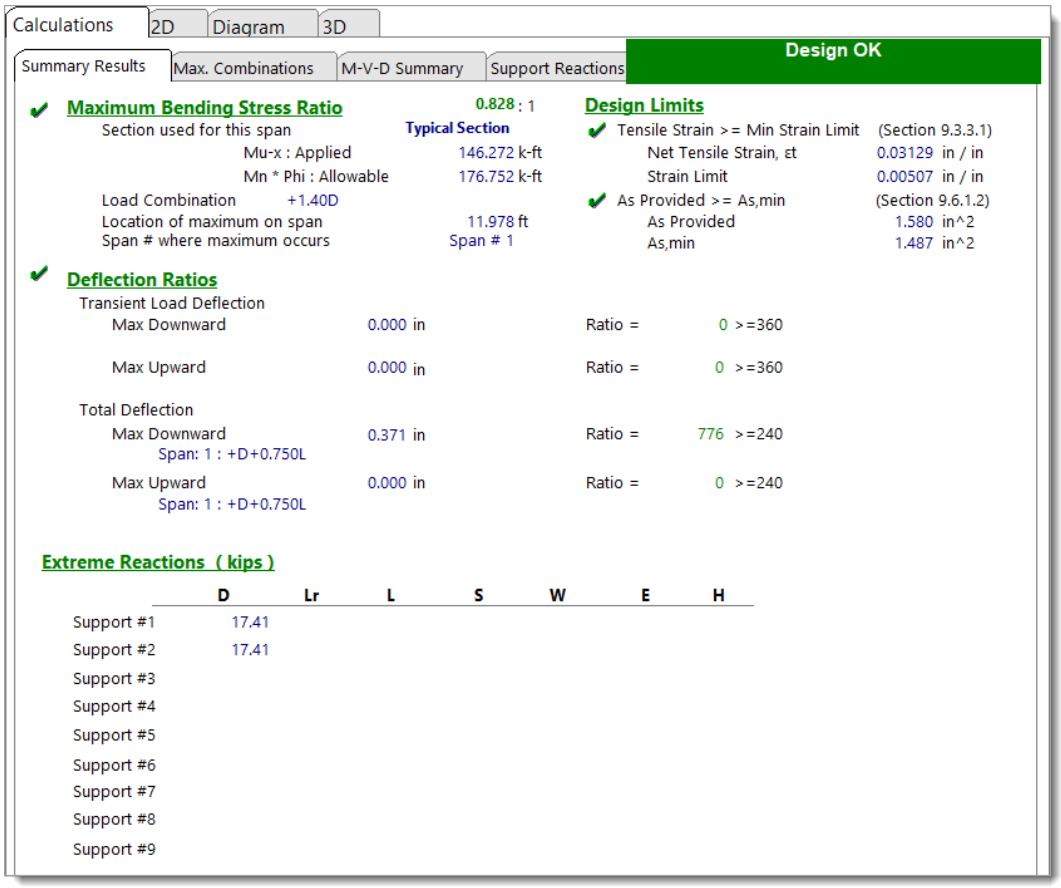
Results Tabs

This set of tabs provides detailed results for the current calculation. The tabs in the upper right-hand corner of the screen allow you to select the four major areas available for review: **Calculations**, **2D Sketch**, **Diagram**, and **3D Rendering**.

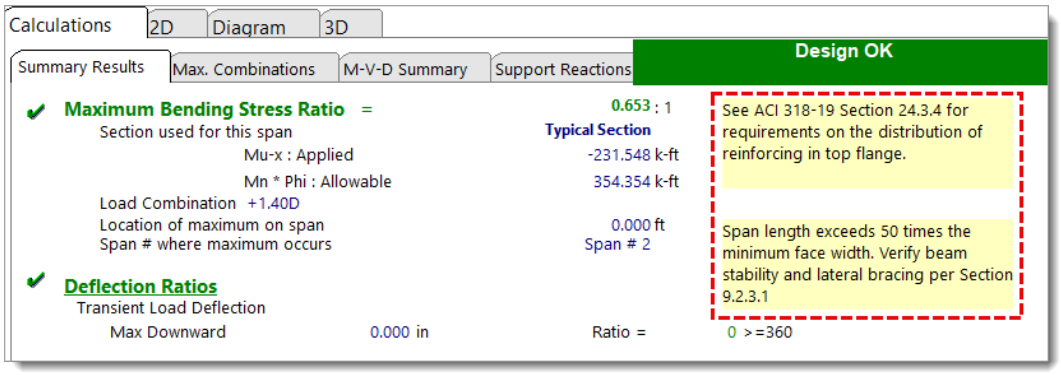
Calculations

The Calculations tab offers the following results subtabs:

Summary Results provides details for shear, moment and deflection for the governing load combinations, as well as the governing prescriptive design limit checks required by the ACI. Shear results are not shown here. They are summarized on a separate tab that gives the required stirrup layout.

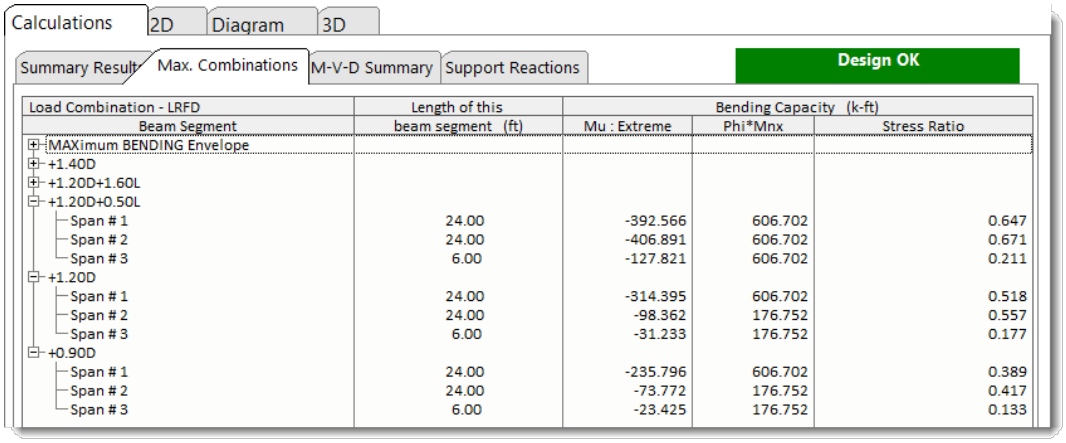


Depending on your beam configuration, related advisory messages may appear in the Summary Results tab. For example:



These checks are outside the scope of this module's automated checks and must be verified by the user independently.

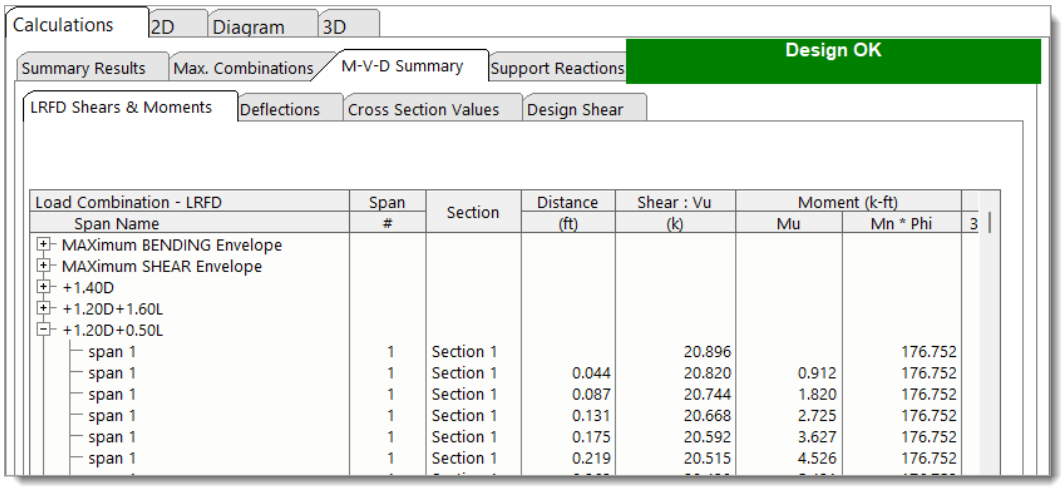
Max. Combinations provides detailed results for each beam segment for each load combination. These results are a consolidation to the highly detailed incremental results on the M-V-D Summary tab.



Load Combination - LRFD / Beam Segment	Length of this beam segment (ft)	Bending Capacity (k-ft) Mu : Extreme	Phi*Mnx	Stress Ratio
MAXimum BENDING Envelope				
+1.40D				
+1.20D+1.60L				
+1.20D+0.50L				
Span # 1	24.00	-392.566	606.702	0.647
Span # 2	24.00	-406.891	606.702	0.671
Span # 3	6.00	-127.821	606.702	0.211
+1.20D				
Span # 1	24.00	-314.395	606.702	0.518
Span # 2	24.00	-98.362	176.752	0.557
Span # 3	6.00	-31.233	176.752	0.177
+0.90D				
Span # 1	24.00	-235.796	606.702	0.389
Span # 2	24.00	-73.772	176.752	0.417
Span # 3	6.00	-23.425	176.752	0.133

M-V-D Summary - LRFD Shears & Moments shows highly detailed moment and shear results for each beam and for each load combination. For multi-span beams using Automatic Unbalanced Live Load Placement there may be thousands of lines of results.

© 2026 ENERCALC, LLC



Load Combination - LRFD / Span Name	Span #	Section	Distance (ft)	Shear : Vu (k)	Moment (k-ft) Mu	Moment (k-ft) Mn * Phi	3
MAXimum BENDING Envelope							
MAXimum SHEAR Envelope							
+1.40D							
+1.20D+1.60L							
+1.20D+0.50L							
span 1	1	Section 1		20.896		176.752	
span 1	1	Section 1	0.044	20.820	0.912	176.752	
span 1	1	Section 1	0.087	20.744	1.820	176.752	
span 1	1	Section 1	0.131	20.668	2.725	176.752	
span 1	1	Section 1	0.175	20.592	3.627	176.752	
span 1	1	Section 1	0.219	20.515	4.526	176.752	

Minimum Steel Reinforcement Calculations ($A_{s,min}$)

Calculations for minimum steel reinforcement ($A_{s,min}$) are performed in accordance with ACI Sections 9.6.1.2 and 9.6.1.3 and reported accordingly:

1. The module evaluates equations (a) and (b) from ACI Section 9.6.1.2.
2. After determining the reinforcement ratio (ρ) required to meet flexural strength, the module checks Section 9.6.1.3 which specifies that the minimum steel area calculated from Section 9.6.1.2 does not need to exceed 4/3 of the A_s required for strength.
3. From these checks, the governing ρ_{min} is identified, and the associated $A_{s,min}$ value is determined and reported.

If the area of flexural reinforcement anywhere along the beam is less than $A_{s,min}$, the module will issue a warning letting you know to adjust the design.

Strain Limit Checks

This module calculates the net tensile strain, εt, at nominal strength to verify tension-controlled behavior in accordance with ACI 318 Section 9.3.3.1 and Table 21.2.2.

The net tensile strain limit is defined as:

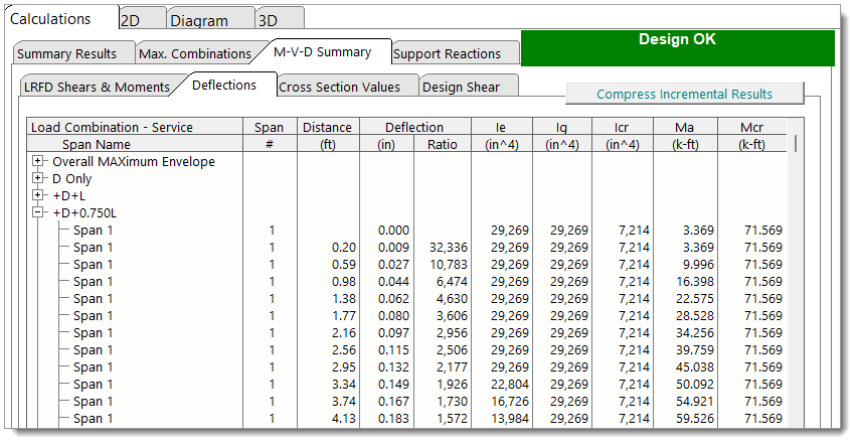
Strain Limit = 0.004 *[ACI 318-14 Section 9.3.3.1]*

Strain Limit = ε_{ty} + 0.003, where $\varepsilon_{ty} = f_y / E$ *[ACI 318-19 / 25 Section 9.3.3.1 and Table 21.2.2]*

The net tensile strain, ε_t, is determined using a strain compatibility analysis of the cross section. This analysis is performed for each unique cross section with the top fiber in tension and with the bottom fiber in tension. The resulting values are also reported on the *Cross Section Values* tab.

© 2026 ENERCALC, LLC

M-V-D Summary - Deflections shows highly detailed deflection results for all load combinations. When each load combination is expanded by clicking the **[+]** icon, you will see the deflections along the entire beam. You will also see the effective moment of inertia used in that region. (Remember that the effective moment of inertia is calculated based on service-level moments at many locations along the length of each span.)



Load Combination - Service / Span Name	Span #	Distance (ft)	Deflection (in)	Deflection Ratio	Ie (in^4)	Ig (in^4)	Icr (in^4)	Ma (k-ft)	Mcr (k-ft)
Overall MAXimum Envelope									
D Only									
+D+L									
+D+0.750L									
Span 1	1		0.000		29,269	29,269	7,214	3.369	71.569
Span 1	1	0.20	0.009	32,336	29,269	29,269	7,214	3.369	71.569
Span 1	1	0.59	0.027	10,783	29,269	29,269	7,214	9.996	71.569
Span 1	1	0.98	0.044	6,474	29,269	29,269	7,214	16.398	71.569
Span 1	1	1.38	0.062	4,630	29,269	29,269	7,214	22.575	71.569
Span 1	1	1.77	0.080	3,606	29,269	29,269	7,214	28.528	71.569
Span 1	1	2.16	0.097	2,956	29,269	29,269	7,214	34.256	71.569
Span 1	1	2.56	0.115	2,506	29,269	29,269	7,214	39.759	71.569
Span 1	1	2.95	0.132	2,177	29,269	29,269	7,214	45.038	71.569
Span 1	1	3.34	0.149	1,926	22,804	29,269	7,214	50.092	71.569
Span 1	1	3.74	0.167	1,730	16,726	29,269	7,214	54.921	71.569
Span 1	1	4.13	0.183	1,572	13,984	29,269	7,214	59.526	71.569

ACI 318-14: The module calculates I_e using Branson's formula as outlined in ACI 318-14 Equation 24.2.3.5(a), but with powers of 4 instead of 3:

$$I_e = \left(\frac{M_{cr}}{M_a}\right)^4 I_g + \left[1 - \left(\frac{M_{cr}}{M_a}\right)^4\right] I_{cr}$$

The "fourth-power" version of Branson's formula, as recommended by ACI 435R-20, provides a more accurate representation of stiffness variations along the member for section-based analyses. In contrast, the "third-power" version in Equation 24.2.3.5(a) represents an averaged I_e over the full member length, which is not suitable for the section-based analysis performed by the Concrete Beam module.

ACI 318-19: The effective moment of inertia, I_e, is calculated based on ACI 318-19 Table 24.2.3.5:

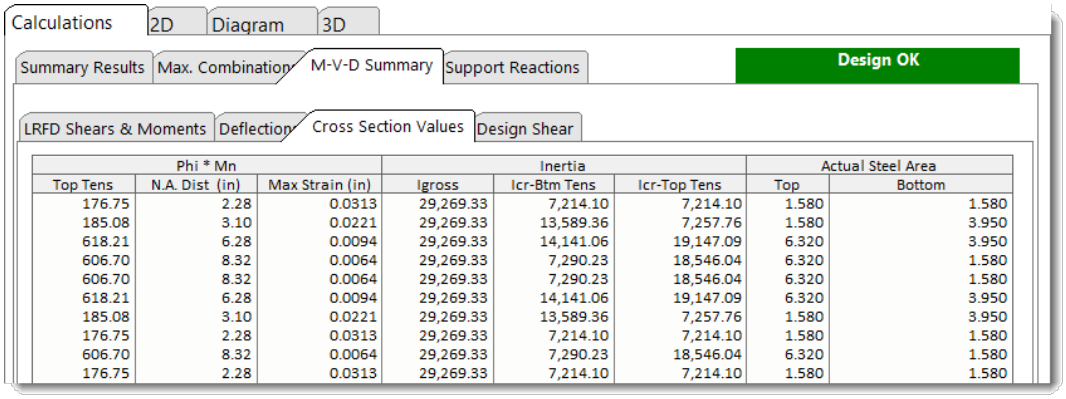
$$M_a \le (2/3)M_{cr}: \qquad I_e = I_g$$

$$M_a > (2/3)M_{cr}: \qquad I_e = \frac{I_{cr}}{1 - \left(\frac{(2/3)M_{cr}}{M_a}\right)^2 \left(1 - \frac{I_{cr}}{I_g}\right)}$$

M-V-D Summary - Cross Section Values shows the moment capacities and moment of inertia for all of the identified cross sections. The module has examined all of the spans you defined and looked for identical reinforcing layouts. It has eliminated

© 2026 ENERCALC, LLC

the duplicates, and for simplicity, it only lists the unique reinforced cross sections here.



Phi * Mn			Inertia			Actual Steel Area	
Top Tens	N.A. Dist (in)	Max Strain (in)	Igross	Icr-Btm Tens	Icr-Top Tens	Top	Bottom
176.75	2.28	0.0313	29,269.33	7,214.10	7,214.10	1.580	1.580
185.08	3.10	0.0221	29,269.33	13,589.36	7,257.76	1.580	3.950
618.21	6.28	0.0094	29,269.33	14,141.06	19,147.09	6.320	3.950
606.70	8.32	0.0064	29,269.33	7,290.23	18,546.04	6.320	1.580
606.70	8.32	0.0064	29,269.33	7,290.23	18,546.04	6.320	1.580
618.21	6.28	0.0094	29,269.33	14,141.06	19,147.09	6.320	3.950
185.08	3.10	0.0221	29,269.33	13,589.36	7,257.76	1.580	3.950
176.75	2.28	0.0313	29,269.33	7,214.10	7,214.10	1.580	1.580
606.70	8.32	0.0064	29,269.33	7,290.23	18,546.04	6.320	1.580
176.75	2.28	0.0313	29,269.33	7,214.10	7,214.10	1.580	1.580

Steel Area for ρw reports the calculated values used to determine the tension reinforcement ratio (ρ_w) which is needed for calculating concrete shear capacity per Table 22.5.5.1.

As Top: Total area of tension reinforcement when the top of the beam is in tension. Taken as the sum of the areas of longitudinal bars located above the N.A. Dist value for "When Top in Tension" where N.A. Dist is measured from the bottom of the beam.

d Top: Distance from the bottom of the beam to the centroid of the bars included in "As Top".

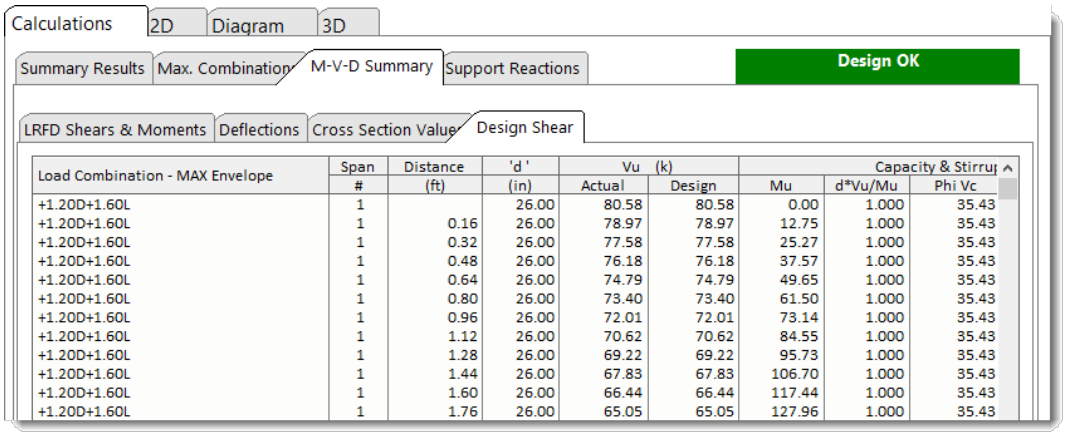
As Bottom: Total area of tension reinforcement when the bottom of the beam is in tension. Taken as the sum of the areas of longitudinal bars located below the N.A. Dist value for "When Bottom in Tension" where N.A. Dist is measured from the top of the beam.

d Bottom: Distance from the top of the beam to the centroid of the bars included in "As Bottom".

The tension reinforcement ratio is then calculated as $A_s / b_w d$ using either As Top and d Top or As Bottom and d Bottom, depending on the sign of the moment (M_u) at a given location along the beam.

M-V-D Summary - Design Shear shows the shear stirrup requirements along the span(s) as required by the governing load combinations that generate the highest shear at each section.

© 2026 ENERCALC, LLC



Load Combination - MAX Envelope	Span #	Distance (ft)	'd ' (in)	Vu (k) Actual	Vu (k) Design	Capacity & Stirrup Mu	d*Vu/Mu	Phi Vc
+1.20D+1.60L	1		26.00	80.58	80.58	0.00	1.000	35.43
+1.20D+1.60L	1	0.16	26.00	78.97	78.97	12.75	1.000	35.43
+1.20D+1.60L	1	0.32	26.00	77.58	77.58	25.27	1.000	35.43
+1.20D+1.60L	1	0.48	26.00	76.18	76.18	37.57	1.000	35.43
+1.20D+1.60L	1	0.64	26.00	74.79	74.79	49.65	1.000	35.43
+1.20D+1.60L	1	0.80	26.00	73.40	73.40	61.50	1.000	35.43
+1.20D+1.60L	1	0.96	26.00	72.01	72.01	73.14	1.000	35.43
+1.20D+1.60L	1	1.12	26.00	70.62	70.62	84.55	1.000	35.43
+1.20D+1.60L	1	1.28	26.00	69.22	69.22	95.73	1.000	35.43
+1.20D+1.60L	1	1.44	26.00	67.83	67.83	106.70	1.000	35.43
+1.20D+1.60L	1	1.60	26.00	66.44	66.44	117.44	1.000	35.43
+1.20D+1.60L	1	1.76	26.00	65.05	65.05	127.96	1.000	35.43

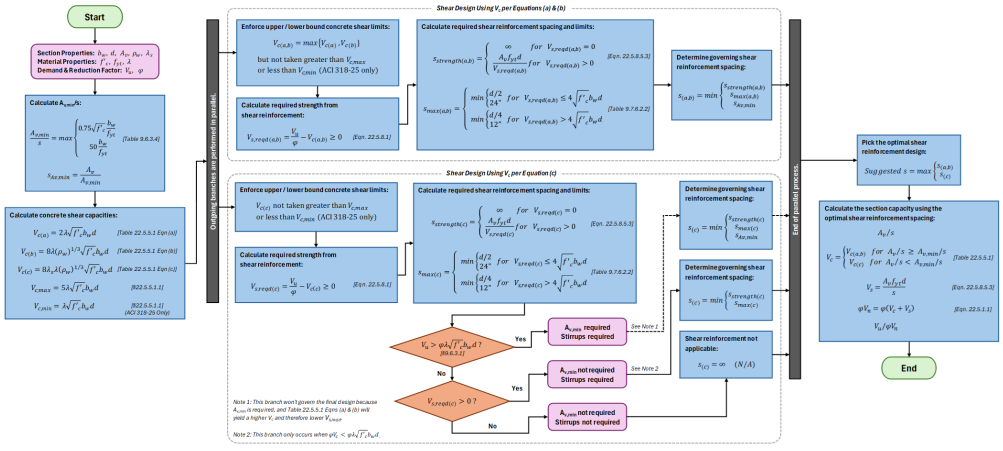
Shear Design Provisions for ACI 318-19 and beyond:

For ACI 318-19 and beyond, the provisions for determining nominal concrete shear strength (V_c) per §22.5.5.1 have been updated from previous versions of the ACI.

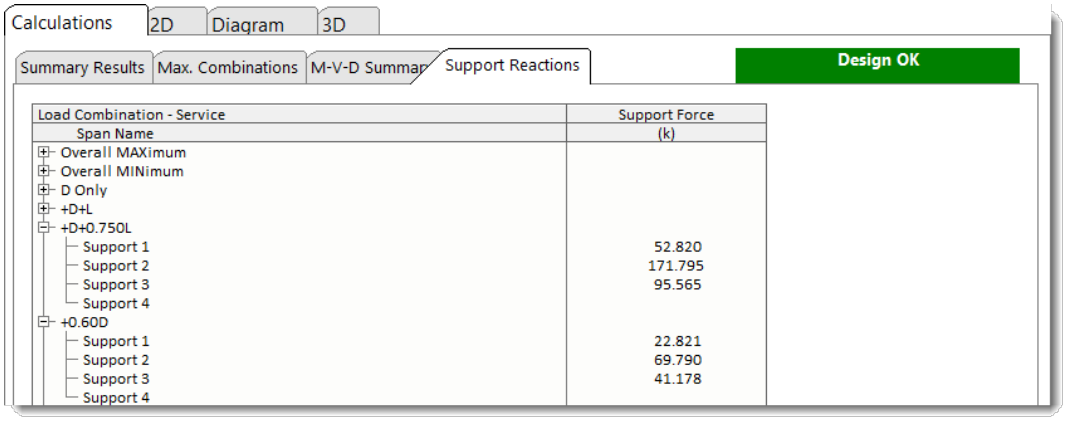
To select the appropriate equation for V_c from Table 22.5.5.1, it is necessary to know the relationship between the provided area of shear reinforcement (A_v) and $A_{v,min}$, where $A_{v,min}$ is calculated per §9.6.3.4 This creates a circular dependency since A_v is needed to determine V_c, but V_c is required to calculate the nominal shear strength due to shear reinforcement (V_s) and the corresponding A_v.

To address this circular logic, both scenarios outlined in Table 22.5.5.1 are evaluated by calculating V_c using equations (a, b) and (c). The required stirrup layout for each scenario is then determined, and the layout that produces the most favorable results is reported. Note, the Concrete Beam module only enforces $A_{v,min}$ where $V_u > \varphi\lambda\sqrt{f'_c}b_w d$ per §9.6.3.1.

This logic is outlined in the flowchart below:

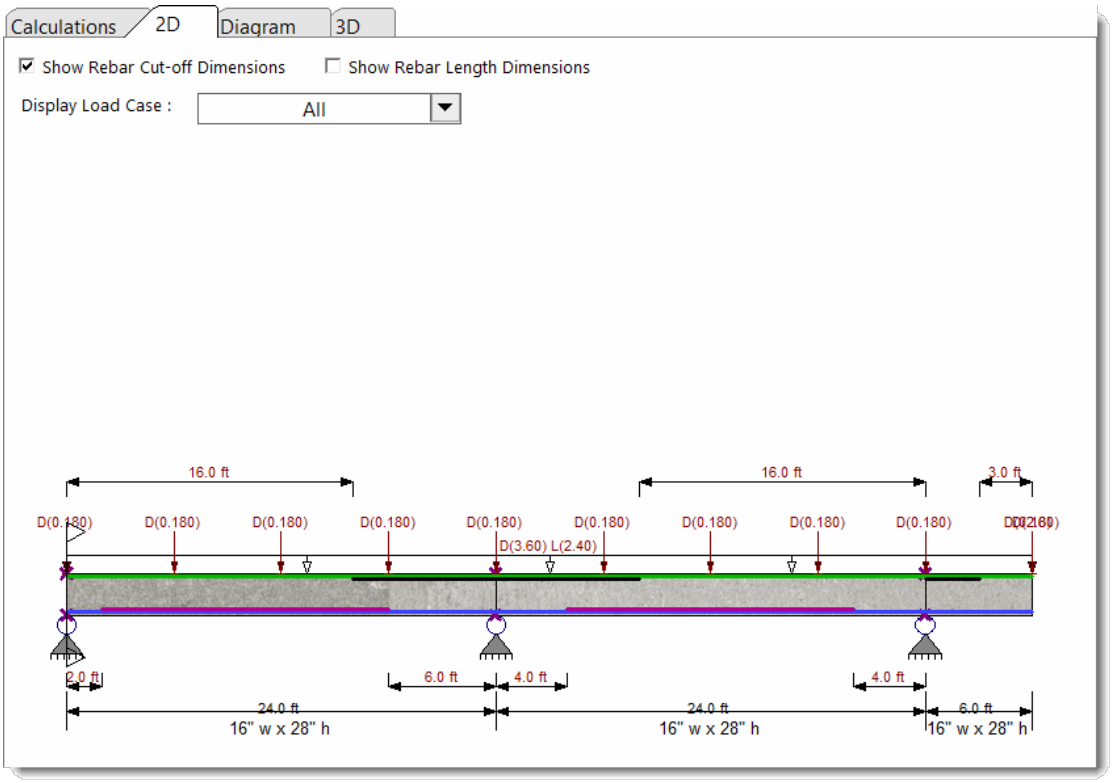


Support Reactions shows reactions for each support for each load condition.



2D Sketch

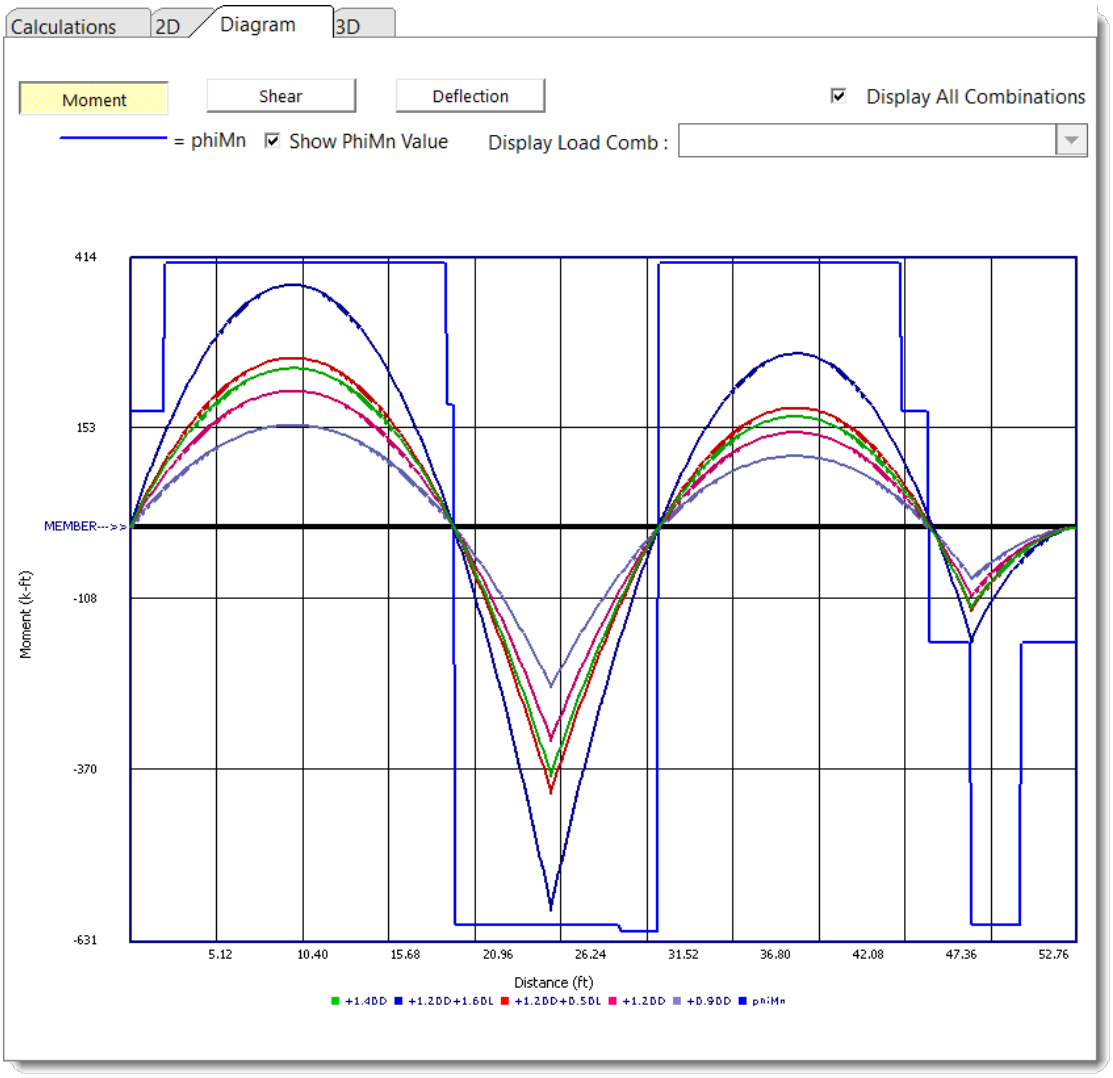
The 2D Sketch tab provides a graphic representation of the beam currently being designed:



Diagram

The Diagram tab offers the ability to view shear, moment, and deflection diagrams for selected load combinations, as well as a moment envelope:

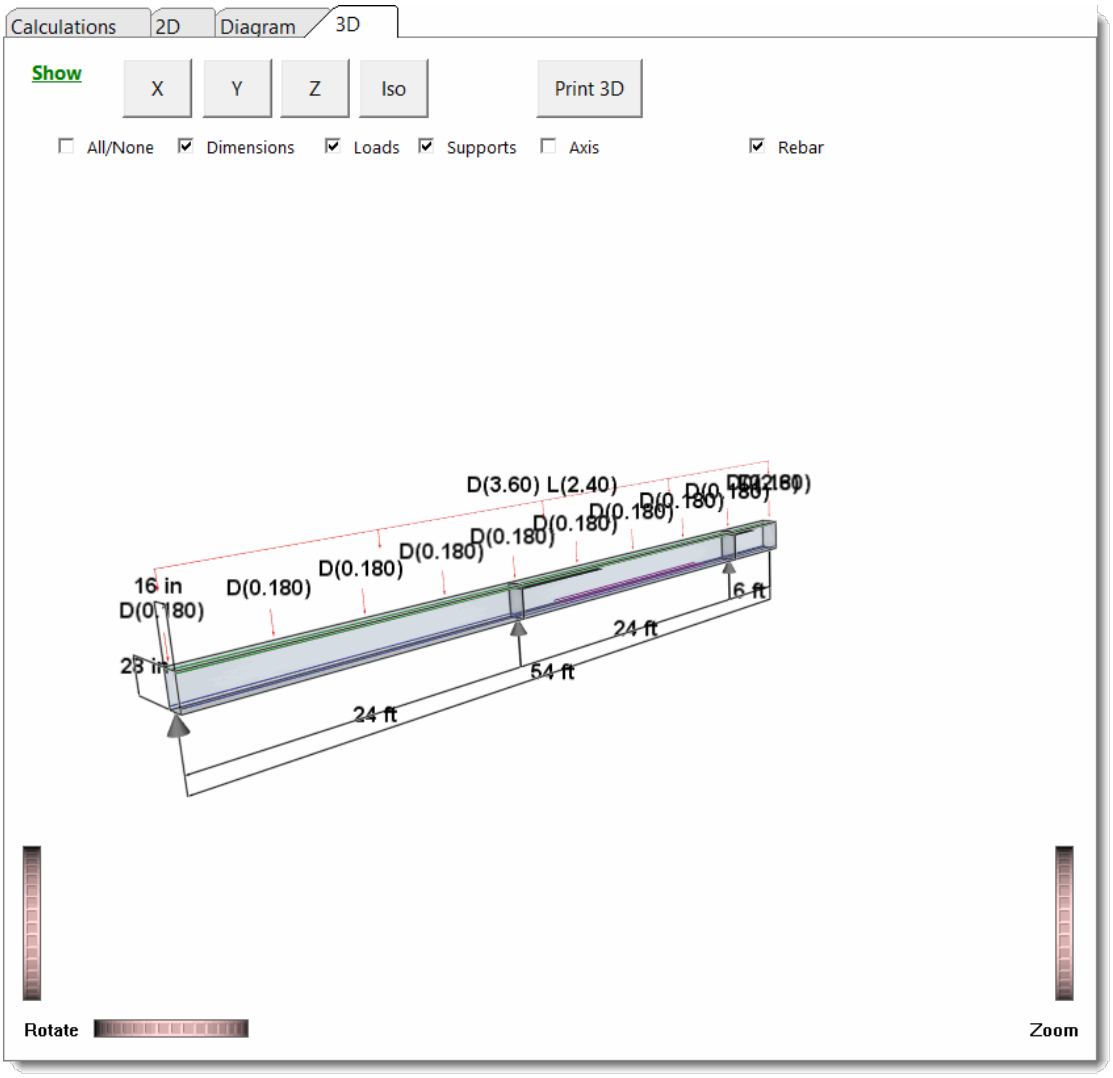
© 2026 ENERCALC, LLC



3D Rendering

The 3D Rendering tab offers a 3D view of the beam, with controls to display options:

© 2026 ENERCALC, LLC



13.4.6 Masonry Beam

Last revised: Tuesday, February 3, 2026 at 06:12 PM

This module provides analysis and design for masonry beams and lintels subject to vertical and lateral loads. Beams can have fixed or pinned ends for most typical conditions, and the user can specify rebar sets within the depth of the beam.

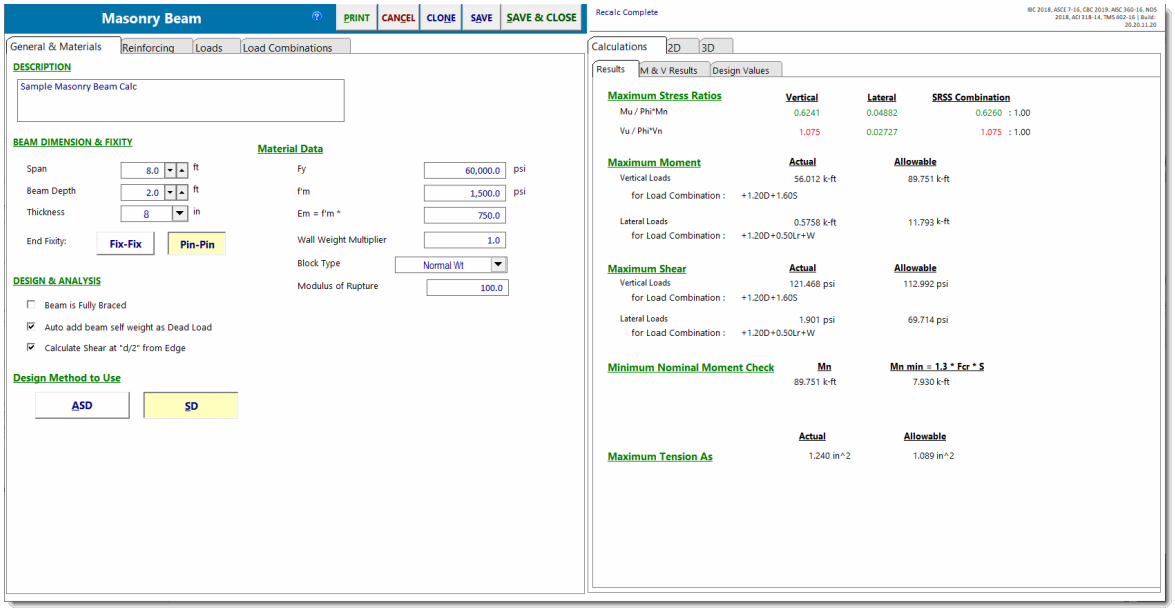
Vertical loads can be dead and live uniform and concentrated loads. A beam can have up to four loads of each type, and the uniform loads can be full or partial length.

The module also provides analysis for both seismic and wind loads. Wind load can be specified as well as seismic factors that apply to the beam's weight.

To allow the module to model different masonry block types, you can specify either lightweight or medium weight block, and additionally enter a self-weight multiplication factor.

© 2026 ENERCALC, LLC

For both the vertical and lateral bending and shear directions, the module calculates allowable bending moments and shear stresses. Also, for both directions, actual moments and shears due to all entered loads are calculated. Final results consist of combined stress ratio calculations for all combinations or dead, live, seismic, and wind vertical and lateral moments and shears.



Unique Features

- This module calculates all vertical and lateral moments and shears and combines them for all possible stress ratios. This provides a thorough evaluation of combined stresses for seismic and wind design.
- The module also provides the ability to modify the material weight, automatically reverse seismic and/or wind loadings, and model both fix-fix and pin-pin end fixity conditions.

Assumptions & Limitations

When the beam's fixity is set to Fixed, both vertical and lateral bending are considered fixed.

General & Materials

This tab provides data entry for the beam dimensions, material properties and lateral loads.

Span and **Lintel Depth** are used to calculate the beam bending and shears.

Thickness is the nominal masonry thickness. The true thickness is determined from internal tables.

End Fixity controls whether the analysis will consider the lintel to be pinned or fixed.

© 2026 ENERCALC, LLC

Beam is Fully Braced dictates whether the module checks the unbraced length or not.

Auto add beam self weight as Dead Load tells the module to calculate the member self-weight and add it as a uniform vertical load across the full span.

Calculate Shear at d/2 from edge controls the critical location for the shear check.

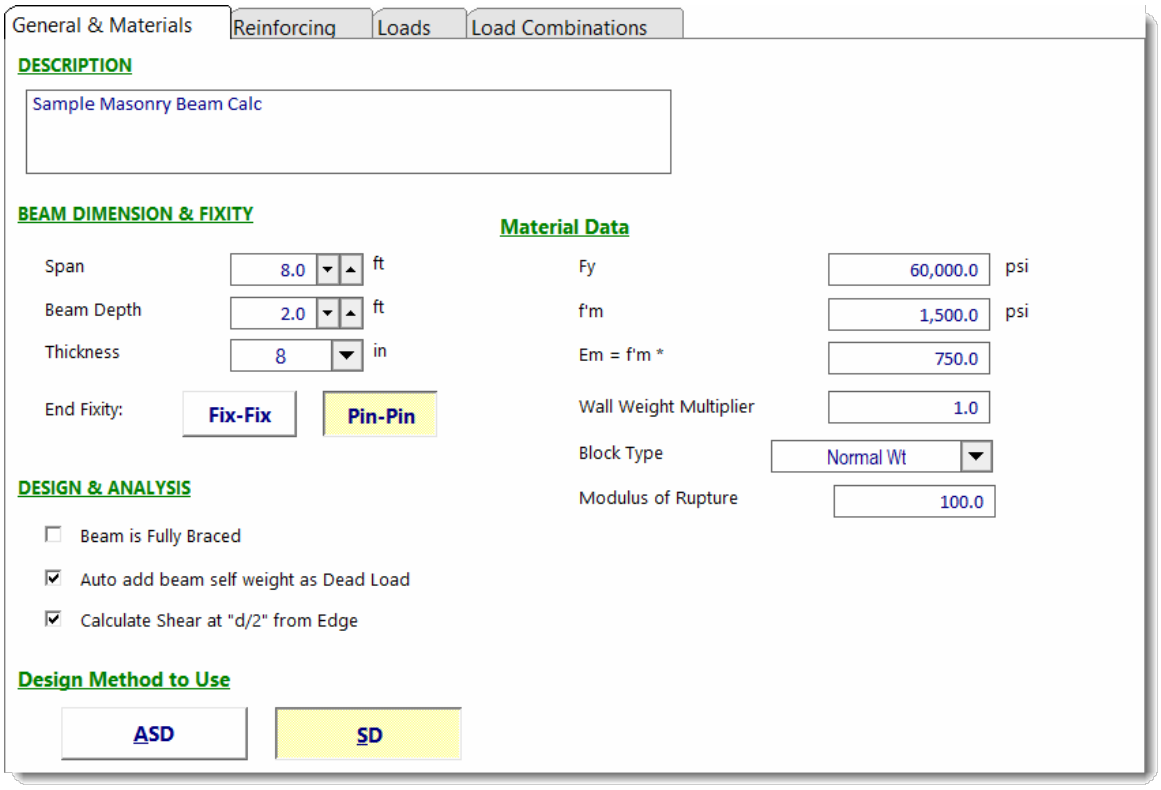
Material Data defines the allowable stresses for masonry and reinforcing steel.

Wall Weight Multiplier allows the user to factor the lintel self-weight that is pulled from internal tables.

Block Type selects the density of the CMU used for self-weight.

Modulus of Rupture collects MOR for masonry.

Design Method provides the option to select Allowable Stress Design or Strength Design.

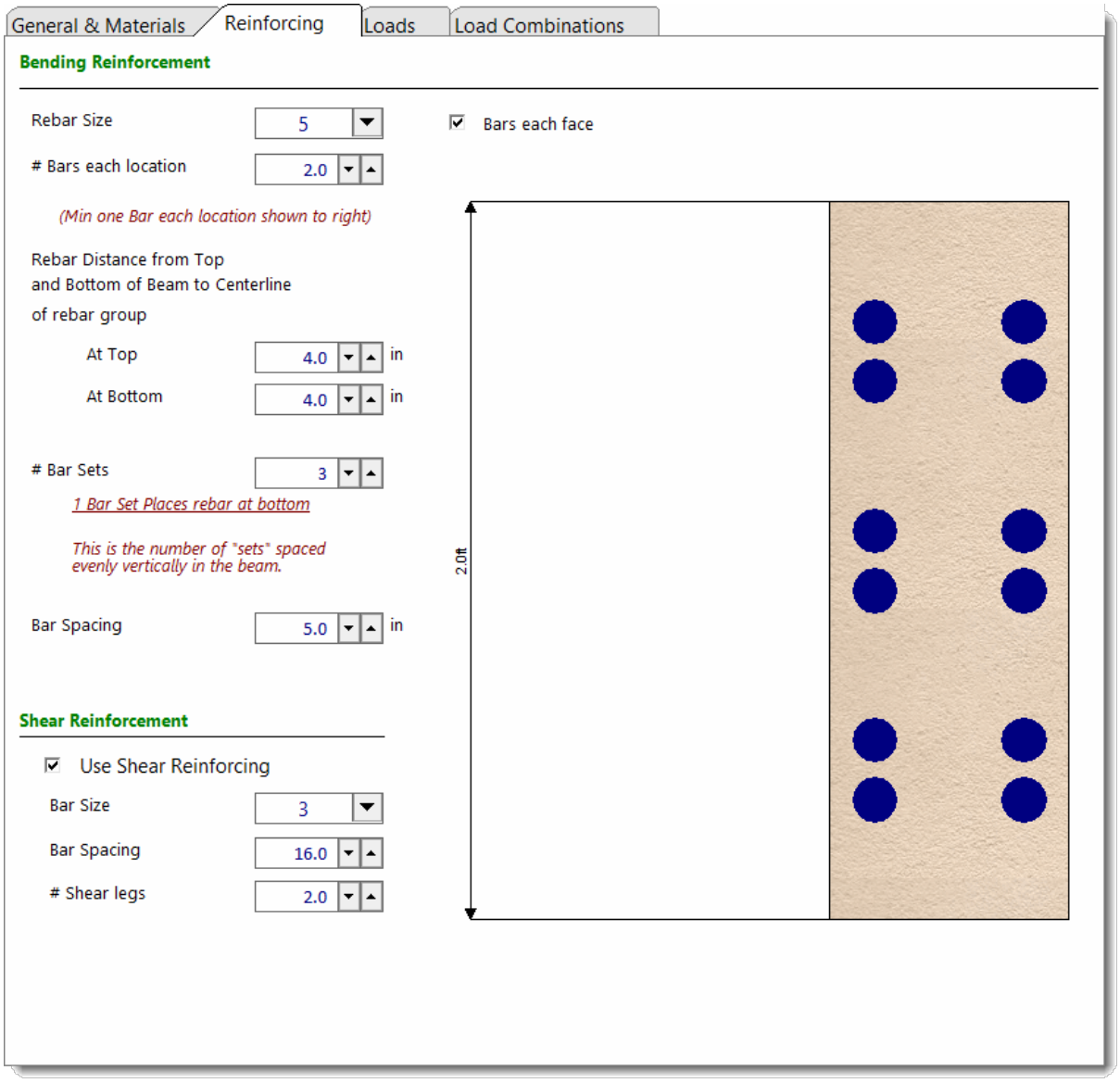


© 2026 ENERCALC, LLC

Reinforcing

This tab allows you to specify the longitudinal reinforcing in the beam.

Note: All longitudinal reinforcing bars are assumed to be fully developed. No attempt is made to compare the moment diagram with a longitudinal rebar development diagram, so engineering judgment should be applied in situations where end fixity is assumed and/or where heavy concentrated loads cause significant moments at locations close to the development zone of the provided reinforcement.



Rebar Size

Enter the rebar size used for all longitudinal bar sets to be used in the lintel.

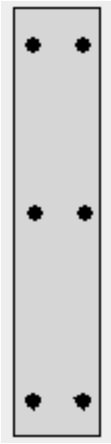
© 2026 ENERCALC, LLC

Bars Each Face (checkbox)

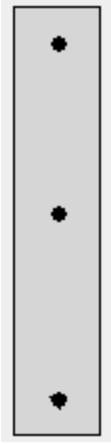
Selected implies that the lintel is reinforced with two layers of reinforcing separated by the value specified in the "Bar Spacing" input.

Deselected implies that the lintel is reinforced with one layer of reinforcing located at the middle of the width of the lintel.

The beam below **DOES** have bars each face:



The beam below does **NOT** have bars each face:



Bar Spacing

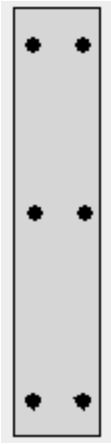
This is the clear distance between the bars on each face in a bar set. It is assumed that the bar set is centered within the width of the lintel. The value of "d" used for lateral bending strength calculations is calculated as: Actual Masonry Thickness - (Actual Masonry Thickness - Bar Spacing) / 2

© 2026 ENERCALC, LLC

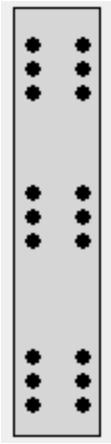
 # Bars Each Location

Enter the number of individual rebars to consider at each reinforced location.

The beam below has **one** bar at each reinforced location:



The beam below has **three** bars at each reinforced location:



Rebar Distance from Top & Bottom of Beam to Centerline of rebar group

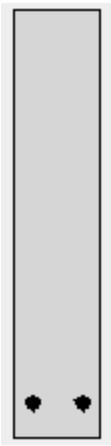
Distance from the top and bottom of the member to the center of area of the respective bar set. These distances will be used as "d" for vertical bending strength calculations.

© 2026 ENERCALC, LLC

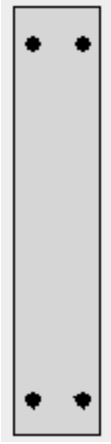
Bar Sets

Enter the number of bar sets in the lintel. A value of one indicates that the lintel has bottom reinforcing only.

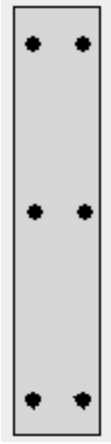
The beam below has **one** bar set:



The beam below has **two** bar sets:



The beam below has **three** bar sets:



© 2026 ENERCALC, LLC

Shear Reinforcement

When you click [**Yes**] you can specify the vertical shear reinforcement used in the lintel. The results will then reflect the allowable & actual shear stress ratios.

© 2026 ENERCALC, LLC

Loads

This tab allows you to specify all loads applied to the lintel.

Point Loads and Distributed Loads

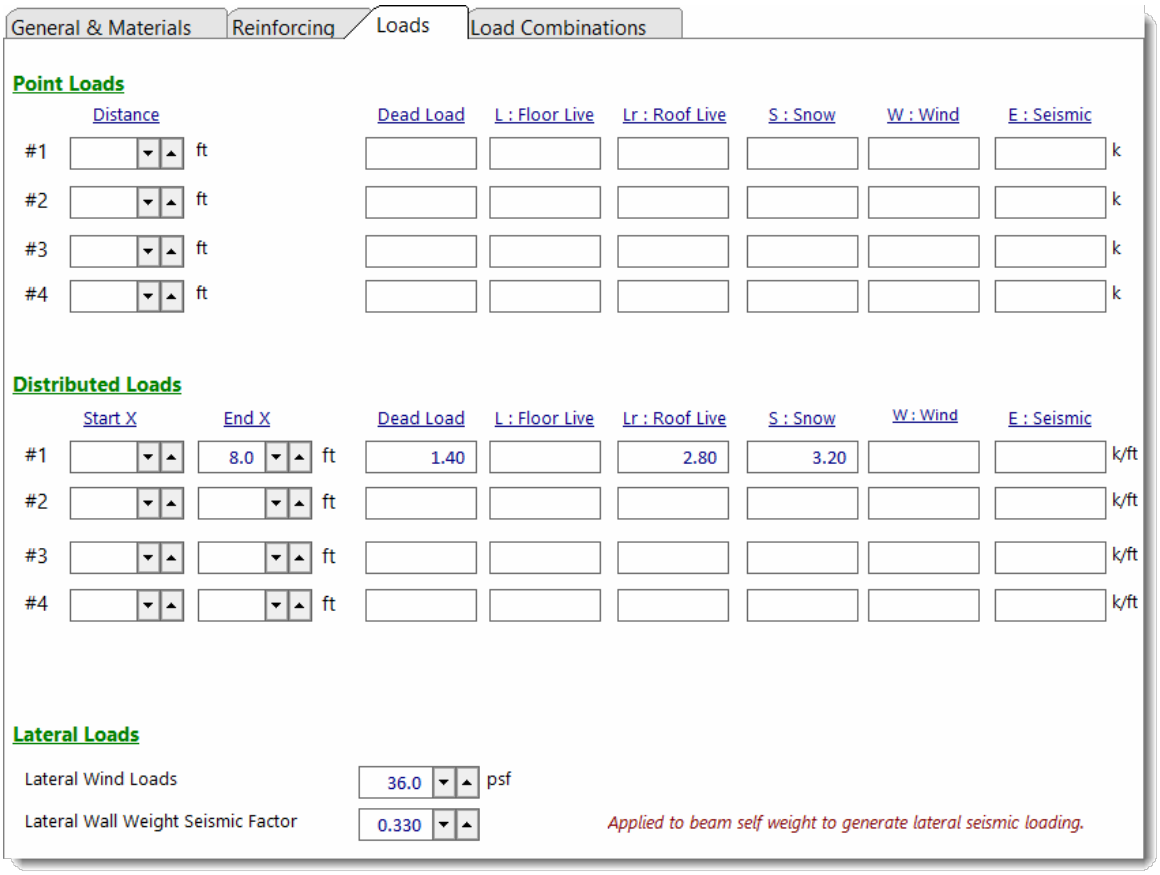
The Point Loads and Distributed Loads categories are assumed to be vertical loads.

Lateral Loads

The Lateral Loads category allows you to specify wind and seismic loads that are applied horizontally, perpendicular to the span of the lintel.

Lateral Wind Loads: This is a wind pressure which acts along the entire span length and depth of the beam. The resulting lateral demands are based on an equivalent uniform line load equal to the product of the wind pressure and the beam depth.

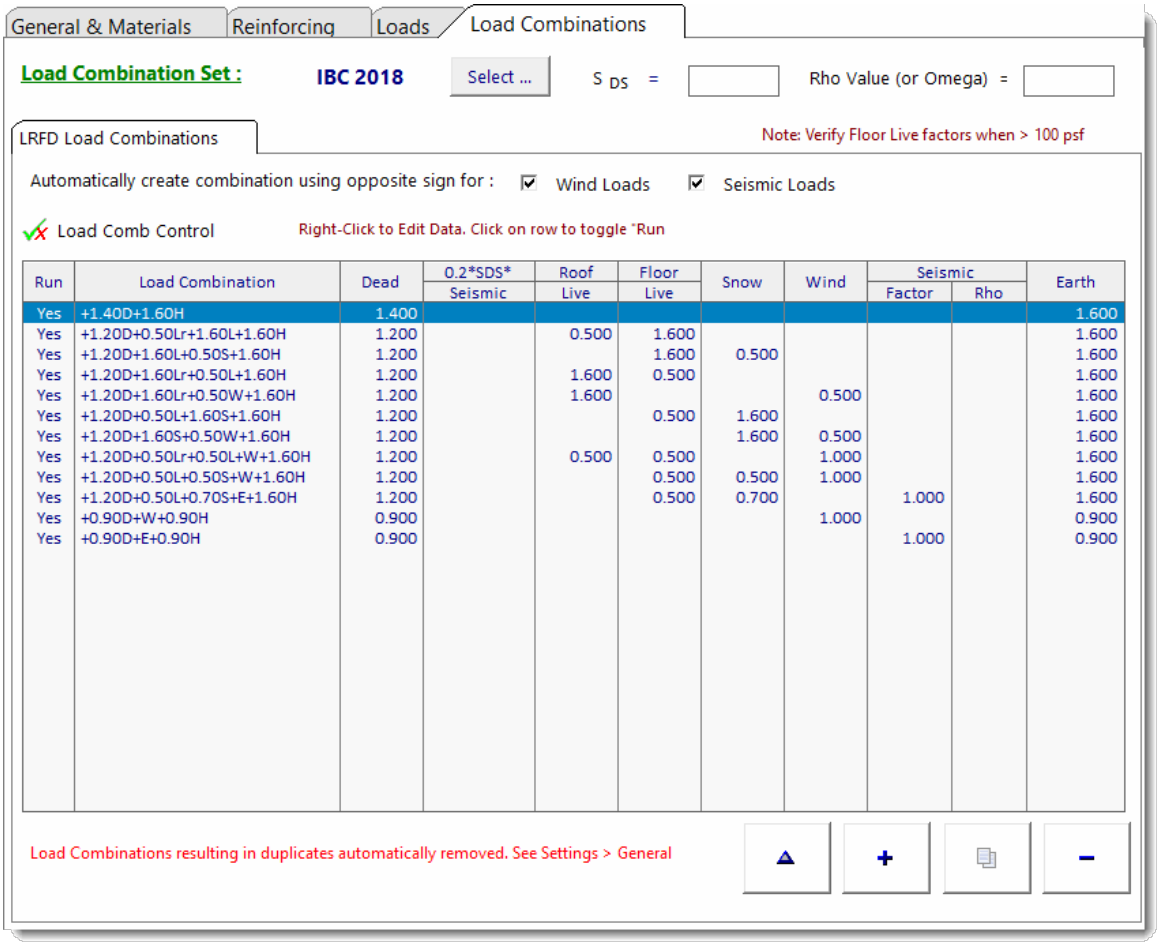
Lateral Wall Weight Seismic Weight Factor: This is a multiplier applied to the lintel's self-weight to create a laterally applied uniform load.



© 2026 ENERCALC, LLC

Load Combinations

This is the typical load combination tab used throughout **SEL**.



Run	Load Combination	Dead	0.2*SDS* Seismic	Roof Live	Floor Live	Snow	Wind	Seismic Factor	Seismic Rho	Earth
Yes	+1.40D+1.60H	1.400								1.600
Yes	+1.20D+0.50Lr+1.60L+1.60H	1.200		0.500	1.600					1.600
Yes	+1.20D+1.60L+0.50S+1.60H	1.200			1.600	0.500				1.600
Yes	+1.20D+1.60Lr+0.50L+1.60H	1.200		1.600	0.500					1.600
Yes	+1.20D+1.60Lr+0.50W+1.60H	1.200		1.600			0.500			1.600
Yes	+1.20D+0.50L+1.60S+1.60H	1.200			0.500	1.600				1.600
Yes	+1.20D+1.60S+0.50W+1.60H	1.200				1.600	0.500			1.600
Yes	+1.20D+0.50Lr+0.50L+W+1.60H	1.200		0.500	0.500		1.000			1.600
Yes	+1.20D+0.50L+0.50S+W+1.60H	1.200			0.500	0.500	1.000			1.600
Yes	+1.20D+0.50L+0.70S+E+1.60H	1.200			0.500	0.700		1.000		1.600
Yes	+0.90D+W+0.90H	0.900					1.000			0.900
Yes	+0.90D+E+0.90H	0.900						1.000		0.900

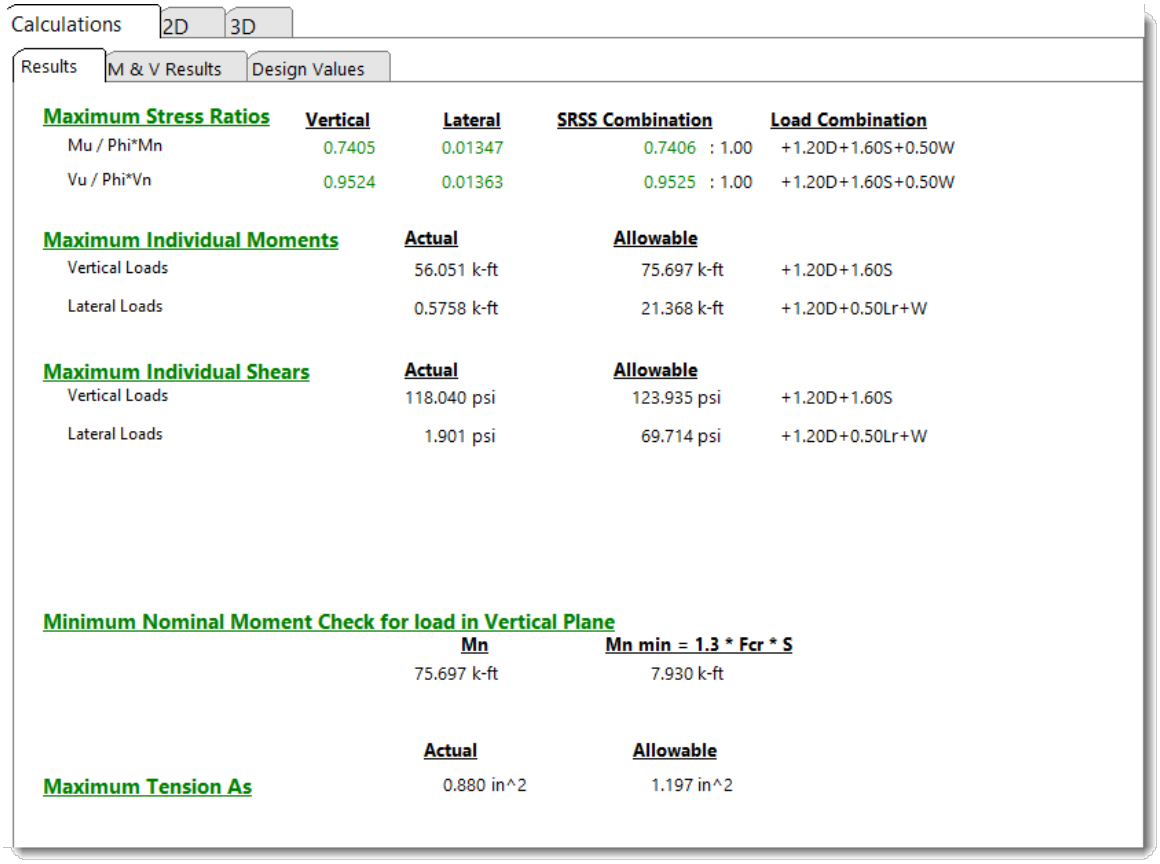
Output & Graphics Tabs

Results

This tab summarizes the calculated moments, shears and combined stress ratios for the lintel.

© 2026 ENERCALC, LLC

Ratios for bending and shear are provided for both vertical and lateral load applications as well as an SRSS calculation for combined vertical & lateral stresses from the controlling load combination.



Calculations | 2D | 3D

Results | M & V Results | Design Values

Maximum Stress Ratios

	Vertical	Lateral	SRSS Combination	Load Combination
Mu / Phi*Mn	0.7405	0.01347	0.7406 : 1.00	+1.20D+1.60S+0.50W
Vu / Phi*Vn	0.9524	0.01363	0.9525 : 1.00	+1.20D+1.60S+0.50W

Maximum Individual Moments

	Actual	Allowable	
Vertical Loads	56.051 k-ft	75.697 k-ft	+1.20D+1.60S
Lateral Loads	0.5758 k-ft	21.368 k-ft	+1.20D+0.50Lr+W

Maximum Individual Shears

	Actual	Allowable	
Vertical Loads	118.040 psi	123.935 psi	+1.20D+1.60S
Lateral Loads	1.901 psi	69.714 psi	+1.20D+0.50Lr+W

Minimum Nominal Moment Check for load in Vertical Plane

Mn	Mn min = 1.3 * Fcr * S
75.697 k-ft	7.930 k-ft

	Actual	Allowable
Maximum Tension As	0.880 in^2	1.197 in^2

The "Allowable Maximum Tension As" represents the maximum permissible cross-sectional area of flexural tensile reinforcement to ensure the net tensile strain (ε_t) limits specified by the TMS design standard are satisfied:

TMS 402-16: $\varepsilon_t = 1.5 * f_y / E_s$ *[§ 9.3.3.2.1(a)]*

TMS 402-22: $\varepsilon_t = 0.003 + f_y / E_s$ *[§ 9.3.3.2.4 and Table 9.1.4]*

Using the principles of similar triangles based on the elastic strain distribution, the value for $A_{s,max}$ can be solved for directly.

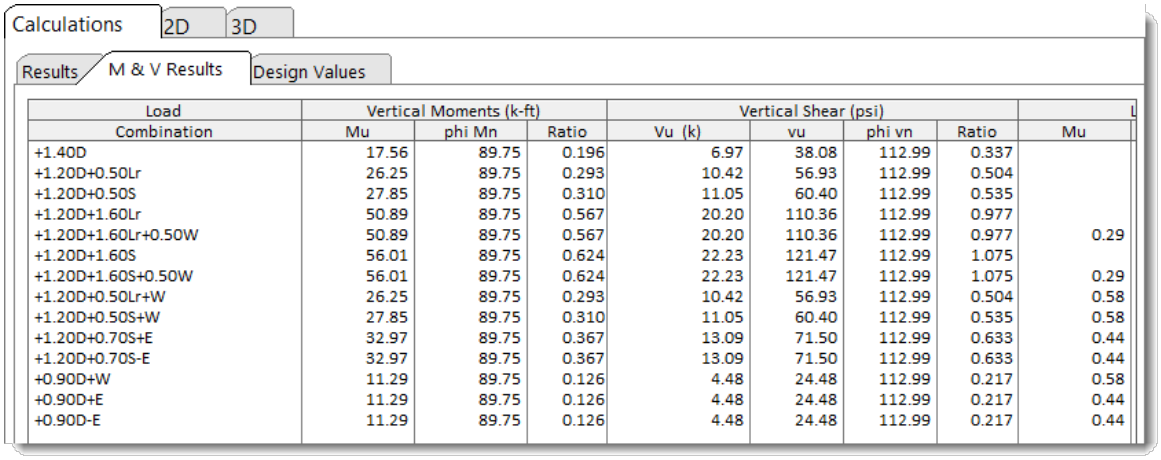
$$a = 0.8 * d * \frac{0.0025}{\varepsilon_t + 0.0025}$$

$$A_{s,max} = 0.8 f'_m a b / f_y$$

© 2026 ENERCALC, LLC

M & V Results

This tab provides full details for all actual and allowable stresses for all load combinations.



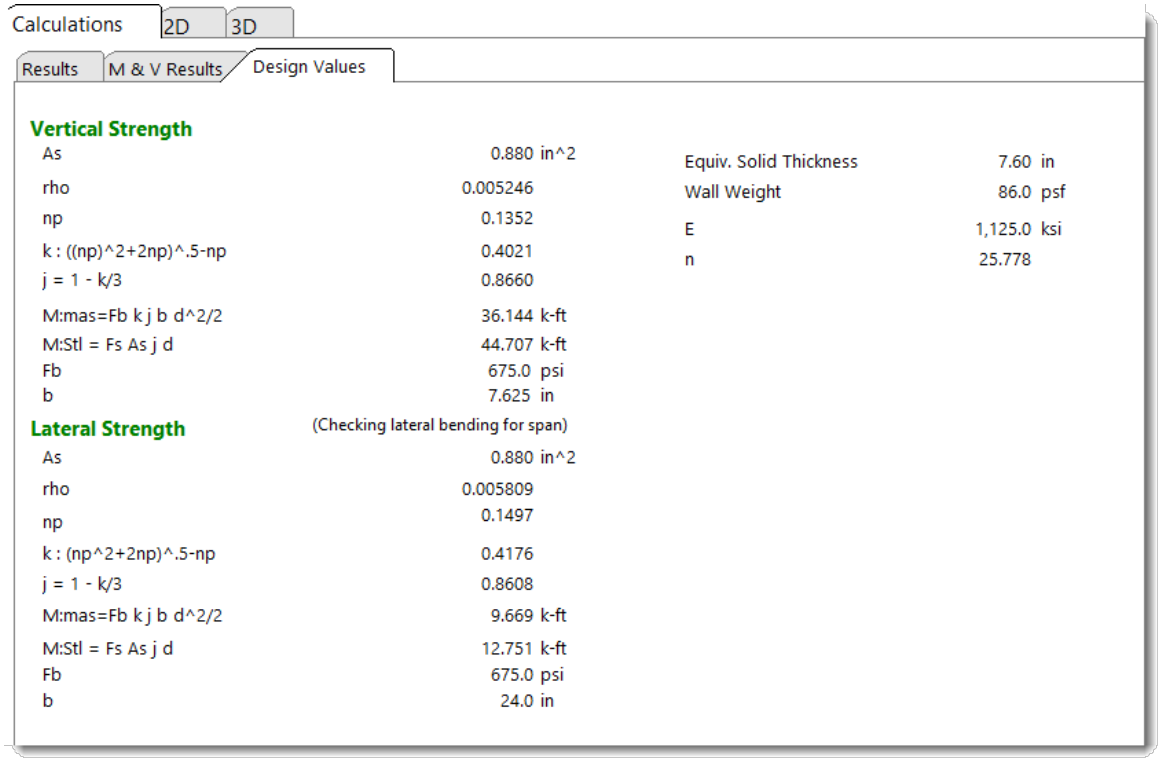
Calculations | 2D | 3D

Results | M & V Results | Design Values

Load Combination	Vertical Moments (k-ft) Mu	phi Mn	Ratio	Vertical Shear (psi) Vu (k)	vu	phi vn	Ratio	L Mu
+1.40D	17.56	89.75	0.196	6.97	38.08	112.99	0.337	
+1.20D+0.50Lr	26.25	89.75	0.293	10.42	56.93	112.99	0.504	
+1.20D+0.50S	27.85	89.75	0.310	11.05	60.40	112.99	0.535	
+1.20D+1.60Lr	50.89	89.75	0.567	20.20	110.36	112.99	0.977	
+1.20D+1.60Lr+0.50W	50.89	89.75	0.567	20.20	110.36	112.99	0.977	0.29
+1.20D+1.60S	56.01	89.75	0.624	22.23	121.47	112.99	1.075	
+1.20D+1.60S+0.50W	56.01	89.75	0.624	22.23	121.47	112.99	1.075	0.29
+1.20D+0.50Lr+W	26.25	89.75	0.293	10.42	56.93	112.99	0.504	0.58
+1.20D+0.50S+W	27.85	89.75	0.310	11.05	60.40	112.99	0.535	0.58
+1.20D+0.70S+E	32.97	89.75	0.367	13.09	71.50	112.99	0.633	0.44
+1.20D+0.70S-E	32.97	89.75	0.367	13.09	71.50	112.99	0.633	0.44
+0.90D+W	11.29	89.75	0.126	4.48	24.48	112.99	0.217	0.58
+0.90D+E	11.29	89.75	0.126	4.48	24.48	112.99	0.217	0.44
+0.90D-E	11.29	89.75	0.126	4.48	24.48	112.99	0.217	0.44

Design Values Tab

This tab summarizes allowable stress calculations. When ASD is used, the tab looks like this:



Calculations | 2D | 3D

Results | M & V Results | Design Values

Vertical Strength

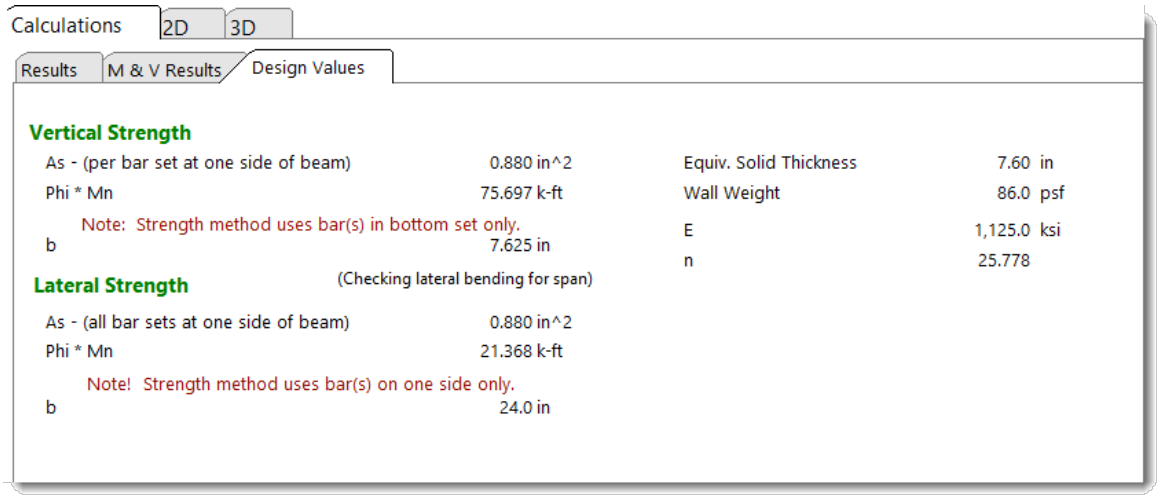
As	0.880	in^2	Equiv. Solid Thickness	7.60	in
rho	0.005246		Wall Weight	86.0	psf
np	0.1352		E	1,125.0	ksi
k : ((np)^2+2np)^.5-np	0.4021		n	25.778	
j = 1 - k/3	0.8660				
M:mas=Fb k j b d^2/2	36.144	k-ft			
M:Stl = Fs As j d	44.707	k-ft			
Fb	675.0	psi			
b	7.625	in			

Lateral Strength (Checking lateral bending for span)

As	0.880	in^2
rho	0.005809	
np	0.1497	
k : (np^2+2np)^.5-np	0.4176	
j = 1 - k/3	0.8608	
M:mas=Fb k j b d^2/2	9.669	k-ft
M:Stl = Fs As j d	12.751	k-ft
Fb	675.0	psi
b	24.0	in

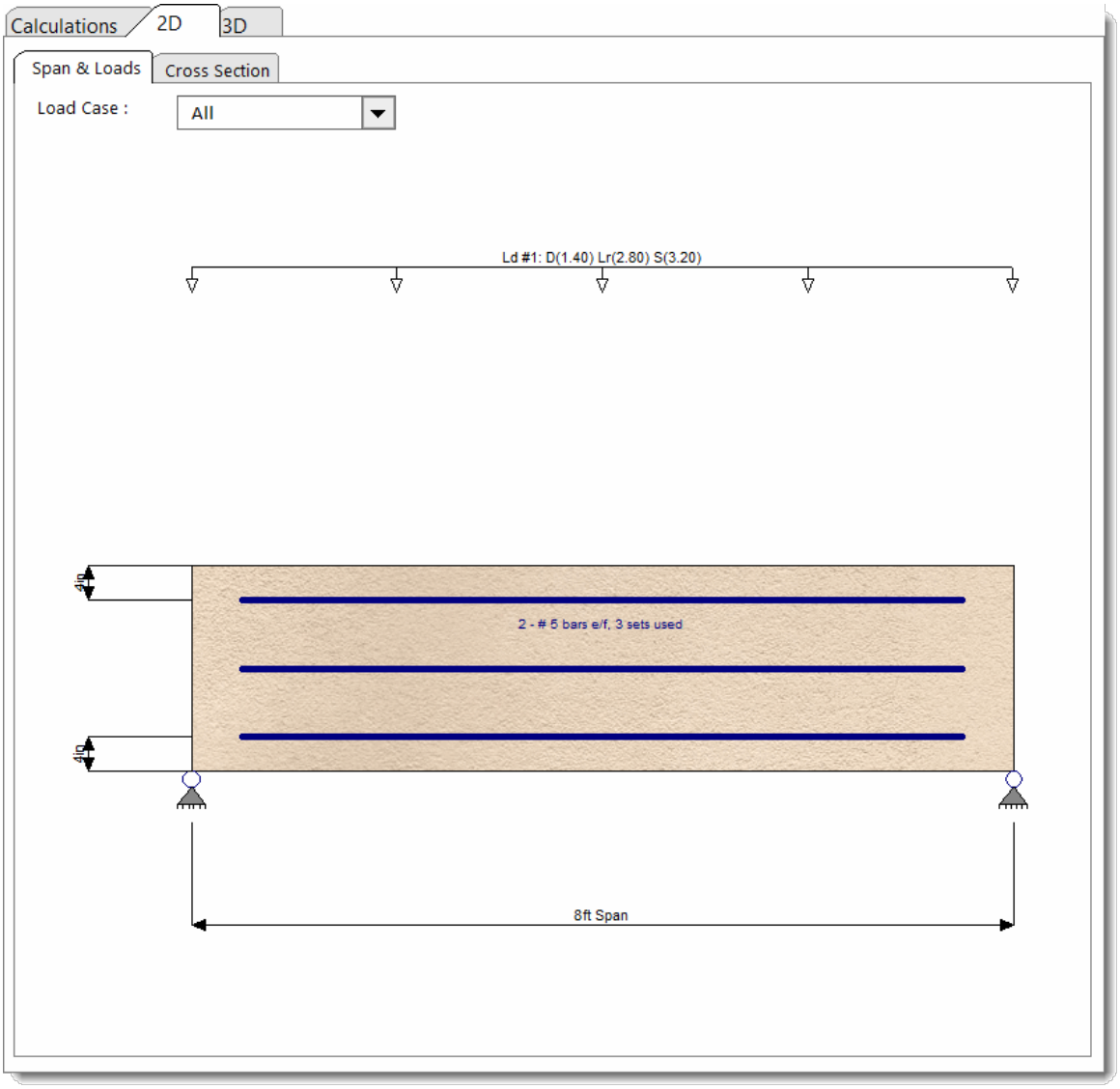
© 2026 ENERCALC, LLC

And when SD is used, the tab looks like this:

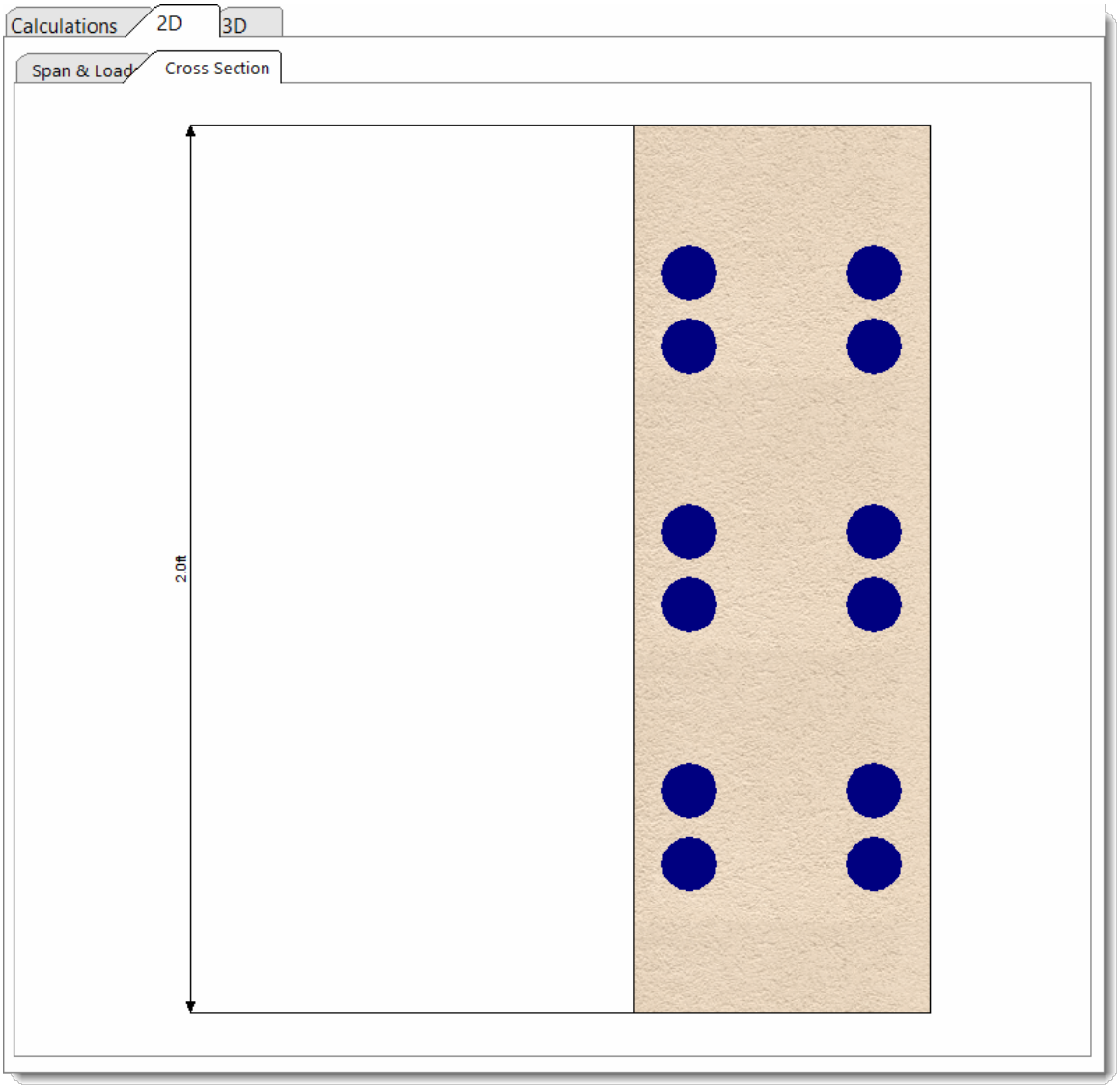


© 2026 ENERCALC, LLC

2D Sketch (Span & Loads subtab)

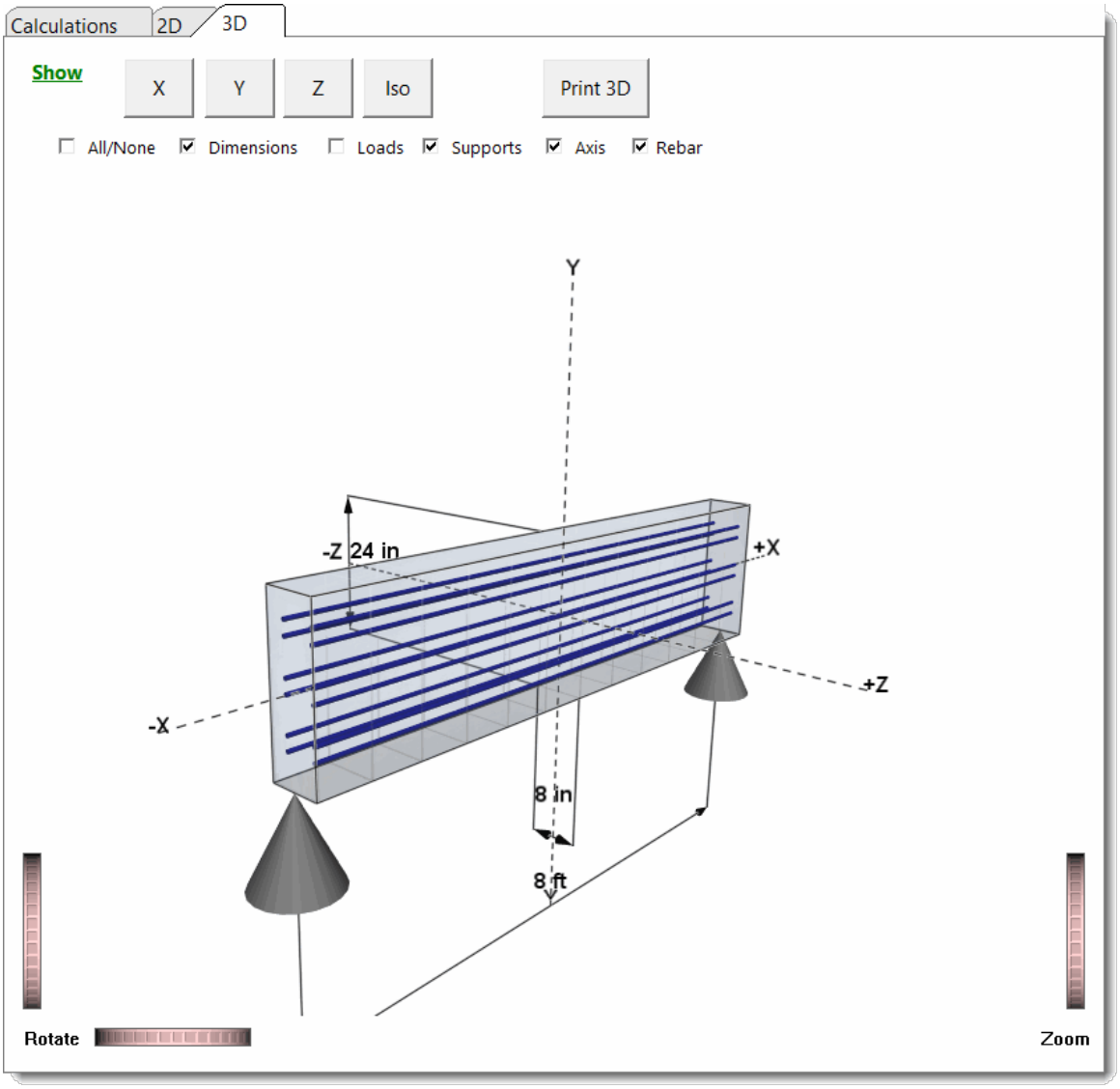


2D Sketch (Cross Section subtab)



© 2026 ENERCALC, LLC

3D Rendering



13.4.7 Wood Beam

Last revised: Tuesday, February 3, 2026 at 06:12 PM

Verification Example

In this section, for each input tab we will review only the items that are unique to the WOOD material type.

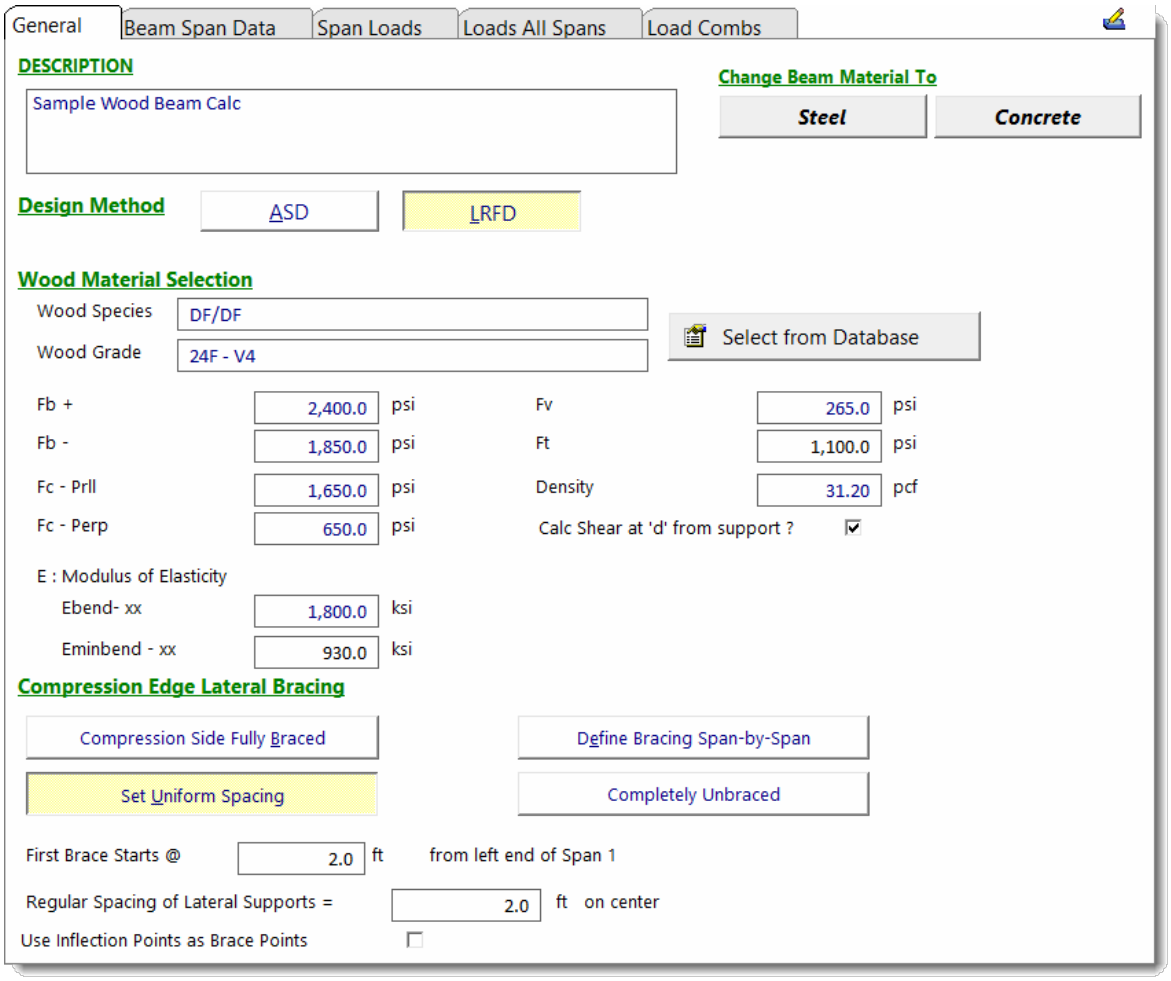
For general information on the typical data input for all beams see the Beams 684 topic.

This module offers complete design of single and multi-span wood members. Among its capabilities are:

- Single or multi-span (up to 8-span) beams.
- End fixity can be pinned, fixed, free or a combination thereof.
- Analysis is according to NDS.
- ASD or LRFD design methods can be selected. Values of K_F and phi are automatically determined and applied for the LRFD method.
- A complete wood section database is provided. This includes sawn, glued-laminated and selected engineered wood products.
- A complete wood species database is provided. All values are per the latest NDS.
- Unbraced compression edge lengths can be specified in a variety of ways.
- Automatic member selection is provided.
- You can specify values for C_M, C_t and C_r.
- C_F or C_V is automatically provided. In the case of C_F, the value is also based on species stress grade. Note: 2018 NDS allows Cv to be greater than OR less than 1.0 for EWP. But some EWP manufacturers limit their values of Cv to be <= 1.0. So as of January 12, 2023, ENERCALC SEL conservatively limits the value of Cv to be <= 1.0 for EWPs.

General

© 2026 ENERCALC, LLC



Beam Material

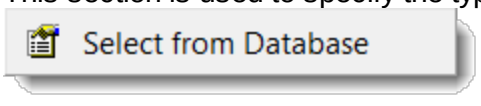
Clicking one of these buttons changes the material type used for the beam.

Design Method

For wood & steel you can select ASD or LRFD design methods. Concrete design always uses ultimate strength design (LRFD).

Design Values

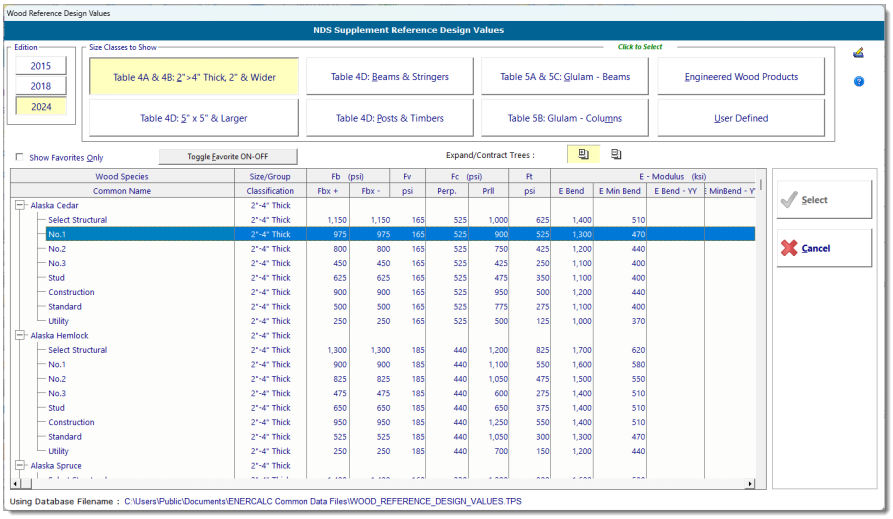
This section is used to specify the type of wood that will be used. Use the



button to access the standard wood reference design values database and select a material.

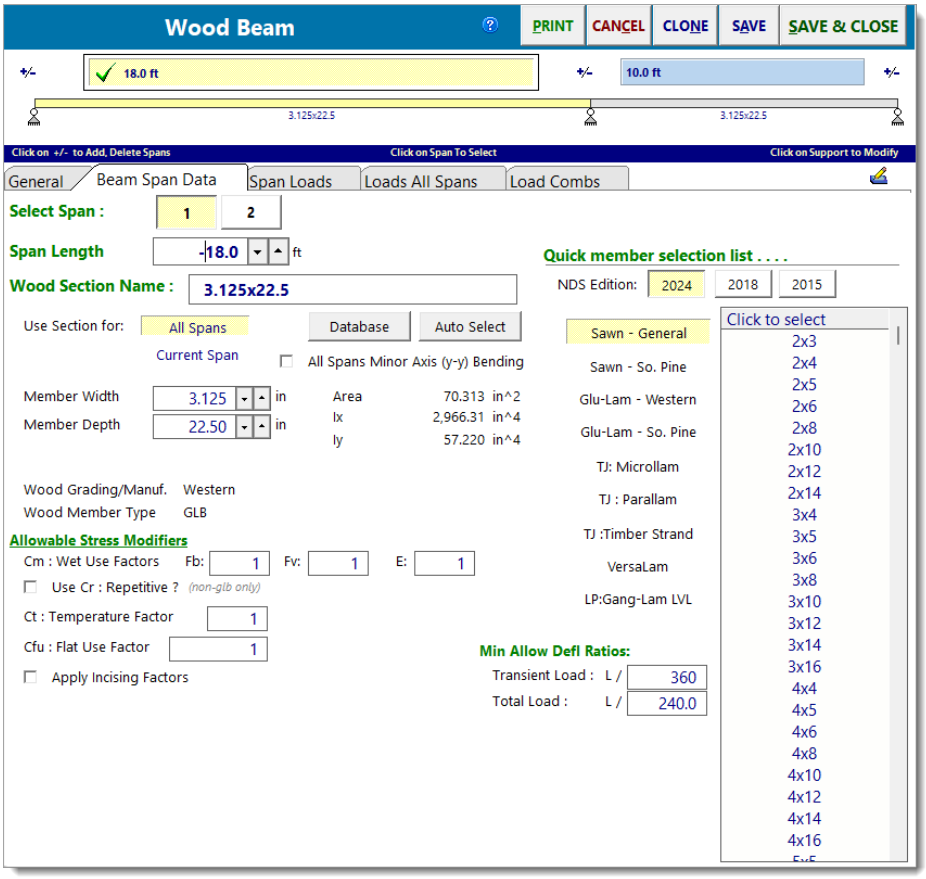
© 2026 ENERCALC, LLC

These values can be edited right on the screen. HOWEVER there are other pieces of information, such as size factors for certain sizes of members, that are stored separately.



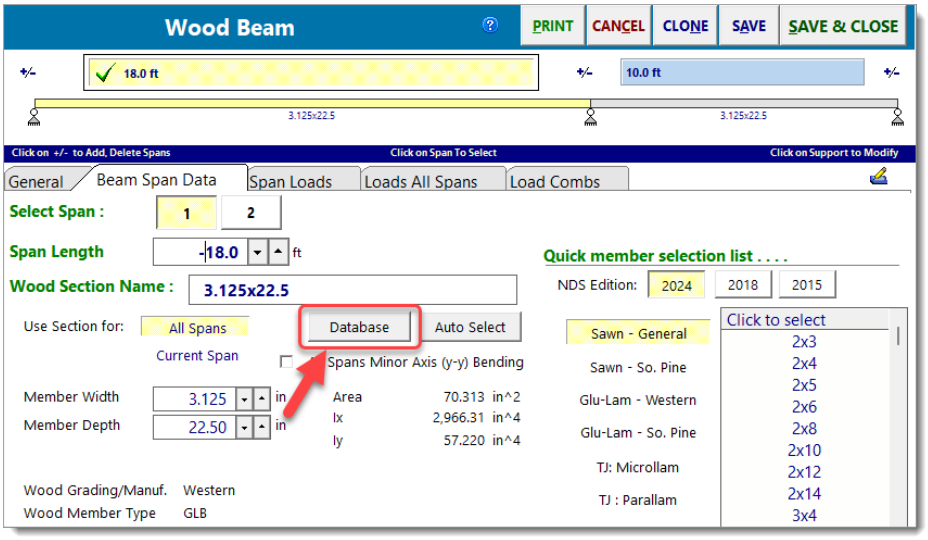
Beam Span Data

This tab is used to define span length and section information for the beam:

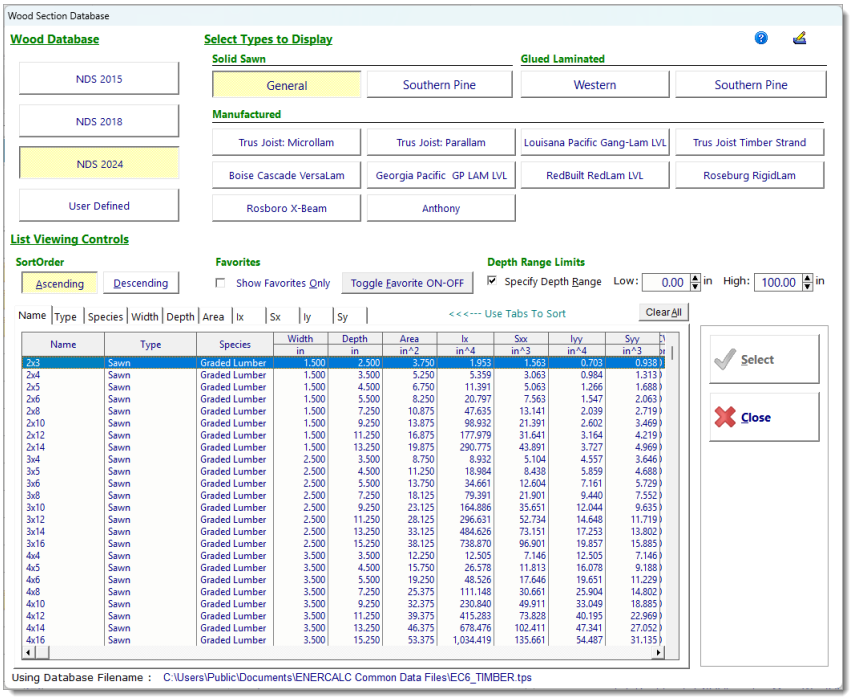


© 2026 ENERCALC, LLC

The button indicated in the screen capture below is used to display the Wood Section Database.

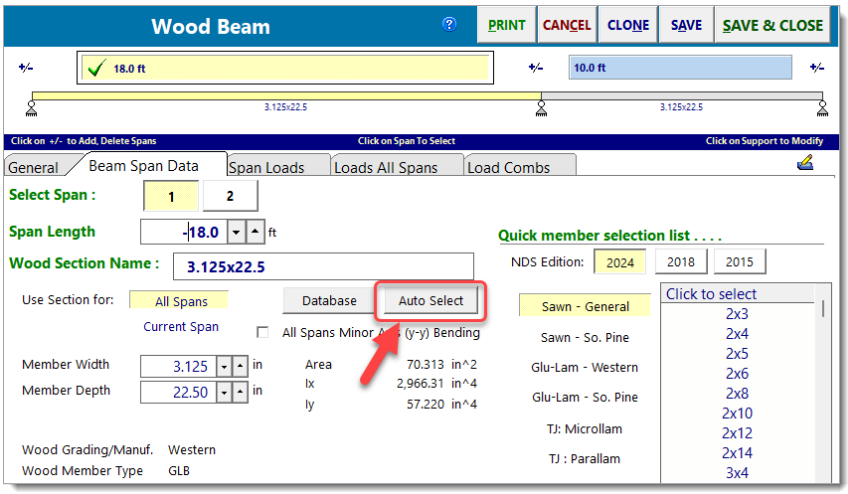


The Wood Section Database contains an extensive number of solid-sawn members, glulam members, and engineered wood products commonly used in the USA.

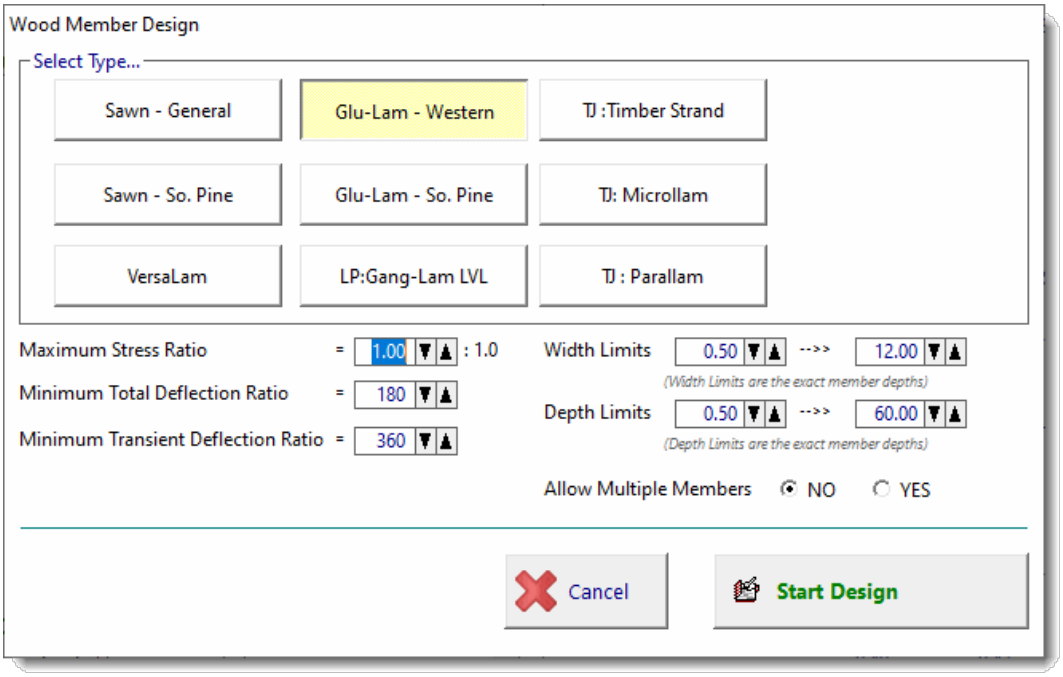


© 2026 ENERCALC, LLC

The button shown bubbled in the screen capture below is used to display the Wood Member Design dialog.



The Wood Member Design dialog allows you to choose the type of member to be selected and to specify limits on the permissible stress ratio, deflection ratio and selected member size.



Note: The factor named Maximum Stress Ratio does not act as a multiplier on the specified deflection ratios.

Span Loads

No differences from other materials.

© 2026 ENERCALC, LLC

Loads All Spans

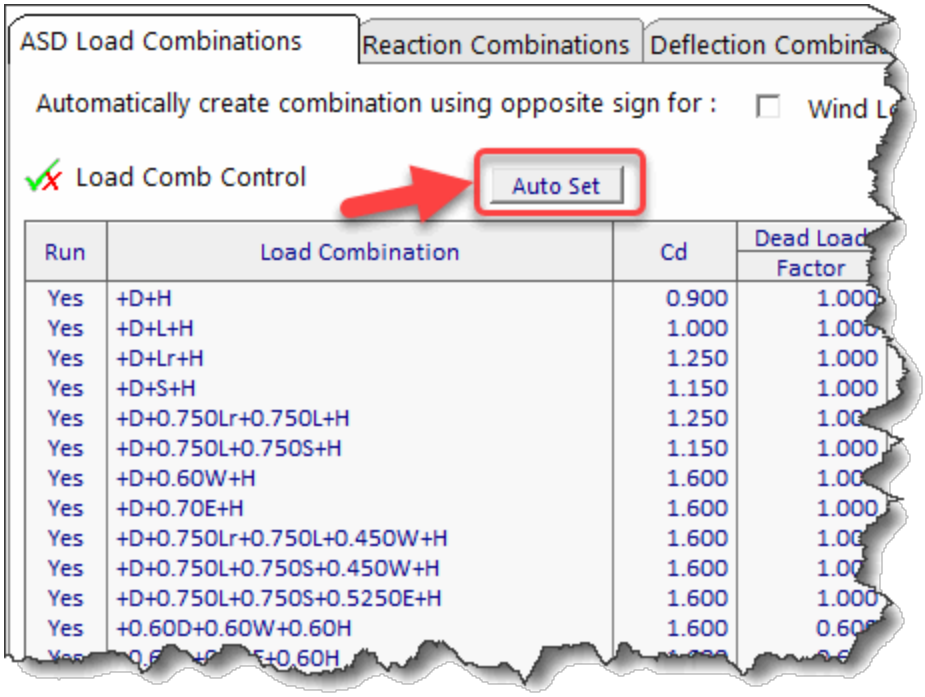
No differences from other materials.

Load Combinations

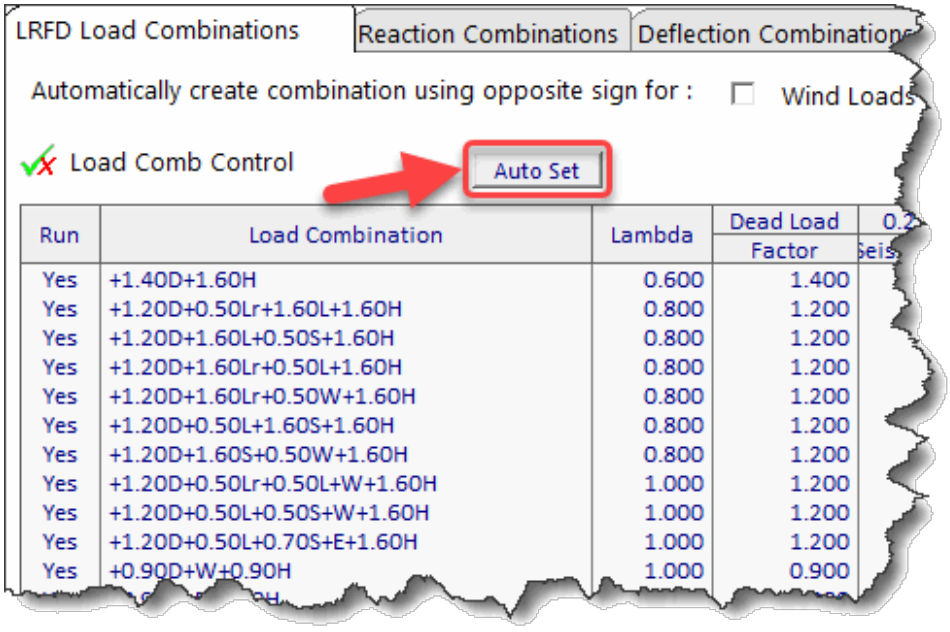
For wood members you will see entries for load duration factors. When ASD is used, the Load Duration Factor is referred to as C_D. When LRFD is used, the Load Duration Factor is referred to as λ.

© 2026 ENERCALC, LLC

Note that C_D and λ can be automatically set for all load combinations by clicking the Auto Set button button at the top of the column of values. When that button is clicked, the program will automatically determine the proper value for C_D or λ according to the NDS based on the shortest duration load type included in each of the load combinations.



Run	Load Combination	Cd	Dead Load Factor
Yes	+D+H	0.900	1.000
Yes	+D+L+H	1.000	1.000
Yes	+D+Lr+H	1.250	1.000
Yes	+D+S+H	1.150	1.000
Yes	+D+0.750Lr+0.750L+H	1.250	1.00
Yes	+D+0.750L+0.750S+H	1.150	1.000
Yes	+D+0.60W+H	1.600	1.00
Yes	+D+0.70E+H	1.600	1.000
Yes	+D+0.750Lr+0.750L+0.450W+H	1.600	1.00
Yes	+D+0.750L+0.750S+0.450W+H	1.600	1.00
Yes	+D+0.750L+0.750S+0.5250E+H	1.600	1.000
Yes	+0.60D+0.60W+0.60H	1.600	0.60



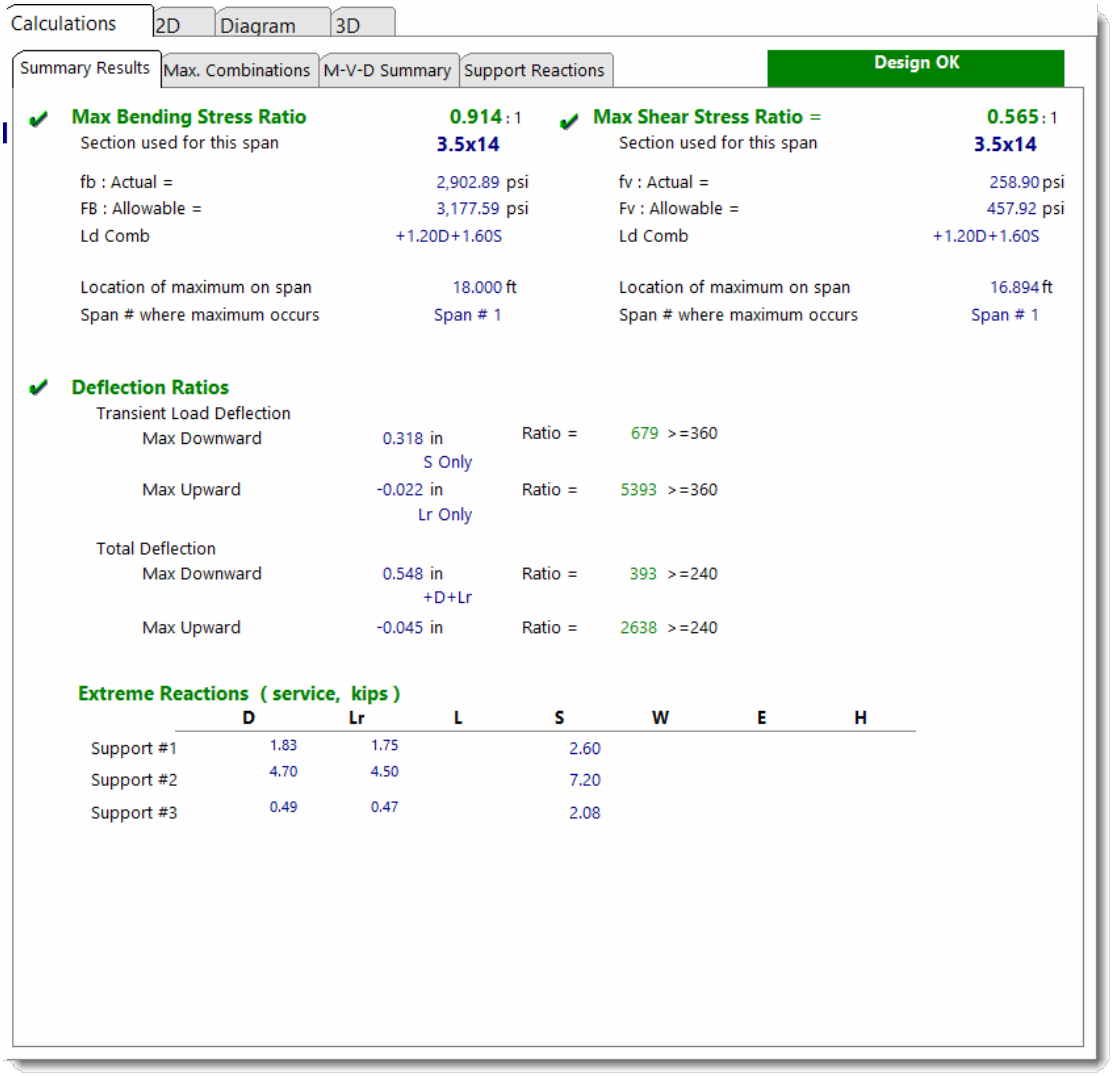
Run	Load Combination	Lambda	Dead Load Factor	0.2 Seis
Yes	+1.40D+1.60H	0.600	1.400	
Yes	+1.20D+0.50Lr+1.60L+1.60H	0.800	1.200	
Yes	+1.20D+1.60L+0.50S+1.60H	0.800	1.200	
Yes	+1.20D+1.60Lr+0.50L+1.60H	0.800	1.200	
Yes	+1.20D+1.60Lr+0.50W+1.60H	0.800	1.200	
Yes	+1.20D+0.50L+1.60S+1.60H	0.800	1.200	
Yes	+1.20D+1.60S+0.50W+1.60H	0.800	1.200	
Yes	+1.20D+0.50Lr+0.50L+W+1.60H	1.000	1.200	
Yes	+1.20D+0.50L+0.50S+W+1.60H	1.000	1.200	
Yes	+1.20D+0.50L+0.70S+E+1.60H	1.000	1.200	
Yes	+0.90D+W+0.90H	1.000	0.900	

© 2026 ENERCALC, LLC

Results Tabs: This set of tabs provides detailed results for the current calculation. The tabs in the upper right-hand corner of the screen allow you to select the major areas available for review: Calculations, 2D Sketch, Diagram and 3D Rendering.

The Calculations tab offers the following results options:

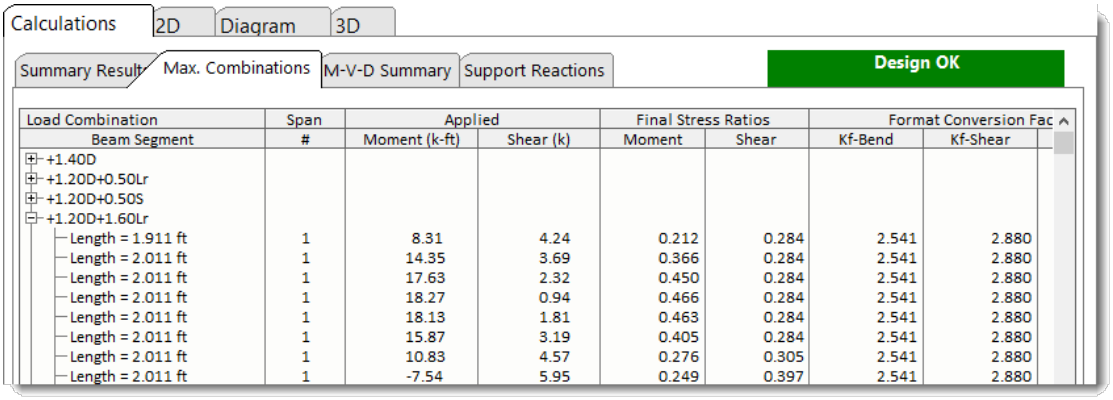
Summary Results provides details for shear, moment and deflection for the governing load combinations.



© 2026 ENERCALC, LLC

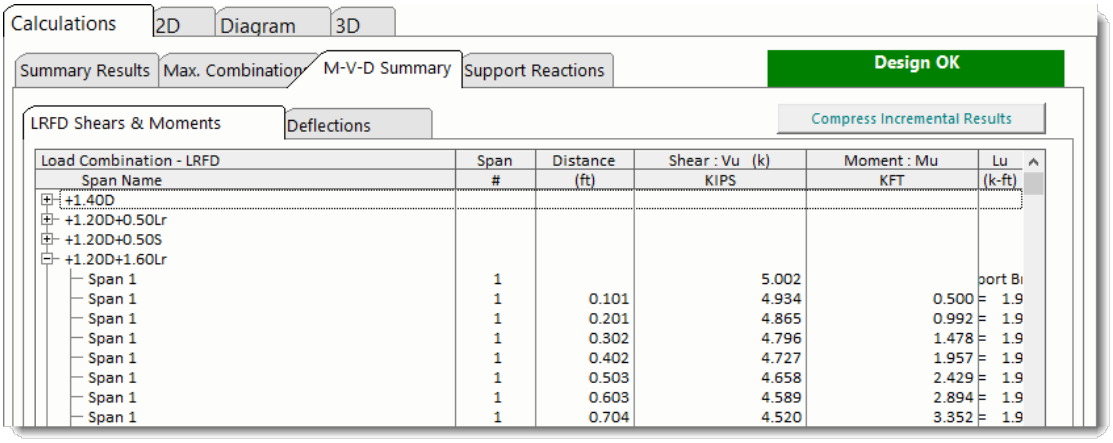
Max. Combinations provides detailed results for each beam segment for each load combination. The leftmost column lists the load combinations and the unbraced length being considered.

These results are a consolidation of the highly detailed incremental results presented on the M-V-D Summary tab.



Load Combination / Beam Segment	Span #	Applied Moment (k-ft)	Applied Shear (k)	Final Stress Ratios Moment	Final Stress Ratios Shear	Format Conversion Fac Kf-Bend	Format Conversion Fac Kf-Shear
+1.40D							
+1.20D+0.50Lr							
+1.20D+0.50S							
+1.20D+1.60Lr							
Length = 1.911 ft	1	8.31	4.24	0.212	0.284	2.541	2.880
Length = 2.011 ft	1	14.35	3.69	0.366	0.284	2.541	2.880
Length = 2.011 ft	1	17.63	2.32	0.450	0.284	2.541	2.880
Length = 2.011 ft	1	18.27	0.94	0.466	0.284	2.541	2.880
Length = 2.011 ft	1	18.13	1.81	0.463	0.284	2.541	2.880
Length = 2.011 ft	1	15.87	3.19	0.405	0.284	2.541	2.880
Length = 2.011 ft	1	10.83	4.57	0.276	0.305	2.541	2.880
Length = 2.011 ft	1	-7.54	5.95	0.249	0.397	2.541	2.880

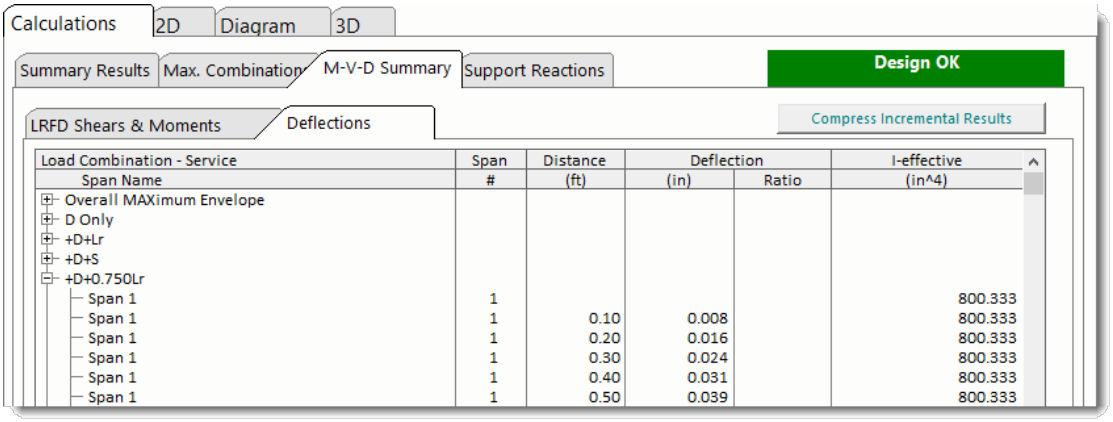
M-V-D Summary - Shears & Moments shows highly detailed moment and shear information for each beam and for each load combination. For multi-span beams using Automatic Unbalanced Live Load Placement there may be thousands of lines of results.



Load Combination - LRFD / Span Name	Span #	Distance (ft)	Shear : Vu (k) KIPS	Moment : Mu KFT	Lu (k-ft)
+1.40D					
+1.20D+0.50Lr					
+1.20D+0.50S					
+1.20D+1.60Lr					
Span 1	1		5.002		port B
Span 1	1	0.101	4.934	0.500	= 1.9
Span 1	1	0.201	4.865	0.992	= 1.9
Span 1	1	0.302	4.796	1.478	= 1.9
Span 1	1	0.402	4.727	1.957	= 1.9
Span 1	1	0.503	4.658	2.429	= 1.9
Span 1	1	0.603	4.589	2.894	= 1.9
Span 1	1	0.704	4.520	3.352	= 1.9

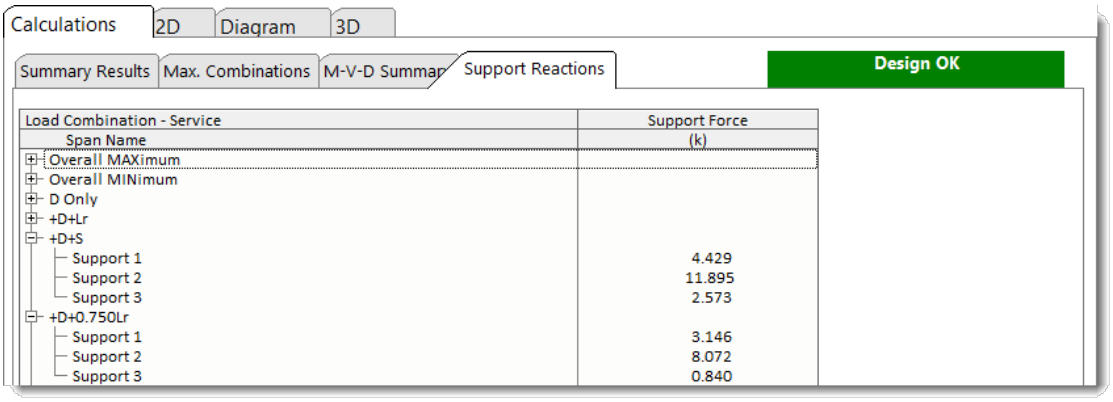
© 2026 ENERCALC, LLC

M-V-D Summary - Deflections shows highly detailed deflection results for all load combinations.



Load Combination - Service / Span Name	Span #	Distance (ft)	Deflection (in)	Deflection Ratio	I-effective (in^4)
Overall MAXimum Envelope					
D Only					
+D+Lr					
+D+S					
+D+0.750Lr					
Span 1	1				800.333
Span 1	1	0.10	0.008		800.333
Span 1	1	0.20	0.016		800.333
Span 1	1	0.30	0.024		800.333
Span 1	1	0.40	0.031		800.333
Span 1	1	0.50	0.039		800.333

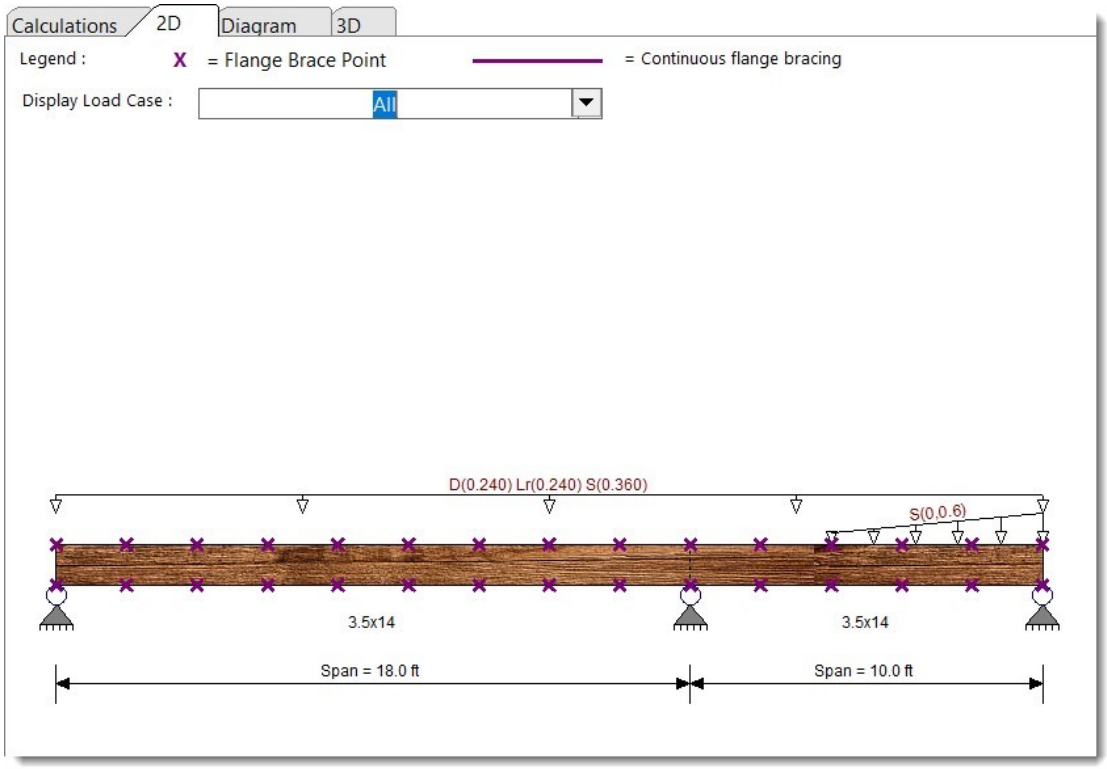
Support Reactions shows reactions for each support for each load condition.



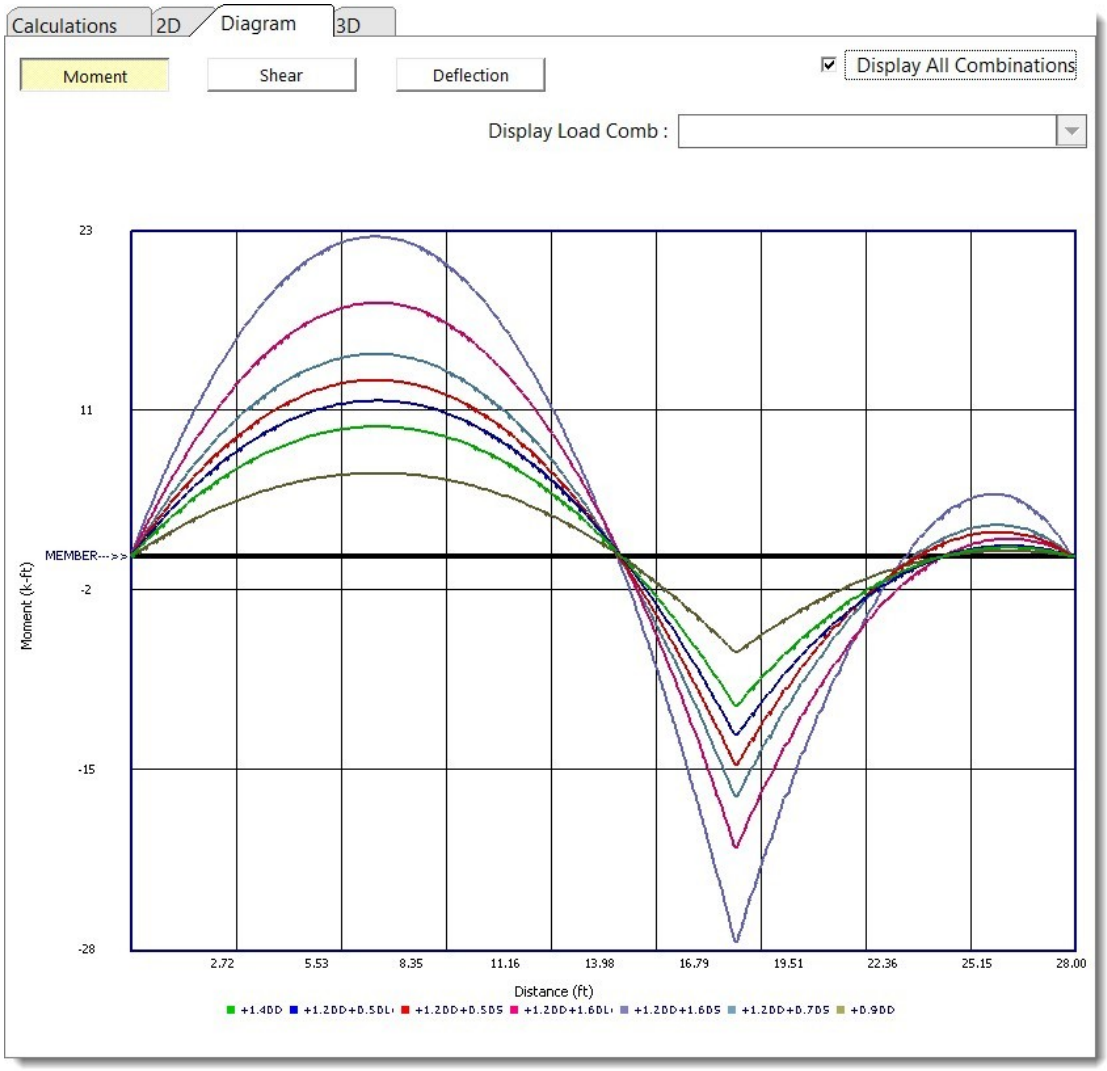
Load Combination - Service / Span Name	Support Force (k)
Overall MAXimum	
Overall MINimum	
D Only	
+D+Lr	
+D+S	
Support 1	4.429
Support 2	11.895
Support 3	2.573
+D+0.750Lr	
Support 1	3.146
Support 2	8.072
Support 3	0.840

© 2026 ENERCALC, LLC

The 2D Sketch tab provides a graphic representation of the beam currently being designed:

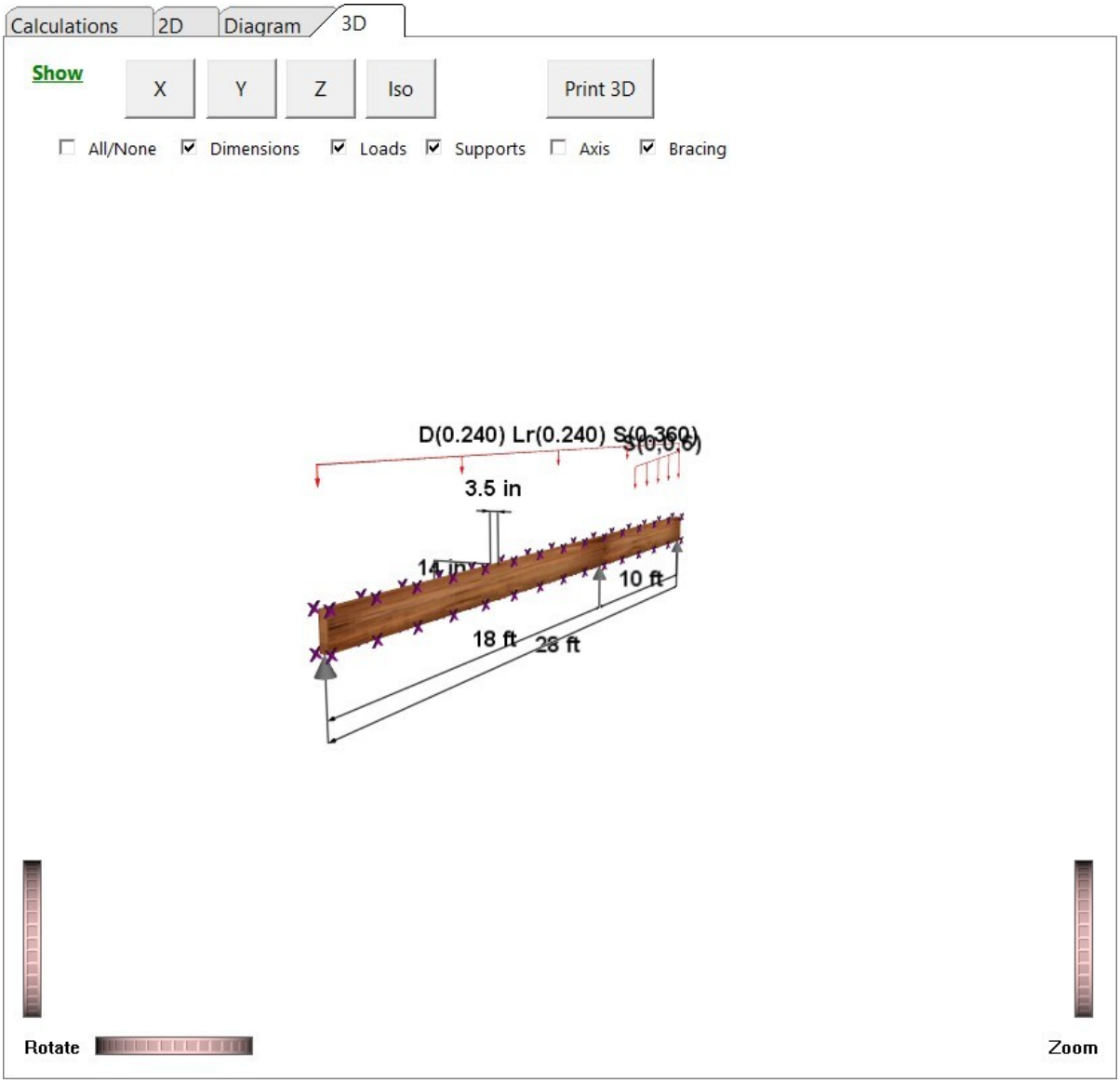


The Diagram tab offers the ability to view shear, moment, and deflection diagrams for selected load combinations:



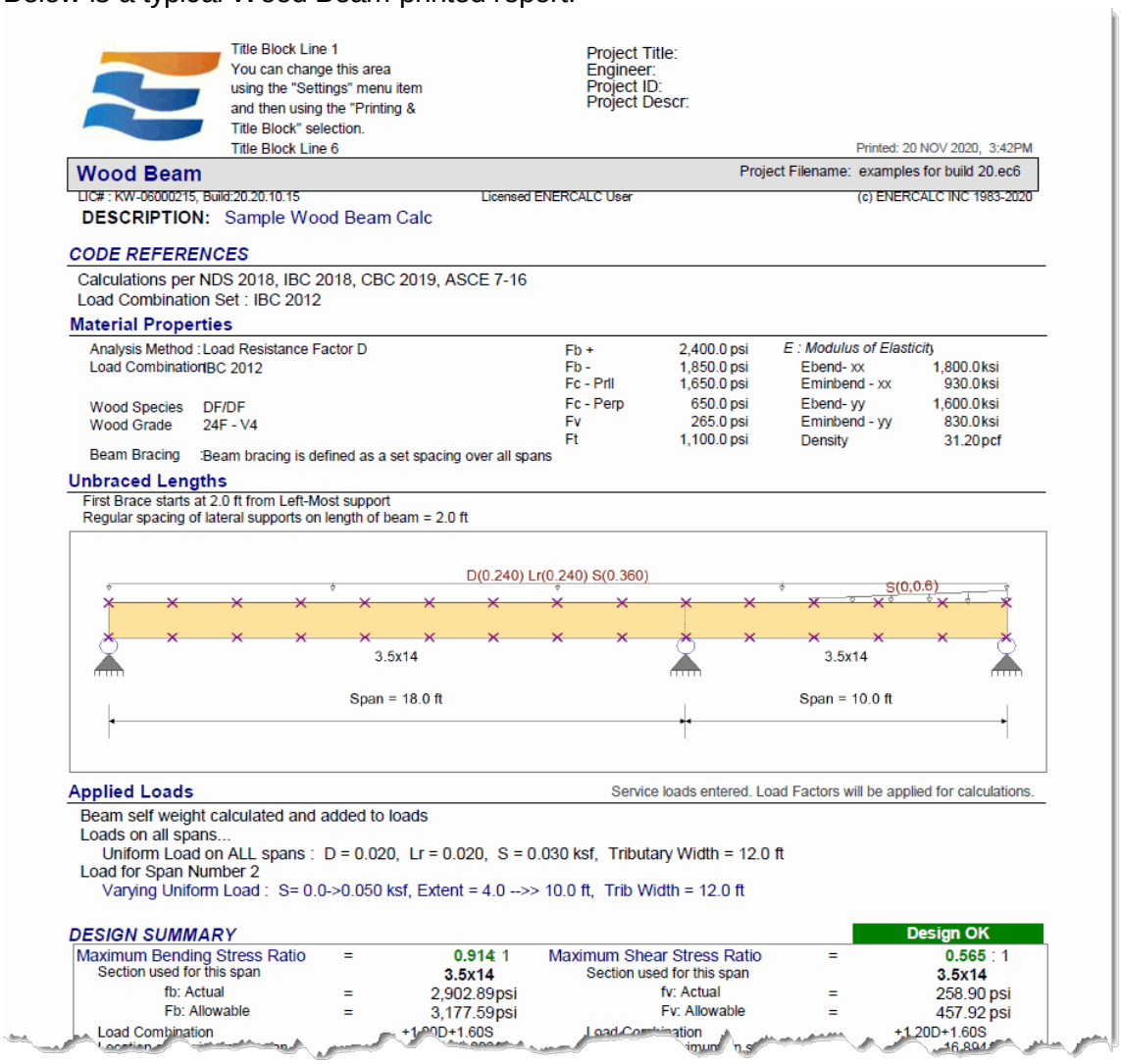
© 2026 ENERCALC, LLC

The 3D Rendering tab provides a graphic representation of the beam currently being designed, and offers many display options:



REPORTS

Below is a typical Wood Beam printed report:



Title Block Line 1
You can change this area
using the "Settings" menu item
and then using the "Printing &
Title Block" selection.
Title Block Line 6

Project Title:
Engineer:
Project ID:
Project Descr:

Printed: 20 NOV 2020, 3:42PM

Wood Beam

Project Filename: examples for build 20.ec6

LIC# : KW-06000215, Build:20.20.10.15 Licensed ENERCALC User (c) ENERCALC INC 1983-2020

DESCRIPTION: Sample Wood Beam Calc

CODE REFERENCES

Calculations per NDS 2018, IBC 2018, CBC 2019, ASCE 7-16
Load Combination Set : IBC 2012

Material Properties

Analysis Method : Load Resistance Factor D
Load Combination IBC 2012

Wood Species DF/DF
Wood Grade 24F - V4

Beam Bracing :Beam bracing is defined as a set spacing over all spans

		E : Modulus of Elasticity	
Fb +	2,400.0 psi	Ebend- xx	1,800.0 ksi
Fb -	1,850.0 psi	Eminbend - xx	930.0 ksi
Fc - Prll	1,650.0 psi	Ebend- yy	1,600.0 ksi
Fc - Perp	650.0 psi	Eminbend - yy	830.0 ksi
Fv	265.0 psi	Density	31.20 pcf
Ft	1,100.0 psi		

Unbraced Lengths

First Brace starts at 2.0 ft from Left-Most support
Regular spacing of lateral supports on length of beam = 2.0 ft

Applied Loads Service loads entered. Load Factors will be applied for calculations.

Beam self weight calculated and added to loads
Loads on all spans...
Uniform Load on ALL spans : D = 0.020, Lr = 0.020, S = 0.030 ksf, Tributary Width = 12.0 ft
Load for Span Number 2
Varying Uniform Load : S= 0.0->0.050 ksf, Extent = 4.0 -->> 10.0 ft, Trib Width = 12.0 ft

DESIGN SUMMARY **Design OK**

Maximum Bending Stress Ratio	=	0.914 : 1	Maximum Shear Stress Ratio	=	0.565 : 1
Section used for this span		3.5x14	Section used for this span		3.5x14
fb: Actual	=	2,902.89 psi	fv: Actual	=	258.90 psi
Fb: Allowable	=	3,177.59 psi	Fv: Allowable	=	457.92 psi
Load Combination		+1.20D+1.60S	Load Combination		+1.20D+1.60S

13.4.8 Wood Ledger

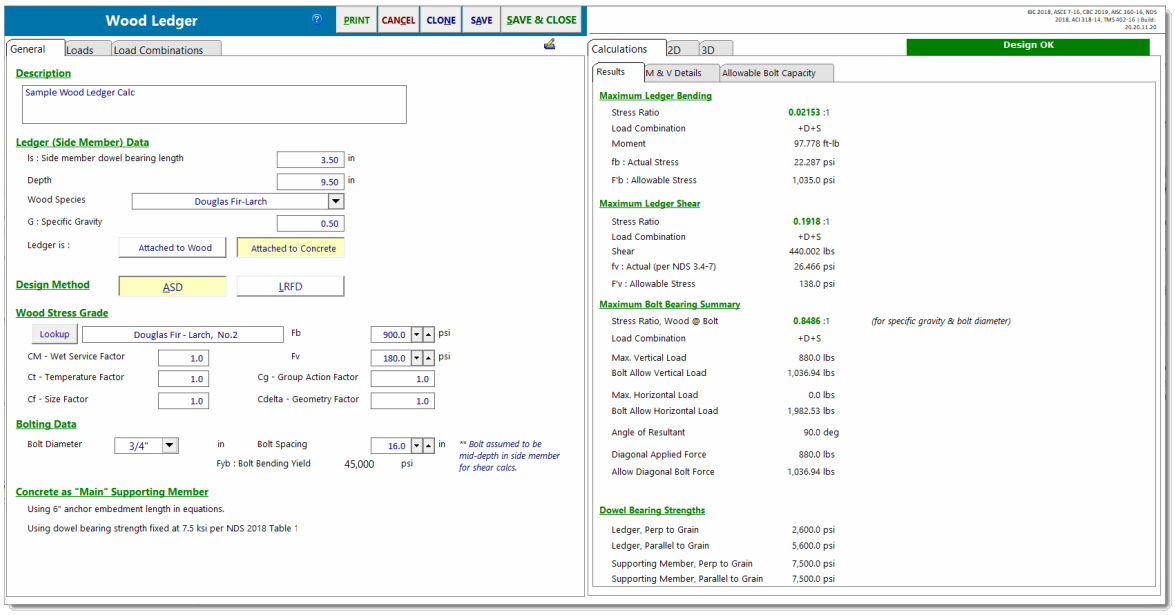
Last revised: Tuesday, February 3, 2026 at 06:12 PM

The wood ledger module provides the ability to calculate moments and shears in the ledger, as well as actual and allowable bearing loads on the attaching bolts.

The module allows the ledger to be attached to concrete or another wood member, and it automatically calculates the proper allowable bolt values.

All calculations are according to NDS and IBC. Allowable bolt capacity at an angle to the grain is calculated using Hankinson's formula.

© 2026 ENERCALC, LLC



General

This tab collects information on the ledger size and stress grade, and bolt information.

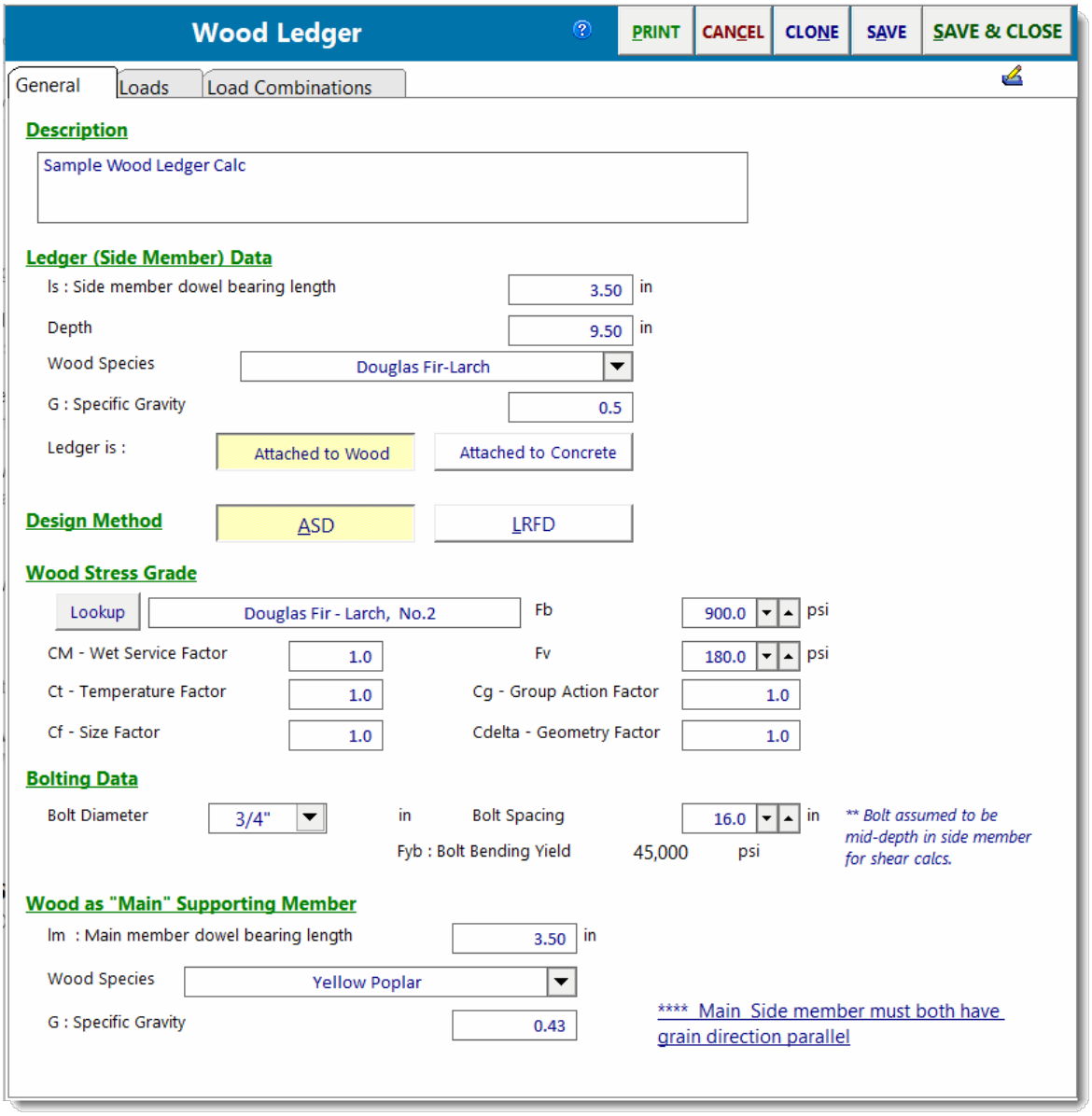
Ledger Data: Enter the actual dimensions (not nominal) and the wood species of the ledger. The specific gravity will be retrieved from the internal databases (you can also revise the specific gravity).

ASD or LRFD: Select the design method to be used. Load factoring and allowable stress calculations will applied accordingly. Values of K_F and phi are automatically determined and applied for the LRFD method.

Wood Stress Grade: Use the [**Browse**] button to access the built-in NDS reference design values database and retrieve the Fb and Fv values. You can also edit these values separately.

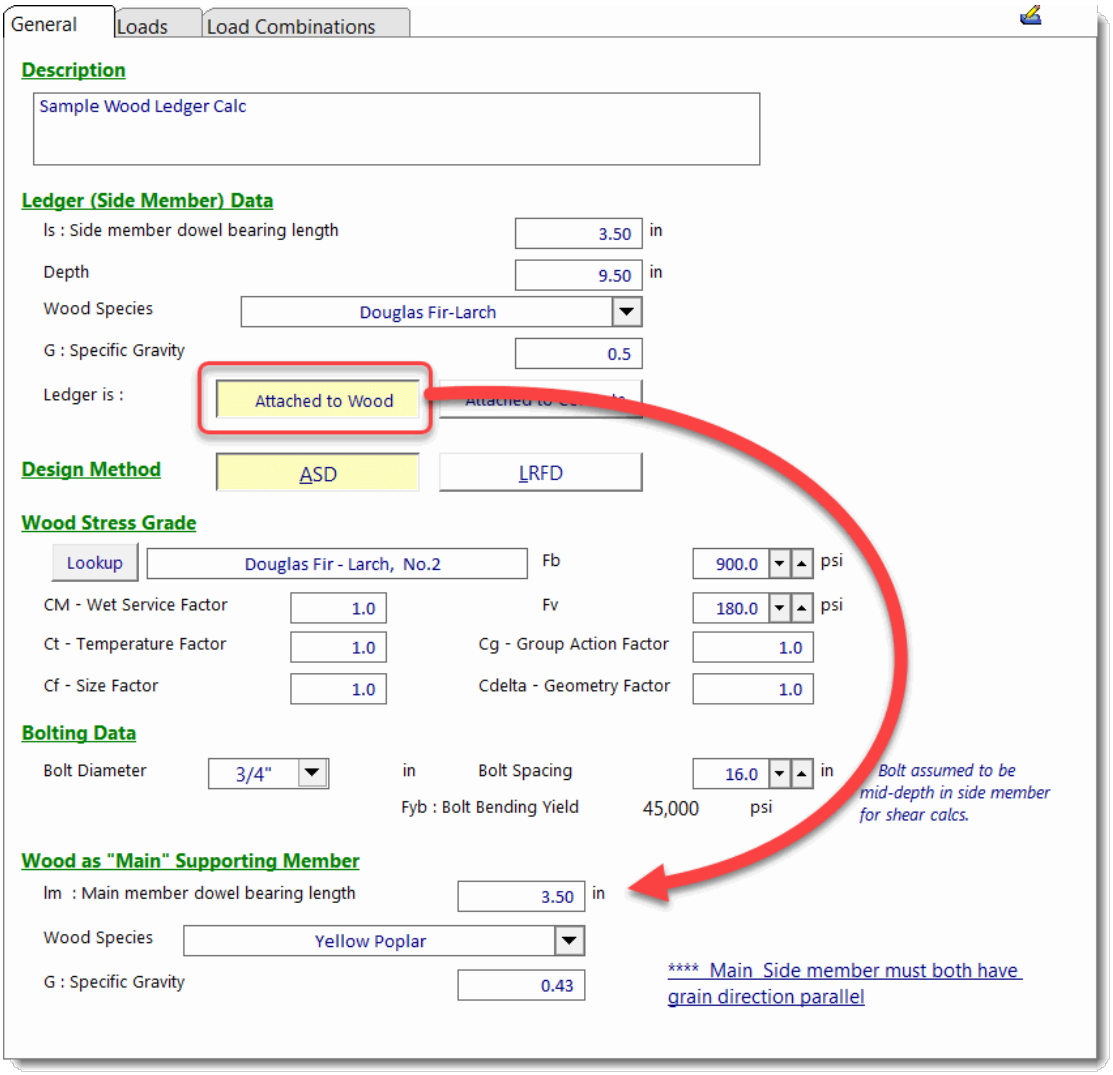
© 2026 ENERCALC, LLC

Bolting Data: Enter the bolt diameter and spacing. The yield strength of the bolt is fixed at 45 ksi in this module to remain consistent with the assumptions in the NDS.



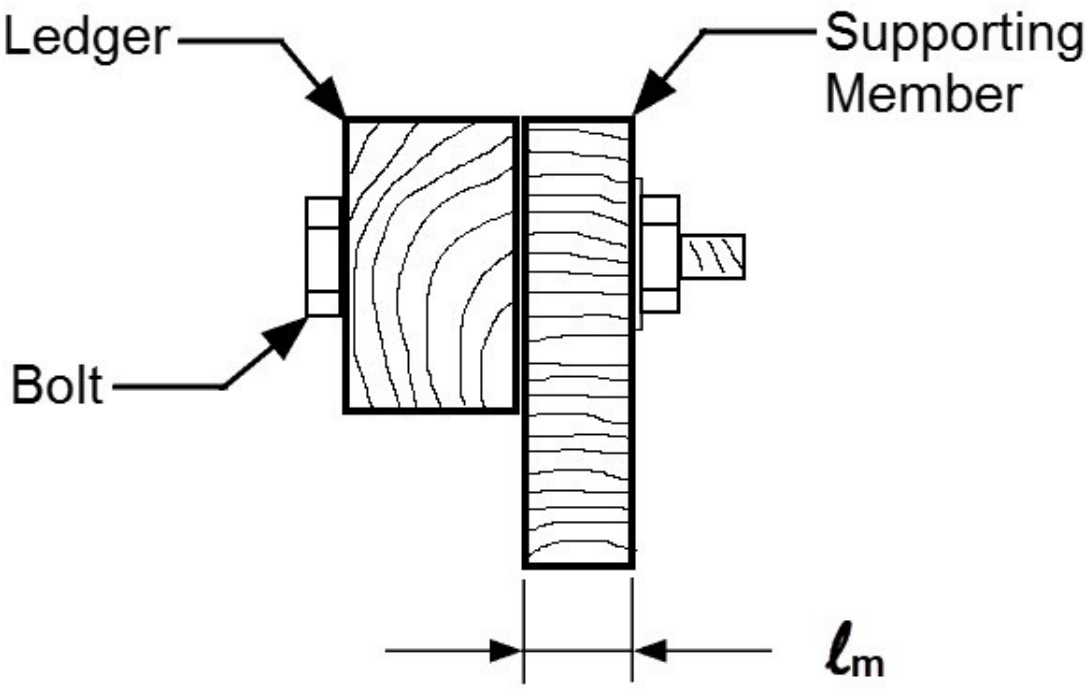
When ledger is "Attached to Wood"

When the user specifies that the ledger is attached to wood, the display will change to allow the width and species of the supporting member to be entered.



© 2026 ENERCALC, LLC

The following sketch clarifies the width dimension required for a wood supporting member:



Loads

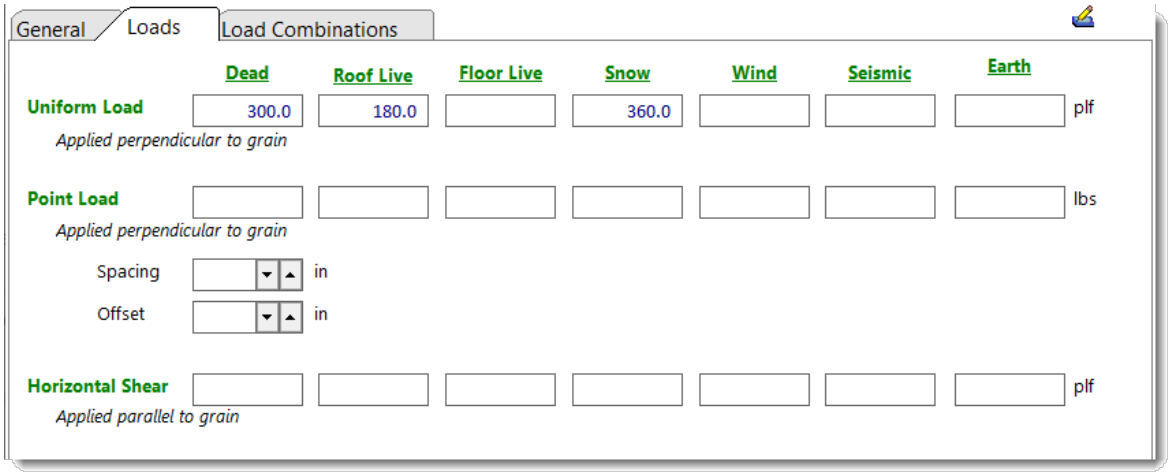
This tab allows you to apply vertical and horizontal loads to the ledger. All loads are assumed to act in the plane of the wall. Vertical loads might come from gravity loading on supported members. Horizontal loading might come from diaphragm action, such as the floor system dragging lateral wind or seismic load into the ledger. When both vertical and horizontal loads are applied, the resulting load will be at an angle to the bolt that is somewhere between 0 degrees (for purely horizontal loading) and 90 degrees (for purely vertical loading).

One set of uniform load values can be specified, and it will be considered to act consistently along all areas of the ledger. For calculations purposes, this module considers the ledger a continuous beam over multiple supports.

One set of repeating point loads can be specified, where the input collects the magnitude, the starting location, and the spacing between subsequent loads. The module will consider all locations of the point loads over enough ledger spans between bolts to determine the governing case. For example, say your ledger bolting is set to 36" and the point load is set to 15". There will be multiple point loads between bolts, and on the NEXT span, the point loads will be in different relative positions with respect to the bolts. The module analyzes all conditions of that bolt pattern over enough ledger spans to determine the governing point load offset.

© 2026 ENERCALC, LLC

One set of horizontal (lateral) shear loads is allowed, in order to simulate wind or seismic load applied to the ledger and acting in the plane of the supporting wall.

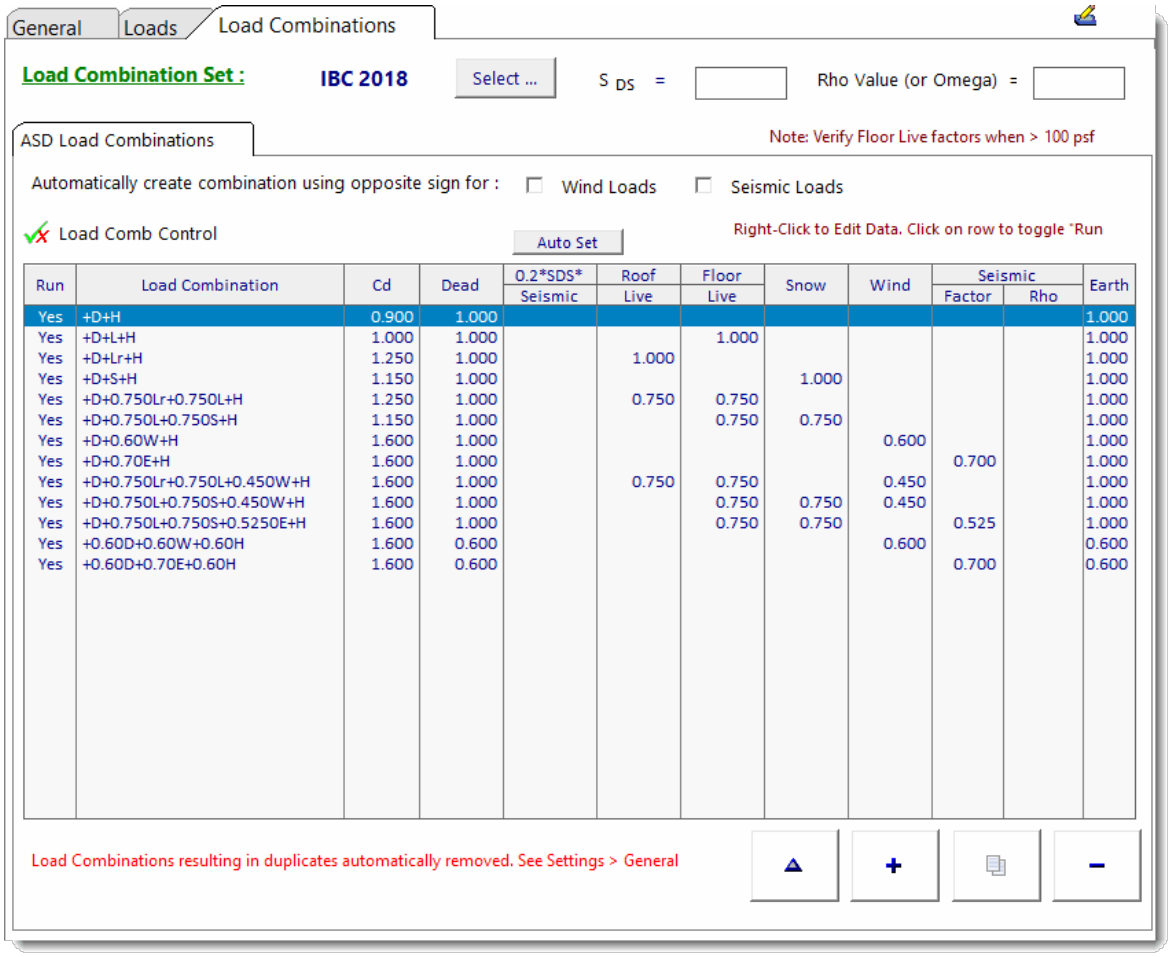


Load Combinations Tab

This is the typical load combination tab with entries for load duration factors C_D (ASD) and λ (LRFD).

© 2026 ENERCALC, LLC

Selecting the [**Auto Reverse Wind Factors**] and/or [**Auto Reverse Seismic Factors**] buttons creates additional load combinations that insert "-W" and "-E" whenever W and E are used.



Run	Load Combination	Cd	Dead	0.2*SDS* Seismic	Roof Live	Floor Live	Snow	Wind	Seismic Factor	Seismic Rho	Earth
Yes	+D+H	0.900	1.000								1.000
Yes	+D+L+H	1.000	1.000			1.000					1.000
Yes	+D+Lr+H	1.250	1.000		1.000						1.000
Yes	+D+S+H	1.150	1.000				1.000				1.000
Yes	+D+0.750Lr+0.750L+H	1.250	1.000		0.750	0.750					1.000
Yes	+D+0.750L+0.750S+H	1.150	1.000			0.750	0.750				1.000
Yes	+D+0.60W+H	1.600	1.000					0.600			1.000
Yes	+D+0.70E+H	1.600	1.000						0.700		1.000
Yes	+D+0.750Lr+0.750L+0.450W+H	1.600	1.000		0.750	0.750		0.450			1.000
Yes	+D+0.750L+0.750S+0.450W+H	1.600	1.000			0.750	0.750	0.450			1.000
Yes	+D+0.750L+0.750S+0.5250E+H	1.600	1.000			0.750	0.750		0.525		1.000
Yes	+0.60D+0.60W+0.60H	1.600	0.600					0.600			0.600
Yes	+0.60D+0.70E+0.60H	1.600	0.600						0.700		0.600

Results

This set of tabs provides detailed results for the current calculation. The tabs in the upper right-hand corner of the screen allow you to select the three major areas available for review: Calculations, 2D Sketch and 3D Rendering.

The Calculations tab offers the following results options:

Results tab presents a summary of the design for the current member by reporting the following:

Maximum Ledger Bending shows the load combination, applied moment and actual and allowable bending stresses. Note that no slenderness is considered for the flexural design of the ledger, as it is assumed to be fully braced.

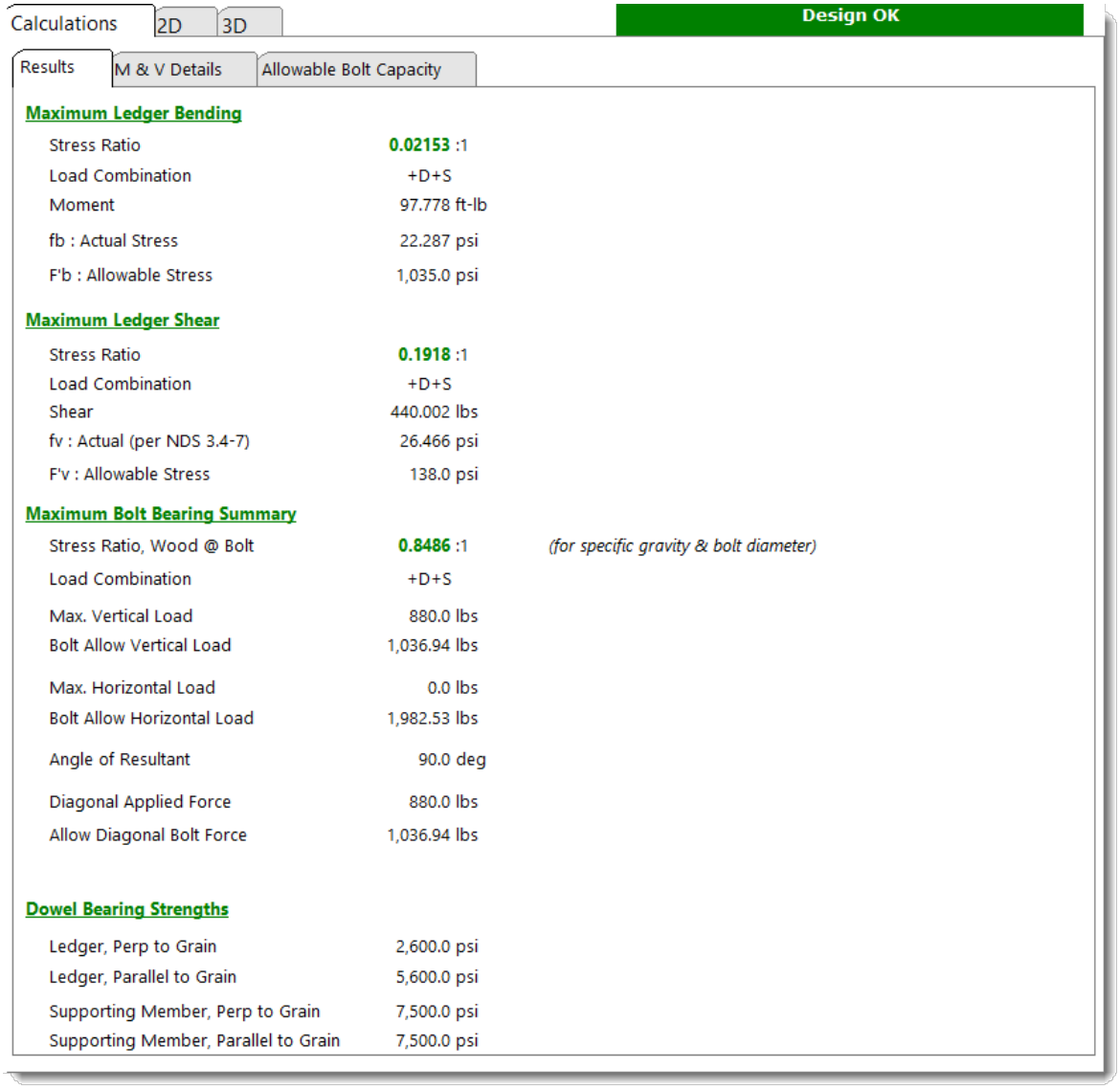
Maximum Ledger Shear shows the load combination, applied shear and actual and allowable shear stresses for vertical loads only. These stresses are taken at the point of

© 2026 ENERCALC, LLC

bolt support of the ledger and no subtraction of uniform loads within a distance "ledger depth" from that support is considered.

Maximum Bolt Bearing Summary Shows the vertical and horizontal components of force acting on the bolt for the governing condition. "Allow Diagonal Bolt Force" is the result of allowable parallel to grain and perpendicular to grain bolt capacities used for the "Angle of Resultant" in a Hankinson formula calculation.

Dowel Bearing Strengths give the allowable stress and resulting allowable bolt bearing stress considering NDS procedures and strengths and bolt size the user has entered.



Calculations | 2D | 3D — Design OK

Results | M & V Details | Allowable Bolt Capacity

Maximum Ledger Bending

Stress Ratio	**0.02153** :1	
Load Combination	+D+S	
Moment	97.778 ft-lb	
fb : Actual Stress	22.287 psi	
F'b : Allowable Stress	1,035.0 psi	

Maximum Ledger Shear

Stress Ratio	**0.1918** :1	
Load Combination	+D+S	
Shear	440.002 lbs	
fv : Actual (per NDS 3.4-7)	26.466 psi	
F'v : Allowable Stress	138.0 psi	

Maximum Bolt Bearing Summary

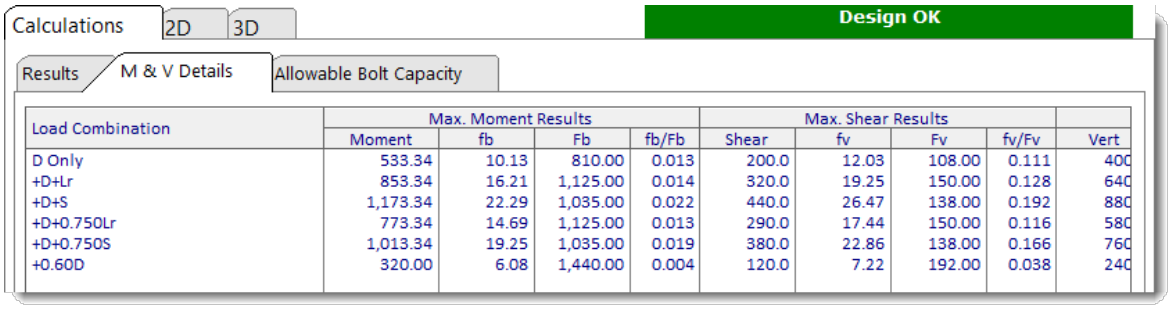
Stress Ratio, Wood @ Bolt	**0.8486** :1	*(for specific gravity & bolt diameter)*
Load Combination	+D+S	
Max. Vertical Load	880.0 lbs	
Bolt Allow Vertical Load	1,036.94 lbs	
Max. Horizontal Load	0.0 lbs	
Bolt Allow Horizontal Load	1,982.53 lbs	
Angle of Resultant	90.0 deg	
Diagonal Applied Force	880.0 lbs	
Allow Diagonal Bolt Force	1,036.94 lbs	

Dowel Bearing Strengths

Ledger, Perp to Grain	2,600.0 psi
Ledger, Parallel to Grain	5,600.0 psi
Supporting Member, Perp to Grain	7,500.0 psi
Supporting Member, Parallel to Grain	7,500.0 psi

© 2026 ENERCALC, LLC

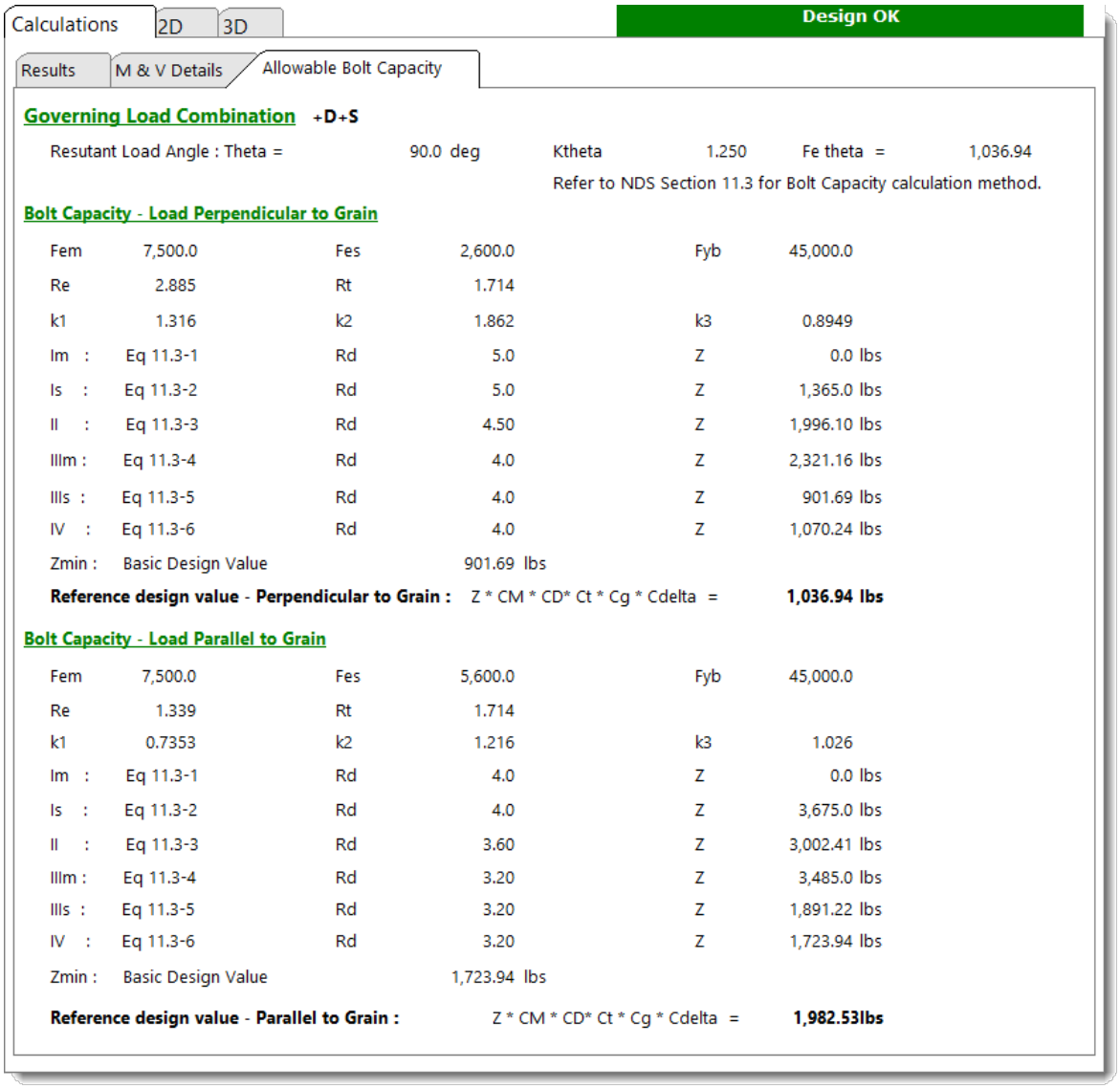
M & V Details tab summarizes the design values according to load combination for moment, shear and bolt forces.



Load Combination	Max. Moment Results				Max. Shear Results				
	Moment	fb	Fb	fb/Fb	Shear	fv	Fv	fv/Fv	Vert
D Only	533.34	10.13	810.00	0.013	200.0	12.03	108.00	0.111	400
+D+Lr	853.34	16.21	1,125.00	0.014	320.0	19.25	150.00	0.128	640
+D+S	1,173.34	22.29	1,035.00	0.022	440.0	26.47	138.00	0.192	880
+D+0.750Lr	773.34	14.69	1,125.00	0.013	290.0	17.44	150.00	0.116	580
+D+0.750S	1,013.34	19.25	1,035.00	0.019	380.0	22.86	138.00	0.166	760
+0.60D	320.00	6.08	1,440.00	0.004	120.0	7.22	192.00	0.038	240

© 2026 ENERCALC, LLC

Allowable Bolt Capacity tab provides the details of the bolt capacities according to the chosen NDS calculation method.



Calculations | 2D | 3D

Design OK

Results | M & V Details | Allowable Bolt Capacity

Governing Load Combination +D+S

Resutant Load Angle : Theta = 90.0 deg Ktheta 1.250 Fe theta = 1,036.94

Refer to NDS Section 11.3 for Bolt Capacity calculation method.

Bolt Capacity - Load Perpendicular to Grain

Fem	7,500.0	Fes	2,600.0	Fyb	45,000.0
Re	2.885	Rt	1.714		
k1	1.316	k2	1.862	k3	0.8949
Im :	Eq 11.3-1	Rd	5.0	Z	0.0 lbs
Is :	Eq 11.3-2	Rd	5.0	Z	1,365.0 lbs
II :	Eq 11.3-3	Rd	4.50	Z	1,996.10 lbs
IIIm :	Eq 11.3-4	Rd	4.0	Z	2,321.16 lbs
IIIs :	Eq 11.3-5	Rd	4.0	Z	901.69 lbs
IV :	Eq 11.3-6	Rd	4.0	Z	1,070.24 lbs

Zmin : Basic Design Value 901.69 lbs

Reference design value - Perpendicular to Grain : Z * CM * CD* Ct * Cg * Cdelta = **1,036.94 lbs**

Bolt Capacity - Load Parallel to Grain

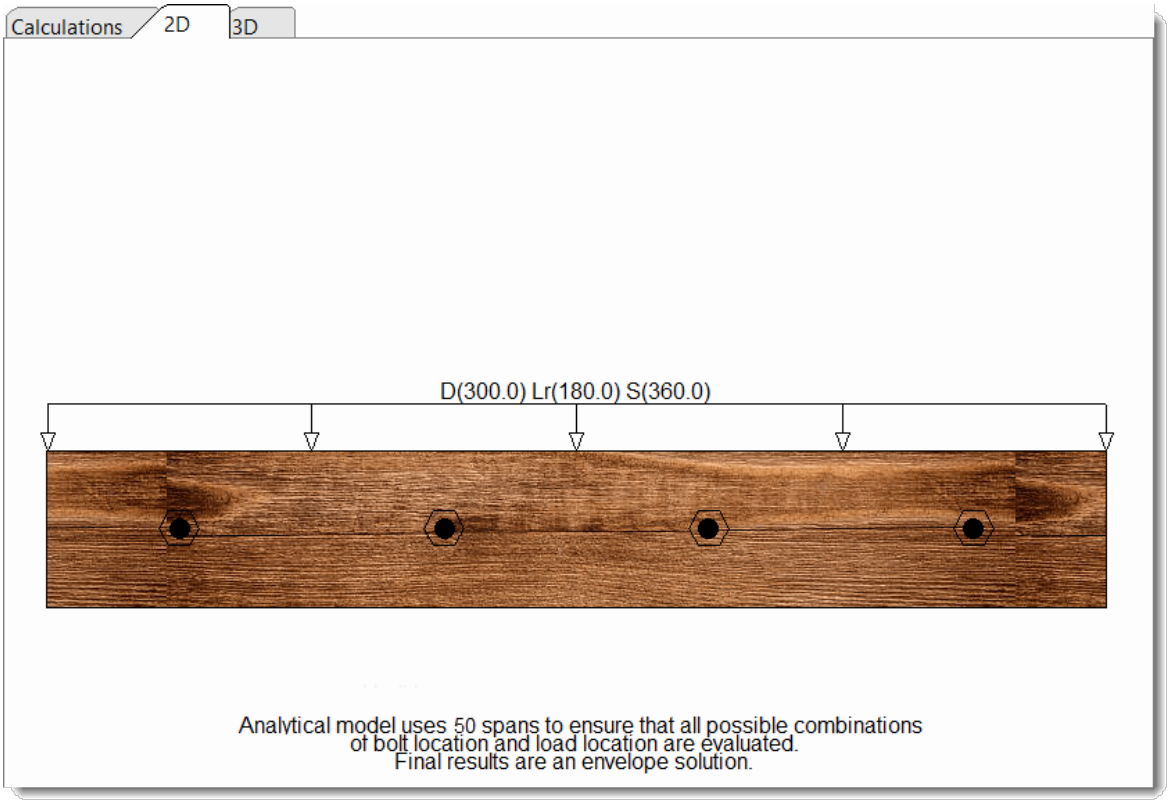
Fem	7,500.0	Fes	5,600.0	Fyb	45,000.0
Re	1.339	Rt	1.714		
k1	0.7353	k2	1.216	k3	1.026
Im :	Eq 11.3-1	Rd	4.0	Z	0.0 lbs
Is :	Eq 11.3-2	Rd	4.0	Z	3,675.0 lbs
II :	Eq 11.3-3	Rd	3.60	Z	3,002.41 lbs
IIIm :	Eq 11.3-4	Rd	3.20	Z	3,485.0 lbs
IIIs :	Eq 11.3-5	Rd	3.20	Z	1,891.22 lbs
IV :	Eq 11.3-6	Rd	3.20	Z	1,723.94 lbs

Zmin : Basic Design Value 1,723.94 lbs

Reference design value - Parallel to Grain : Z * CM * CD* Ct * Cg * Cdelta = **1,982.53lbs**

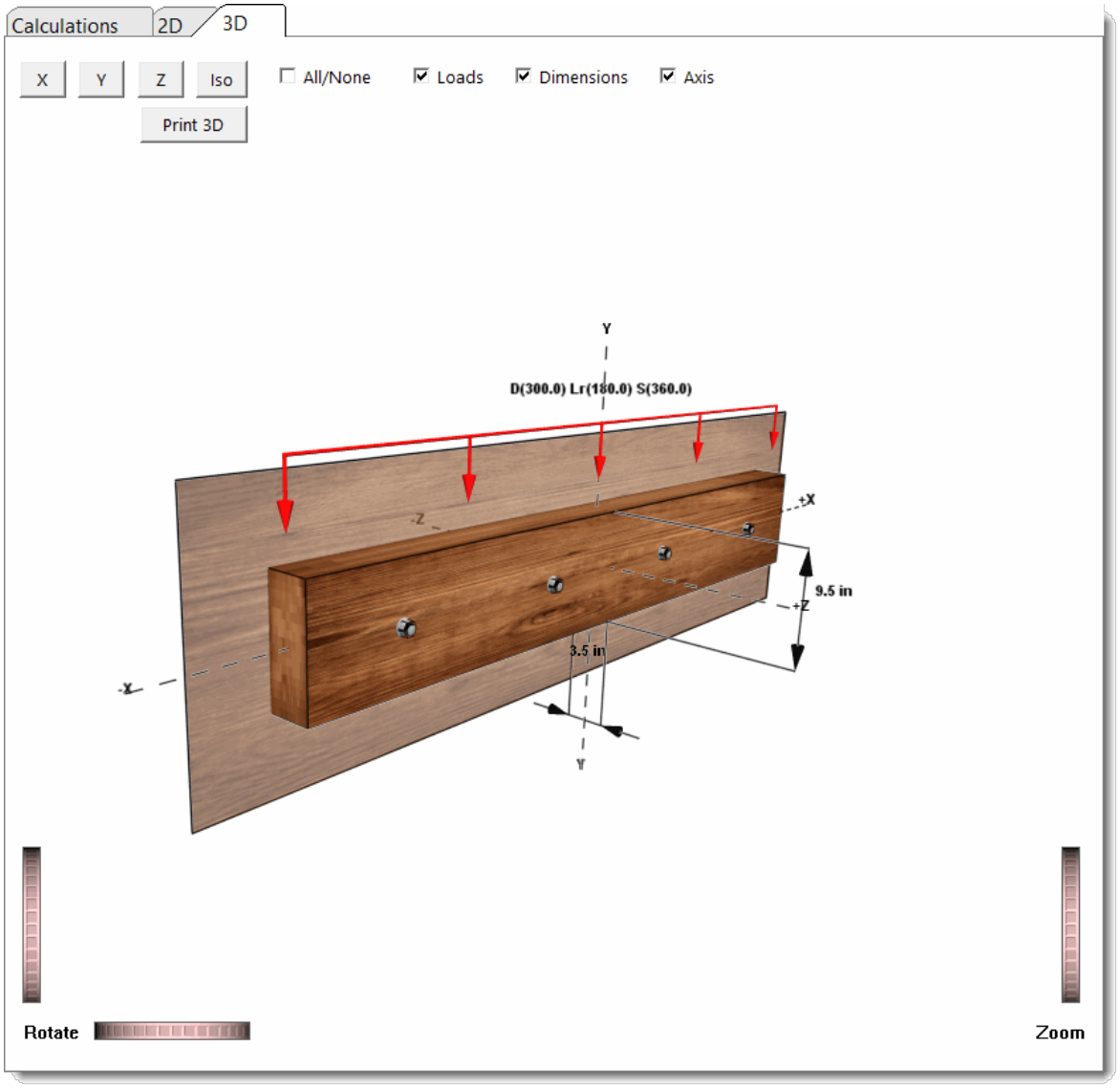
© 2026 ENERCALC, LLC

The 2D Sketch tab provides a graphic representation of the ledger currently being designed:



© 2026 ENERCALC, LLC

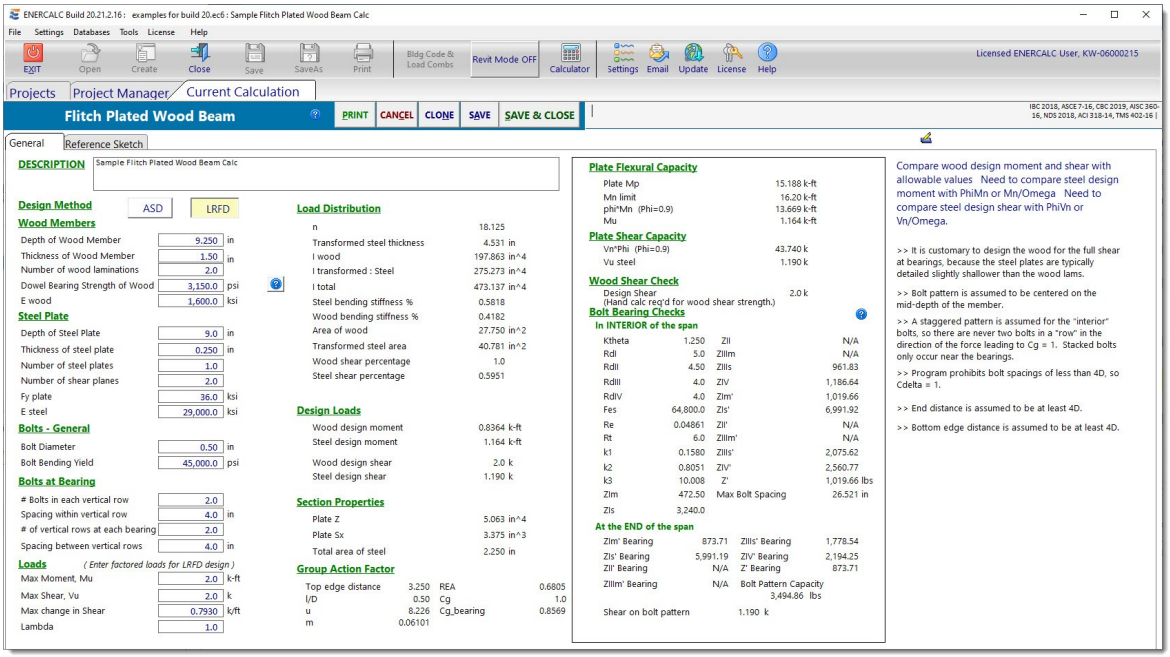
The 3D Rendering tab displays a 3D graphic of the member and offers display options:



13.4.9 Flitch Plated Wood Beam

Last revised: Tuesday, February 3, 2026 at 06:12 PM

The Flitch Plated Wood Beam module provides an efficient way to evaluate a simply-supported wood beam reinforced with one or more steel flitch plates. It allows both ASD and LRFD, but for simplicity it assumes that the values are already factored appropriately and combined. The module then performs the load distribution to determine the percentage of load attributable to the wood member(s) and to the steel plate(s). Finally, the specified bolts are checked for bearing in accordance with NDS requirements.



Design Method

Specify ASD or LRFD.

Wood Members

Specify depth and thickness of one individual wood lamination.

Number of laminations input allows multiple of the defined individual wood lamination.

Dowel bearing strength of wood comes from NDS Table 12.3.3, which is provided for convenience with a click of the related help button.

E wood is the modulus of elasticity that will be used. Enter the value of E', as no further adjustment factors are applied to this value.

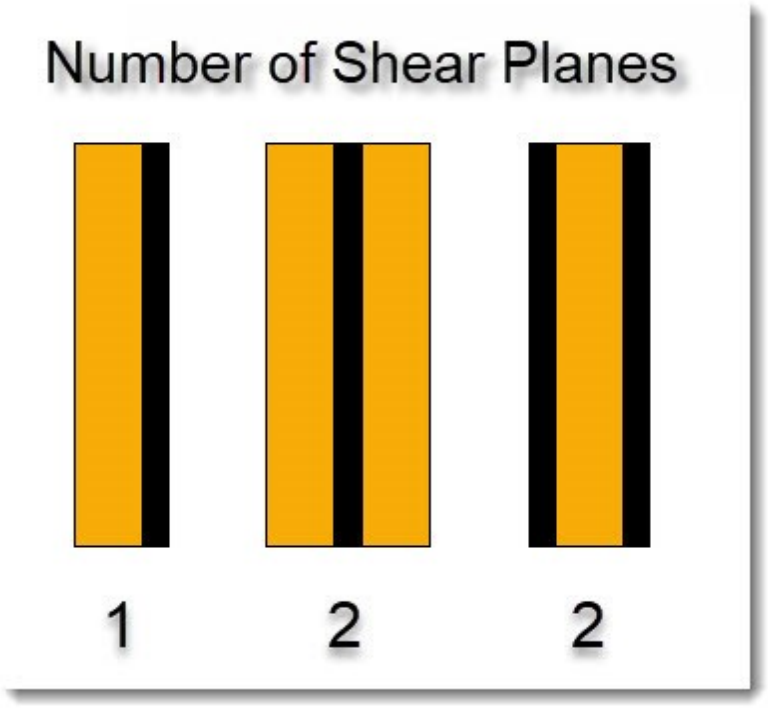
© 2026 ENERCALC, LLC

Steel Plate

Specify the depth and thickness of one individual steel plate.

Number of steel plates input allows multiple of the defined individual steel plate.

Number of shear planes affects the bolt bearing calculation.



Specify yield strength and modulus of elasticity of the steel plate(s).

Bolts - General

Specify the bolt diameter.

Specify the bolt bending yield strength.

Bolts at Bearing

Specify the number of bolts in each vertical row at the bearing and the spacing within those vertical rows. Then specify the number of vertical rows that occur at each bearing and the spacing between those rows.

© 2026 ENERCALC, LLC

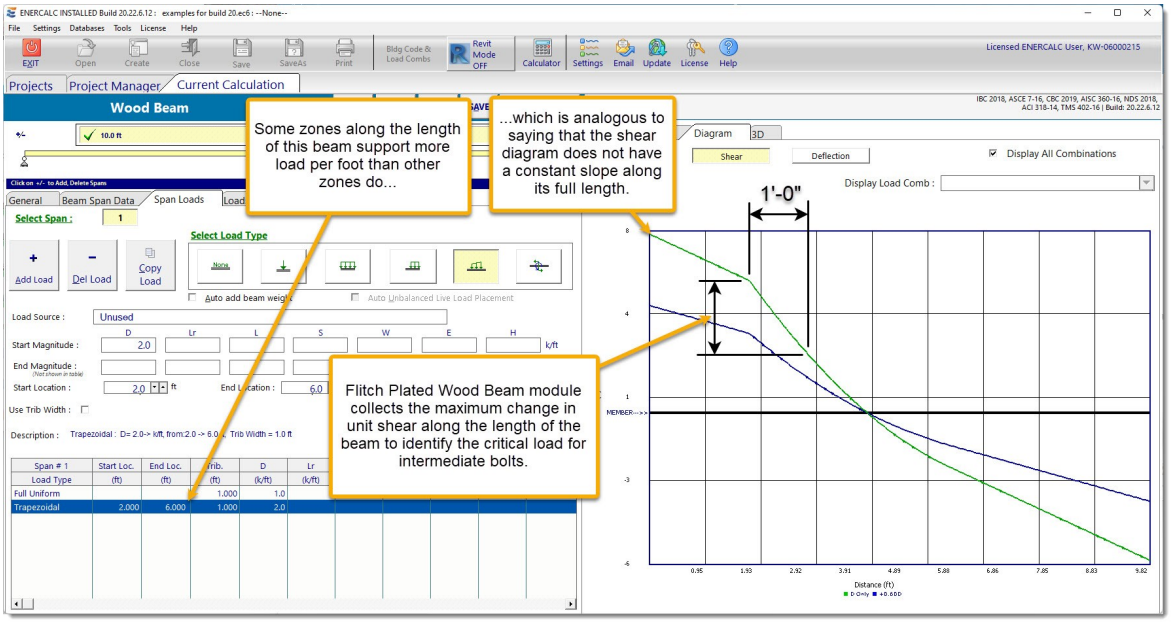
Loads

Specify the design loads.

If ASD is used, the input represents the maximum moment, maximum shear, maximum change in shear due to service-level load combinations, and the Load Duration factor, Cd.

If LRFD is used, the input represents the maximum moment, maximum shear, maximum change in shear due to strength-level load combinations, and the Load Duration factor, Lambda.

Note that the following screen capture is NOT from the Flitch Plated Wood Beam module. It is from the Wood Beam module, which supports full loading functions and displays shear diagrams. The screen capture is included to illustrate one method of determining the input value "Maximum Change in Shear" as required by the Flitch Plated Wood Beam module.



Load Distribution

The Load Distribution category presents the calculations to determine the relative flexural stiffness of the wood and the steel based on transformed moments of inertia. It also shows the calculations to determine the relative shear force distribution between the steel and the wood.

Note: The wood members are always conservatively designed to carry the full shear force, because common practice is to detail the steel flitch plates slightly shorter than the wood member to avoid conflicts at the bearing. Therefore the wood must always carry the full shear force at the bearing.

© 2026 ENERCALC, LLC

Design Loads

The Design Loads section applies the relative percentages determined in the Load Distribution section to proportion the design moment and design shear between the wood and the steel.

Section Properties

Section properties of the steel plate are presented.

Group Action Factor

The Group Action Factor section reports the values of Cg (for the capacity of bolts in the interior of the span) and Cg_bearing (for the capacity of bolts at the bearings). It also displays the variables used in determining the two group action factors.

Note: A staggered pattern is assumed for the "interior" bolts, so there are never two bolts in a row in the direction of the force leading to Cg = 1. Stacked bolts only occur near the bearings, so Cg_bearing may have a value less than 1.0.

Plate Flexural Capacity

The Plate Flexural Capacity section presents the calcs to determine the flexural capacity of the plate. The value of Ma or Mu attributable to the plate is reported for easy verification of the flexural capacity.

Note: Flexural buckling is assumed to be prevented by adequate bracing.

Plate Shear Capacity

The Plate Shear Capacity section presents the calcs to determine the shear capacity of the plate. The value of Va or Vu attributable to the plate is reported for easy verification of the shear capacity.

Wood Shear Check

The Wood Shear Check section echoes the design shear force required of the wood member(s). This is convenient for running a quick check on the wood shear capacity by hand, or in the Wood Beam module.

© 2026 ENERCALC, LLC

Bolt Bearing Checks in the Interior of the Span

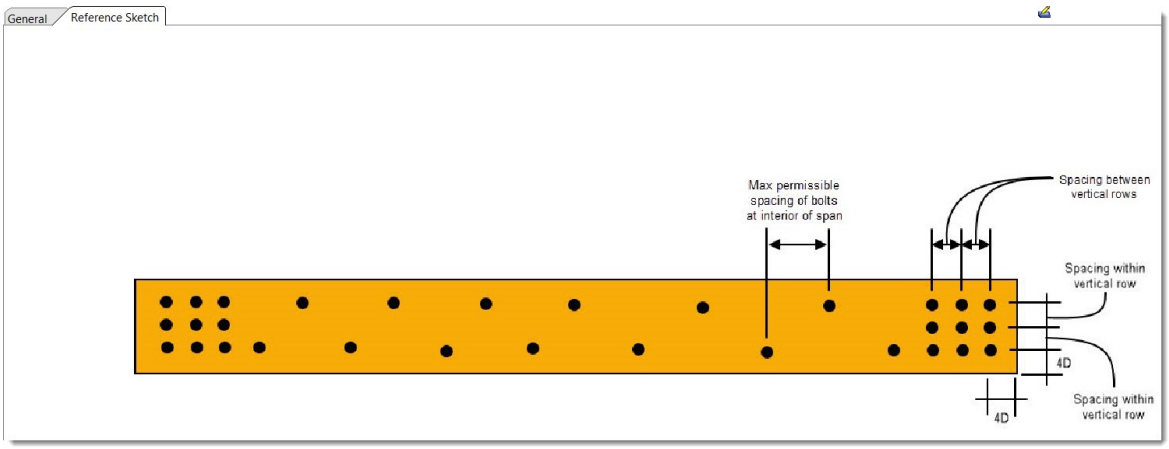
This section determines the capacity of the individual bolts in the staggered pattern in the interior of the span (not at the bearings). It presents the variables used in the checks of the various bolt failure modes, along with the capacities associated with each failure mode. The value of Z' is the lowest of Zlm', Zls', Zll', Zlllm', Zllls' and ZlV'. Using the Max change in Shear input value, this section reports a theoretical maximum permissible spacing of the bolts in the interior of the span, assuming that the bolts must transfer the "Steel bending stiffness" percentage of the "Max change in Shear" from the wood to the steel plate(s).

Bolt Bearing Checks at the End of the Span

This section determines the capacity of the individual bolts in the pattern at the end of the span (at the bearings). Values are reported for each individual failure mode. The value of Z' Bearing is the lowest of Zlm' Bearing, Zls' Bearing, Zll' Bearing, Zlllm' Bearing, Zllls' Bearing and ZlV' Bearing. It also presents the capacity of the entire pattern at the bearing, and reports the load on those bolts for easy comparison.

Reference Sketch

The reference sketch is provided to clarify the dimensions collected in the input.



13.5 Columns

ENERCALC SEL has a single basic column design module that supports Wood, Steel, Concrete and Masonry material types.

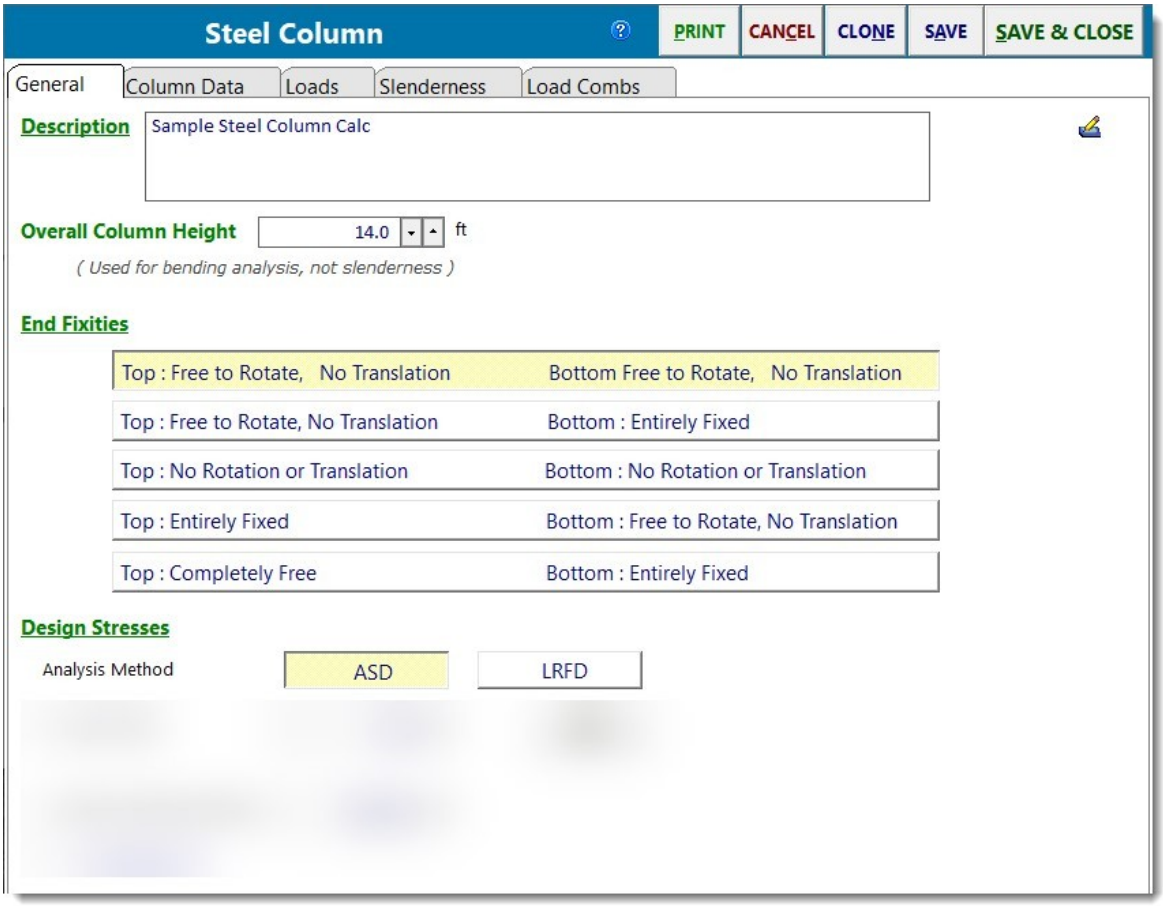
There are some portions of the graphical user interface that remain consistent for all materials, and this topic will focus on those items. For detailed information about each of the material-specific calculation modules, please review the respective topics below.

Note: The Column modules are not intended for the design of tension members. Column modules should not be applied in situations where the member experiences net tension.

© 2026 ENERCALC, LLC

General

The screen capture below shows the portions of the General tab that remain constant for all four material types.



You can easily select a different column material by clicking one of the four material buttons. When you do, the program will load the user interface that is specific to the chosen material.

Overall Column Height is the total height of the column and does not have anything to do with slenderness lengths. This length is used for three things:

- to describe the overall height of the column for the purpose of calculating self weight (if specified),
- to locate the topmost point of load application, and
- to perform the bending analysis when lateral loads are applied.

Rotational End Fixities let you specify how the ends of the columns are or are not attached to boundary conditions. Each condition explicitly describes the translational and rotational boundary conditions at both the top and bottom of the column.

© 2026 ENERCALC, LLC

Column Data

This tab varies depending on the selected material type. For detailed information on each material-specific calculation module (e.g., Steel Column, Wood Column), see the corresponding help topic.

Loads

Load Reference Guide: Clicking this button opens a pop-up window that displays the positive sign conventions used in each module. For module-specific details, such as those for Steel Column, Wood Column, etc., refer to the corresponding help topic for that module.

Vertical Loads: Vertical loads can be applied to any location along the height of the column. You use this tab to build a table of applied loads with the [**Add**], [**Copy**] and [**Delete**] buttons.

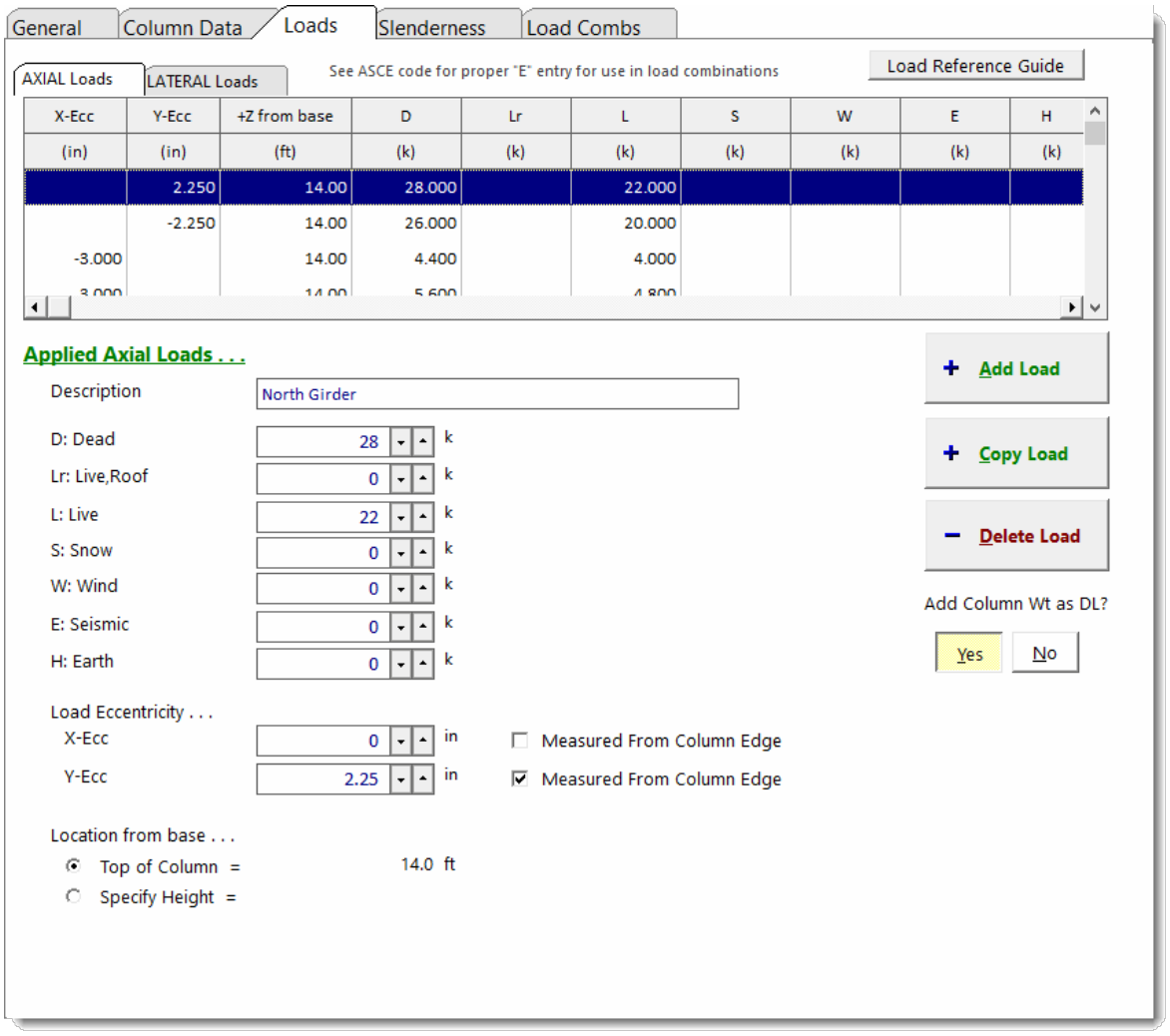
Include Self Weight option tells the module to automatically calculate the weight of the column and add it as an additional dead load (which will be factored per "D" load combination factor and applied at the top of the column).

Description lets you describe each load you are applying.

Load Eccentricity has "X" and "Y" eccentricity locations so the loads can create Y-Y and X-X axis moments respectively.

Location from Base is where you specify the vertical location of the axial load with respect to the bottom of the column.

© 2026 ENERCALC, LLC



Lateral Loads: On this tab you can specify loads that will be applied along the X or Y axes of the column (non-axial loads).

> **Note!** ***Moments applied at fixed ends have no effect on column design, and will be ignored, producing no effect on the column, and will also NOT appear as reactions.***

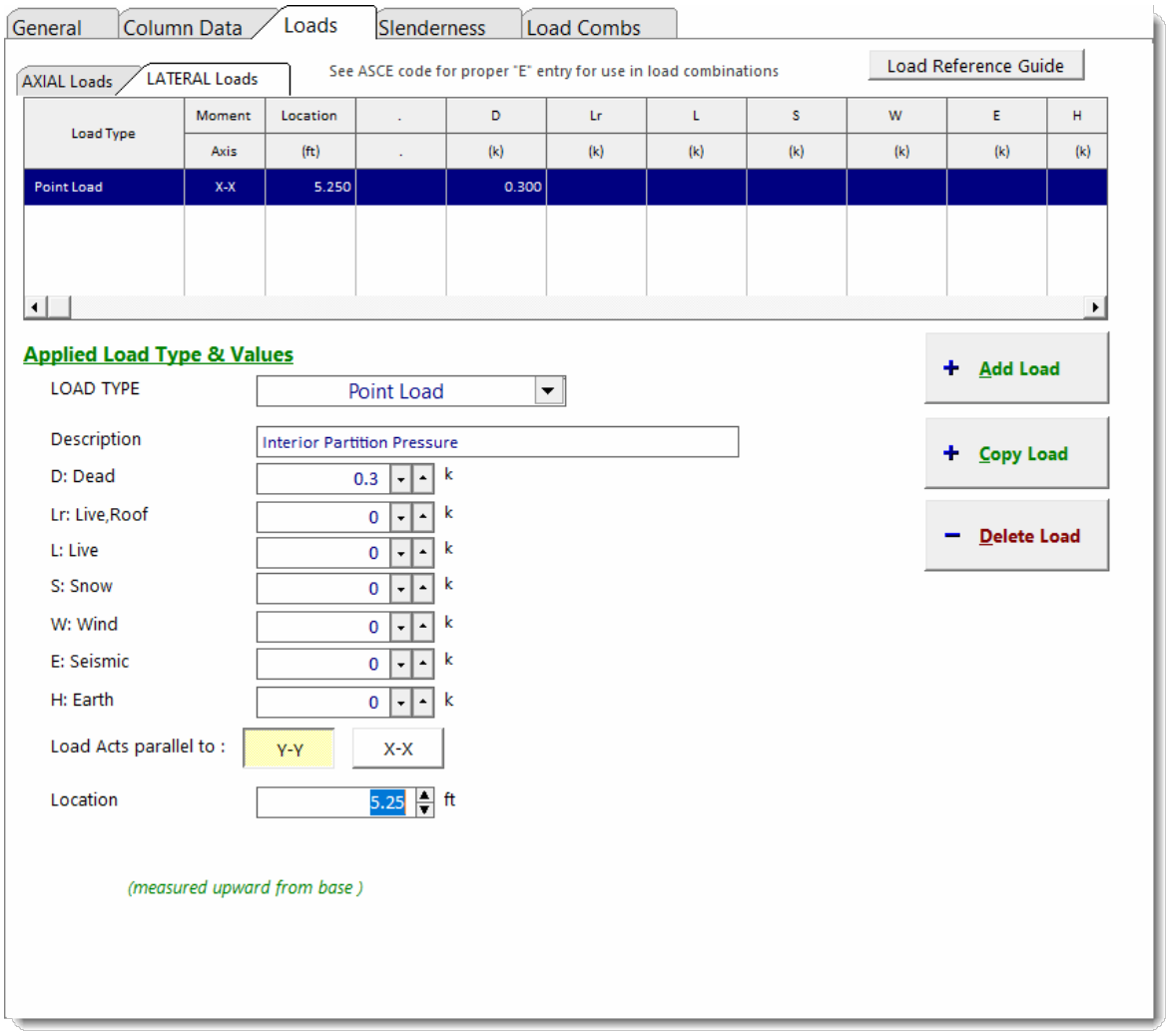
Applied Load Type & Values has a drop-down list box where you can choose Full Uniform, Partial Uniform, Point Load and Moment load types.

Description lets you describe each load you are applying.

Moment Axis is where you specify about which column axis the load creates its applied moment.

Location from Base is where you specify the vertical location of the lateral load with respect to the bottom of the column.

© 2026 ENERCALC, LLC



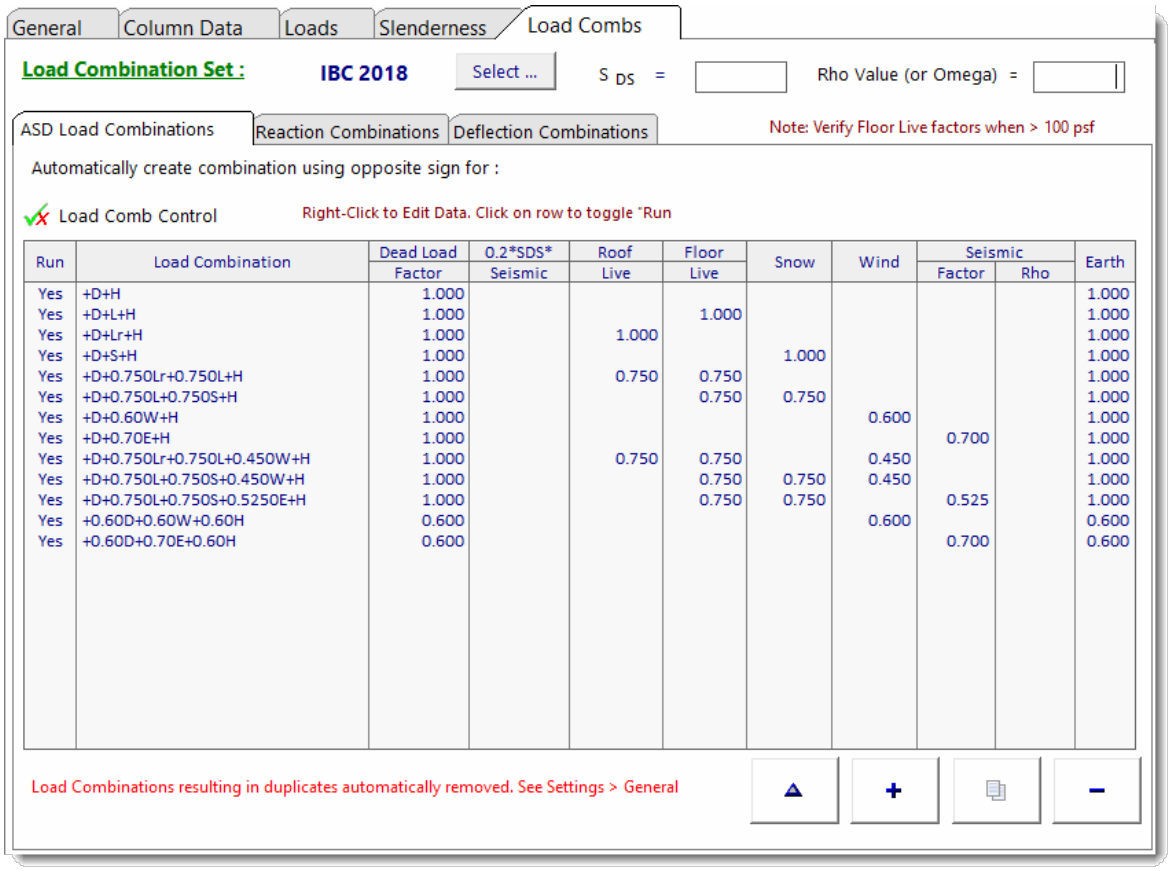
Load Combinations

This tab allows you to specify the load combinations to use for the analysis. The tabs change appearance slightly between ASD and LRFD selections. There are also optional load duration factor entries for wood and masonry design.

© 2026 ENERCALC, LLC

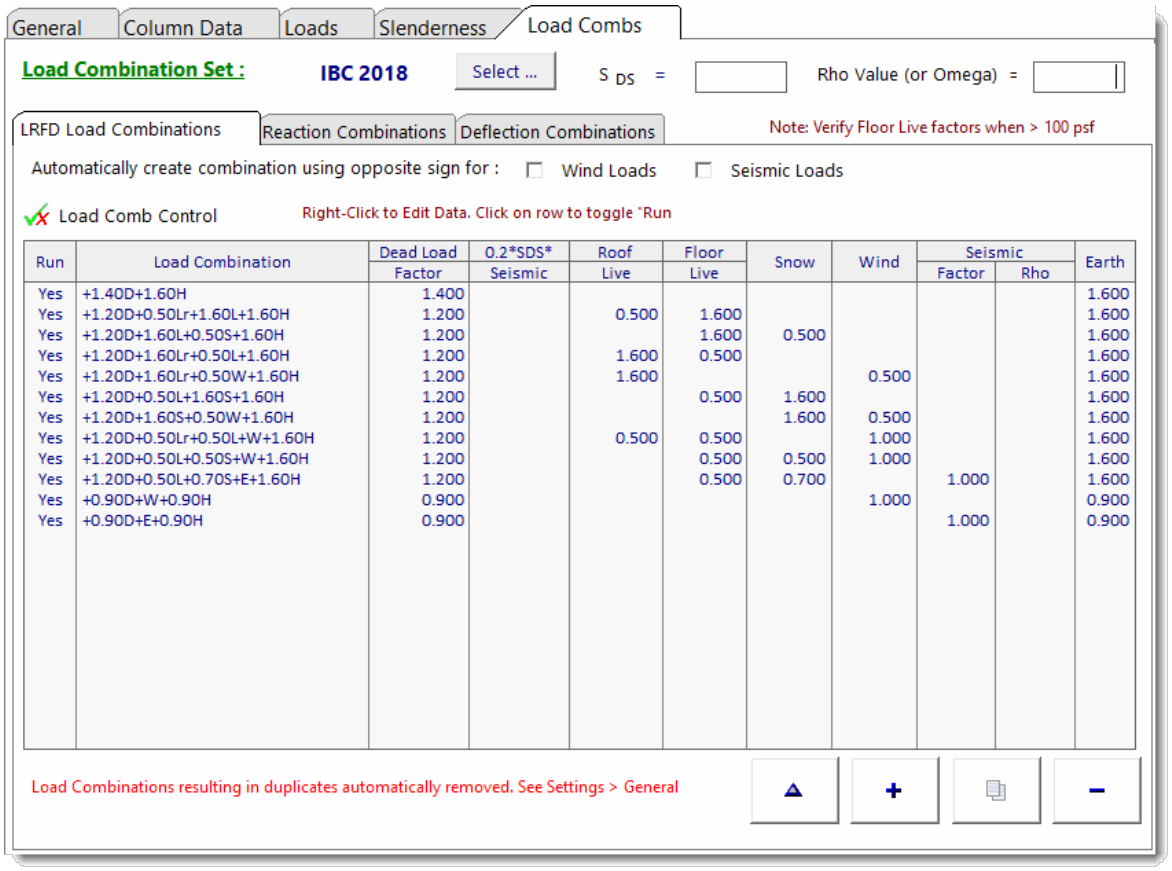
Please see the screen captures below for variations based on the selected material.

Steel ASD



Run	Load Combination	Dead Load Factor	0.2*SDS* Seismic	Roof Live	Floor Live	Snow	Wind	Seismic Factor	Seismic Rho	Earth
Yes	+D+H	1.000								1.000
Yes	+D+L+H	1.000			1.000					1.000
Yes	+D+Lr+H	1.000		1.000						1.000
Yes	+D+S+H	1.000				1.000				1.000
Yes	+D+0.750Lr+0.750L+H	1.000		0.750	0.750					1.000
Yes	+D+0.750L+0.750S+H	1.000			0.750	0.750				1.000
Yes	+D+0.60W+H	1.000					0.600			1.000
Yes	+D+0.70E+H	1.000						0.700		1.000
Yes	+D+0.750Lr+0.750L+0.450W+H	1.000		0.750	0.750		0.450			1.000
Yes	+D+0.750L+0.750S+0.450W+H	1.000			0.750	0.750	0.450			1.000
Yes	+D+0.750L+0.750S+0.5250E+H	1.000			0.750	0.750		0.525		1.000
Yes	+0.60D+0.60W+0.60H	0.600					0.600			0.600
Yes	+0.60D+0.70E+0.60H	0.600						0.700		0.600

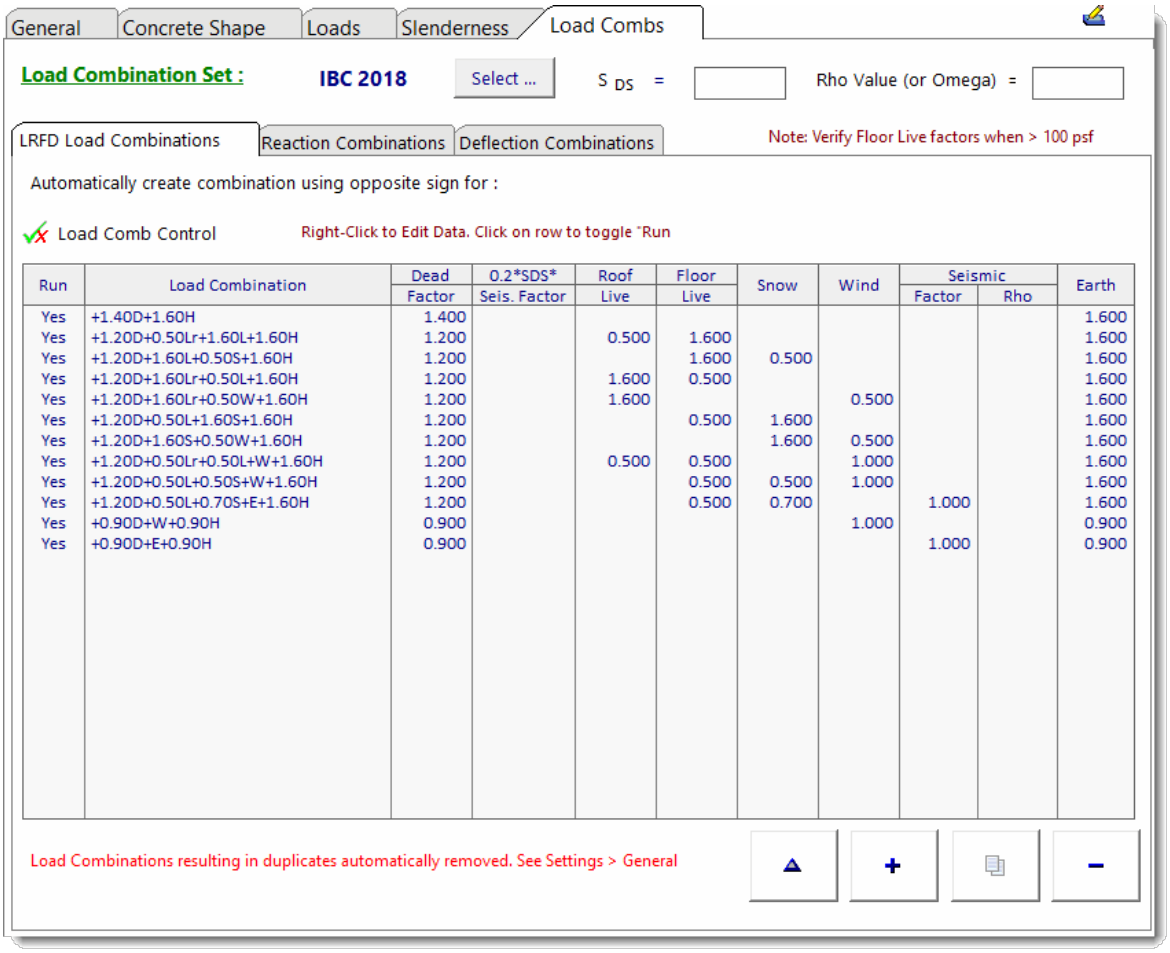
Steel LRFD



Run	Load Combination	Dead Load Factor	0.2*SDS* Seismic	Roof Live	Floor Live	Snow	Wind	Seismic Factor	Seismic Rho	Earth
Yes	+1.40D+1.60H	1.400								1.600
Yes	+1.20D+0.50Lr+1.60L+1.60H	1.200		0.500	1.600					1.600
Yes	+1.20D+1.60L+0.50S+1.60H	1.200			1.600	0.500				1.600
Yes	+1.20D+1.60Lr+0.50L+1.60H	1.200		1.600	0.500					1.600
Yes	+1.20D+1.60Lr+0.50W+1.60H	1.200		1.600			0.500			1.600
Yes	+1.20D+0.50L+1.60S+1.60H	1.200			0.500	1.600				1.600
Yes	+1.20D+1.60S+0.50W+1.60H	1.200				1.600	0.500			1.600
Yes	+1.20D+0.50Lr+0.50L+W+1.60H	1.200		0.500	0.500		1.000			1.600
Yes	+1.20D+0.50L+0.50S+W+1.60H	1.200			0.500	0.500	1.000			1.600
Yes	+1.20D+0.50L+0.70S+E+1.60H	1.200			0.500	0.700		1.000		1.600
Yes	+0.90D+W+0.90H	0.900					1.000			0.900
Yes	+0.90D+E+0.90H	0.900						1.000		0.900

© 2026 ENERCALC, LLC

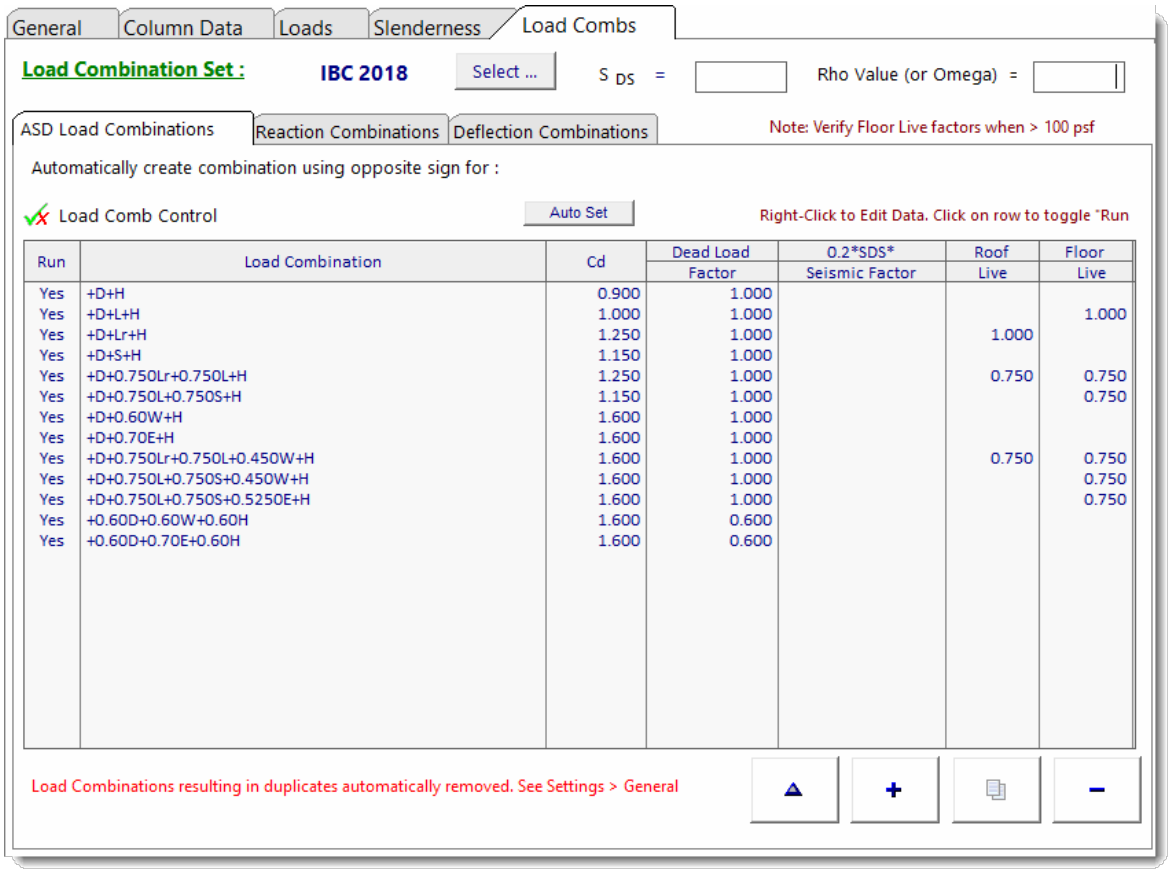
Concrete LRFD



Run	Load Combination	Dead Factor	0.2*SDS* Seis. Factor	Roof Live	Floor Live	Snow	Wind	Seismic Factor	Seismic Rho	Earth
Yes	+1.40D+1.60H	1.400								1.600
Yes	+1.20D+0.50Lr+1.60L+1.60H	1.200		0.500	1.600					1.600
Yes	+1.20D+1.60L+0.50S+1.60H	1.200			1.600	0.500				1.600
Yes	+1.20D+1.60Lr+0.50L+1.60H	1.200		1.600	0.500					1.600
Yes	+1.20D+1.60Lr+0.50W+1.60H	1.200		1.600			0.500			1.600
Yes	+1.20D+0.50L+1.60S+1.60H	1.200			0.500	1.600				1.600
Yes	+1.20D+1.60S+0.50W+1.60H	1.200				1.600	0.500			1.600
Yes	+1.20D+0.50Lr+0.50L+W+1.60H	1.200		0.500	0.500		1.000			1.600
Yes	+1.20D+0.50L+0.50S+W+1.60H	1.200			0.500	0.500	1.000			1.600
Yes	+1.20D+0.50L+0.70S+E+1.60H	1.200			0.500	0.700		1.000		1.600
Yes	+0.90D+W+0.90H	0.900					1.000			0.900
Yes	+0.90D+E+0.90H	0.900						1.000		0.900

© 2026 ENERCALC, LLC

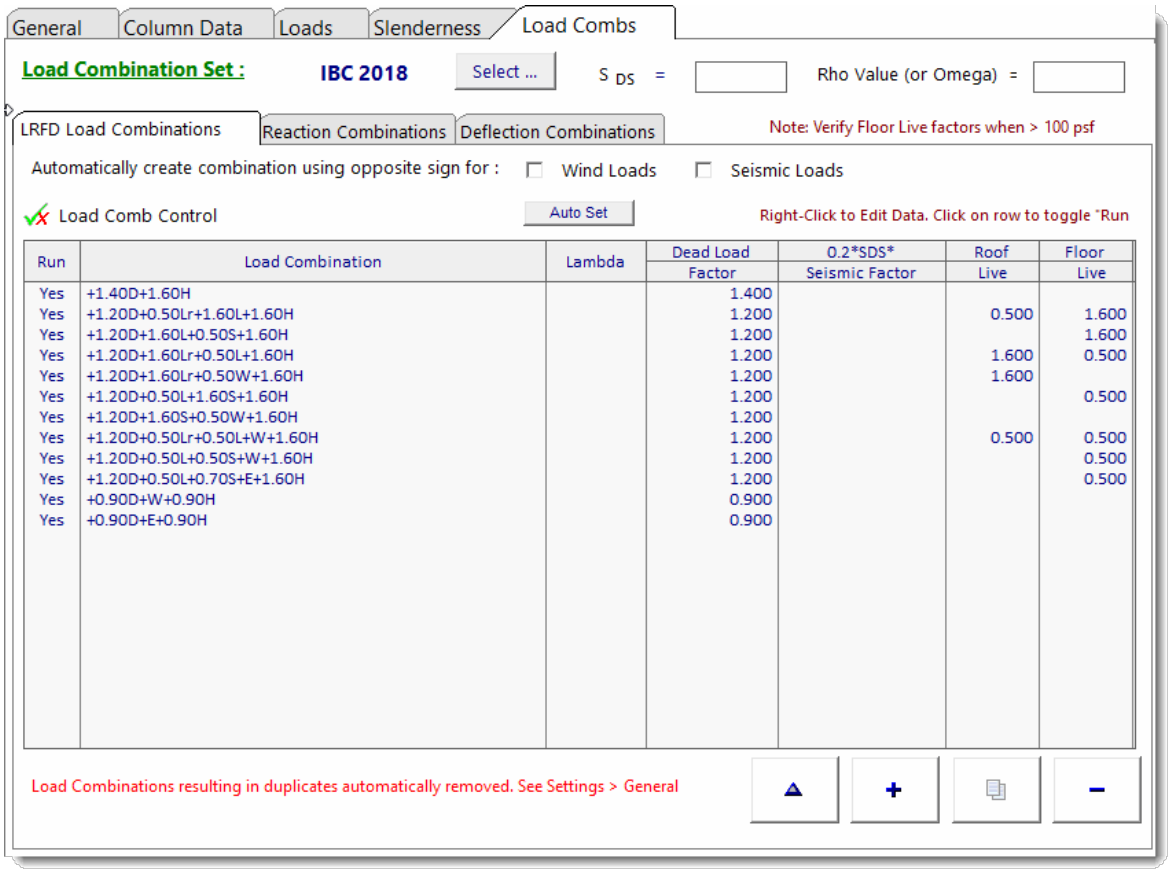
Wood ASD



Run	Load Combination	Cd	Dead Load Factor	0.2*SDS* Seismic Factor	Roof Live	Floor Live
Yes	+D+H	0.900	1.000			
Yes	+D+L+H	1.000	1.000			1.000
Yes	+D+Lr+H	1.250	1.000		1.000	
Yes	+D+S+H	1.150	1.000			
Yes	+D+0.750Lr+0.750L+H	1.250	1.000		0.750	0.750
Yes	+D+0.750L+0.750S+H	1.150	1.000			0.750
Yes	+D+0.60W+H	1.600	1.000			
Yes	+D+0.70E+H	1.600	1.000			
Yes	+D+0.750Lr+0.750L+0.450W+H	1.600	1.000		0.750	0.750
Yes	+D+0.750L+0.750S+0.450W+H	1.600	1.000			0.750
Yes	+D+0.750L+0.750S+0.5250E+H	1.600	1.000			0.750
Yes	+0.60D+0.60W+0.60H	1.600	0.600			
Yes	+0.60D+0.70E+0.60H	1.600	0.600			

© 2026 ENERCALC, LLC

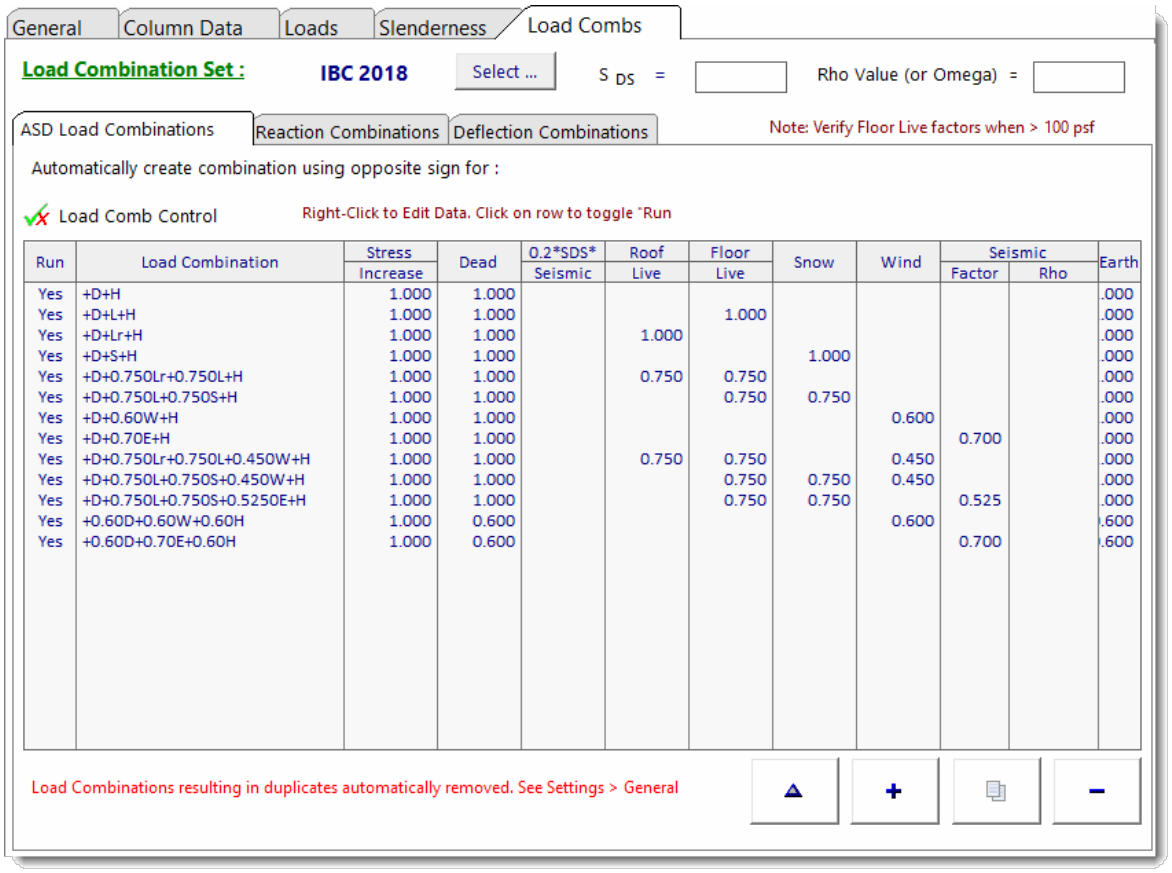
Wood LRFD



Run	Load Combination	Lambda	Dead Load Factor	0.2*SDS* Seismic Factor	Roof Live	Floor Live
Yes	+1.40D+1.60H		1.400			
Yes	+1.20D+0.50Lr+1.60L+1.60H		1.200		0.500	1.600
Yes	+1.20D+1.60L+0.50S+1.60H		1.200			1.600
Yes	+1.20D+1.60Lr+0.50L+1.60H		1.200		1.600	0.500
Yes	+1.20D+1.60Lr+0.50W+1.60H		1.200		1.600	
Yes	+1.20D+0.50L+1.60S+1.60H		1.200			0.500
Yes	+1.20D+1.60S+0.50W+1.60H		1.200			
Yes	+1.20D+0.50Lr+0.50L+W+1.60H		1.200		0.500	0.500
Yes	+1.20D+0.50L+0.50S+W+1.60H		1.200			0.500
Yes	+1.20D+0.50L+0.70S+E+1.60H		1.200			0.500
Yes	+0.90D+W+0.90H		0.900			
Yes	+0.90D+E+0.90H		0.900			

© 2026 ENERCALC, LLC

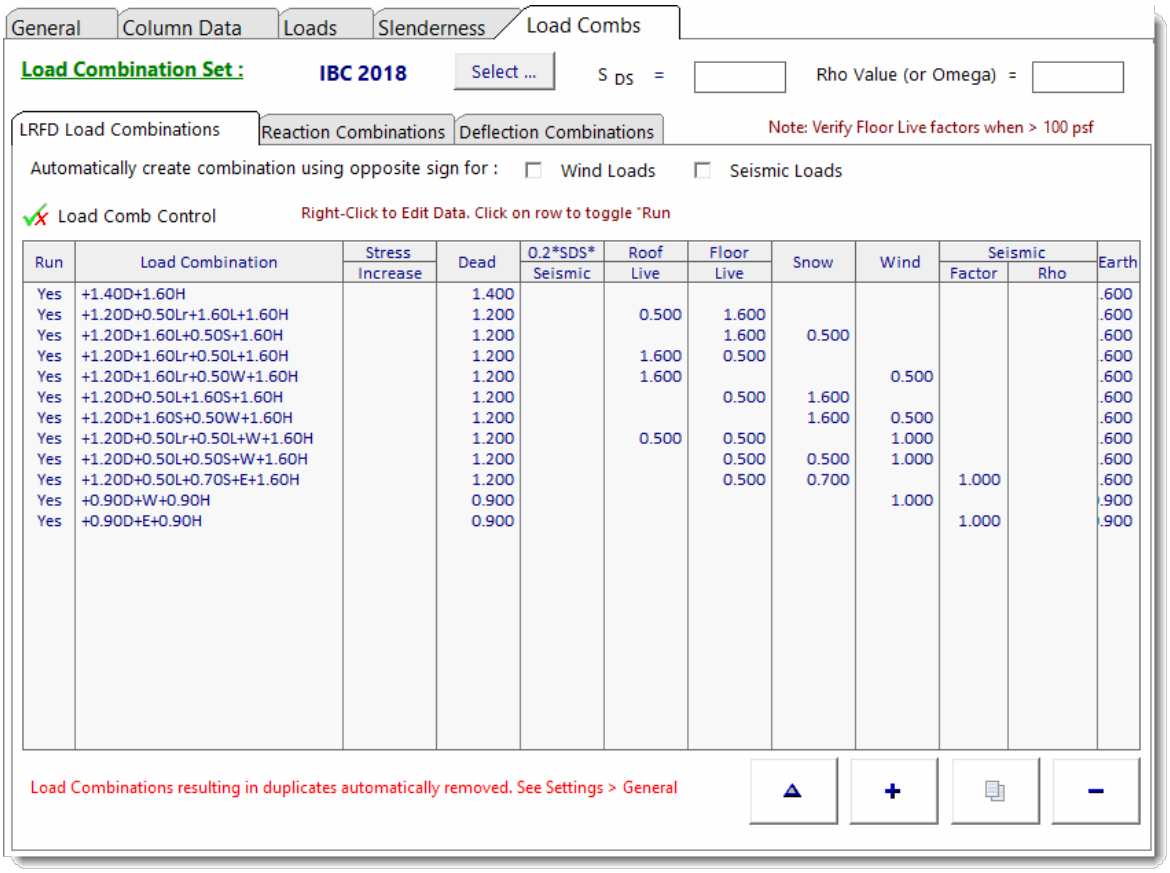
Masonry ASD



Run	Load Combination	Stress Increase	Dead	0.2*SDS* Seismic	Roof Live	Floor Live	Snow	Wind	Seismic Factor	Seismic Rho	Earth
Yes	+D+H	1.000	1.000								.000
Yes	+D+L+H	1.000	1.000			1.000					.000
Yes	+D+Lr+H	1.000	1.000		1.000						.000
Yes	+D+S+H	1.000	1.000				1.000				.000
Yes	+D+0.750Lr+0.750L+H	1.000	1.000		0.750	0.750					.000
Yes	+D+0.750L+0.750S+H	1.000	1.000			0.750	0.750				.000
Yes	+D+0.60W+H	1.000	1.000					0.600			.000
Yes	+D+0.70E+H	1.000	1.000						0.700		.000
Yes	+D+0.750Lr+0.750L+0.450W+H	1.000	1.000		0.750	0.750		0.450			.000
Yes	+D+0.750L+0.750S+0.450W+H	1.000	1.000			0.750	0.750	0.450			.000
Yes	+D+0.750L+0.750S+0.5250E+H	1.000	1.000			0.750	0.750		0.525		.000
Yes	+0.60D+0.60W+0.60H	1.000	0.600					0.600			.600
Yes	+0.60D+0.70E+0.60H	1.000	0.600						0.700		.600

© 2026 ENERCALC, LLC

Masonry SD



13.5.1 All Columns

The following topics generally apply to all column types except where noted specifically.

13.5.1.1 Column Slenderness

The Slenderness tab allows you to specify the column bracing, which will be considered in axial capacity calculations.

In most of the column modules there are two tabs: "Buckling ABOUT X-X" and "Buckling ABOUT Y-Y". Let's start with the all-important definition of the axis reference for slenderness. Note that the nomenclature used to define column slenderness was updated in May 2023 to bring ENERCALC in concert with the way the industry defines column buckling. Prior to May 2023, ENERCALC used definitions that referred to "buckling IN THE DIRECTION OF" an axis. After the change made in May 2023, ENERCALC uses definitions that refer to "buckling ABOUT" an axis.

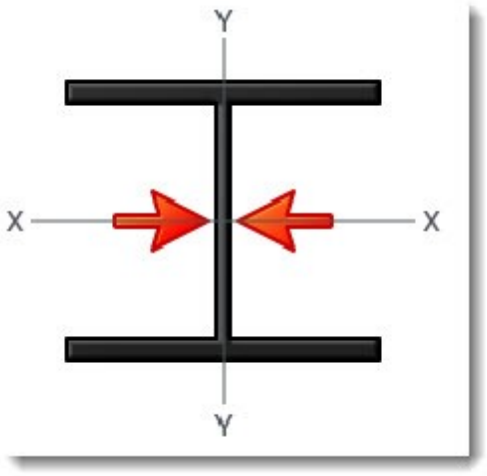
Buckling failure of a column can be thought of as an uncontrolled and excessive deflection about a particular axis. When defining slenderness, one of the important values is the distance between points that brace the column against buckling about a particular axis.

© 2026 ENERCALC, LLC

The column modules ask you to specify the distance between points of bracing that prevent column buckling about the member's local X-X or Y-Y axis.

(The **X-X axis** is always parallel to the "**width**" dimension of the column. The **Y-Y axis** is always parallel to the "**depth**" dimension of the column.)

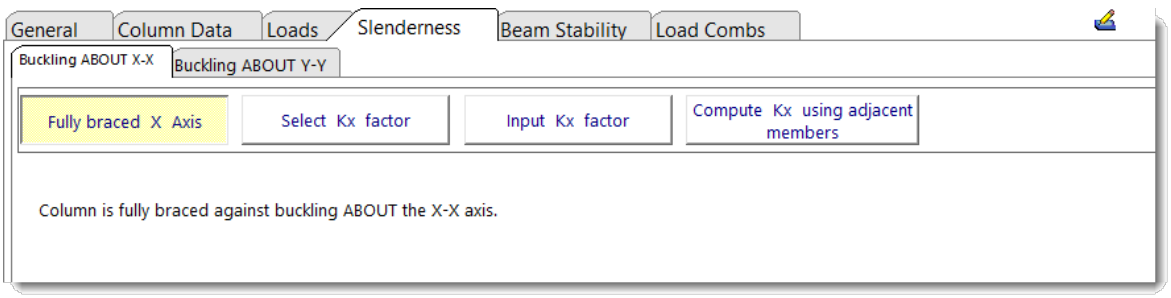
In addition to preventing column buckling about an axis, any brace that is defined in the column modules will ALSO be interpreted as a brace that stabilizes the compression flange of the column against lateral torsional buckling in the flexural design. Here's where things get fun. When we define the spacing of bracing that is effective in resisting column buckling **about the Y axis**, that bracing is in place to prevent the column from **translating in the X axis** direction, like this:



In the screen capture immediately below, we have selected [**Fully braced "X" Axis**]. This means that the column is fully braced against buckling about its X-X axis, which is parallel to the width dimension.

All materials, column fully braced

This selection sets the column as fully braced, and no slenderness effects will be evaluated.

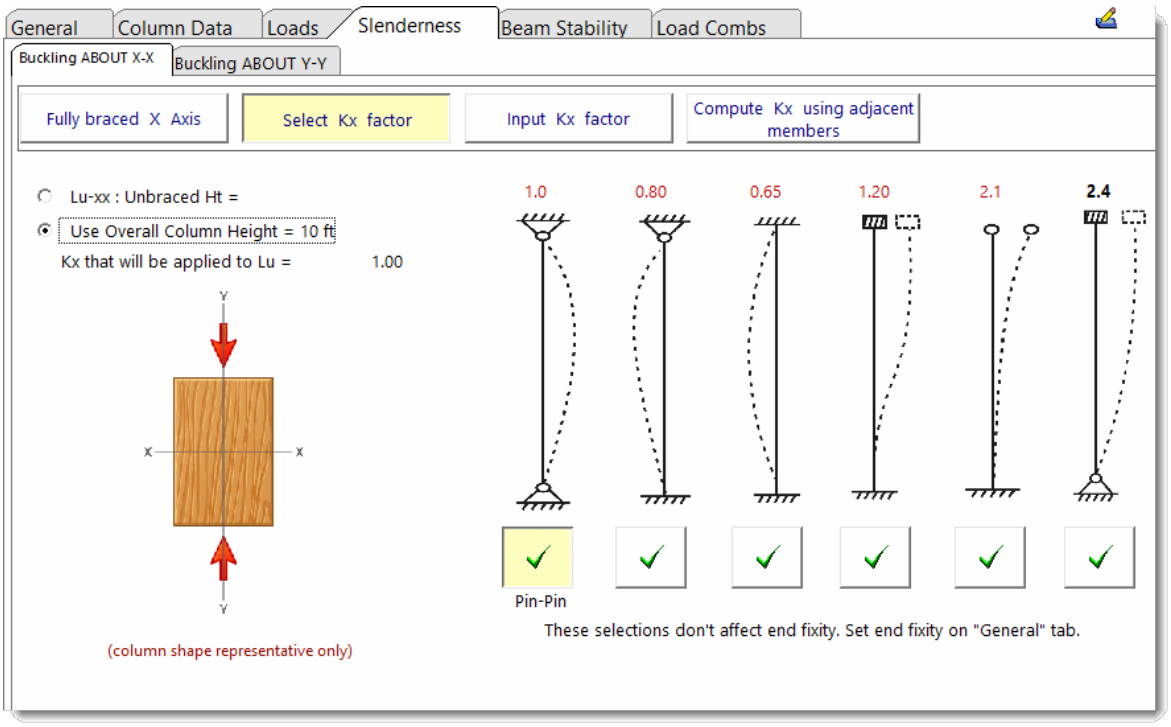


© 2026 ENERCALC, LLC

All materials, typical simple slenderness specification

This selection allows you to enter the unbraced length to use for the column slenderness calculation. Also available are selections for the typical effective length multipliers ("K" factors) for various end conditions.

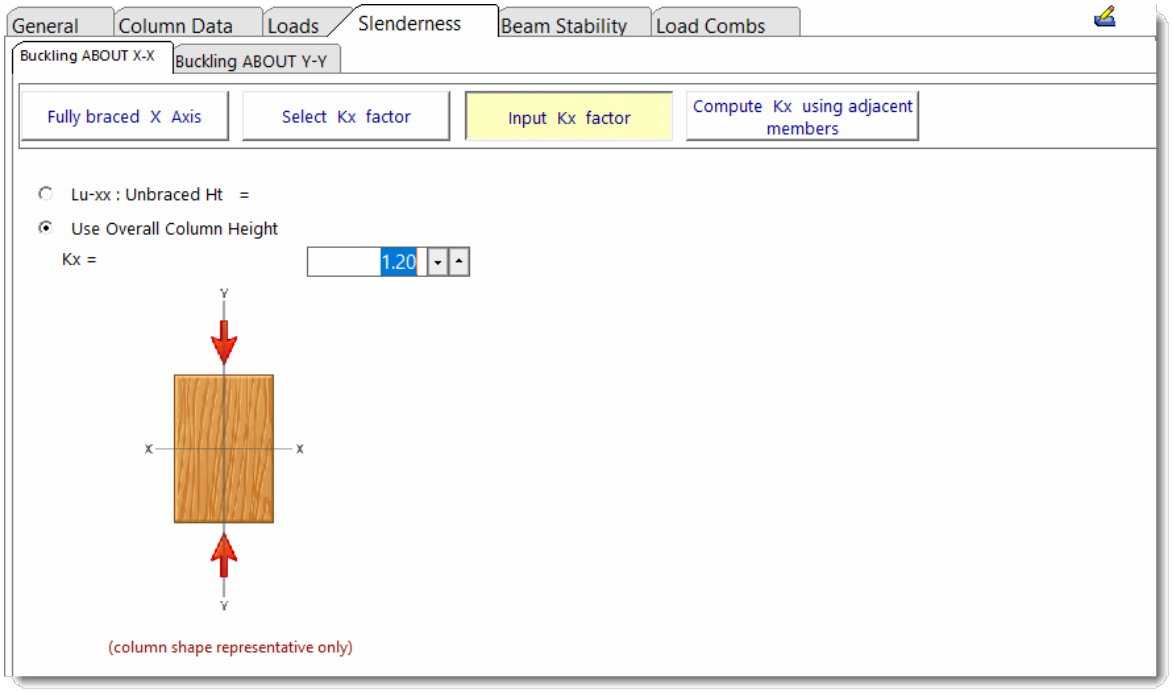
Note! End fixity is specified on a different tab in the column modules. These slenderness factors do not alter the end fixity you specified for the column, nor do they get determined automatically by the end fixities that you specified.



All materials, typical slenderness with user-defined effective length factor "K"

This selection allows you to enter the unbraced height and the effective length factor "K" to use for the column slenderness calculation.

Note! End fixity is specified on a different tab in the column modules. These slenderness factors do not alter the end fixity you specified for the column, nor do they get determined automatically by the end fixities that you specified.

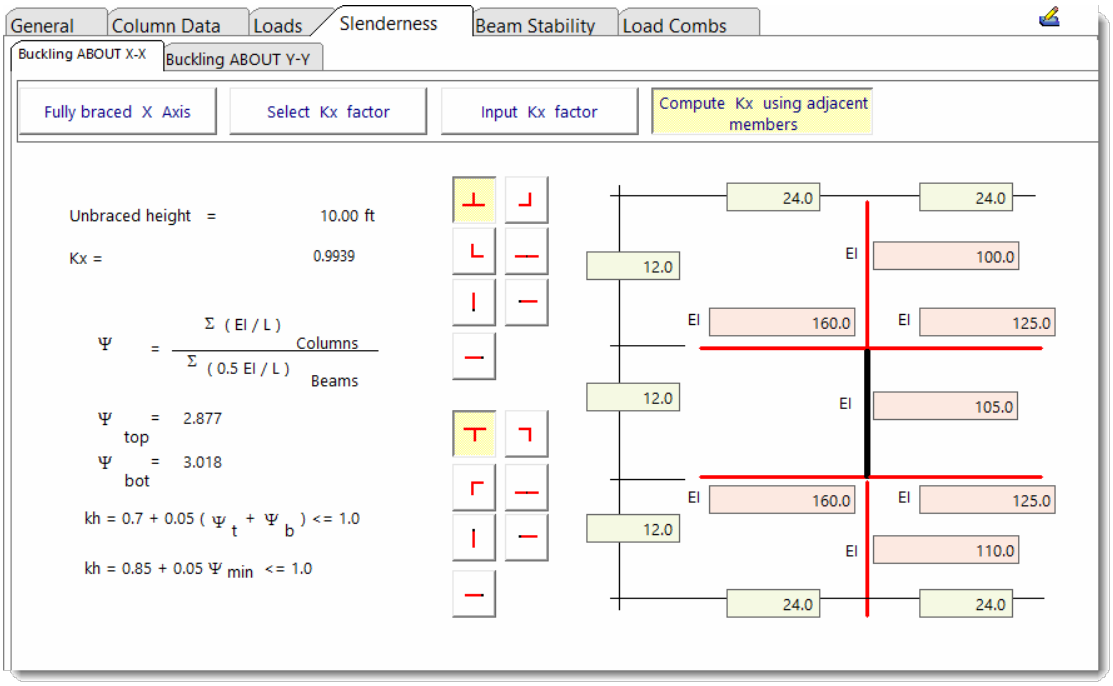


© 2026 ENERCALC, LLC

All Materials, non-sway slenderness calculation by calculating K based on stiffness of adjacent members

There is also an option to Compute K using adjacent members. This advanced selection lets you select the framing condition above and below the column, and using the entered lengths and EI values, it will use standard equations for Non-Sway columns to calculate the effective "K" factor.

Keep in mind that this option is used to define the relative stiffnesses of the framing in the plane of buckling that is being considered. For example, when using the X-X Axis Column Slenderness tab, the plane under consideration is the plane in which the column's X-X axis lies.

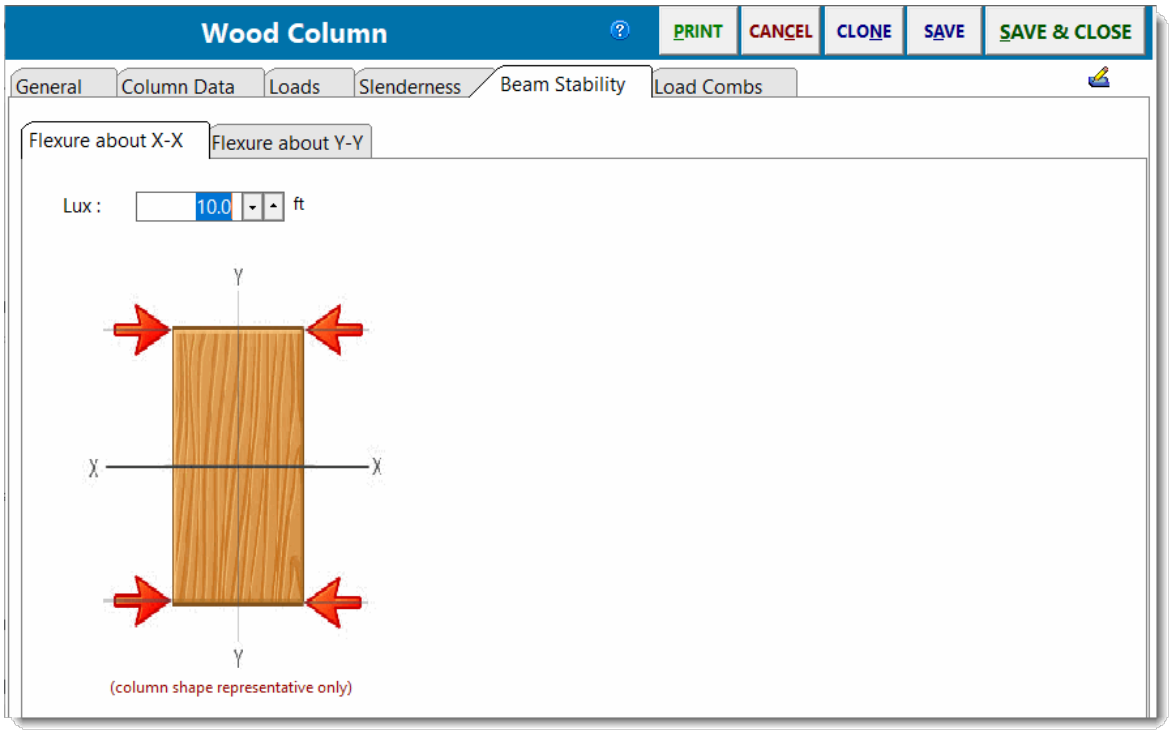


13.5.1.2 Beam Stability (in Steel Column and Wood Column modules)

Older versions of the column modules only had the Column Slenderness tab input to take some sort of guess as to what the bracing condition is for defining the lateral-torsional buckling strength. But there may be times where the bracing that is effective in resisting Euler (column) buckling would not be considered as effective in resisting lateral-torsional buckling. So the Beam Stability tab allows the user to more effectively define the column and the bracing for all of its failure modes.

In the Wood Column module, the program evaluates the dimensions of the column member and offers an input for the unbraced length to apply when evaluating flexure about the strong axis of the member.

© 2026 ENERCALC, LLC



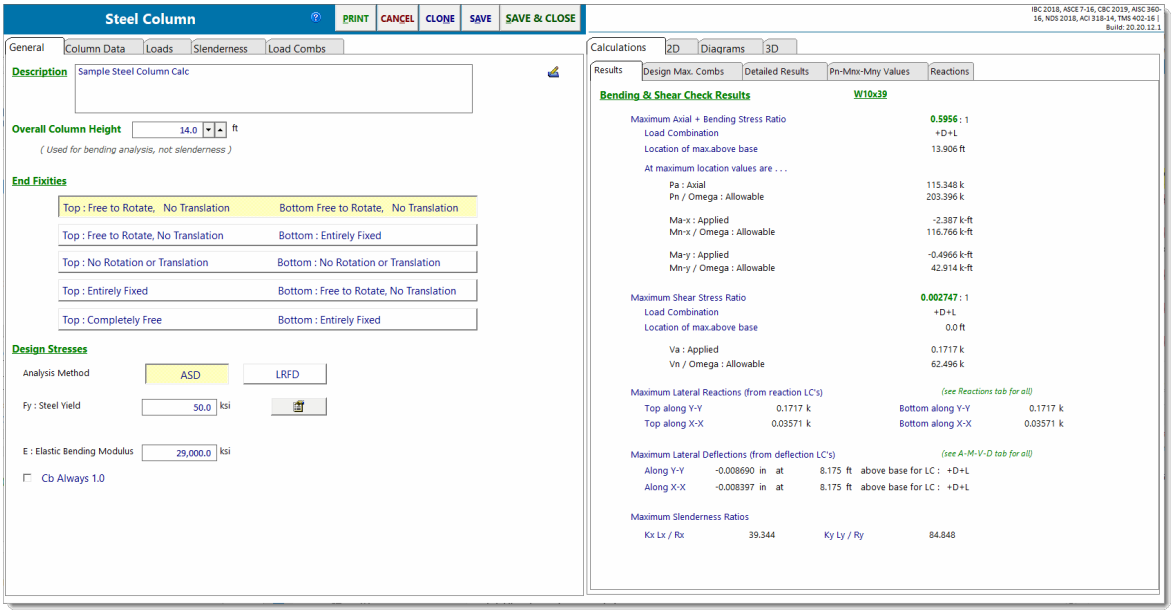
The tab for the weak axis simply indicates that lateral-torsional buckling is not a consideration when flexure occurs about the weak axis.

© 2026 ENERCALC, LLC

13.5.2 Steel Column

Verification Example

This module designs steel columns that are subject to axial loads and lateral bending loads about both axes.



The user can select ASD or LRFD methods and has access to a complete database of steel section sizes.

All calculations are according to AISC 360.

The screen capture below shows the full screen for steel column design. See items below for descriptions of items that are specific to the steel column design module.

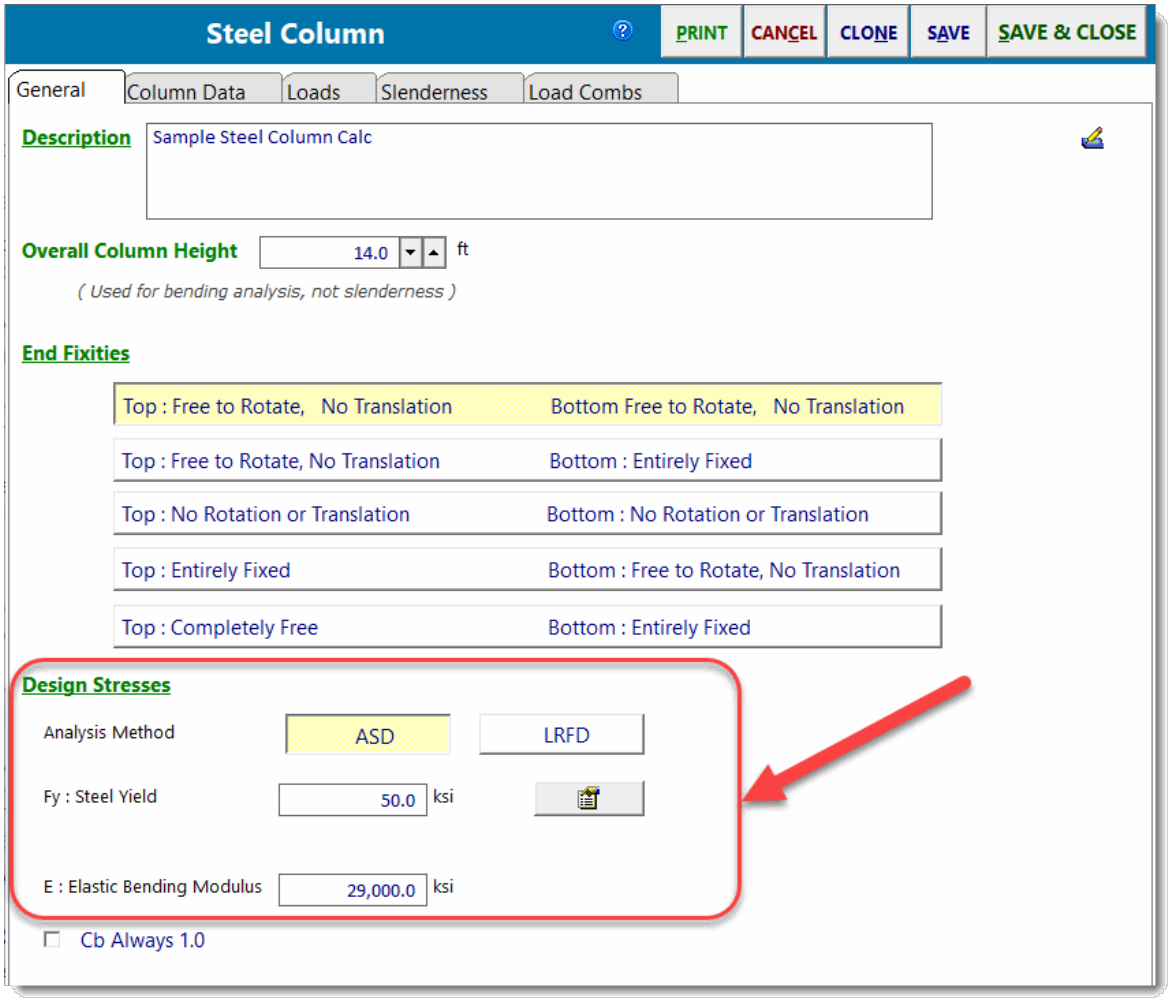
For general description of the module, end fixity, loads, and load combinations click here 810. For slenderness description click here 821.

General

The area indicated in the screen capture below is specific to the steel column selection. Here you can specify ASD or LRFD design procedure and specify the yield strength and elastic modulus of the steel member to be used.

© 2026 ENERCALC, LLC

Click the button to display the Steel Fy Selections dialog to select a standard grade.



Column Data

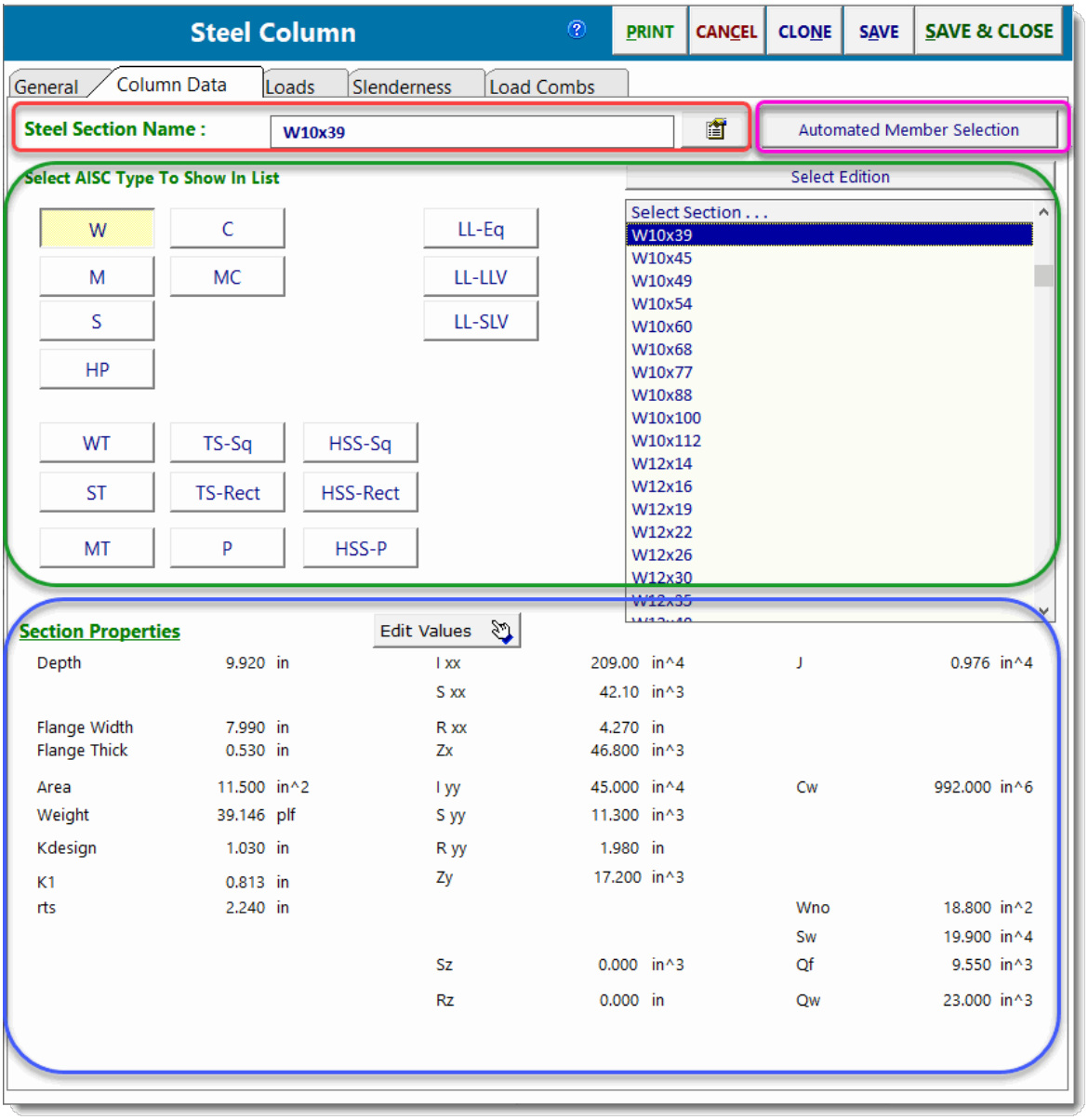
All items on this tab are specific to a steel column.

© 2026 ENERCALC, LLC

The item circled in **red** is where you can type in the typical section name, press [Tab], and the section will be searched and retrieved from the built-in database. Or you can click the button, and you can select a section from the built-in AISC database.

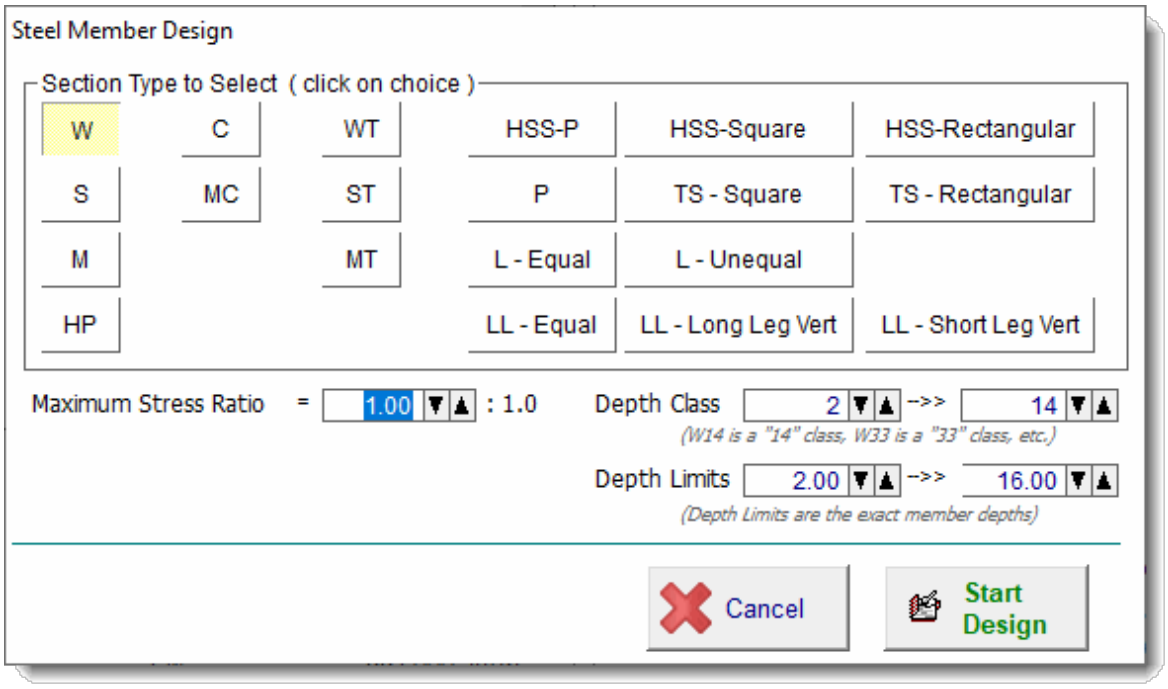
The items circled in **blue** are the section properties for the section you have chosen.

The item circled in **green** is the Quick-List". Click on one of the many buttons with a section letter and the full list of those sections will be displayed to the right. Simply click on a section and it will be assigned to the red and blue areas.



© 2026 ENERCALC, LLC

The button circled in **pink** will display the Steel Member Design dialog (see below).



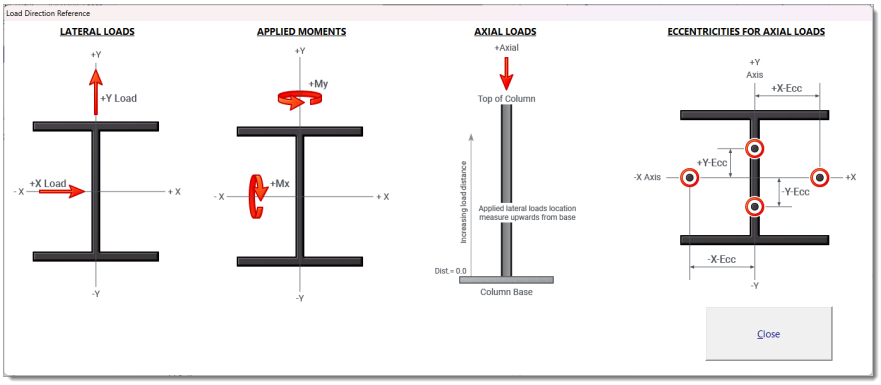
This tool checks all of the steel sections for the selected type (W is selected in this case). A section will be judged to pass if the maximum stress ratio entered is not exceeded and the depth measurements and depth class are not exceeded.

The term "Depth Class" refers to the nominal dimension of the family to which the section belongs (ex: W14). It does not refer to the actual depth of the section. All sections starting with "W14" are of the "14" depth class.

© 2026 ENERCALC, LLC

Loads

Load Reference Guide: Clicking this button opens a pop-up window that displays the positive sign conventions used in the module.



For more information on the Loads tab, please refer to the Columns 810 topic.

Results

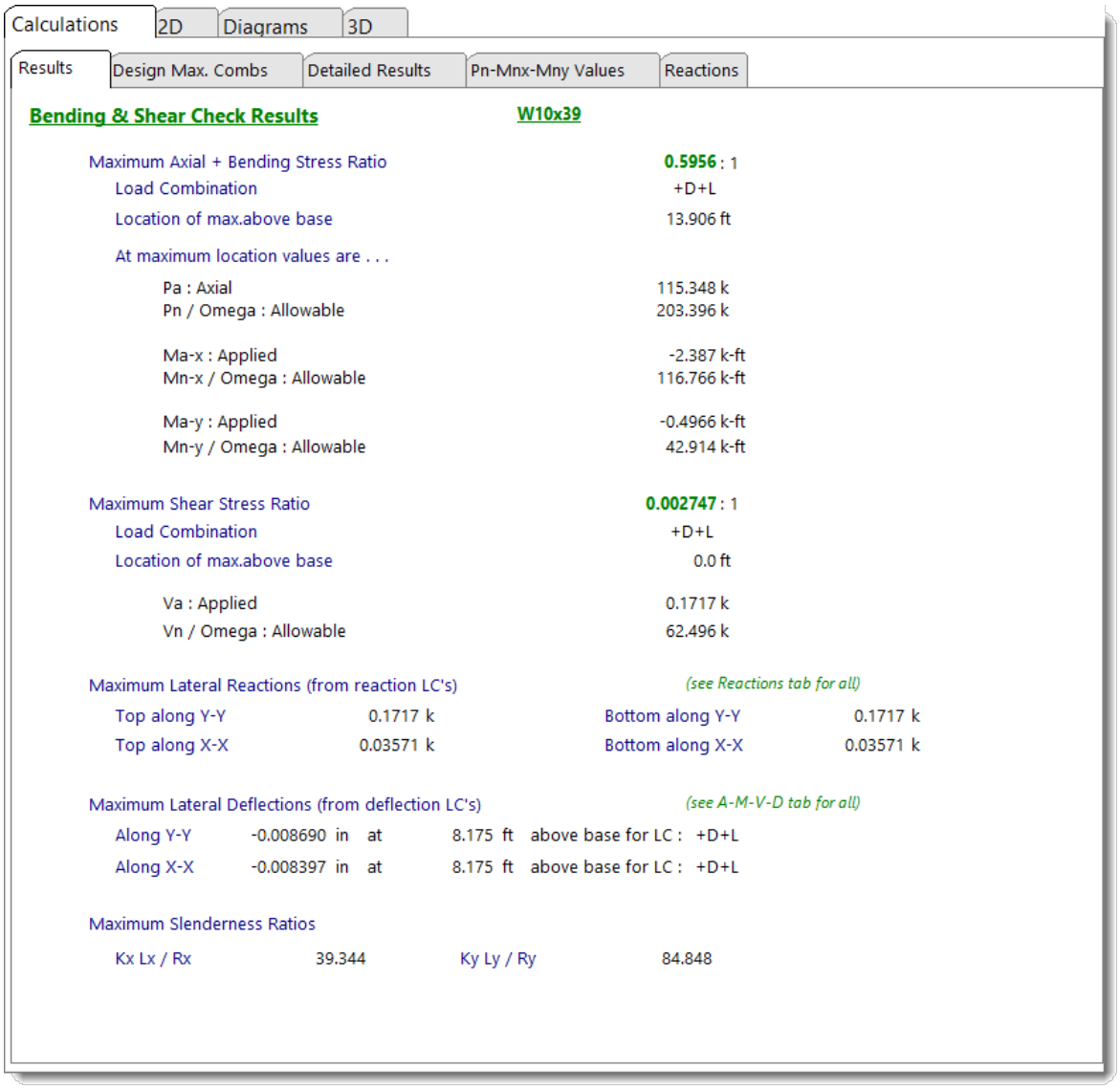
This tab provides a summary of the stress ratios, reactions and deflections for the column.

Max Axial + Bending Stress Ratio is the governing load combination for the column. Listed is the governing load load combination, the AISC formulas that are used, and the location of the maximum stress ratio above the base of the column. Please note that the maximum stress ratio is being reported (along with its location) because it governs the overall design, even though it may not necessarily be the highest axial stress or the highest bending stress experienced anywhere in the column.

Max Shear Stress Ratio will probably never govern for a normally loaded column design. But it is presented here with the governing load combination, location and allowable/actual stress values.

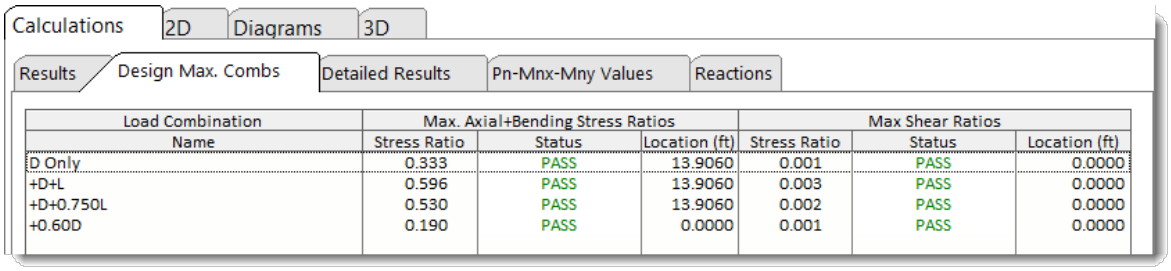
Lateral Load Reactions and **Deflections** are the result of applied lateral loads.

© 2026 ENERCALC, LLC



Design Maximum Combinations

This tab lists the resulting maximum stress ratios for each load combination. This list is created by examining the detailed list (on the next tab) and determining the governing stress ratios for each load combination.



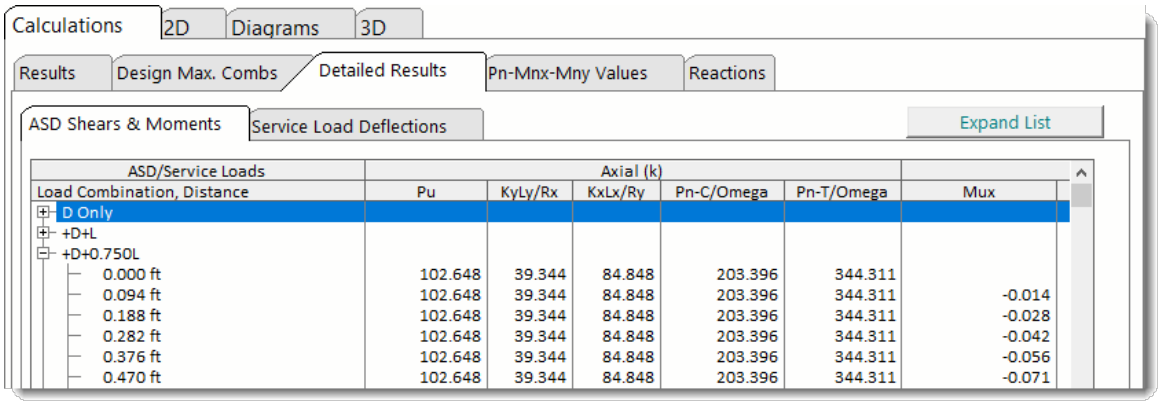
Load Combination	Max. Axial+Bending Stress Ratios			Max Shear Ratios		
Name	Stress Ratio	Status	Location (ft)	Stress Ratio	Status	Location (ft)
D Only	0.333	PASS	13.9060	0.001	PASS	0.0000
+D+L	0.596	PASS	13.9060	0.003	PASS	0.0000
+D+0.750L	0.530	PASS	13.9060	0.002	PASS	0.0000
+0.60D	0.190	PASS	0.0000	0.001	PASS	0.0000

© 2026 ENERCALC, LLC

Detailed Results - Shears & Moments

This tab lists the detailed results at small increments along the height of the column, for each load combination.
For consistency, all of the column headings are taken directly from AISC.

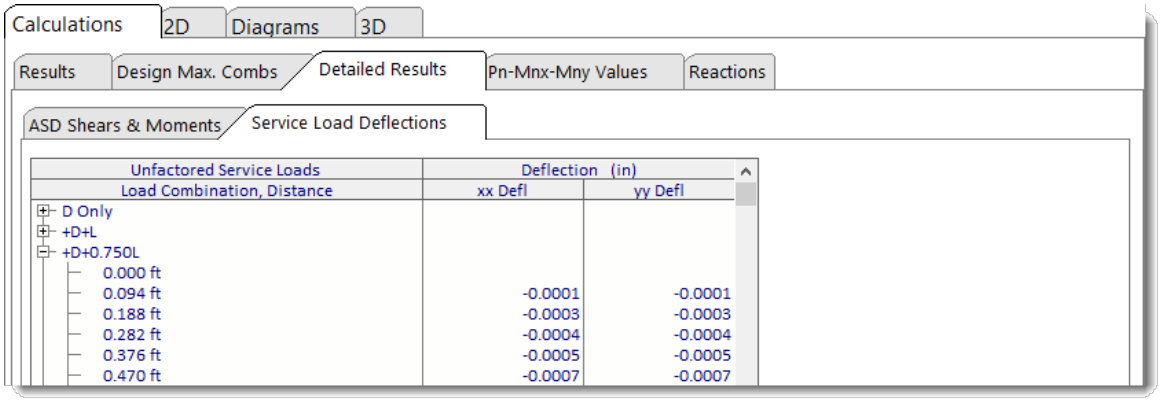
Note! This list scrolls to the right to display more information.



ASD/Service Loads	Axial (k)					
Load Combination, Distance	Pu	KyLy/Rx	KxLx/Ry	Pn-C/Omega	Pn-T/Omega	Mux
D Only						
+D+L						
+D+0.750L						
0.000 ft	102.648	39.344	84.848	203.396	344.311	
0.094 ft	102.648	39.344	84.848	203.396	344.311	-0.014
0.188 ft	102.648	39.344	84.848	203.396	344.311	-0.028
0.282 ft	102.648	39.344	84.848	203.396	344.311	-0.042
0.376 ft	102.648	39.344	84.848	203.396	344.311	-0.056
0.470 ft	102.648	39.344	84.848	203.396	344.311	-0.071

Detailed Results - Service Load Deflections

This tab reports the deflections at incremental locations along the height of the column, for each service load condition (i.e. for individual load cases and for a set of built-in service load combinations), along each axis.

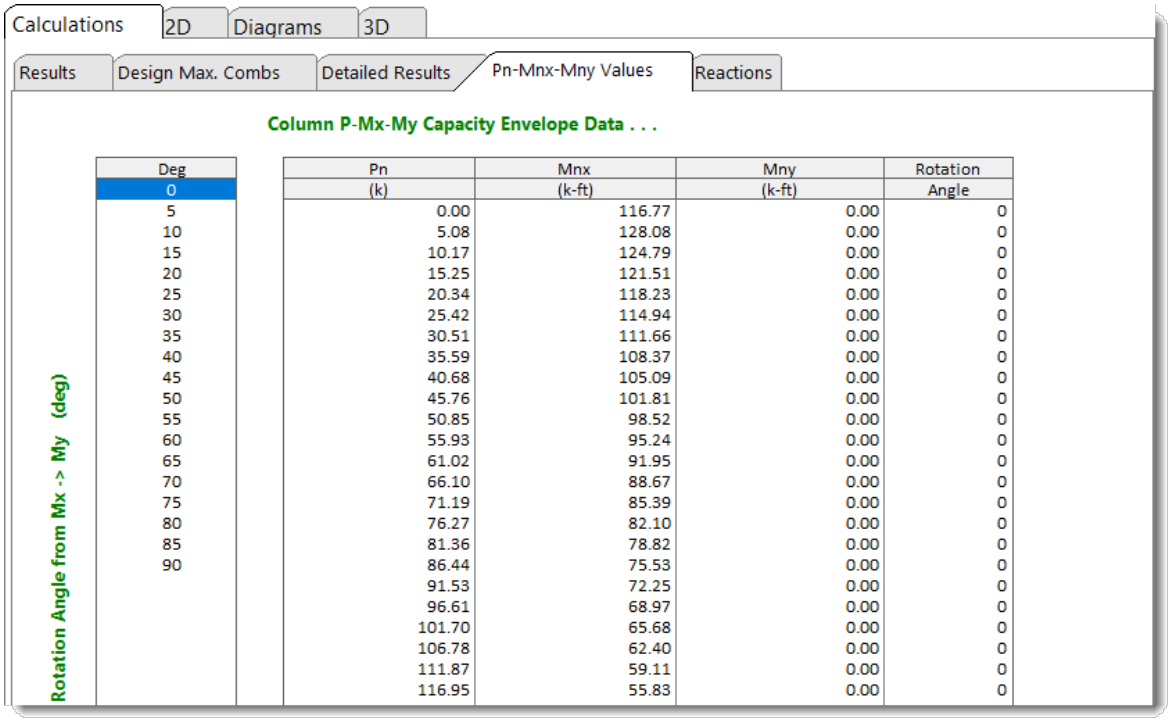


Unfactored Service Loads	Deflection (in)	
Load Combination, Distance	xx Defl	yy Defl
D Only		
+D+L		
+D+0.750L		
0.000 ft		
0.094 ft	-0.0001	-0.0001
0.188 ft	-0.0003	-0.0003
0.282 ft	-0.0004	-0.0004
0.376 ft	-0.0005	-0.0005
0.470 ft	-0.0007	-0.0007

© 2026 ENERCALC, LLC

Pn - Mnx - Mny Values

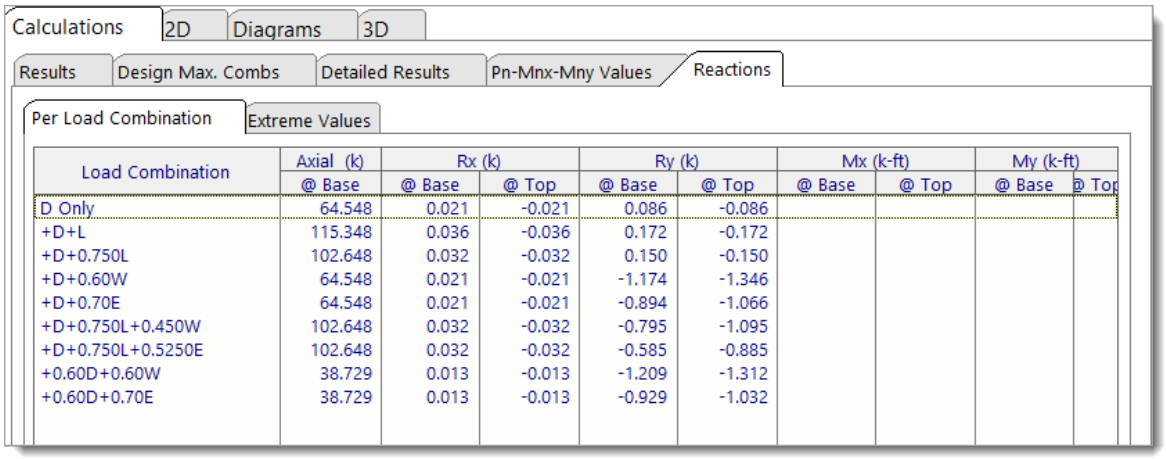
This tab provides capacity envelope data.



Deg	Pn (k)	Mnx (k-ft)	Mny (k-ft)	Rotation Angle
0	0.00	116.77	0.00	0
5	5.08	128.08	0.00	0
10	10.17	124.79	0.00	0
15	15.25	121.51	0.00	0
20	20.34	118.23	0.00	0
25	25.42	114.94	0.00	0
30	30.51	111.66	0.00	0
35	35.59	108.37	0.00	0
40	40.68	105.09	0.00	0
45	45.76	101.81	0.00	0
50	50.85	98.52	0.00	0
55	55.93	95.24	0.00	0
60	61.02	91.95	0.00	0
65	66.10	88.67	0.00	0
70	71.19	85.39	0.00	0
75	76.27	82.10	0.00	0
80	81.36	78.82	0.00	0
85	86.44	75.53	0.00	0
90	91.53	72.25	0.00	0
	96.61	68.97	0.00	0
	101.70	65.68	0.00	0
	106.78	62.40	0.00	0
	111.87	59.11	0.00	0
	116.95	55.83	0.00	0

Reactions - Per Load Combination

This tab displays the support reactions calculated for each load combination defined in the Load Combs > Reaction Combinations tab.



Load Combination	Axial (k)	Rx (k)		Ry (k)		Mx (k-ft)		My (k-ft)	
	@ Base	@ Base	@ Top	@ Base	@ Top	@ Base	@ Top	@ Base	@ Top
D Only	64.548	0.021	-0.021	0.086	-0.086				
+D+L	115.348	0.036	-0.036	0.172	-0.172				
+D+0.750L	102.648	0.032	-0.032	0.150	-0.150				
+D+0.60W	64.548	0.021	-0.021	-1.174	-1.346				
+D+0.70E	64.548	0.021	-0.021	-0.894	-1.066				
+D+0.750L+0.450W	102.648	0.032	-0.032	-0.795	-1.095				
+D+0.750L+0.5250E	102.648	0.032	-0.032	-0.585	-0.885				
+0.60D+0.60W	38.729	0.013	-0.013	-1.209	-1.312				
+0.60D+0.70E	38.729	0.013	-0.013	-0.929	-1.032				

Reactions are reported at both the base and top of the column (where applicable), and include the following:

Axial @ Base (k): Axial reaction at the base of the column.

© 2026 ENERCALC, LLC

- Positive values indicate compression (downward force).
- Negative values indicate tension (uplift).

Rx @ Base/Top (k): Horizontal reaction in the X-direction at the base and top.

- Positive values act in the global +X direction (to the right when looking down at the top of the column).
- Negative values act in the global -X direction (to the left when looking down at the top of the column).

Ry @ Base/Top (k): Horizontal reaction in the Y-direction at the base and top.

- Positive values act in the global +Y direction (upward on screen when looking down at the top of the column).
- Negative values act in the global -Y direction (downward on screen when looking down at the top of the column).

Mx @ Base/Top (k-ft): Moment reaction about the X-axis at the base and top.

- Positive values follow the right-hand rule. Point your right thumb in the direction of the +X-axis (to the right when looking down at the top of the column), and your curled fingers indicate the direction of the moment.
- Negative values follow the right-hand rule. Point your right thumb in the direction of the -X-axis (to the left when looking down at the top of the column), and your curled fingers indicate the direction of the moment.

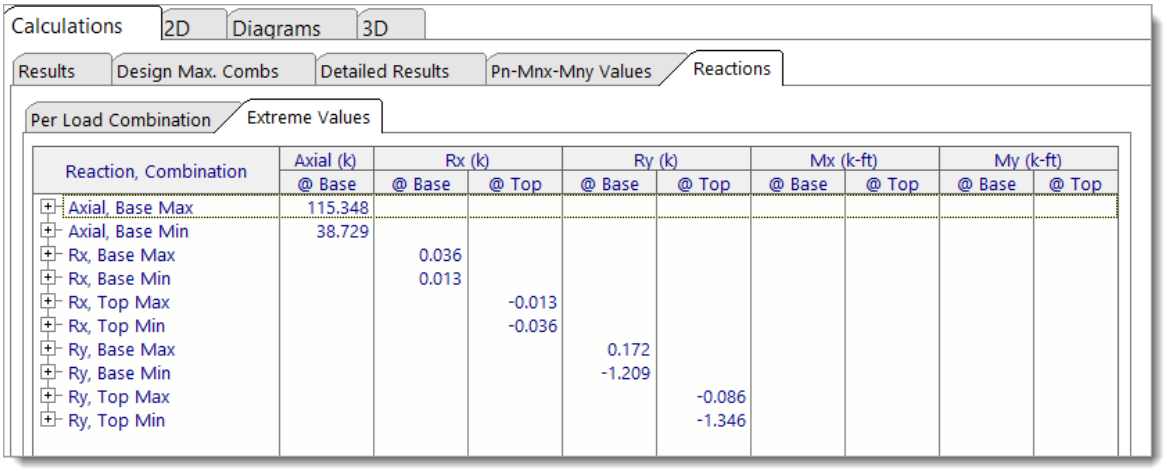
My @ Base/Top (k-ft): Moment reaction about the Y-axis at the base and top.

- Positive values follow the right-hand rule. Point your right thumb in the direction of the +Y-axis (upward on screen when looking down at the top of the column), and your curled fingers indicate the direction of the moment.
- Negative values follow the right-hand rule. Point your right thumb in the direction of the -Y-axis (downward on screen when looking down at the top of the column), and your curled fingers indicate the direction of the moment.

© 2026 ENERCALC, LLC

Reactions - Extreme Values

This tab summarizes the maximum and minimum reaction forces and moments that occur at the top and bottom of the column across all defined load combinations.



Reaction, Combination	Axial (k)	Rx (k)		Ry (k)		Mx (k-ft)		My (k-ft)	
	@ Base	@ Base	@ Top	@ Base	@ Top	@ Base	@ Top	@ Base	@ Top
Axial, Base Max	115.348								
Axial, Base Min	38.729								
Rx, Base Max		0.036							
Rx, Base Min		0.013							
Rx, Top Max			-0.013						
Rx, Top Min			-0.036						
Ry, Base Max				0.172					
Ry, Base Min				-1.209					
Ry, Top Max					-0.086				
Ry, Top Min					-1.346				

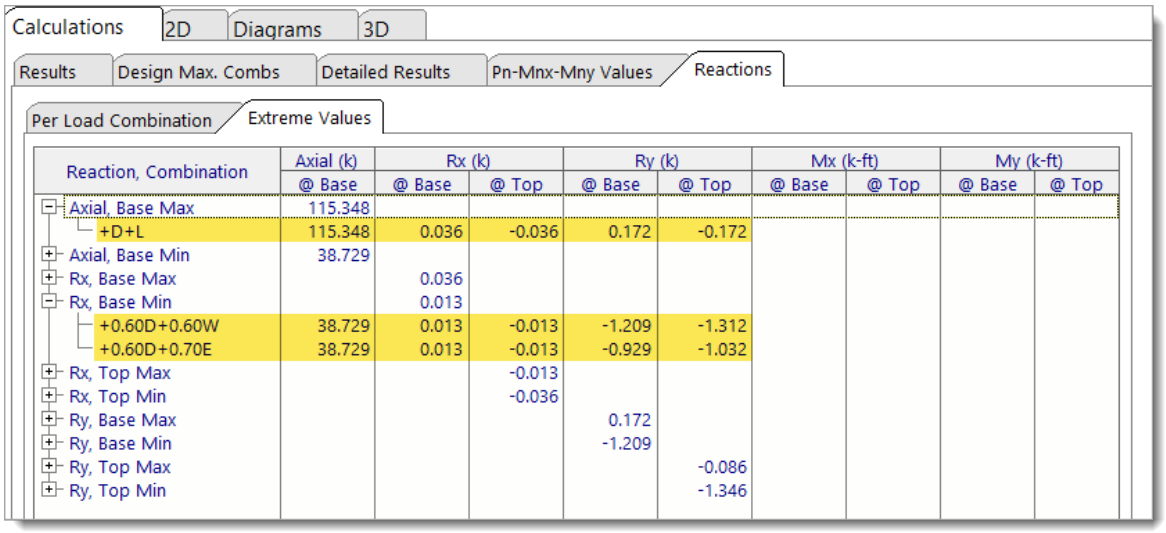
The following values are reported (where applicable):

- **Axial, Base Max** and **Axial, Base Min:** The max and min vertical axial reactions at the base of the column.
- **Rx, Base Max** and **Rx, Base Min:** The max and min horizontal reactions in the X-direction at the base and top.
- **Ry, Base Max** and **Ry, Base Min:** The max and min horizontal reactions in the Y-direction at the base and top.
- **Mx, Base Max** and **Mx, Base Min:** The max and min moment reaction about the X-axis at the base and top.
- **My, Base Max** and **My, Base Min:** The max and min moment reaction about the Y-axis at the base and top.

Notes:

- "Max" values represent the "most positive" results (i.e., closest to positive infinity), while "Min" values represent the "most negative" results (i.e., closest to negative infinity).
- Reaction components with no results are omitted from the table. For example, a cantilever column will not display any Top extreme values, since no reactions exist there.
- Zero values are shown as blank in the table for clarity.

© 2026 ENERCALC, LLC



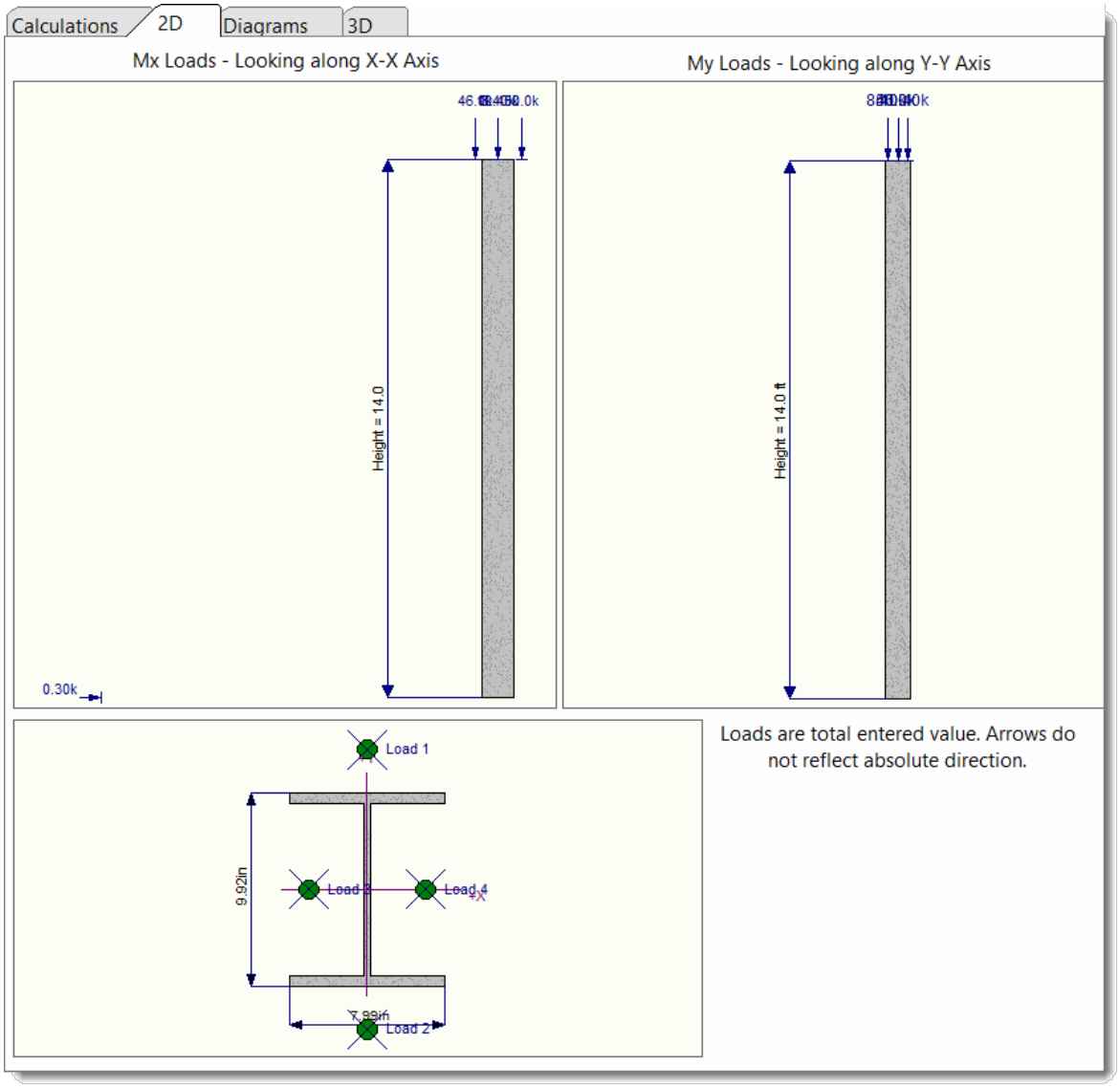
Reaction, Combination	Axial (k)	Rx (k)		Ry (k)		Mx (k-ft)		My (k-ft)	
	@ Base	@ Base	@ Top	@ Base	@ Top	@ Base	@ Top	@ Base	@ Top
Axial, Base Max	115.348								
+D+L	115.348	0.036	-0.036	0.172	-0.172				
Axial, Base Min	38.729								
Rx, Base Max		0.036							
Rx, Base Min		0.013							
+0.60D+0.60W	38.729	0.013	-0.013	-1.209	-1.312				
+0.60D+0.70E	38.729	0.013	-0.013	-0.929	-1.032				
Rx, Top Max			-0.013						
Rx, Top Min			-0.036						
Ry, Base Max				0.172					
Ry, Base Min				-1.209					
Ry, Top Max					-0.086				
Ry, Top Min					-1.346				

The results are presented in a tree-structured table. Each row shows an extreme value and can be expanded to reveal:

- The load combination(s) that produced that extreme value.
- The associated reaction components for those combinations.
- If multiple load combinations result in the same extreme value, all contributing combinations will be listed.

© 2026 ENERCALC, LLC

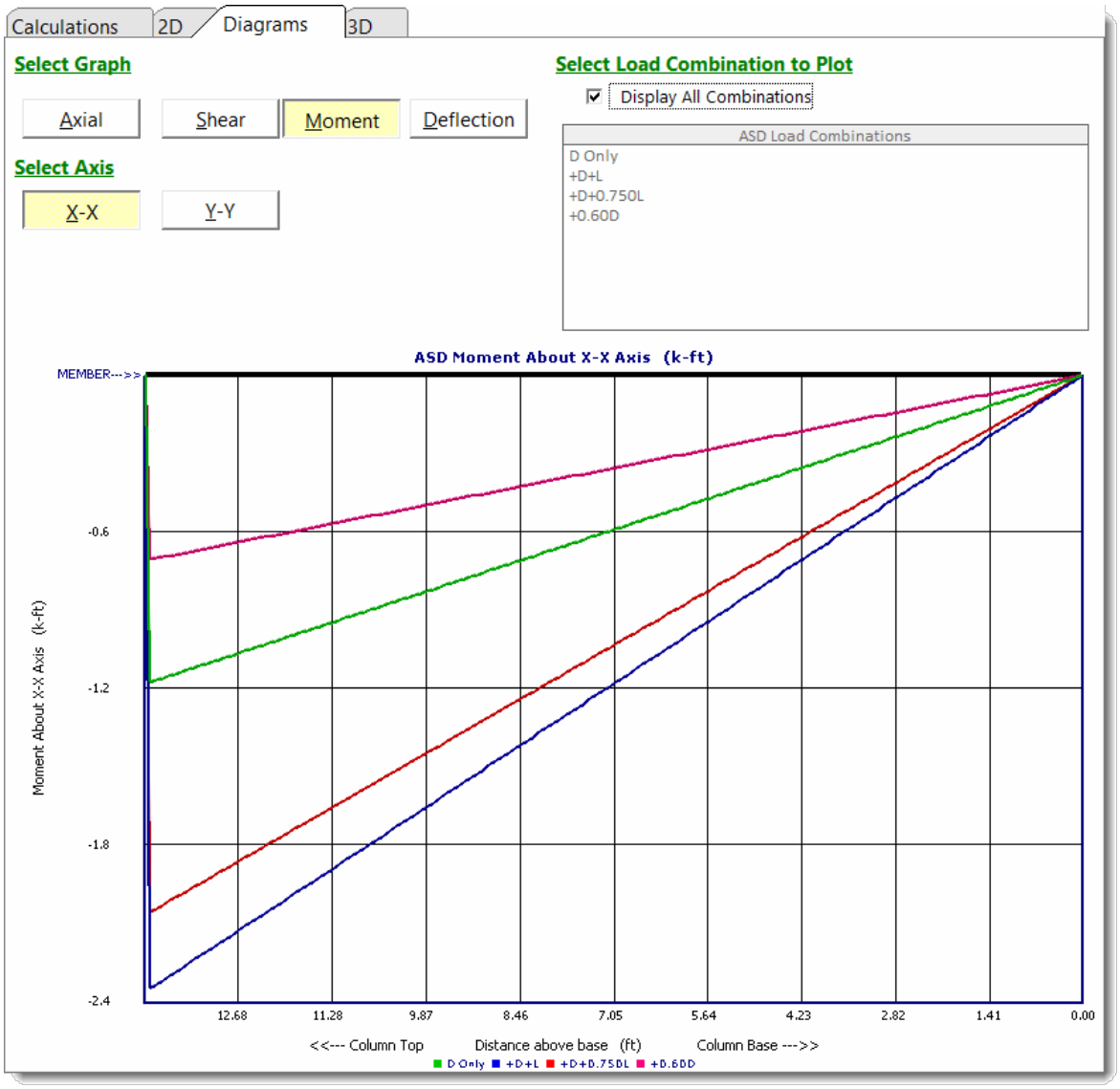
2D Sketch



© 2026 ENERCALC, LLC

Diagrams (Axial-Shear-Moment-Deflection)

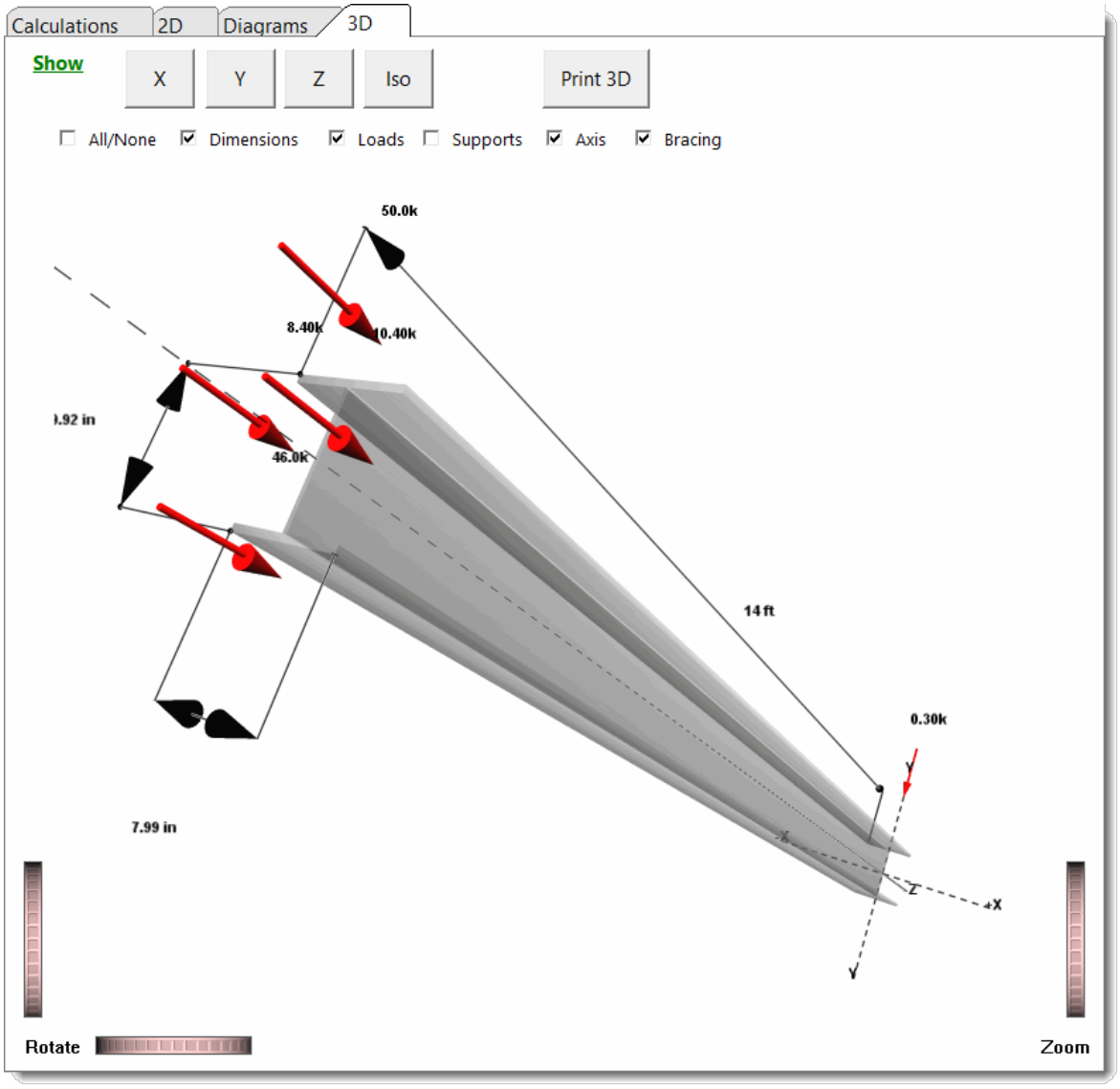
This tab provides comprehensive charting capability to view graphs of Axial load, Shear, Moment, and Deflection along the length of the member. Note that the graphs are oriented such that the right end of the graph represents the column base, and the left end of the graph represents the column top. This was done to maximize the scale of the graph based on the screen area available.



© 2026 ENERCALC, LLC

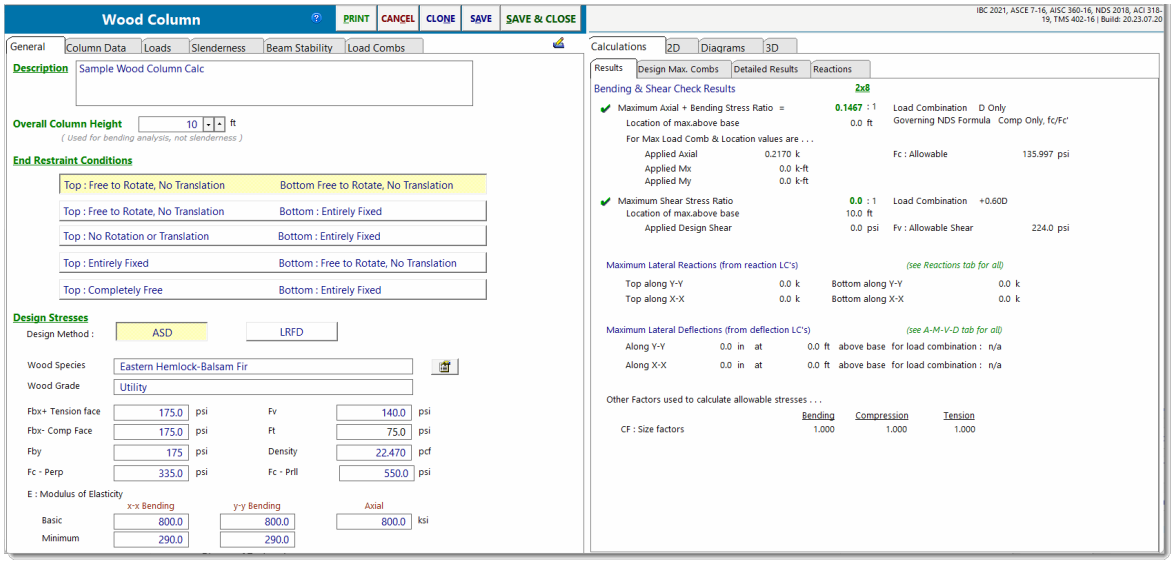
3D Rendering

This tab displays the 3D rendering of the column and offers various display options.



13.5.3 Wood Column

This module designs wood columns that are subject to axial loads and lateral bending loads about both axes.



The user can select ASD or LRFD design methods and has access to a large built-in database of wood sizes and NDS species stress grades. Values of K_F and phi are automatically determined and applied for the LRFD method.

All calculations are according to the NDS code.

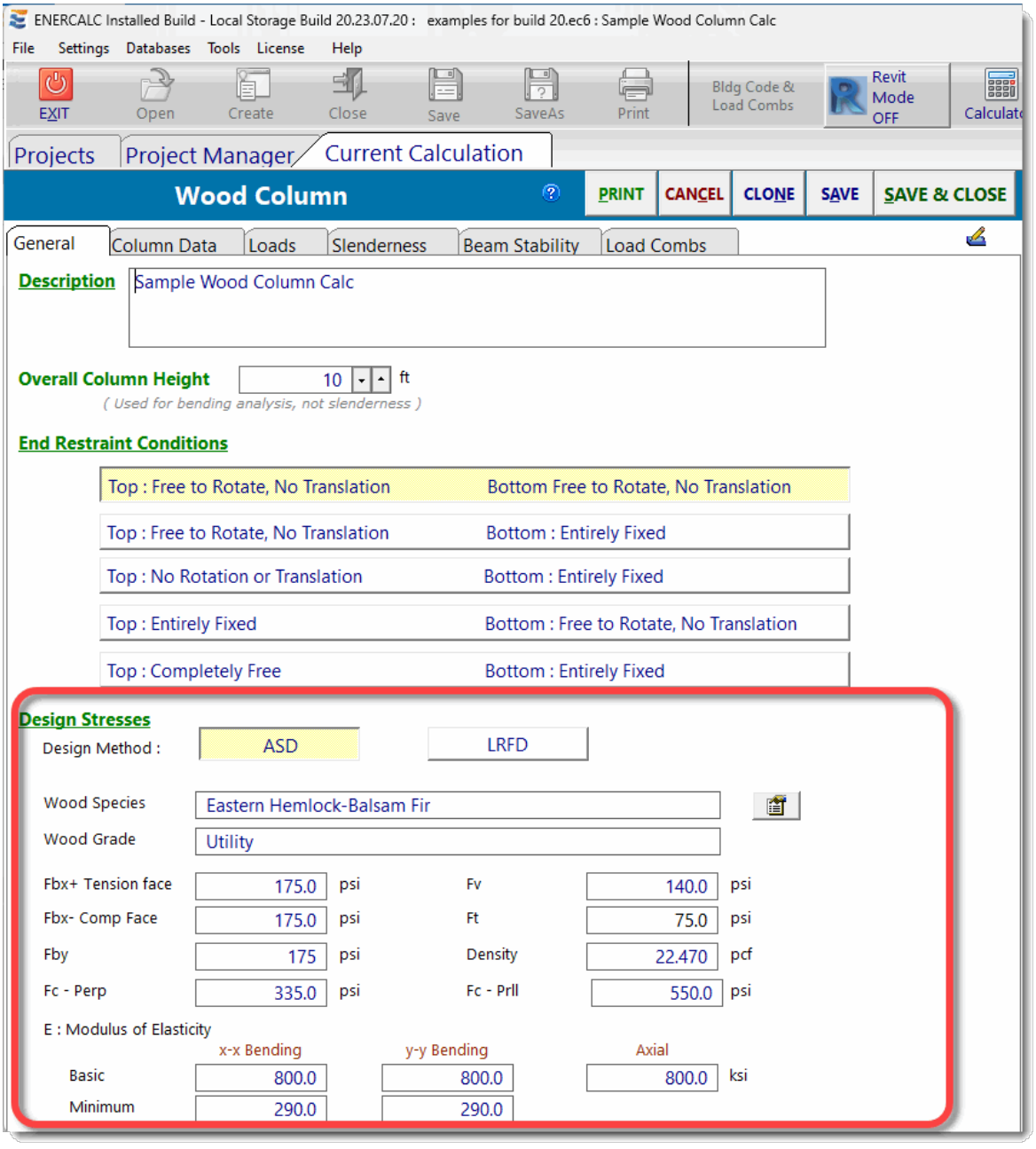
The screen capture below shows the full screen for wood column design. See items below for descriptions of items that are specific to the Wood Column module.

For general description of the module, end fixity, loads, and load combinations click here 810. For slenderness description click here 821.

© 2026 ENERCALC, LLC

General

The Design Stresses area bubbled in the screen capture below is unique to the wood column selection. This area enables you to specify the base design values for the wood species and grade of interest.



You can either enter these values manually, or you can click the button to display the Wood Stress Database.

© 2026 ENERCALC, LLC

Column Data

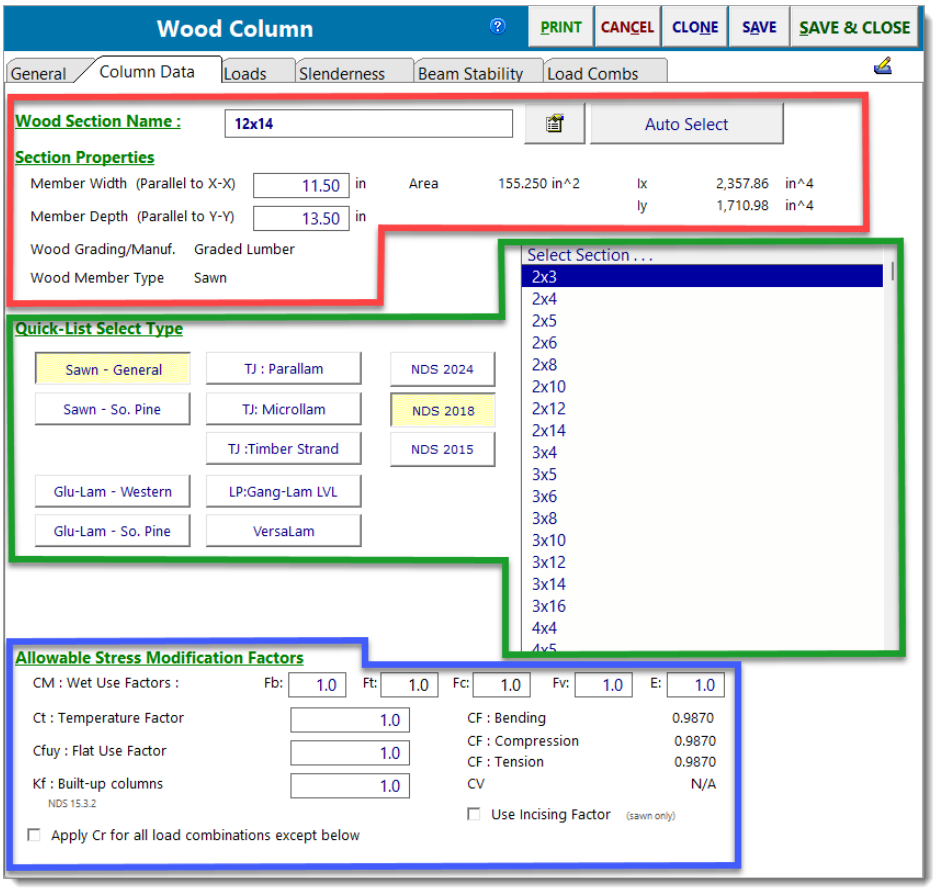
All of the information on this tab is unique to a wood column.

The area bubbled in **red** is where you specify the column cross section. Use the button to display the built-in database of wood sections (solid-sawn, glulam, and engineered wood products are available). You can also enter the values manually.

The area bubbled in **blue** provides allowable stress modification factors that you can specify. Please note that C_F or C_V values are automatically filled in. C_F values are determined from the size and stress grade of the member (No. 1 and Utility grades have different values). C_V values are calculated when a glu-lam section is specified. Note that this section allows for the specification of the Repetitive Member factor. If the Repetitive Member factor option is selected, the program then offers the option to specify a value for the Wall Stud Repetitive Member Factor as defined by the NDS Special Design Provisions for Wind and Seismic (SDPWS). If the option is selected to specify a Wall Stud Repetitive Member Factor, it will only be applied to load combinations that include wind. Be sure to review the appropriate section of SDPWS for requirements on the use of this factor, as well as the values to be used for various sizes of dimension lumber.

The area in **green** provides quick access to the built-in wood section database. Simply click the button of the section type and the list on the right will be populated automatically from the appropriate database. Then just click on a section to have its data loaded into the entries in the **red** area.

© 2026 ENERCALC, LLC

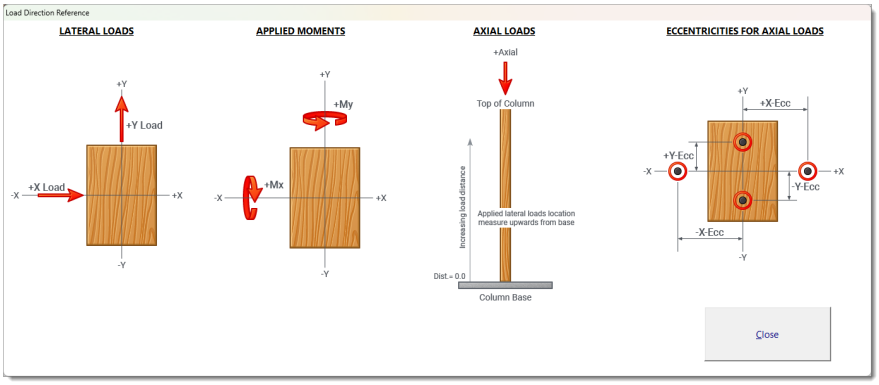


Note: The flat use factor will only be applied if it is specified by the user. In addition, it is important to understand how the factor will be applied in situations where built-up columns are designed. The flat use factor is supposed to be applied when bending occurs about the weak axis of the ***individual laminations***. But the program does not actually understand the orientation of individual laminations in a built-up column as of July 2018. So for consistency, the program is set up to look at the ***overall dimensions*** of a built-up column cross section, and then to apply the flat use factor only when considering bending about the overall weak axis. This requires the designer's consideration, because the weak axis of the overall built-up section may or may not correspond to the weak axis of the individual laminations.

© 2026 ENERCALC, LLC

Loads

Load Reference Guide: Clicking this button opens a pop-up window that displays the positive sign conventions used in the module.



For more information on the Loads tab, please refer to the Columns 810 topic.

Results

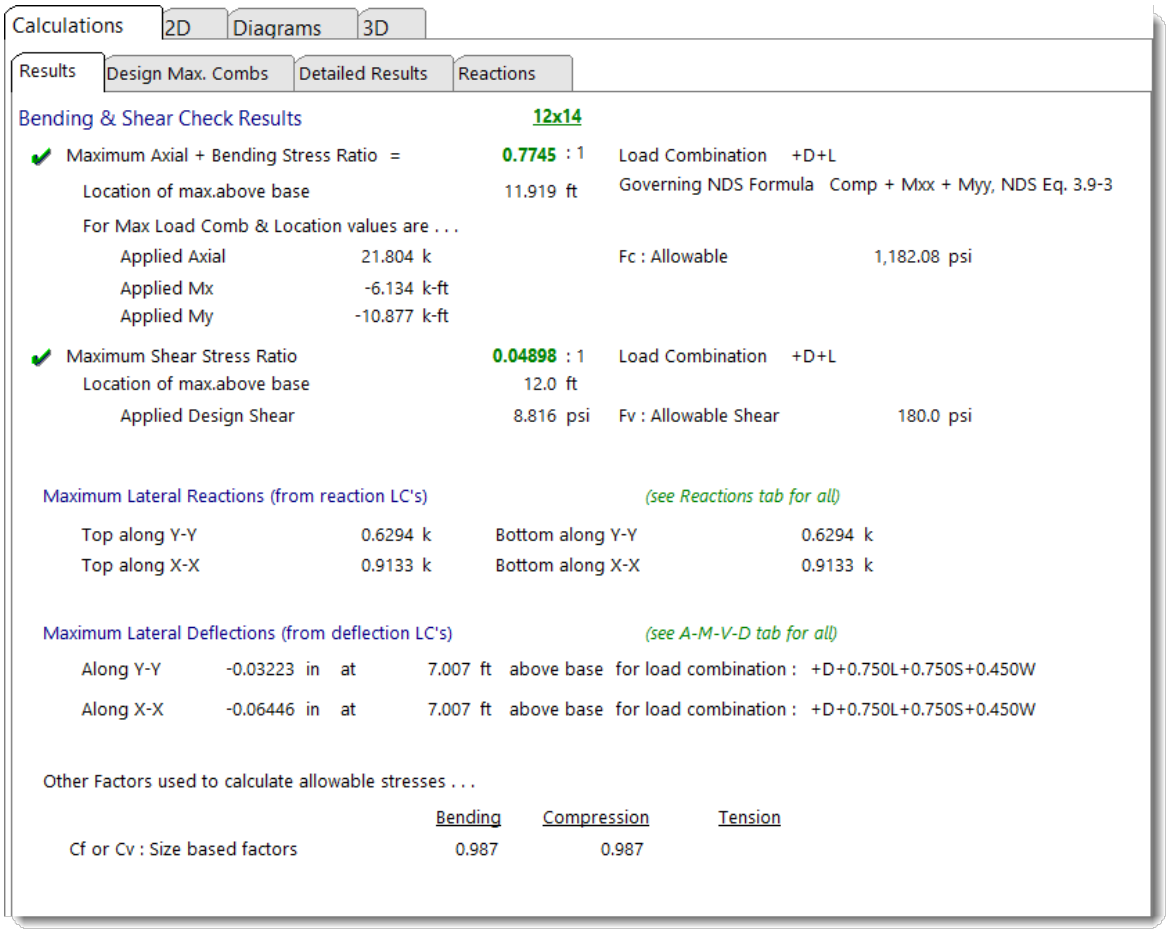
This tab provides a summary of the stress ratios, reactions and deflections for the column.

Max Axial + Bending Stress Ratio is the governing load combination for the column. Listed is the governing load load combination, the NDS formulas that is used and the location of the maximum stress ratio above the base of the column. Please note maximum stress ratio is what is being reported because it governs the design.....not necessarily the highest axial or bending stress.

Max Shear Stress Ratio will probably never govern for a column design but is presented here with the governing load combination, location and allowable/actual stress values.

Lateral Load Reactions and **Deflections** are the result of applied lateral loads.

© 2026 ENERCALC, LLC



Calculations | 2D | Diagrams | 3D

Results | Design Max. Combs | Detailed Results | Reactions

Bending & Shear Check Results 12x14

✔ Maximum Axial + Bending Stress Ratio = 0.7745 : 1 — Load Combination +D+L

Location of max.above base 11.919 ft — Governing NDS Formula Comp + Mxx + Myy, NDS Eq. 3.9-3

For Max Load Comb & Location values are . . .

Applied Axial 21.804 k — Fc : Allowable 1,182.08 psi

Applied Mx -6.134 k-ft

Applied My -10.877 k-ft

✔ Maximum Shear Stress Ratio 0.04898 : 1 — Load Combination +D+L

Location of max.above base 12.0 ft

Applied Design Shear 8.816 psi — Fv : Allowable Shear 180.0 psi

Maximum Lateral Reactions (from reaction LC's) *(see Reactions tab for all)*

Top along Y-Y 0.6294 k — Bottom along Y-Y 0.6294 k

Top along X-X 0.9133 k — Bottom along X-X 0.9133 k

Maximum Lateral Deflections (from deflection LC's) *(see A-M-V-D tab for all)*

Along Y-Y -0.03223 in at 7.007 ft above base for load combination : +D+0.750L+0.750S+0.450W

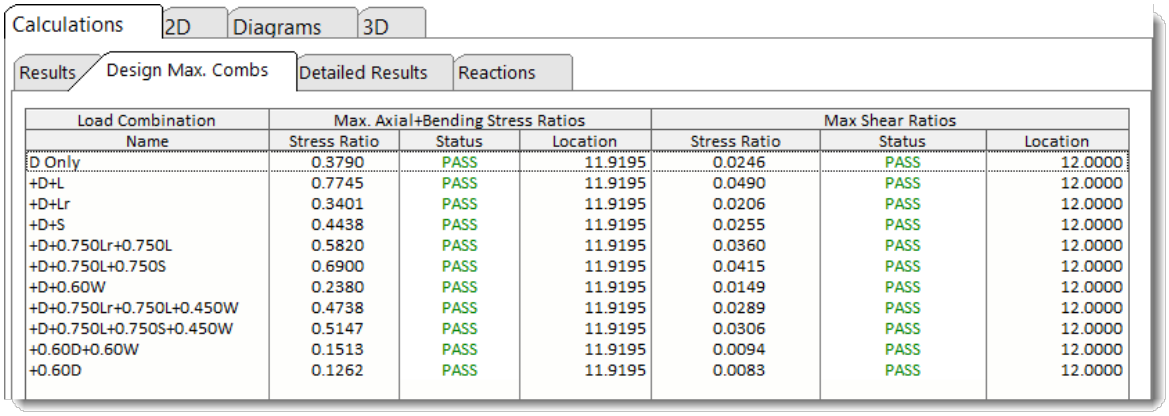
Along X-X -0.06446 in at 7.007 ft above base for load combination : +D+0.750L+0.750S+0.450W

Other Factors used to calculate allowable stresses . . .

	Bending	Compression	Tension
Cf or Cv : Size based factors	0.987	0.987	

Design Maximum Combinations

This tab lists the resulting maximum stress ratios for each load combination. This list is created by examining the detailed list (on the next tab) and determining the governing stress ratios for each load combination.



Calculations | 2D | Diagrams | 3D

Results | Design Max. Combs | Detailed Results | Reactions

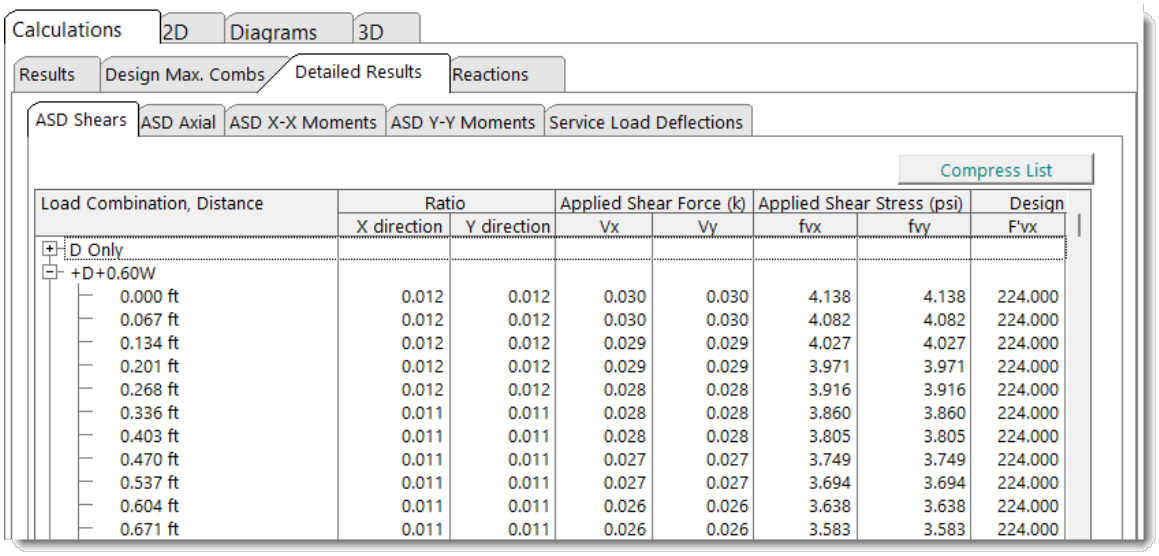
Load Combination	Max. Axial+Bending Stress Ratios			Max Shear Ratios		
Name	Stress Ratio	Status	Location	Stress Ratio	Status	Location
D Only	0.3790	PASS	11.9195	0.0246	PASS	12.0000
+D+L	0.7745	PASS	11.9195	0.0490	PASS	12.0000
+D+Lr	0.3401	PASS	11.9195	0.0206	PASS	12.0000
+D+S	0.4438	PASS	11.9195	0.0255	PASS	12.0000
+D+0.750Lr+0.750L	0.5820	PASS	11.9195	0.0360	PASS	12.0000
+D+0.750L+0.750S	0.6900	PASS	11.9195	0.0415	PASS	12.0000
+D+0.60W	0.2380	PASS	11.9195	0.0149	PASS	12.0000
+D+0.750Lr+0.750L+0.450W	0.4738	PASS	11.9195	0.0289	PASS	12.0000
+D+0.750L+0.750S+0.450W	0.5147	PASS	11.9195	0.0306	PASS	12.0000
+0.60D+0.60W	0.1513	PASS	11.9195	0.0094	PASS	12.0000
+0.60D	0.1262	PASS	11.9195	0.0083	PASS	12.0000

© 2026 ENERCALC, LLC

Detailed Results - Shears

This tab lists the detailed shear results at small increments along the height of the column for each load combination.

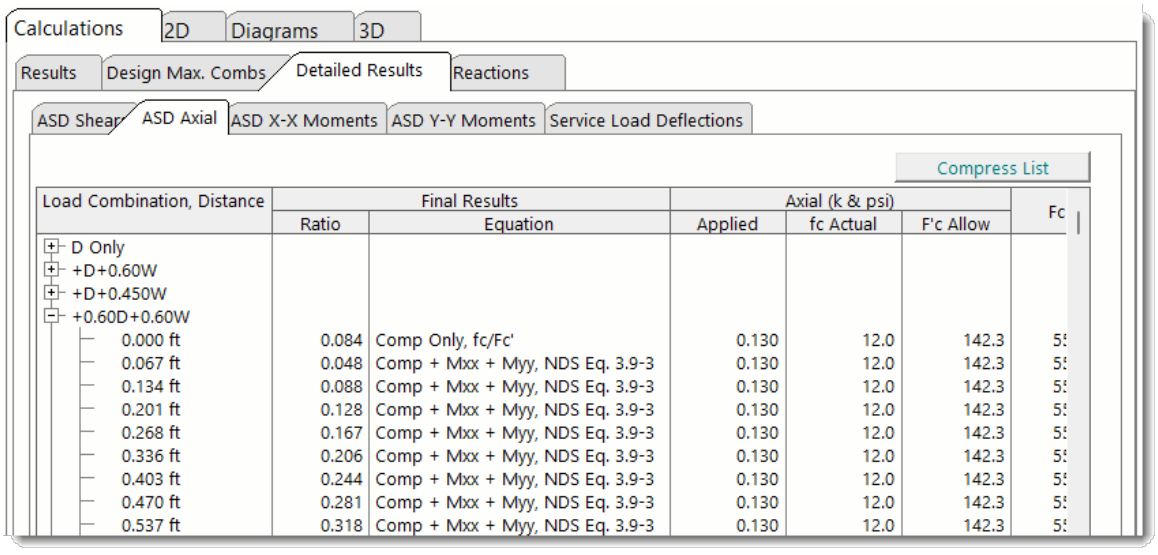
Note! This list scrolls to the right to display more information.



Load Combination, Distance	Ratio		Applied Shear Force (k)		Applied Shear Stress (psi)		Design
	X direction	Y direction	Vx	Vy	fvx	fvy	F'vx
D Only							
+D+0.60W							
0.000 ft	0.012	0.012	0.030	0.030	4.138	4.138	224.000
0.067 ft	0.012	0.012	0.030	0.030	4.082	4.082	224.000
0.134 ft	0.012	0.012	0.029	0.029	4.027	4.027	224.000
0.201 ft	0.012	0.012	0.029	0.029	3.971	3.971	224.000
0.268 ft	0.012	0.012	0.028	0.028	3.916	3.916	224.000
0.336 ft	0.011	0.011	0.028	0.028	3.860	3.860	224.000
0.403 ft	0.011	0.011	0.028	0.028	3.805	3.805	224.000
0.470 ft	0.011	0.011	0.027	0.027	3.749	3.749	224.000
0.537 ft	0.011	0.011	0.027	0.027	3.694	3.694	224.000
0.604 ft	0.011	0.011	0.026	0.026	3.638	3.638	224.000
0.671 ft	0.011	0.011	0.026	0.026	3.583	3.583	224.000

Detailed Results - Axial

This tab lists the detailed axial results at small increments along the height of the column for each load combination.

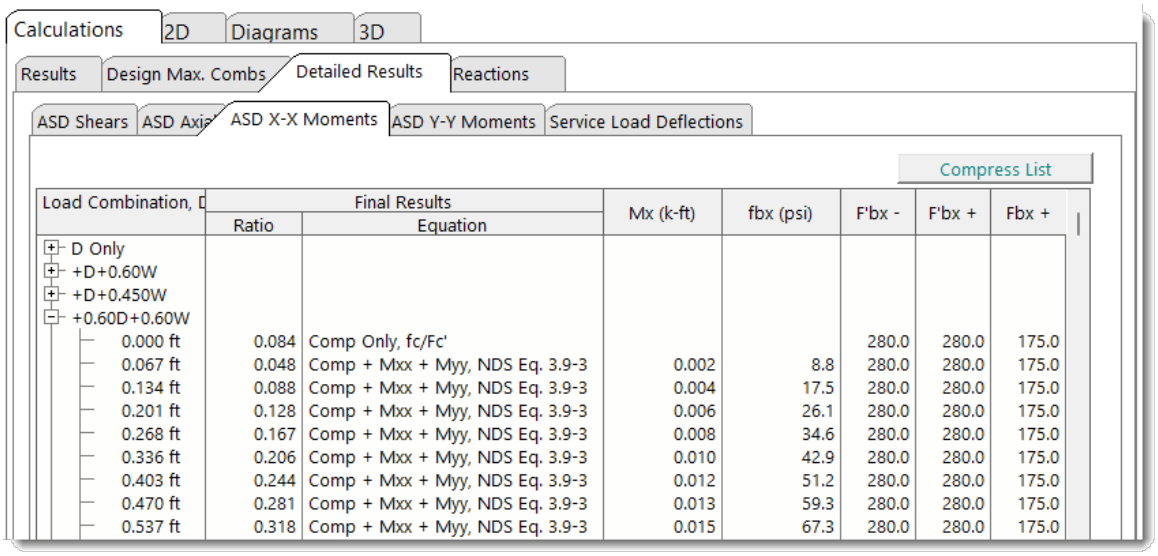


Load Combination, Distance	Final Results		Axial (k & psi)			Fc
	Ratio	Equation	Applied	fc Actual	F'c Allow	
D Only						
+D+0.60W						
+D+0.450W						
+0.60D+0.60W						
0.000 ft	0.084	Comp Only, fc/Fc'	0.130	12.0	142.3	5
0.067 ft	0.048	Comp + Mxx + Myy, NDS Eq. 3.9-3	0.130	12.0	142.3	5
0.134 ft	0.088	Comp + Mxx + Myy, NDS Eq. 3.9-3	0.130	12.0	142.3	5
0.201 ft	0.128	Comp + Mxx + Myy, NDS Eq. 3.9-3	0.130	12.0	142.3	5
0.268 ft	0.167	Comp + Mxx + Myy, NDS Eq. 3.9-3	0.130	12.0	142.3	5
0.336 ft	0.206	Comp + Mxx + Myy, NDS Eq. 3.9-3	0.130	12.0	142.3	5
0.403 ft	0.244	Comp + Mxx + Myy, NDS Eq. 3.9-3	0.130	12.0	142.3	5
0.470 ft	0.281	Comp + Mxx + Myy, NDS Eq. 3.9-3	0.130	12.0	142.3	5
0.537 ft	0.318	Comp + Mxx + Myy, NDS Eq. 3.9-3	0.130	12.0	142.3	5

© 2026 ENERCALC, LLC

Detailed Results - X-X Moments

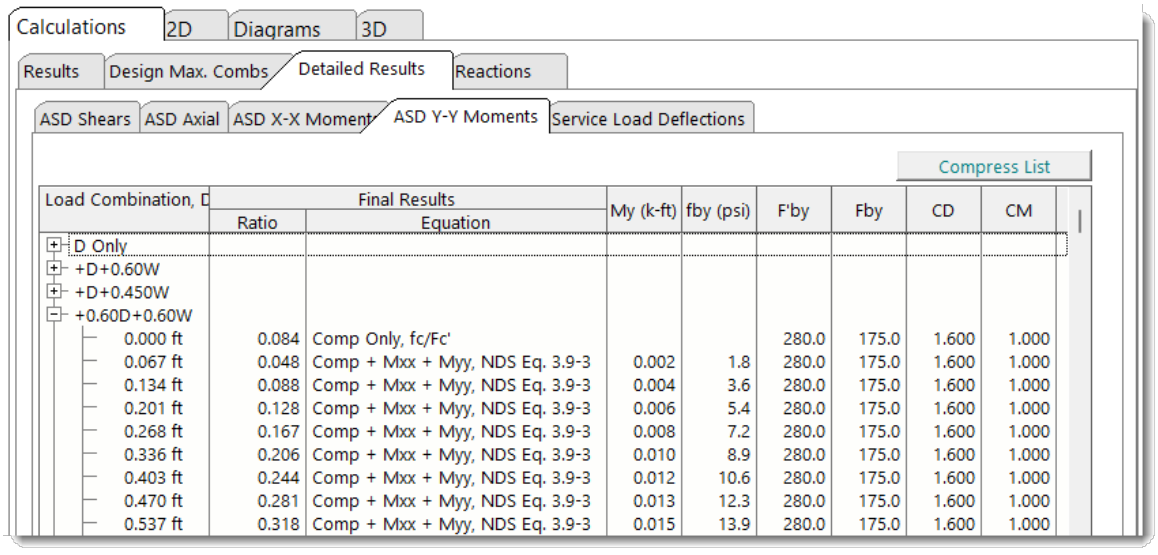
This tab lists the detailed X axis moment results at small increments along the height of the column for each load combination.



Load Combination, D	Final Results Ratio	Final Results Equation	Mx (k-ft)	fbx (psi)	F'bx -	F'bx +	Fbx +
D Only							
+D+0.60W							
+D+0.450W							
+0.60D+0.60W							
0.000 ft	0.084	Comp Only, fc/Fc'			280.0	280.0	175.0
0.067 ft	0.048	Comp + Mxx + Myy, NDS Eq. 3.9-3	0.002	8.8	280.0	280.0	175.0
0.134 ft	0.088	Comp + Mxx + Myy, NDS Eq. 3.9-3	0.004	17.5	280.0	280.0	175.0
0.201 ft	0.128	Comp + Mxx + Myy, NDS Eq. 3.9-3	0.006	26.1	280.0	280.0	175.0
0.268 ft	0.167	Comp + Mxx + Myy, NDS Eq. 3.9-3	0.008	34.6	280.0	280.0	175.0
0.336 ft	0.206	Comp + Mxx + Myy, NDS Eq. 3.9-3	0.010	42.9	280.0	280.0	175.0
0.403 ft	0.244	Comp + Mxx + Myy, NDS Eq. 3.9-3	0.012	51.2	280.0	280.0	175.0
0.470 ft	0.281	Comp + Mxx + Myy, NDS Eq. 3.9-3	0.013	59.3	280.0	280.0	175.0
0.537 ft	0.318	Comp + Mxx + Myy, NDS Eq. 3.9-3	0.015	67.3	280.0	280.0	175.0

Detailed Results - Y-Y Moments

This tab lists the detailed Y axis moment results at small increments along the height of the column for each load combination.

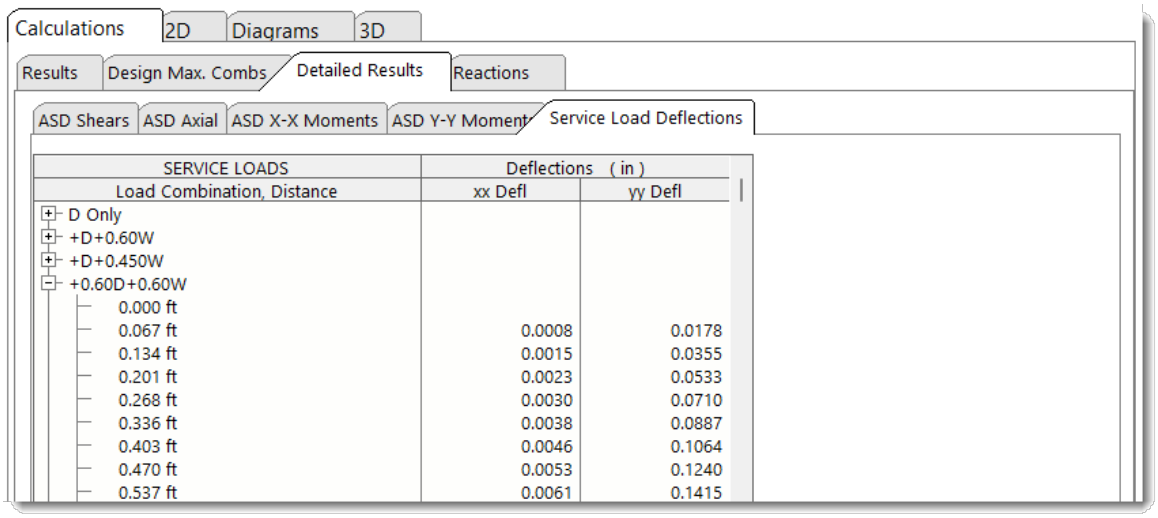


Load Combination, D	Final Results Ratio	Final Results Equation	My (k-ft)	fby (psi)	F'by	Fby	CD	CM
D Only								
+D+0.60W								
+D+0.450W								
+0.60D+0.60W								
0.000 ft	0.084	Comp Only, fc/Fc'			280.0	175.0	1.600	1.000
0.067 ft	0.048	Comp + Mxx + Myy, NDS Eq. 3.9-3	0.002	1.8	280.0	175.0	1.600	1.000
0.134 ft	0.088	Comp + Mxx + Myy, NDS Eq. 3.9-3	0.004	3.6	280.0	175.0	1.600	1.000
0.201 ft	0.128	Comp + Mxx + Myy, NDS Eq. 3.9-3	0.006	5.4	280.0	175.0	1.600	1.000
0.268 ft	0.167	Comp + Mxx + Myy, NDS Eq. 3.9-3	0.008	7.2	280.0	175.0	1.600	1.000
0.336 ft	0.206	Comp + Mxx + Myy, NDS Eq. 3.9-3	0.010	8.9	280.0	175.0	1.600	1.000
0.403 ft	0.244	Comp + Mxx + Myy, NDS Eq. 3.9-3	0.012	10.6	280.0	175.0	1.600	1.000
0.470 ft	0.281	Comp + Mxx + Myy, NDS Eq. 3.9-3	0.013	12.3	280.0	175.0	1.600	1.000
0.537 ft	0.318	Comp + Mxx + Myy, NDS Eq. 3.9-3	0.015	13.9	280.0	175.0	1.600	1.000

© 2026 ENERCALC, LLC

Detailed Results - Service Load Deflections

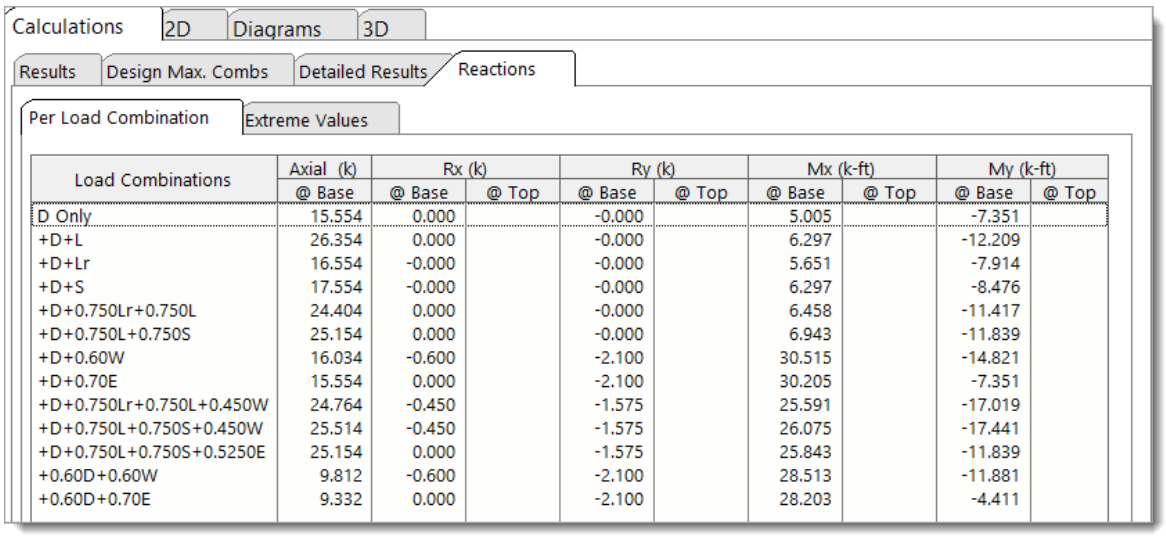
This tab reports the deflections at incremental locations along the height of the column, for each service load condition (i.e. for individual load cases and for a set of built-in service load combinations), along each axis.



SERVICE LOADS Load Combination, Distance	Deflections (in) xx Defl	 yy Defl
D Only		
+D+0.60W		
+D+0.450W		
+0.60D+0.60W		
0.000 ft		
0.067 ft	0.0008	0.0178
0.134 ft	0.0015	0.0355
0.201 ft	0.0023	0.0533
0.268 ft	0.0030	0.0710
0.336 ft	0.0038	0.0887
0.403 ft	0.0046	0.1064
0.470 ft	0.0053	0.1240
0.537 ft	0.0061	0.1415

Reactions - Per Load Combination

This tab displays the support reactions calculated for each load combination defined in the Load Combs > Reaction Combinations tab.



Load Combinations	Axial (k)	Rx (k)		Ry (k)		Mx (k-ft)		My (k-ft)	
	@ Base	@ Base	@ Top	@ Base	@ Top	@ Base	@ Top	@ Base	@ Top
D Only	15.554	0.000		-0.000		5.005		-7.351	
+D+L	26.354	0.000		-0.000		6.297		-12.209	
+D+Lr	16.554	-0.000		-0.000		5.651		-7.914	
+D+S	17.554	-0.000		-0.000		6.297		-8.476	
+D+0.750Lr+0.750L	24.404	0.000		-0.000		6.458		-11.417	
+D+0.750L+0.750S	25.154	0.000		-0.000		6.943		-11.839	
+D+0.60W	16.034	-0.600		-2.100		30.515		-14.821	
+D+0.70E	15.554	0.000		-2.100		30.205		-7.351	
+D+0.750Lr+0.750L+0.450W	24.764	-0.450		-1.575		25.591		-17.019	
+D+0.750L+0.750S+0.450W	25.514	-0.450		-1.575		26.075		-17.441	
+D+0.750L+0.750S+0.5250E	25.154	0.000		-1.575		25.843		-11.839	
+0.60D+0.60W	9.812	-0.600		-2.100		28.513		-11.881	
+0.60D+0.70E	9.332	0.000		-2.100		28.203		-4.411	

Reactions are reported at both the base and top of the column (where applicable), and include the following:

Axial @ Base (k): Axial reaction at the base of the column.

- Positive values indicate compression (downward force).

© 2026 ENERCALC, LLC

- Negative values indicate tension (uplift).

Rx @ Base/Top (k): Horizontal reaction in the X-direction at the base and top.
- Positive values act in the global +X direction (to the right when looking down at the top of the column).
- Negative values act in the global -X direction (to the left when looking down at the top of the column).

Ry @ Base/Top (k): Horizontal reaction in the Y-direction at the base and top.
- Positive values act in the global +Y direction (upward on screen when looking down at the top of the column).
- Negative values act in the global -Y direction (downward on screen when looking down at the top of the column).

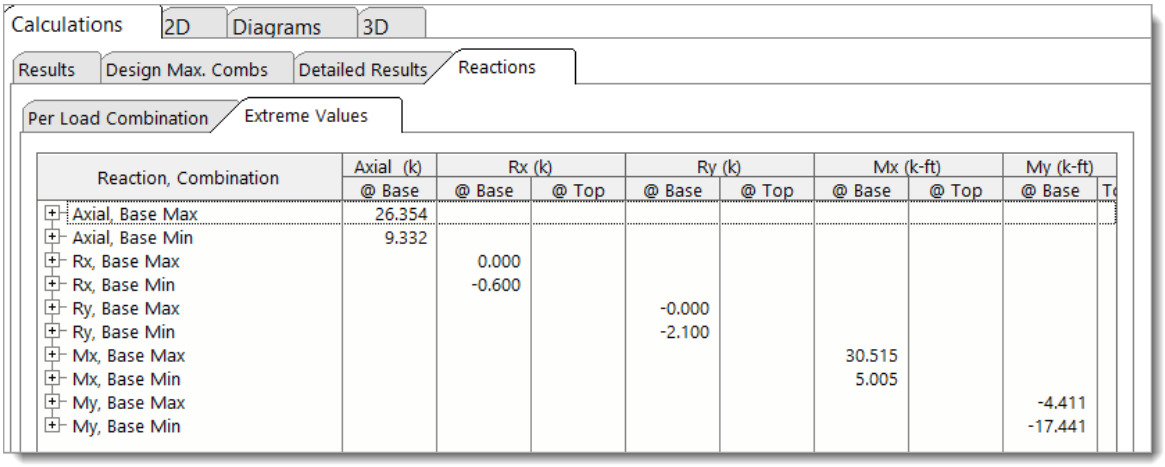
Mx @ Base/Top (k-ft): Moment reaction about the X-axis at the base and top.
- Positive values follow the right-hand rule. Point your right thumb in the direction of the +X-axis (to the right when looking down at the top of the column), and your curled fingers indicate the direction of the moment.
- Negative values follow the right-hand rule. Point your right thumb in the direction of the -X-axis (to the left when looking down at the top of the column), and your curled fingers indicate the direction of the moment.

My @ Base/Top (k-ft): Moment reaction about the Y-axis at the base and top.
- Positive values follow the right-hand rule. Point your right thumb in the direction of the +Y-axis (upward on screen when looking down at the top of the column), and your curled fingers indicate the direction of the moment.
- Negative values follow the right-hand rule. Point your right thumb in the direction of the -Y-axis (downward on screen when looking down at the top of the column), and your curled fingers indicate the direction of the moment.

© 2026 ENERCALC, LLC

Reactions - Extreme Values

This tab summarizes the maximum and minimum reaction forces and moments that occur at the top and bottom of the column across all defined load combinations.



Reaction, Combination	Axial (k)	Rx (k)		Ry (k)		Mx (k-ft)		My (k-ft)	
	@ Base	@ Base	@ Top	@ Base	@ Top	@ Base	@ Top	@ Base	To
Axial, Base Max	26.354								
Axial, Base Min	9.332								
Rx, Base Max		0.000							
Rx, Base Min		-0.600							
Ry, Base Max				-0.000					
Ry, Base Min				-2.100					
Mx, Base Max						30.515			
Mx, Base Min						5.005			
My, Base Max								-4.411	
My, Base Min								-17.441	

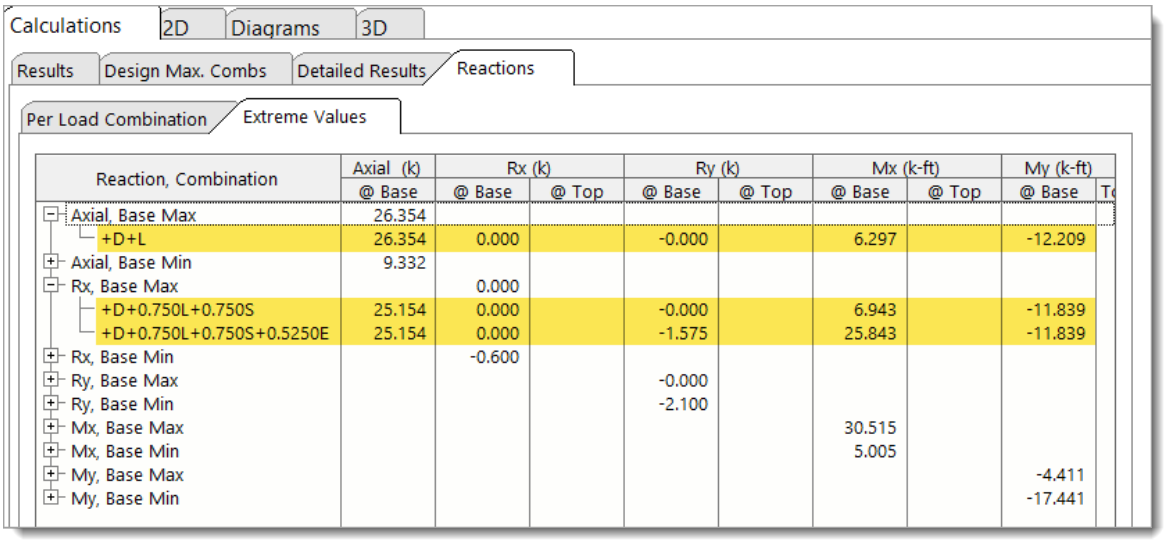
The following values are reported (where applicable):

- **Axial, Base Max** and **Axial, Base Min:** The max and min vertical axial reactions at the base of the column.
- **Rx, Base Max** and **Rx, Base Min:** The max and min horizontal reactions in the X-direction at the base and top.
- **Ry, Base Max** and **Ry, Base Min:** The max and min horizontal reactions in the Y-direction at the base and top.
- **Mx, Base Max** and **Mx, Base Min:** The max and min moment reaction about the X-axis at the base and top.
- **My, Base Max** and **My, Base Min:** The max and min moment reaction about the Y-axis at the base and top.

Notes:

- "Max" values represent the "most positive" results (i.e., closest to positive infinity), while "Min" values represent the "most negative" results (i.e., closest to negative infinity).
- Reaction components with no results are omitted from the table. For example, a cantilever column will not display any Top extreme values, since no reactions exist there.
- Zero values are shown as blank in the table for clarity.

© 2026 ENERCALC, LLC



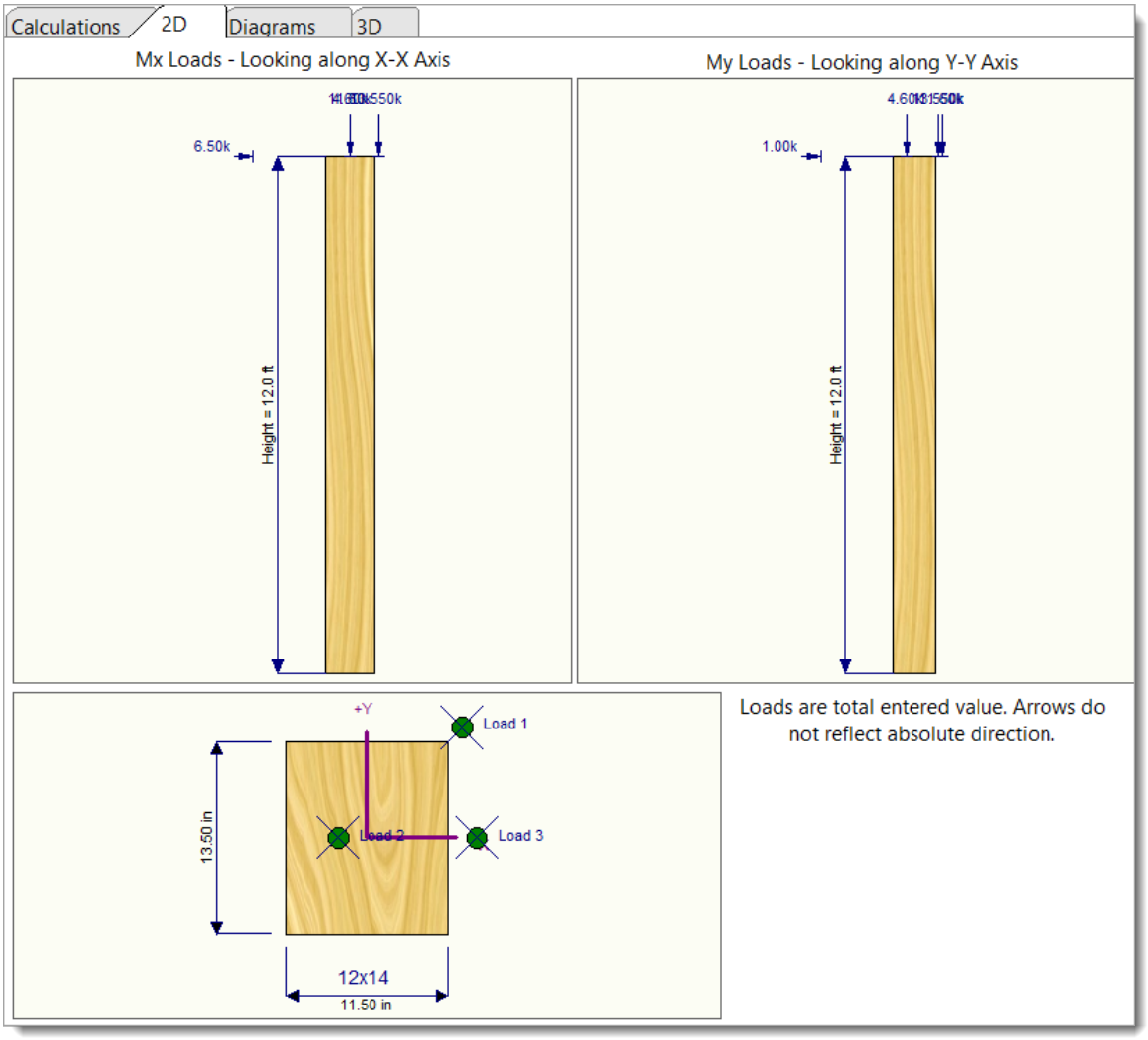
Reaction, Combination	Axial (k)	Rx (k)		Ry (k)		Mx (k-ft)		My (k-ft)	
	@ Base	@ Base	@ Top	@ Base	@ Top	@ Base	@ Top	@ Base	To
Axial, Base Max	26.354								
+D+L	26.354	0.000		-0.000		6.297		-12.209	
Axial, Base Min	9.332								
Rx, Base Max		0.000							
+D+0.750L+0.750S	25.154	0.000		-0.000		6.943		-11.839	
+D+0.750L+0.750S+0.5250E	25.154	0.000		-1.575		25.843		-11.839	
Rx, Base Min		-0.600							
Ry, Base Max				-0.000					
Ry, Base Min				-2.100					
Mx, Base Max						30.515			
Mx, Base Min						5.005			
My, Base Max								-4.411	
My, Base Min								-17.441	

The results are presented in a tree-structured table. Each row shows an extreme value and can be expanded to reveal:

- The load combination(s) that produced that extreme value.
- The associated reaction components for those combinations.
- If multiple load combinations result in the same extreme value, all contributing combinations will be listed.

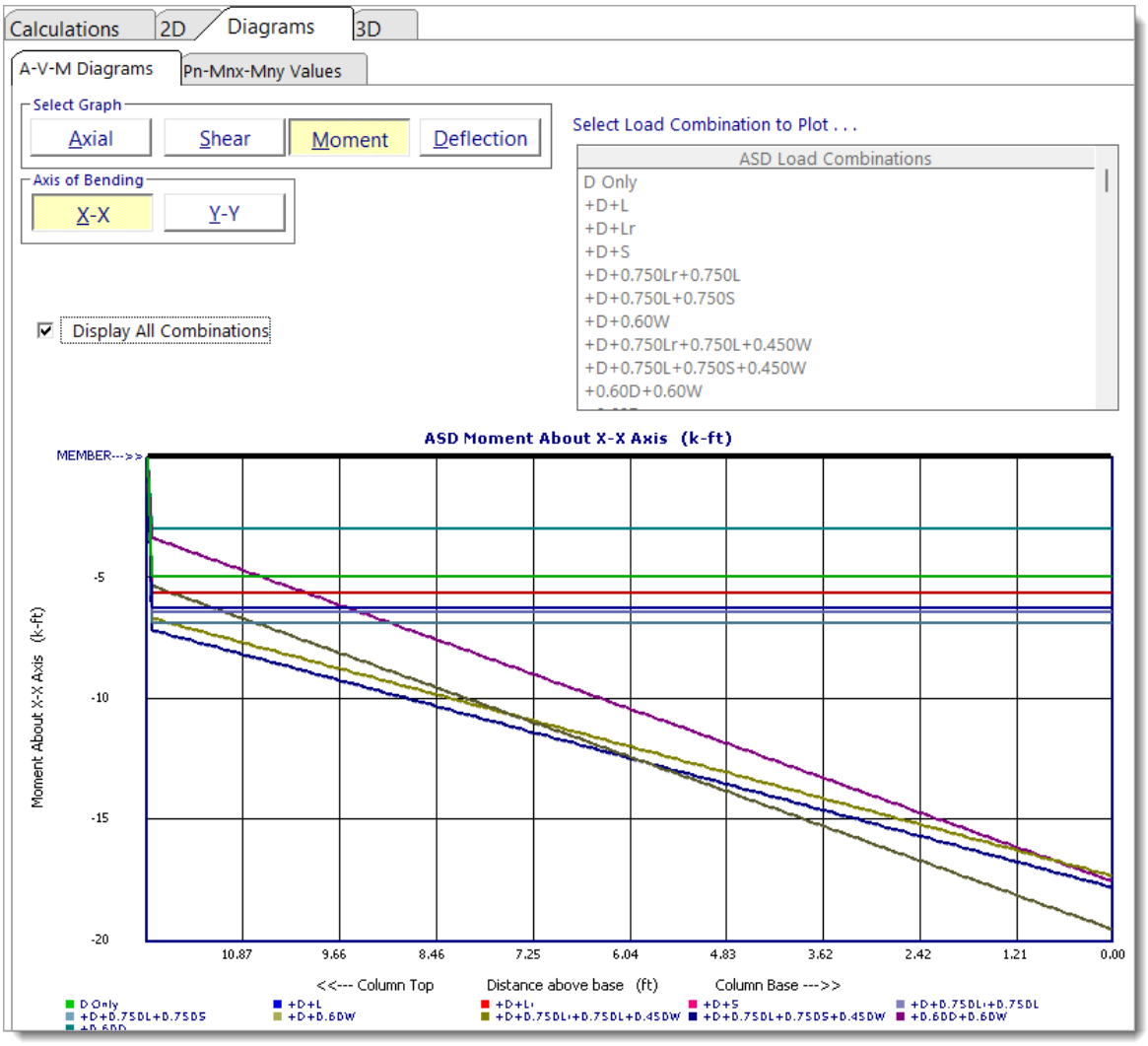
© 2026 ENERCALC, LLC

2D Sketch


Calculations
2D
Diagrams
3D
Mx Loads - Looking along X-X Axis
My Loads - Looking along Y-Y Axis
6.50k
1.00k
Height = 12.0 ft
Height = 12.0 ft
+Y
Load 1
Load 2
Load 3
13.50 in
12x14
11.50 in
Loads are total entered value. Arrows do not reflect absolute direction.

Diagrams - A-V-M-D Diagrams

This tab provides comprehensive charting capability to view graphs of axial load, shear, moment, and deflection along the length of the member. Note that the graphs are oriented such that the right end of the graph represents the column base, and the left end of the graph represents the column top. This was done to maximize the scale of the graph based on the screen area available.

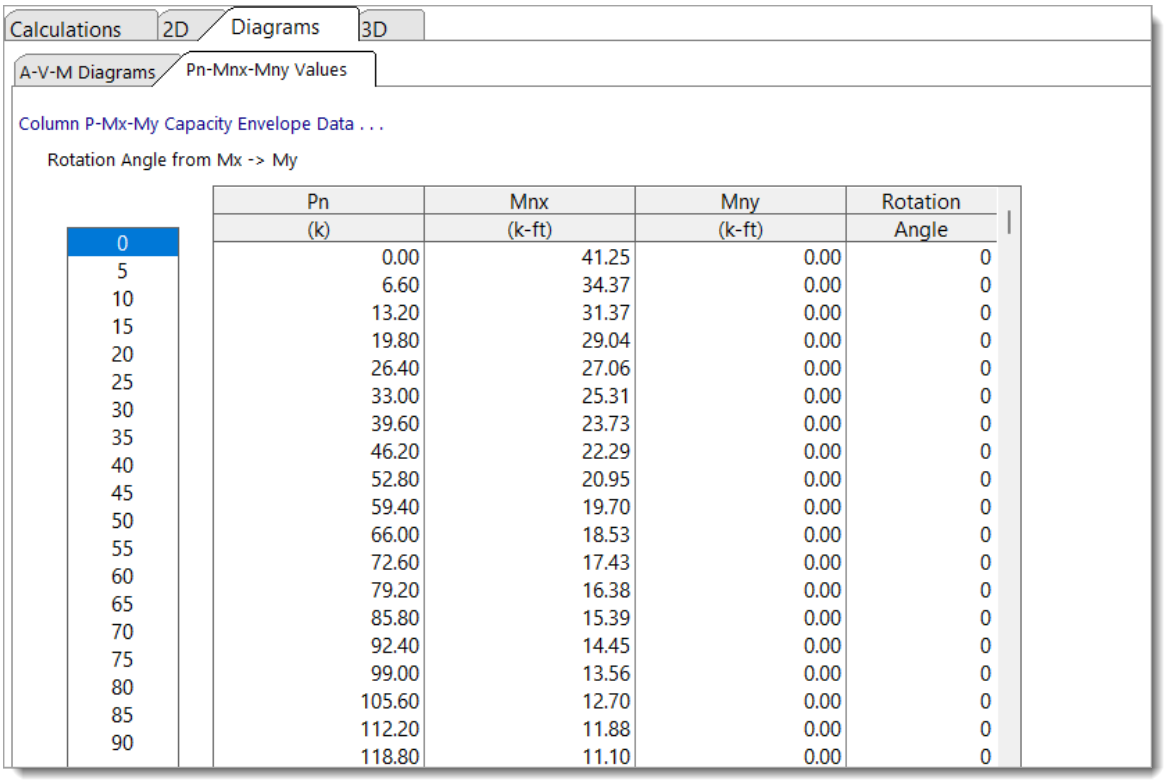


© 2026 ENERCALC, LLC

Diagrams - Pn-Pnx-Mny Values

This tab allows you to see the moment capacities about each axis given a certain allowable axial load.

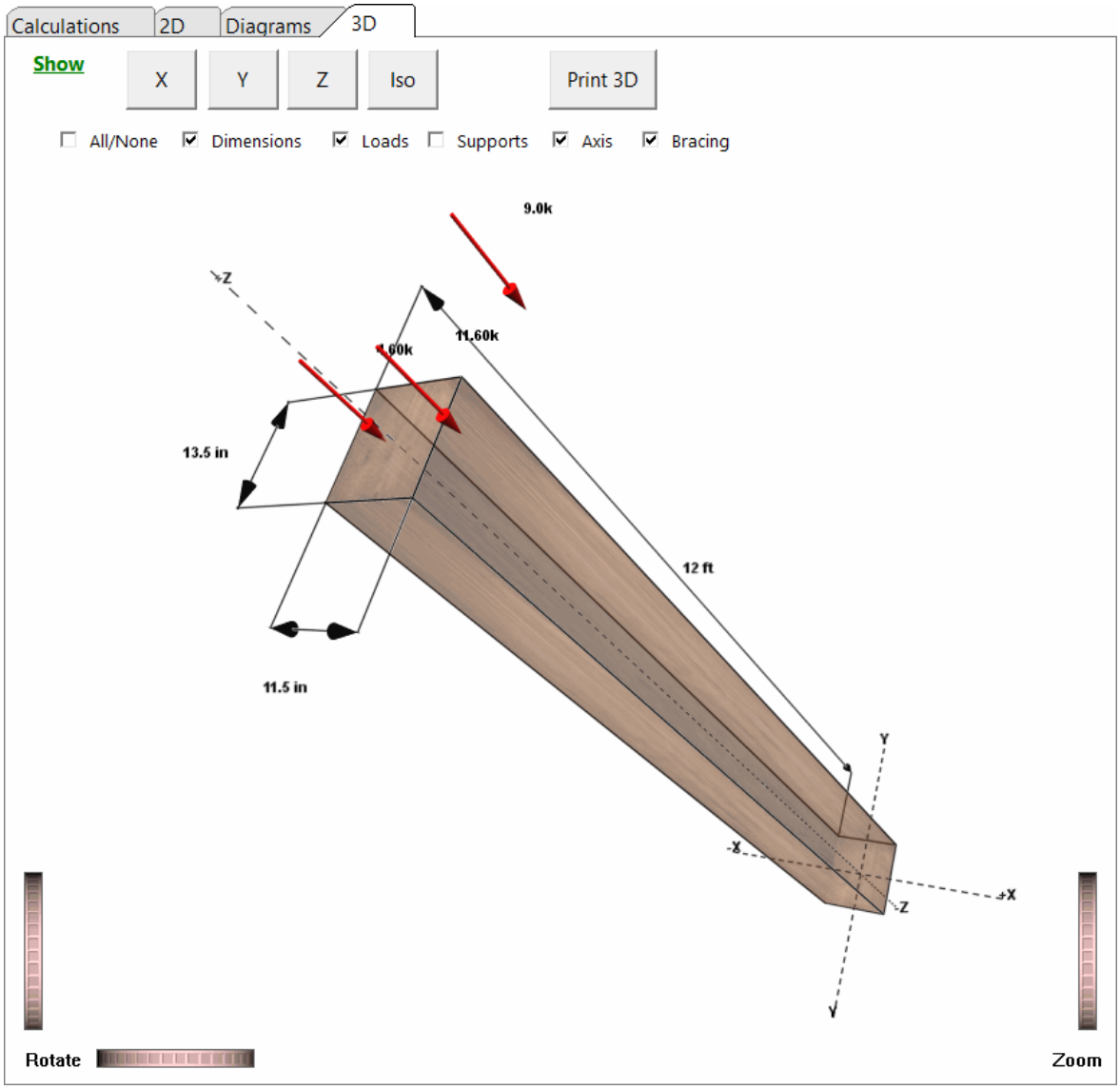
This is mostly for reference and can be considered a reverse application of the load capacity calculations.



	Pn (k)	Mnx (k-ft)	Mny (k-ft)	Rotation Angle
0	0.00	41.25	0.00	0
5	6.60	34.37	0.00	0
10	13.20	31.37	0.00	0
15	19.80	29.04	0.00	0
20	26.40	27.06	0.00	0
25	33.00	25.31	0.00	0
30	39.60	23.73	0.00	0
35	46.20	22.29	0.00	0
40	52.80	20.95	0.00	0
45	59.40	19.70	0.00	0
50	66.00	18.53	0.00	0
55	72.60	17.43	0.00	0
60	79.20	16.38	0.00	0
65	85.80	15.39	0.00	0
70	92.40	14.45	0.00	0
75	99.00	13.56	0.00	0
80	105.60	12.70	0.00	0
85	112.20	11.88	0.00	0
90	118.80	11.10	0.00	0

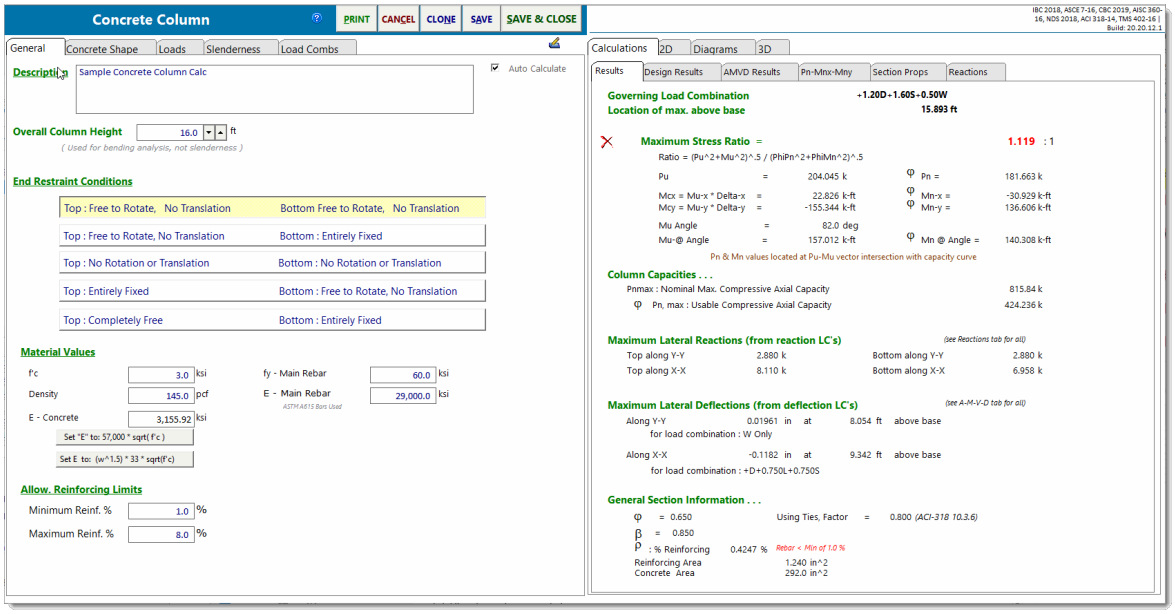
© 2026 ENERCALC, LLC

3D Rendering



13.5.4 Concrete Column

This module designs concrete columns that are subject to axial loads and lateral bending loads about both axes.



The module only uses strength design for concrete.

All calculations are according to the selected version of ACI 318.

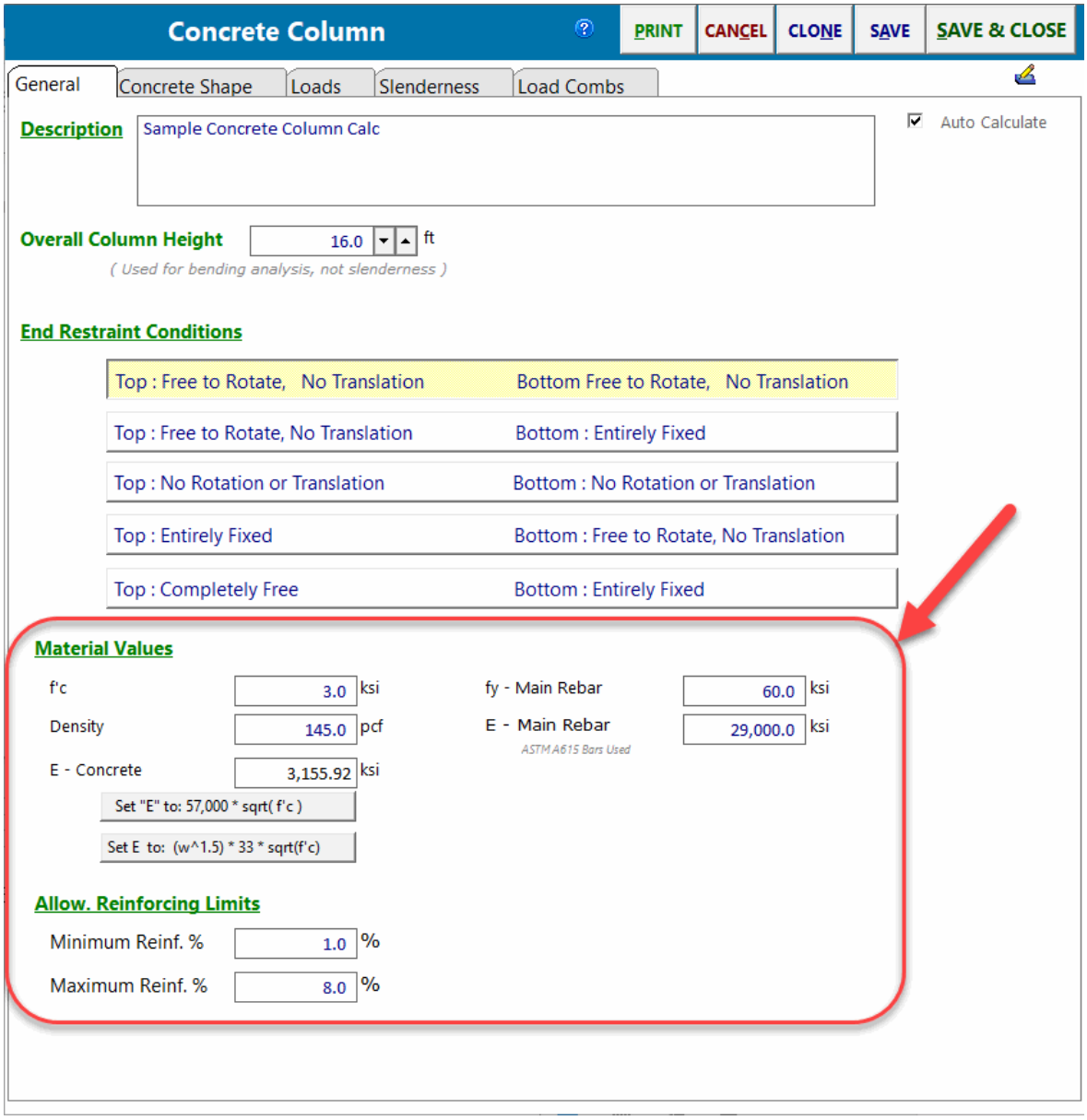
The screen capture below shows the full screen for concrete column design. See items below for descriptions of items that are specific to the concrete column design module.

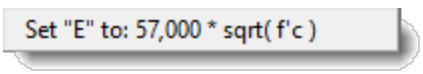
For general description of the module, end fixity, loads, and load combinations click here 810. For slenderness description click here 821.

© 2026 ENERCALC, LLC

General

The area indicated in the screen capture below is specific to the concrete column selection.



The two buttons  and immediately set the value for elastic modulus "E" to the values as described on the button.

© 2026 ENERCALC, LLC

Concrete Shape

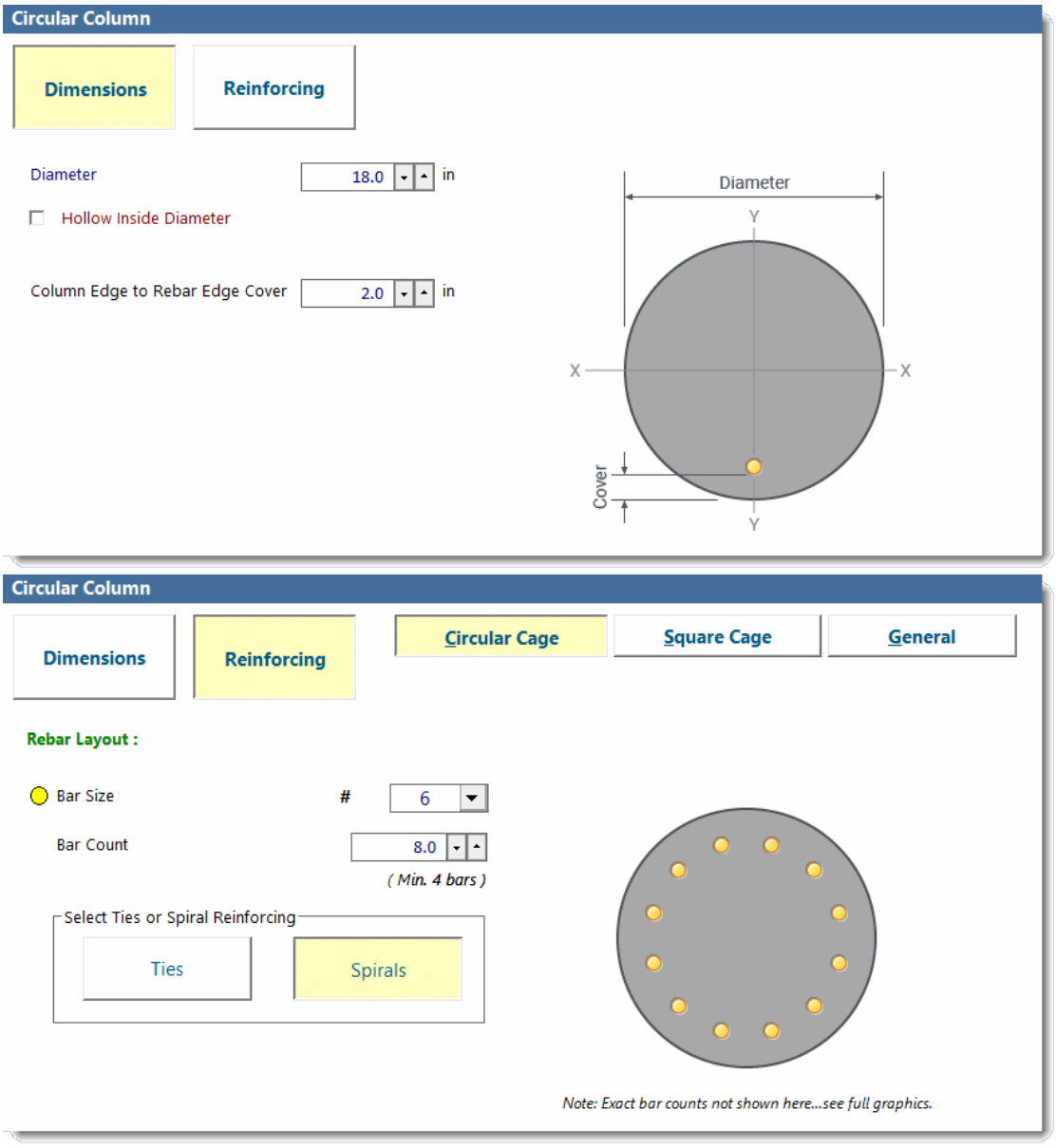
This tab is specific to the concrete column selection. It allows you to select from 12 different column shapes. Simply click the button surrounding the column shape icon and the screen below will change to allow specific data input for measurements and reinforcing layout.

Following the screen capture below we will show ALL the data input areas for ALL the column shapes with descriptions as needed.

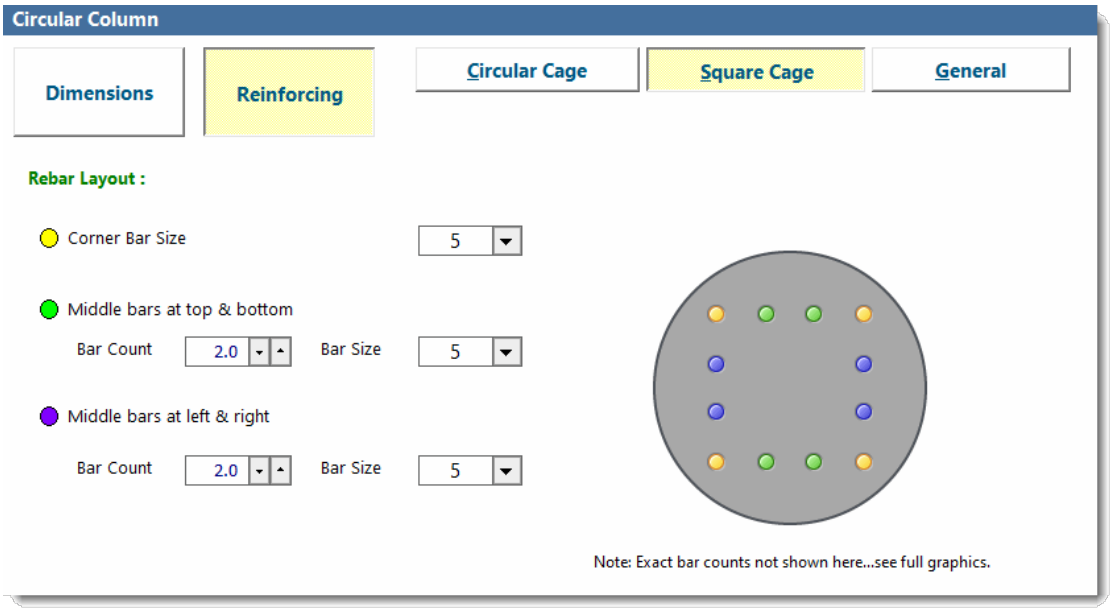
Note that this module does a very detailed biaxial analysis of the column cross section using exact numerical methods.

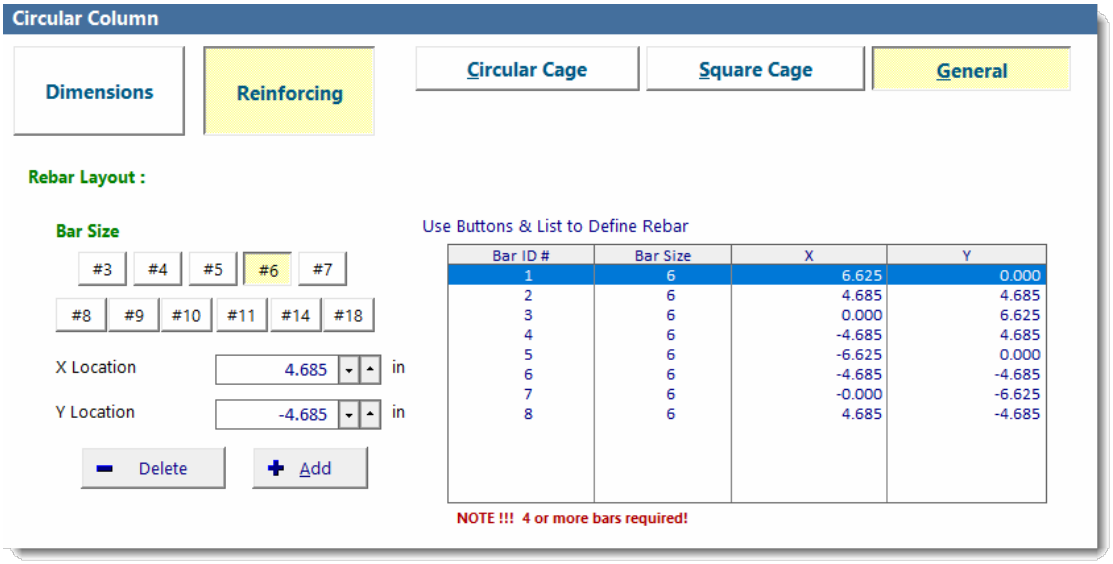
© 2026 ENERCALC, LLC

Circular Column



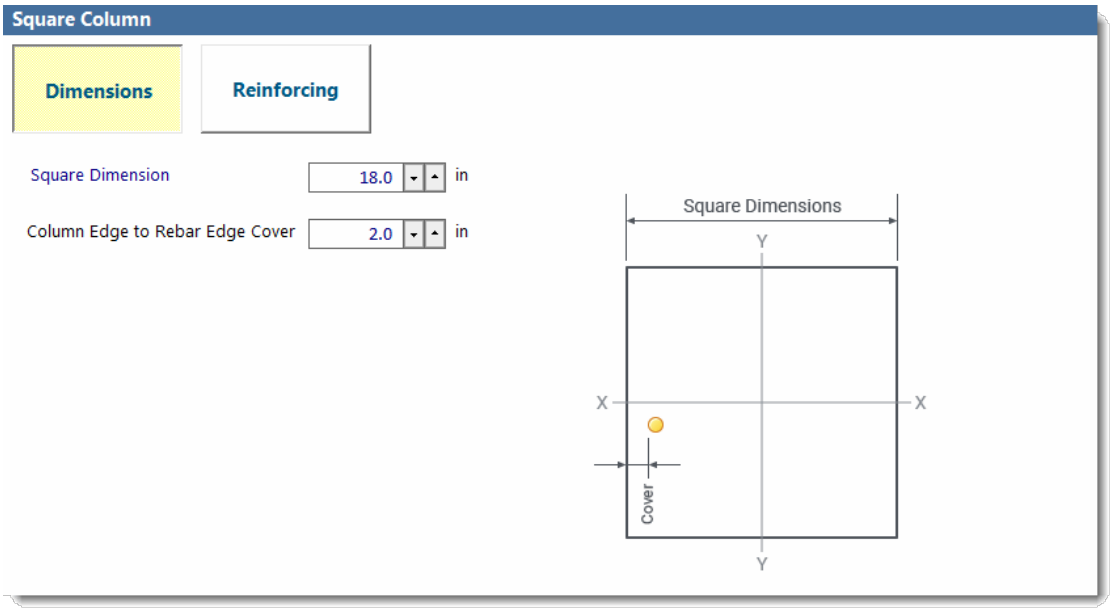
© 2026 ENERCALC, LLC

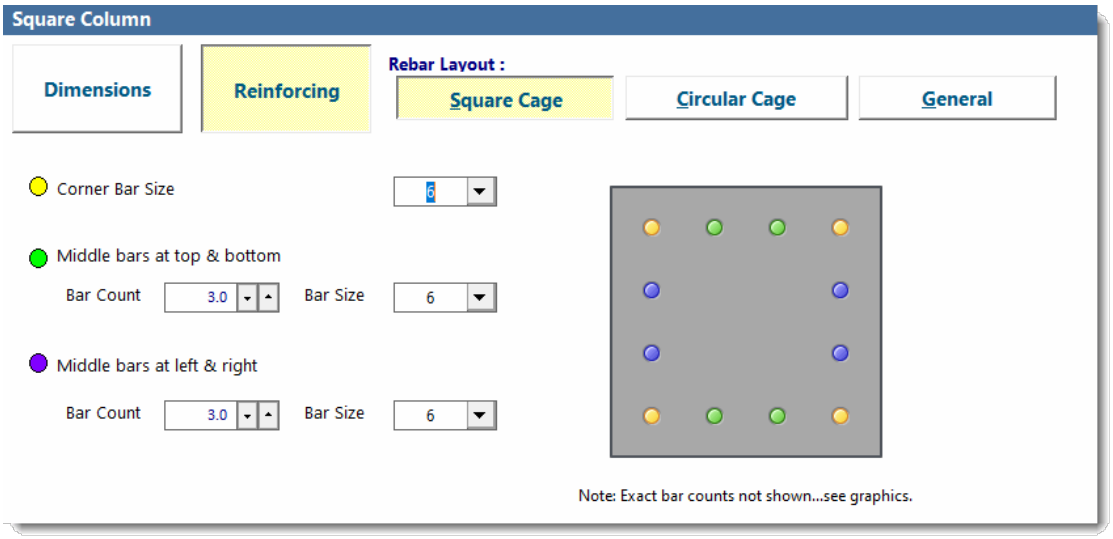




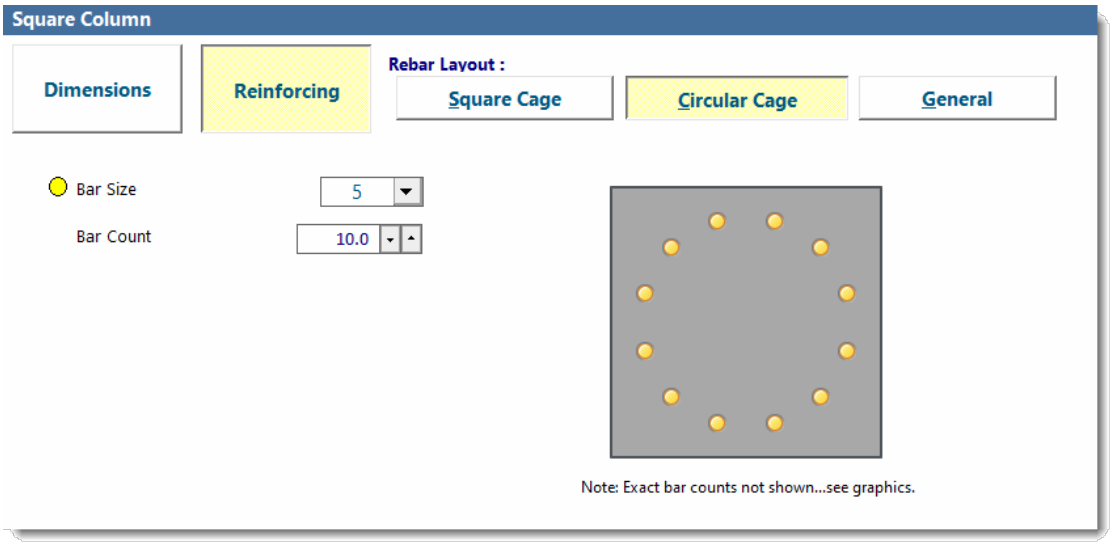
© 2026 ENERCALC, LLC

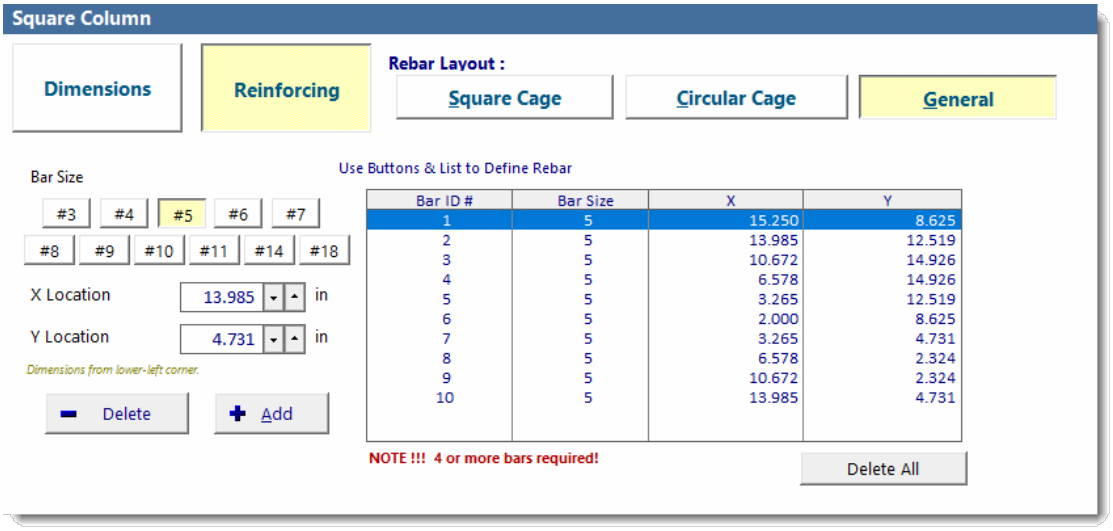
Square Column





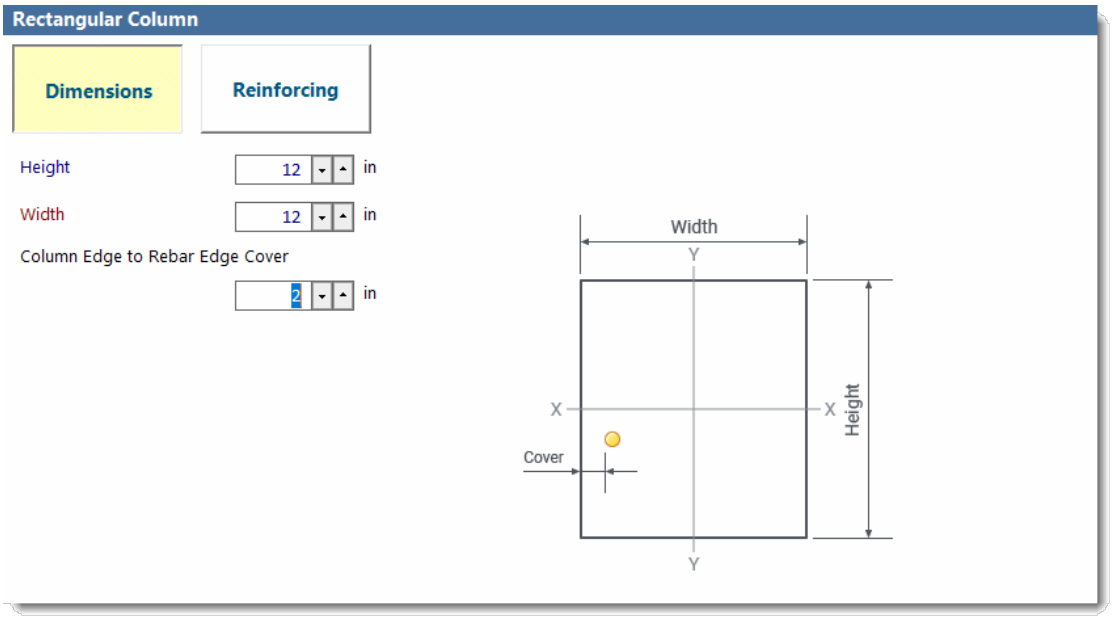
© 2026 ENERCALC, LLC

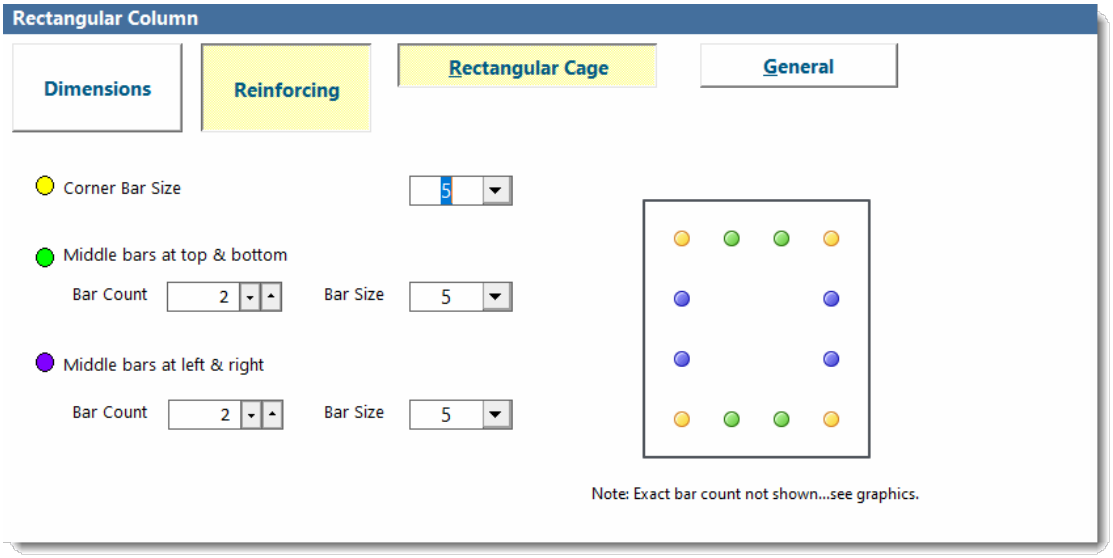




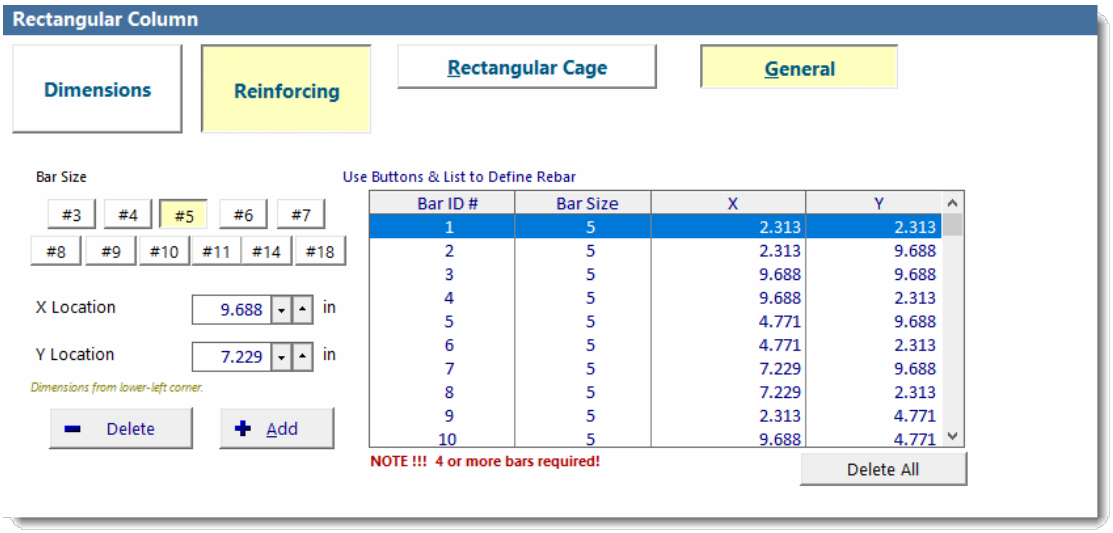
© 2026 ENERCALC, LLC

Rectangular Column



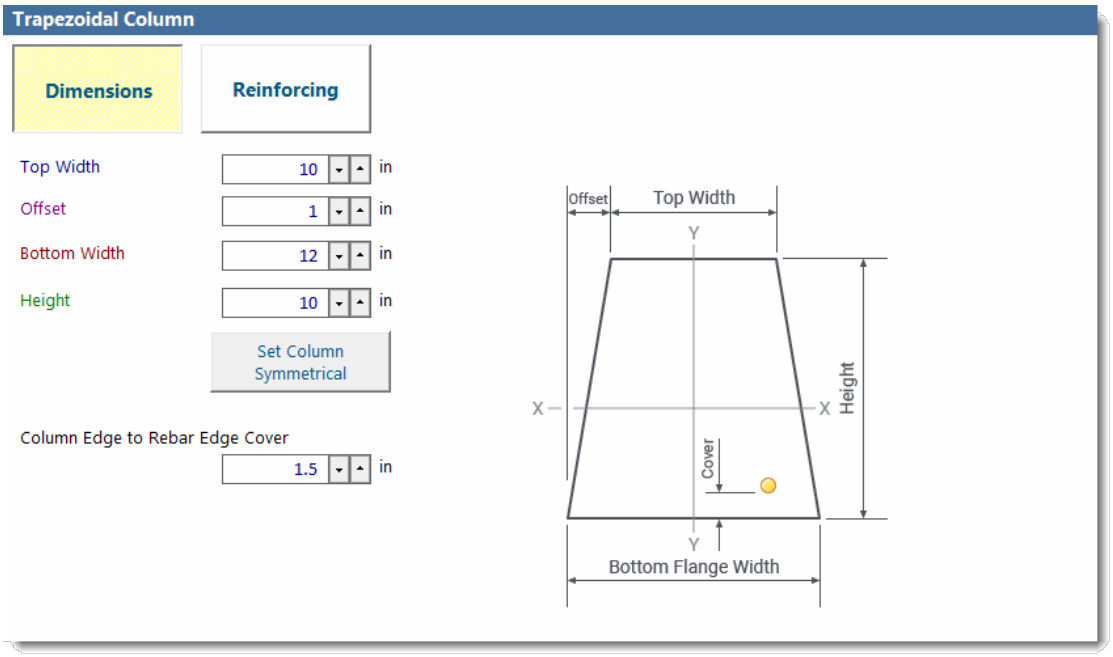


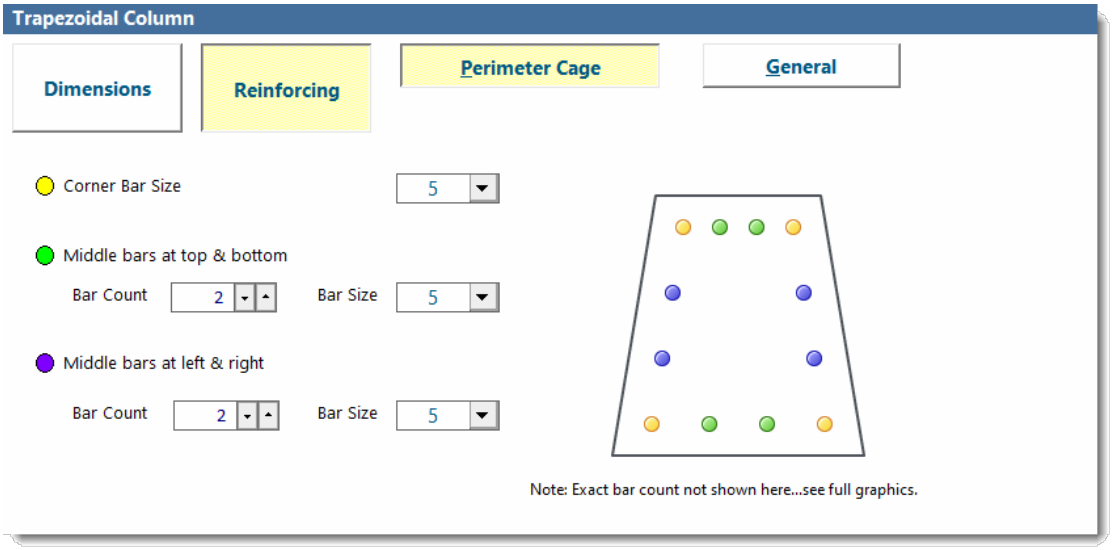
© 2026 ENERCALC, LLC



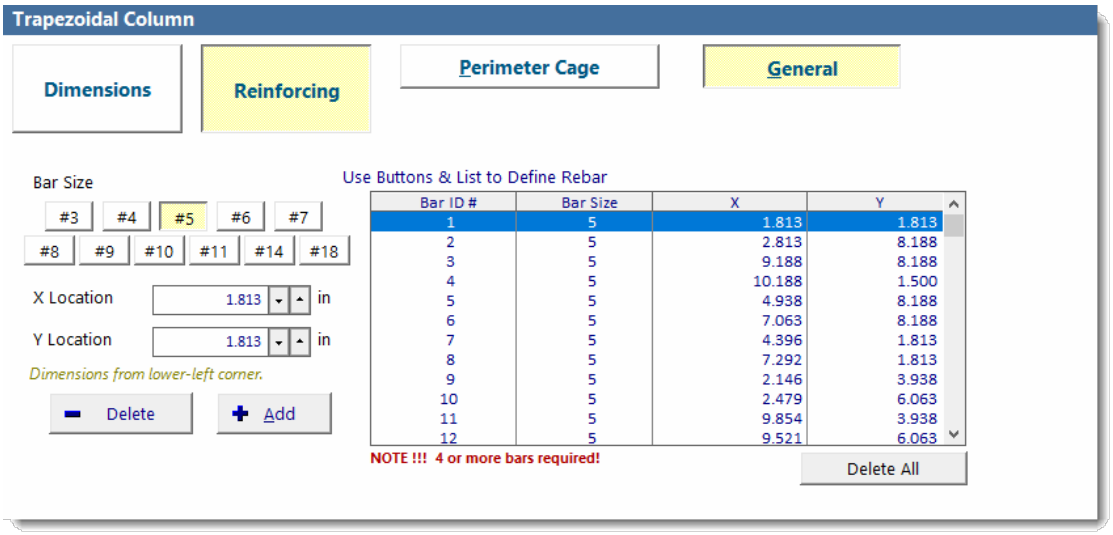
© 2026 ENERCALC, LLC

Trapezoidal Column



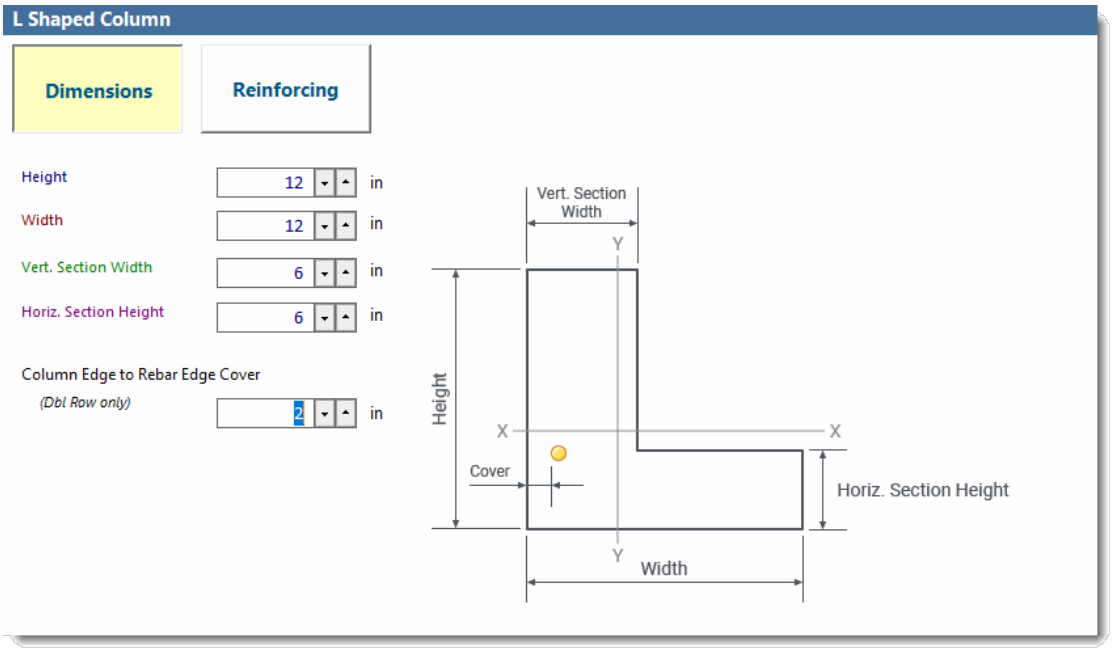


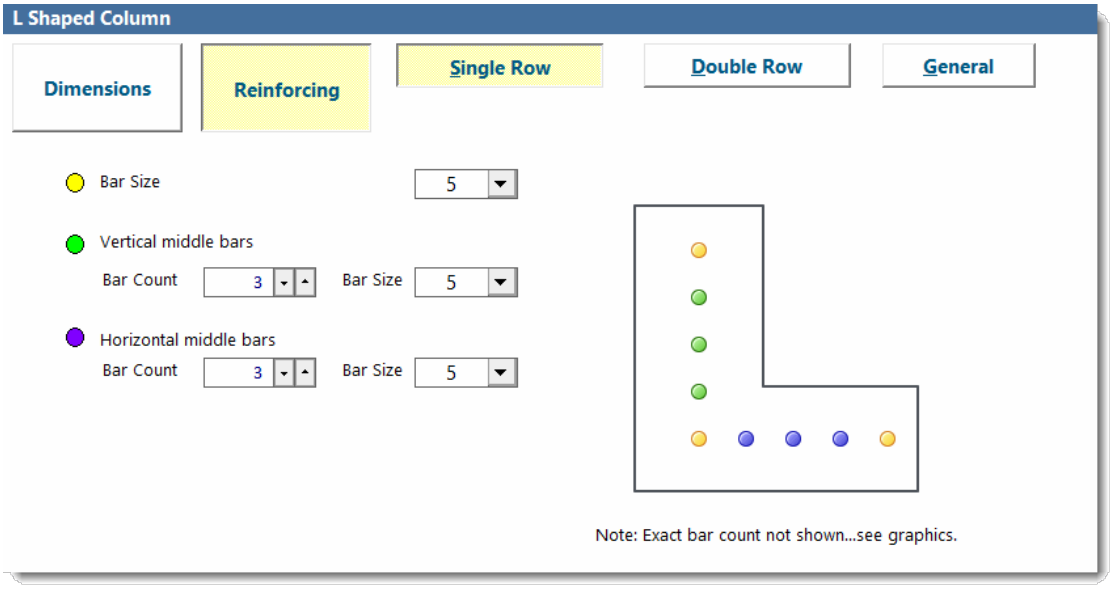
© 2026 ENERCALC, LLC



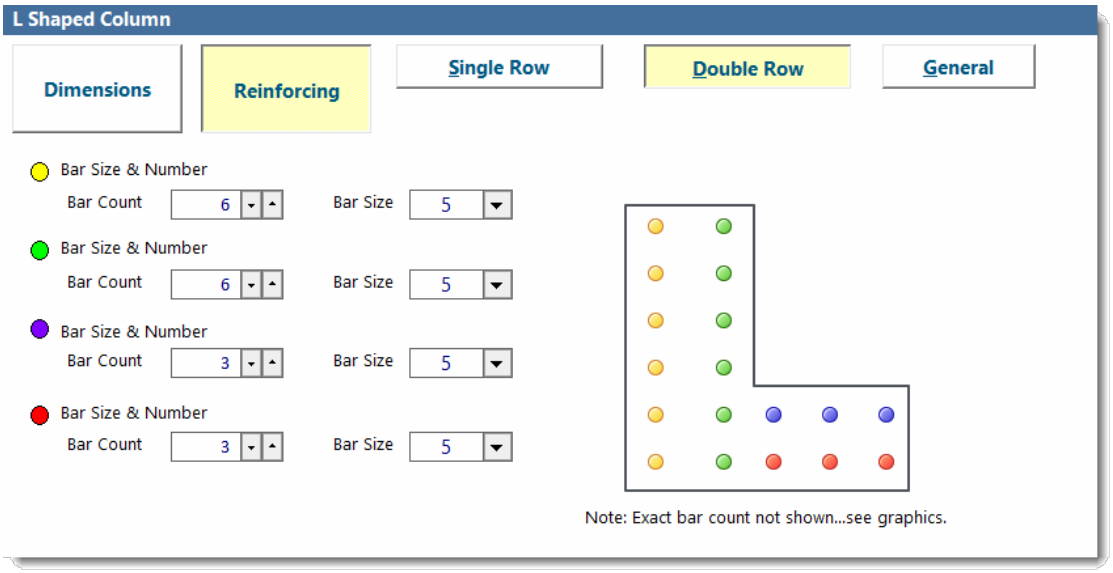
© 2026 ENERCALC, LLC

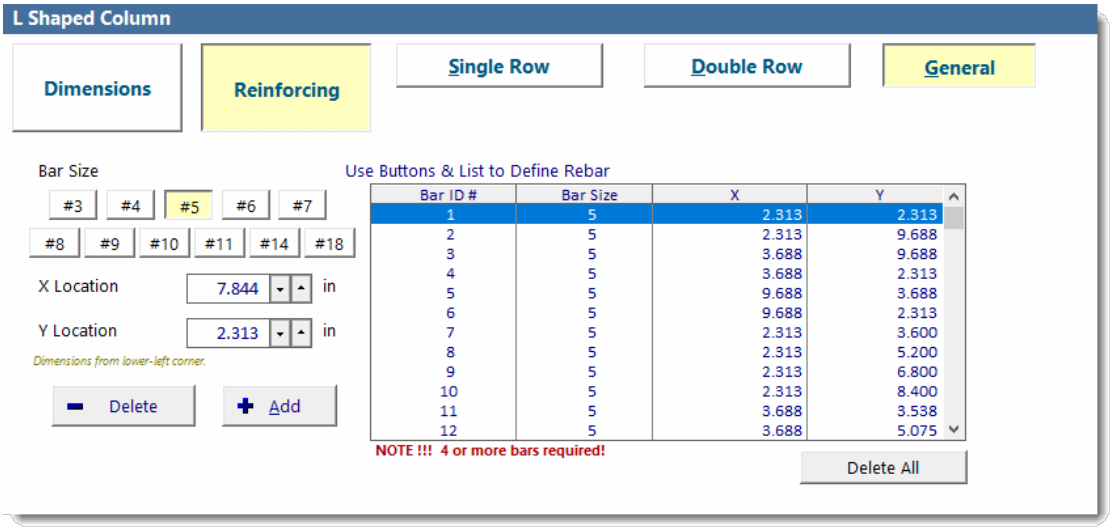
"L" Shaped Column



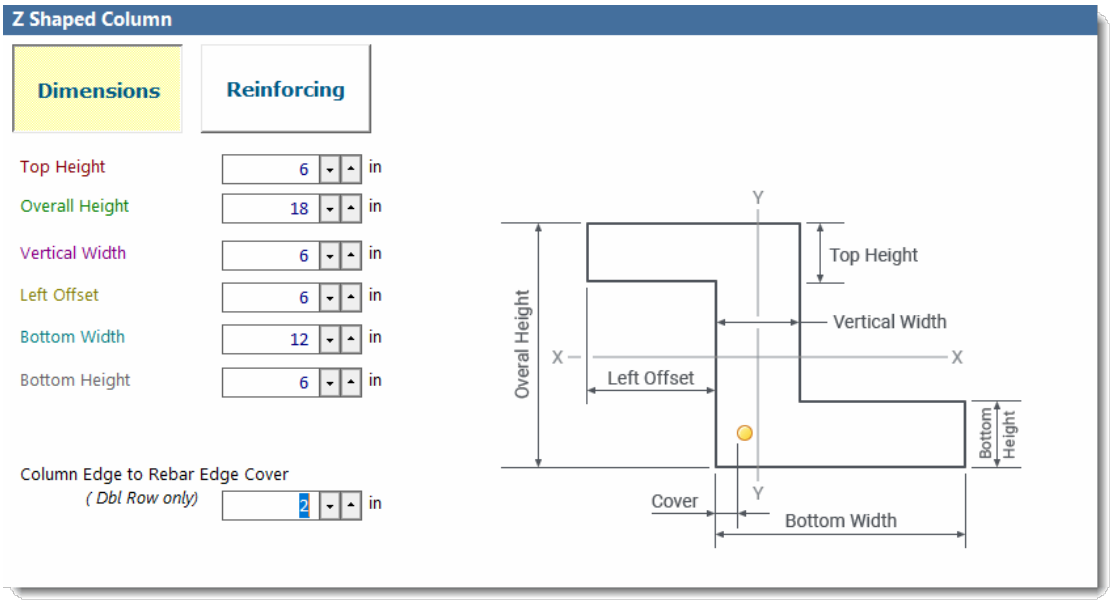


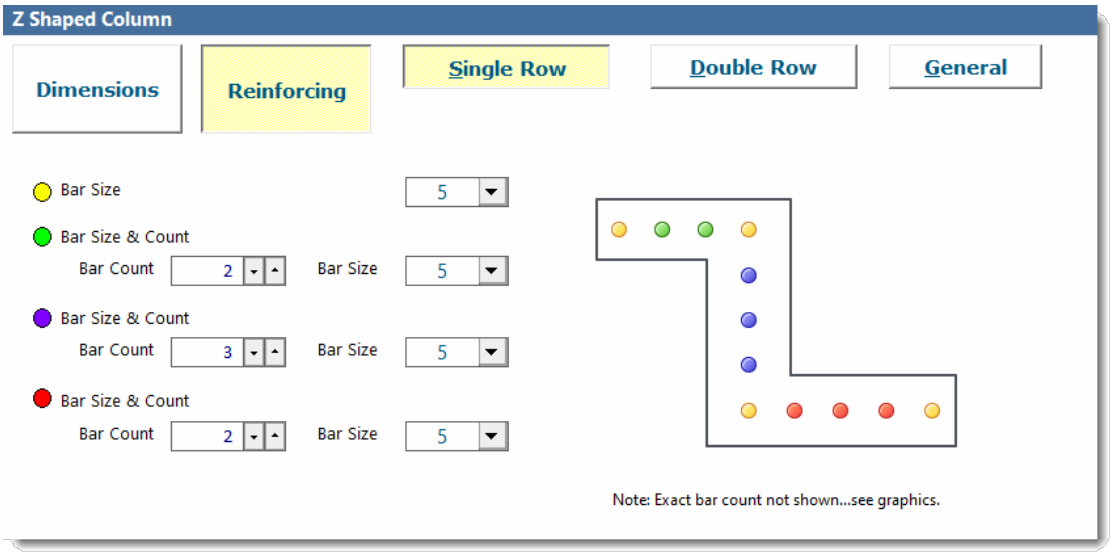
© 2026 ENERCALC, LLC


L Shaped Column
Dimensions
Reinforcing
Single Row
Double Row
General
Bar Size & Number
Bar Count 6
Bar Size 5
Bar Size & Number
Bar Count 6
Bar Size 5
Bar Size & Number
Bar Count 3
Bar Size 5
Bar Size & Number
Bar Count 3
Bar Size 5
Note: Exact bar count not shown...see graphics.

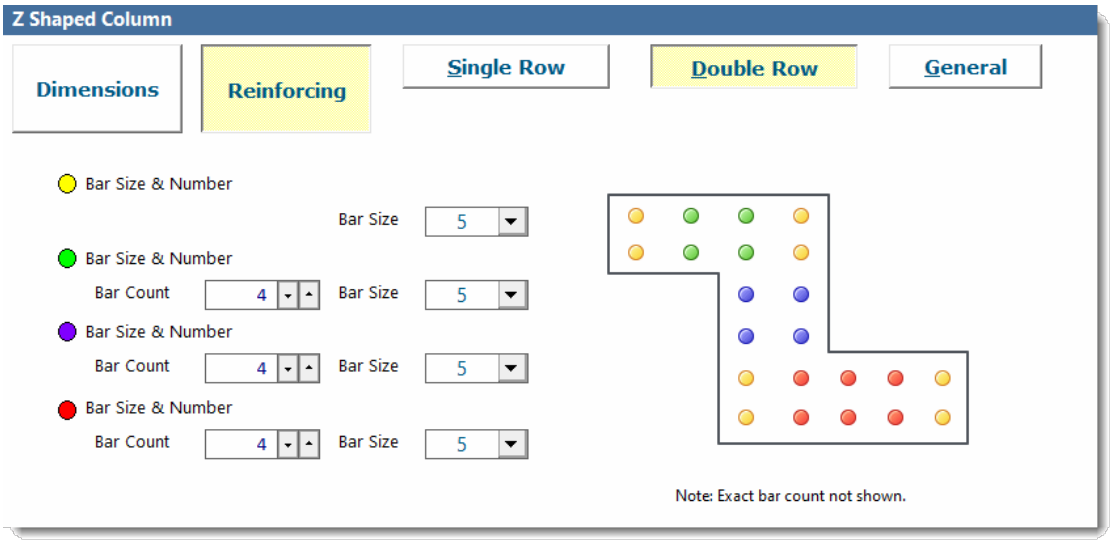

L Shaped Column
Dimensions
Reinforcing
Single Row
Double Row
General
Bar Size
#3 #4 #5 #6 #7
#8 #9 #10 #11 #14 #18
X Location 7.844 in
Y Location 2.313 in
Dimensions from lower-left corner.
Delete
Add
Use Buttons & List to Define Rebar
Bar ID # | Bar Size | X | Y
1 5 2.313 2.313
2 5 2.313 9.688
3 5 3.688 9.688
4 5 3.688 2.313
5 5 9.688 3.688
6 5 9.688 2.313
7 5 2.313 3.600
8 5 2.313 5.200
9 5 2.313 6.800
10 5 2.313 8.400
11 5 3.688 3.538
12 5 3.688 5.075
NOTE !!! 4 or more bars required!
Delete All

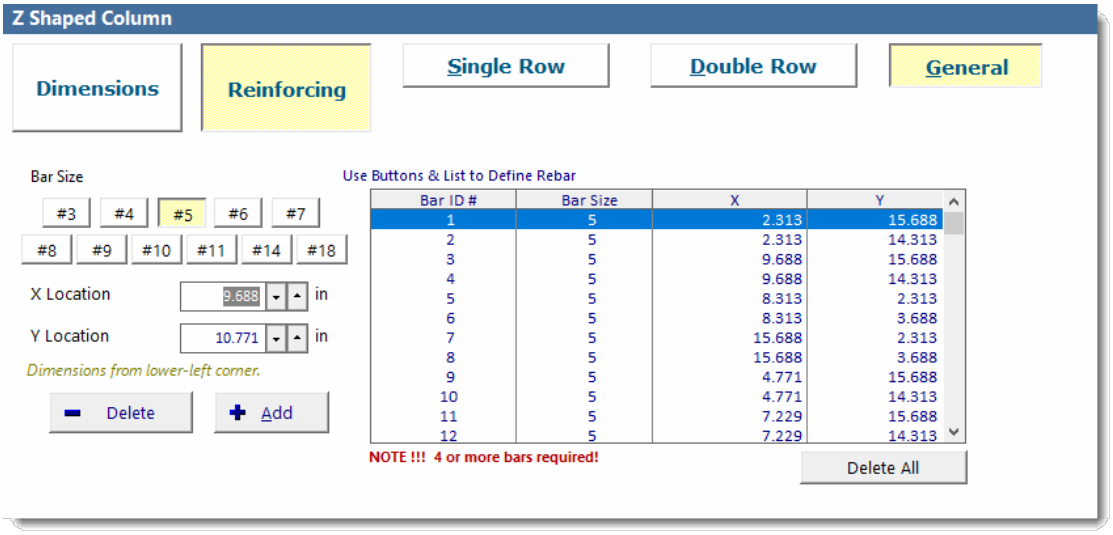
"Z" Shaped Column





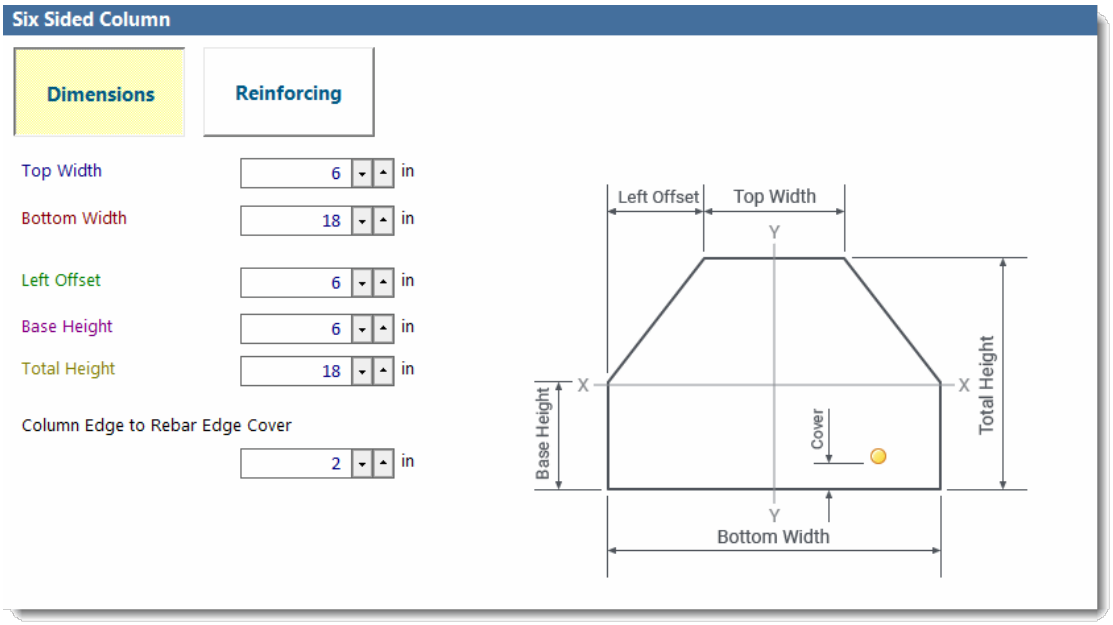
© 2026 ENERCALC, LLC

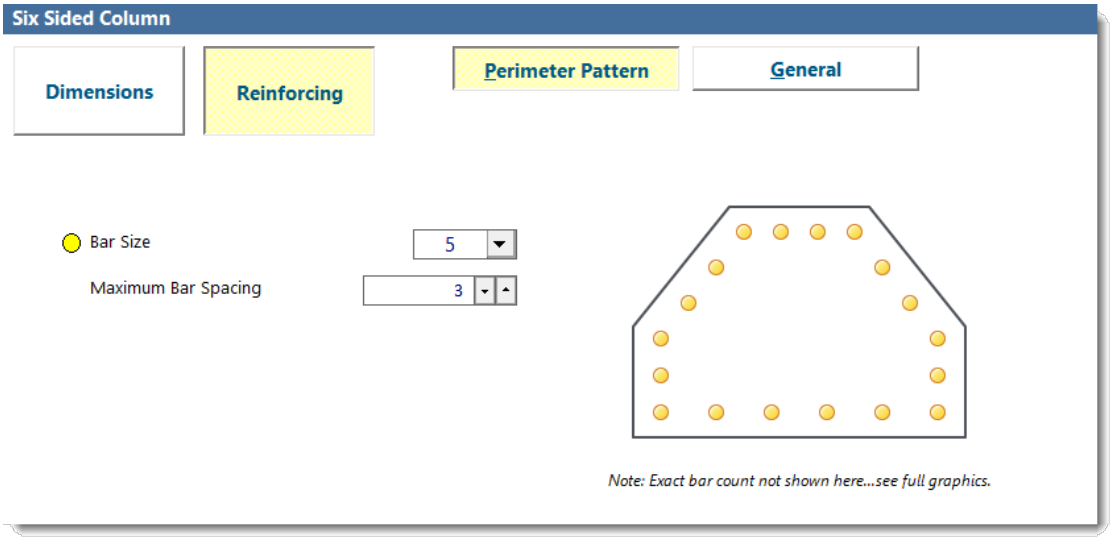




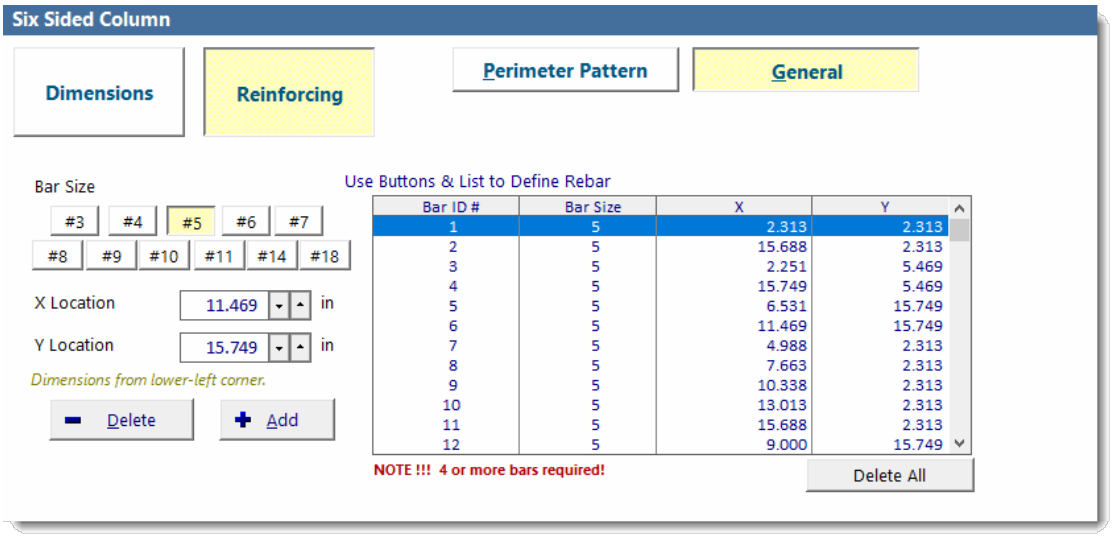
© 2026 ENERCALC, LLC

Six Sided Column



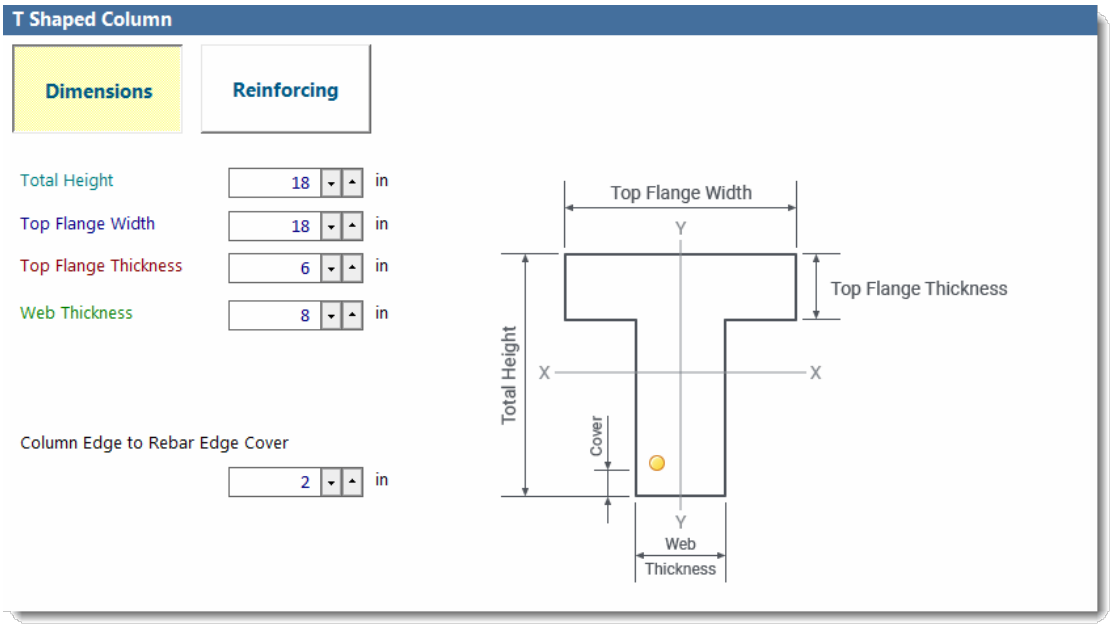


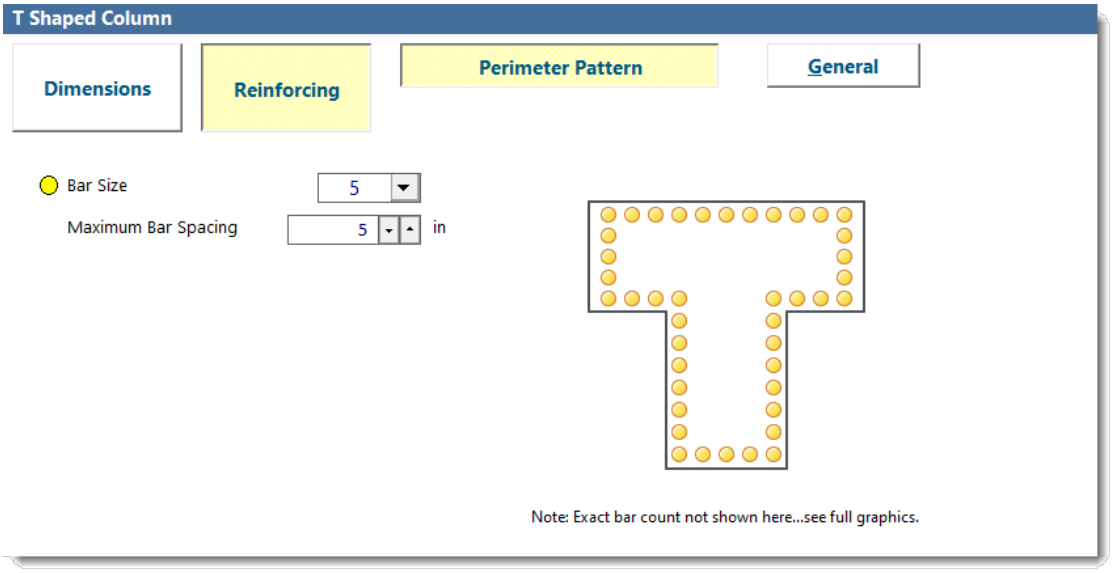
© 2026 ENERCALC, LLC



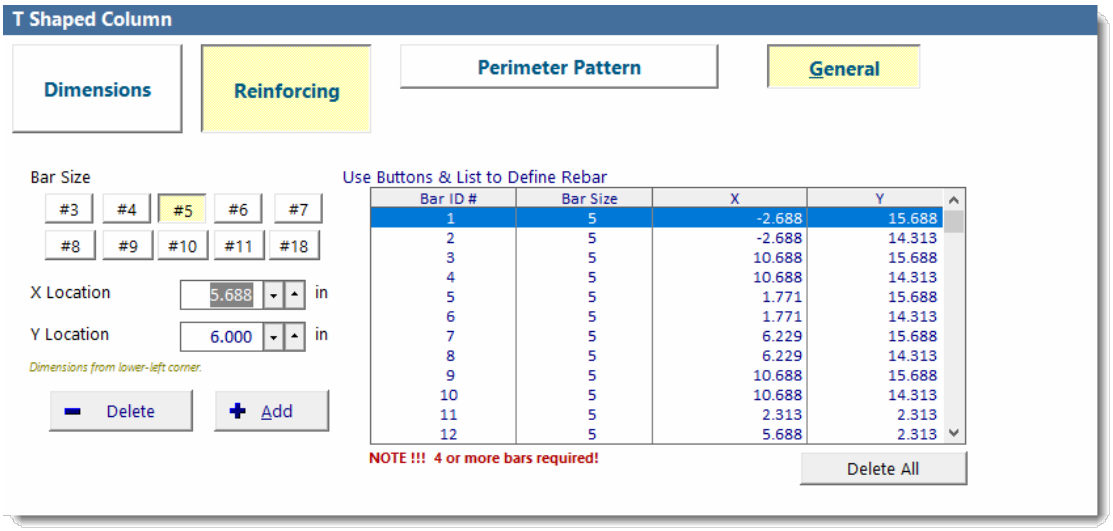
© 2026 ENERCALC, LLC

"T" Shaped Column



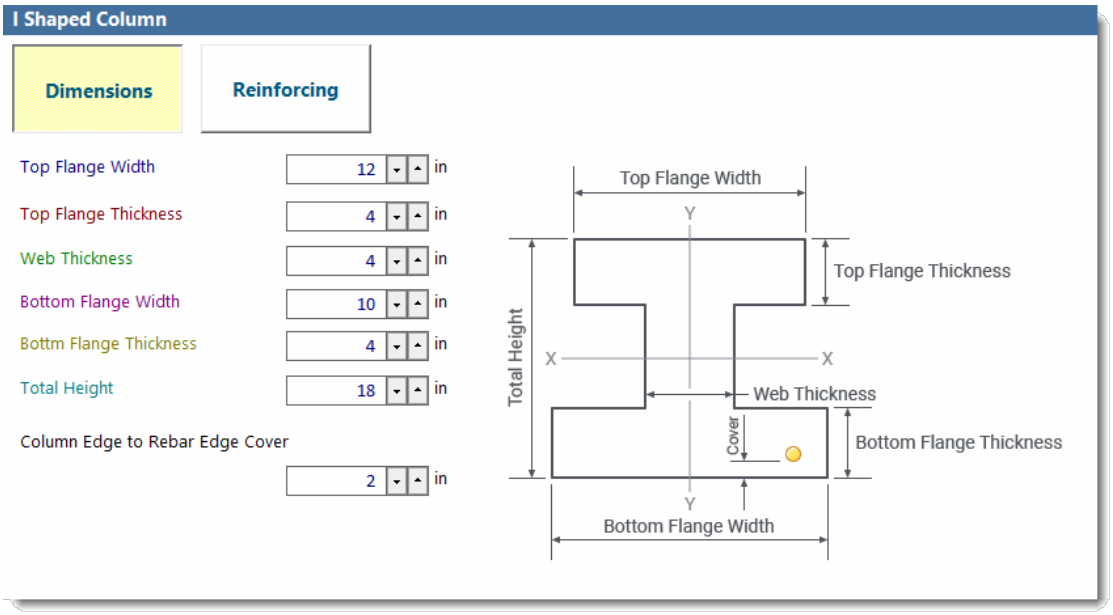


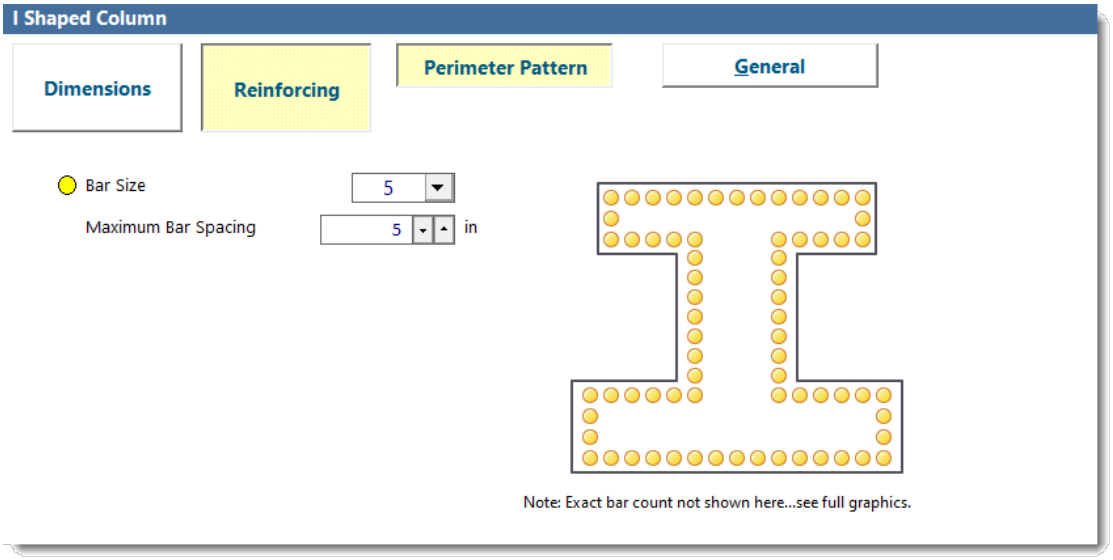
© 2026 ENERCALC, LLC



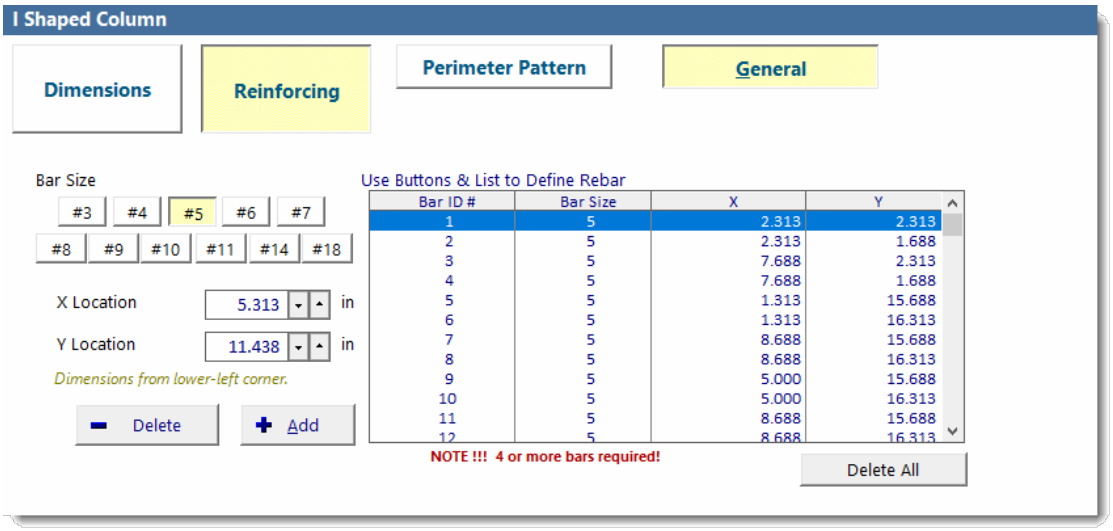
© 2026 ENERCALC, LLC

"I" Shaped Column



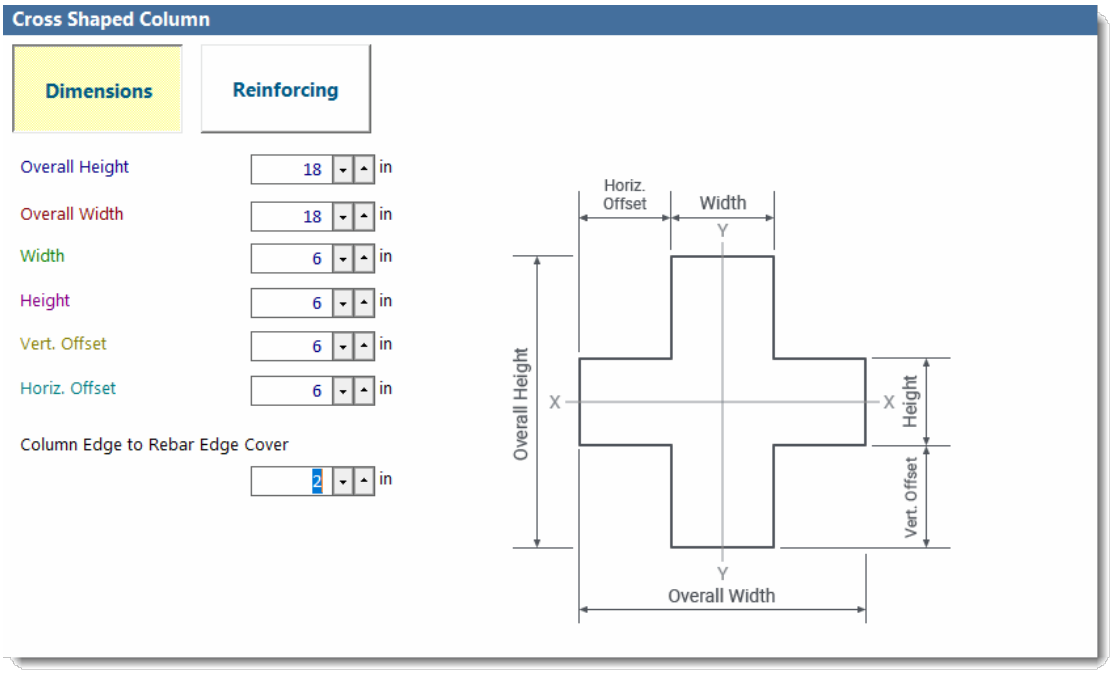


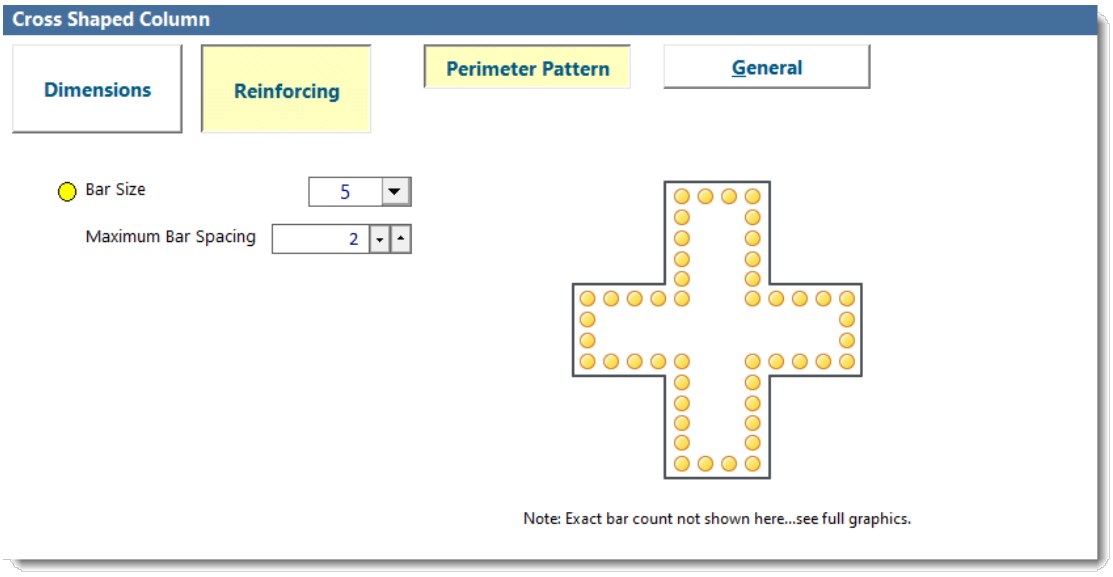
© 2026 ENERCALC, LLC

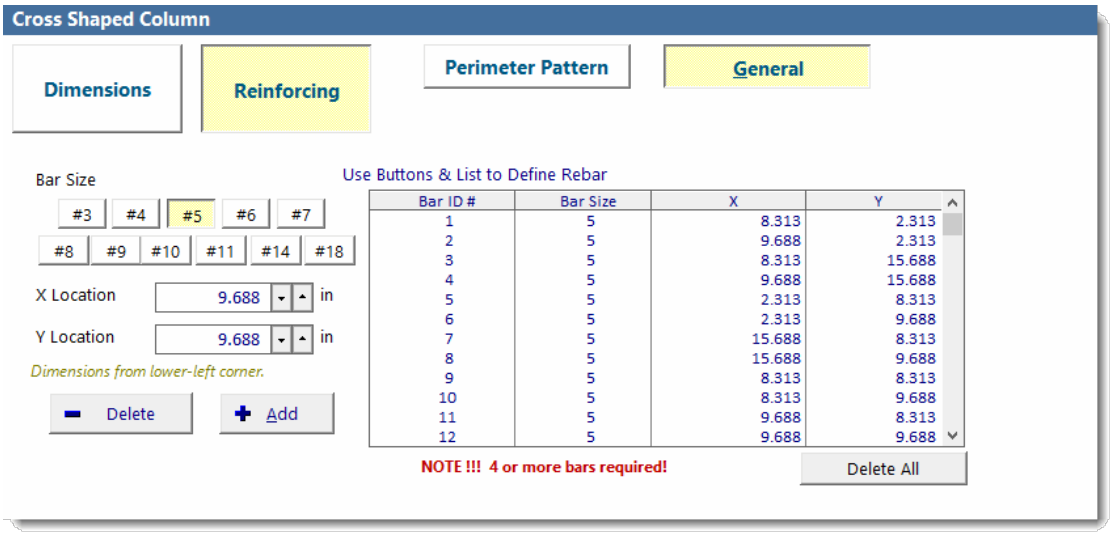


© 2026 ENERCALC, LLC

Cross Shaped Column

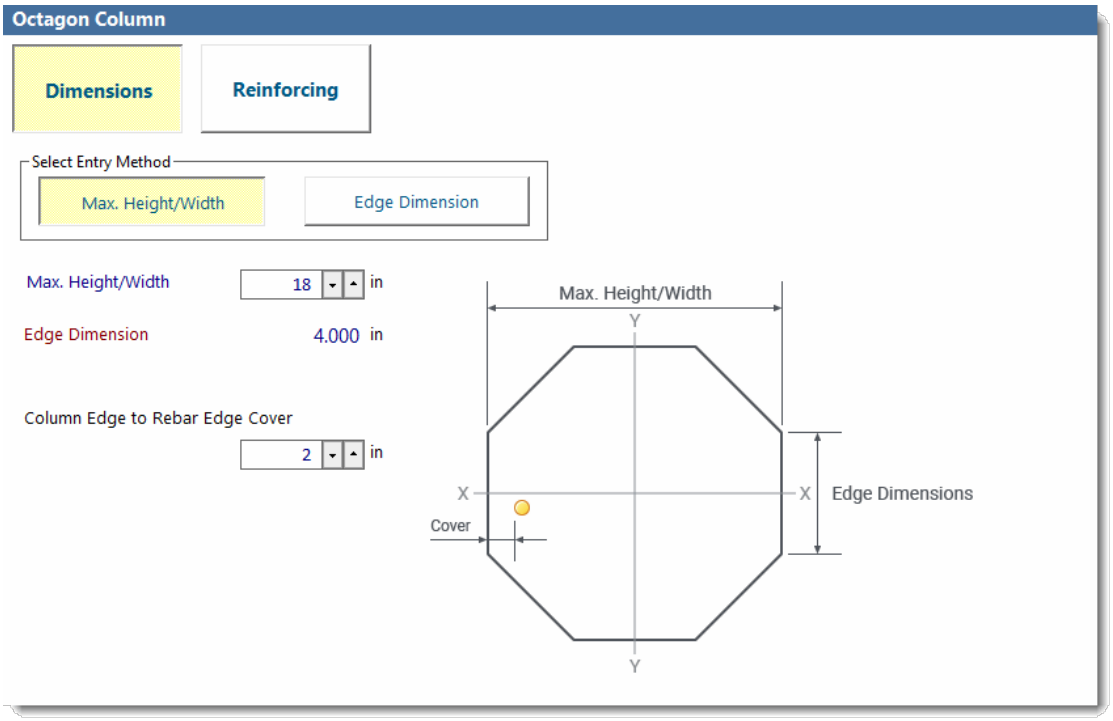


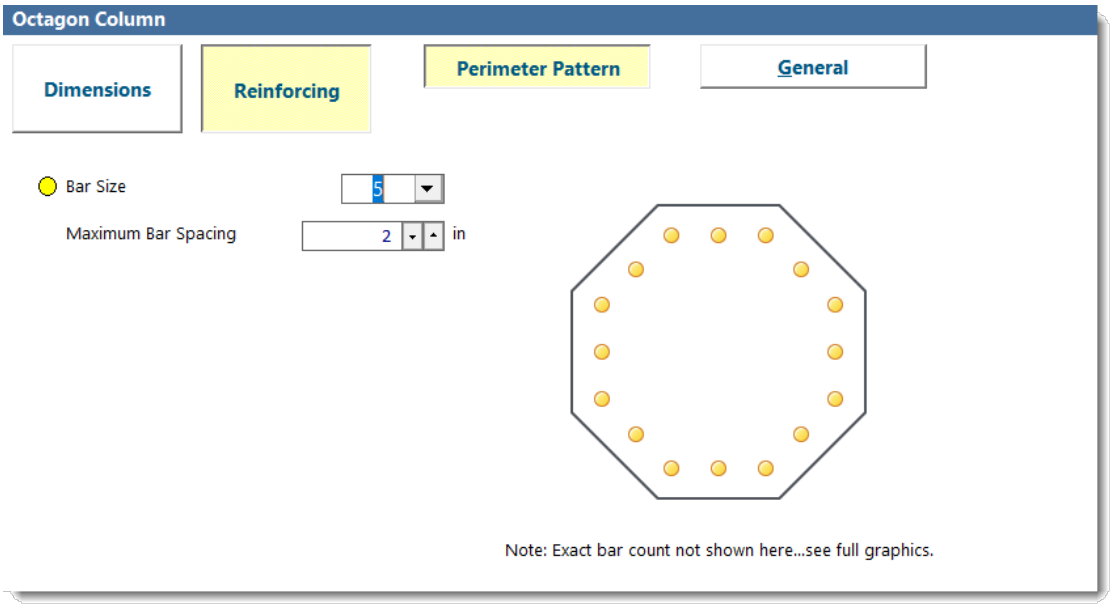




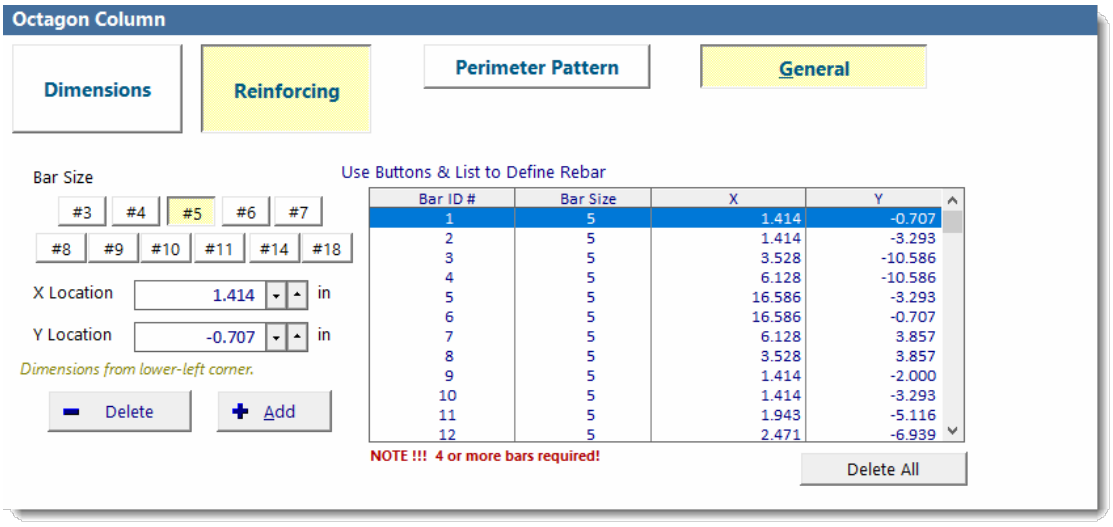
© 2026 ENERCALC, LLC

Octagon Column



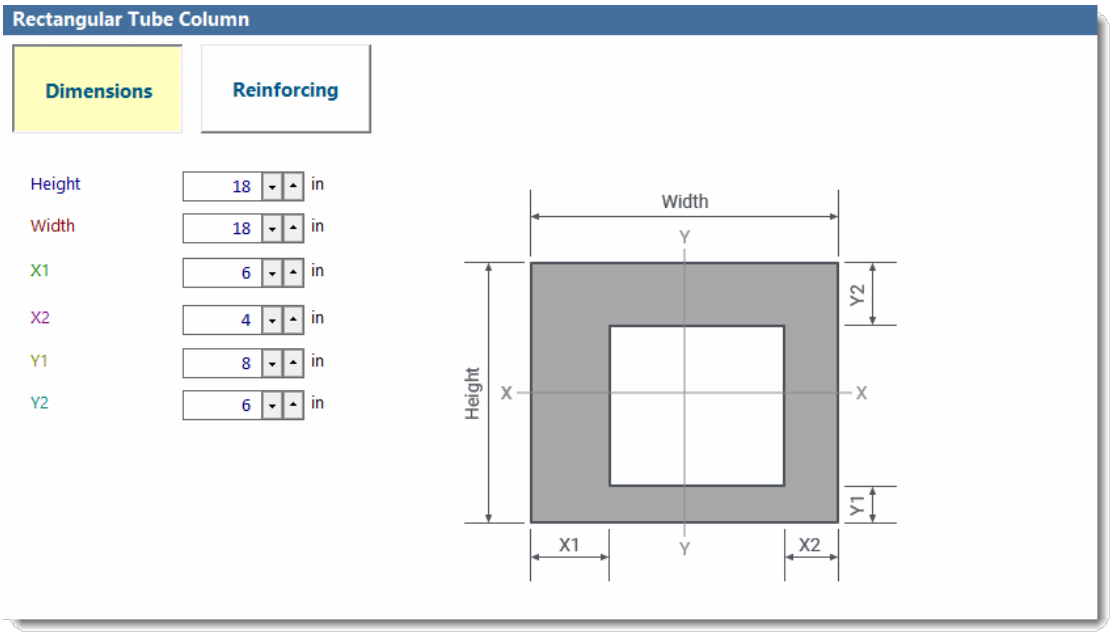


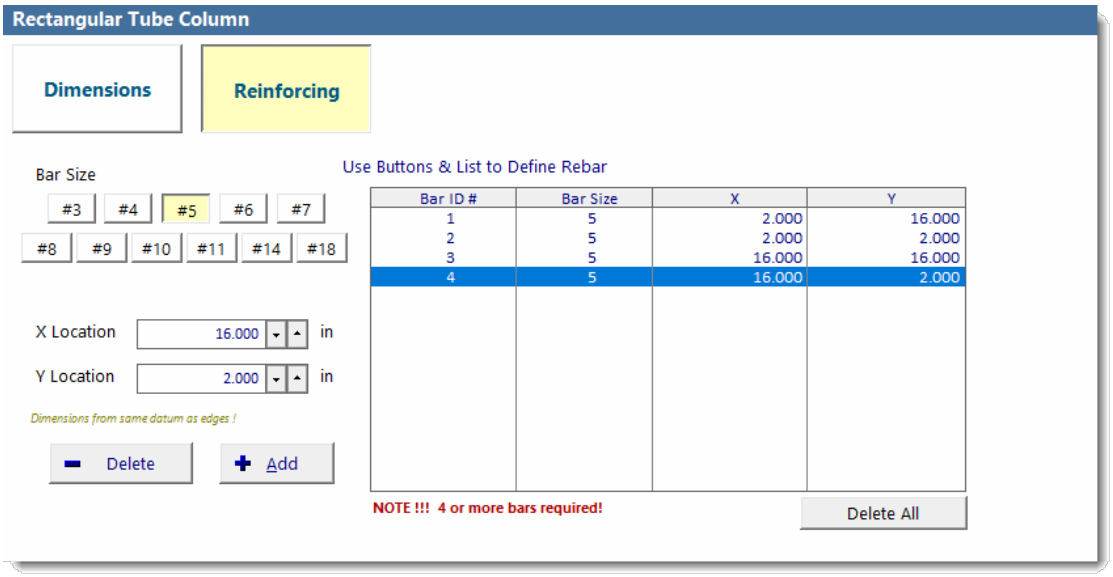
© 2026 ENERCALC, LLC



© 2026 ENERCALC, LLC

Rectangular Tube Column

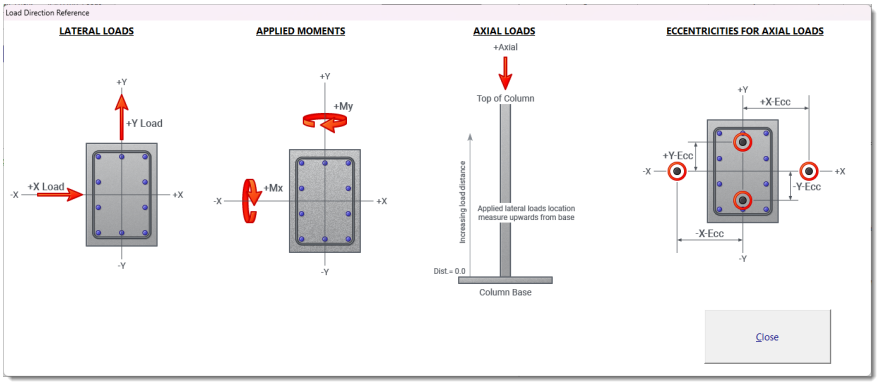




© 2026 ENERCALC, LLC

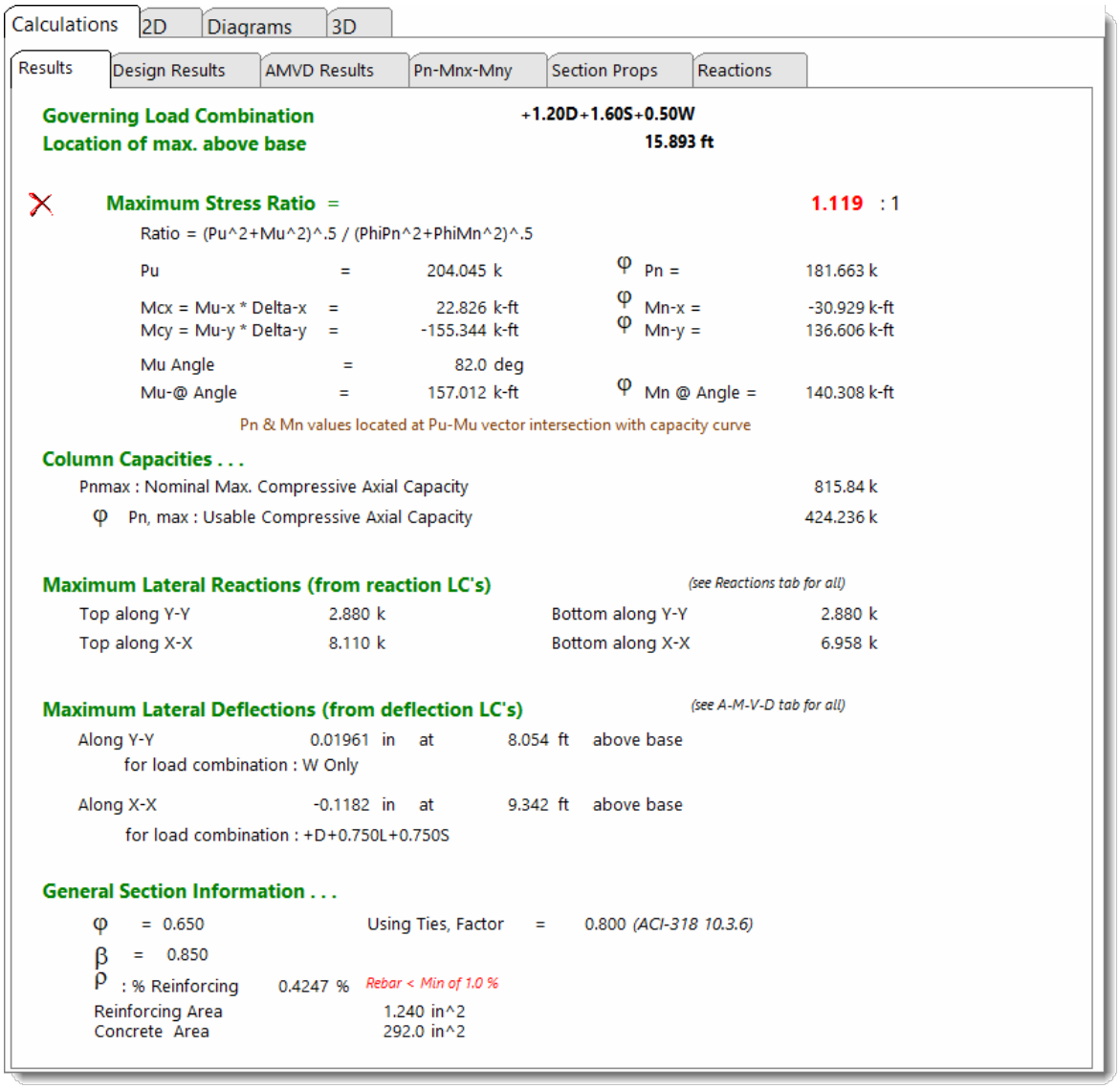
Loads

Load Reference Guide: Clicking this button opens a pop-up window that displays the positive sign conventions used in the module.



For more information on the Loads tab, please refer to the Columns 810 and Sign Convention for Concrete Column 892 topics.

Results



Calculations | 2D | Diagrams | 3D

Results | Design Results | AMVD Results | Pn-Mnx-Mny | Section Props | Reactions

Governing Load Combination +1.20D+1.60S+0.50W
Location of max. above base 15.893 ft

Maximum Stress Ratio = 1.119 : 1

Ratio = (Pu^2+Mu^2)^.5 / (PhiPn^2+PhiMn^2)^.5

Pu	=	204.045 k	φ Pn =	181.663 k	
Mcx = Mu-x * Delta-x	=	22.826 k-ft	φ Mn-x =	-30.929 k-ft	
Mcy = Mu-y * Delta-y	=	-155.344 k-ft	φ Mn-y =	136.606 k-ft	
Mu Angle	=	82.0 deg			
Mu-@ Angle	=	157.012 k-ft	φ Mn @ Angle =	140.308 k-ft	

Pn & Mn values located at Pu-Mu vector intersection with capacity curve

Column Capacities . . .

Pnmax : Nominal Max. Compressive Axial Capacity	815.84 k
φ Pn, max : Usable Compressive Axial Capacity	424.236 k

Maximum Lateral Reactions (from reaction LC's) *(see Reactions tab for all)*

Top along Y-Y	2.880 k	Bottom along Y-Y	2.880 k
Top along X-X	8.110 k	Bottom along X-X	6.958 k

Maximum Lateral Deflections (from deflection LC's) *(see A-M-V-D tab for all)*

Along Y-Y 0.01961 in at 8.054 ft above base
for load combination : W Only

Along X-X -0.1182 in at 9.342 ft above base
for load combination : +D+0.750L+0.750S

General Section Information . . .

φ = 0.650 Using Ties, Factor = 0.800 *(ACI-318 10.3.6)*
β = 0.850
ρ : % Reinforcing 0.4247 % *Rebar < Min of 1.0 %*
Reinforcing Area 1.240 in^2
Concrete Area 292.0 in^2

© 2026 ENERCALC, LLC

Design Results - Load Combination Summary

Calculations | 2D | Diagrams | 3D

Results | Design Results | AMVD Results | Pn-Mnx-Mny | Section Props | Reactions

Load Combination Summary | Load Combination Results

Load Combination	Moment Source		Dist	Pu	Phi * Pn	X-Slenderness		Y-Slenderness		Alpha	Moments @
	X-X	Y-Y	(ft)	(kips)	(kips)	Delta	Delta*Mux	Delta	Delta*Muy	(deg)	Delta*Mu
+1.40D		Actual	15.89	185.79	218.67			1.00	119.47	270.00	119.47
+1.20D+0.50Lr+1.60L		Actual	15.89	209.65	218.37			1.00	136.00	270.00	136.00
+1.20D+1.60L+0.50S		Actual	15.89	211.65	218.37			1.00	137.33	270.00	137.33
+1.20D+1.60Lr+0.50L		Actual	15.89	209.65	218.37			1.00	136.00	270.00	136.00
+1.20D+1.60Lr+0.50W	M2,min	Actual	15.89	197.65	182.96	1.17	21.99	1.17	149.51	82.00	151.12
+1.20D+0.50L+1.60S		Actual	15.89	216.05	218.37			1.00	140.27	270.00	140.27
+1.20D+1.60S+0.50W	M2,min	Actual	15.89	204.05	181.66	1.18	22.83	1.17	155.34	82.00	157.01
+1.20D+0.50Lr+0.50L+W	M2,min	Actual	15.89	183.25	185.30	1.16	20.13	1.15	136.63	82.00	138.10
+1.20D+0.50L+0.50S+W	M2,min	Actual	15.89	185.25	184.05	1.16	20.39	1.16	138.40	82.00	139.89
+1.20D+0.50L+0.70S		Actual	15.89	190.85	218.67			1.00	123.47	270.00	123.47
+0.90D+W	Actual	Actual	15.89	119.43	190.33	1.10	12.63	1.10	84.11	81.00	85.06
+0.90D		Actual	15.89	119.43	218.67			1.00	76.80	270.00	76.80

Design Results - Load Combination Results

Calculations | 2D | Diagrams | 3D

Results | Design Results | AMVD Results | Pn-Mnx-Mny | Section Props | Reactions

Load Combination Summary | Load Combination Results

Load Combination	Dist. From	Factored Applied (k & k-ft)			elta : Mag. Factor		Magnified Moments (k-ft)		Delta*Mu
Name	Base (ft)	Pu	Alpha (deg)	Mu@Alpha	X-X	Y-Y	DeltaX*Mux	DeltaY*Muy	@ Alpha
+1.40D	15.89	185.79	270.0	119.47		1.000		-119.47	119.47
+1.20D+0.50Lr+1.60L	15.89	209.65	270.0	136.00		1.000		-136.00	136.00
+1.20D+1.60L+0.50S	15.89	211.65	270.0	137.33		1.000		-137.33	137.33
+1.20D+1.60Lr+0.50L	15.89	209.65	270.0	136.00		1.000		-136.00	136.00
+1.20D+1.60Lr+0.50W	15.89	197.65	82.0	129.37	1.171	1.168	21.99	-149.51	151.12
+1.20D+0.50L+1.60S	15.89	216.05	270.0	140.27		1.000		-140.27	140.27
+1.20D+1.60S+0.50W	15.89	204.05	82.0	133.68	1.178	1.174	22.83	-155.34	157.01
+1.20D+0.50Lr+0.50L+W	15.89	183.25	82.0	119.67	1.157	1.154	20.13	-136.63	138.10
+1.20D+0.50L+0.50S+W	15.89	185.25	82.0	121.02	1.159	1.156	20.39	-138.40	139.89
+1.20D+0.50L+0.70S	15.89	190.85	270.0	123.47		1.000		-123.47	123.47
+0.90D+W	15.89	119.43	81.0	77.66	1.097	1.095	12.63	-84.11	85.06
+0.90D	15.89	119.43	270.0	76.80		1.000		-76.80	76.80

Note: The value of Euler Buckling load (Pc) is calculated using the following formula from ACI 318 Section R6.6.4.4.4: $EI = 0.25E_c I_g$.

© 2026 ENERCALC, LLC

A-M-V-D Results - Strength Design Results

Calculations | 2D | Diagrams | 3D

Results | Design Results | AMVD Results | Pn-Mnx-Mny | Section Props | Reactions | Expand Results

Strength Design Results | Service Load Deflections

FACTORED LOADS Load Combination, Distance	Pu (k)	Moment (k-ft) Mu - xx	 Mu - yy	Shear (k) Vu - xx	 Vu - yy
+1.40D					
+1.20D+0.50Lr+1.60L					
+1.20D+1.60L+0.50S					
+1.20D+1.60Lr+0.50L					
+1.20D+1.60Lr+0.50W					
0.000 ft	197.645			6.720	-1.440
0.107 ft	197.645	0.154	-0.723	6.737	-1.421
0.215 ft	197.645	0.305	-1.447	6.754	-1.401
0.322 ft	197.645	0.455	-2.173	6.772	-1.382
0.430 ft	197.645	0.602	-2.901	6.789	-1.363

A-M-V-D Results - Service Load Deflections

Calculations | 2D | Diagrams | 3D

Results | Design Results | AMVD Results | Pn-Mnx-Mny | Section Props | Reactions

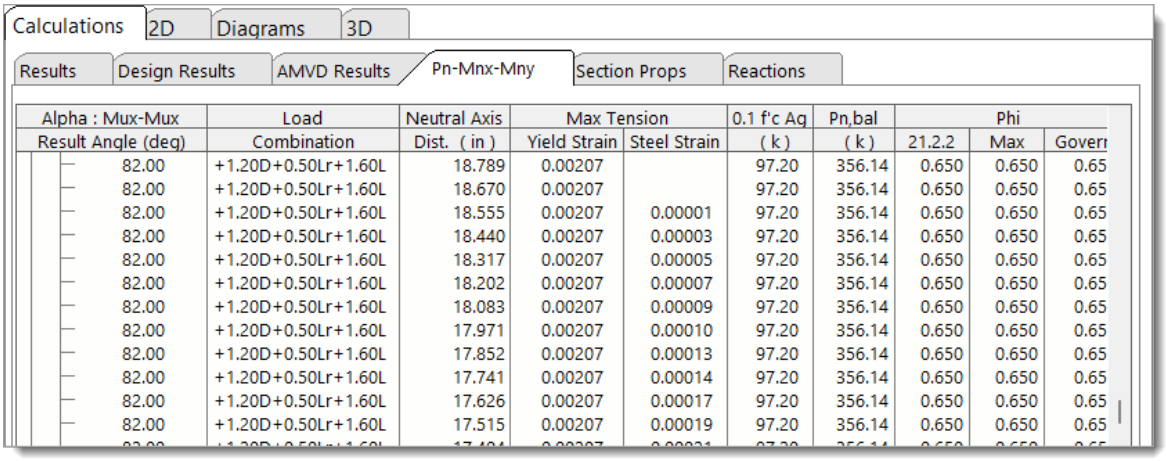
Strength Design Results | Service Load Deflections

SERVICE LOADS Load Combination, Distance	Deflections (in) xx Defl	 yy Defl	Moment of Inertia (in^4) Ixx	 Iyy
D Only				
+D+L				
+D+Lr				
+D+S				
+D+0.750Lr+0.750L				
+D+0.750L+0.750S				
0.000 ft			8,669.83	8,541.83
0.107 ft	-0.0020		8,669.83	8,541.83
0.215 ft	-0.0041		8,669.83	8,541.83
0.322 ft	-0.0061		8,669.83	8,541.83
0.430 ft	-0.0082		8,669.83	8,541.83

Note: Deflections are based on Ig.

© 2026 ENERCALC, LLC

P-Mx-My Capacities



Alpha : Mux-Mux	Load	Neutral Axis	Max Tension		0.1 f'c Ag	Pn,bal	Phi		
Result Angle (deg)	Combination	Dist. (in)	Yield Strain	Steel Strain	(k)	(k)	21.2.2	Max	Goverr
82.00	+1.20D+0.50Lr+1.60L	18.789	0.00207		97.20	356.14	0.650	0.650	0.65
82.00	+1.20D+0.50Lr+1.60L	18.670	0.00207		97.20	356.14	0.650	0.650	0.65
82.00	+1.20D+0.50Lr+1.60L	18.555	0.00207	0.00001	97.20	356.14	0.650	0.650	0.65
82.00	+1.20D+0.50Lr+1.60L	18.440	0.00207	0.00003	97.20	356.14	0.650	0.650	0.65
82.00	+1.20D+0.50Lr+1.60L	18.317	0.00207	0.00005	97.20	356.14	0.650	0.650	0.65
82.00	+1.20D+0.50Lr+1.60L	18.202	0.00207	0.00007	97.20	356.14	0.650	0.650	0.65
82.00	+1.20D+0.50Lr+1.60L	18.083	0.00207	0.00009	97.20	356.14	0.650	0.650	0.65
82.00	+1.20D+0.50Lr+1.60L	17.971	0.00207	0.00010	97.20	356.14	0.650	0.650	0.65
82.00	+1.20D+0.50Lr+1.60L	17.852	0.00207	0.00013	97.20	356.14	0.650	0.650	0.65
82.00	+1.20D+0.50Lr+1.60L	17.741	0.00207	0.00014	97.20	356.14	0.650	0.650	0.65
82.00	+1.20D+0.50Lr+1.60L	17.626	0.00207	0.00017	97.20	356.14	0.650	0.650	0.65
82.00	+1.20D+0.50Lr+1.60L	17.515	0.00207	0.00019	97.20	356.14	0.650	0.650	0.65

Yield Strain: Yield strain of reinforcement taken as f_y / E.

Steel Strain: Net tensile strain (ε_t) in the extreme layer of longitudinal tension reinforcement at nominal strength.

Phi:

- ACI 318-14: Calculated per Table 21.2.2, where φ is obtained by linear interpolation between 0.65 (or 0.75 for spiral ties) and 0.90 based on a net tensile strain (ε_t) ranging from the yield strain (ε_{ty}) to 0.005, respectively.
- ACI 318-19: Calculated per Table 21.2.2, where φ is obtained by linear interpolation between 0.65 (or 0.75 for spiral ties) and 0.90 based on a net tensile strain (ε_t) ranging from the yield strain (ε_{ty}) to ε_{ty} + 0.003, respectively.
- ACI 318-25: Taken as the minimum between the following two values:
  - Phi - 21.2.2: Calculated per Table 21.2.2, where φ is obtained by linear interpolation between 0.65 (or 0.75 for spiral ties) and 0.90 based on a net tensile strain (ε_t) ranging from the yield strain (ε_{ty}) to ε_{ty} + 0.003, respectively.
  - Phi - Max: Calculated per Section 21.2.2.3, where φ Max is obtained by linear interpolation between 0.65 (or 0.75 for spiral ties) and 0.90 based on a nominal axial strength (P_n) ranging from the nominal axial strength at the balanced strain condition ($P_{n,bal}$) to 0.1 f'_c A_g, respectively.

© 2026 ENERCALC, LLC

Section Properties

Calculations | 2D | Diagrams | 3D

Results | Design Results | AMVD Results | Pn-Mnx-Mny | Section Props | Reactions

Alpha | Sx | Sy | Ix | Iy | Ixy | Rx | Ry

Alpha	Sxx	Syy	Ixx	Iyy	Ixy	rxx	ryy	Cracked Section
deg	in^3	in^3	in^4	in^4	in^4	in	in	Icracked (in^3)
0	951.725	937.674	8,669.826	8,541.826	-35.507	5.449	5.409	2,535.522
15	781.879	764.785	8,679.006	8,532.647	1.250	5.452	5.406	2,526.104
30	702.800	686.526	8,668.576	8,543.077	37.672	5.449	5.409	2,451.444
45	678.927	665.247	8,641.333	8,570.320	64.000	5.440	5.418	2,227.672
60	697.611	691.669	8,604.576	8,607.077	73.179	5.428	5.429	1,868.016
75	771.892	774.721	8,568.154	8,643.499	62.750	5.417	5.441	1,457.408
90	937.674	951.725	8,541.826	8,669.826	35.507	5.409	5.449	1,127.642
105	764.785	781.879	8,532.647	8,679.006	-1.250	5.406	5.452	947.607
120	686.526	702.800	8,543.077	8,668.576	-37.672	5.409	5.449	840.028
135	665.247	678.927	8,570.320	8,641.333	-64.000	5.418	5.440	741.924

Reactions

Calculations | 2D | Diagrams | 3D

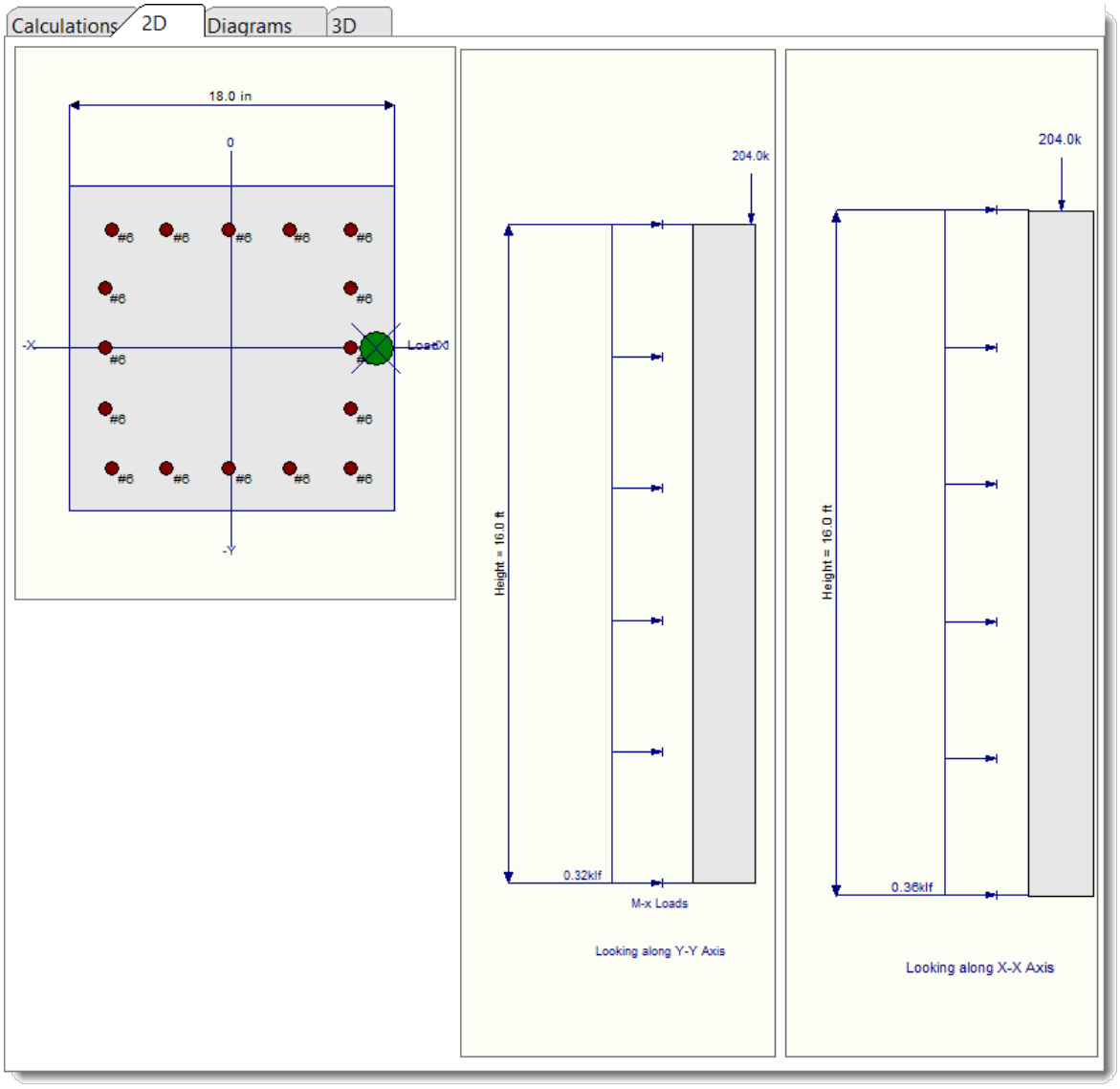
Results | Design Results | AMVD Results | Pn-Mnx-Mny | Section Props | Reactions

Service Load Reactions . . .

SERVICE LOADS	Axial (k)	X-X Reaction (k)		Y-Y Reaction (k)		My (kft)		Mx (kft)	
Load Combination	@ Base	@ Base	@ Top	@ Base	@ Top	@ Base	@ Top	@ Base	To
D Only	132.704	5.333	5.333						
+D+L	156.704	6.333	6.333						
+D+Lr	156.704	6.333	6.333						
+D+S	160.704	6.500	6.500						
+D+0.750Lr+0.750L	168.704	6.833	6.833						
+D+0.750L+0.750S	171.704	6.958	6.958						
+D+0.60W	132.704	3.797	6.869	1.728	1.728				
+D+0.750Lr+0.750L+0.450W	168.704	5.681	7.985	1.296	1.296				
+D+0.750L+0.750S+0.450W	171.704	5.806	8.110	1.296	1.296				
+0.60D+0.60W	79.623	1.664	4.736	1.728	1.728				
+0.60D	79.623	3.200	3.200						
Lr Only	24.000	1.000	1.000						
L Only	24.000	1.000	1.000						
S Only	28.000	1.167	1.167						
W Only		2.560	2.560	2.880	2.880				

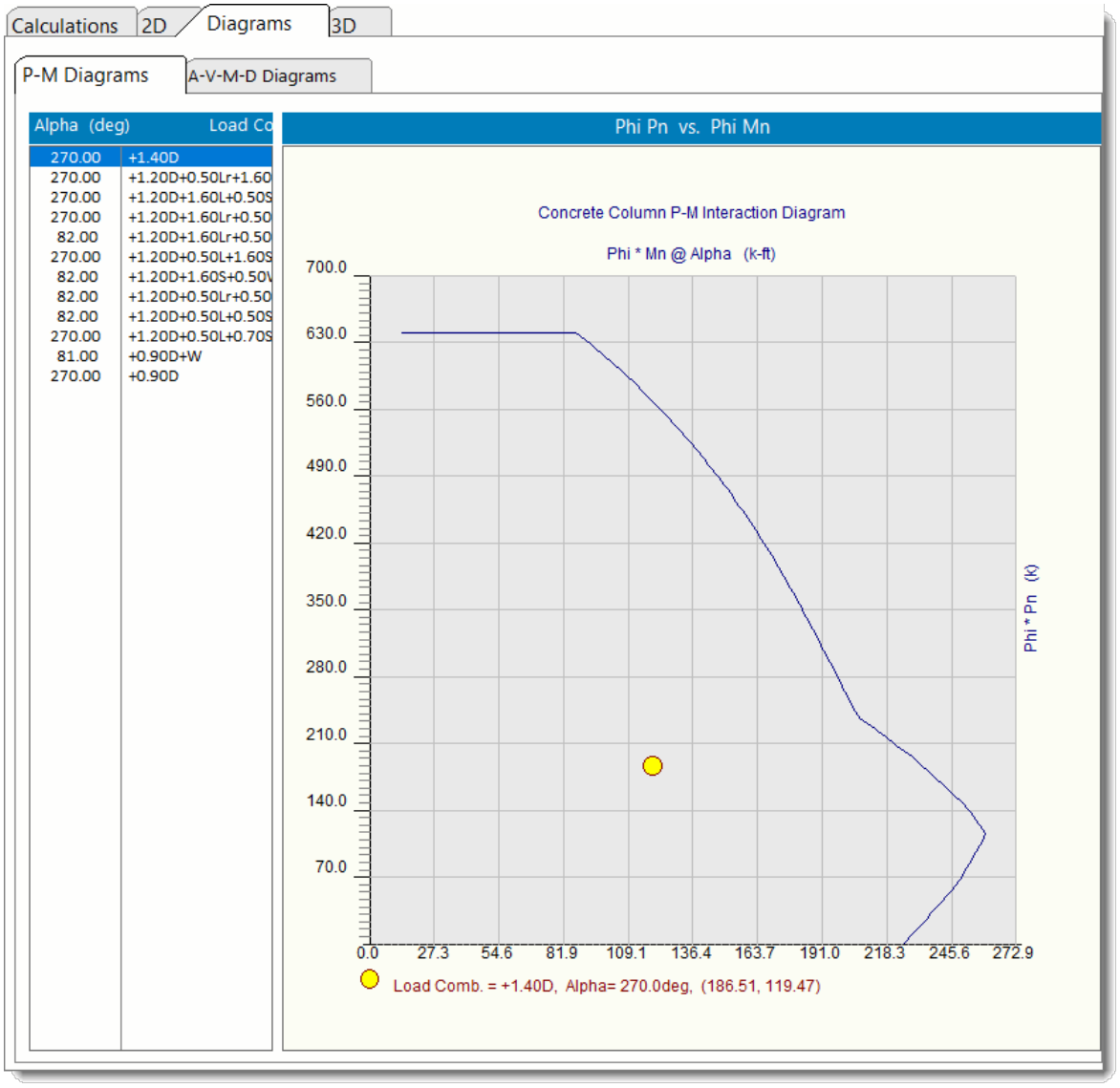
© 2026 ENERCALC, LLC

2D Sketch



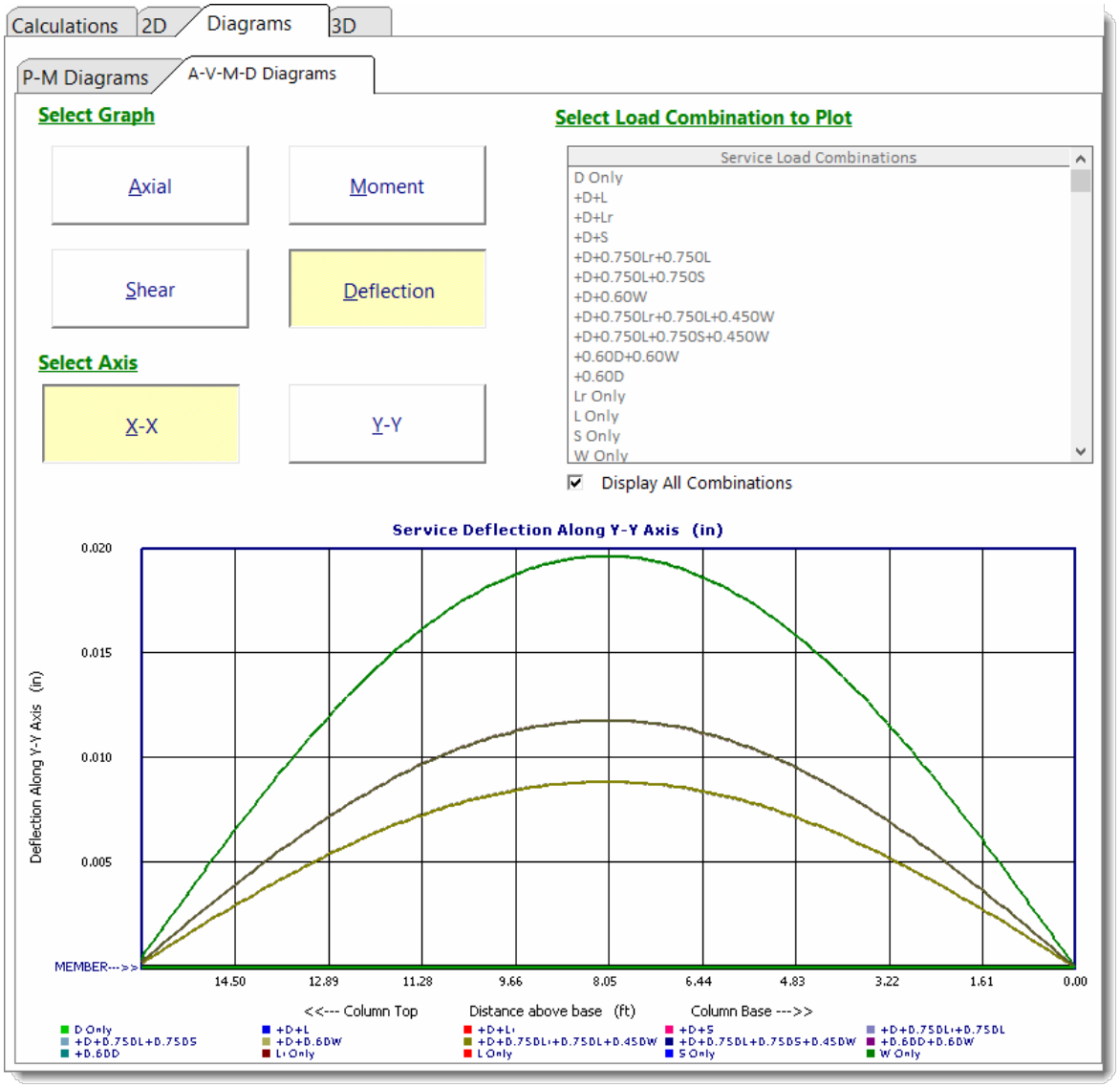
© 2026 ENERCALC, LLC

Diagrams - P-M Diagrams

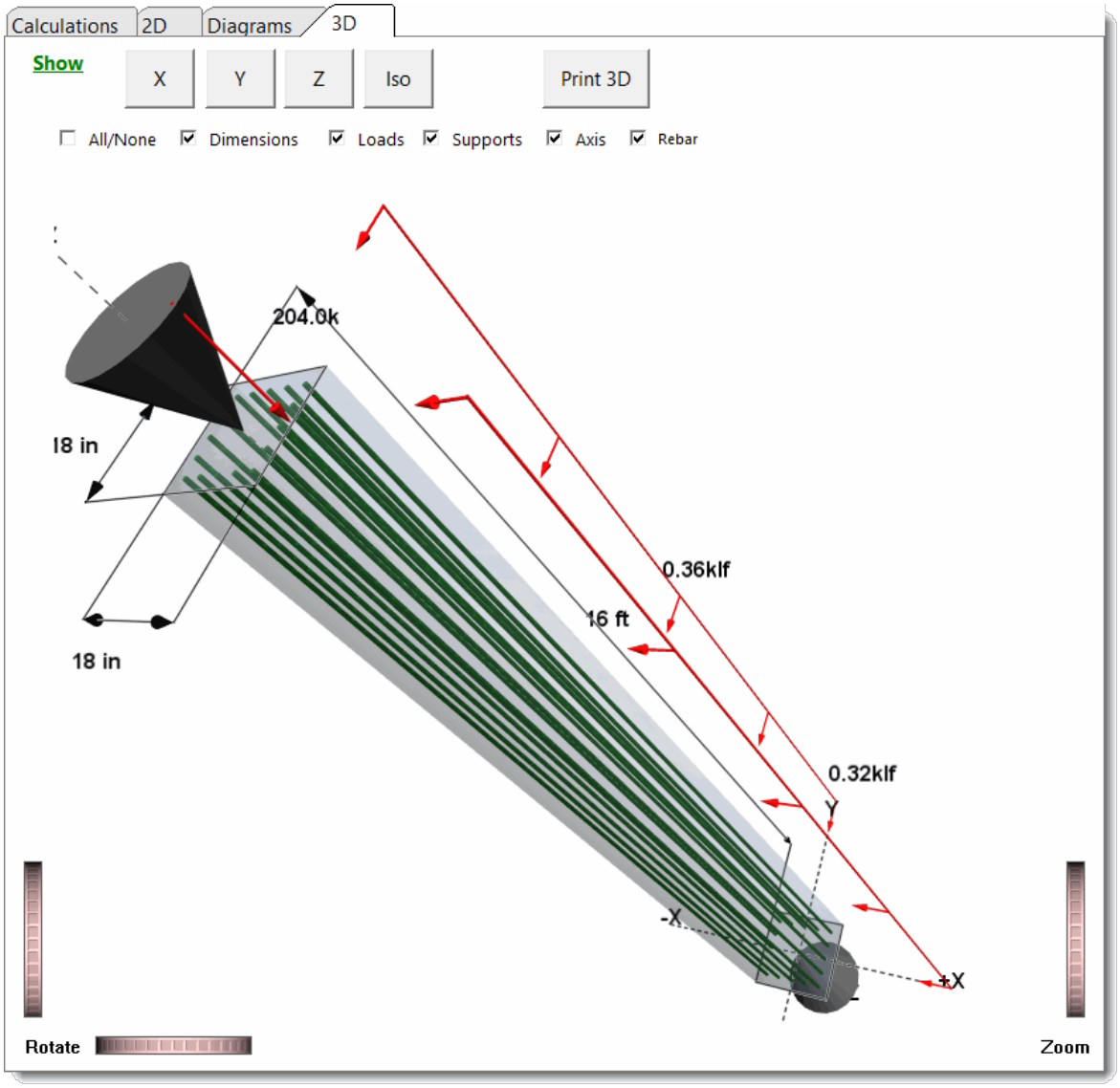


© 2026 ENERCALC, LLC

Diagrams - A-V-M-D Diagrams



3D Rendering



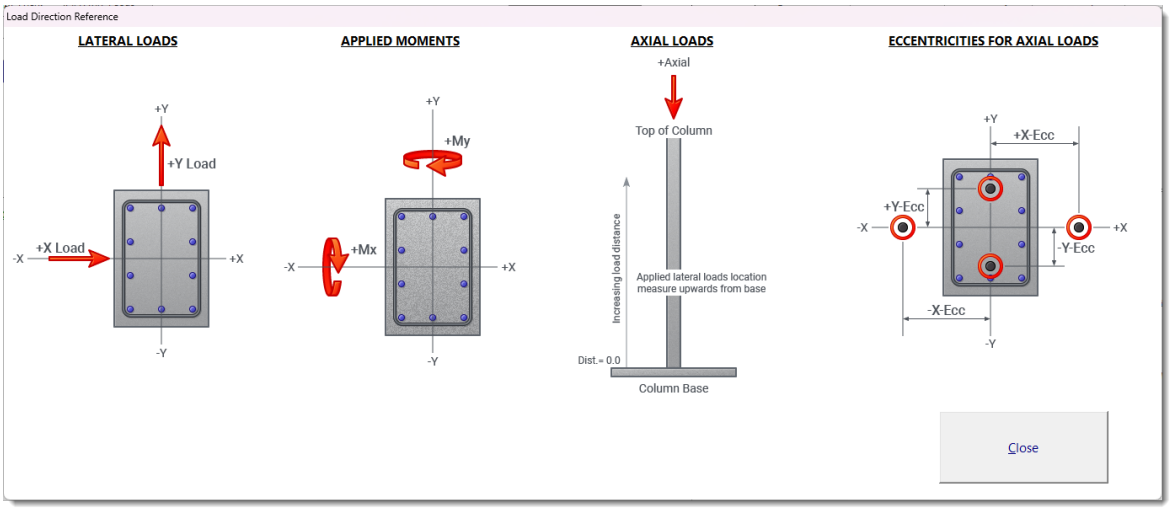
Theoretical Basis

Click here to view Concrete Column Module Theoretical Basis.

13.5.4.1 Sign Convention for Concrete Column

Coordinate Axis System (Column in Plan View)

© 2026 ENERCALC, LLC



Applied Axial Load Sign Convention

Applied axial loads are treated as downward loads. They act in the same direction as gravity.

Applied Shear Load Sign Convention

Applied shear loads respect the logical convention.

Positive shears act in the positive direction of the selected axis.

Negative shears act in the negative direction of the selected axis.

Applied Moment Sign Convention

Applied moments follow the right hand rule.

To visualize the direction of action of a positive Mx, point your right thumb in the +X direction, and the natural curl of your right fingers will indicate the direction of action of a positive Mx.

To visualize the direction of action of a negative Mx, point your right thumb in the -X direction, and the natural curl of your right fingers will indicate the direction of action of a negative Mx.

To visualize the direction of action of a positive My, point your right thumb in the +Y direction, and the natural curl of your right fingers will indicate the direction of action of a positive My.

© 2026 ENERCALC, LLC

To visualize the direction of action of a negative My, point your right thumb in the -Y direction, and the natural curl of your right fingers will indicate the direction of action of a negative My.

Internal Axial Loads Sign Convention

Internal axial loads reported on Calculations > AMVD Results > Strength Design Results follow the convention that positive values represent compression.

Internal Moments Sign Convention

Internal moments reported on Calculations > AMVD Results > Strength Design Results follow this convention:

A positive Mx means the +Y surface is in tension, and the -Y surface is in compression.

A negative Mx means the -Y surface is in tension, and the +Y surface is in compression.

A positive My means the +X surface is in tension, and the -X surface is in compression.

A negative My means the -X surface is in tension, and the +X surface is in compression.

Reactions Sign Convention

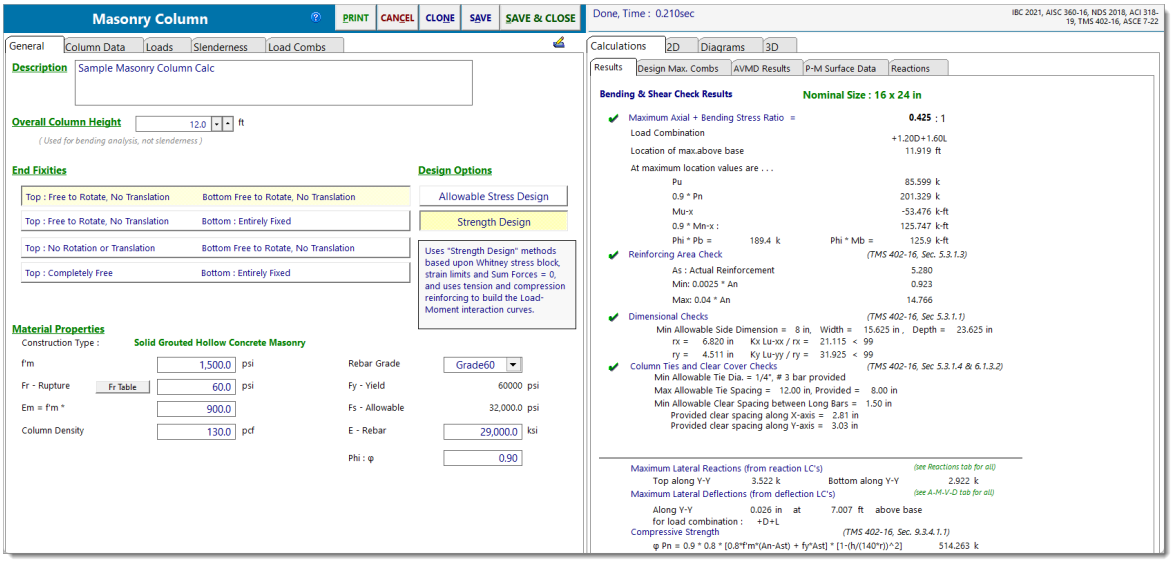
Axial reactions are reported as positive when they are upward.

Moment Reactions follow the right-hand rule explained above in Applied Moment Sign Convention

© 2026 ENERCALC, LLC

13.5.5 Masonry Column

This module designs masonry columns that are subject to axial loads and lateral bending loads about one axis.



The user can select ASD or LRFD methods.

All calculations are according to the ACI 530 or TMS 402, depending upon the selected design standard.

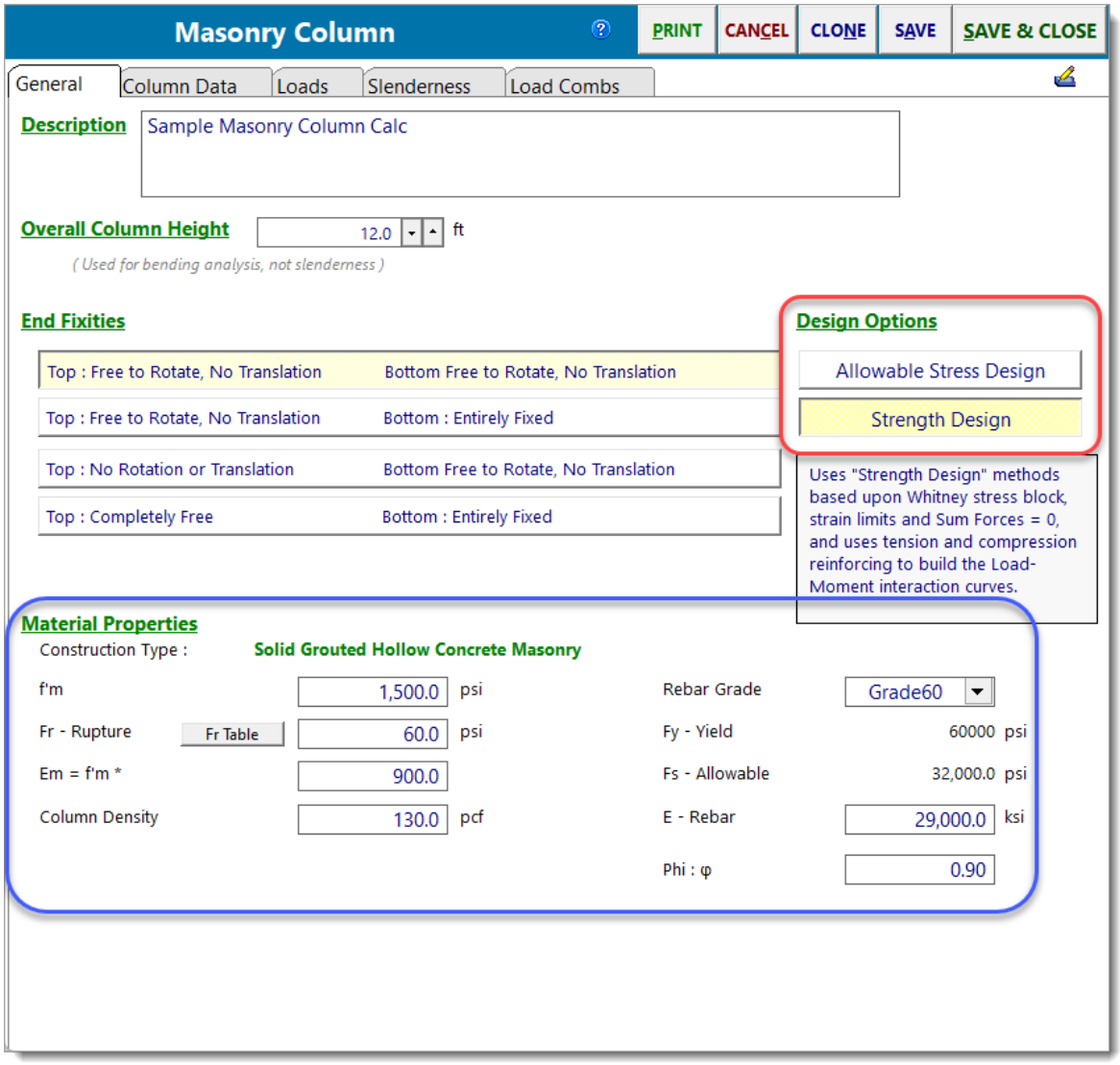
The screen capture below shows the full screen for masonry column design. See items below for descriptions of items that are specific to the masonry column design module.

For general description of the module, end fixity, loads, and load combinations click here 810. For slenderness description click here 821.

© 2026 ENERCALC, LLC

General

The area shown in the screen capture below is specific to the masonry column selection. You have the choice of using Working Stress or Strength Design methods.



The design method is selected with the toggle shown above in the red bubble. Material properties are collected in the area shown in the blue bubble above.

The column capacity is determined by creating a P-M interaction diagram, so that the effect of compressive force is included in the calculation of allowable moment capacity. For working stress this will result in significantly higher capacities than the older methods that calculated an actual stress ratio using (fa/Fa + fb/Fb).

© 2026 ENERCALC, LLC

Column Data

All of the information on this tab is specific to masonry column design.

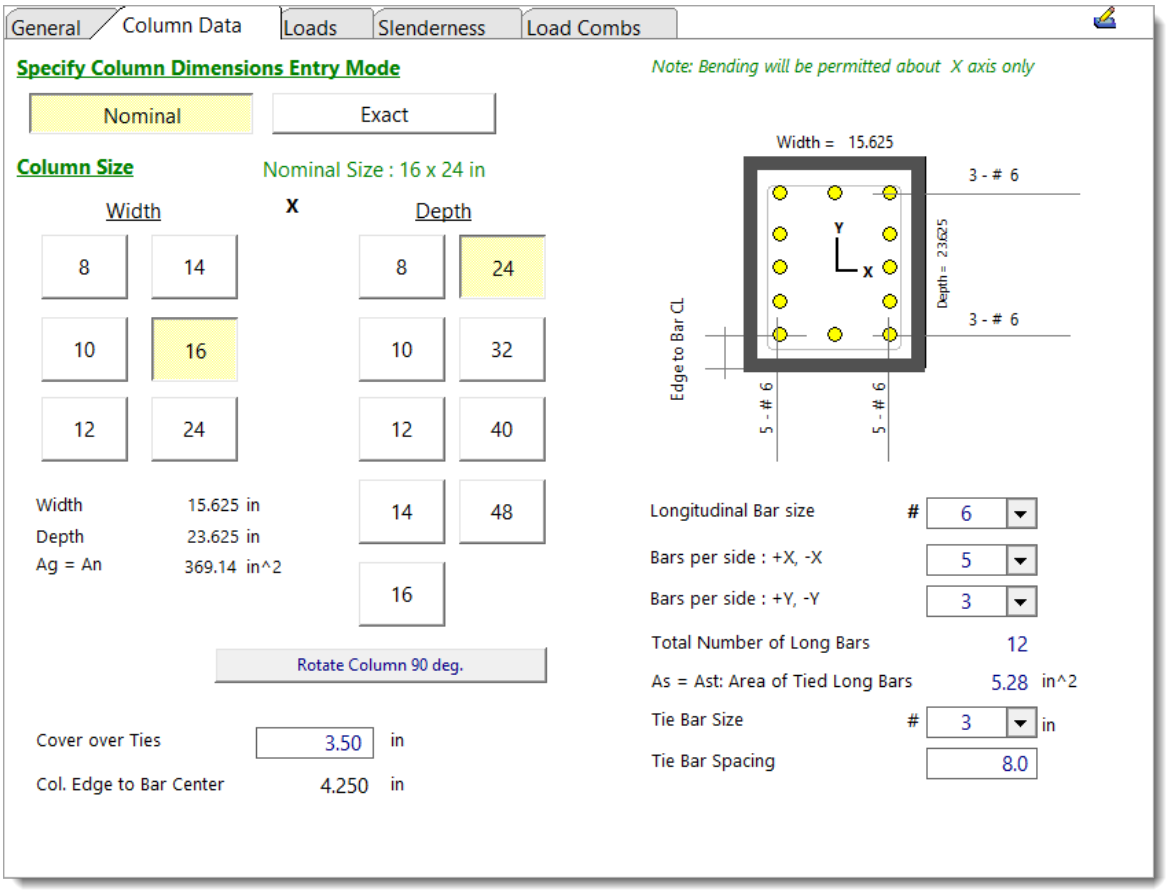
This tab collects the cross section size, reinforcing, and orientation of the column.

Section definition is made easy by buttons for the common nominal dimensions of a masonry column. Click the Width and Depth buttons and you will see the actual dimension appear in the bottom of the area. Note that "Width" is parallel to the "x-x" axis and "Depth" is parallel to the "y-y" axis.

To define the longitudinal reinforcement, use the "Bar Size" and "Bars per Side" drop-down menus. Bars on the "+X, -X" faces are placed parallel to the depth of the column (left and right sides in the cross-sectional view). Bars on the "+Y, -Y" faces are placed parallel to the width of the column (top and bottom sides in the cross-sectional view). The schematic graphic will change accordingly as your input changes. You can also use the 2D Sketch to see a scaled cross-sectional view of the column and to help verify that the reinforcement is defined correctly.

The calculation of the total area of laterally tied longitudinal reinforcing steel (A_{st}) assumes that the lateral tie detailing meets the requirements of TMS 402 §5.3.4.4(c), and therefore all defined bars will be included in the A_{st} calculation. Users must ensure that their column tie detailing aligns with these assumptions and/or confirm that this module is suitable for their specific design requirements. If the clear spacing between bars exceeds 6 inches, a warning will appear in the Results tab, reminding users to check their column tie detailing for compliance.

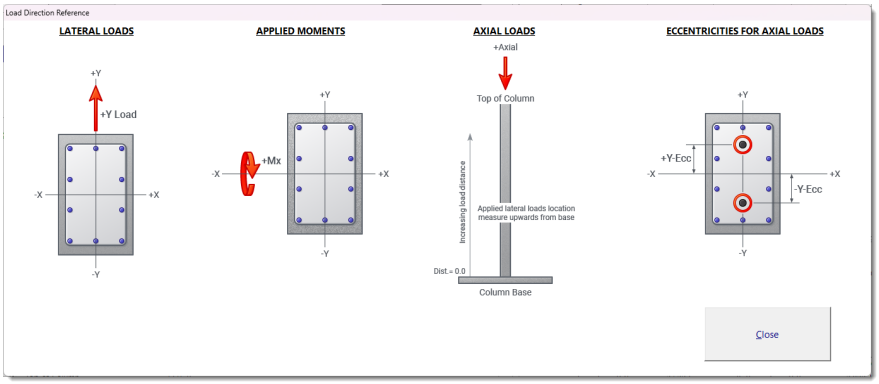
© 2026 ENERCALC, LLC



© 2026 ENERCALC, LLC

Loads

Load Reference Guide: Clicking this button opens a pop-up window that displays the positive sign conventions used in the module.



For more information on the Loads tab, please refer to the Columns 810 topic.

Results

Calculations | 2D | Diagrams | 3D

Results | Design Max. Combs | AVMD Results | P-M Surface Data | Reactions

Bending & Shear Check Results **Nominal Size : 16 x 24 in**

✔ Maximum Axial + Bending Stress Ratio = **0.425** : 1

Load Combination +1.20D+1.60L

Location of max.above base 11.919 ft

At maximum location values are . . .

Pu 85.599 k

0.9 * Pn 201.329 k

Mu-x -53.476 k-ft

0.9 * Mn-x : 125.747 k-ft

Phi * Pb = 189.4 k Phi * Mb = 125.9 k-ft

✔ Reinforcing Area Check *(TMS 402-16, Sec. 5.3.1.3)*

As : Actual Reinforcement 5.280

Min: 0.0025 * An 0.923

Max: 0.04 * An 14.766

✔ Dimensional Checks *(TMS 402-16, Sec 5.3.1.1)*

Min Allowable Side Dimension = 8 in, Width = 15.625 in , Depth = 23.625 in

rx = 6.820 in Kx Lu-xx / rx = 21.115 < 99

ry = 4.511 in Ky Lu-yy / ry = 31.925 < 99

✔ Column Ties and Clear Cover Checks *(TMS 402-16, Sec 5.3.1.4 & 6.1.3.2)*

Min Allowable Tie Dia. = 1/4", # 3 bar provided

Max Allowable Tie Spacing = 12.00 in, Provided = 8.00 in

Min Allowable Clear Spacing between Long Bars = 1.50 in

Provided clear spacing along X-axis = 2.81 in

Provided clear spacing along Y-axis = 3.03 in

Maximum Lateral Reactions (from reaction LC's) *(see Reactions tab for all)*

Top along Y-Y 3.522 k Bottom along Y-Y 2.922 k

Maximum Lateral Deflections (from deflection LC's) *(see A-M-V-D tab for all)*

Along Y-Y 0.026 in at 7.007 ft above base

for load combination : +D+L

Compressive Strength *(TMS 402-16, Sec. 9.3.4.1.1)*

φ Pn = 0.9 * 0.8 * [0.8*f'm*(An-Ast) + fy*Ast] * [1-(h/(140*r))^2] 514.263 k

Design Maximum Combinations

This tab summarizes the maximum stress ratios for each load combination.

Calculations | 2D | Diagrams | 3D

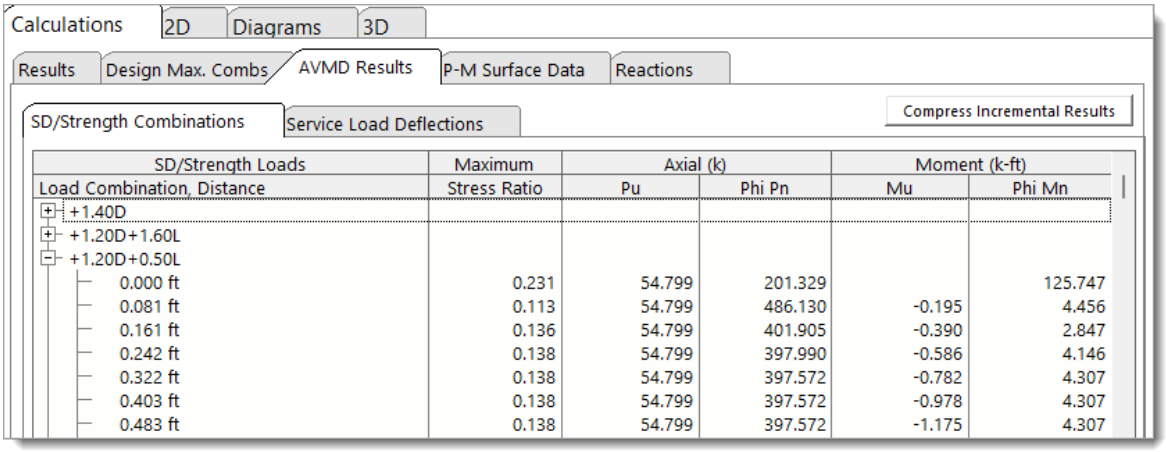
Results | Design Max. Combs | AVMD Results | P-M Surface Data | Reactions

Load Combination	Max. Bending Ratios			Axial Values (k)		Moment Values (k-ft)	
Name	Stress Ratio	Status	Location (ft)	Actual	Allowable	Actual	Allowable
+1.40D	0.2214	PASS	11.92	47.599	215.049	-27.778	125.356
+1.20D+1.60L	0.4252	PASS	11.92	85.599	201.329	-53.476	125.747
+1.20D+0.50L	0.2631	PASS	11.92	54.799	208.386	-33.081	125.639
+1.20D	0.1898	PASS	11.92	40.799	215.049	-23.810	125.356
+0.90D	0.1423	PASS	11.92	30.599	215.049	-17.858	125.356

© 2026 ENERCALC, LLC

AVMD Results: SD/Strength Combinations

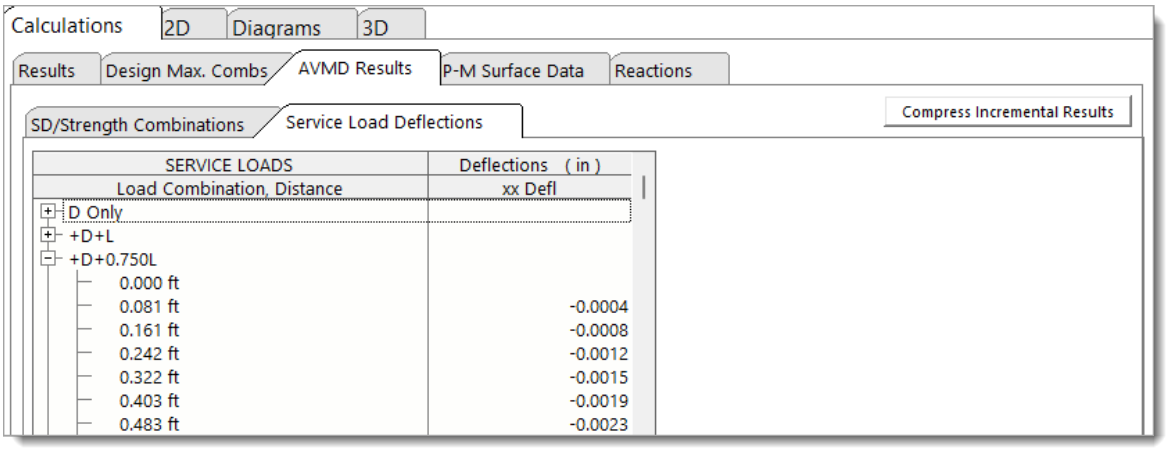
This tab presents the very detailed allowable and actual values for each load combination.



SD/Strength Loads Load Combination, Distance	Maximum Stress Ratio	Axial (k) Pu	 Phi Pn	Moment (k-ft) Mu	 Phi Mn
+1.40D					
+1.20D+1.60L					
+1.20D+0.50L					
0.000 ft	0.231	54.799	201.329		125.747
0.081 ft	0.113	54.799	486.130	-0.195	4.456
0.161 ft	0.136	54.799	401.905	-0.390	2.847
0.242 ft	0.138	54.799	397.990	-0.586	4.146
0.322 ft	0.138	54.799	397.572	-0.782	4.307
0.403 ft	0.138	54.799	397.572	-0.978	4.307
0.483 ft	0.138	54.799	397.572	-1.175	4.307

AVMD Results: Service Load Deflections

This tab summarizes the lateral deflections of the column at increments along its height. These values will be nonzero only if lateral loads are applied or the axial load is applied with an eccentricity.

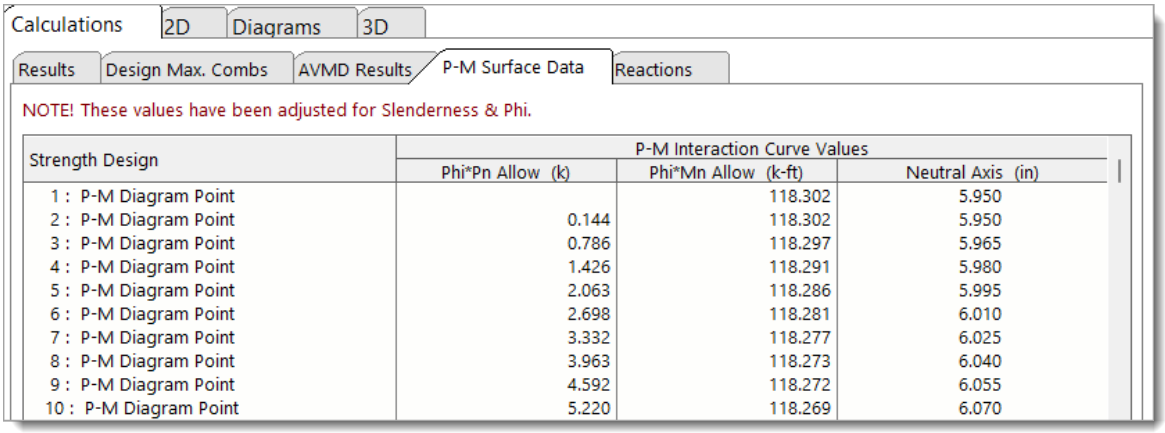


SERVICE LOADS Load Combination, Distance	Deflections (in) xx Defl
D Only	
+D+L	
+D+0.750L	
0.000 ft	
0.081 ft	-0.0004
0.161 ft	-0.0008
0.242 ft	-0.0012
0.322 ft	-0.0015
0.403 ft	-0.0019
0.483 ft	-0.0023

© 2026 ENERCALC, LLC

P-M Surface Data

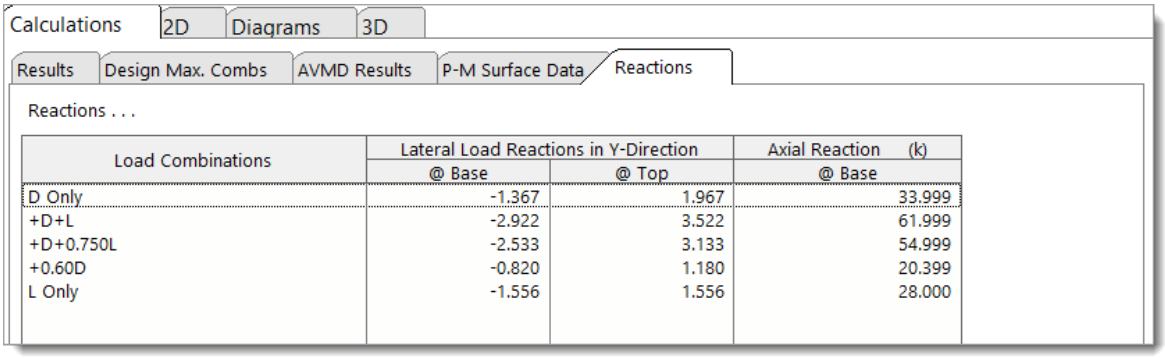
This tab lists the full analysis results for the column.



Strength Design	P-M Interaction Curve Values		
	Phi*Pn Allow (k)	Phi*Mn Allow (k-ft)	Neutral Axis (in)
1 : P-M Diagram Point		118.302	5.950
2 : P-M Diagram Point	0.144	118.302	5.950
3 : P-M Diagram Point	0.786	118.297	5.965
4 : P-M Diagram Point	1.426	118.291	5.980
5 : P-M Diagram Point	2.063	118.286	5.995
6 : P-M Diagram Point	2.698	118.281	6.010
7 : P-M Diagram Point	3.332	118.277	6.025
8 : P-M Diagram Point	3.963	118.273	6.040
9 : P-M Diagram Point	4.592	118.272	6.055
10 : P-M Diagram Point	5.220	118.269	6.070

Reactions

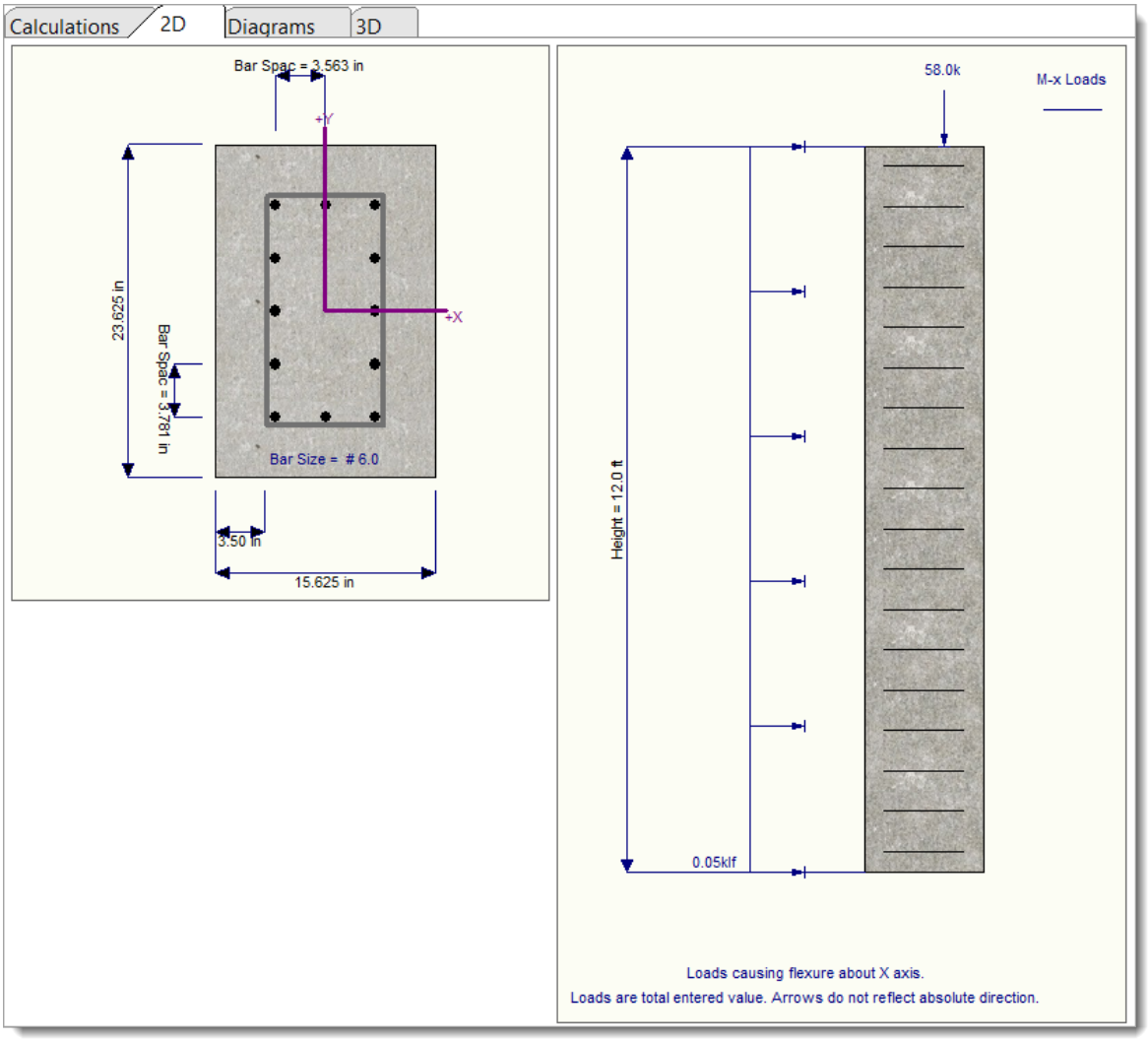
This tab reports axial reactions and lateral reactions.



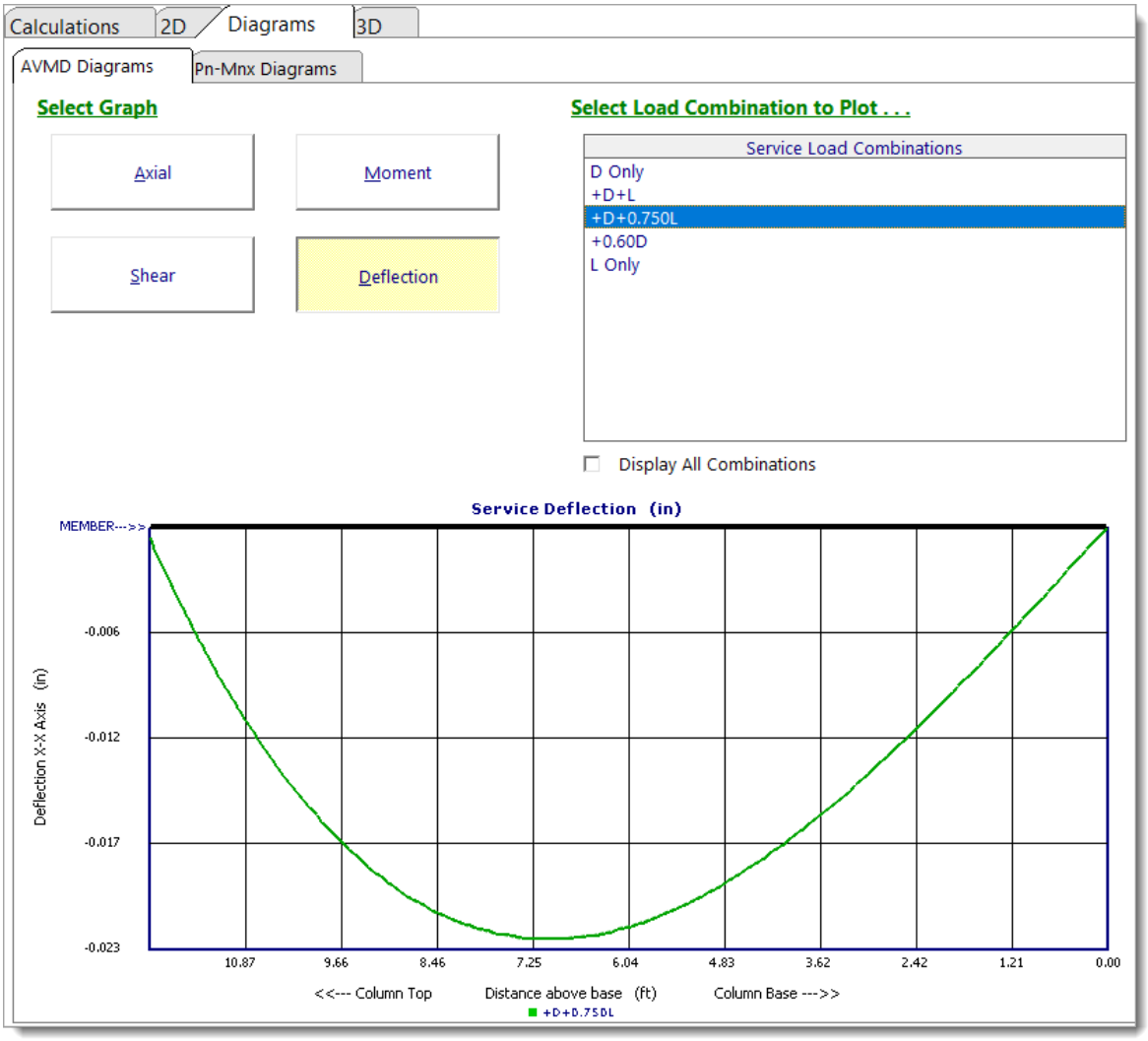
Load Combinations	Lateral Load Reactions in Y-Direction		Axial Reaction (k)
	@ Base	@ Top	@ Base
D Only	-1.367	1.967	33.999
+D+L	-2.922	3.522	61.999
+D+0.750L	-2.533	3.133	54.999
+0.60D	-0.820	1.180	20.399
L Only	-1.556	1.556	28.000

© 2026 ENERCALC, LLC

2D Sketch

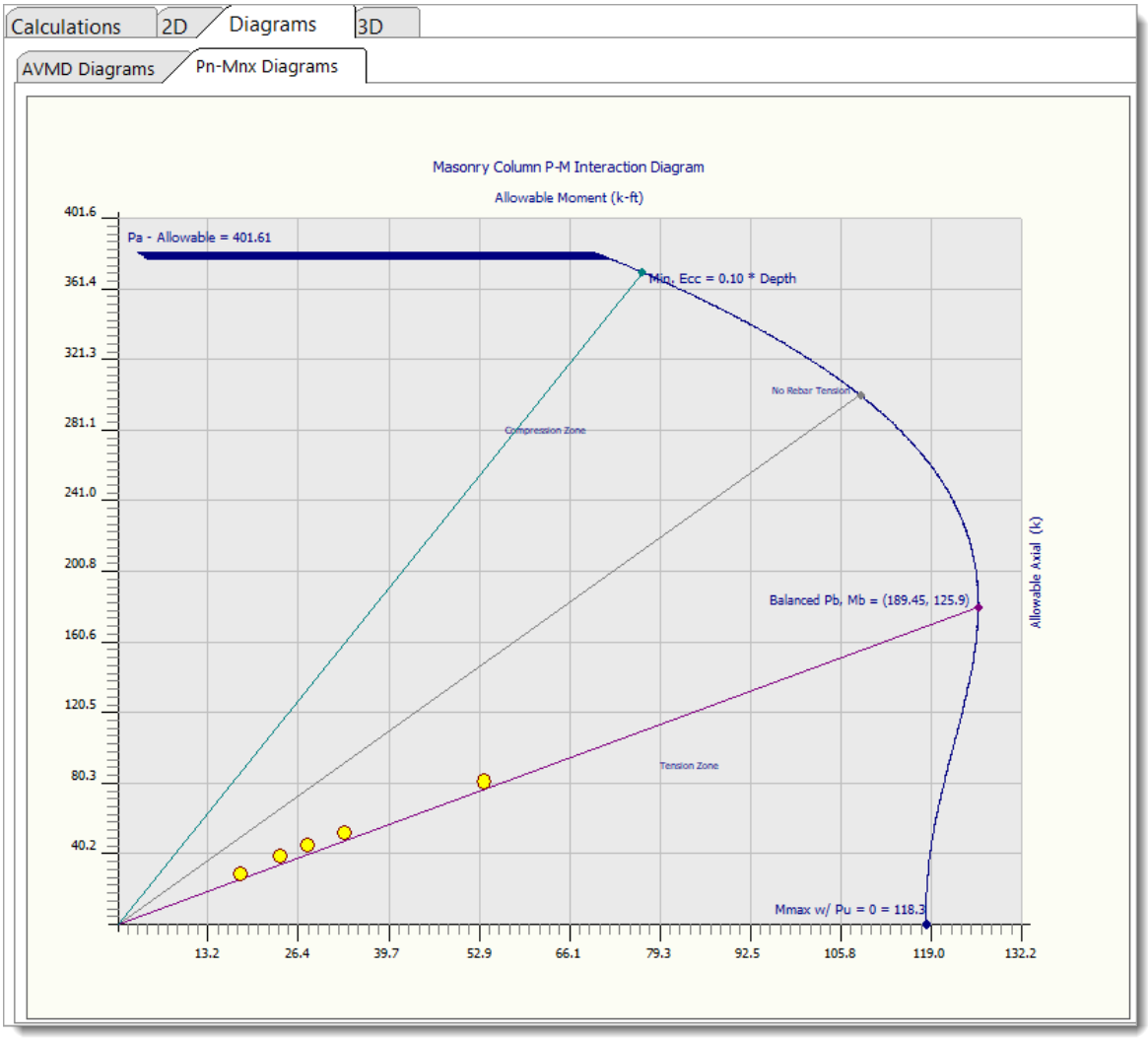

Calculations
2D
Diagrams
3D
Bar Spac = 3.563 in
+Y
+X
23.625 in
Bar Spac = 3.781 in
Bar Size = # 6.0
3.50 in
15.625 in
58.0k
M-x Loads
Height = 12.0 ft
0.05klf
Loads causing flexure about X axis.
Loads are total entered value. Arrows do not reflect absolute direction.

AVMD Diagrams



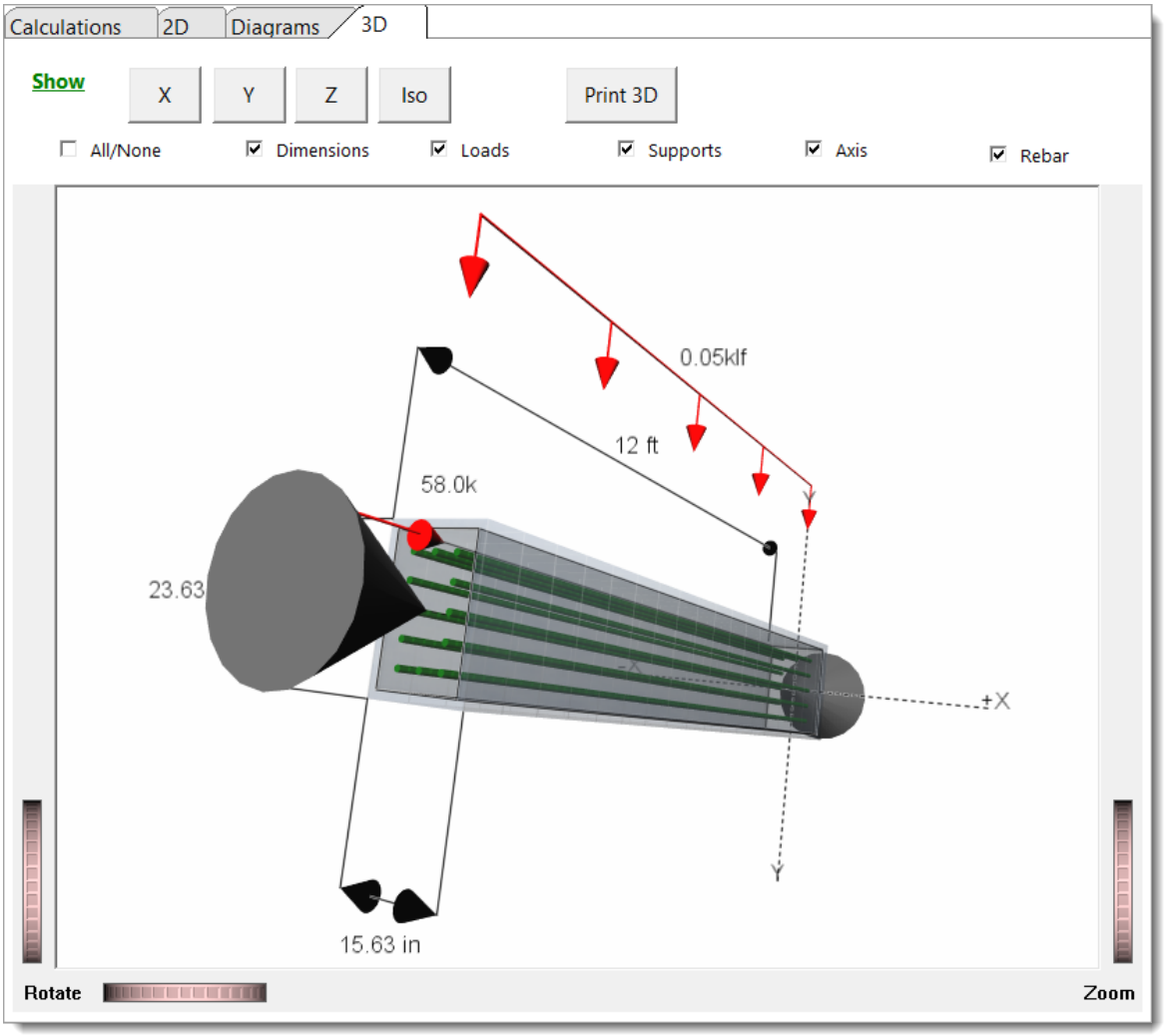
© 2026 ENERCALC, LLC

Pn-Mnx Diagram



© 2026 ENERCALC, LLC

3D Rendering



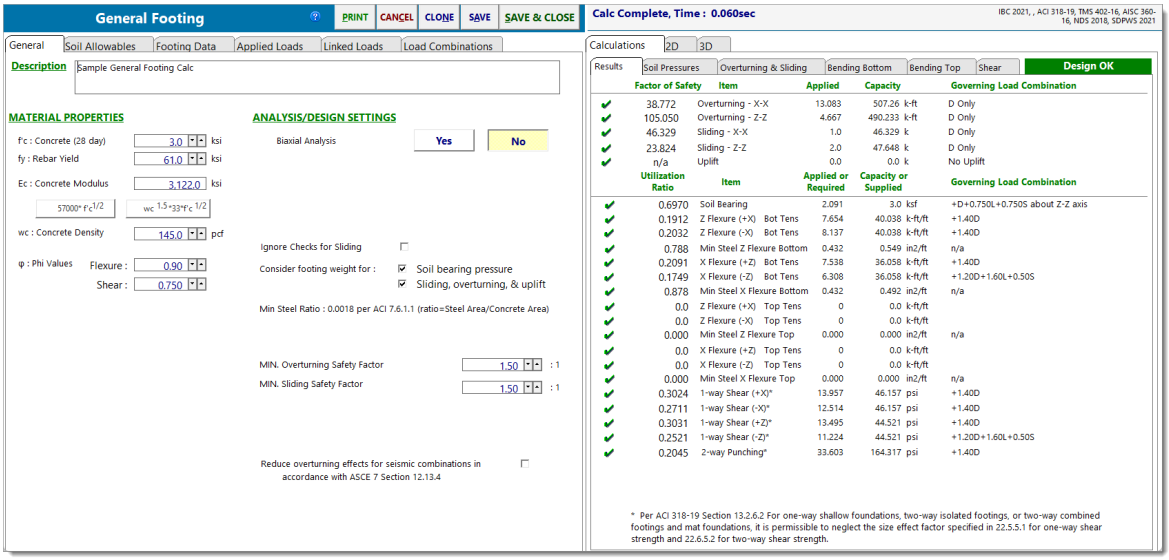
13.6 Foundations

Please select a subtopic.

© 2026 ENERCALC, LLC

13.6.1 General Footing

This module provides analysis of a rectangular footing with applied axial load, overburden, moment and shear loads.

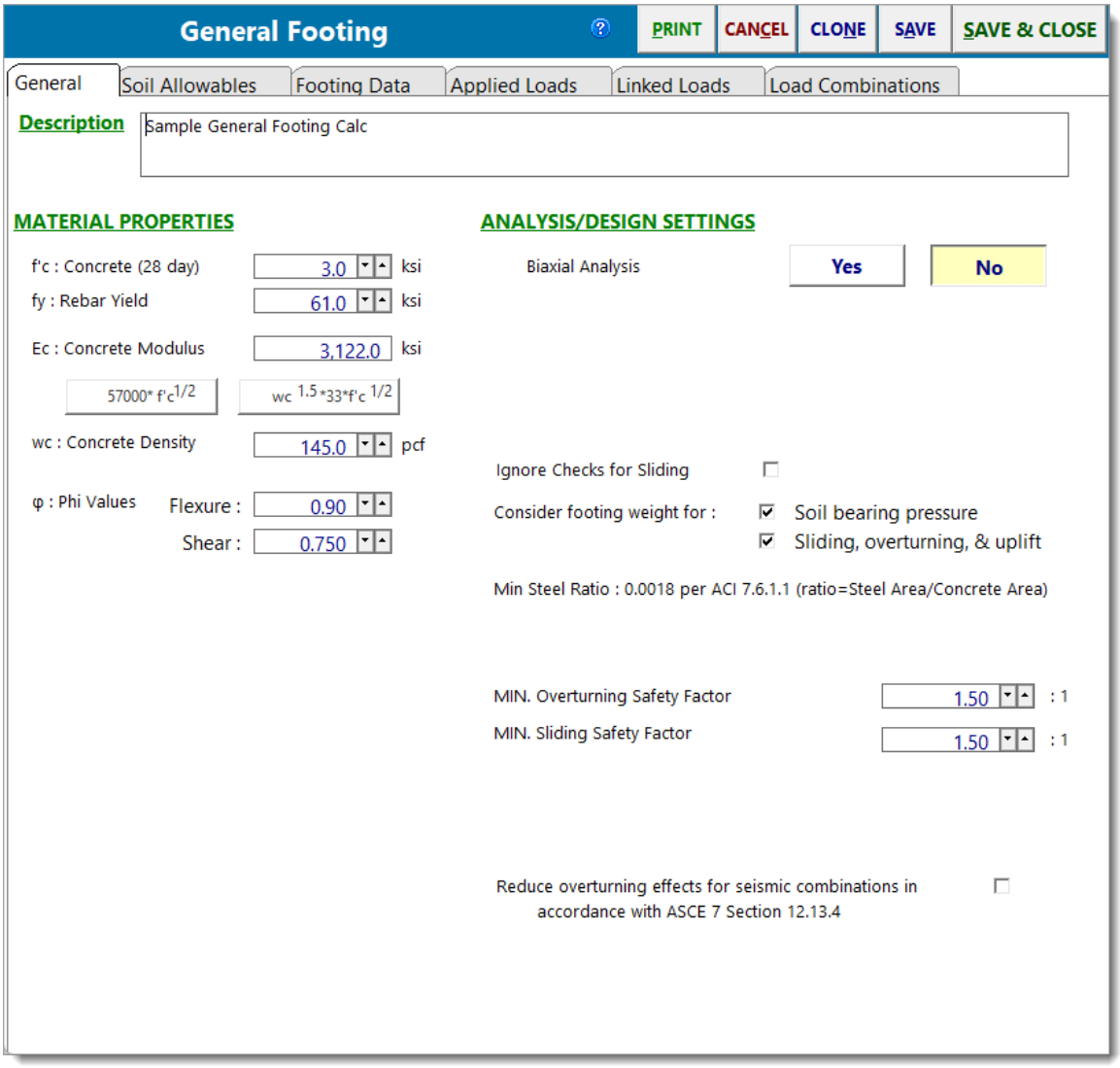


The module allows you to move the axial load application position off-center of the footing, and provides automatic calculation of allowable soil bearing pressure increases based on footing dimensions and/or depth below surface.

The module checks service load soil pressure, overturning stability, sliding stability, flexure at each of the four pedestal faces, 1-way shear at 'd' from each of the four pedestal faces, and punching shear along a perimeter located 'd/2' from the pedestal faces.

© 2026 ENERCALC, LLC

General



Click to Calculate (Button is only visible when biaxial analysis is selected)

Due to the iterative nature of the calculations that are required for a biaxial analysis, it would be undesirable to rerun the entire analysis and design every time an input parameter changes. So for reasons of efficiency, the program automatically goes into manual recalculation mode when biaxial analysis is selected. Click this button any time you wish to recalculate with the current input parameters.

f'c

28-day compressive strength of the concrete.

fy

Yield point stress of reinforcing.

© 2026 ENERCALC, LLC

Ec

Modulus of elasticity of concrete.

Concrete Density

The density of the concrete is used to calculate the self-weight of the pedestal and footing when that option is selected. Note that code modifications for lightweight concrete are not applied in this module. The purpose of this input is mainly to allow the user to specify something in the range of 145 to 150 pcf.

Phi Values

Enter the capacity reduction values to be applied to Vn and Mn.

Biaxial Analysis

Select Yes or No to indicate if a biaxial analysis is to be performed. If a biaxial analysis is performed, the solution will consider moments applied simultaneously about the two orthogonal axes of the footing. If a biaxial analysis is NOT performed, the solution will consider moments applied about the two orthogonal axes to be acting non-concurrently.

Amount of Edge Length for M & V (Only displayed when biaxial analysis is selected)

When calculating shear and moment for footings where the maximum soil pressure values occur at the corners, this value specifies the fraction (as a decimal) of the footing dimension from the edge to use when calculating moments and shears due to variable soil pressure in that region. A smaller value for this variable will produce a more conservative design, because it will focus on a narrower strip which is experiencing the highest soil bearing pressure.

Ignore Checks for Sliding

Select this option if sliding is not a design consideration for any particular reason.

Consider footing weight when determining soil bearing pressure

Select this option to have the module calculate the self-weight of the footing and apply it as a downward load when determining soil bearing pressures. The self-weight will be multiplied by the dead load factor in each of the Soil Bearing Pressure load combinations.

Note: This option should generally be selected. Deselecting this option can lead to incorrect soil bearing pressure calculations in footings with moment. If the goal is to try to compare soil bearing pressures to net allowable pressures, then it would be advisable to use the option on the Soil Allowables tab to "Increase Bearing by Footing Weight" or adjust the allowable bearing pressure manually if the design parameters warrant further adjustment.

© 2026 ENERCALC, LLC

Consider footing weight when determining sliding, overturning and uplift

Select this option to have the module calculate the self-weight of the footing and apply it as a downward load when determining sliding, overturning and uplift factors of safety. The self-weight will be multiplied by the dead load factor in each of the Stability load combinations.

Min Steel Ratio

This is set to 0.0018 per ACI 7.6.1.1. A warning message will be triggered if the section is under-reinforced.

NOTE: This is not T&S steel. It is a minimum that must be met for flexural steel. This means that if the footing has net uplift and requires a top mat of reinforcing, this ratio also applies to the area of steel provided in the top mat. It is not something that can be "shared" between the top and bottom mats like T&S steel can.

Minimum overturning safety factor

Enter the minimum allowable ratio of resisting moment to overturning moment. If the actual ratio is less than the specified minimum ratio, it will trigger a message that overturning stability is not satisfied.

Minimum sliding safety factor

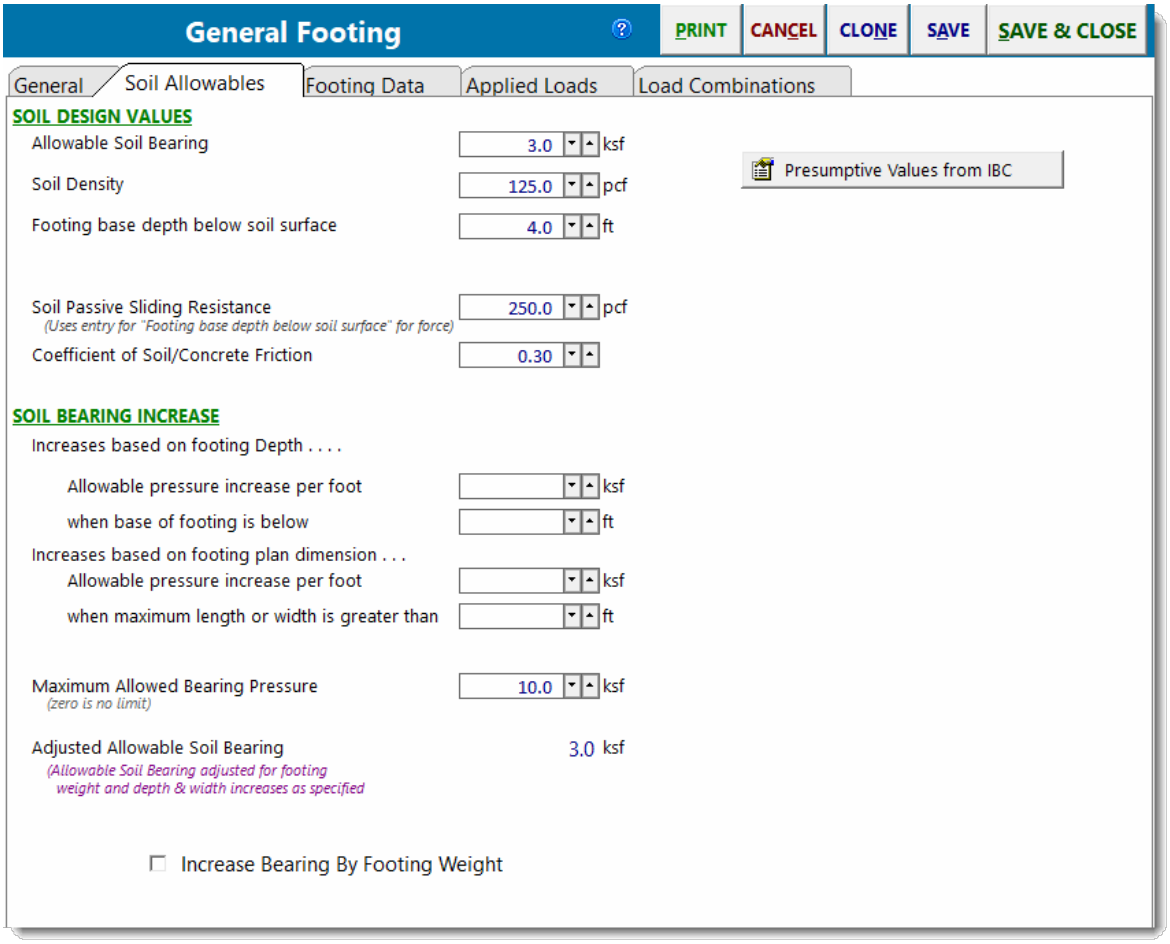
Enter the minimum allowable ratio of resisting force to sliding force. If the actual ratio is less than the specified minimum ratio, it will trigger a message that sliding stability is not satisfied.

Reduce overturning effects for seismic combinations in accordance with ASCE 7 Section 12.13.4

Select this option to apply the provisions of the referenced section of ASCE 7.

© 2026 ENERCALC, LLC

Soil Allowable Values



Allowable Soil Bearing

Enter the allowable soil bearing pressure that the soil can resist. This is a service load resistance and will be compared to calculated service load soil pressures (loads not factored as in strength design).

Soil density

Enter the density of the soil in pcf.

Footing base depth below soil surface

The distance from the bottom of the footing to the top of the soil. This value is used to determine allowable soil bearing pressure increases and soil passive sliding resistance, but it is not used in any other calculations in this module.

© 2026 ENERCALC, LLC

Increase bearing by footing weight

Click [**Yes**] to tell the module to calculate the weight of one square foot (plan view) of footing weight and add it to the allowable soil bearing value. This has the effect of not penalizing the soil for the self weight of the footing, and is useful for situations where the geotechnical engineering report provides allowable net bearing pressures.

Soil passive sliding resistance

Enter the value of passive soil pressure resistance to sliding. This value will be used to determine a component of sliding resistance that is generated by the passive pressure of the soil. The sliding resistance due to passive pressure is then added to the sliding resistance due to friction to determine the total resistance to sliding for each load combination.

Coefficient of Soil/Concrete Friction

Enter the coefficient of friction between soil and footing to use in sliding resistance calculations.

Soil Bearing Increase

This section allows you to specify some dimensions that, when exceeded, will automatically increase the allowable soil bearing pressure.

Increases based on footing depth: Provides a method to automatically apply increases to the basic allowable soil bearing pressure based on footing depth below some reference depth. Collects the following parameters:

Allowable pressure increase per foot: Specifies the amount that the basic allowable soil bearing pressure can be increased for each foot of depth below some reference depth.

When base of footing is below: Specifies the required depth in order to start realizing incremental increases in the allowable soil bearing pressure on the basis of footing depth.

Example: Assume the following: Basic Allowable Soil Bearing Pressure = 3 ksf. Footing base is 6'-0" below soil surface. The Geotechnical report specifies that a 0.15 ksf increase in bearing pressure is allowed for each foot of depth when the base is deeper than 4' below top of soil. Since you've indicated that the footing is 6' below the soil surface, the module will automatically calculate the adjusted allowable soil bearing pressure to be 3 ksf + (6' - 4') * 0.15 ksf = 3.30 ksf.

© 2026 ENERCALC, LLC

Increases based on footing plan dimension: Provides a method to automatically apply increases to the basic allowable soil bearing pressure based on footing dimensions greater than some reference dimension. Collects the following parameters:

Allowable pressure increase per foot: Specifies the amount that the basic allowable soil bearing pressure can be increased for each foot of length or width greater than some reference dimension.

When maximum length or width is greater than: Specifies the required dimension in order to start realizing incremental increases in the allowable soil bearing pressure on the basis of footing dimension.

Example: Assume the following: Basic Allowable Soil Bearing Pressure = 3 ksf. Footing measures 12'-0" x 6'-0". The geotechnical report specifies that a 0.15 ksf increase in soil bearing pressure is allowed for each foot when the largest plan dimension of the footing is greater than 4'. The module will automatically calculate the adjusted allowable soil bearing pressure to be 3 ksf + (12' - 4') * 0.15 ksf = 4.2 ksf.

Note: Increases based on footing depth and plan dimensions are cumulative.

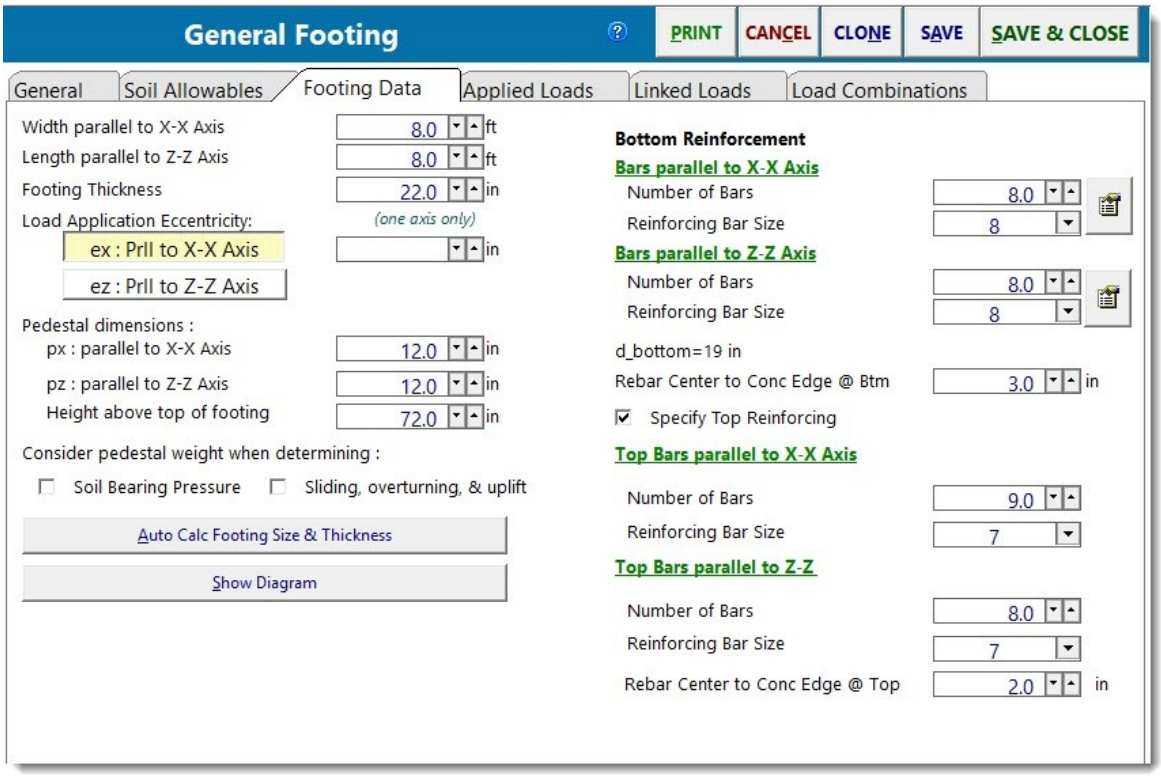
Maximum Allowed Bearing Pressure: Allows the specification of an upper limit on the soil bearing pressure that cannot be exceeded, regardless of the above increases.

Adjusted Soil Bearing Pressure: Reports the allowable soil bearing pressure adjusted for footing weight and depth and dimensions as specified.

© 2026 ENERCALC, LLC

Footing Data

This tab is where you enter the footing and pedestal dimensions.



Width, Length & Thickness: defines the overall dimensions of the footing.

Load Location: defines the offset from the center of the footing where the axial load is applied. If a biaxial analysis is NOT being used, then only one direction can be used.

Pedestal dimensions: If a concrete pedestal bears on the footing, its dimensions can be specified here. The px and pz dimensions are used to define the locations on all four sides where one-way shear, two-way shear and bending moment are calculated. If you enter a nonzero height, then you can choose to have the weight of that prism calculated and added as dead load. Any applied overburden loads will be omitted from the area defined as the pedestal dimension along the xx and yy axis, regardless of the specified height of the prism.

Note: If no pedestal is defined, then the load location will be treated as the face of the pedestal when determining the critical locations to check shear and flexure.

Bottom Reinforcement: (Always present)

Rebar Definitions: Specify the number and size of bottom rebar parallel to each axis.

© 2026 ENERCALC, LLC

Rebar Location: Specify the distance from the center of the bottom rebar to the bottom surface of the footing. Note that this is the dimension to the center of the rebar, not the clear cover.

Specify Top Reinforcement: Provides a checkbox to explicitly state whether top rebar is provided or not.

Rebar Definitions: Specify the number and size of top rebar parallel to each axis.

Rebar Location: Specify the distance from the center of the top rebar to the bottom surface of the footing. Note that this is the dimension to the center of the rebar, not the clear cover.

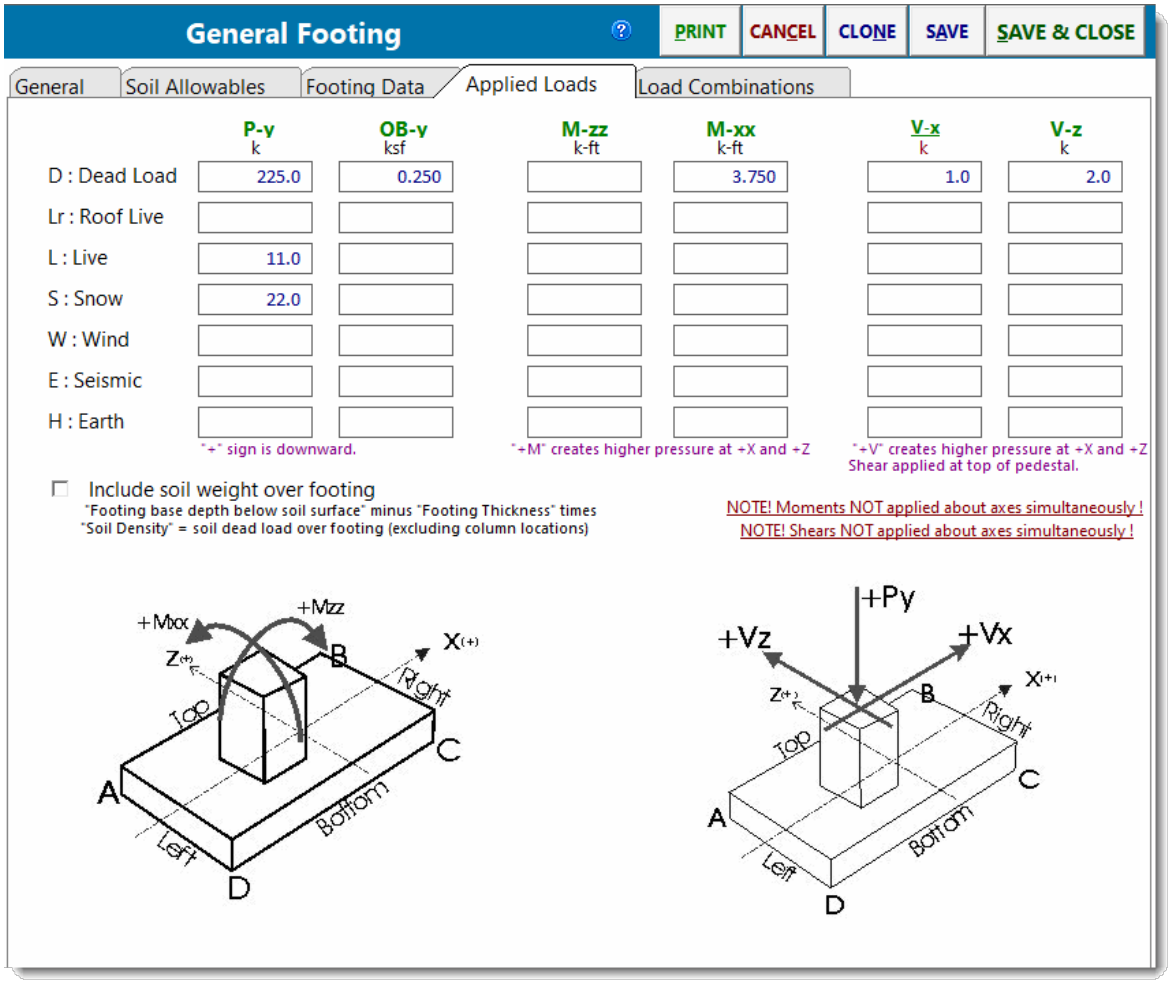
Note: Bars are assumed to be fully developed at the locations where they are required. It is the engineer's responsibility to validate that assumption. The program is not taking rebar development length into consideration.

Consider Pedestal Weight When Determining: This option allows the user to specify whether or not the self-weight of the pedestal is to be considered when determining the soil bearing pressure, and separately, whether or not the self-weight of the pedestal is to be considered when performing the checks for sliding, overturning, and uplift.

© 2026 ENERCALC, LLC

Applied Loads

This tab allows you to specify the axial load, shear, and moment applied to the pedestal, as well as an overburden load applied to the entire plan dimension of the footing (except the area designated as the pedestal).



Enter vertical loads with a positive sign for downward direction. Shear loads are applied at the location of the pedestal. If the pedestal is specified to have a height, the shear will be applied at that height and will create a moment on the footing equal to Shear Load * (Footing Thickness + Pedestal Height).

Note! This module will not allow a net uplift on the footing. If the result of the factored axial loads (dead, live, wind, etc) produces a negative load sign, the module will not recalculate and will notify you of which load combination resulted in net uplift.

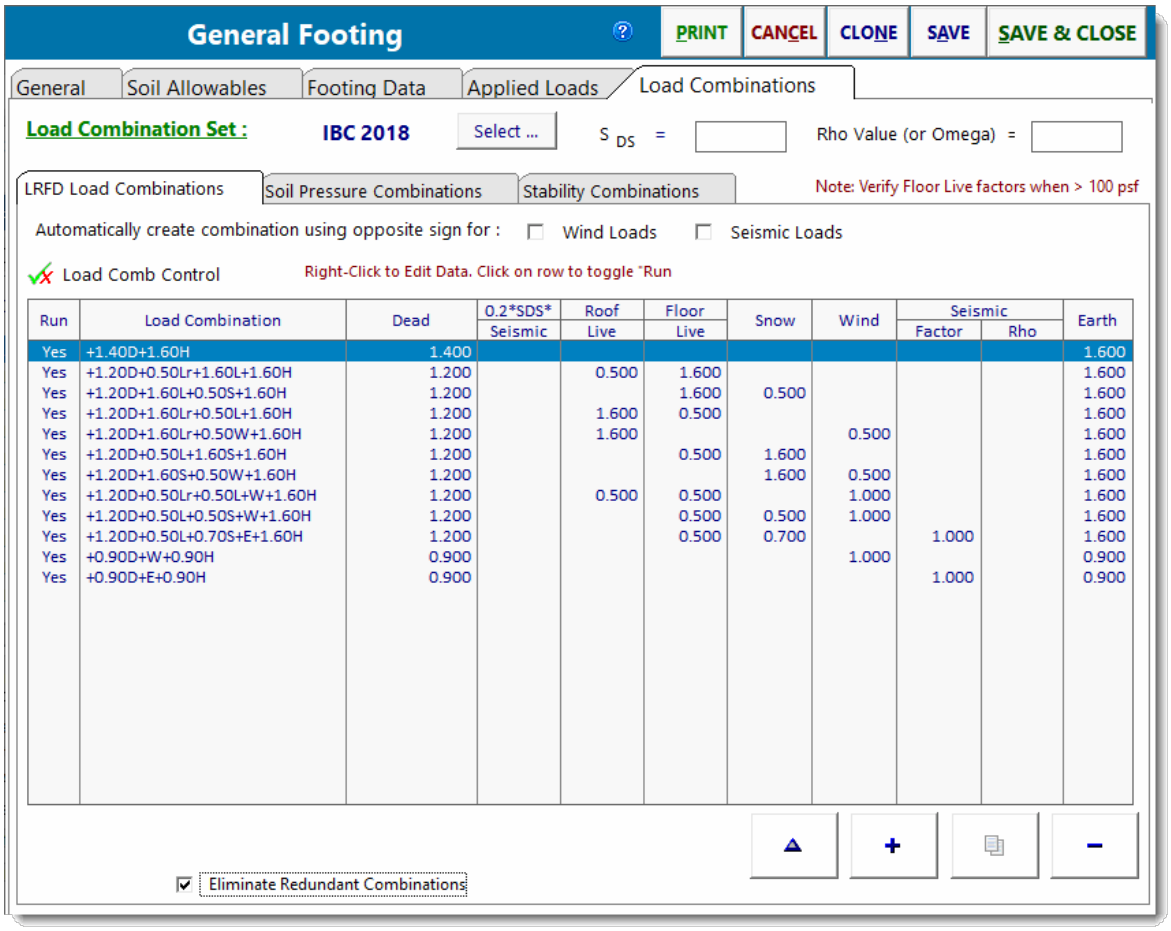
© 2026 ENERCALC, LLC

Linked Loads

See Linked Loads Overview 192.

Load Combinations

The Load Combination tab offers three subtabs: LRFD Load Combinations, Soil Pressure Combinations, and Stability Combinations. They are all based on the selected Load Combination Set.



Run	Load Combination	Dead	0.2*SDS* Seismic	Roof Live	Floor Live	Snow	Wind	Seismic Factor	Seismic Rho	Earth
Yes	+1.40D+1.60H	1.400								1.600
Yes	+1.20D+0.50Lr+1.60L+1.60H	1.200		0.500	1.600					1.600
Yes	+1.20D+1.60L+0.50S+1.60H	1.200			1.600	0.500				1.600
Yes	+1.20D+1.60Lr+0.50L+1.60H	1.200		1.600	0.500					1.600
Yes	+1.20D+1.60Lr+0.50W+1.60H	1.200		1.600			0.500			1.600
Yes	+1.20D+0.50L+1.60S+1.60H	1.200			0.500	1.600				1.600
Yes	+1.20D+1.60S+0.50W+1.60H	1.200				1.600	0.500			1.600
Yes	+1.20D+0.50Lr+0.50L+W+1.60H	1.200		0.500	0.500		1.000			1.600
Yes	+1.20D+0.50L+0.50S+W+1.60H	1.200			0.500	0.500	1.000			1.600
Yes	+1.20D+0.50L+0.70S+E+1.60H	1.200			0.500	0.700		1.000		1.600
Yes	+0.90D+W+0.90H	0.900					1.000			0.900
Yes	+0.90D+E+0.90H	0.900						1.000		0.900

The LRFD load combinations are used to calculate moments and shears in the footing for use in determining stresses and required reinforcing. The combinations on this tab are at the strength-level.

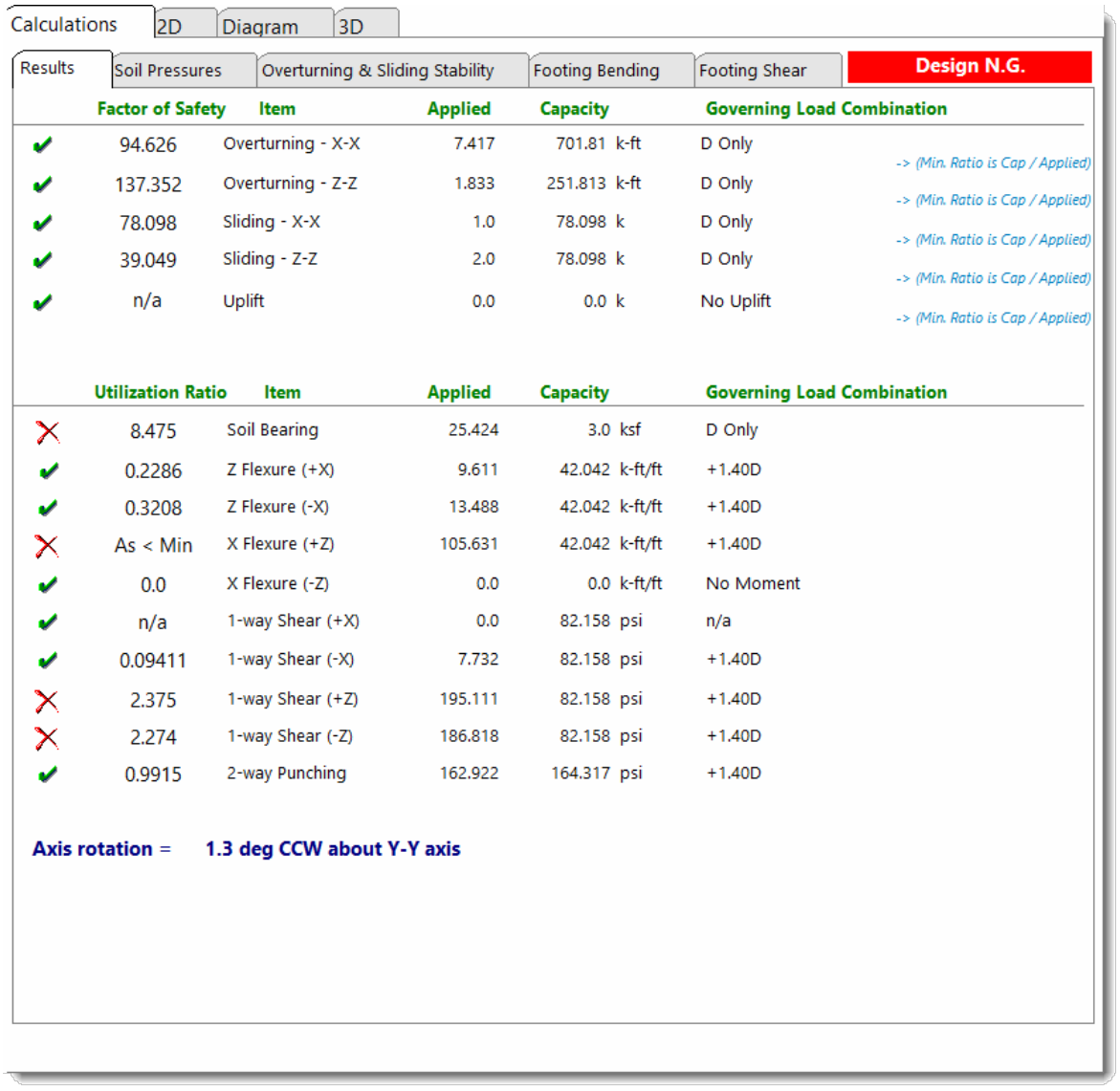
The combinations on the other two tabs are at the service-level, and are used for checking soil bearing pressure and stability, respectively.

© 2026 ENERCALC, LLC

Note: The General Footing module is applying the factored loads to the footing and determining a different eccentricity than was determined using the service loads for the soil bearing pressure check.

Results

This tab provides a summary of all calculated values. The stress ratios, applied & allowable values and load combination for those governing values are reported.



Calculations | 2D | Diagram | 3D

Results | Soil Pressures | Overturning & Sliding Stability | Footing Bending | Footing Shear | **Design N.G.**

	Factor of Safety	Item	Applied	Capacity	Governing Load Combination
✔	94.626	Overturning - X-X	7.417	701.81 k-ft	D Only -> (Min. Ratio is Cap / Applied)
✔	137.352	Overturning - Z-Z	1.833	251.813 k-ft	D Only -> (Min. Ratio is Cap / Applied)
✔	78.098	Sliding - X-X	1.0	78.098 k	D Only -> (Min. Ratio is Cap / Applied)
✔	39.049	Sliding - Z-Z	2.0	78.098 k	D Only -> (Min. Ratio is Cap / Applied)
✔	n/a	Uplift	0.0	0.0 k	No Uplift -> (Min. Ratio is Cap / Applied)

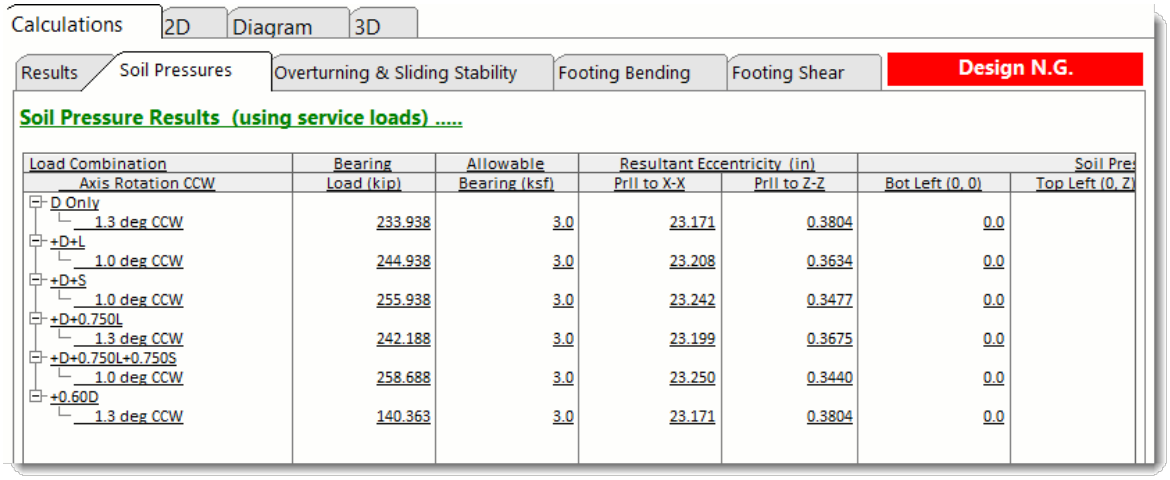
	Utilization Ratio	Item	Applied	Capacity	Governing Load Combination
✘	8.475	Soil Bearing	25.424	3.0 ksf	D Only
✔	0.2286	Z Flexure (+X)	9.611	42.042 k-ft/ft	+1.40D
✔	0.3208	Z Flexure (-X)	13.488	42.042 k-ft/ft	+1.40D
✘	As < Min	X Flexure (+Z)	105.631	42.042 k-ft/ft	+1.40D
✔	0.0	X Flexure (-Z)	0.0	0.0 k-ft/ft	No Moment
✔	n/a	1-way Shear (+X)	0.0	82.158 psi	n/a
✔	0.09411	1-way Shear (-X)	7.732	82.158 psi	+1.40D
✘	2.375	1-way Shear (+Z)	195.111	82.158 psi	+1.40D
✘	2.274	1-way Shear (-Z)	186.818	82.158 psi	+1.40D
✔	0.9915	2-way Punching	162.922	164.317 psi	+1.40D

Axis rotation = 1.3 deg CCW about Y-Y axis

© 2026 ENERCALC, LLC

Soil Pressures

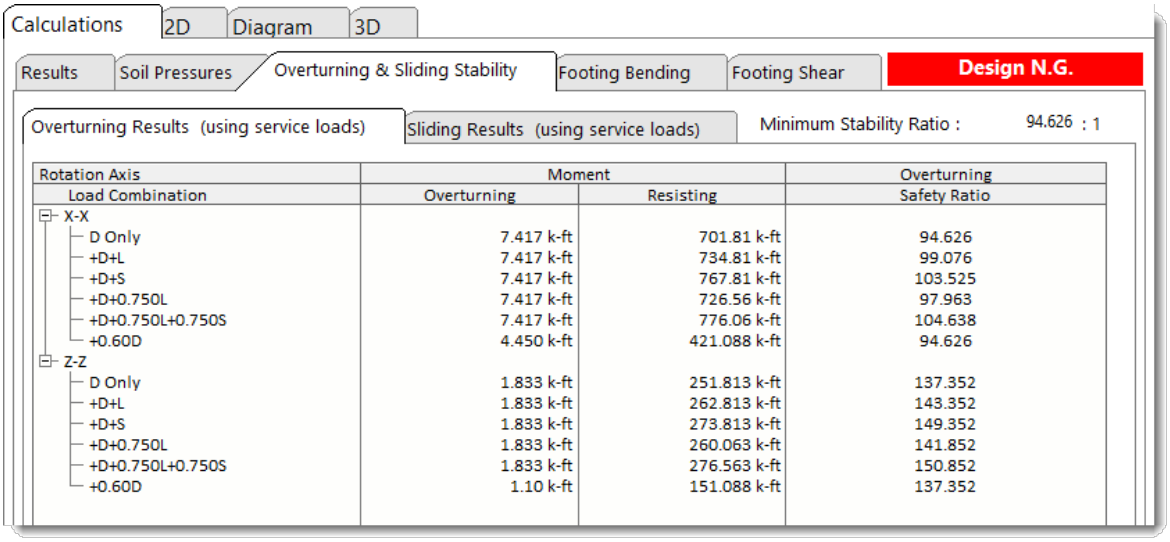
This tab summarizes the calculated service load soil bearing pressure for moments & shears applied about the specified axis, for each load combination.



Load Combination / Axis Rotation CCW	Bearing Load (kip)	Allowable Bearing (ksf)	Resultant Eccentricity (in) Prll to X-X	Prll to Z-Z	Soil Pres Bot Left (0, 0)	Top Left (0, Z)
D Only						
1.3 deg CCW	233.938	3.0	23.171	0.3804	0.0	
+D+L						
1.0 deg CCW	244.938	3.0	23.208	0.3634	0.0	
+D+S						
1.0 deg CCW	255.938	3.0	23.242	0.3477	0.0	
+D+0.750L						
1.3 deg CCW	242.188	3.0	23.199	0.3675	0.0	
+D+0.750L+0.750S						
1.0 deg CCW	258.688	3.0	23.250	0.3440	0.0	
+0.60D						
1.3 deg CCW	140.363	3.0	23.171	0.3804	0.0	

Overturning & Sliding Stability

This tab provides the calculations for overturning and resisting moment of the footing about each axis for each load combination, and sliding and resisting forces in each direction for each load combination.



Rotation Axis / Load Combination	Moment Overturning	Moment Resisting	Overturning Safety Ratio
X-X			
D Only	7.417 k-ft	701.81 k-ft	94.626
+D+L	7.417 k-ft	734.81 k-ft	99.076
+D+S	7.417 k-ft	767.81 k-ft	103.525
+D+0.750L	7.417 k-ft	726.56 k-ft	97.963
+D+0.750L+0.750S	7.417 k-ft	776.06 k-ft	104.638
+0.60D	4.450 k-ft	421.088 k-ft	94.626
Z-Z			
D Only	1.833 k-ft	251.813 k-ft	137.352
+D+L	1.833 k-ft	262.813 k-ft	143.352
+D+S	1.833 k-ft	273.813 k-ft	149.352
+D+0.750L	1.833 k-ft	260.063 k-ft	141.852
+D+0.750L+0.750S	1.833 k-ft	276.563 k-ft	150.852
+0.60D	1.10 k-ft	151.088 k-ft	137.352

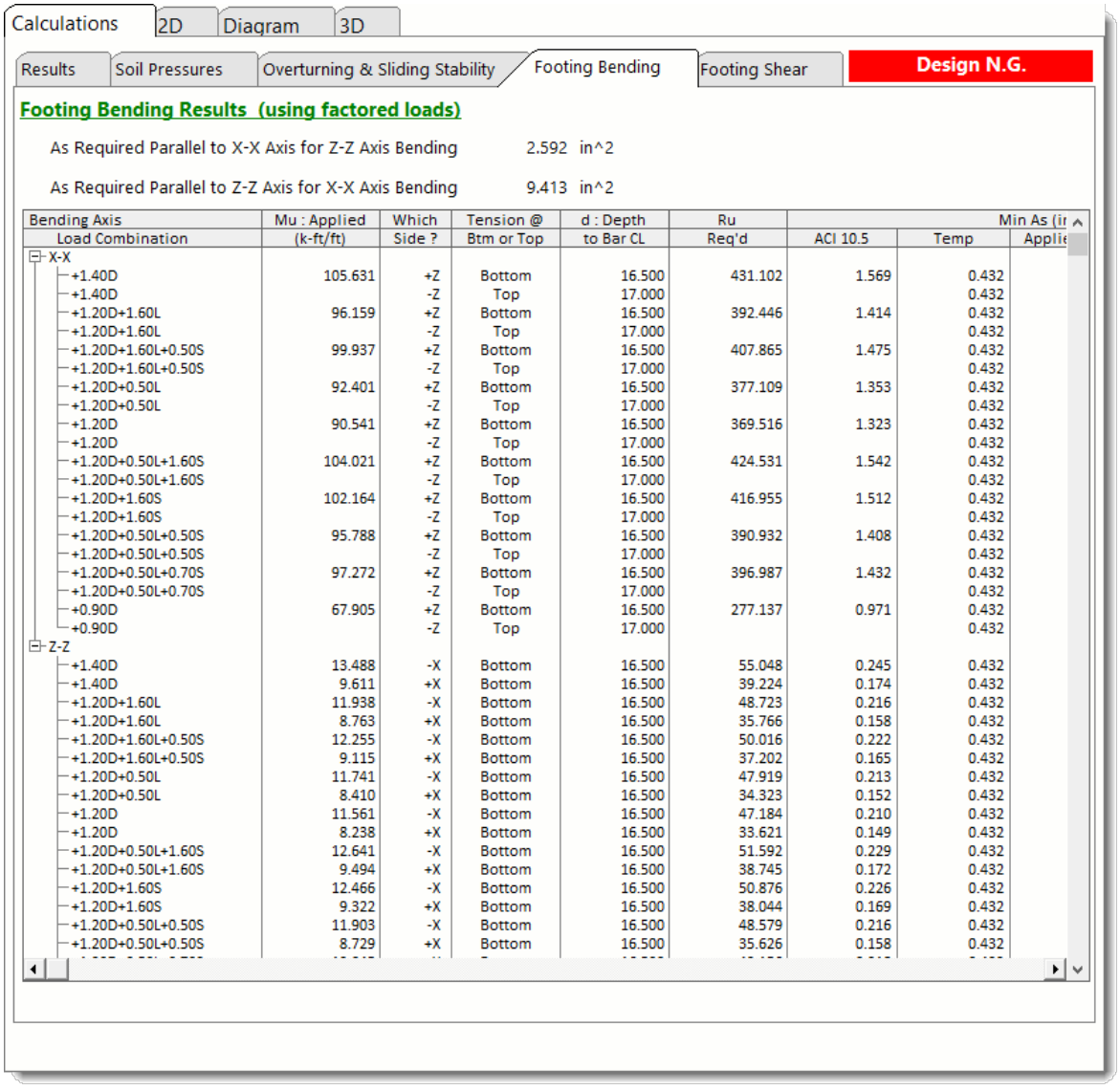
Note that the program is set up to look for overturning and resisting forces individually. For example, take the situation where the footing is subjected to equal and opposite shears at a given elevation. Common sense dictates that these forces cancel each other, and the footing experiences no net applied overturning moment from them. But the program treats one of the two equal and opposite forces as an *overturning* force, and the other as a

© 2026 ENERCALC, LLC

resisting force. So for these two forces, there IS a net overturning moment reported, but the resisting moment ALSO considers the effect of the opposing load, so the accounting used to determine the overturning ratio is proper.

Footing Bending

This tab provides a summary of the calculated factored load moment at all four edges of the pedestal perimeter for each load combination. It indicates whether the named load combination produces tension on the top surface of the footing or the bottom.



Calculations | 2D | Diagram | 3D

Results | Soil Pressures | Overturning & Sliding Stability | Footing Bending | Footing Shear | Design N.G.

Footing Bending Results (using factored loads)

As Required Parallel to X-X Axis for Z-Z Axis Bending 2.592 in^2

As Required Parallel to Z-Z Axis for X-X Axis Bending 9.413 in^2

Bending Axis / Load Combination	Mu : Applied (k-ft/ft)	Which Side ?	Tension @ Btm or Top	d : Depth to Bar CL	Ru Req'd	Min As (ir ACI 10.5	Min As (ir Temp	Min As (ir Applie
X-X								
+1.40D	105.631	+Z	Bottom	16.500	431.102	1.569	0.432	
+1.40D		-Z	Top	17.000			0.432	
+1.20D+1.60L	96.159	+Z	Bottom	16.500	392.446	1.414	0.432	
+1.20D+1.60L		-Z	Top	17.000			0.432	
+1.20D+1.60L+0.50S	99.937	+Z	Bottom	16.500	407.865	1.475	0.432	
+1.20D+1.60L+0.50S		-Z	Top	17.000			0.432	
+1.20D+0.50L	92.401	+Z	Bottom	16.500	377.109	1.353	0.432	
+1.20D+0.50L		-Z	Top	17.000			0.432	
+1.20D	90.541	+Z	Bottom	16.500	369.516	1.323	0.432	
+1.20D		-Z	Top	17.000			0.432	
+1.20D+0.50L+1.60S	104.021	+Z	Bottom	16.500	424.531	1.542	0.432	
+1.20D+0.50L+1.60S		-Z	Top	17.000			0.432	
+1.20D+1.60S	102.164	+Z	Bottom	16.500	416.955	1.512	0.432	
+1.20D+1.60S		-Z	Top	17.000			0.432	
+1.20D+0.50L+0.50S	95.788	+Z	Bottom	16.500	390.932	1.408	0.432	
+1.20D+0.50L+0.50S		-Z	Top	17.000			0.432	
+1.20D+0.50L+0.70S	97.272	+Z	Bottom	16.500	396.987	1.432	0.432	
+1.20D+0.50L+0.70S		-Z	Top	17.000			0.432	
+0.90D	67.905	+Z	Bottom	16.500	277.137	0.971	0.432	
+0.90D		-Z	Top	17.000			0.432	
Z-Z								
+1.40D	13.488	-X	Bottom	16.500	55.048	0.245	0.432	
+1.40D	9.611	+X	Bottom	16.500	39.224	0.174	0.432	
+1.20D+1.60L	11.938	-X	Bottom	16.500	48.723	0.216	0.432	
+1.20D+1.60L	8.763	+X	Bottom	16.500	35.766	0.158	0.432	
+1.20D+1.60L+0.50S	12.255	-X	Bottom	16.500	50.016	0.222	0.432	
+1.20D+1.60L+0.50S	9.115	+X	Bottom	16.500	37.202	0.165	0.432	
+1.20D+0.50L	11.741	-X	Bottom	16.500	47.919	0.213	0.432	
+1.20D+0.50L	8.410	+X	Bottom	16.500	34.323	0.152	0.432	
+1.20D	11.561	-X	Bottom	16.500	47.184	0.210	0.432	
+1.20D	8.238	+X	Bottom	16.500	33.621	0.149	0.432	
+1.20D+0.50L+1.60S	12.641	-X	Bottom	16.500	51.592	0.229	0.432	
+1.20D+0.50L+1.60S	9.494	+X	Bottom	16.500	38.745	0.172	0.432	
+1.20D+1.60S	12.466	-X	Bottom	16.500	50.876	0.226	0.432	
+1.20D+1.60S	9.322	+X	Bottom	16.500	38.044	0.169	0.432	
+1.20D+0.50L+0.50S	11.903	-X	Bottom	16.500	48.579	0.216	0.432	
+1.20D+0.50L+0.50S	8.729	+X	Bottom	16.500	35.626	0.158	0.432	

Ru Req'd: R_u is the "design moment factor", also sometimes referred to as "K_u". It is a normalized measure of bending demand and is calculated as: $R_u = M_u/(\phi b d^2)$

© 2026 ENERCALC, LLC

R_u can then be used to determine the required reinforcement ratio for strength:

$$\rho = (0.85f'_c/f_y) * \left[1 - \sqrt{1 - R_u/(0.383f'_c)}\right]$$

Once the required reinforcement ratio is known, the minimum required steel area based on the applied moment can be calculated as: $\text{Min As Based on Applied Mu} = \rho * (bd)$

Footing Shear

This tab provides a summary of the calculated factored load shear at all four edges of the pedestal perimeter for each load combination. Two-way or punching shear is also calculated.

For footings designed per ACI 318-14 and earlier, and ACI 318-25 and later:

The nominal shear strength follows the one-way shear provisions of ACI 318-14 §22.5.5.1 / ACI 318-25 §13.2.6.2:

$\varphi v_n = \varphi * 2 * \lambda * \sqrt{f'_c}$, where φ is specified by the user on the *General* tab.

For footings designed per ACI 318-19:

The nominal shear strength follows the one-way shear provisions of §22.5 and Table 22.5.5.1 (referenced by §7.5.3.1):

$\varphi v_n = \varphi * 8 * \lambda_s * \lambda * (\rho_w)^{1/3} * \sqrt{f'_c}$, where φ is specified by the user on the *General* tab.

Note: The one-way shear capacity assumes no transverse reinforcement in the footing. Therefore, $A_v < A_{v,min}$, and equations (a) and (b) from Table 22.5.5.1 are not applicable.

It is also acceptable to neglect the size effect factor (λ_s) for one-way shallow foundations per §13.2.6.3, so $\lambda_s = 1.0$.

This value is also subject to additional limits per §22.5.5.1.1:

Lower bound: $\varphi * \lambda * \sqrt{f'_c}$

Upper bound: $\varphi * 5 * \lambda * \sqrt{f'_c}$

© 2026 ENERCALC, LLC

Calculations | 2D | 3D

Results | Soil Pressures | Overturning & Sliding | Bending | Shear

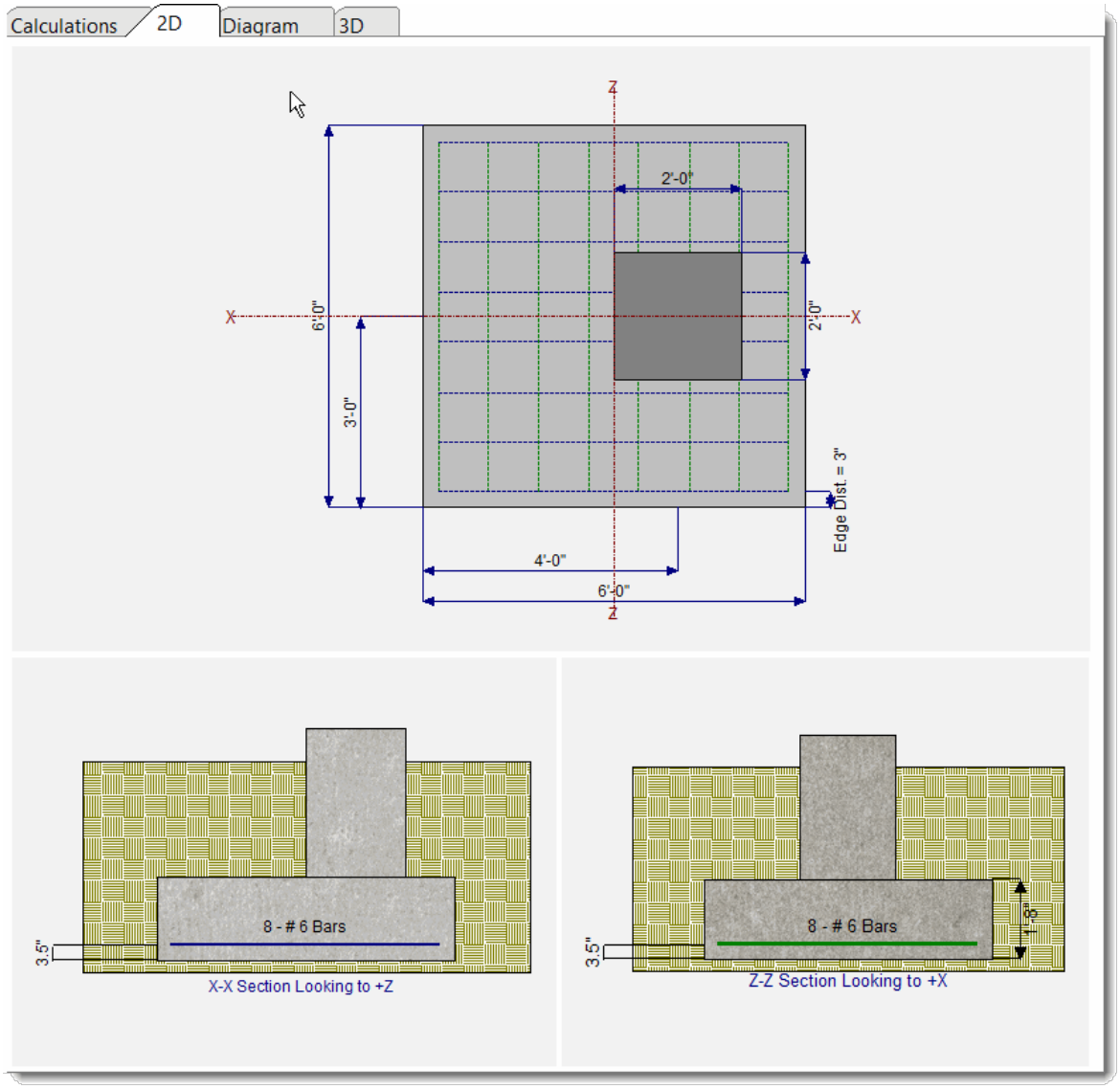
Design OK

Footing Shear Results (using factored loads) .....

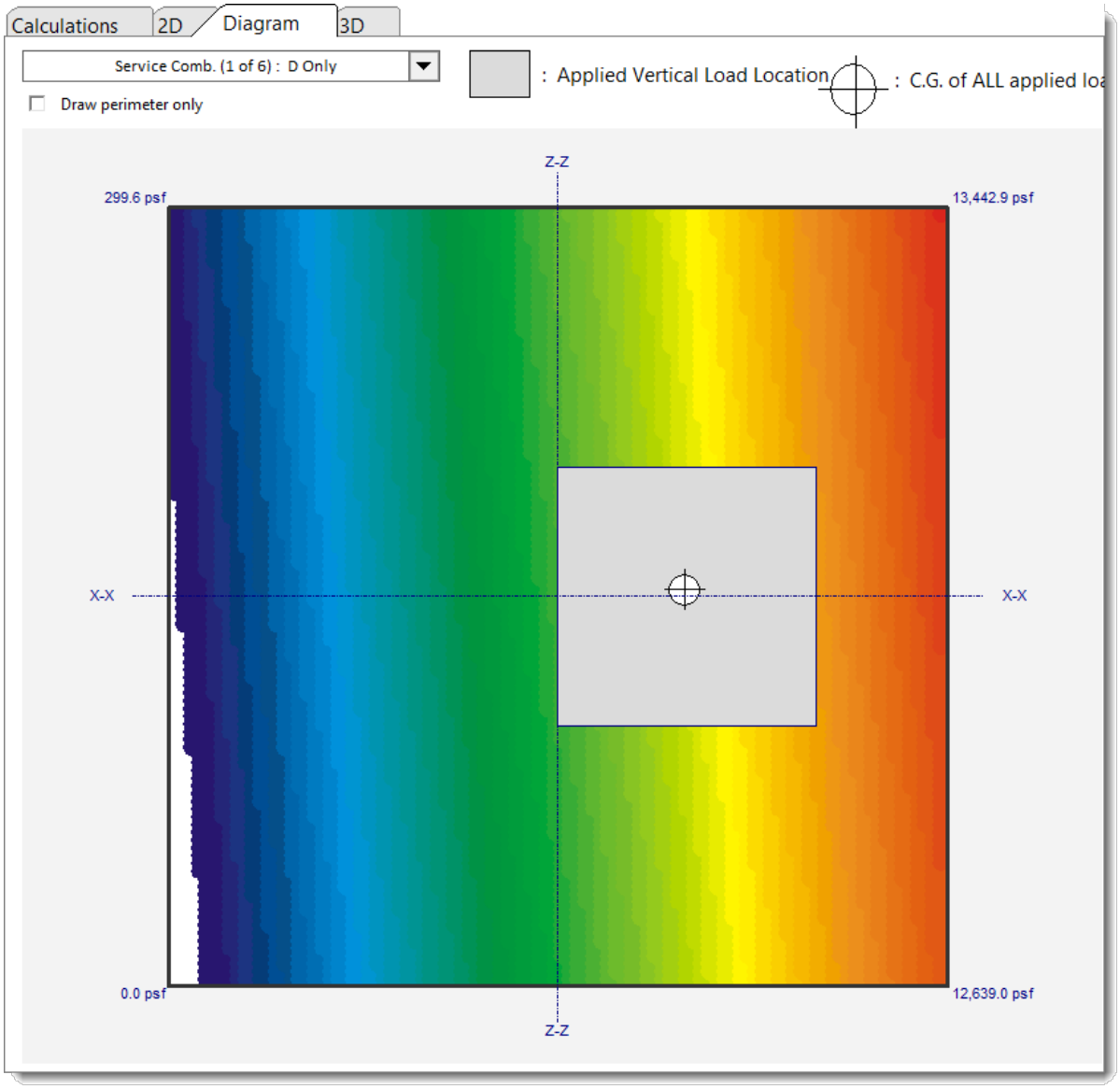
Load Combination	λ	λs	One-Way Shear on +X and -X Edges					
			Vu @ -X	Vu @ +X	Max.	ρw (X)	φVc/(Length * d)	Vu/φ
+1.40D	1.000	1.000	12.514	13.957	13.957	0.00277	46.2	
+1.20D+1.60L	1.000	1.000	12.137	13.440	13.440	0.00277	46.2	
+1.20D+1.60L+0.50S	1.000	1.000	12.452	13.770	13.770	0.00277	46.2	
+1.20D+0.50L	1.000	1.000	11.167	12.425	12.425	0.00277	46.2	
+1.20D	1.000	1.000	10.726	11.964	11.964	0.00277	46.2	
+1.20D+0.50L+1.60S	1.000	1.000	12.175	13.480	13.480	0.00277	46.2	
+1.20D+1.60S	1.000	1.000	11.734	13.018	13.018	0.00277	46.2	
+1.20D+0.50L+0.50S	1.000	1.000	11.482	12.755	12.755	0.00277	46.2	
+1.20D+0.50L+0.70S	1.000	1.000	11.608	12.887	12.887	0.00277	46.2	
+0.90D	1.000	1.000	8.045	8.973	8.973	0.00277	46.2	

© 2026 ENERCALC, LLC

2D Sketch

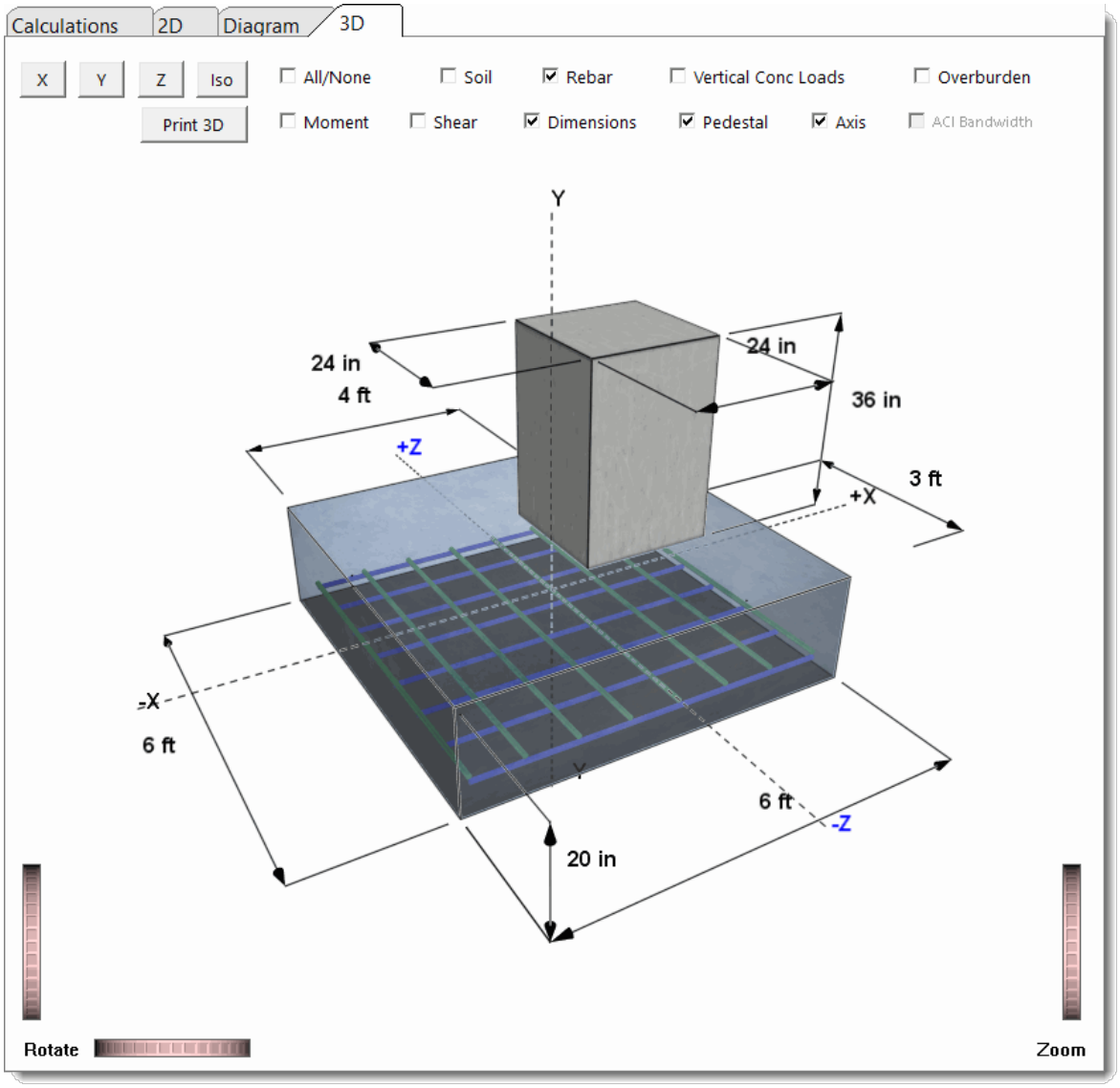

Calculations
2D
Diagram
3D
Z
2'-0"
X
6'-0"
X
2'-0"
3'-0"
Edge Dist. = 3"
4'-0"
6'-0"
Z
8 - # 6 Bars
3.5"
X-X Section Looking to +Z
8 - # 6 Bars
3.5"
1'-8"
Z-Z Section Looking to +X

Diagram



© 2026 ENERCALC, LLC

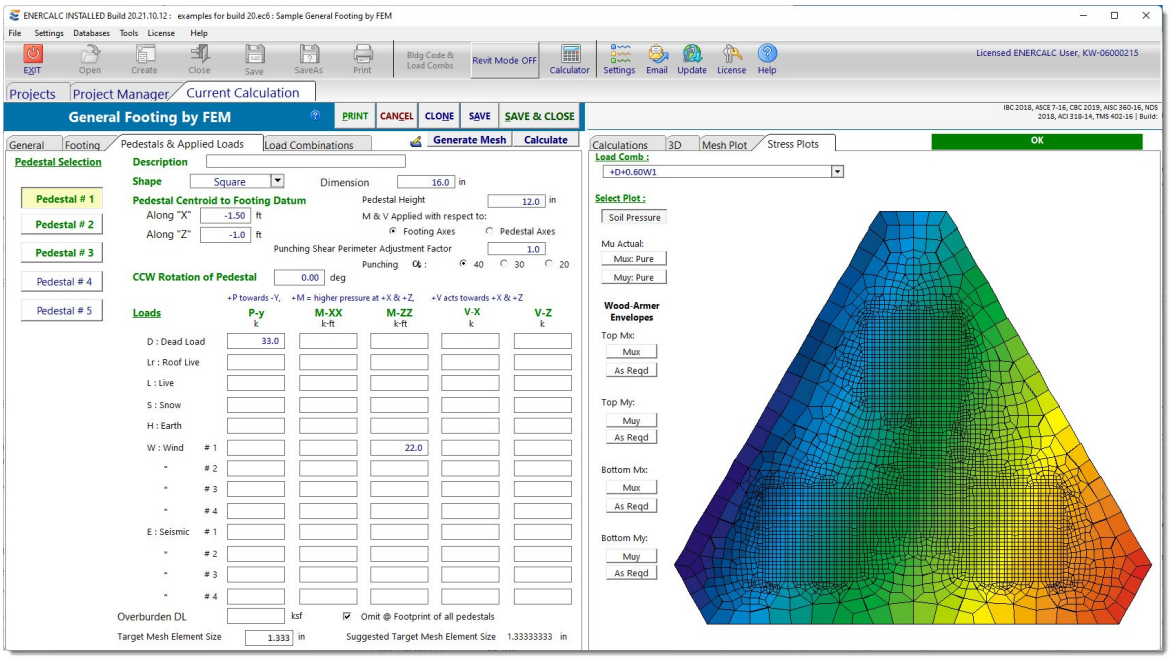
3D Rendering



13.6.2 General Footing by FEM

The General Footing by FEM module is useful for footings of a variety of shapes and supporting up to five pedestals.

© 2026 ENERCALC, LLC



The module gathers geometric info, soil properties, and loading data, and then generates a finite element model that is analyzed behind the scenes in ENERCALC 3D. The results are then passed back to the module for review and refinement.

This process allows for considerably more flexibility than some of the other footing modules. For example, this module supports many common shapes of footings. It also allows square, rectangular and round pedestals to be modeled, and the square and rectangular pedestals can even be rotated. A full biaxial analysis is performed using compression-only springs for each load combination, including service-level combinations for soil pressure evaluation and stability checks, and strength-level combinations for concrete design.

The module allows the user to control the mesh density, and to review the resulting mesh that is automatically created prior to running the analysis. It also offers two options for the stiffness of the footing. One option is to use the calculated stiffness based on material properties, thickness, and specified cracking factors. A second option is to consider the footing as a "Rigid Body", which evaluates the footing as if it was an infinitely stiff plate.

General Tab

Concrete & Rebar Properties

Enter material properties and design parameters. Note that code modifications for lightweight concrete are not applied in this module. The purpose of this input is mainly to allow the user to specify something in the range of 145 to 150 pcf.

© 2026 ENERCALC, LLC

Stability Settings

Set minimum allowable factors of safety for sliding and overturning checks.

Soil Design Values

Specify allowable soil bearing pressure, subgrade modulus, soil density, coefficient of friction and passive pressure.

Footing Tab

Footing Shape

Select the desired footing shape and specify the data necessary to define the plan dimensions of the footing.

Suggested Footing Target Element Size

The module suggests a value to be used as the Footing Target Element Size. The value generally represents one-fiftieth of the longest dimension of the footing.

Footing Target Element Size

This field is used to specify the desired target size of the finite elements that will be used to model the footing. Note that this input is specified in units of inches, and that this is only a target size. The meshing procedure will attempt to achieve this size for as many elements as possible, but it is not a rule or a limit. The smaller the target element size, the larger the model (roughly a squared function).

Footing Thickness

Specify the thickness of the footing in units of inches.

Footing Stiffness

This section offers two options. The first is to treat the footing as a rigid body. The second is to consider the actual stiffness of the footing based on thickness, material properties, and the specified cracking factor.

© 2026 ENERCALC, LLC

Pedestals & Applied Loads Tab

Pedestal Selection

Click the desired button to enter geometric and loading data for a particular pedestal. Up to five pedestals can be specified per footing.

Description

The description field is available for a text description of each individual pedestal.

Shape

The shape dropdown list box offers Unused, Square, Rectangular and Round. Dimension fields are displayed based on the selected shape. Note that pedestal dimensions are always collected in units of inches, even if the "pedestal" is long, like a wall. Enter the Pedestal Height (in inches) above the top of the footing.

Pedestal Centroid to Footing Datum

This area collects dimensions to locate the centroid of the pedestal. Note that dimensions are in units of feet, and that the pedestal outline is visible on the Mesh Plot for reference.

Punching Shear Perimeter Adjustment Factor

This factor acts as a multiplier on the calculated punching perimeter. The punching perimeter is calculated as follows:

- Square Pedestal: 4 * (d/2 + Pedestal dim + d/2)
- Rectangular Pedestal: 2 * [(d/2 + Pedestal X dim + d/2) + (d/2 + Pedestal Z dim + d/2)]
- Round Pedestal: pi * (d/2 + diameter + d/2)

So by applying the Punching Shear Perimeter Adjustment Factor to the calculated punching perimeter, it is possible to account for situations where the punching perimeter falls partially outside the footing, such as in corner and edge conditions. Note that this factor is separate and distinct from the Punching Alpha s Factor described in the next section. For example, if it is determined that only three quarters of the calculated punching perimeter actually falls within the footprint of the footing for a particular corner or edge column, then the Punching Shear Perimeter Adjustment Factor can be specified as 0.75 for that column.

© 2026 ENERCALC, LLC

Note that the module collects clear cover, not an actual value of d. And the size of the bars is not known. So the program makes an allowance of 1/2 inch for the rebar radius when calculating d. The value of d is calculated as Footing Thickness minus clear cover minus an additional one-half inch to account for the radius of the rebar.

Punching Alpha s Factor

Based on the position of a pedestal (interior, edge, or corner), ACI requires the use of a specific value for the Alpha s factor: 40, 30 or 20 respectively.

CW Rotation

This field allows square and rectangular pedestals to be rotated by a specified number of degrees. Positive = CW. Negative = CCW.

Loads

Loads can be specified for each of the primary load types. Positive values of Py act downward. Positive Mxx and Mzz tend to create higher soil pressure on the positive X and positive Z sides of the footing, respectively. Positive Vx and positive Vz act in the positive X and positive Z directions, respectively. Moment and shear can be defined with respect to the footing axis system or the pedestal axis system. Overburden DL can be specified over the entire footing. It can also be omitted at the footprint of pedestals.

Loading is facilitated by the fact that the module allows up to four independent wind load cases and four independent seismic load cases for each model. This means that one calculation can include cases such as:

1. Net wind to the **East**,
2. Net wind to the **West**,
3. Net wind to the **North**, and
4. Net wind to the **South**.

It could also be set up to run:

1. Net wind in the **East-West** direction with **positive** internal pressure,
2. Net wind in the **East-West** direction with **negative** internal pressure,
3. Net wind in the **North-South** direction with **positive** internal pressure, and
4. Net wind in the **North-South** direction with **negative** internal pressure.

© 2026 ENERCALC, LLC

Likewise, this increased loading capacity means that a single calculation could include seismic load cases such as:

1. Seismic in the **East-West** direction with **positive** eccentricity,
2. Seismic in the **East-West** direction with **negative** eccentricity,
3. Seismic in the **North-South** direction with **positive** eccentricity, and
4. Seismic in the **North-South** direction with **negative** eccentricity.

It could also be set up to run:

1. 100% Seismic in the **East-West** direction with a concurrent 30% seismic to the **North**,
2. 100% Seismic in the **East-West** direction with a concurrent 30% seismic to the **South**,
3. 100% Seismic in the **North-South** direction with a concurrent 30% seismic to the **East**, and
4. 100% Seismic in the **North-South** direction with a concurrent 30% seismic to the **West**.

Note: Footing self-weight is applied automatically as Dead Load. Pedestal self-weight is not automatically accounted for.

© 2026 ENERCALC, LLC

Suggested Target Mesh Element Size

The module suggests a value to be used as the Target Mesh Element Size at the pedestal footprint.

Footing Target Element Size

This field is used to specify the desired target mesh element size of the finite elements that will be used to model the footing within the footprint of the selected pedestal. Note that this input is specified in units of inches, and that this is only a target size. The meshing procedure will attempt to achieve this size for as many elements as possible, but it is not a rule or a limit. The smaller the target element size, the larger the model (roughly a squared function).

Load Combinations Tab

Select the desired load combination set with the Select button. Specify Sds and Rho values to be incorporated into the generated load combinations if desired. Combination sets are automatically generated for strength-level (concrete design), and service-level (soil pressure, sliding, overturning). Each load combination in each set can be individually modified and/or turned on or off as desired.

Calculations Tab

Results

Displays a summary of failure modes, factors of safety or utilization ratios, design values, and governing load combinations.

Soil Pressure

Reports the maximum soil pressure and associated load combination, as well as soil pressures for each shell for each load combination.

Settlement

Reports the maximum settlement and associated load combination, as well as settlement for each node for each load combination.

Actual Mu & Vu

© 2026 ENERCALC, LLC

Reports the maximum Mxx, Myy, Vxx and Vyy with associated load combinations, as well as values of Mxx, Myy, Vxx and Vyy for each shell for each load combination.

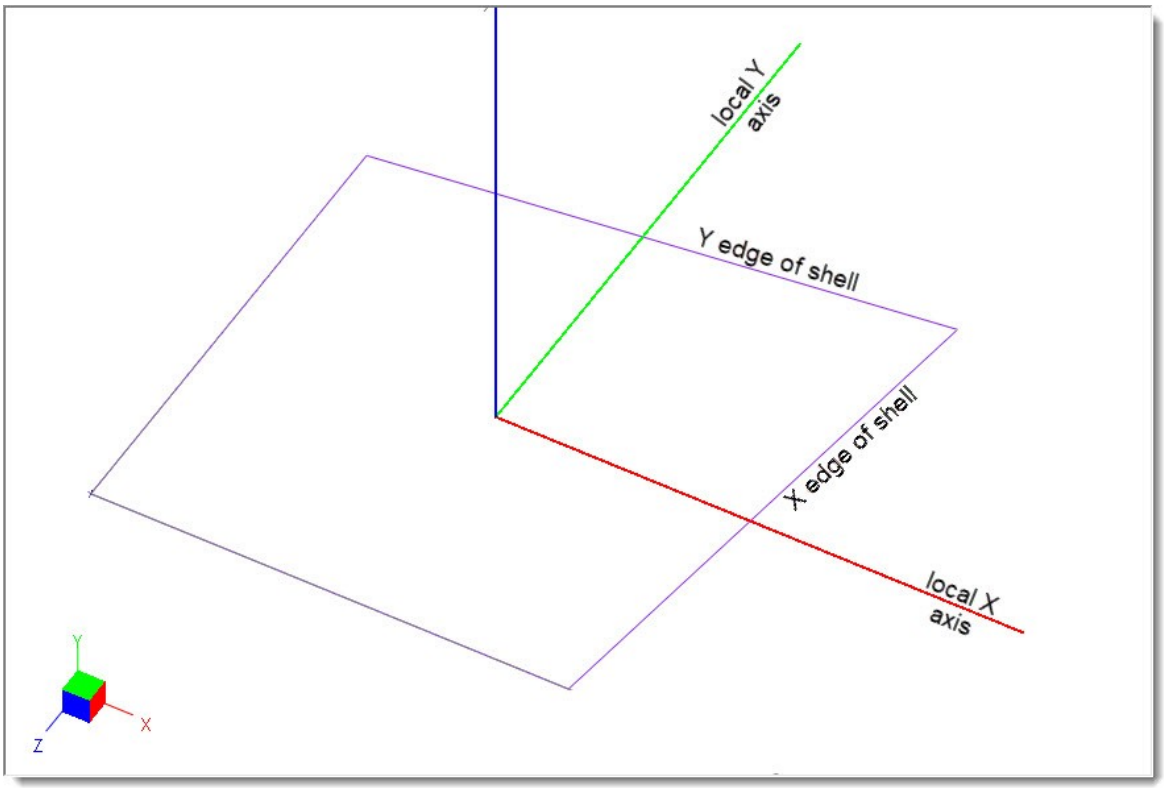
Wood-Armer Results

Reports envelope results of Wood-Armer moments and required areas of steel for the top surface and the bottom surface at each edge of each shell.

Technical Note:

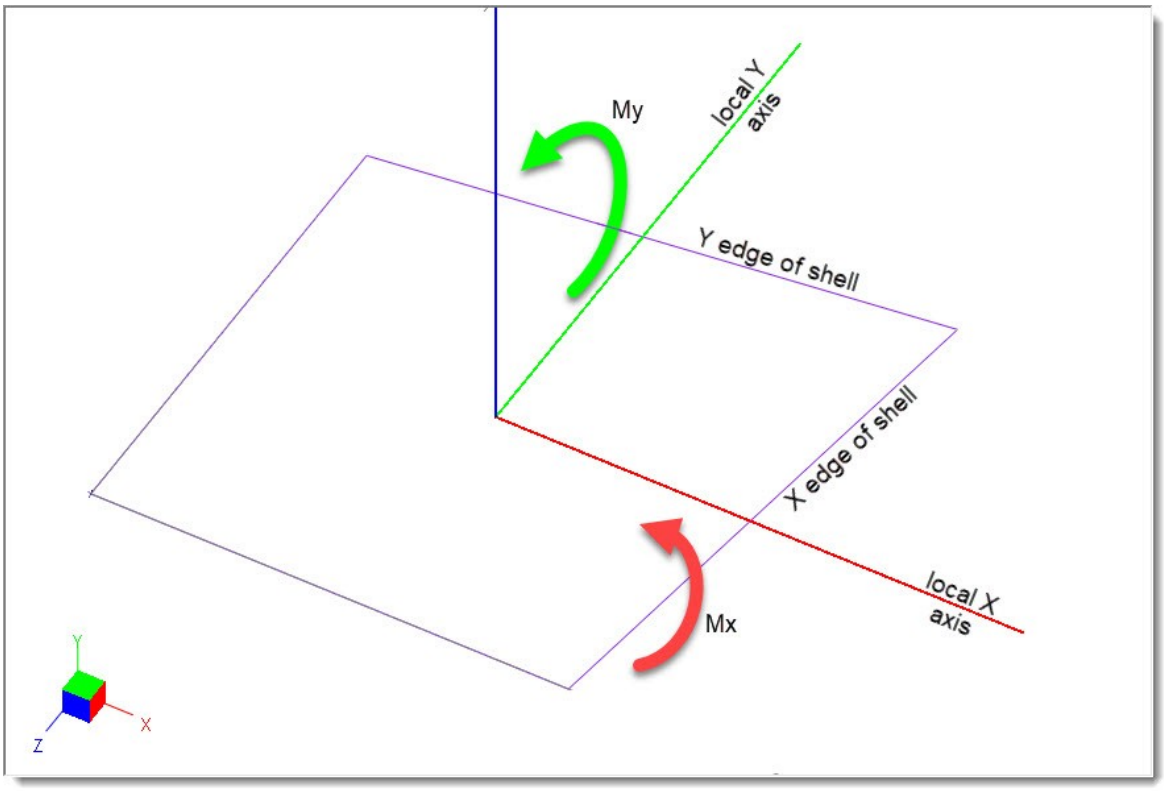
The General Footing x FEM module uses ENERCALC 3D as the underlying solver.

ENERCALC 3D refers to shell edges as follows. The X edge of the shell is the edge that the local X axis protrudes through, and the Y edge of the shell is the edge that the local Y axis protrudes through.



When ENERCALC 3D refers to shell moments, it uses the following convention. Mx represents the moment on the X edge of the shell, and My represents the moment on the Y edge of the shell.

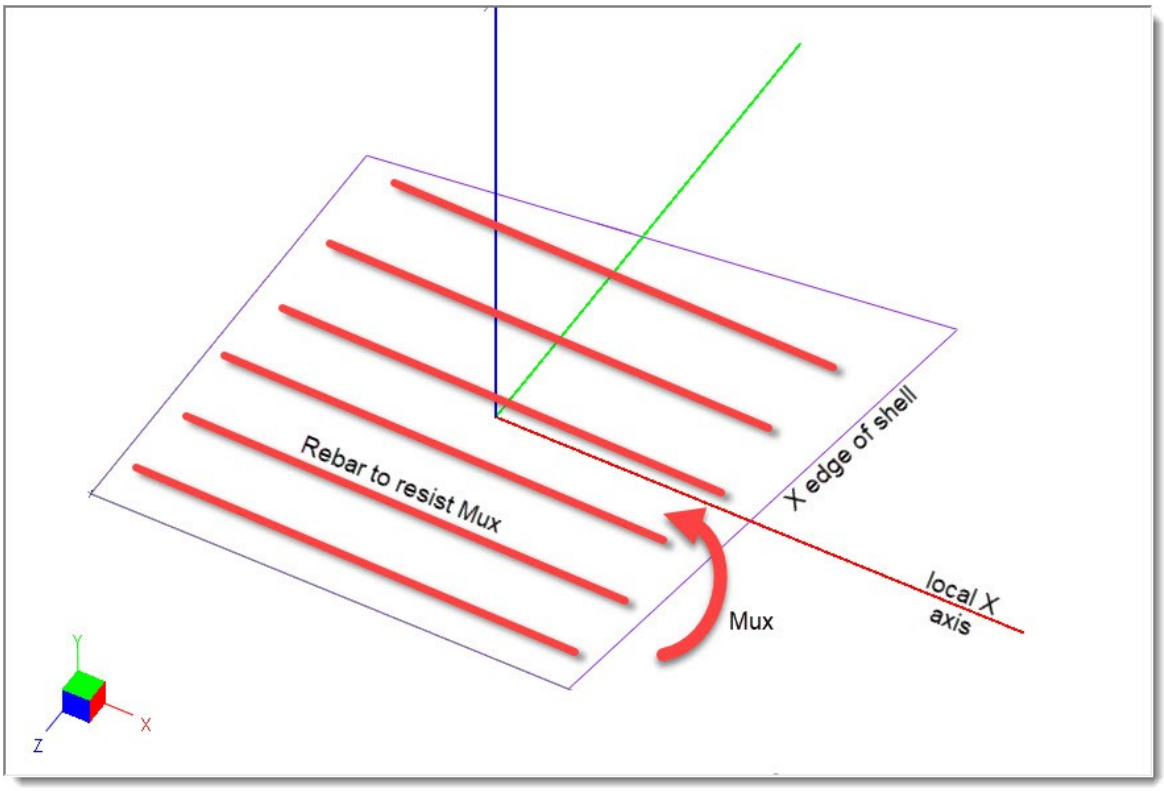
© 2026 ENERCALC, LLC



The General Footing by FEM module automatically orients all shells so that their local X axis coincides with the global X axis (both in direction and in algebraic sense). Following the right-hand rule, this also means that the local Y axis is parallel to the global Z axis, and the positive sense of the local Y axis points in the opposite direction from the positive sense of the global Z axis, as shown in the image above.

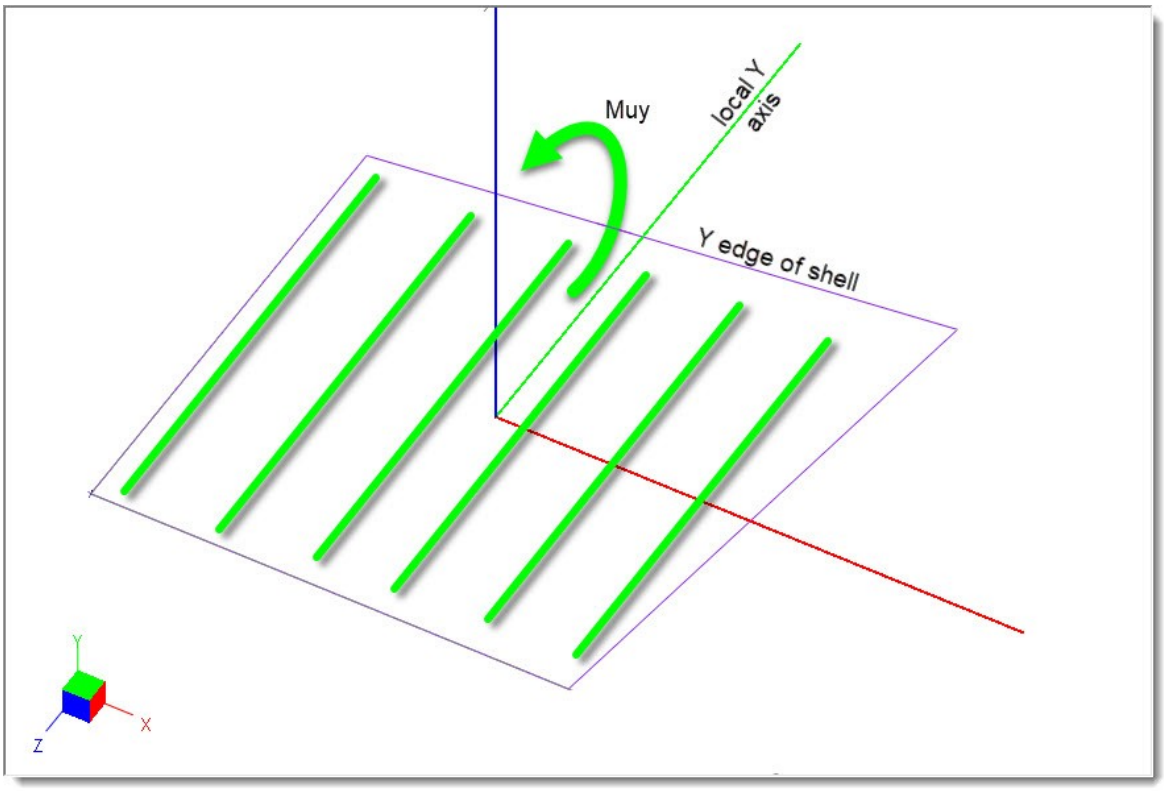
Wood-Armer moments Mux are ***not*** moments about the global X axis. Instead, they act on the X edge of the shell, meaning that they would be resisted by reinforcing steel placed ***parallel*** to the global X axis (perpendicular to the global Z axis), as shown below.

© 2026 ENERCALC, LLC



Likewise, Wood-Armer moments Muy act on the Y edge of the shell, meaning that they act ***about*** the local and global X axis, and would be resisted by reinforcing steel placed parallel to the local y (global Z) axis, as shown below.

© 2026 ENERCALC, LLC

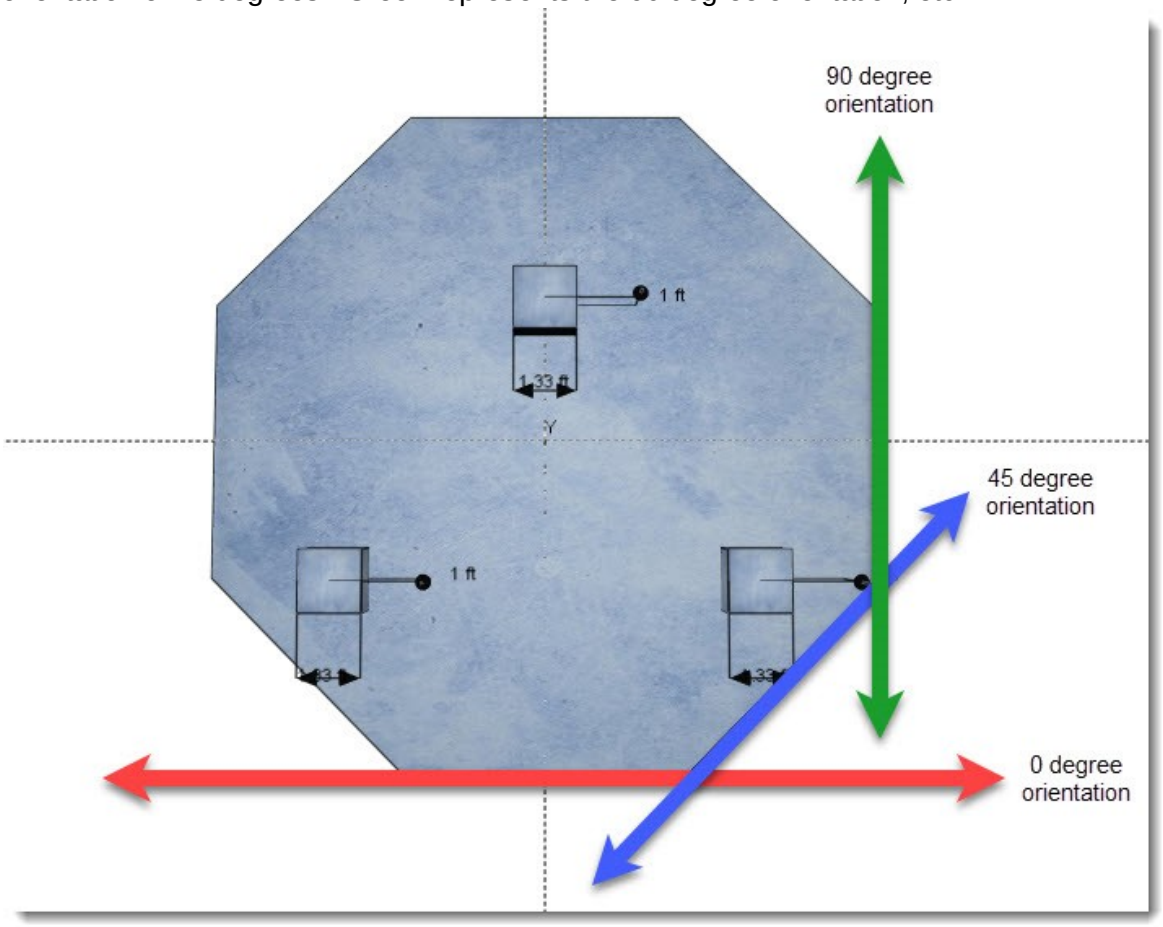


© 2026 ENERCALC, LLC

Overturning

Reports the overall overturning stability ratio and the associated load combination and orientation of the overturning axis. Also reports the controlling overturning ratio for each load combination, including the overturning moment, the resisting moment, and the orientation of the overturning axis.

Overturning is evaluated about many axes starting with the axis oriented at zero degrees, represented by the red line in the screen capture below. The blue line represents a CCW orientation of 45 degrees. Green represents the 90 degree orientation, etc.



Sliding

Reports the lowest sliding ratio and the associated load combination. Also reports the sliding ratio for each load combination, including the vertical load, friction coefficient, sliding resistance, and applied force.

Punching

© 2026 ENERCALC, LLC

Reports the punching shear and resistance.

Shell Info

Reports the area and bounding node info for each shell in the model.

3D Tab

Displays a 3D rendering of the footing. Display options allow control over the individual elements to be shown.

Mesh Plot Tab

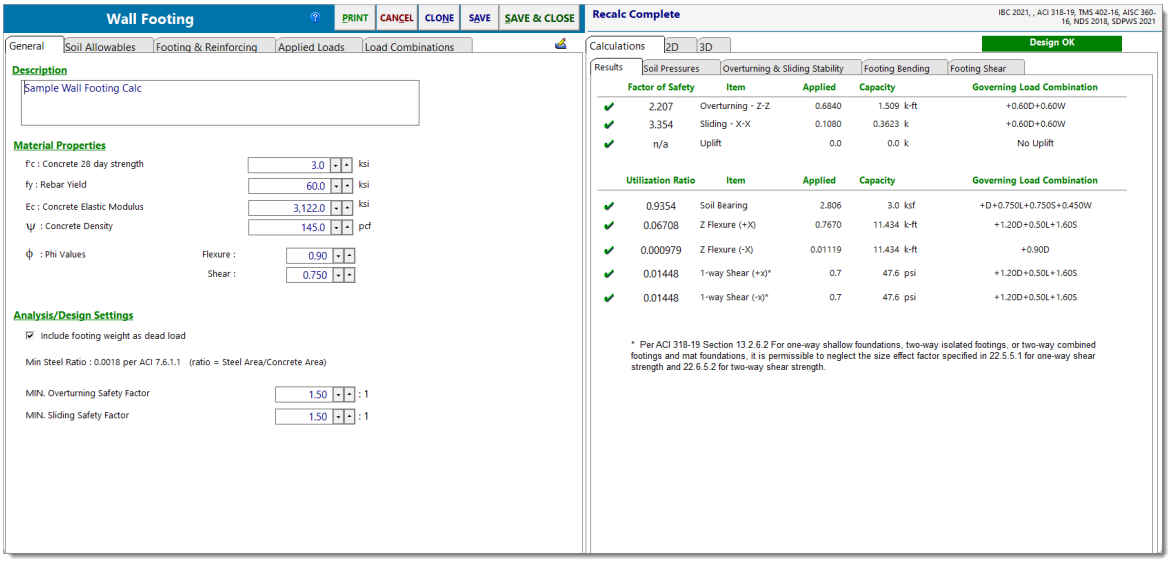
Displays a graphical depiction of the underlying finite element model mesh. Click the Regenerate button to refresh the mesh display.

Stress Plots Tab

Displays plots that graphically depict soil pressure, pure flexural moments, Wood-Armer design moments, and area of steel required to satisfy Wood-Armer moments.

13.6.3 Wall Footing

This module provides analysis of a unit strip of a continuous wall footing.



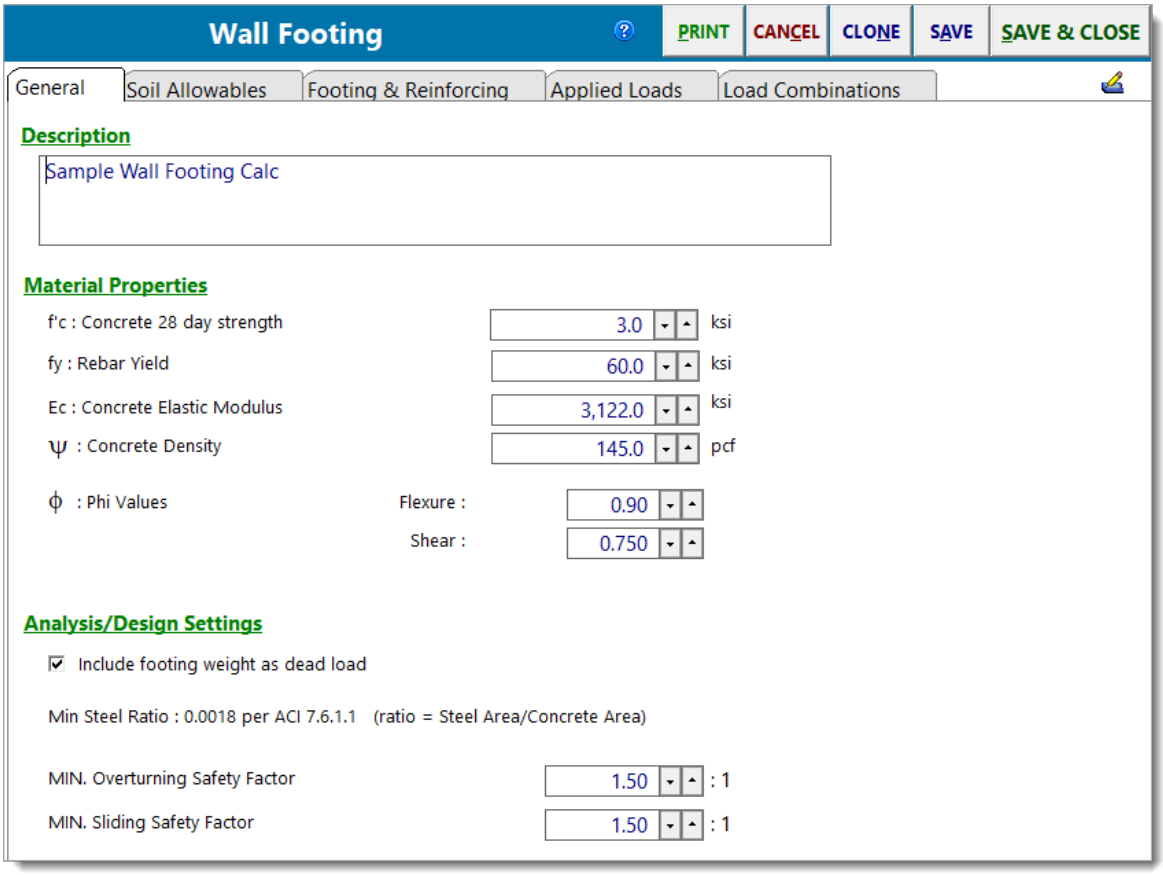
© 2026 ENERCALC, LLC

Axial load, moment, and shear loads can be applied. Overburden loads can be specified, and will apply to the surface area of the footing (except the area covered by the wall). The module also provides automatic calculation of allowable soil bearing pressure increases based on footing width and/or depth below surface.

The module checks service load soil pressure, overturning stability, sliding stability, uplift stability, footing flexure and one-way footing shear.

General

This tab collects material property values, strength reduction factors, and other settings that influence the design.



f'c

28-day compressive strength of the concrete.

fy

Yield point stress of reinforcing.

Ec

Modulus of elasticity of concrete.

© 2026 ENERCALC, LLC

Concrete Density

The density of the concrete is used to calculate the self weight of the footing when the option is selected. Note that code modifications for lightweight concrete are not applied in this module. The purpose of this input is mainly to allow the user to specify something in the range of 145 to 150 pcf.

Phi Values

Enter the capacity reduction values to be applied to Vn and Mn.

Include footing weight as dead load

Click [**Yes**] to have the module calculate the weight of the footing and apply it as a downward load. The footing self weight will be multiplied by the dead load factor in each load combination.

Min Steel Ratio

This is set to 0.0018 per ACI 7.6.1.1. A warning message will be triggered if the section is under-reinforced.

NOTE: This is not T&S steel. It is a minimum that must be met for flexural steel. It is not something that can be "shared" between the top and bottom mats like T&S steel can.

Minimum Overturning Safety Factor

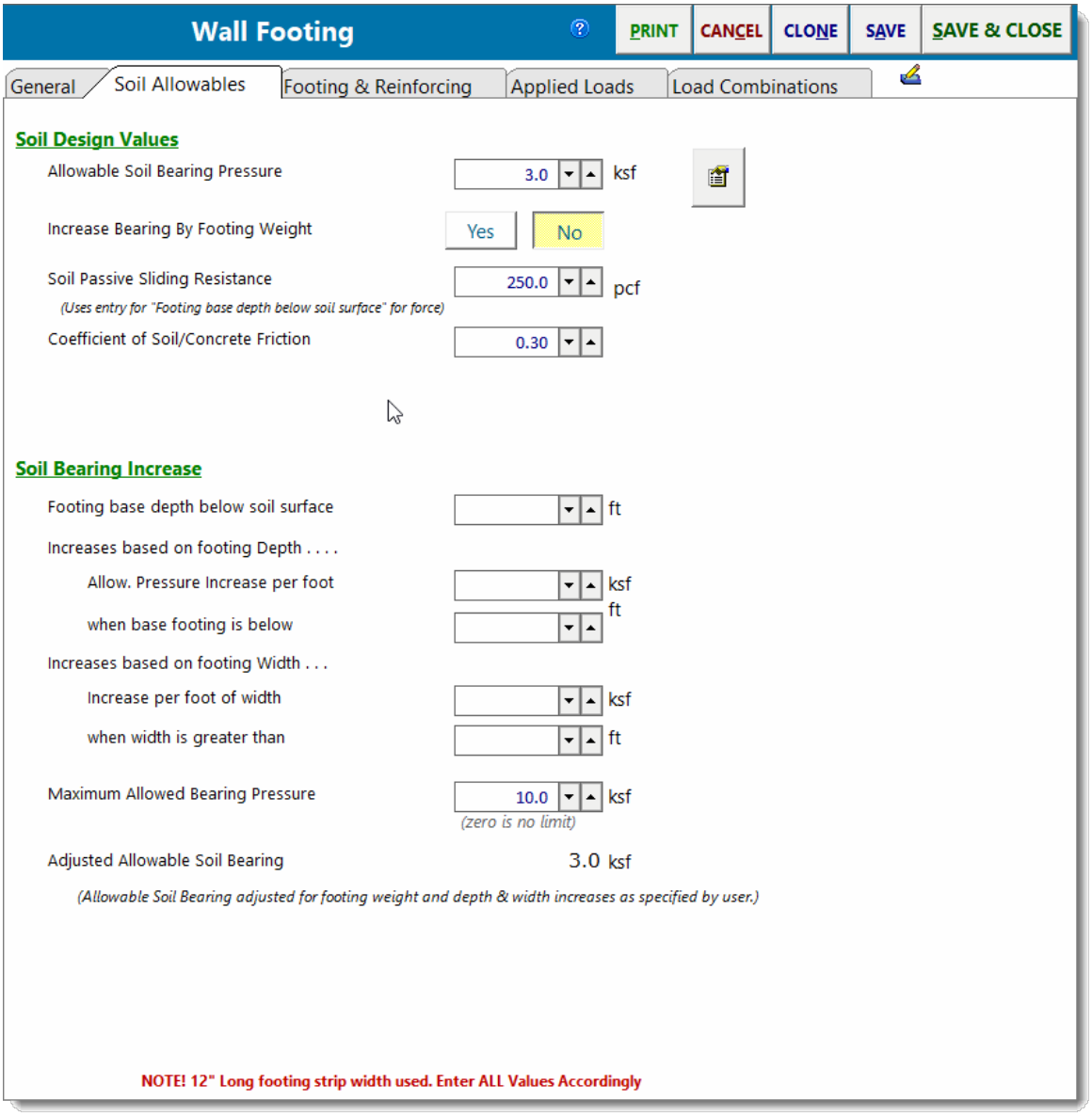
Enter the minimum allowable ratio of resisting moment to overturning moment. If the actual ratio is less than the specified minimum ratio, it will trigger a message that overturning stability is not satisfied.

Minimum Sliding Safety Factor

Enter the minimum allowable ratio of resisting force to sliding force. If the actual ratio is less than the specified minimum ratio, it will trigger a message that sliding stability is not satisfied.

© 2026 ENERCALC, LLC

Soil Allowables



Allowable Soil Bearing Pressure

Enter the allowable soil bearing pressure. This is a service load resistance and will be compared to calculated service load soil pressures (loads not factored as in strength design).

Increase Bearing by Footing Weight

Click [**Yes**] to tell the module to calculate the weight of one square foot (plan view) of footing and add it to the allowable soil bearing value. This has the effect of not penalizing the soil for the self weight of the footing, and is useful for situations where the geotechnical engineering report provides allowable net bearing pressures.

© 2026 ENERCALC, LLC

Soil Passive Sliding Resistance

Enter the value of passive soil pressure resistance to sliding. This value will be used to determine a component of sliding resistance that is generated by the passive pressure of the soil. The sliding resistance due to passive pressure is then added to the sliding resistance due to friction to determine the total resistance to sliding for each load combination.

Coefficient of Soil/Concrete Friction

Enter the coefficient of friction between soil and footing to use in sliding resistance calculations.

Soil Bearing Increase

This section allows you to specify some dimensions that, when exceeded, will automatically increase the allowable soil bearing pressure.

Footing base depth below soil surface: The distance from the bottom of the footing to the top of the soil. This value is used to determine allowable soil bearing pressure increases and soil passive sliding resistance, but it is NOT used in any other calculations in this module.

Increases based on footing depth: Provides a method to automatically apply increases to the basic allowable soil bearing pressure based on footing depth below some reference depth. Collects the following parameters:

Allowable pressure increase per foot: Specifies the amount that the basic allowable soil bearing pressure can be increased for each foot of depth below some reference depth.

When base of footing is below: Specifies the required depth in order to start realizing incremental increases in the allowable soil bearing pressure on the basis of footing depth.

Example: Assume the following: Basic Allowable Soil Bearing Pressure = 3 ksf. Footing base is 6'-0" below soil surface. The Geotechnical report specifies that a 0.15 ksf increase in bearing pressure is allowed for each foot of depth when the base is deeper than 4' below top of soil. Since you've indicated that the footing is 6' below the soil surface, the module will automatically calculate the adjusted allowable soil bearing pressure to be 3 ksf + (6' - 4') * 0.15 ksf = 3.30 ksf.

© 2026 ENERCALC, LLC

Increases based on footing width: Provides a method to automatically apply increases to the basic allowable soil bearing pressure based on footing width greater than some reference dimension. Collects the following parameters:

Allowable pressure increase per foot: Specifies the amount that the basic allowable soil bearing pressure can be increased for each foot of width greater than some reference dimension.

When maximum length or width is greater than: Specifies the required dimension in order to start realizing incremental increases in the allowable soil bearing pressure on the basis of footing width.

Example: Assume the following: Basic Allowable Soil Bearing Pressure = 3 ksf. Footing measures 6'-0" wide. The Geotechnical report specifies that a 0.15 ksf increase in soil bearing pressure is allowed for each foot when the width of the footing is greater than 4'-0". The module will automatically calculate the adjusted allowable soil bearing pressure to be 3 ksf + (6' - 4') * 0.15 ksf = 3.3 ksf.

Maximum Allowed Bearing Pressure: Provides a way to specify an upper limit on the adjusted allowable bearing pressure.

Note: Increases based on footing depth and width are cumulative.

Footing & Reinforcing

Dimensions

Footing Width: Define the width of the footing.

Wall Width: Define the width of the supported wall.

Wall center offset from footing centerline: Define the dimension between the centerline of the wall and the centerline of the footing. Positive offsets shift the wall toward the right edge of the footing.

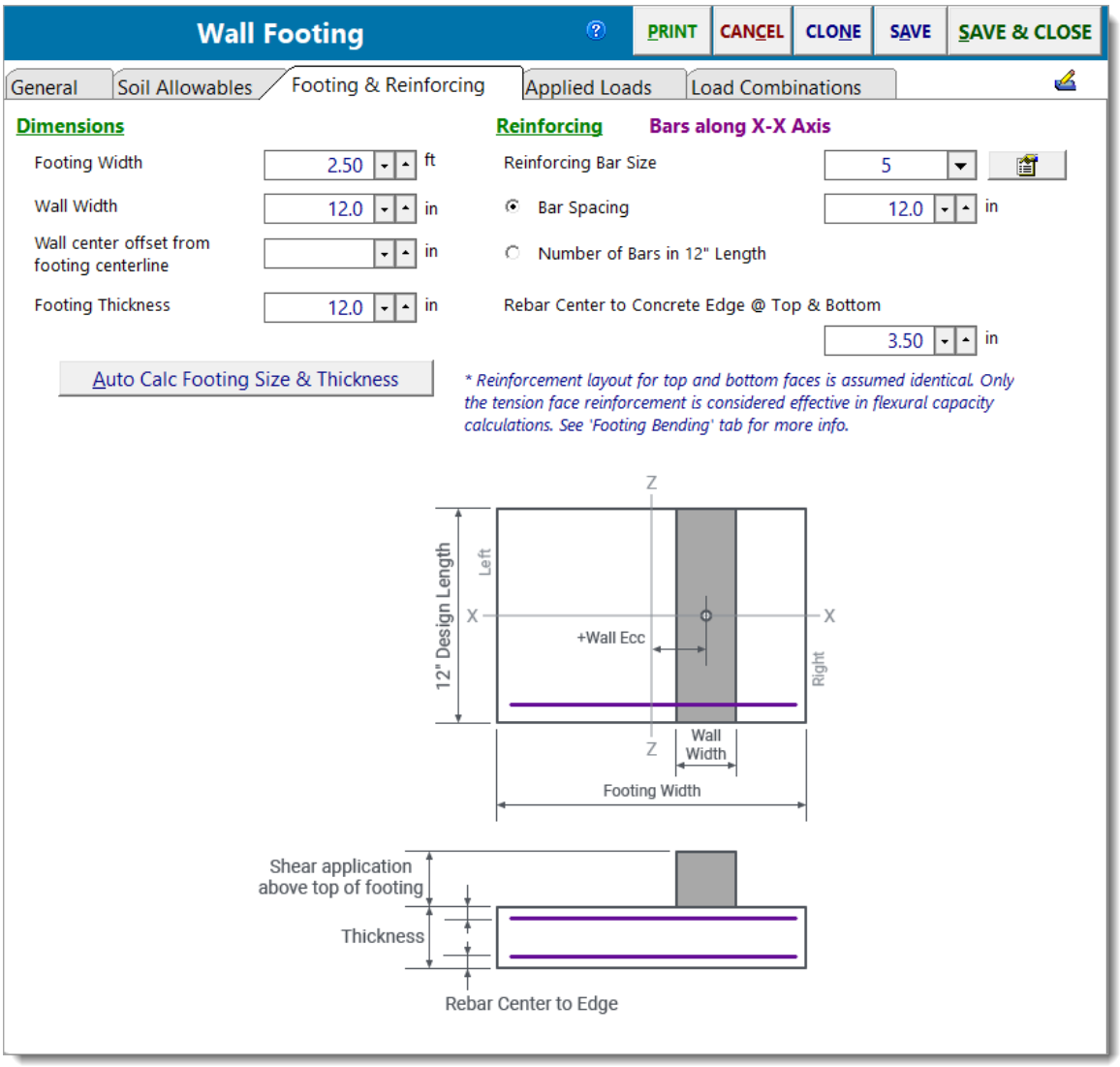
Footing Thickness: Define the thickness of the footing.

Auto Calculate Footing Size & Thickness: Provides an automated routine to increase footing dimensions until soil pressures are satisfied and one-way shear is acceptable.

Note: Any applied overburden loads will be omitted from the area occupied by the wall.

© 2026 ENERCALC, LLC

Reinforcing



Reinforcing Bar Size: Indicate the rebar size to consider for the bars that run parallel to the footing width.

Rebar Spacing: Provides an option to either specify an explicit value for the rebar spacing, or to specify the number of bars in a 12-inch length.

Rebar Center to Concrete Edge @ Top & Bottom: Specify the clear cover plus 1/2 the diameter of the rebar.

Note: The reinforcement layout for the top and bottom faces of the footing is assumed to be identical, with only the tension face reinforcement considered effective in flexural capacity calculations.

© 2026 ENERCALC, LLC

- The moment resulting from downward loads which create bottom face tension is compared to the moment capacity assuming the specified reinforcement bars are located on the bottom of the footing.
- The moment resulting from net uplift which creates top face tension is compared to the moment capacity assuming the same number and size of bars are also present in a top mat.

The Footing Bending tab clearly indicates which surface is in tension, aiding in the decision of whether a top mat is necessary.

Applied Loads

Wall Footing PRINT CANCEL CLONE SAVE SAVE & CLOSE

General | Soil Allowables | Footing & Reinforcing | Applied Loads | Load Combinations

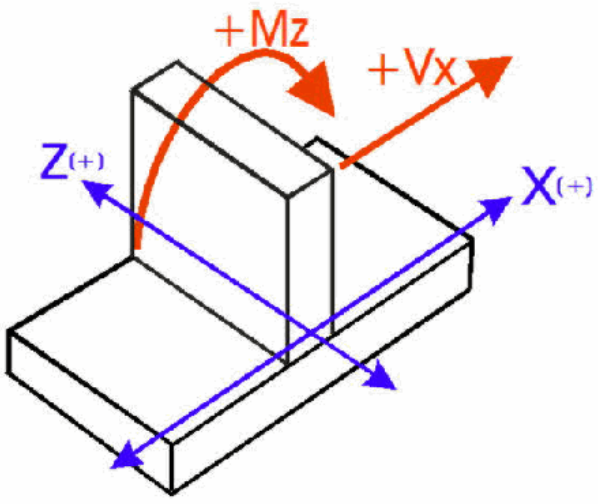
	Vertical Load * P-y		Overburden * OB-y		Moments M-zz		Shears V-x	
D : Dead Load	1.20	k	0.30	ksf	0.60	k-ft		k
Lr : Roof Live	0.60	k		ksf	0.30	k-ft		k
L : Live	0.80	k		ksf	0.40	k-ft		k
S : Snow	1.0	k		ksf	0.50	k-ft		k
W : Wind		k		ksf		k-ft	0.180	k
E : Seismic*		k		ksf		k-ft		k
H : Earth		k		ksf		k-ft		k

* " P " = Gravity load, "+" sign is downward.

Shear application above top of footing = 24.0 in

"OB" = Overburden load, "+" sign is downward, uniform load over footing but not in "wall" area.

Note : Wall Location is specified on "Footing Size" tab.



NOTE! 12" Long footing strip width used. Enter ALL Values Accordingly

© 2026 ENERCALC, LLC

Vertical Load

Provides input fields for vertical loads. Vertical loads are specified in units of kips per foot, and they are considered to act at the center of the width of the wall. Overburden loads are specified in units of kips per square foot, and they are considered to act on the top surface area of the footing, excluding the area occupied by the wall.

Overburden

Provides input fields for overburden pressures. Overburden pressures are specified in units of kips per square foot, and they are considered to act at the top surface area of the footing, excluding the area occupied by the wall.

Moments

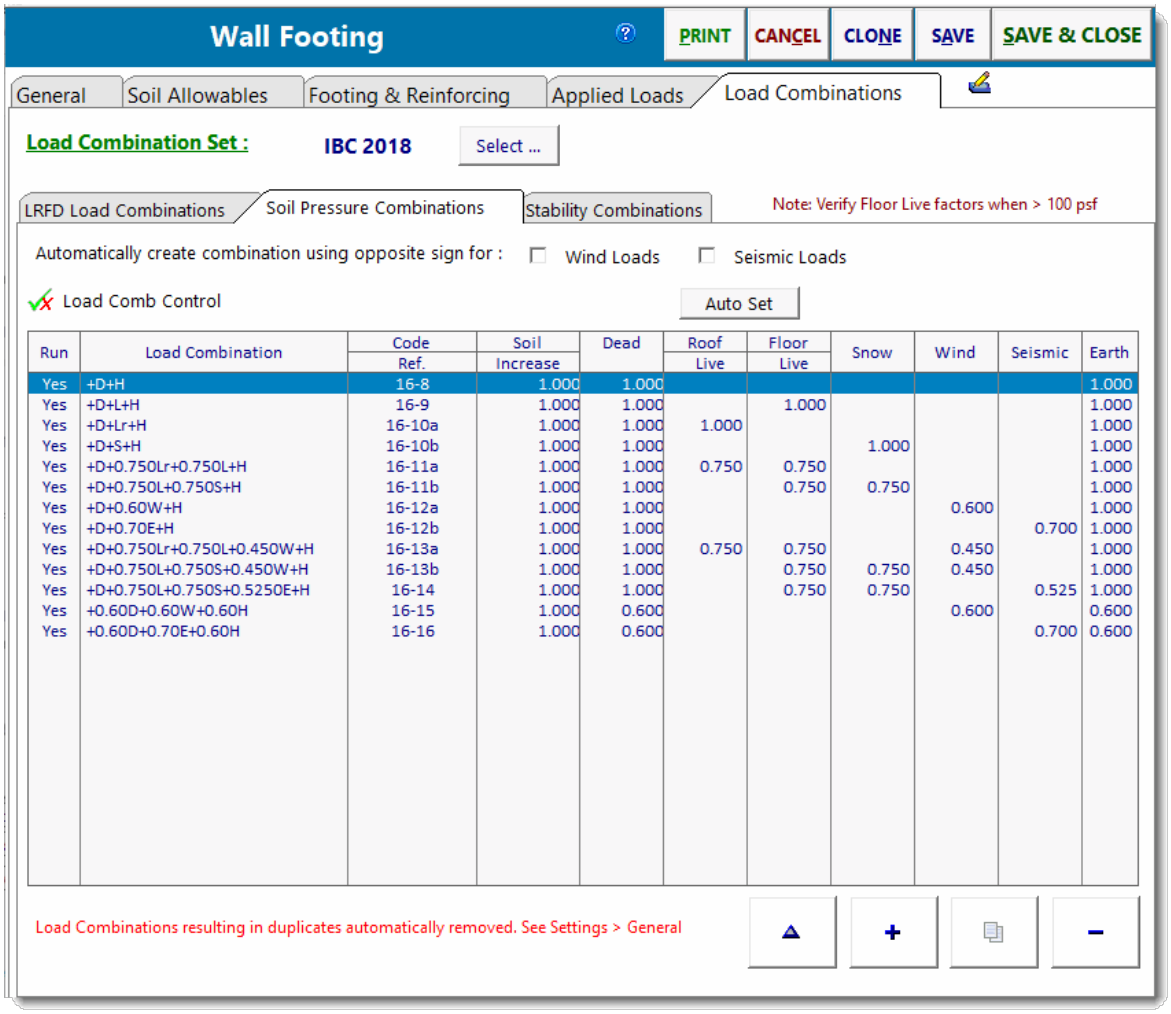
Provides input fields for moments. Moments are specified in units of foot kips per foot.

Shears

Provides input fields for shears. Shears are specified in units of kips per foot, and they are considered to act at the height specified in the field named **Shear application above top of footing**. Shears will produce a moment equal to the shear force times the distance from the bottom of the footing to the location of application of the shear force.

© 2026 ENERCALC, LLC

Load Combinations



Run	Load Combination	Code Ref.	Soil Increase	Dead	Roof Live	Floor Live	Snow	Wind	Seismic	Earth
Yes	+D+H	16-8	1.000	1.000						1.000
Yes	+D+L+H	16-9	1.000	1.000		1.000				1.000
Yes	+D+Lr+H	16-10a	1.000	1.000	1.000					1.000
Yes	+D+S+H	16-10b	1.000	1.000			1.000			1.000
Yes	+D+0.750Lr+0.750L+H	16-11a	1.000	1.000	0.750	0.750				1.000
Yes	+D+0.750L+0.750S+H	16-11b	1.000	1.000		0.750	0.750			1.000
Yes	+D+0.60W+H	16-12a	1.000	1.000				0.600		1.000
Yes	+D+0.70E+H	16-12b	1.000	1.000					0.700	1.000
Yes	+D+0.750Lr+0.750L+0.450W+H	16-13a	1.000	1.000	0.750	0.750		0.450		1.000
Yes	+D+0.750L+0.750S+0.450W+H	16-13b	1.000	1.000		0.750	0.750	0.450		1.000
Yes	+D+0.750L+0.750S+0.5250E+H	16-14	1.000	1.000		0.750	0.750		0.525	1.000
Yes	+0.60D+0.60W+0.60H	16-15	1.000	0.600				0.600		0.600
Yes	+0.60D+0.70E+0.60H	16-16	1.000	0.600					0.700	0.600

The Load Combinations tab is used to specify the load combinations to be used in the design. The LRFD Load Combinations tab controls the combinations that are used for reinforced concrete design checks. The Soil Pressure Combinations tab controls the combinations that are used for evaluating soil bearing pressure. A Soil Increase factor can be applied on a load combination by load combination basis, as permitted by the geotechnical engineering report. The Stability Combinations tab controls the load combinations that are used to perform the serviceability checks for Overturning, Sliding, and Uplift.

These tabs allow the user to select from load combination sets that are supplied with the program or to select from custom load combination sets that have been created and saved on the user's machine. It is also possible to unlock the selected load combination set and make edits to the factors directly in this view.

The user has control over which combinations are run and which are ignored. Finally, these tabs allow the user to specify whether the program should consider the algebraic sign of the specified load factors on wind loads and seismic loads to be reversible or not.

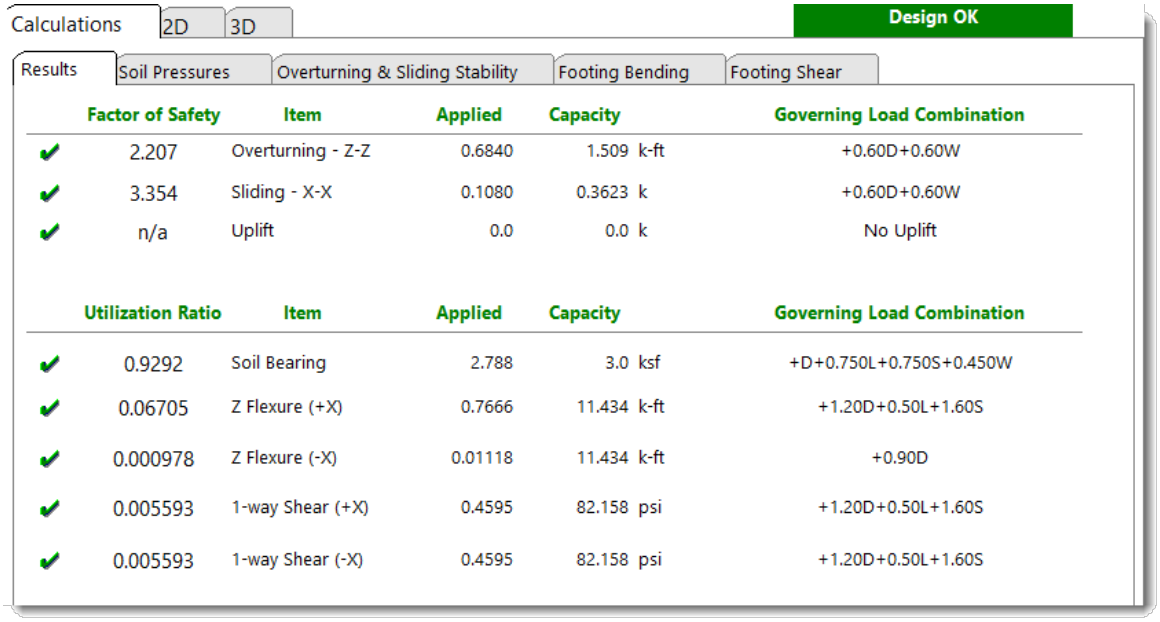
© 2026 ENERCALC, LLC

This can be a convenient way to ensure that these loads are investigated as acting in both positive and negative directions, if that is the design intent. Note, however, that if selected, the algebraic sign reversal will be applied to ALL wind loads and/or ALL seismic loads including horizontally AND vertically applied loads.

Calculations

Results

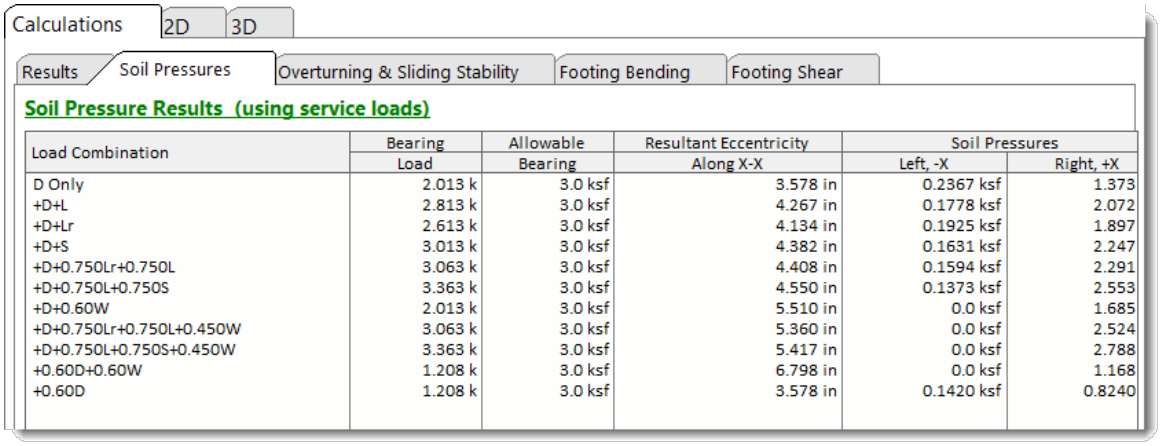
This tab summarizes the controlling values (highest utilization ratio) for each design consideration, from all of the load combinations that have been run. For the controlling load combination, it presents the Applied load, the Capacity or available resisting load, the ratio of the Applied to the Capacity, and the governing load combination that produces this controlling ratio.



© 2026 ENERCALC, LLC

Soil Pressures

For each service load combination, this tab presents the total vertical load, the resultant eccentricity, the soil pressures on the left and the right ends of the footing, the allowable soil pressure, and the ratio of the actual to the allowable soil pressure.

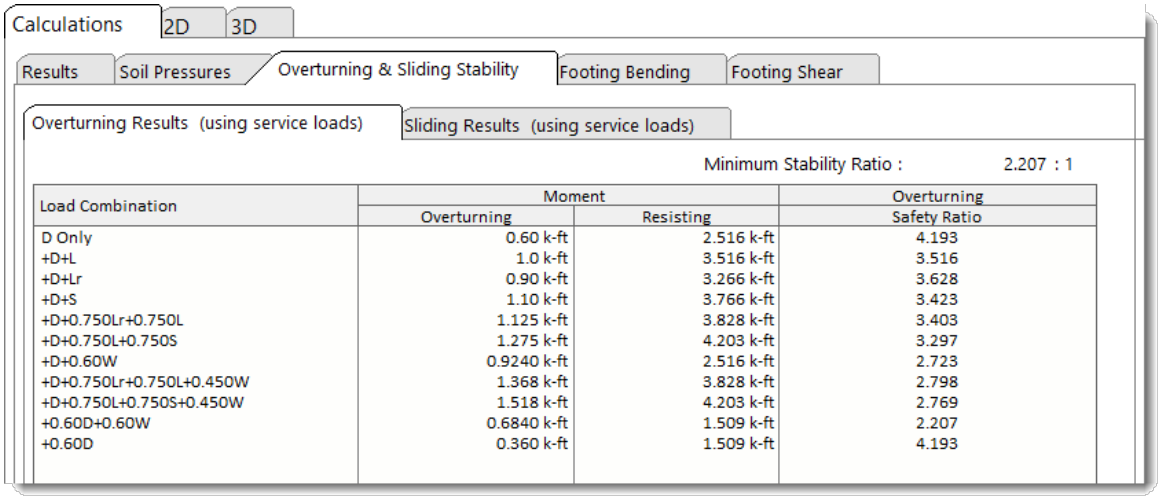


Soil Pressure Results (using service loads)

Load Combination	Bearing Load	Allowable Bearing	Resultant Eccentricity Along X-X	Soil Pressures Left, -X	Soil Pressures Right, +X
D Only	2.013 k	3.0 ksf	3.578 in	0.2367 ksf	1.373
+D+L	2.813 k	3.0 ksf	4.267 in	0.1778 ksf	2.072
+D+Lr	2.613 k	3.0 ksf	4.134 in	0.1925 ksf	1.897
+D+S	3.013 k	3.0 ksf	4.382 in	0.1631 ksf	2.247
+D+0.750Lr+0.750L	3.063 k	3.0 ksf	4.408 in	0.1594 ksf	2.291
+D+0.750L+0.750S	3.363 k	3.0 ksf	4.550 in	0.1373 ksf	2.553
+D+0.60W	2.013 k	3.0 ksf	5.510 in	0.0 ksf	1.685
+D+0.750Lr+0.750L+0.450W	3.063 k	3.0 ksf	5.360 in	0.0 ksf	2.524
+D+0.750L+0.750S+0.450W	3.363 k	3.0 ksf	5.417 in	0.0 ksf	2.788
+0.60D+0.60W	1.208 k	3.0 ksf	6.798 in	0.0 ksf	1.168
+0.60D	1.208 k	3.0 ksf	3.578 in	0.1420 ksf	0.8240

Overturning & Sliding Stability - Overturning Results

For each service load combination, this tab presents the overturning moment, the resisting moment and the ratio of the resisting to overturning moment about the left and right edges of the footing.



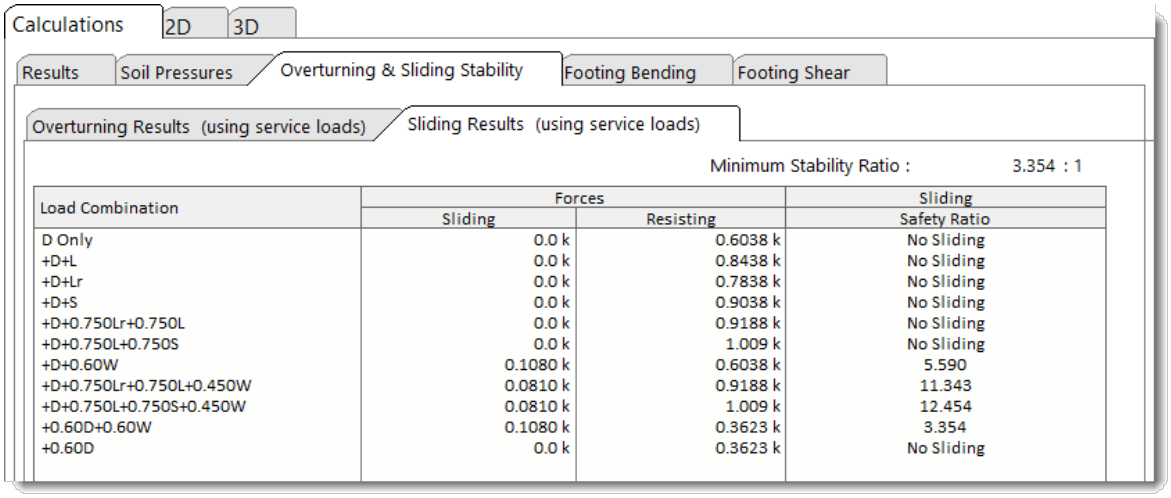
Minimum Stability Ratio : 2.207 : 1

Load Combination	Moment Overturning	Moment Resisting	Overturning Safety Ratio
D Only	0.60 k-ft	2.516 k-ft	4.193
+D+L	1.0 k-ft	3.516 k-ft	3.516
+D+Lr	0.90 k-ft	3.266 k-ft	3.628
+D+S	1.10 k-ft	3.766 k-ft	3.423
+D+0.750Lr+0.750L	1.125 k-ft	3.828 k-ft	3.403
+D+0.750L+0.750S	1.275 k-ft	4.203 k-ft	3.297
+D+0.60W	0.9240 k-ft	2.516 k-ft	2.723
+D+0.750Lr+0.750L+0.450W	1.368 k-ft	3.828 k-ft	2.798
+D+0.750L+0.750S+0.450W	1.518 k-ft	4.203 k-ft	2.769
+0.60D+0.60W	0.6840 k-ft	1.509 k-ft	2.207
+0.60D	0.360 k-ft	1.509 k-ft	4.193

© 2026 ENERCALC, LLC

Overturning & Sliding Stability - Sliding Results

For each service load combination, this tab presents the sliding force, the resisting force, and the ratio of the resisting to sliding force.

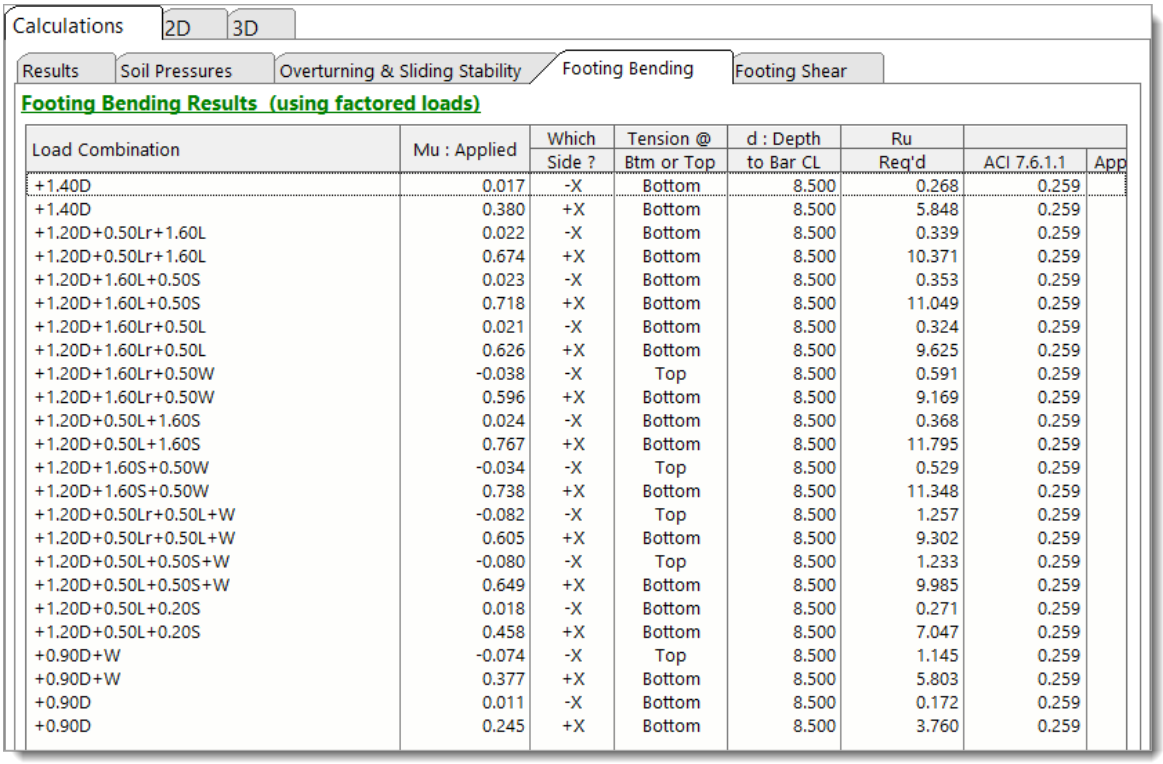


Load Combination	Forces		Sliding
	Sliding	Resisting	Safety Ratio
D Only	0.0 k	0.6038 k	No Sliding
+D+L	0.0 k	0.8438 k	No Sliding
+D+Lr	0.0 k	0.7838 k	No Sliding
+D+S	0.0 k	0.9038 k	No Sliding
+D+0.750Lr+0.750L	0.0 k	0.9188 k	No Sliding
+D+0.750L+0.750S	0.0 k	1.009 k	No Sliding
+D+0.60W	0.1080 k	0.6038 k	5.590
+D+0.750Lr+0.750L+0.450W	0.0810 k	0.9188 k	11.343
+D+0.750L+0.750S+0.450W	0.0810 k	1.009 k	12.454
+0.60D+0.60W	0.1080 k	0.3623 k	3.354
+0.60D	0.0 k	0.3623 k	No Sliding

© 2026 ENERCALC, LLC

Footing Bending

This tab reports the results of the flexural design on a load combination by load combination basis.



Footing Bending Results (using factored loads)

Load Combination	Mu : Applied	Which Side ?	Tension @ Btm or Top	d : Depth to Bar CL	Ru Req'd	ACI 7.6.1.1	App
+1.40D	0.017	-X	Bottom	8.500	0.268	0.259	
+1.40D	0.380	+X	Bottom	8.500	5.848	0.259	
+1.20D+0.50Lr+1.60L	0.022	-X	Bottom	8.500	0.339	0.259	
+1.20D+0.50Lr+1.60L	0.674	+X	Bottom	8.500	10.371	0.259	
+1.20D+1.60L+0.50S	0.023	-X	Bottom	8.500	0.353	0.259	
+1.20D+1.60L+0.50S	0.718	+X	Bottom	8.500	11.049	0.259	
+1.20D+1.60Lr+0.50L	0.021	-X	Bottom	8.500	0.324	0.259	
+1.20D+1.60Lr+0.50L	0.626	+X	Bottom	8.500	9.625	0.259	
+1.20D+1.60Lr+0.50W	-0.038	-X	Top	8.500	0.591	0.259	
+1.20D+1.60Lr+0.50W	0.596	+X	Bottom	8.500	9.169	0.259	
+1.20D+0.50L+1.60S	0.024	-X	Bottom	8.500	0.368	0.259	
+1.20D+0.50L+1.60S	0.767	+X	Bottom	8.500	11.795	0.259	
+1.20D+1.60S+0.50W	-0.034	-X	Top	8.500	0.529	0.259	
+1.20D+1.60S+0.50W	0.738	+X	Bottom	8.500	11.348	0.259	
+1.20D+0.50Lr+0.50L+W	-0.082	-X	Top	8.500	1.257	0.259	
+1.20D+0.50Lr+0.50L+W	0.605	+X	Bottom	8.500	9.302	0.259	
+1.20D+0.50L+0.50S+W	-0.080	-X	Top	8.500	1.233	0.259	
+1.20D+0.50L+0.50S+W	0.649	+X	Bottom	8.500	9.985	0.259	
+1.20D+0.50L+0.20S	0.018	-X	Bottom	8.500	0.271	0.259	
+1.20D+0.50L+0.20S	0.458	+X	Bottom	8.500	7.047	0.259	
+0.90D+W	-0.074	-X	Top	8.500	1.145	0.259	
+0.90D+W	0.377	+X	Bottom	8.500	5.803	0.259	
+0.90D	0.011	-X	Bottom	8.500	0.172	0.259	
+0.90D	0.245	+X	Bottom	8.500	3.760	0.259	

Footing Shear tab

This tab reports the results of the shear design on a load combination by load combination basis.

For footings designed per ACI 318-14 and earlier, and ACI 318-25 and later:

The nominal shear strength follows the one-way shear provisions of ACI 318-14 §22.5.5.1 / ACI 318-25 §13.2.6.2:

$\varphi v_n = \varphi * 2 * \lambda * \sqrt{(f'_c)}$, where $\varphi = 0.75$.

For footings designed per ACI 318-19:

The nominal shear strength follows the one-way shear provisions of §22.5 and Table 22.5.5.1 (referenced by §7.5.3.1):

$\varphi v_n = \varphi * 8 * \lambda_s * \lambda * (\rho_w)^{1/3} * \sqrt{(f'_c)}$, where $\varphi = 0.75$.

Note: The one-way shear capacity assumes no transverse reinforcement in the footing. Therefore, $A_v < A_{v,min}$, and equations (a) and (b) from Table 22.5.5.1 are not applicable.

© 2026 ENERCALC, LLC

It is also acceptable to neglect the size effect factor (λ_s) for one-way shallow foundations per §13.2.6.3, so $\lambda_s = 1.0$.

This value is also subject to additional limits per §22.5.5.1.1:

Lower bound: $\varphi * \lambda * \sqrt{(f'_c)}$

Upper bound: $\varphi * 5 * \lambda * \sqrt{(f'_c)}$

Calculations | 2D | 3D

Results | Soil Pressures | Overturning & Sliding Stability | Footing Bending | Footing Shear

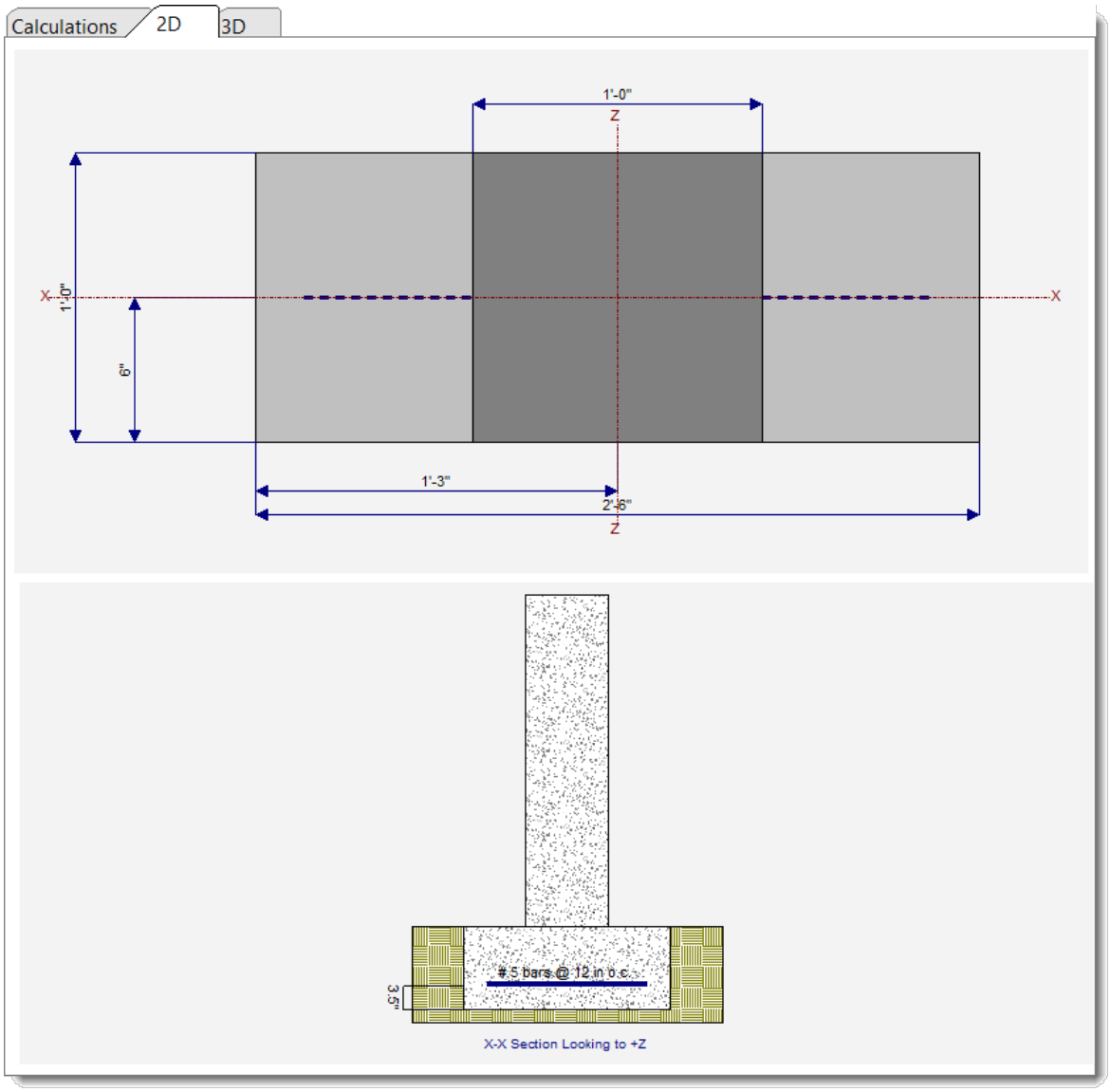
Footing Shear Results (using factored loads)

Load Combination	One Way Shear @ "d" from Face of Wall				
	vu @ -X (psi)	vu @ +X (psi)	vu Max (psi)	Φ vn (psi)	vu / Φ vn
+1.40D	0.3	0.3	0.3	47.6	0.007
+1.20D+0.50Lr+1.60L	0.6	0.6	0.6	47.6	0.013
+1.20D+1.60L+0.50S	0.6	0.6	0.6	47.6	0.014
+1.20D+1.60Lr+0.50L	0.6	0.6	0.6	47.6	0.012
+1.20D+1.60Lr+0.50W	0.5	0.5	0.5	47.6	0.010
+1.20D+0.50L+1.60S	0.7	0.7	0.7	47.6	0.014
+1.20D+1.60S+0.50W	0.6	0.6	0.6	47.6	0.013
+1.20D+0.50Lr+0.50L+W	0.4	0.4	0.4	47.6	0.009
+1.20D+0.50L+0.50S+W	0.5	0.5	0.5	47.6	0.010
+1.20D+0.50L+0.70S	0.5	0.5	0.5	47.6	0.011
+0.90D+W	0.2	0.2	0.2	47.6	0.005
+0.90D	0.2	0.2	0.2	47.6	0.005

© 2026 ENERCALC, LLC

2D Sketch

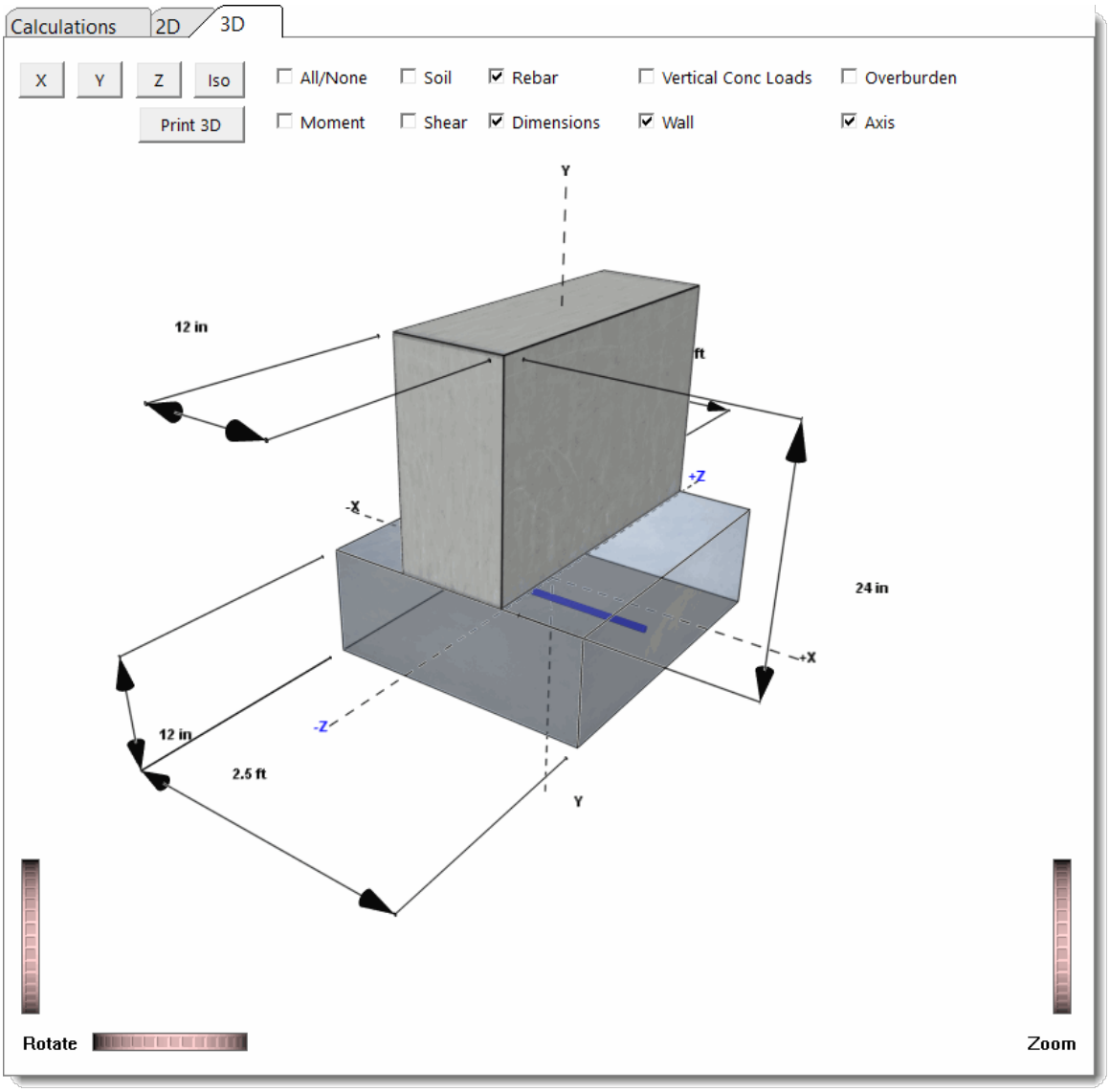
This tab presents plan and section views of the footing:



© 2026 ENERCALC, LLC

3D Rendering

This tab presents the 3D rendering of the footing:



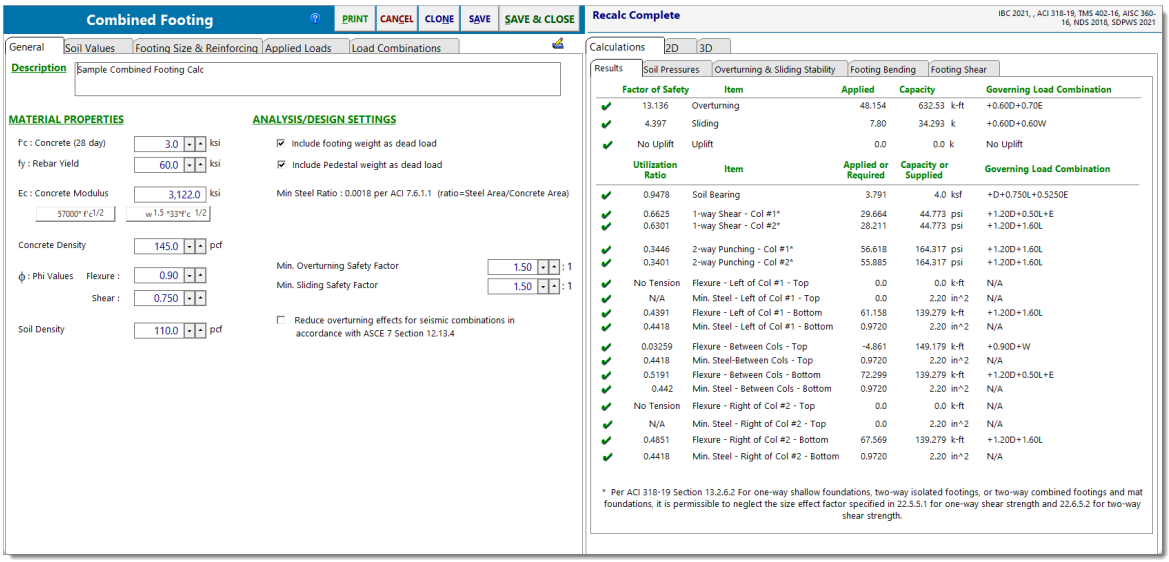
13.6.4 Combined Footing

This module provides analysis of a rectangular footing with two applied axial, moment and shear loads. Overburden loads can also be specified, and will apply to the surface area of the footing (except the areas covered by the two piers). The module allows you to position the application of the pier loads as necessary, and provides automatic calculation of allowable soil bearing pressure increases based on footing dimensions and/or depth below surface.

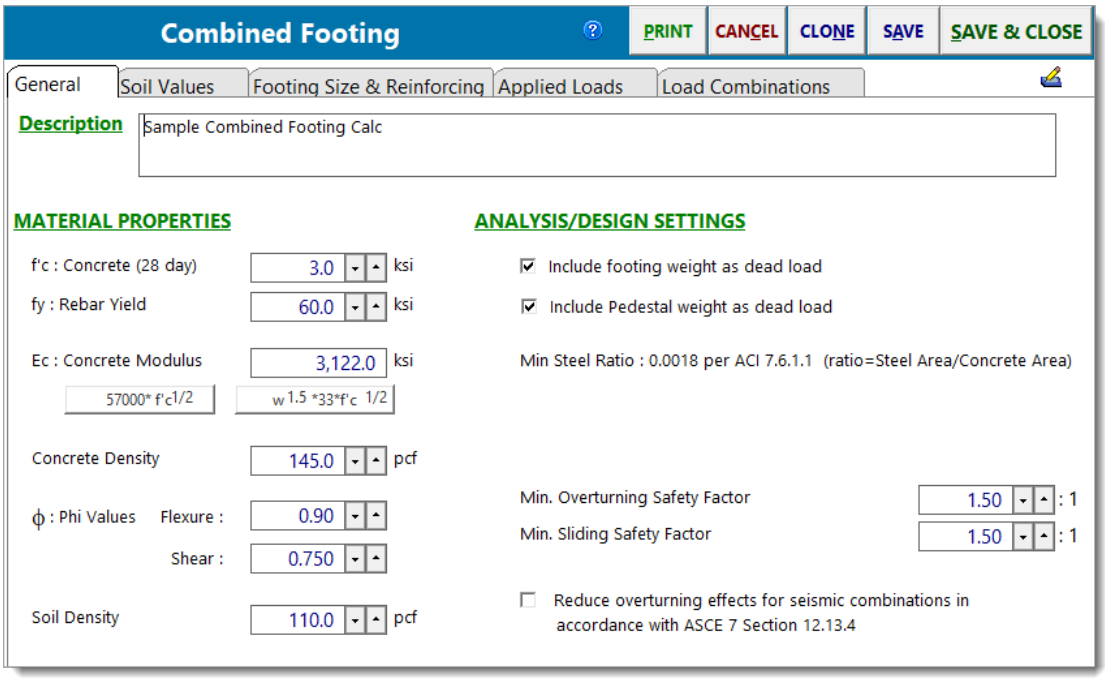
The module checks service load soil pressure, overturning stability, sliding stability, uplift stability, flexure left and right of each pedestal, 1-way shear at 'd' from each of the

© 2026 ENERCALC, LLC

pedestals, and punching shear along a perimeter located 'd/2' from the pedestal faces. The module does not evaluate the footing for flexure about the Length axis.



General



f'c

28-day compressive strength of the concrete.

fy

Yield point stress of reinforcing.

© 2026 ENERCALC, LLC

Ec

Modulus of elasticity of concrete.

Concrete Density

The density of the concrete is used to calculate the self weight of the pedestals and footing when those options are selected. Note that code modifications for lightweight concrete are not applied in this module. The purpose of this input is mainly to allow the user to specify something in the range of 145 to 150 pcf.

Phi Values

Enter the capacity reduction values to be applied to Vn and Mn.

Soil Density

Enter the density of soil in units of lb per cubic ft.

Include footing weight as dead load

Click [**Yes**] to have the module calculate the weight of the footing and apply it as a downward load. The footing self weight will be multiplied by the dead load factor in each load combination.

Include pedestal weight as dead load

Click [**Yes**] to have the module calculate the weight of the pedestals and apply it as downward loads. The pedestal self weight will be multiplied by the dead load factor in each load combination.

Min Steel Ratio

This is set to 0.0018 per ACI 7.6.1.1. A warning message will be triggered if the section is under-reinforced.

NOTE: This is not T&S steel. It is a minimum that must be met for flexural steel. This means that if the footing has net uplift and requires a top mat of reinforcing, this ratio also applies to the area of steel provided in the top mat. It is not something that can be "shared" between the top and bottom mats like T&S steel can.

Minimum Overturning Safety Factor

Enter the minimum allowable ratio of resisting moment to overturning moment. If the actual ratio is less than the specified minimum ratio, it will trigger a message that overturning stability is not satisfied.

Minimum Sliding Safety Factor

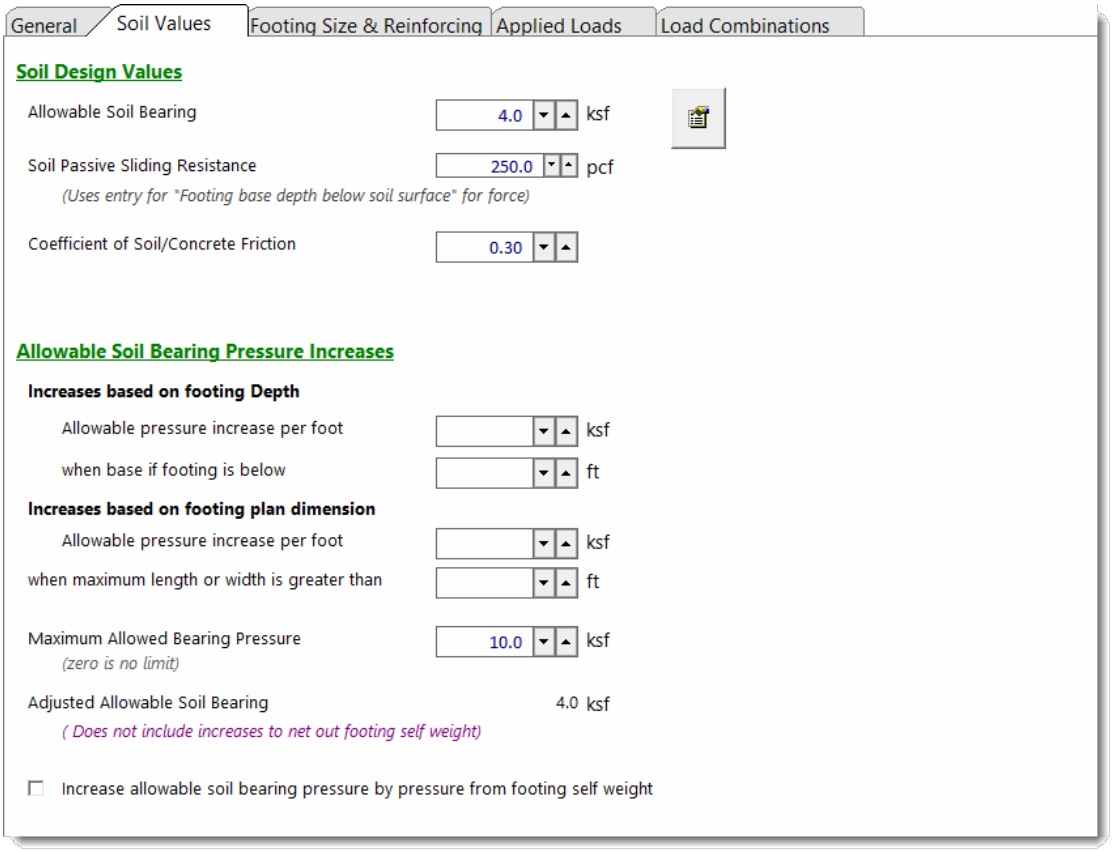
Enter the minimum allowable ratio of resisting force to sliding force. If the actual ratio is less than the specified minimum ratio, it will trigger a message that sliding stability is not satisfied.

Reduce overturning effects for seismic combinations in accordance with ASCE 7 Section 12.13.4

Select this option to apply the provisions of the referenced section of ASCE 7.

© 2026 ENERCALC, LLC

Soil Values



Allowable Soil Bearing

Enter the allowable soil bearing pressure that the soil can resist. This is a service load resistance and will be compared to calculated service load soil pressures.

Soil Passive Sliding Resistance

Enter the value of passive soil pressure resistance to sliding. This value will be used to determine a component of sliding resistance that is generated by the passive pressure of the soil. The sliding resistance due to passive pressure is then added to the sliding resistance due to friction to determine the total resistance to sliding for each load combination.

Coefficient of Soil/Concrete Friction

Enter the coefficient of friction between soil and footing to use in sliding resistance calculations.

© 2026 ENERCALC, LLC

Soil Bearing Increase

This section allows you to specify some dimensions that, when exceeded, will automatically increase the allowable soil bearing pressure.

Increases based on footing depth: Provides a method to automatically apply increases to the basic allowable soil bearing pressure based on footing depth below some reference depth. Collects the following parameters:

Allowable pressure increase per foot: Specifies the amount that the basic allowable soil bearing pressure can be increased for each foot of depth below some reference depth.

When base of footing is below: Specifies the required depth in order to start realizing incremental increases in the allowable soil bearing pressure on the basis of footing depth.

Example: Assume the following: Basic Allowable Soil Bearing Pressure = 3 ksf. Footing base is 6'-0" below soil surface. The Geotechnical report specifies that a 0.15 ksf increase in bearing pressure is allowed for each foot of depth when the base is deeper than 4' below top of soil. Since you've indicated that the footing is 6' below the soil surface, the module will automatically calculate the adjusted allowable soil bearing pressure to be 3 ksf + (6' - 4') * 0.15 ksf = 3.30 ksf.

Increases based on footing plan dimension: Provides a method to automatically apply increases to the basic allowable soil bearing pressure based on footing dimensions greater than some reference dimension. Collects the following parameters:

Allowable pressure increase per foot: Specifies the amount that the basic allowable soil bearing pressure can be increased for each foot of width or length greater than some reference dimension.

When maximum length or width is greater than: Specifies the required dimension in order to start realizing incremental increases in the allowable soil bearing pressure on the basis of footing dimension.

Example: Assume the following: Basic Allowable Soil Bearing Pressure = 3 ksf. Footing measures 12'-0" x 6'-0". The Geotechnical report specifies that a 0.15 ksf increase in soil bearing pressure is allowed for each foot when the largest plan dimension of the footing is greater than 4'. The module will automatically calculate the adjusted allowable soil bearing pressure to be 3 ksf + (12' - 4') * 0.15 ksf = 4.2 ksf.

Maximum Allowed Bearing Pressure: Provides a way to specify an upper limit on the adjusted allowable bearing pressure.

Note: Increases based on footing depth and plan dimensions are cumulative.

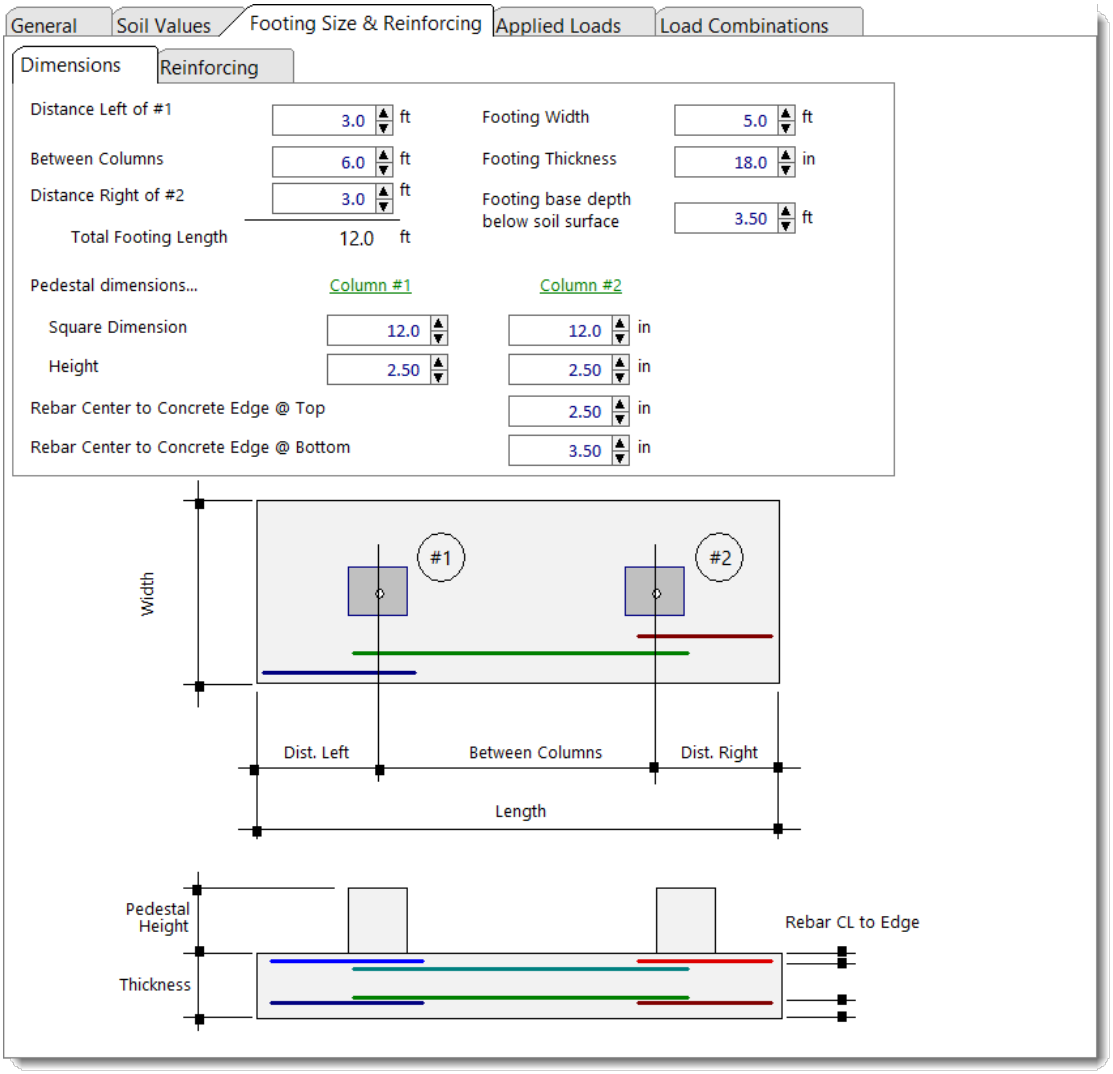
© 2026 ENERCALC, LLC

Increase Bearing by Footing Weight

Click [**Yes**] to tell the module to calculate the weight of one square foot (plan view) of footing and add it to the allowable soil bearing value. This has the effect of not penalizing the soil for the self weight of the footing, and is useful for situations where the geotechnical engineering report provides allowable net bearing pressures.

Footing Size & Reinforcing

Dimensions tab



Projection on Left, Distance Between Columns, Projection on Right: Define the dimensions of the footing in the Length direction.

Footing Width: Define the dimension of the Width direction.

Footing Thickness: Define the total thickness of the footing.

© 2026 ENERCALC, LLC

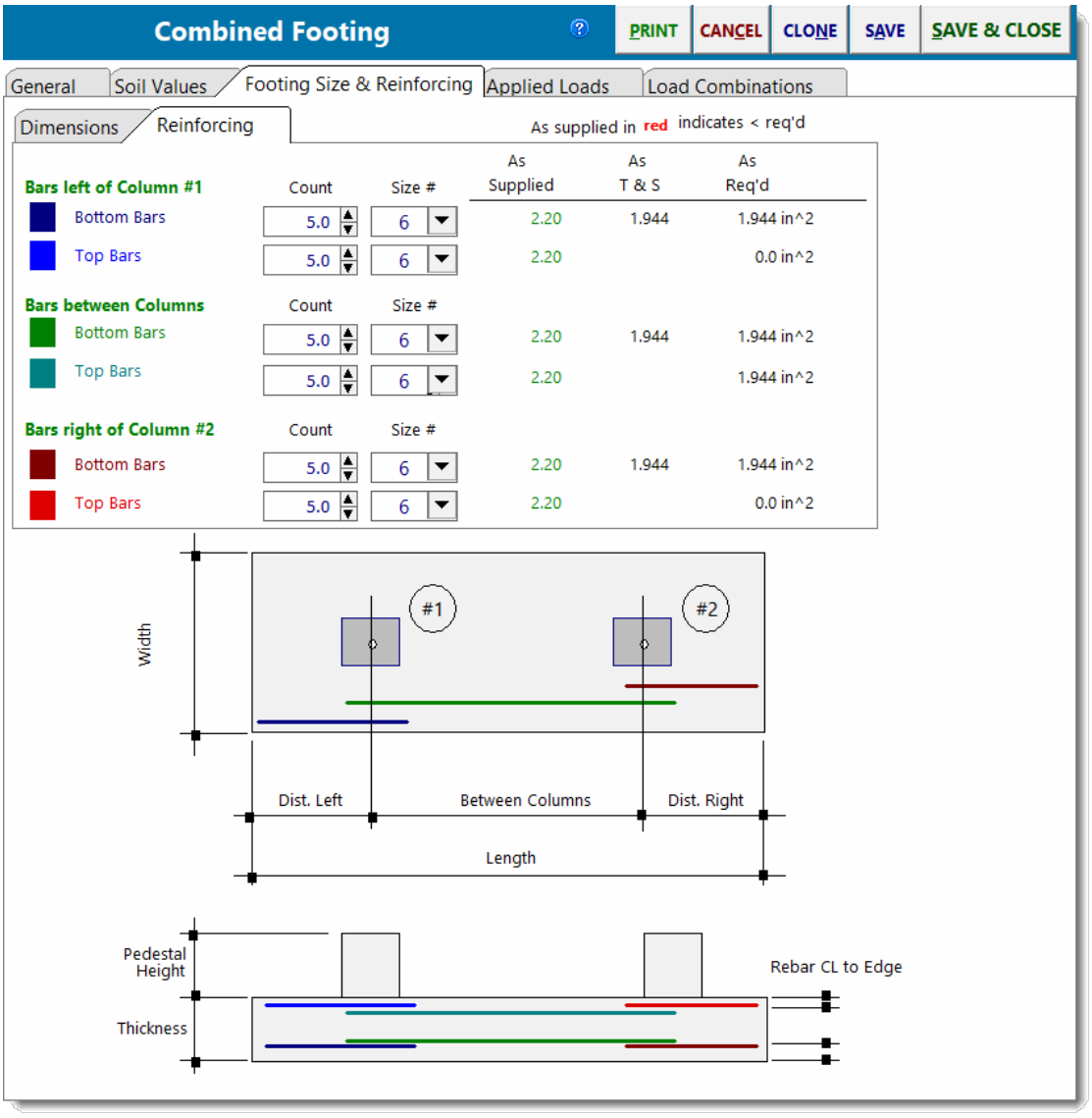
Footing base depth below soil surface: The distance from the bottom of the footing to the top of the soil. This value is used to determine allowable soil bearing pressure increases and soil passive sliding resistance, but it is not used in any other calculations in this module.

Pedestal dimensions: If concrete pedestals bear on the footing, their dimensions can be specified here. Pedestals are assumed to be square, and they are assumed to be centered on the Width dimension of the footing.

Note: Any applied overburden loads will be omitted from the area occupied by the pedestals.

© 2026 ENERCALC, LLC

Reinforcing tab

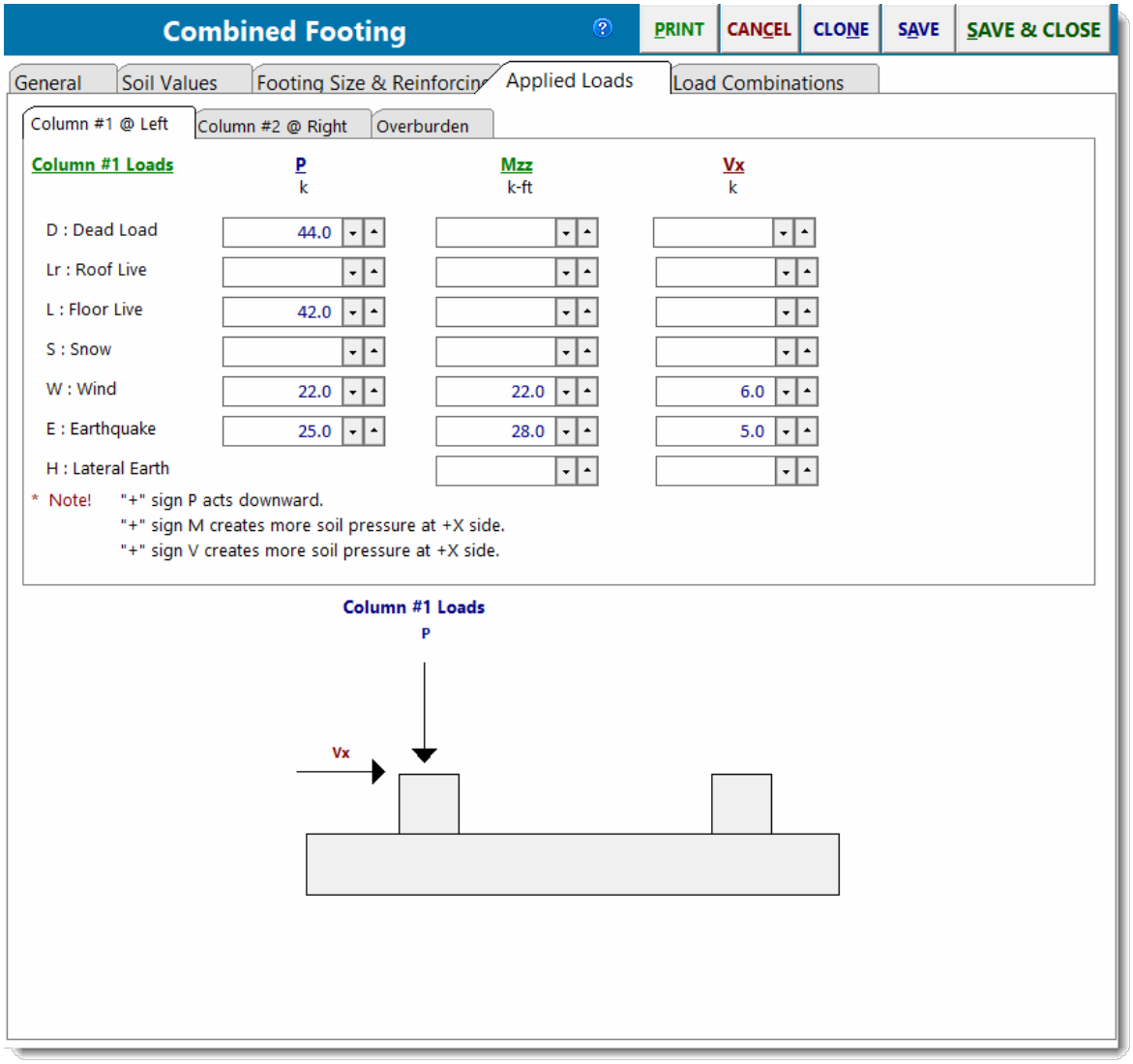


Reinforcing parallel to the Length dimension can be defined separately for the left and right projections of the footing and for the area between the columns. Input fields are provided to define top bars and bottom bars separately.

Note: Bars are assumed to be fully developed at the locations where they are required. It is the engineer's responsibility to validate that assumption. The program is not taking rebar development length into consideration.

© 2026 ENERCALC, LLC

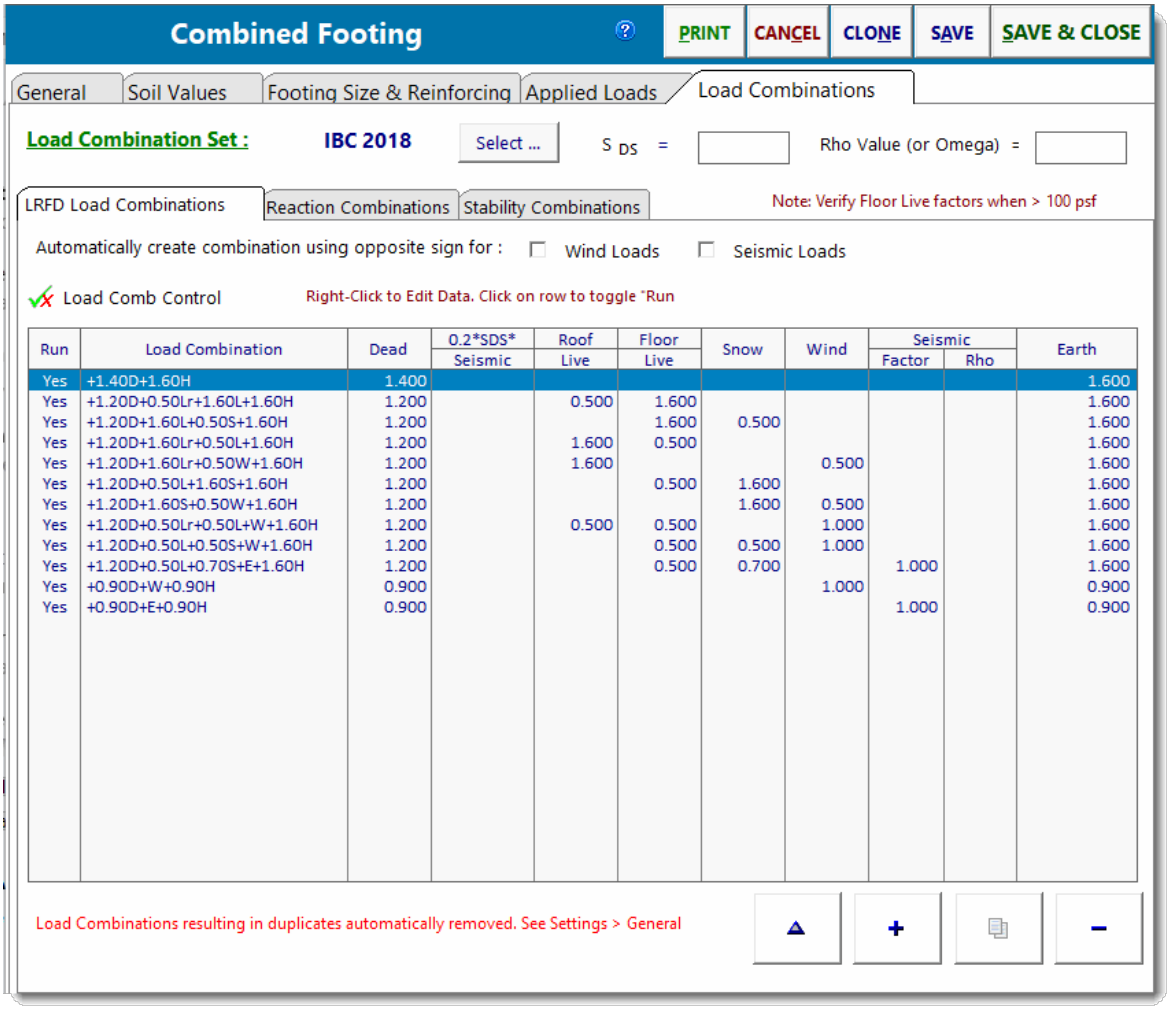
Applied Loads



The Applied Loads tab provides sub-tabs for Column #1 (the column on the left), Column #2 (the column on the right), and Overburden. The two column load tabs offer input fields for vertical loads, moment about the Width axis, and shear in the Length direction. The Overburden tab provides input fields for a uniform vertical pressure that will be applied to the entire surface area of the footing with the exception of the areas occupied by the pedestals.

© 2026 ENERCALC, LLC

Load Combinations



Run	Load Combination	Dead	0.2*SDS* Seismic	Roof Live	Floor Live	Snow	Wind	Seismic Factor	Seismic Rho	Earth
Yes	+1.40D+1.60H	1.400								1.600
Yes	+1.20D+0.50Lr+1.60L+1.60H	1.200		0.500	1.600					1.600
Yes	+1.20D+1.60L+0.50S+1.60H	1.200			1.600	0.500				1.600
Yes	+1.20D+1.60Lr+0.50L+1.60H	1.200		1.600	0.500					1.600
Yes	+1.20D+1.60Lr+0.50W+1.60H	1.200		1.600			0.500			1.600
Yes	+1.20D+0.50L+1.60S+1.60H	1.200			0.500	1.600				1.600
Yes	+1.20D+1.60S+0.50W+1.60H	1.200				1.600	0.500			1.600
Yes	+1.20D+0.50Lr+0.50L+W+1.60H	1.200		0.500	0.500		1.000			1.600
Yes	+1.20D+0.50L+0.50S+W+1.60H	1.200			0.500	0.500	1.000			1.600
Yes	+1.20D+0.50L+0.70S+E+1.60H	1.200			0.500	0.700		1.000		1.600
Yes	+0.90D+W+0.90H	0.900					1.000			0.900
Yes	+0.90D+E+0.90H	0.900						1.000		0.900

The Load Combinations tab is used to specify the load combinations to be used in the design. The Service Combinations tab controls the load combinations that are used to perform the serviceability checks for Soil Bearing, Overturning, Sliding, and Uplift. The Factored Combinations tab controls the load combinations that are used to perform the strength checks for Flexure, One-Way Shear, and Two-Way Punching Shear.

These tabs allow the user to select from load combination sets that are supplied with the program or to select from custom load combination sets that have been created and saved on the user's machine. It is also possible to unlock the selected load combination set and make edits to the factors directly in this view. The user has control over which combinations are run and which are ignored. A Soil Increase factor can be applied on a load combination by load combination basis, as permitted by the geotechnical engineering report.

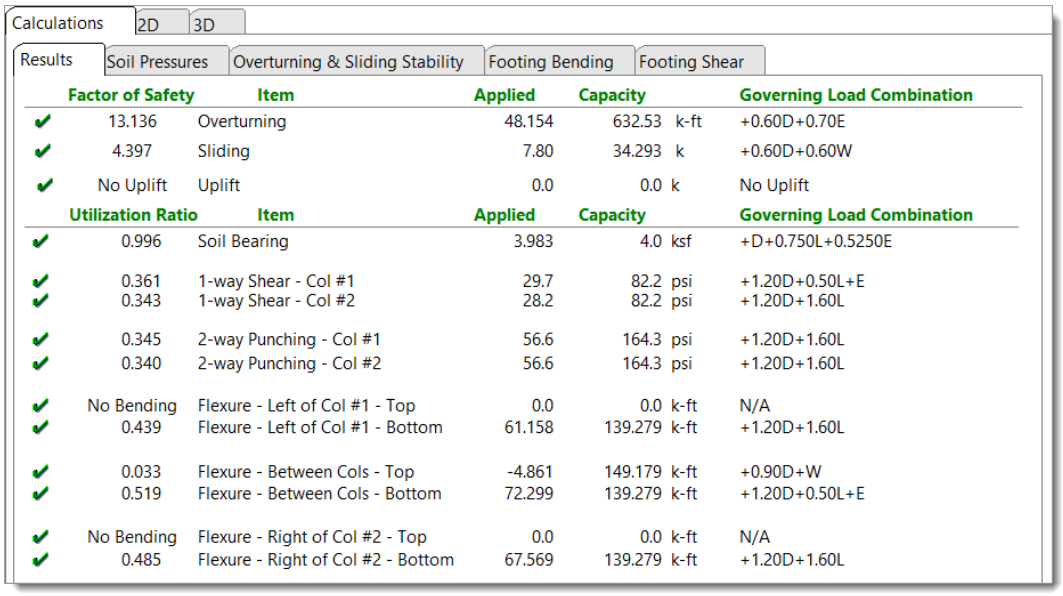
Finally, this tab allows the user to specify whether the program should consider the algebraic sign of the specified load factors on wind loads and seismic loads to be reversible or not. This can be a convenient way to ensure that these loads are investigated

© 2026 ENERCALC, LLC

as acting in both positive and negative directions, if that is the design intent. Note, however, that if selected, the algebraic sign reversal will be applied to ALL wind loads and/or ALL seismic loads including horizontally AND vertically applied loads.

Calculations

Results



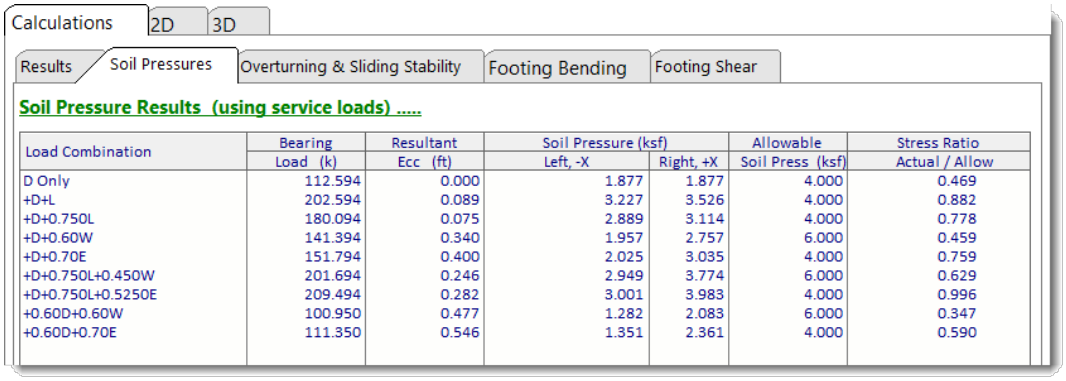
Calculations | 2D | 3D

Results | Soil Pressures | Overturning & Sliding Stability | Footing Bending | Footing Shear

Factor of Safety	Item	Applied	Capacity		Governing Load Combination
✔ 13.136	Overturning	48.154	632.53	k-ft	+0.60D+0.70E
✔ 4.397	Sliding	7.80	34.293	k	+0.60D+0.60W
✔ No Uplift	Uplift	0.0	0.0	k	No Uplift
Utilization Ratio	**Item**	**Applied**	**Capacity**		**Governing Load Combination**
✔ 0.996	Soil Bearing	3.983	4.0	ksf	+D+0.750L+0.5250E
✔ 0.361	1-way Shear - Col #1	29.7	82.2	psi	+1.20D+0.50L+E
✔ 0.343	1-way Shear - Col #2	28.2	82.2	psi	+1.20D+1.60L
✔ 0.345	2-way Punching - Col #1	56.6	164.3	psi	+1.20D+1.60L
✔ 0.340	2-way Punching - Col #2	56.6	164.3	psi	+1.20D+1.60L
✔ No Bending	Flexure - Left of Col #1 - Top	0.0	0.0	k-ft	N/A
✔ 0.439	Flexure - Left of Col #1 - Bottom	61.158	139.279	k-ft	+1.20D+1.60L
✔ 0.033	Flexure - Between Cols - Top	-4.861	149.179	k-ft	+0.90D+W
✔ 0.519	Flexure - Between Cols - Bottom	72.299	139.279	k-ft	+1.20D+0.50L+E
✔ No Bending	Flexure - Right of Col #2 - Top	0.0	0.0	k-ft	N/A
✔ 0.485	Flexure - Right of Col #2 - Bottom	67.569	139.279	k-ft	+1.20D+1.60L

This tab summarizes the controlling values (highest utilization ratio) for each design consideration, from all of the load combinations that have been run. For the controlling load combination, it presents the Applied load, the Capacity or available resisting load, the ratio of the Applied to the Capacity, and the governing load combination that produces this controlling ratio.

Soil Pressures



Calculations | 2D | 3D

Results | Soil Pressures | Overturning & Sliding Stability | Footing Bending | Footing Shear

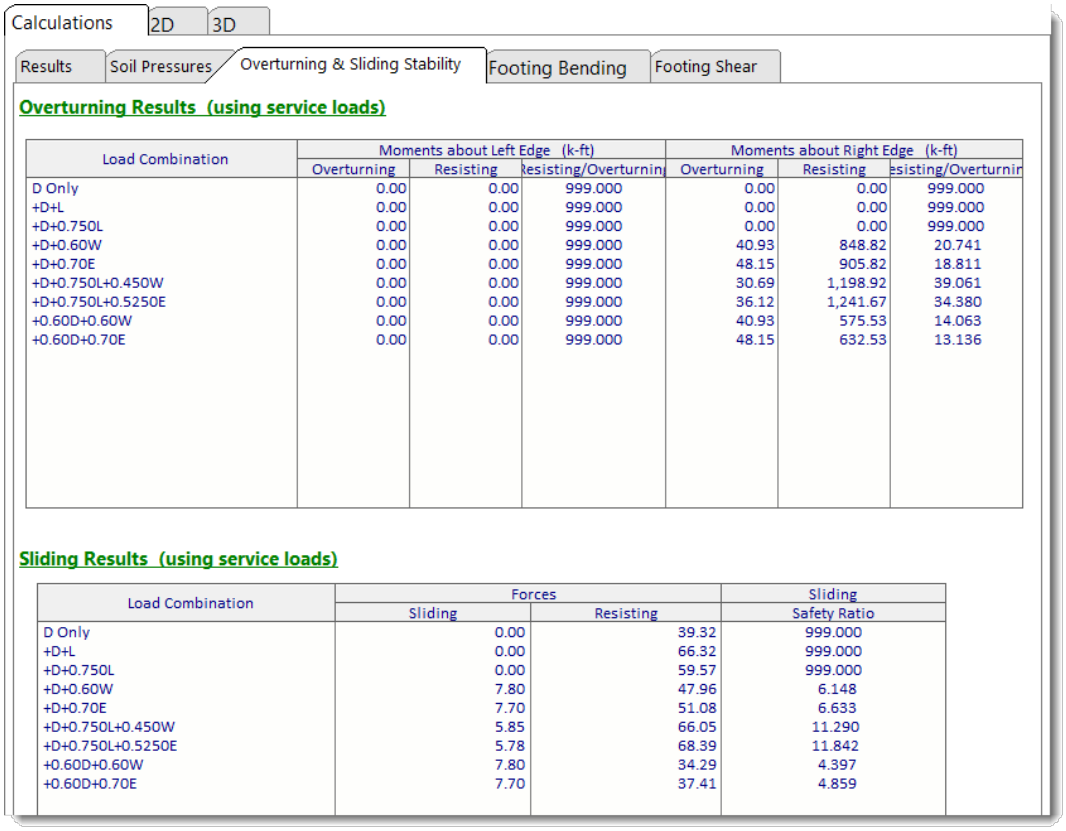
Soil Pressure Results (using service loads) .....

Load Combination	Bearing Load (k)	Resultant Ecc (ft)	Soil Pressure (ksf) Left, -X	Right, +X	Allowable Soil Press (ksf)	Stress Ratio Actual / Allow
D Only	112.594	0.000	1.877	1.877	4.000	0.469
+D+L	202.594	0.089	3.227	3.526	4.000	0.882
+D+0.750L	180.094	0.075	2.889	3.114	4.000	0.778
+D+0.60W	141.394	0.340	1.957	2.757	6.000	0.459
+D+0.70E	151.794	0.400	2.025	3.035	4.000	0.759
+D+0.750L+0.450W	201.694	0.246	2.949	3.774	6.000	0.629
+D+0.750L+0.5250E	209.494	0.282	3.001	3.983	4.000	0.996
+0.60D+0.60W	100.950	0.477	1.282	2.083	6.000	0.347
+0.60D+0.70E	111.350	0.546	1.351	2.361	4.000	0.590

© 2026 ENERCALC, LLC

For each service load combination, this tab presents the total vertical load, the resultant eccentricity, the soil pressures on the left and the right ends of the footing, the allowable soil pressure, and the ratio of the actual to the allowable soil pressure.

Overturning & Sliding Stability

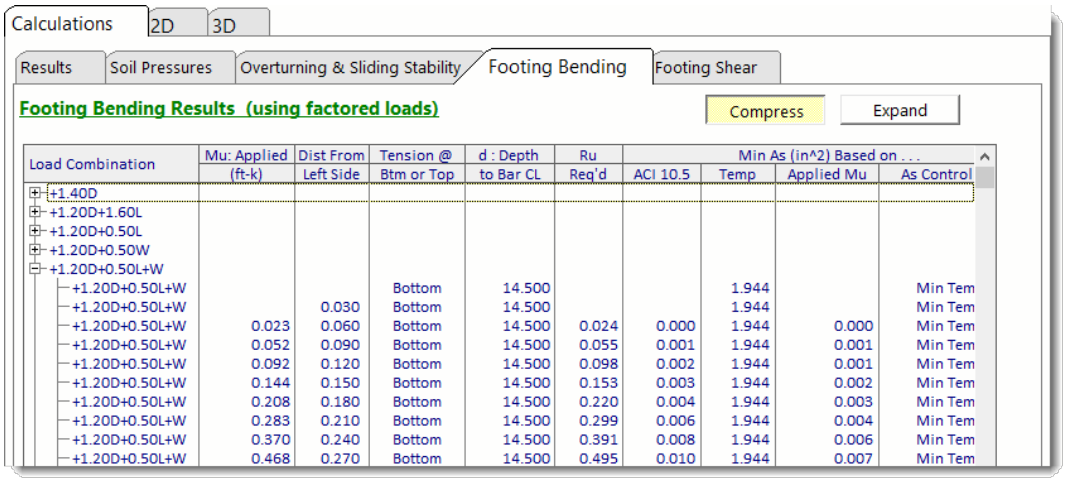


For each service load combination, this tab presents the overturning moment, the resisting moment and the ratio of the resisting to overturning moment about the left and right edges of the footing. It also reports the sliding force, the resisting force, and the ratio of the resisting to sliding force.

Note that the program is set up to look for overturning and resisting forces individually. For example, take the situation where the footing is subjected to equal and opposite shears at a given elevation. Common sense dictates that these forces cancel each other, and the footing experiences no net applied overturning moment from them. But the program treats one of the two equal and opposite forces as an overturning force, and the other as a resisting force. So for these two forces, there IS a net overturning moment reported, but the resisting moment ALSO considers the effect of the opposing load, so the accounting used to determine the overturning ratio is proper.

© 2026 ENERCALC, LLC

Footing Bending



Load Combination	Mu: Applied (ft-k)	Dist From Left Side	Tension @ Btm or Top	d : Depth to Bar CL	Ru Req'd	Min As (in^2) Based on . . . ACI 10.5	Temp	Applied Mu	As Control
+1.40D									
+1.20D+1.60L									
+1.20D+0.50L									
+1.20D+0.50W									
+1.20D+0.50L+W									
+1.20D+0.50L+W			Bottom	14.500			1.944		Min Tem
+1.20D+0.50L+W		0.030	Bottom	14.500			1.944		Min Tem
+1.20D+0.50L+W	0.023	0.060	Bottom	14.500	0.024	0.000	1.944	0.000	Min Tem
+1.20D+0.50L+W	0.052	0.090	Bottom	14.500	0.055	0.001	1.944	0.001	Min Tem
+1.20D+0.50L+W	0.092	0.120	Bottom	14.500	0.098	0.002	1.944	0.001	Min Tem
+1.20D+0.50L+W	0.144	0.150	Bottom	14.500	0.153	0.003	1.944	0.002	Min Tem
+1.20D+0.50L+W	0.208	0.180	Bottom	14.500	0.220	0.004	1.944	0.003	Min Tem
+1.20D+0.50L+W	0.283	0.210	Bottom	14.500	0.299	0.006	1.944	0.004	Min Tem
+1.20D+0.50L+W	0.370	0.240	Bottom	14.500	0.391	0.008	1.944	0.006	Min Tem
+1.20D+0.50L+W	0.468	0.270	Bottom	14.500	0.495	0.010	1.944	0.007	Min Tem

This tab reports the results of the flexural design on a load combination by load combination basis, at small increments along the length of the footing.

Footing Shear

This tab reports the results of the one-way shear design to the left and right of each column and the two-way shear design for each column on a load combination by load combination basis.

One-Way Shear

© 2026 ENERCALC, LLC

Calculations | 2D | 3D

Results | Soil Pressures | Overturning & Sliding Stability | Footing Bending | Footing Shear

Footing Shear Results (using factored loads) .....

Load Combination	One Way Shear @ "d" from Edge of Column #1								
	Vu Left (k)	vu Left (psi)	Φ vn Left (psi)	Ratio	Vu Right (k)	vu Right (psi)	Φ vn Right (psi)	Ratio	Vu L (k
+1.40D	12.253	13.2	44.8	0.294	12.255	13.2	50.1	0.263	
+1.20D+1.60L	23.607	25.4	44.8	0.567	23.560	25.3	50.1	0.506	2
+1.20D+0.50L	14.598	15.7	44.8	0.351	14.584	15.7	50.1	0.313	1
+1.20D+0.50W	11.098	11.9	44.8	0.267	16.716	18.0	50.1	0.359	1
+1.20D+0.50L+W	15.788	17.0	44.8	0.379	27.009	29.0	50.1	0.580	1
+1.20D+0.50L+E	16.292	17.5	44.8	0.391	27.599	29.7	50.1	0.592	1
+0.90D+W	9.068	9.8	44.8	0.218	20.303	21.8	50.1	0.436	1
+0.90D+E	9.571	10.3	44.8	0.230	20.893	22.5	50.1	0.449	1

Calculations | 2D | 3D

Results | Soil Pressures | Overturning & Sliding Stability | Footing Bending | Footing Shear

Footing Shear Results (using factored loads) .....

Load Combination	One Way Shear @ "d" from Edge of Column #2								Perime
	Vu Left (k)	vu Left (psi)	Φ vn Left (psi)	Ratio	Vu Right (k)	vu Right (psi)	Φ vn Right (psi)	Ratio	Pu C (k
+1.40D	9.755	10.5	50.1	0.209	9.747	10.5	41.6	0.252	4
+1.20D+1.60L	20.942	22.5	50.1	0.450	21.167	22.8	41.6	0.548	9
+1.20D+0.50L	12.293	13.2	50.1	0.264	12.359	13.3	41.6	0.320	5
+1.20D+0.50W	11.758	12.6	50.1	0.252	6.390	6.9	41.6	0.165	5
+1.20D+0.50L+W	19.085	20.5	49.0	0.419	8.429	9.1	41.6	0.218	7
+1.20D+0.50L+E	19.967	21.5	50.1	0.429	9.267	10.0	41.6	0.240	8
+0.90D+W	13.063	14.0	49.0	0.287	2.337	2.5	41.6	0.060	5
+0.90D+E	13.945	15.0	49.0	0.306	3.175	3.4	41.6	0.082	5

The maximum factored one-way shear stress is calculated as follows: $v_u = V_u / (B * d)$

Where:

- V_u = Maximum one-way shear force calculated at a distance of "d" from the left and right faces of each column.
- B = [Footing Width] as input on the "Footing Size & Reinforcing" tab.
- d = Distance from extreme compression fiber to centroid of tension reinforcement as reported on the "Footing Bending" tab at the associated critical section for shear.

<u>For footings designed per ACI 318-14 and earlier, and ACI 318-25 and later:</u>

The nominal shear strength follows the one-way shear provisions of ACI 318-14 §22.5.5.1 / ACI 318-25 §13.2.6.2:
$\varphi v_n = \varphi * 2 * \lambda * \sqrt{(f'_c)}$, where φ is specified by the user on the *General* tab.

<u>For footings designed per ACI 318-19:</u>

The nominal shear strength follows the one-way shear provisions of §22.5 and Table 22.5.5.1 (referenced by §7.5.3.1):
$\varphi v_n = \varphi * 8 * \lambda_s * \lambda * (\rho_w)^{1/3} * \sqrt{(f'_c)}$, where φ is specified by the user on the *General* tab.

© 2026 ENERCALC, LLC

Note: The one-way shear capacity assumes no transverse reinforcement in the footing. Therefore, $A_v < A_{v,min}$, and equations (a) and (b) from Table 22.5.5.1 are not applicable.
It is also acceptable to neglect the size effect factor (λ_s) for one-way shallow foundations per §13.2.6.3, so $\lambda_s = 1.0$.

This value is also subject to additional limits per §22.5.5.1.1:
Lower bound: $\varphi * \lambda * \sqrt{(f'_c)}$
Upper bound: $\varphi * 5 * \lambda * \sqrt{(f'_c)}$

Two-Way Shear

Calculations | 2D | 3D

Results | Soil Pressures | Overturning & Sliding Stability | Footing Bending | Footing Shear

Footing Shear Results (using factored loads) .....

Load Combination	Two Way Shear (psi) @ Perimeter "d/2" From Col Edge							
	Pu Col #1 (k)	vu (psi)	Φ vn (psi)	Ratio	Pu Col #2 (k)	vu (psi)	Φ vn (psi)	Ratio
+1.40D	48.092	28.2	164.3	0.172	41.565	20.0	164.3	0.122
+1.20D+1.60L	96.540	56.6	164.3	0.345	83.030	40.0	164.3	0.243
+1.20D+0.50L	58.509	34.3	164.3	0.209	50.440	24.3	164.3	0.148
+1.20D+0.50W	51.008	29.9	164.3	0.182	42.172	20.3	164.3	0.124
+1.20D+0.50L+W	78.082	45.8	164.3	0.279	63.530	30.6	164.3	0.186
+1.20D+0.50L+E	80.530	47.2	164.3	0.287	65.267	31.4	164.3	0.191
+0.90D+W	50.489	29.6	164.3	0.180	39.810	19.2	164.3	0.117
+0.90D+E	52.937	31.0	164.3	0.189	41.547	20.0	164.3	0.122

The maximum factored two-way shear stress is calculated as follows: $v_u = P_u / (b_0 * d)$

Where:

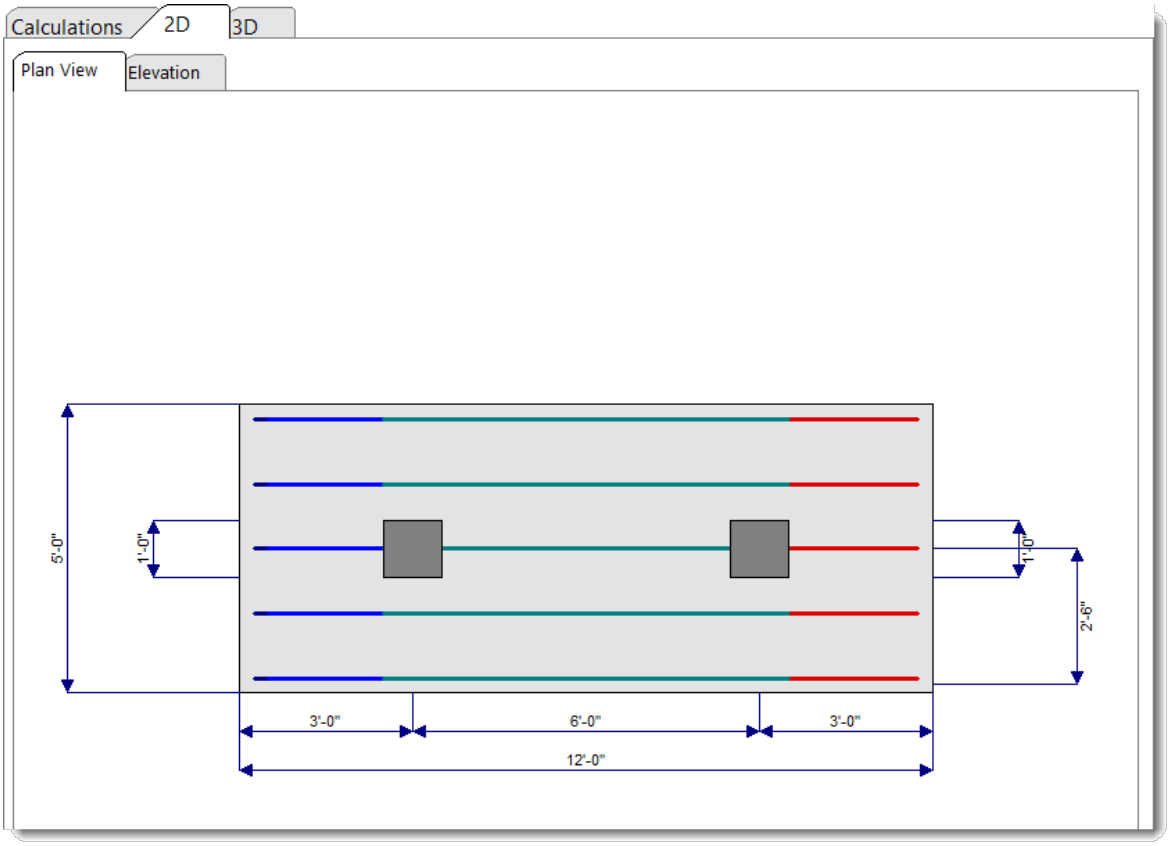
- P_u = Maximum factored axial force at column.
- b_0 = 4 * ([Pedestal Dimension] + d) ; Perimeter of critical section for two-way shear taken at d/2 from face of column.
- d = Distance from extreme compression fiber to centroid of tension reinforcement as reported on the "Footing Bending" tab at the associated critical section for shear.

For ACI 318-14 and ACI 318-19, the nominal shear strength for one-way shear is calculated per the provisions of §22.6 and Table 22.6.5.2.

- The size effect modification factor, λ_s, is taken as 1.0 per ACI 318-19 §13.2.6.2.
- Columns/pedestals are assumed square, therefore $\beta = 1.0$.
- α_s is taken as 40 for interior columns per §22.6.5.3.

© 2026 ENERCALC, LLC

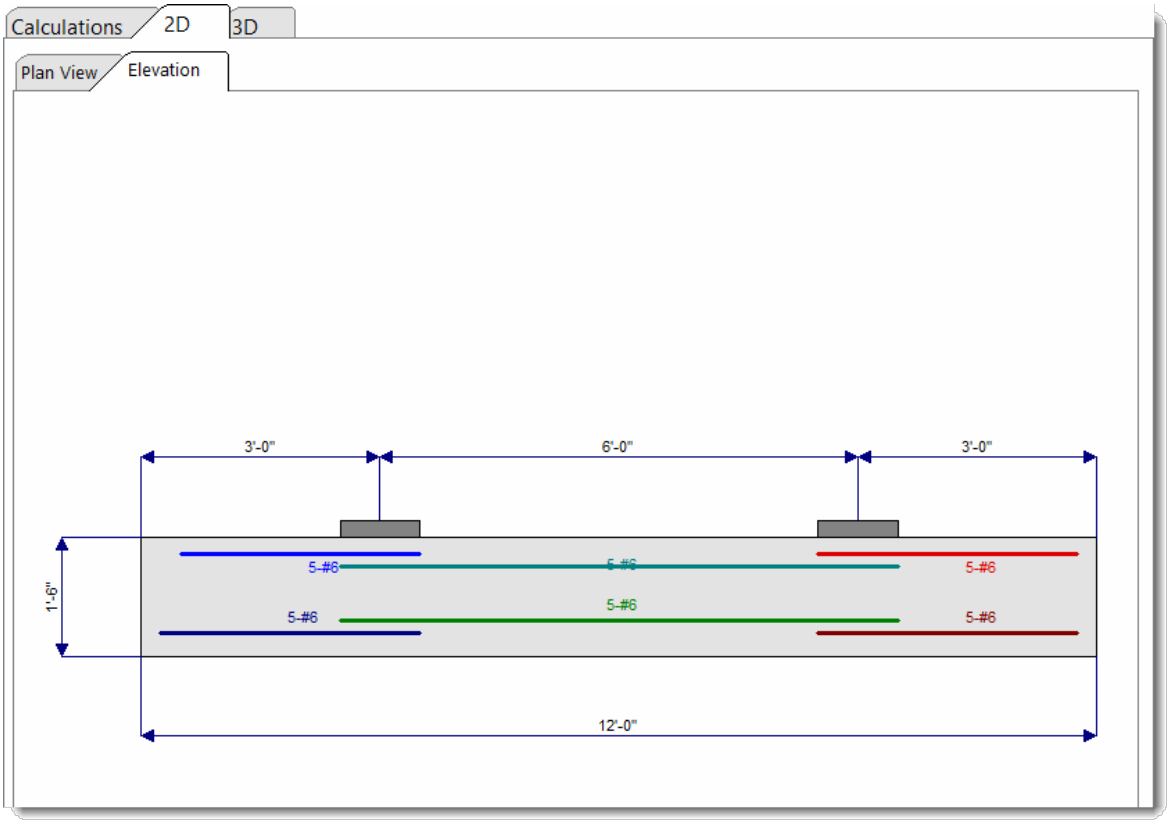
2D Sketch - Plan



This tab provides a plan view of the footing with rebar callouts and overall dimensions.

© 2026 ENERCALC, LLC

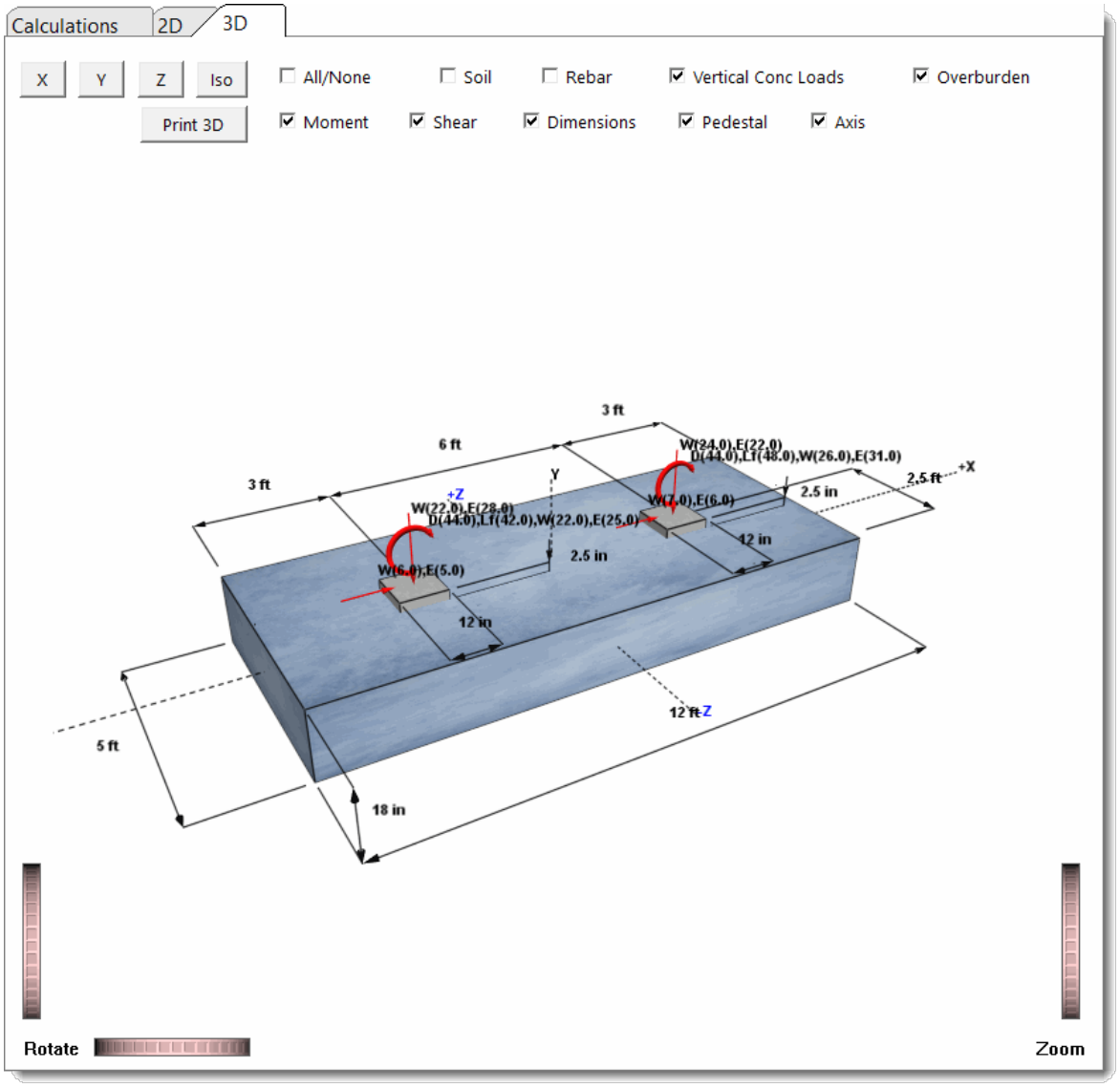
2D Sketch - Elevation



This tab provides an elevation view of the footing with rebar callouts and overall dimensions.

© 2026 ENERCALC, LLC

3D Rendering



This tab provides a 3D rendering of the footing with various view options.

13.6.5 Beam on Elastic Foundation

The **Beam on Elastic Foundation** module is a single-span version of the **Concrete Beam** module that provides special support conditions to represent an idealized, compression-only, continuous elastic support.

The significant variation between the **Beam on Elastic Foundation** module and the **Concrete Beam** module is the fact that the Beam on Elastic Foundation module collects a Soil Subgrade Modulus for use in modeling the spring support.

© 2026 ENERCALC, LLC

Note that code modifications for lightweight concrete are not applied in this module. The purpose of this input is mainly to allow the user to specify something in the range of 145 to 150 pcf.

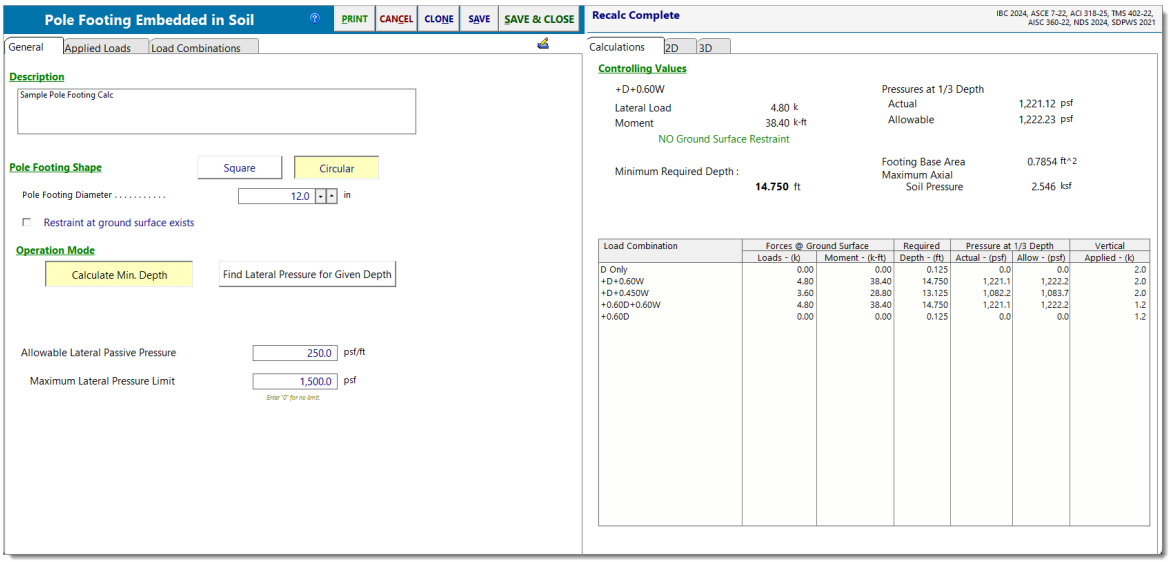
Note: This module will stop if it finds any locations where the soil springs tend to go into tension, so be sure to use realistic load combinations. For instance, don't run +W Only or +Lr Only, without the corresponding dead load that would also be applied concurrently. Also be careful when allowing automatic reversals of wind or seismic loads that could result in net tension at the soil interface.

Refer to the Concrete Beam 752 module information in this document for other details.

13.6.6 Pole Footing Embedded in Soil

Verification Example

This module determines actual soil pressures and required depths for pole footings primarily supporting lateral loads.



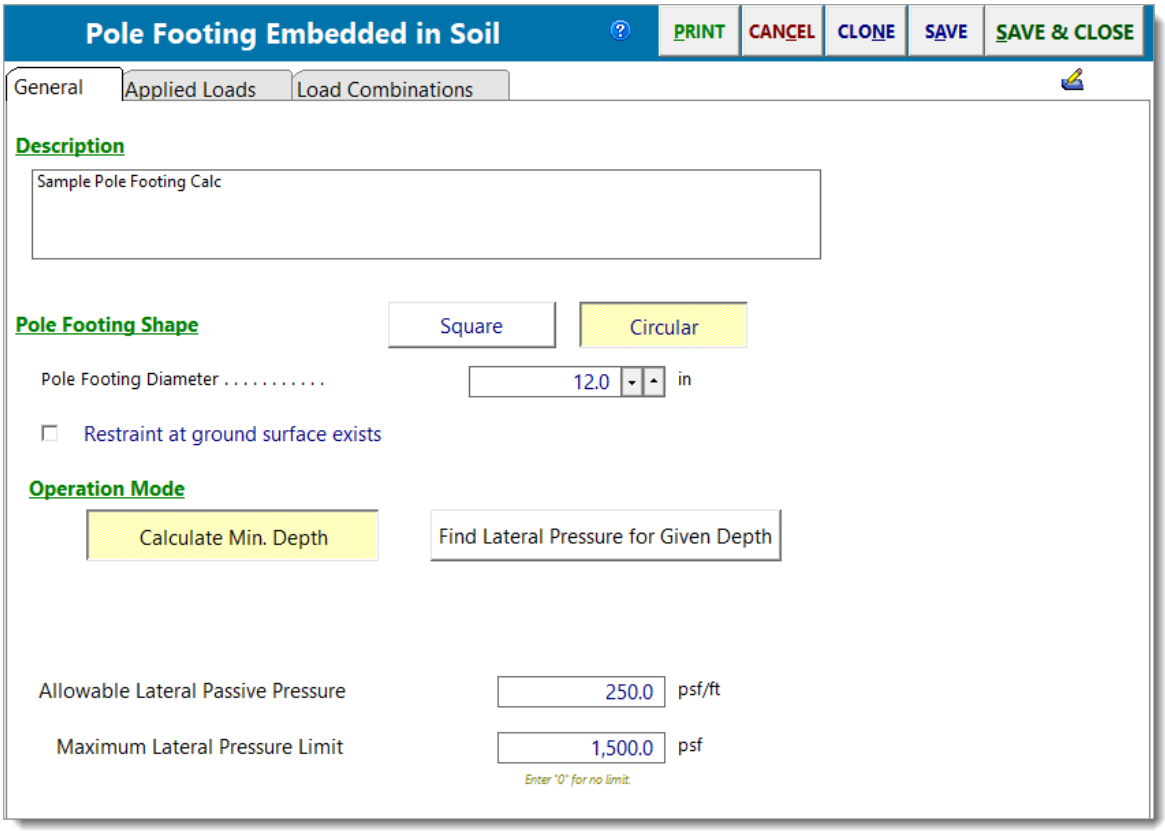
Such footings are commonly called "flagpole footings".

Since applied top moment generates lateral soil pressures that usually govern the design, these footings typically have a depth/width ratio of 2:1 and greater.

Cases with and without lateral restraint at the ground surface are allowed. Evaluation of actual and allowable pressures is in accordance with the IBC Section entitled "Embedded posts and poles".

© 2026 ENERCALC, LLC

General



Pole Footing Shape

Use this section to specify whether the pole is round or rectangular (assumed square).

Footing Width/Diameter

Enter the width or diameter of the pole footing. Width is measured perpendicular to force direction. If the pole is specified as square, the module will multiply the value entered for footing width 1.41 to determine an equivalent width dimension for calculations.

Restraint at Ground Surface

Specify whether the footing is free at the ground surface or restrained and cannot translate. A restrained footing indicates that a concrete slab or other rigid element prevents translation of the pole footing at the ground surface, but does not prevent rotation. When specifying a restrained footing, you must assure yourself that the final force required to provide the restraint can actually be developed by the restraining construction.

When ground surface restraint is present, the lateral pressure value at the bottom of the pole will govern the design.

© 2026 ENERCALC, LLC

Operation Mode

This setting provides an option to select from two different modes of operation as follows:

Calculate Minimum Depth: In this mode, the module will iterate to determine the minimum embedment depth required to make the actual lateral soil pressure lower than the allowable soil pressure.

Find Lateral Pressure for Given Depth: In this mode, the module will calculate the lateral earth pressures caused by the specified pole size, embedment depth and applied loads. When this option is selected, a Pole Footing Embedment Depth input field will appear as shown below:

Pole Footing Embedment Depth

The actual embedment depth of the pole footing. (Visible for "Find Lateral Pressure for Given Depth" mode only).

Allowable Lateral Passive Pressure

The allowable lateral passive pressure that the soil can withstand. This value is entered as pounds per square foot per foot of embedment depth.

Maximum Lateral Pressure Limit

This value is used to specify an upper limit on the Allowable Lateral Passive Pressure, so that it does not increase in an uncontrolled manner as the embedment depth increases. This value is entered as pounds per square foot.

Solution Procedure

Assume:

- Allowable lateral passive pressure = 200 psf/ft
- Maximum lateral pressure limit = 3000 psf
- No restraint at ground surface.

The solution progresses as follows:

1. The program begins with a shallow assumed embedment depth.
2. It calculates one-third of the embedment depth.
3. It compares that value to 12 feet and uses the smaller of the two.
4. It multiplies the governing depth by the allowable lateral passive pressure (psf/ft).
5. It compares that result to the Maximum Lateral Pressure Limit and uses the smaller value.
6. The IBC formula is used to determine the actual passive pressure for the assumed embedment depth.
7. If the actual pressure exceeds the allowable pressure, the embedment depth is increased and the process repeats.
8. When the actual pressure is less than or equal to the allowable pressure, the program rounds the embedment depth up slightly and reports that value.

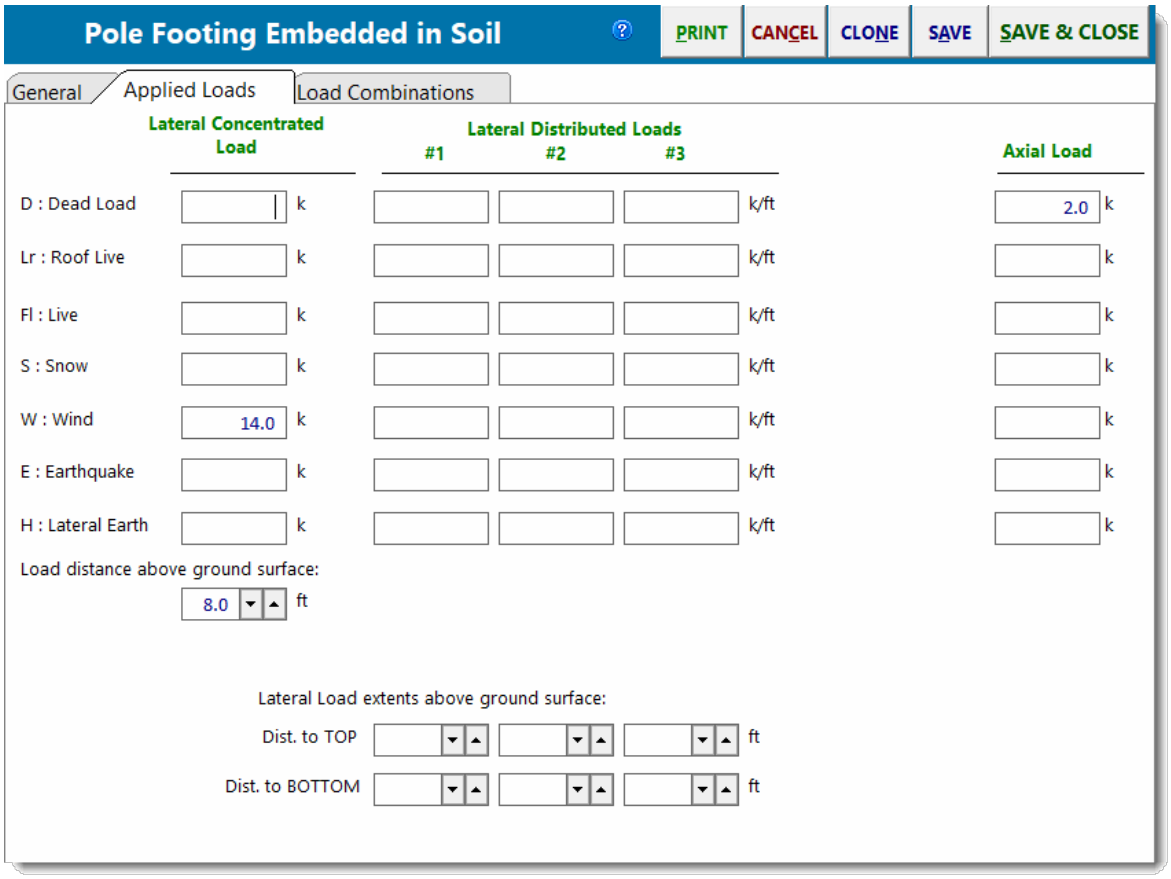
Example: If the embedment depth reaches 42 feet during iteration:

1. One-third embedment depth = 42 / 3 = 14 ft

© 2026 ENERCALC, LLC

2. Compare to 12 ft limit → use 12 ft
3. Allowable lateral passive pressure = 200 psf/ft * 12 ft = 2400 psf
4. Compare to maximum limit → 2400 psf < 3000 psf → use 2400 psf
5. The program checks whether the actual pressure at 42 feet is less than or equal to 2400 psf. If not, it increases the embedment and repeats.

Applied Loads



This module allows many types of loads to be applied to a pole footing embedded in soil.

Lateral Concentrated Loads

Module allows one concentrated load with various load types to be applied at a specified distance above the surface of the soil.

Lateral Distributed Loads

You can apply a uniform lateral load to the pole by specifying the magnitude of the load and the starting and ending locations.

Applied Moments (Only displayed when restraint at ground surface exists)

You can apply a concentrated moment. No "height" entry is required, because it is purely a rotational force.

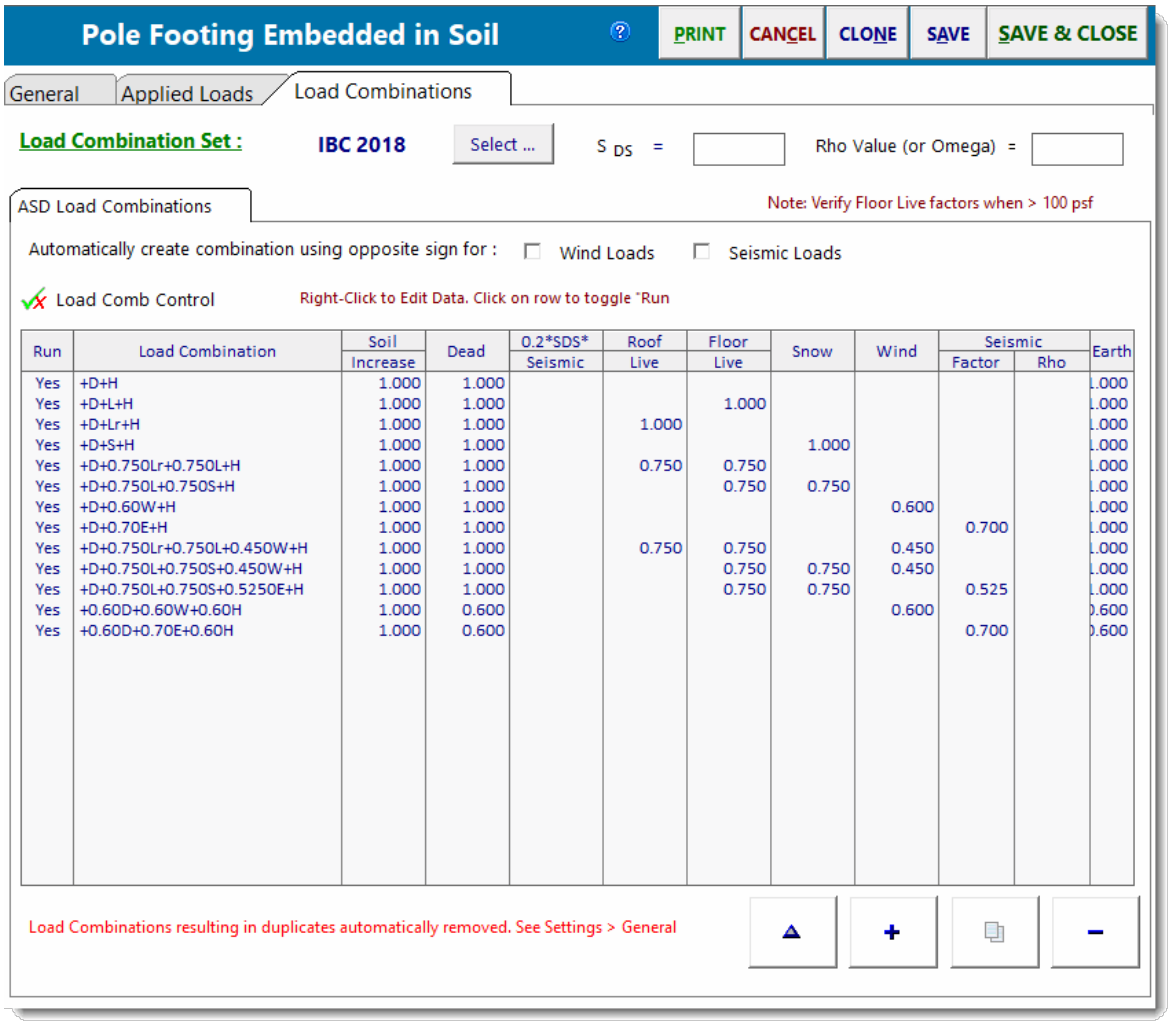
© 2026 ENERCALC, LLC

Vertical Load

You can also apply a vertical load so that the module can calculate the vertical bearing load on the footing for each load combination.

Load Combinations

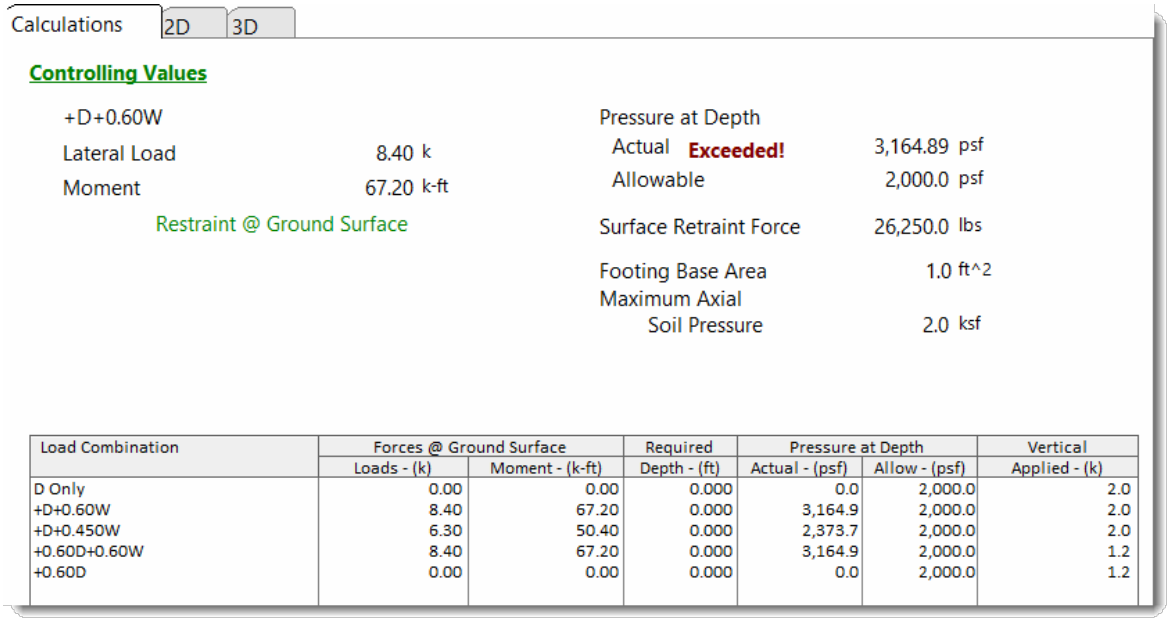
Use this tab to specify the load combinations you want the module to analyze.



Run	Load Combination	Soil Increase	Dead	0.2*SDS* Seismic	Roof Live	Floor Live	Snow	Wind	Seismic Factor	Seismic Rho	Earth
Yes	+D+H	1.000	1.000								.000
Yes	+D+L+H	1.000	1.000			1.000					.000
Yes	+D+Lr+H	1.000	1.000		1.000						.000
Yes	+D+S+H	1.000	1.000				1.000				.000
Yes	+D+0.750Lr+0.750L+H	1.000	1.000		0.750	0.750					.000
Yes	+D+0.750L+0.750S+H	1.000	1.000			0.750	0.750				.000
Yes	+D+0.60W+H	1.000	1.000					0.600			.000
Yes	+D+0.70E+H	1.000	1.000						0.700		.000
Yes	+D+0.750Lr+0.750L+0.450W+H	1.000	1.000		0.750	0.750		0.450			.000
Yes	+D+0.750L+0.750S+0.450W+H	1.000	1.000			0.750	0.750	0.450			.000
Yes	+D+0.750L+0.750S+0.5250E+H	1.000	1.000			0.750	0.750		0.525		.000
Yes	+0.60D+0.60W+0.60H	1.000	0.600					0.600			0.600
Yes	+0.60D+0.70E+0.60H	1.000	0.600						0.700		0.600

Calculations

The calculations tab provides a summary of the results.



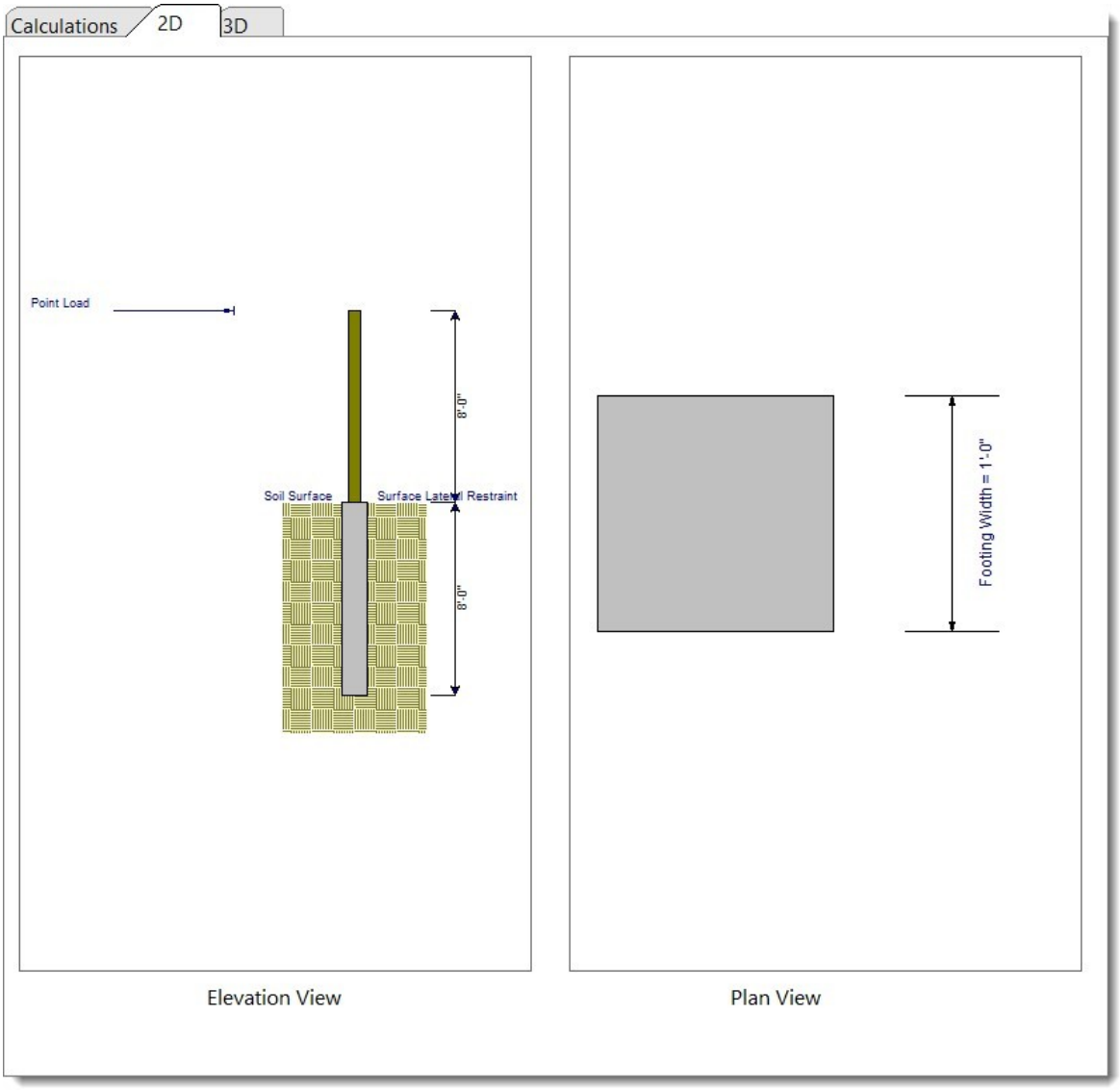
Load Combination	Forces @ Ground Surface		Required	Pressure at Depth		Vertical
	Loads - (k)	Moment - (k-ft)	Depth - (ft)	Actual - (psf)	Allow - (psf)	Applied - (k)
D Only	0.00	0.00	0.000	0.0	2,000.0	2.0
+D+0.60W	8.40	67.20	0.000	3,164.9	2,000.0	2.0
+D+0.450W	6.30	50.40	0.000	2,373.7	2,000.0	2.0
+0.60D+0.60W	8.40	67.20	0.000	3,164.9	2,000.0	1.2
+0.60D	0.00	0.00	0.000	0.0	2,000.0	1.2

The table reports the resulting forces, moments and required depth for each load combination.

The Controlling Values area provides information for the most severe load combination.

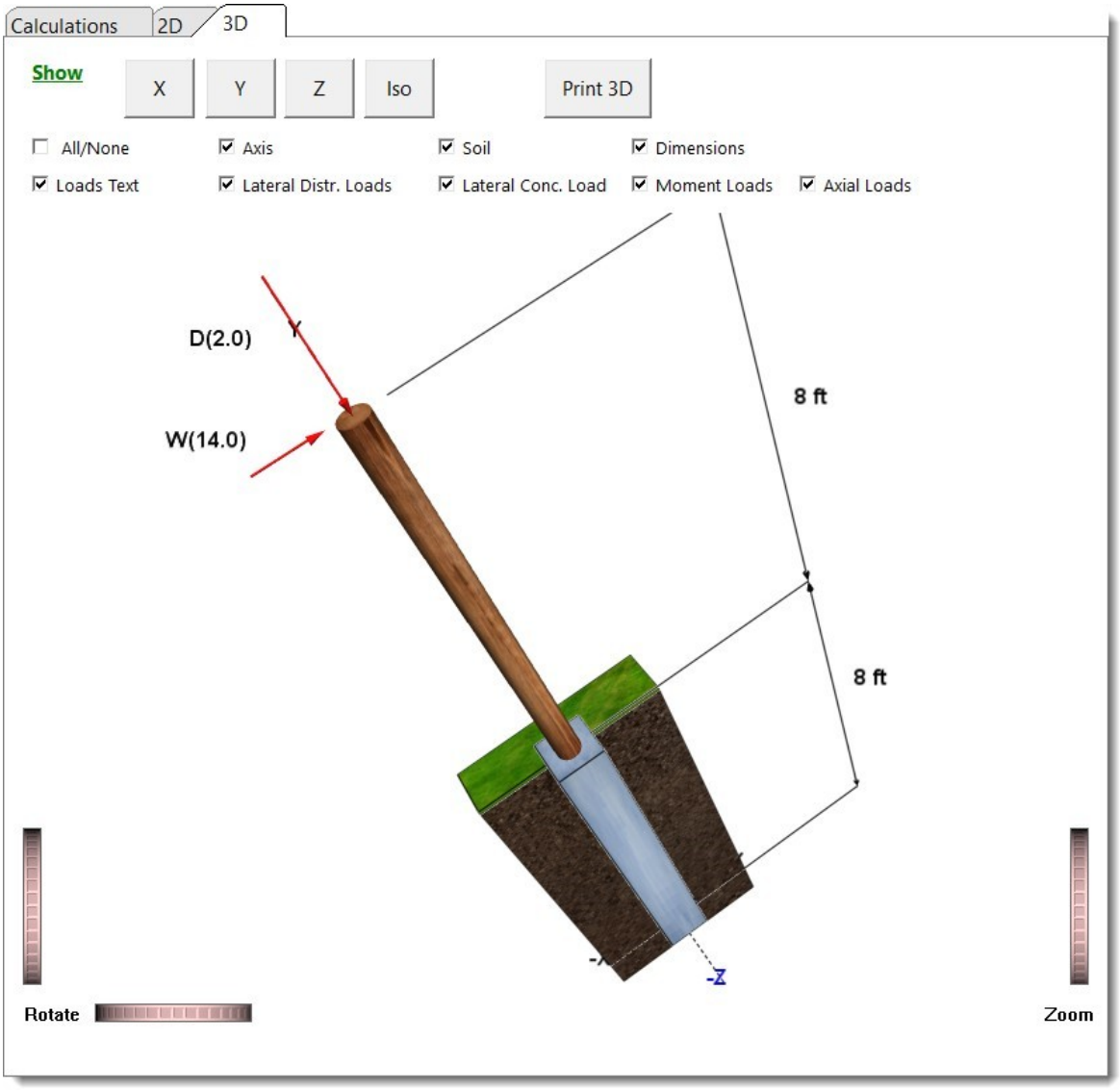
© 2026 ENERCALC, LLC

2D Sketch



© 2026 ENERCALC, LLC

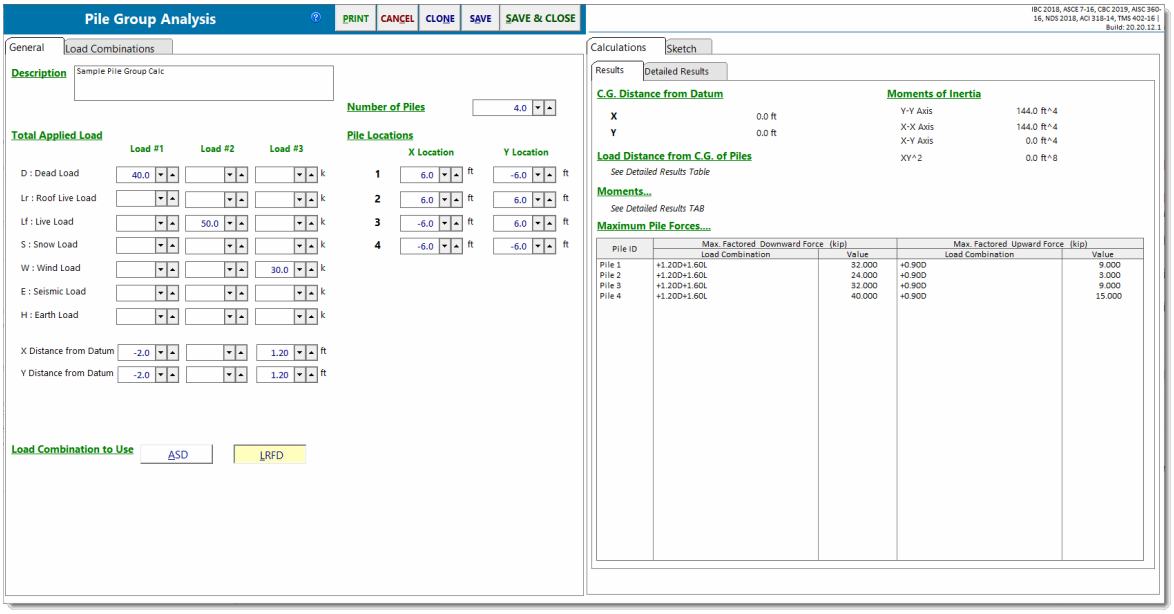
3D Rendering



© 2026 ENERCALC, LLC

13.6.7 Pile Group Analysis

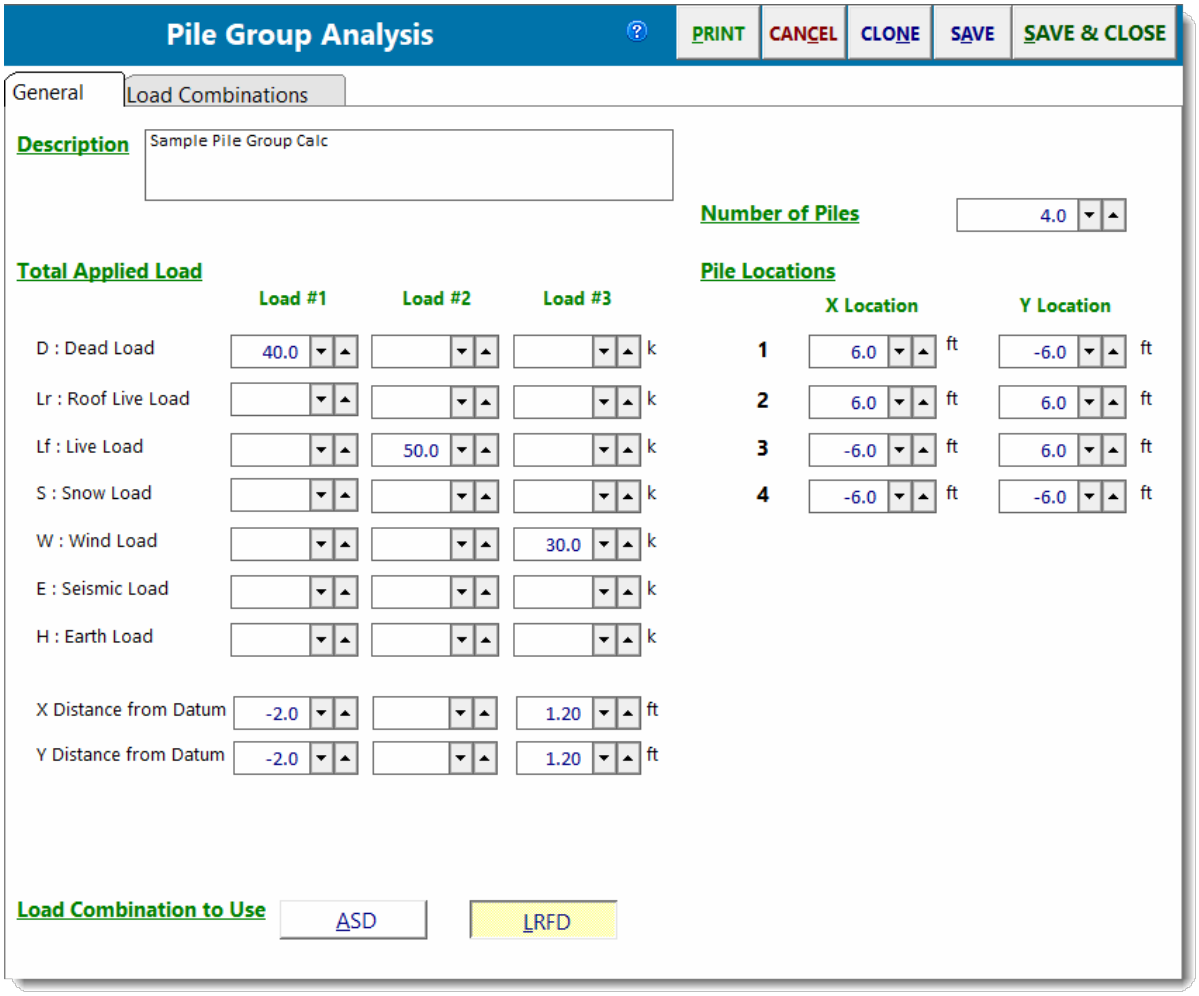
This module considers a concentrated load applied to a rigid pile cap and distributes it to a group of piles.



Force distribution is performed assuming a rigid pile cap and that all piles have equal vertical load resistance. Distribution of loads to each pile due to the effect of load eccentricity is determined using a skew bending analysis. This considers simultaneous action about both X and Y axes. The module is also an efficient method for determining loads on a pile group in the as-driven arrangement.

© 2026 ENERCALC, LLC

General



NOTE! Establish an X & Y Coordinate system prior to entering pile and load locations. Module requires a 2-dimensional pile group to be defined. It will not report results for a collinear group, i.e. a single line of piles.

Total Applied Axial Load

Enter the total vertical load to be distributed to the piles in the pile group using the coordinate system you have defined.

Note: Only vertical loads are allowed; no lateral shears.

X & Y Distance to Load

Enter the distance from the X & Y datum (0,0) to the location of the applied vertical load.

© 2026 ENERCALC, LLC

Number of Piles

This entry defines the number of piles in the group. As you change the number of piles, the number of data entry locations will match the specified number of piles.

Pile Locations: distance from Datum to the pile

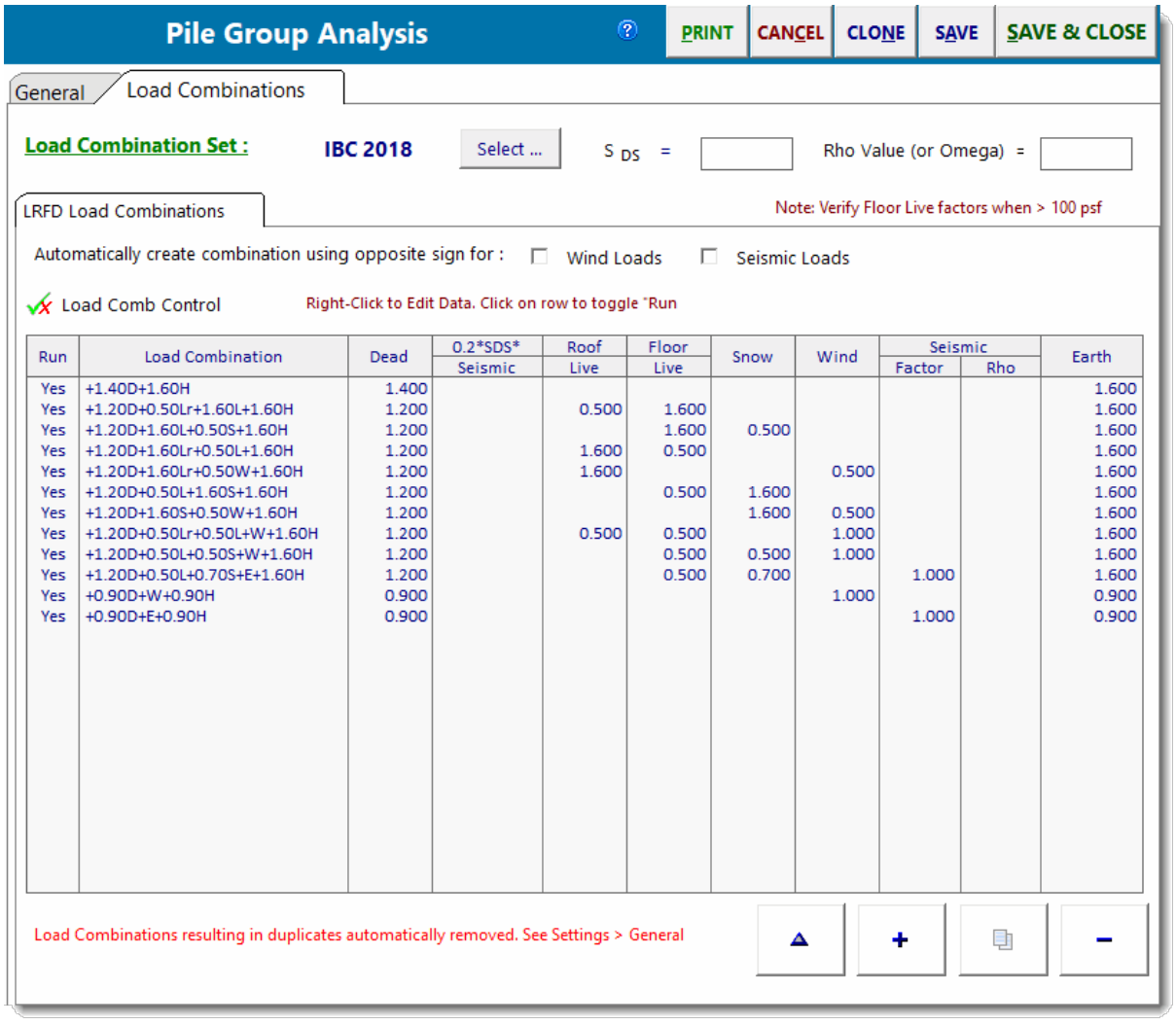
Enter the distance from the X & Y datum (0,0) to the center of each pile location.

Load Combination to use

This selection will switch the load combinations shown on the Load Combinations tab between Service and Factored design combinations.

Load Combinations

This tab allows you to specify the load combinations to be considered.

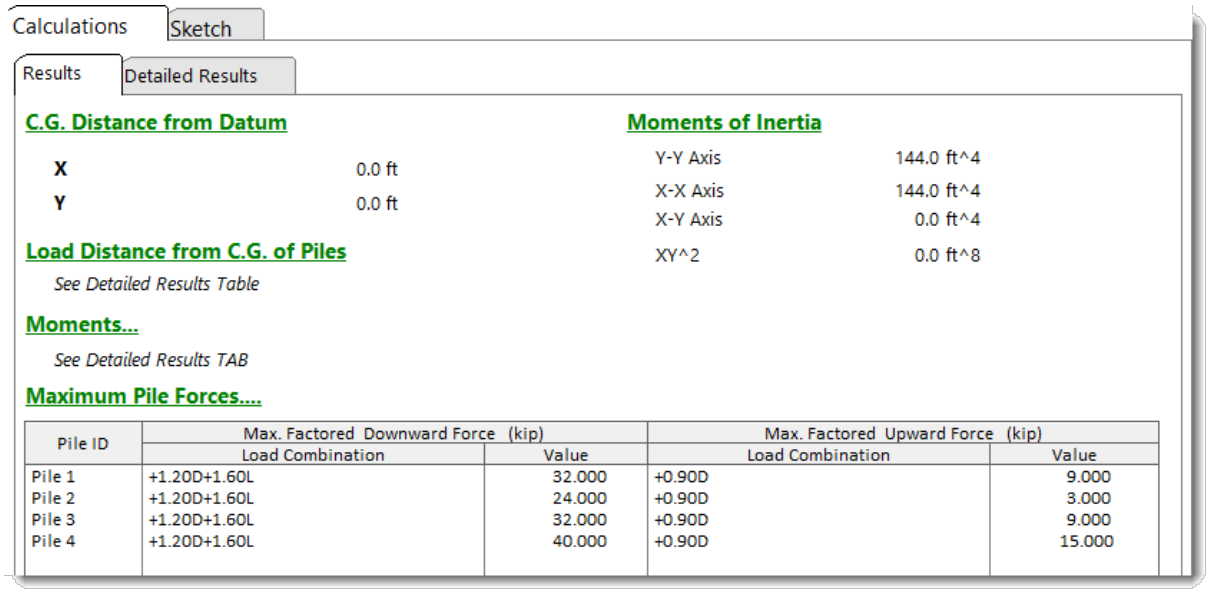


Run	Load Combination	Dead	0.2*SDS* Seismic	Roof Live	Floor Live	Snow	Wind	Seismic Factor	Seismic Rho	Earth
Yes	+1.40D+1.60H	1.400								1.600
Yes	+1.20D+0.50Lr+1.60L+1.60H	1.200		0.500	1.600					1.600
Yes	+1.20D+1.60L+0.50S+1.60H	1.200			1.600	0.500				1.600
Yes	+1.20D+1.60Lr+0.50L+1.60H	1.200		1.600	0.500					1.600
Yes	+1.20D+1.60Lr+0.50W+1.60H	1.200		1.600			0.500			1.600
Yes	+1.20D+0.50L+1.60S+1.60H	1.200			0.500	1.600				1.600
Yes	+1.20D+1.60S+0.50W+1.60H	1.200				1.600	0.500			1.600
Yes	+1.20D+0.50Lr+0.50L+W+1.60H	1.200		0.500	0.500		1.000			1.600
Yes	+1.20D+0.50L+0.50S+W+1.60H	1.200			0.500	0.500	1.000			1.600
Yes	+1.20D+0.50L+0.70S+E+1.60H	1.200			0.500	0.700		1.000		1.600
Yes	+0.90D+W+0.90H	0.900					1.000			0.900
Yes	+0.90D+E+0.90H	0.900						1.000		0.900

© 2026 ENERCALC, LLC

Results

This tab summarizes the overall calculated values for the pile group and lists the maximum factored load for each pile and the load combination that produced the maximum load.



Calculations | Sketch

Results | Detailed Results

C.G. Distance from Datum

X	0.0 ft
Y	0.0 ft

Moments of Inertia

Y-Y Axis	144.0 ft^4
X-X Axis	144.0 ft^4
X-Y Axis	0.0 ft^4
XY^2	0.0 ft^8

Load Distance from C.G. of Piles

See Detailed Results Table

Moments...

See Detailed Results TAB

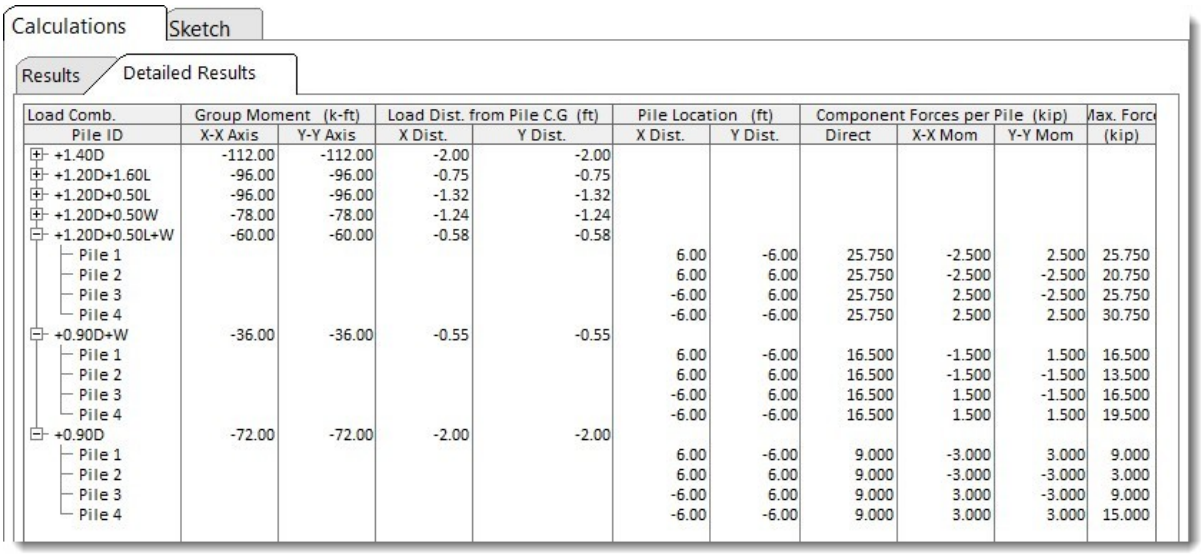
Maximum Pile Forces....

Pile ID	Max. Factored Downward Force (kip)		Max. Factored Upward Force (kip)	
	Load Combination	Value	Load Combination	Value
Pile 1	+1.20D+1.60L	32.000	+0.90D	9.000
Pile 2	+1.20D+1.60L	24.000	+0.90D	3.000
Pile 3	+1.20D+1.60L	32.000	+0.90D	9.000
Pile 4	+1.20D+1.60L	40.000	+0.90D	15.000

© 2026 ENERCALC, LLC

Detailed Results

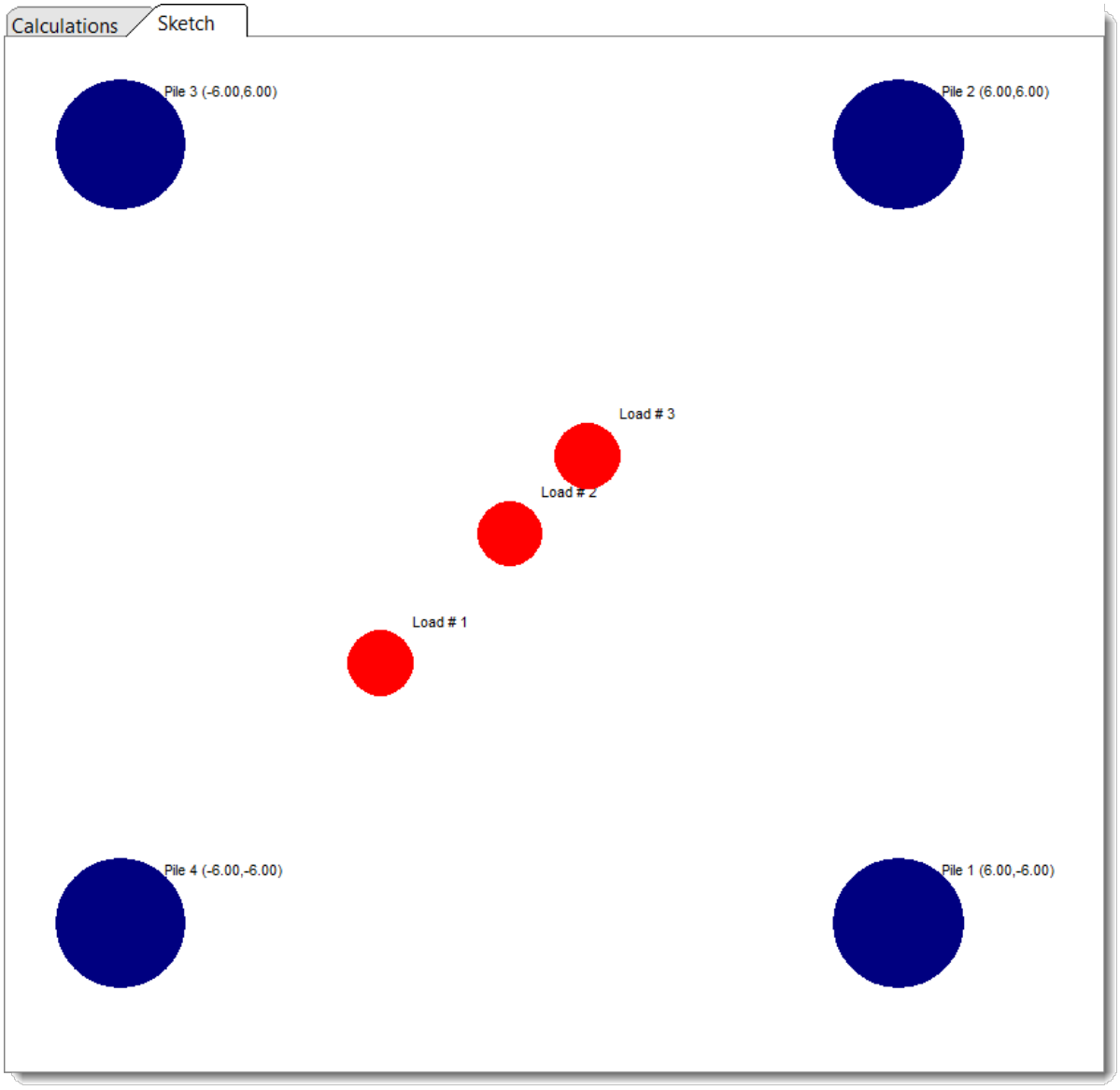
This tab gives the detailed calculations for each pile for each load combination. It indicates the Direct force as well as the component of axial load that is due to the net moment applied to the entire group. This latter effect will be observed for any pile that is located at a distance from the center of gravity of the pile group.



Load Comb. / Pile ID	Group Moment (k-ft) X-X Axis	Group Moment (k-ft) Y-Y Axis	Load Dist. from Pile C.G (ft) X Dist.	Load Dist. from Pile C.G (ft) Y Dist.	Pile Location (ft) X Dist.	Pile Location (ft) Y Dist.	Component Forces per Pile (kip) Direct	Component Forces per Pile (kip) X-X Mom	Component Forces per Pile (kip) Y-Y Mom	Max. Force (kip)
+1.40D	-112.00	-112.00	-2.00	-2.00						
+1.20D+1.60L	-96.00	-96.00	-0.75	-0.75						
+1.20D+0.50L	-96.00	-96.00	-1.32	-1.32						
+1.20D+0.50W	-78.00	-78.00	-1.24	-1.24						
+1.20D+0.50L+W	-60.00	-60.00	-0.58	-0.58						
Pile 1					6.00	-6.00	25.750	-2.500	2.500	25.750
Pile 2					6.00	6.00	25.750	-2.500	-2.500	20.750
Pile 3					-6.00	6.00	25.750	2.500	-2.500	25.750
Pile 4					-6.00	-6.00	25.750	2.500	2.500	30.750
+0.90D+W	-36.00	-36.00	-0.55	-0.55						
Pile 1					6.00	-6.00	16.500	-1.500	1.500	16.500
Pile 2					6.00	6.00	16.500	-1.500	-1.500	13.500
Pile 3					-6.00	6.00	16.500	1.500	-1.500	16.500
Pile 4					-6.00	-6.00	16.500	1.500	1.500	19.500
+0.90D	-72.00	-72.00	-2.00	-2.00						
Pile 1					6.00	-6.00	9.000	-3.000	3.000	9.000
Pile 2					6.00	6.00	9.000	-3.000	-3.000	3.000
Pile 3					-6.00	6.00	9.000	3.000	-3.000	9.000
Pile 4					-6.00	-6.00	9.000	3.000	3.000	15.000

© 2026 ENERCALC, LLC

2D Sketch



© 2026 ENERCALC, LLC

Design Basis:

Detailed Item Summary

☞ *Items with ‡ by their names are calculated values; all others are input.*

Design Data

Total Axial Load — Enter the total Vertical load to be distributed to the piles in the pile group using the coordinate system you have defined. This indicates the distance from the datum (0,0) point to the point of load application.

☞ ***Note!*** *Only vertical loads are allowed; no lateral shears.*

Calculated Values

‡ **Center of Gravity** — Using a simple center of gravity calculation assuming each pile is of equal resistance, the neutral axis of the pile group about both axes is determined.

‡ **Load Ecc. from CG** — After the center of gravity of the pile group is located, the eccentricity of the applied load to the C.G. is calculated and will be used to determine the X-X and Y-Y axis moments on the pile group.

Group Inertia About Axis — Ixx and Iyy are calculated by using:

$I = SUM (A * d^2)$

where...

d = Distance of each pile from the center of gravity

‡ **X-X & Y-Y Moments** — Using the applied load and eccentricity from the pile group center of gravity, the X-X and Y-Y axis moments on the pile group are calculated. This will be used in the equations detailed below to determine the loads to each pile.

Summary Of Pile Loads

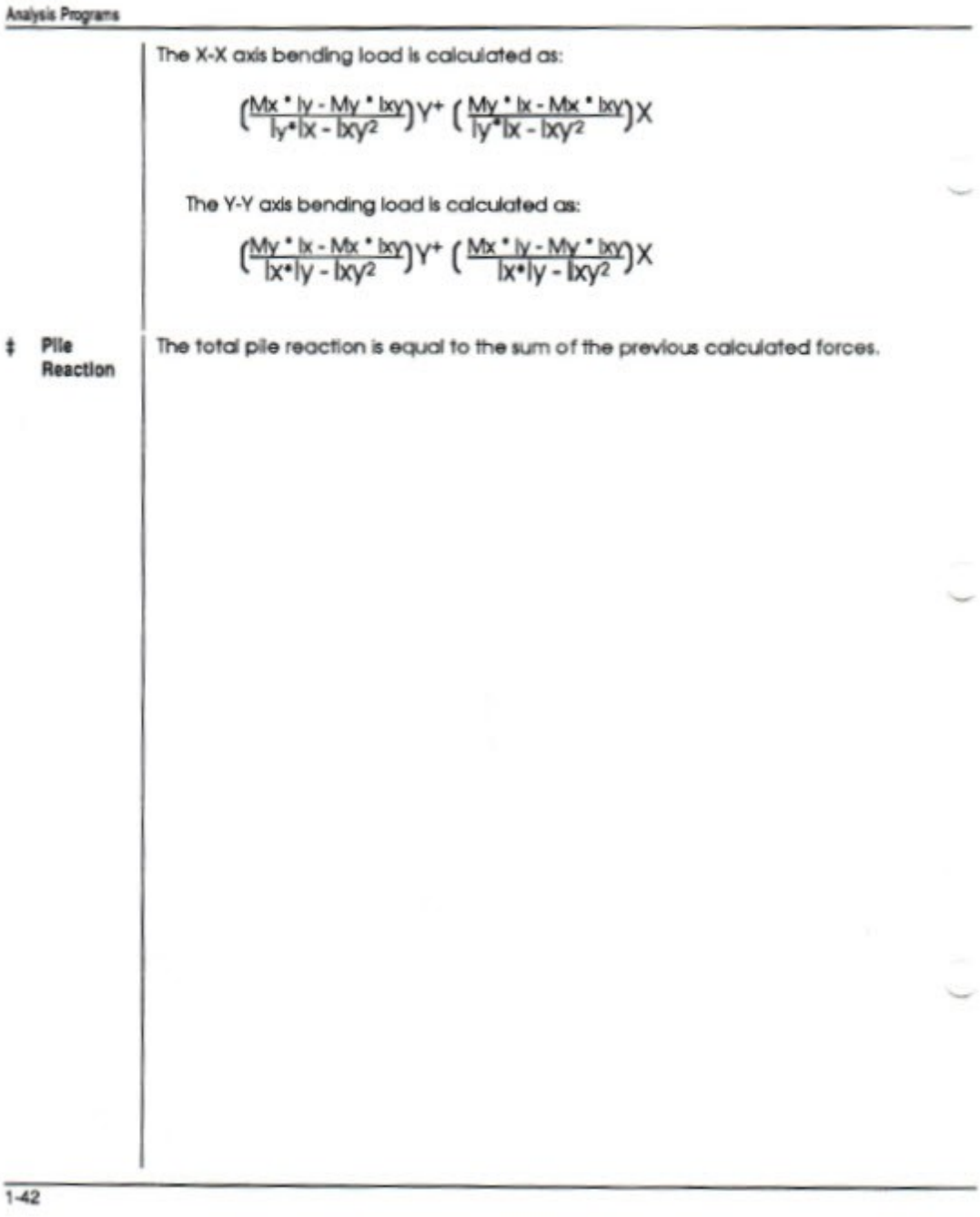
Pile Number — Reference number for your convenience.

Coordinates — According to the user defined coordinate system, enter the "X" and "Y" distances of each individual pile from datum (0,0).

Load/# Pile — This equals the total applied load divided by the number of piles.

‡ **Force from Rotation** — Represents the force applied to each pile as a result of the induced moment about the "X" and "Y" axes.

© 2026 ENERCALC, LLC



Analysis Programs

The X-X axis bending load is calculated as:

$$\left(\frac{Mx * Iy - My * Ixy}{Iy * Ix - Ixy^2}\right)Y + \left(\frac{My * Ix - Mx * Ixy}{Iy * Ix - Ixy^2}\right)X$$

The Y-Y axis bending load is calculated as:

$$\left(\frac{My * Ix - Mx * Ixy}{Ix * Iy - Ixy^2}\right)Y + \left(\frac{Mx * Iy - My * Ixy}{Ix * Iy - Ixy^2}\right)X$$

‡ **Pile Reaction**

The total pile reaction is equal to the sum of the previous calculated forces.

1-42

13.6.8 Point Load on Slab

Design Approach and Sources

This module calculates the load-carrying capacity of an unreinforced concrete slab-on-grade subjected to isolated concentrated loads, such as storage rack legs not supported by a building structure. This application is outside the scope of ACI 318.

© 2026 ENERCALC, LLC

The design approach implemented in this module is based on the simplified analytical method presented by Azzi & Laird (2008), derived from the nonlinear finite-element research of Shentu, Jiang, and Hsu (1997).

This method is based on an elasto-plastic representation of slab behavior and is intended to estimate the ultimate load-carrying capacity of a slab-on-grade subjected to a concentrated load. Unlike traditional linear-elastic approaches, this formulation accounts for cracking and post-cracking behavior of the concrete slab.

For further information see:

(1) Shentu, Jiang, and Hsu (1997). *Load-Carrying Capacity for Concrete Slabs on Grade.* Journal of Structural Engineering, ASCE.

(2) Azzi and Laird (2008). *Load-Carrying Capacity of Concrete Slabs-on-Grade Subject to Concentrated Loads.* STRUCTURE Magazine.

(3) Jlilati, Brown, and Tang (2019). *Comparative Investigation of Two Design Methods for Slab-on-Grade Foundations Subjected to Concentrated Loads.* ISEC Press.

(4) Federal Emergency Management Agency (FEMA). (2005). *Seismic Considerations for Steel Storage Racks Located in Areas Accessible to the Public (FEMA 460).* Prepared by the Building Seismic Safety Council.

Slab Capacity Equations

The nominal load-carrying capacity is calculated using the following equation:

$$P_n = \beta * 1.72 [(k_s * R_1 / E_c) 10{,}000 + 3.60] * F_r * d^2$$ *[Equation 1, Azzi and Laird (2008)]*

Where

k_s: Modulus of subgrade reaction of the soil, pci

R_1: sqrt(Plate Width * Plate Length) / 2, inches

E_c: Concrete elastic modulus, psi

F_r: Tensile strength of the concrete = f_t' * sqrt(f'c), psi

d: Slab thickness, inches

β: Load reduction factor for thick slabs; Azzi & Laird (2008) use β = 1.0 for slab thicknesses less than 7 inches and β = 0.85 for slab thicknesses of 7 inches or greater

f_t': Factor used to calculate the tensile strength in flexure of concrete; a value of 7.5 is commonly used for normalweight concrete, though alternative values may be selected based on engineering judgment and supporting research

© 2026 ENERCALC, LLC

ASD and LRFD Implementation

When ASD is selected, the nominal load-carrying capacity (P_n) is reduced using a user-defined Factor of Safety (FS) and compared directly to the applied load (P_u):

$$FS = P_n / P_u$$

When LRFD is selected, the nominal load-carrying capacity (P_n) is reduced using a user-defined strength reduction factor (Φ). The Factor of Safety remains available as an additional user-defined input. In this case, it is generally expected that Φ will account for the primary resistance reduction and that the Factor of Safety may therefore be set to 1.0; however, users may specify alternative values consistent with their design approach and engineering judgment.

$$FS = \Phi * P_n / P_u$$

This module provides flexibility in how resistance reductions are applied. Users should ensure that the combined effect of Φ and FS reflects their intended design assumptions and should avoid unintentionally double-counting safety margins.

Factor of Safety Considerations

This module allows the user to specify a Factor of Safety (FS) to reduce the nominal load-carrying capacity for design use. Earlier applications of this method commonly used a Factor of Safety of 3. Experimental studies evaluating this approach indicate that a Factor of Safety of 6 may be more appropriate for design applications. Users should select a Factor of Safety consistent with their intended performance objectives and applicable engineering judgment.

These Factor of Safety values are most directly applicable when the method is used in an ASD framework. When LRFD is selected, users should ensure that the combined effect of the selected strength reduction factor (Φ) and any additional Factor of Safety reflects their intended performance objectives.

Minimum Adjacent Load

The nominal load-carrying capacity (P_n) equation assumes that the load acting on the slab is unique and no other nearby loads are affecting the calculation.

To assist in the evaluation of slabs-on-grade, this module also provides a calculation of the distance that the closest load may be without affecting the calculated slab capacity. The calculation given below is based on research by Packard, Pickett and Ray, and Spears and Panarese, and is also discussed in ACI 360R-92.

© 2026 ENERCALC, LLC

In this module the distance is calculated as 1.5 * "Radius of Relative Stiffness" given by the following equation:

$$b = [\, E_c * d^3 / (12 * (1-u^2) * k_s)]^{0.25} \qquad \textit{[Equation 3, Azzi and Laird (2008)]}$$

Where

b: Radius of relative stiffness, inches

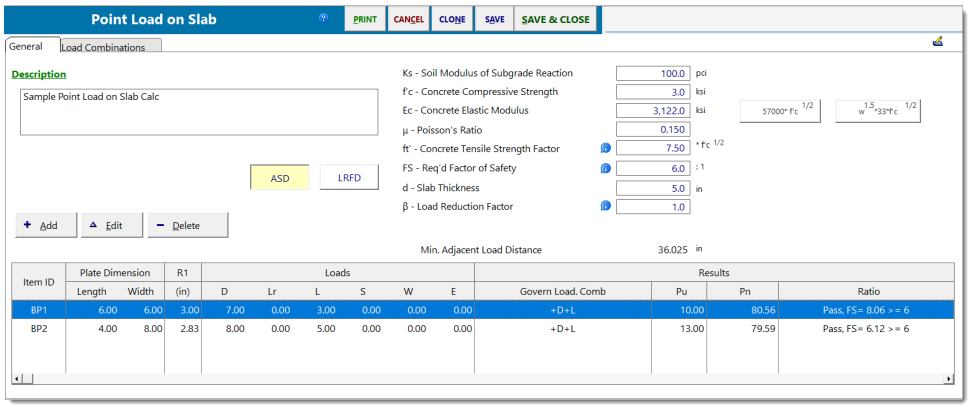
E_c: Concrete elastic modulus, psi

d: Slab thickness, inches

u: Poisson's ratio

k_s: Modulus of subgrade reaction of the soil, pci

General



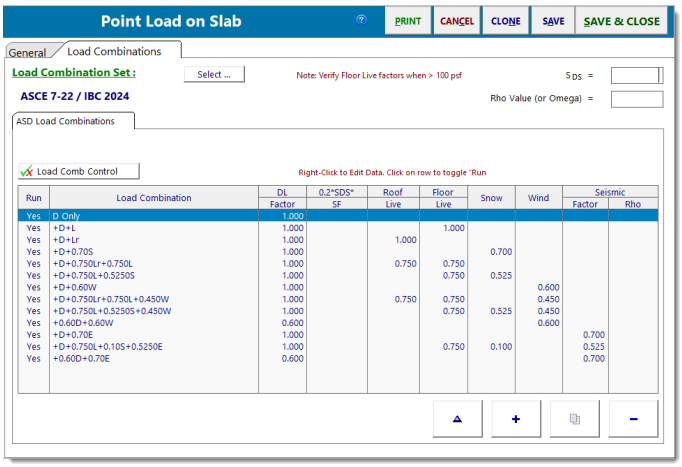
This module is designed to allow the user to create a table of loads applied to a particular concrete slab and supporting soil with one set of material properties.

You can then use the [**Add**], [**Edit**] and [**Delete**] buttons to add a set of applied loads and base plate dimensions. From this data all load combinations are used to determine the maximum axial force. For the plate dimension you specify, the maximum load capacity for the point load application is calculated and compared with your required factor of safety.

Load Combinations

This Load Combination set used is based on the selected design method, ASD or LRFD.

© 2026 ENERCALC, LLC



13.7 Walls

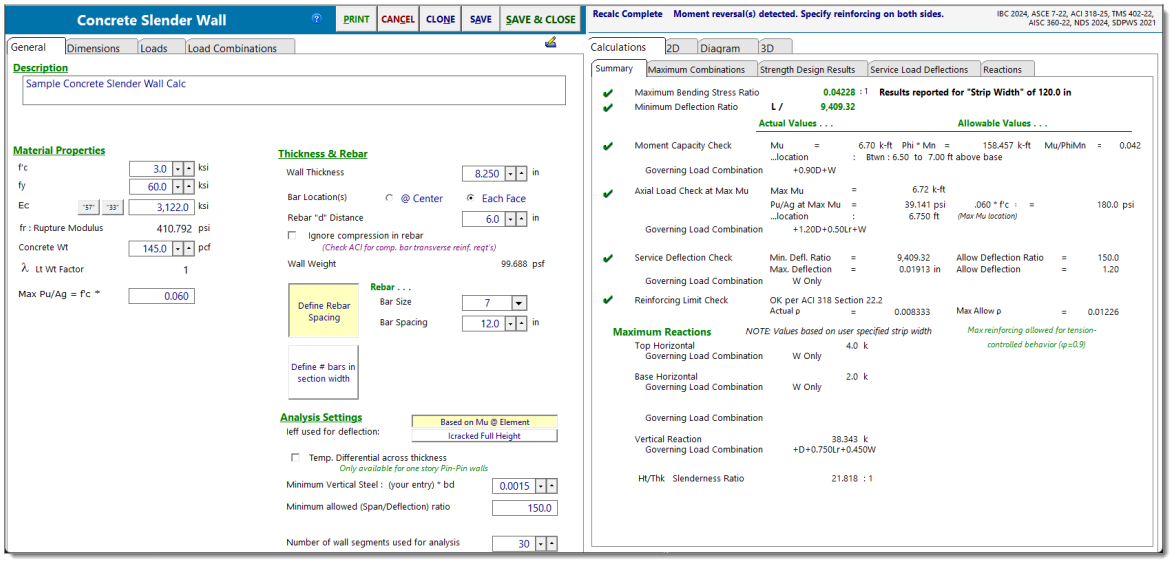
Please select a subtopic.

13.7.1 Slender Walls

Please select a subtopic.

13.7.1.1 Concrete Slender Wall

This program provides analysis of concrete wall panels that stand vertically and have applied vertical and out-of-plane lateral loads.



The wall panel is analyzed using strength design procedures. Out-of-plane moments within the wall are created by eccentric axial loads, applied lateral loads, lateral self weight loads and moments induced due to the wall weight acting at an eccentricity when it deflects (P-Delta moments).

© 2026 ENERCALC, LLC

The ACI slender wall procedure, introduced in ACI 318-99, was first adopted by the IBC 2000 and subsequent code editions. As quoted in ACI 318R-05 Commentary, Section 14.8 is based on the corresponding requirements in 1997 UBC and experimental research presented in the 1982 "Test Report by SCCACI-SEAOSC". Analytical study of the current IBC/ACI provisions for concrete wall panels showed the ACI procedure does not correspond to a bilinear load-deflection characteristic observed in the SEAOSC tests and significantly underestimates the service load deflection.

The ENERCALC Concrete Slender Wall program uses basic principles of structural mechanics to model the wall as a series of beam segments. For each segment the actual moment is used to calculate member stiffness using the $I_{effective}$ equations developed by Peter H. Bischoff. Since these changes to wall stiffness affect the wall deflection profile, the program performs an iterative analysis of calculating moments (including P-Delta effects). The results are deflection curves almost exactly matching the SCCACI-SEAOSC test results. This makes this program far more accurate at calculating wall deflections and P-Delta effects than the simple equations in the ACI code.

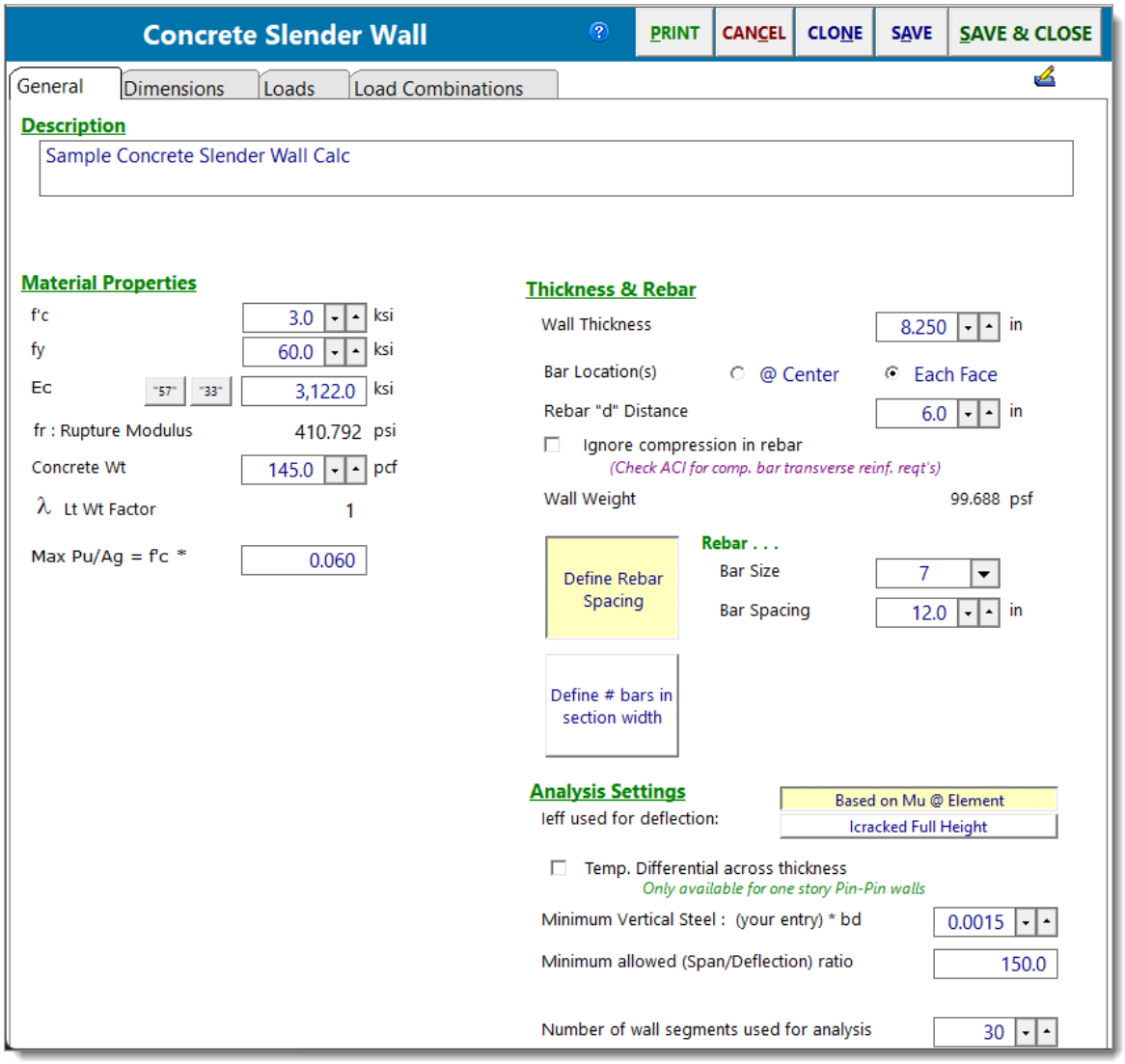
Capabilities

This module provides these capabilities:

- One or two story slender tilt-up concrete walls
- Iterative process accounts for P-delta (12 iterations are used)
- Optional parapet
- Axial loads with optional eccentricities
- Wind, seismic and user defined lateral loads creating bending on the wall panel
- Variable strip width to model the wall panel
- Temperature differential can be specified across thickness of wall to add curvature
- Rebar location at center of wall or two layers of reinforcing at each side
- Bottom of wall can be fixed or pinned for moment resistance
- Top of wall can be pinned or free
- A reveal can be defined and cross section properties modified for reduced thickness and optionally add rebar
- The effects of wall openings can be addressed by modeling the solid panel between or adjacent to openings and then using superposition to apply the loads above and below openings.

© 2026 ENERCALC, LLC

General



Material Properties

f'c

28-day compressive strength of the concrete.

Fy

Yield point stress of reinforcing

Ec

Modulus of elasticity of concrete. You can enter the value or click the ["**57**"] button to set Ec = 57000 * sqrt(f'c), or click the ["**33**"] button to set Ec = 33 * sqrt(f'c) * ConcWeight$^{1.5}$.

© 2026 ENERCALC, LLC

Fr: Rupture Modulus
The modulus of rupture for concrete, calculated as $f_r = 7.5\lambda\sqrt{f'_c}$.

Concrete Wt
Weight of concrete in pounds per cubic foot.

Lambda
Factor to account for lightweight concrete.

Max Pu/Ag = f'c * <entry>
Enter a multiplier less than 1.0 which will be applied to f'c to determine the maximum allowable factored axial stress.

Thickness & Rebar

Wall Thickness
Total wall thickness

Bar Location
You can select bar placement at the center of the wall thickness or at each face. When selecting "Each Face" the module performs calculations considering both bars, unless the option is also selected to Ignore compression in rebar.

Rebar "d" Distance
Enter the distance between the outside surface of the wall to the centerline of the rebar. For bars each face this measurement can be from either face.

Wall Weight
The internally calculated wall weight considering the concrete weight and wall thickness entered.

Bar Size
Enter the US customary rebar size number.

Rebar [Spacing] / [# in Width]
These two options indicate how you will specify the rebar quantity in your design strip.

[**Spacing**] will change the entry so that you can enter a spacing in inches for the rebar.

[**# in Width**] changes the entry so you can enter the number of bars in your design strip, where the width of the design strip is entered on the Dimensions tab.

Note: When using the "Each Face" option, the "# Bars in Width" specifies the number of bars **on each face** within the design strip width.

© 2026 ENERCALC, LLC

Analysis Settings

Ieff used for Deflection

The module offers the option to use $I_{effective}$ based on the moment in the individual wall elements or to use $I_{cracked}$ for the full height of the wall.

Temperature Differential across thickness

This input is used to describe the temperature change between each face of the wall. A temperature change induces a slight curvature into the wall because the hotter side expands, resulting in a slightly higher out-of-plane deflection.

Minimum Vertical Steel: %/100

Minimum steel as a percentage of the gross wall area.

Minimum Allowed (Span/Deflection) Ratio

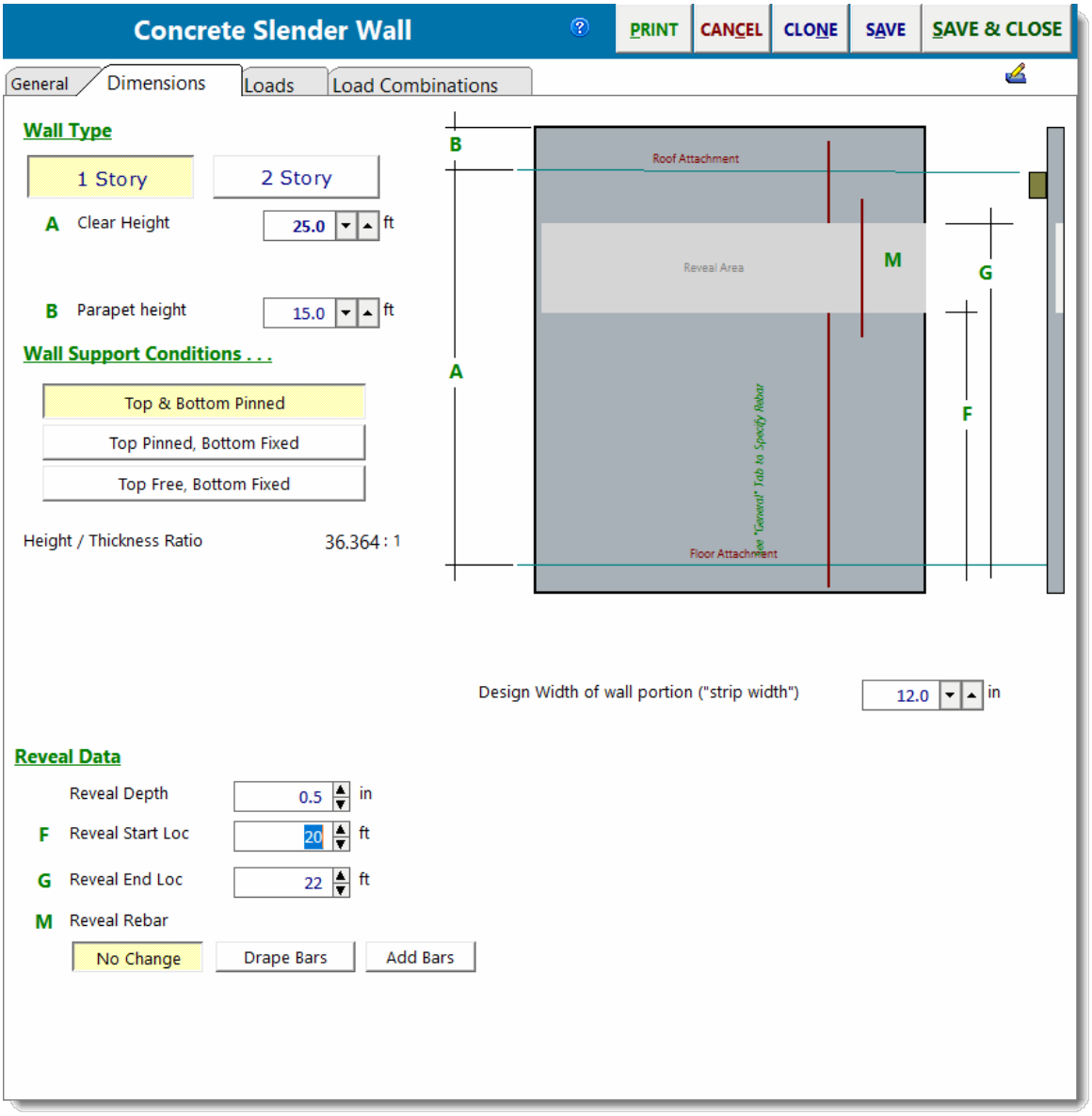
This setting establishes the minimum allowable ratio of span length to service load deflection. If a lower actual Span/Deflection ratio occurs (meaning greater deflection), a warning message will be displayed.

Number of wall elements for FE solver to use

This module divides the wall design strip into segments from the base to the top for analysis purposes. Use this entry to define the number of segments to use. Experience demonstrates that approximately 30-40 segments gives a good balance between the iterative P-Delta analysis reaching convergence and excessive calculation time.

© 2026 ENERCALC, LLC

Dimensions



Clear Height

Span of the wall between the base and the first lateral support. For one-story walls this is the top support. For 2-story walls this prompt will change to be "1st story height".

Parapet Height

Distance the wall extends above the topmost lateral support (i.e. extension above the clear height for one-story wall, extension above the 2nd story height for 2-story walls).

Wall Support Conditions

Controls how the top and bottom of the wall are restrained for moments and lateral movement.

© 2026 ENERCALC, LLC

[Top & Bottom Pinned]
Base of wall is restrained against movement out of plane and vertically, rotates freely. Top of wall restrained against out of plane movement and can move vertically and rotate freely.

[Top Pinned, Bottom Fixed]
Base of wall is restrained against movement about all three axes. Top of wall restrained against out of plane movement and can move vertically and rotate freely.

[Top Free, Bottom Fixed]
Base of wall is restrained against movement about all three axes. Top of wall is completely free making this a cantilevered wall.

Reveal Data

A reveal is a portion of the wall that is recessed from the rest of the surface. It is formed by placing thin blockout material (typically styrofoam) in the forms prior to concrete placement. It is used to create architectural effects. The reveal reduces the structural thickness of the wall. This module calculates section properties for this reduced section in the portion of the wall where the reveal has been formed.

Reveal Depth

Depth of reveal measured from outside face of wall. A 1" reveal in a 6" wall gives a net structural thickness of 5".

Reveal Start Location, Reveal End Location

Distances measured upwards from bottom of wall that define the start and end points of the reveal.

Reveal Rebar

This selection defines how the module should consider the reveal area to be reinforced.
No Change means that the reinforcing stays where it is as defined by the "Rebar 'd' Distance" entered on the General tab. This option results in an offset rebar location within the remaining structural thickness, because the reveal takes away part of the concrete.
Drape Bars tells the module to move the rebar inward to give the same dimension between the rebar and face of wall. For walls with bars at "Center" this moves the bar to the center of the remaining structural thickness. When bars are specified on each face, this option moves only one of the bars inward.
Add Bars enables you to add additional reinforcing in the area of the reveal. The location of the main rebar is as described in the "No Change" option above.

© 2026 ENERCALC, LLC

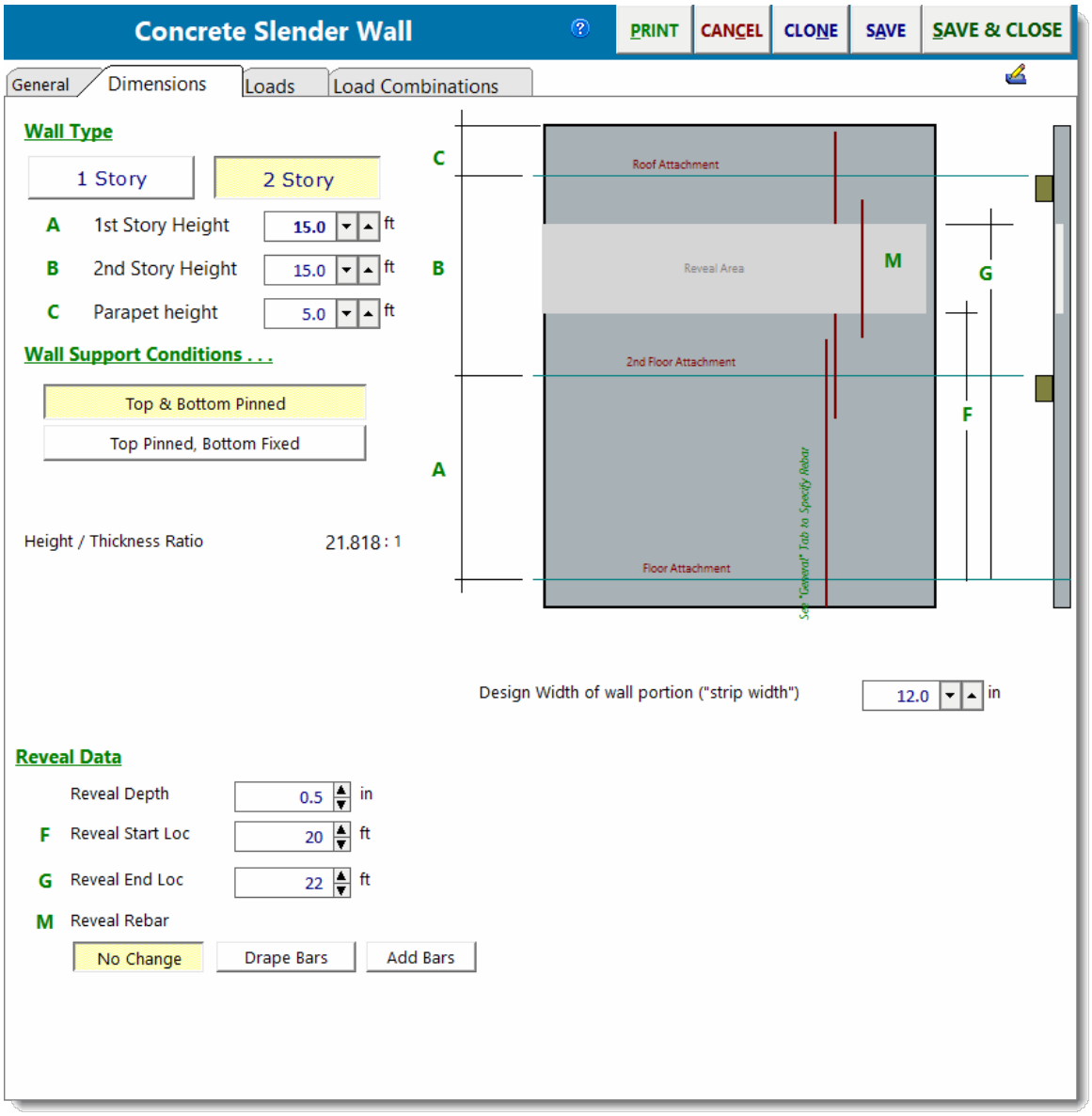
Design Width of wall portion ("strip width")

This module performs its analysis for this width. Results are for either this width or a 12" width as noted where the results are provided.

Note that applied loads either are applied to the entire strip width (as for concentrated vertical and lateral loads) or are entered on a per-foot basis when they are uniform loads.

Two Story...

When a two-story wall is selected, this tab changes slightly to provide the 2nd story height and remove the pure cantilevered support option.



© 2026 ENERCALC, LLC

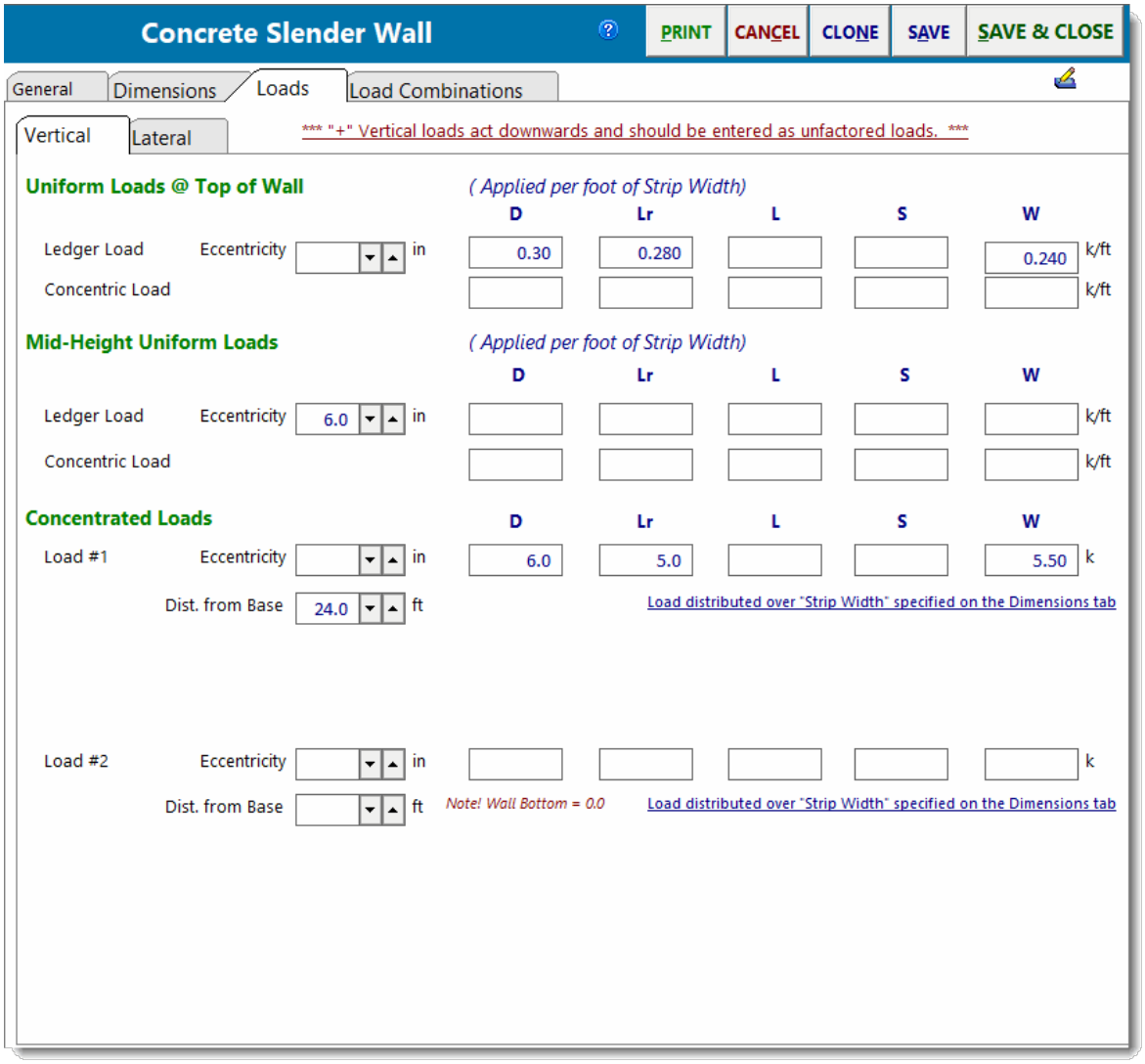
1st Story Height
Distance from the bottom of the wall to the first lateral support.

2nd Story Height
Distance from the first lateral support to the top lateral support.

Loads - Vertical

A variety of vertical loads are available. Note the hints describing whether the load is per foot or on the entire strip width.

All loads that are entered on this tab will be multiplied by the load factors specified on the Load Combination sub-tabs. So these magnitudes should be specified with those load factors in mind.



© 2026 ENERCALC, LLC

Ledger Load

This is a per-foot vertical load applied to the wall at an optional eccentricity. So if you have a 48" strip width and specify a 1 k/ft dead load then the strip will have a total of 4 kip applied due to the 1 k/ft entry.

Eccentricity

Describes an offset from the mid-thickness of the wall panel, which is the default location of application of a vertical load. Enter this value as a positive number when the load is shifted toward the inside of the wall.

Concentric Load

This is a per-foot vertical load applied concentrically to the wall. So if you have a 48" strip width and specify a 1 k/ft dead load then the strip will have a total of 4 kip applied due to the 1 k/ft entry.

Mid-Height Vertical Uniform Load

This load entry is only shown for 2-story walls. It allows you to specify two uniform loads applied at the "1st Story" height, one of which can have an eccentricity from the wall center.

Concentrated Loads

These are single concentrated vertical loads applied to the wall "strip width" with an optional eccentricity.

Distance from Base is the height at which the load is applied.

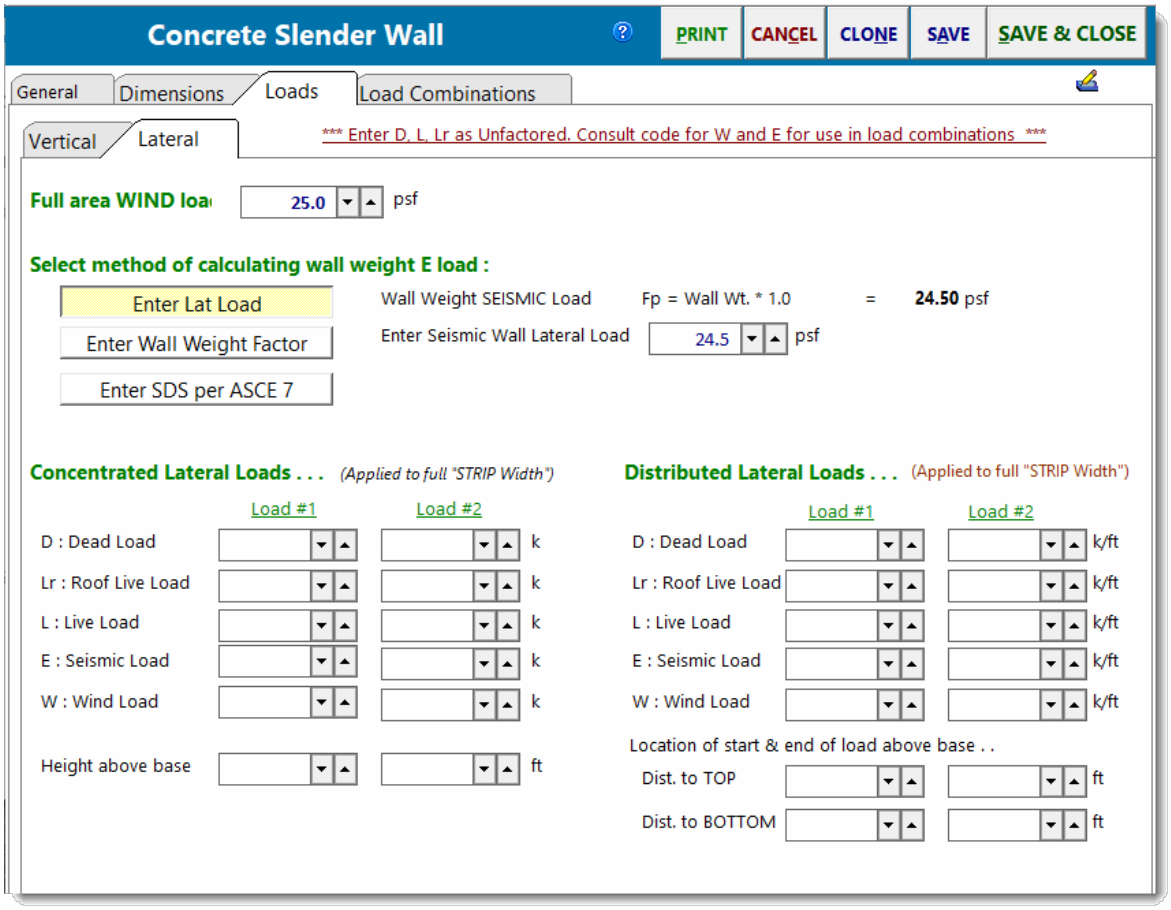
Eccentricity describes an offset from the mid-thickness of the wall panel, which is the default location of application of a vertical load. Enter this value as a positive number when the load is shifted toward the inside of the wall.

Loads - Lateral

Lateral loads are applied perpendicular to the plane of the wall and are almost always seismic or wind. These loads create out of plane deflection of the wall, which the module will use to develop P-Delta effects to calculate secondary moments in the wall. Recall that this module divides the wall into small segments and calculates the allowable and actual forces and deflections for each small segment. In this way the lateral loads are properly modeled on what is effectively a beam with variable stiffness due to the state of cracking in each segment.

© 2026 ENERCALC, LLC

All loads that are entered on this tab will be multiplied by the load factors specified on the Load Combination sub-tabs. So these magnitudes should be specified with those load factors in mind. All lateral loads must be entered as positive values.



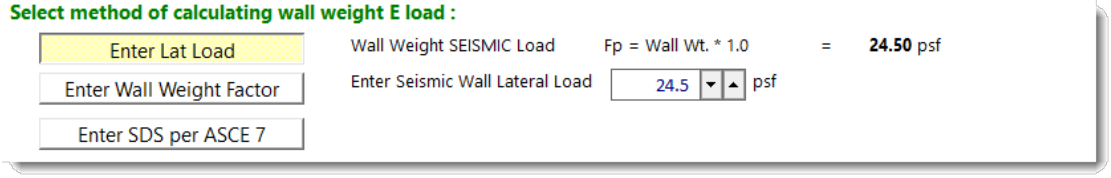
Full area WIND Load

Enter the wind load that will be applied to the wall in the out-of-plane direction. This load will only be applied to one surface of the wall, and as such, the magnitude must take into consideration both the internal and external pressures.

Wall Weight SEISMIC Load

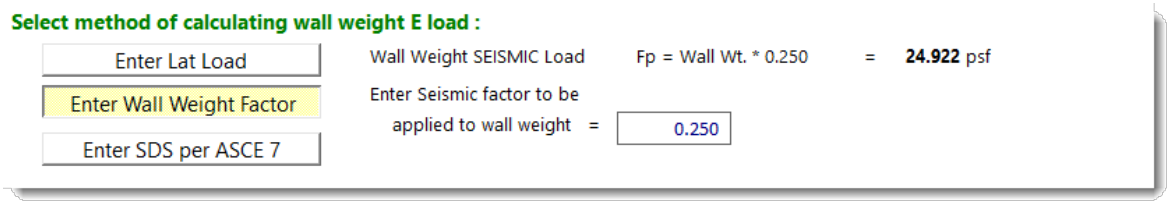
This section offers three options to specify the seismic load that will be applied to the wall in the out-of-plane direction:

Enter Lateral Load: This entry is a simple net load applied to the wall (but will still be factored by the load combination factors for "E").

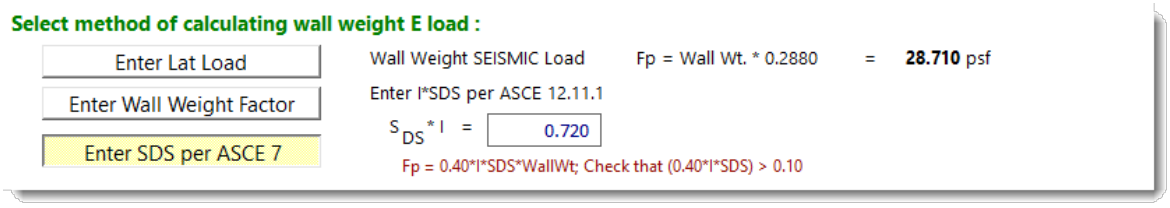


© 2026 ENERCALC, LLC

Enter Wall Weight Factor: Enter a number that will be multiplied by the self-weight of the wall. For example, if you enter 0.25 and the wall weighs 80 psf, then a 20.00 psf out-of-plane load will be calculated and applied to the wall using the load combination factors for "E".



Enter S_{DS} per ASCE-05: Enter the (S_{DS} * I) value as prescribed by the ASCE code for the building location. The minimum calculated load value of 10 psf or (0.4 * Value Entered * Wall Weight) will be applied to the wall using the load combination factors for "E".



Fp

This is the actual seismic load applied perpendicular to the plane of the wall, which represents the wall's seismic self weight load.

Concentrated Lateral Loads

This is an added lateral load applied perpendicular to the plane of the wall. It acts on the full "Strip Width" and is factored by the load combination factors corresponding to the type of load.

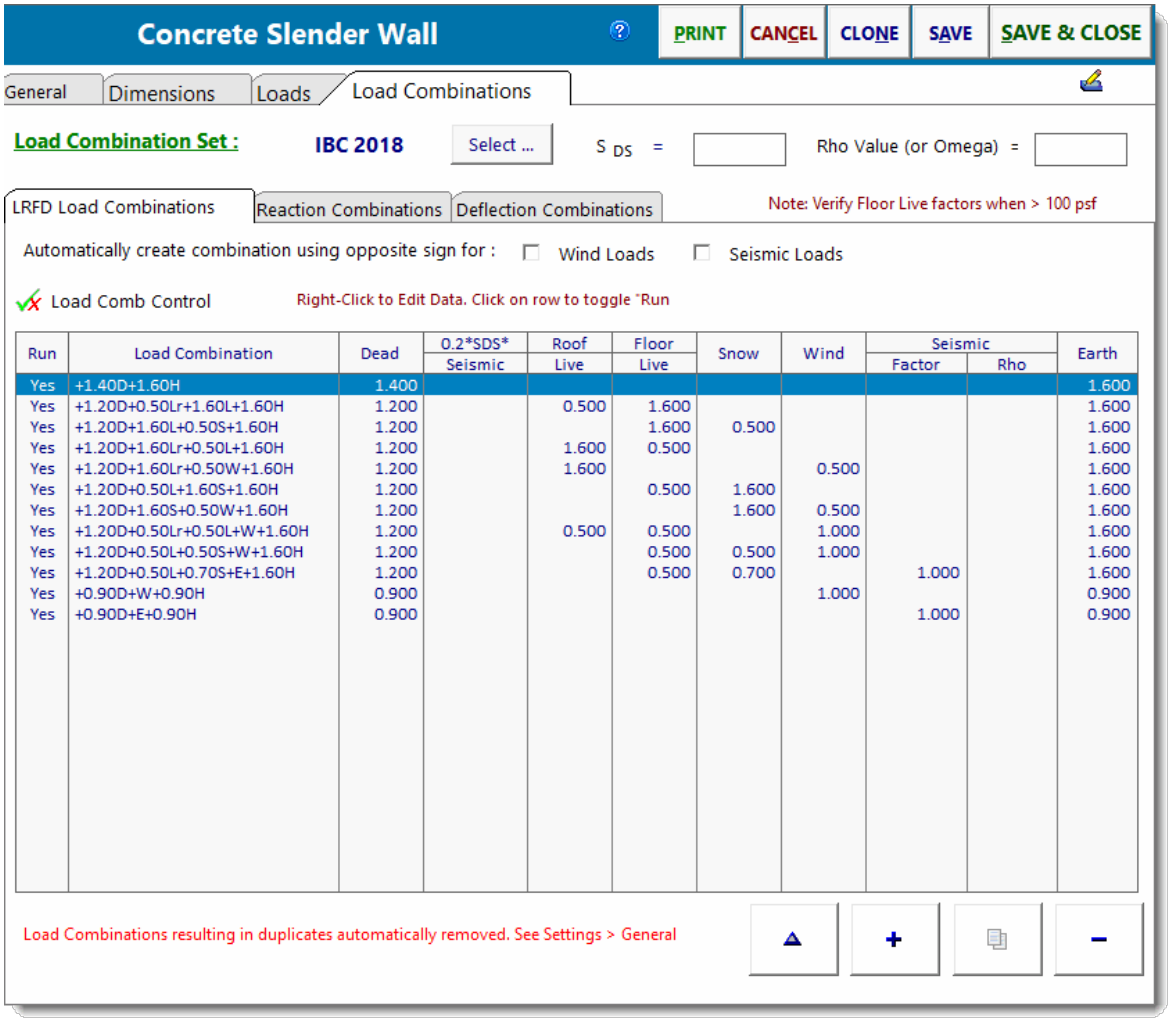
Distributed Lateral Loads

This is an added lateral uniform load applied out-of-plane to the wall. It acts on the full "Strip Width" and is factored by the load combination factors corresponding to the type of load. You also enter the start and end distance of the load extent above the base of the wall.

© 2026 ENERCALC, LLC

Load Combinations

Typical load combination information as used throughout **ENERCALC SEL**.

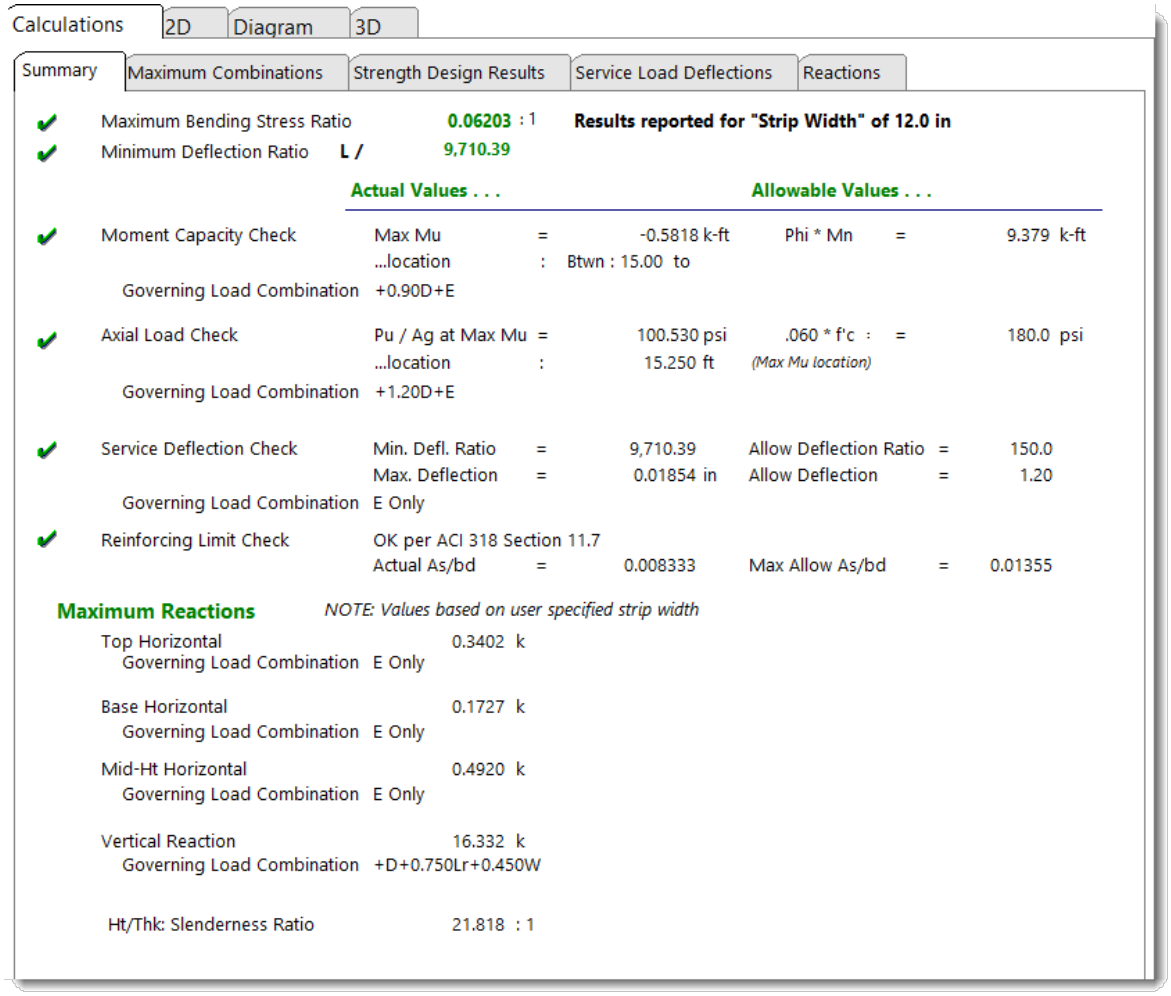


Run	Load Combination	Dead	0.2*SDS* Seismic	Roof Live	Floor Live	Snow	Wind	Seismic Factor	Seismic Rho	Earth
Yes	+1.40D+1.60H	1.400								1.600
Yes	+1.20D+0.50Lr+1.60L+1.60H	1.200		0.500	1.600					1.600
Yes	+1.20D+1.60L+0.50S+1.60H	1.200			1.600	0.500				1.600
Yes	+1.20D+1.60Lr+0.50L+1.60H	1.200		1.600	0.500					1.600
Yes	+1.20D+1.60Lr+0.50W+1.60H	1.200		1.600			0.500			1.600
Yes	+1.20D+0.50L+1.60S+1.60H	1.200			0.500	1.600				1.600
Yes	+1.20D+1.60S+0.50W+1.60H	1.200				1.600	0.500			1.600
Yes	+1.20D+0.50Lr+0.50L+W+1.60H	1.200		0.500	0.500		1.000			1.600
Yes	+1.20D+0.50L+0.50S+W+1.60H	1.200			0.500	0.500	1.000			1.600
Yes	+1.20D+0.50L+0.70S+E+1.60H	1.200			0.500	0.700		1.000		1.600
Yes	+0.90D+W+0.90H	0.900					1.000			0.900
Yes	+0.90D+E+0.90H	0.900						1.000		0.900

© 2026 ENERCALC, LLC

Summary

This tab presents the critical results as calculated by the module.



Maximum Bending Stress Ratio

The module looks at the detailed results for ALL strength design load combinations at all "segments" in the wall and pulls out the maximum factored load bending stress ***ratio*** to present here as the governing condition.

Minimum Deflection Ratio

The module looks at the detailed results for ALL service load combinations at all "segments" in the wall and pulls out the minimum service load deflection ratio (meaning maximum deflection) to present here as the governing condition.

Moment Capacity Check

For the condition of maximum bending stress ratio, the actual applied and allowable bending moments are given along with the governing load combination.

© 2026 ENERCALC, LLC

Axial Load Check

The module determines the actual axial stress in the segment (and for the load combination) that has the highest ***bending moment***, and reports that axial stress as Pu/Ag. The allowable value is the result of the user's entry for maximum percentage of f'c to use.

Service Deflection Check

For the condition of minimum deflection ratio (meaning maximum deflection) the ratio, deflection, allowable minimum ratio, allowable deflection (based on allowable ratio) and governing load combination are reported.

Reinforcing Limit Check

The module checks all portions of the wall for reinforcing (including differently reinforced first and second stories and reveal areas). It reports the maximum reinforcing ratio and compares it with the maximum percentage of balanced section analysis As allowed.

Minimum Moment Check

ACI specifies that a wall section in bending shall have a minimum strength phiMn that is greater than the cracking strength Mcr = Sgross * Fr.

Maximum Reactions

This gives a summary of the maximum reactions (both out-of-plane and vertical) along with the load combination that creates them.

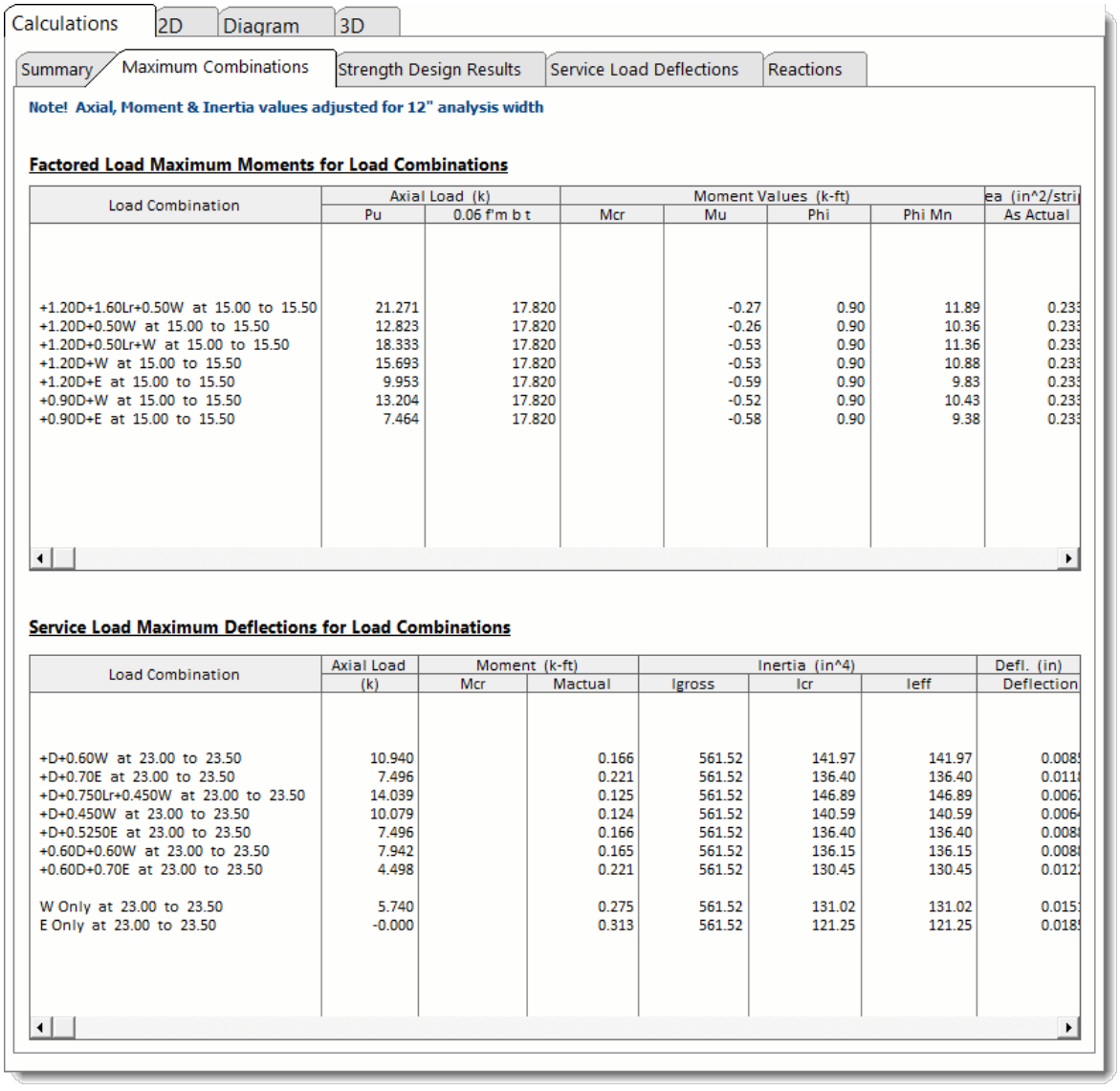
© 2026 ENERCALC, LLC

Maximum Combinations

This tab provides a summary of the governing values for each load combination for both factored load axial & bending and service load deflections.

Factored Load Maximum Moments for Load Combinations: The module looks through the result set for each load combination and identifies the location above the base of the wall at which the maximum condition is found. Note that "Aseff" is the effective area of steel and is influenced by the axial compression in that segment.

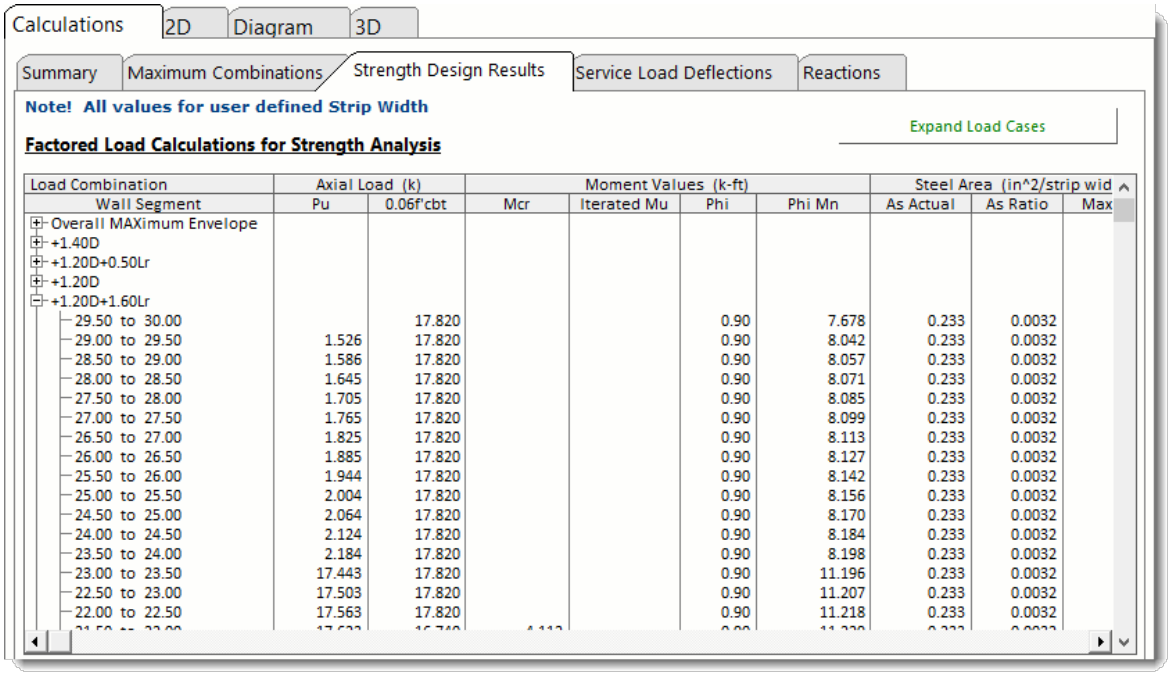
Service Load Maximum Deflections for Load Combinations: The module looks through the result set for each load combination and identifies the location above the base of the wall at which the maximum out-of-plane deflection is found. The value for "Ieff" is specific to the segment at that location and is based on the actual moment and Bischoff's equation for calculating effective moment of inertia.



© 2026 ENERCALC, LLC

Strength Design Results

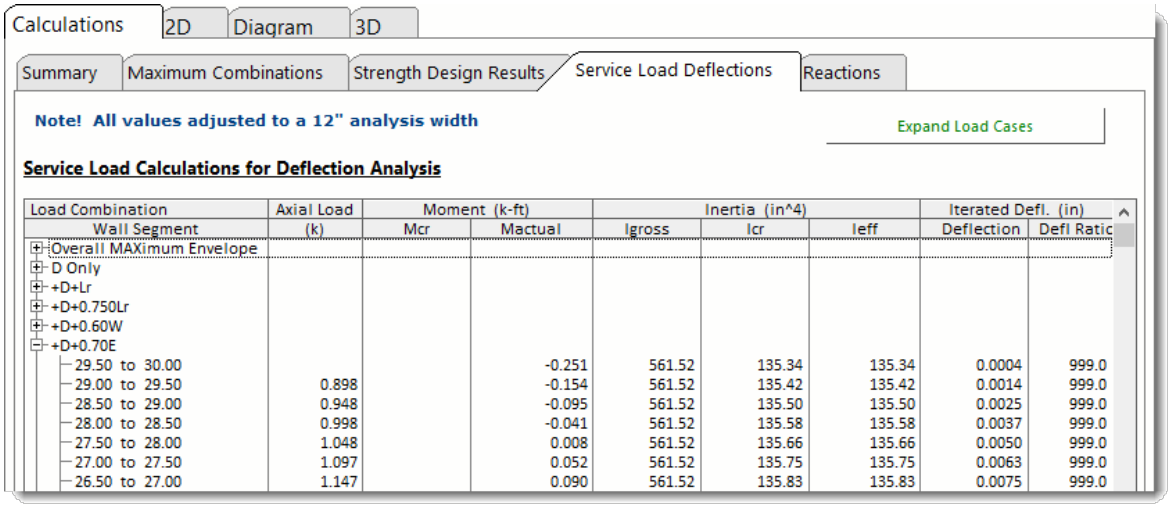
This tab provides an extremely detailed summary of the factored axial load, moments, effective steel area and moment of inertia at each wall analysis segment for each load combination.



Load Combination / Wall Segment	Axial Load (k) Pu	Axial Load (k) 0.06f'cbt	Moment Values (k-ft) Mcr	Moment Values (k-ft) Iterated Mu	Moment Values (k-ft) Phi	Moment Values (k-ft) Phi Mn	Steel Area (in^2/strip wid As Actual	Steel Area (in^2/strip wid As Ratio	Steel Area (in^2/strip wid Max
Overall MAXimum Envelope									
+1.40D									
+1.20D+0.50Lr									
+1.20D									
+1.20D+1.60Lr									
29.50 to 30.00		17.820			0.90	7.678	0.233	0.0032	
29.00 to 29.50	1.526	17.820			0.90	8.042	0.233	0.0032	
28.50 to 29.00	1.586	17.820			0.90	8.057	0.233	0.0032	
28.00 to 28.50	1.645	17.820			0.90	8.071	0.233	0.0032	
27.50 to 28.00	1.705	17.820			0.90	8.085	0.233	0.0032	
27.00 to 27.50	1.765	17.820			0.90	8.099	0.233	0.0032	
26.50 to 27.00	1.825	17.820			0.90	8.113	0.233	0.0032	
26.00 to 26.50	1.885	17.820			0.90	8.127	0.233	0.0032	
25.50 to 26.00	1.944	17.820			0.90	8.142	0.233	0.0032	
25.00 to 25.50	2.004	17.820			0.90	8.156	0.233	0.0032	
24.50 to 25.00	2.064	17.820			0.90	8.170	0.233	0.0032	
24.00 to 24.50	2.124	17.820			0.90	8.184	0.233	0.0032	
23.50 to 24.00	2.184	17.820			0.90	8.198	0.233	0.0032	
23.00 to 23.50	17.443	17.820			0.90	11.196	0.233	0.0032	
22.50 to 23.00	17.503	17.820			0.90	11.207	0.233	0.0032	
22.00 to 22.50	17.563	17.820			0.90	11.218	0.233	0.0032	

Service Load Deflections

This tab provides an extremely detailed summary of the service axial load, moments, effective moment of inertia and calculated deflection at each wall analysis segment for each load combination.

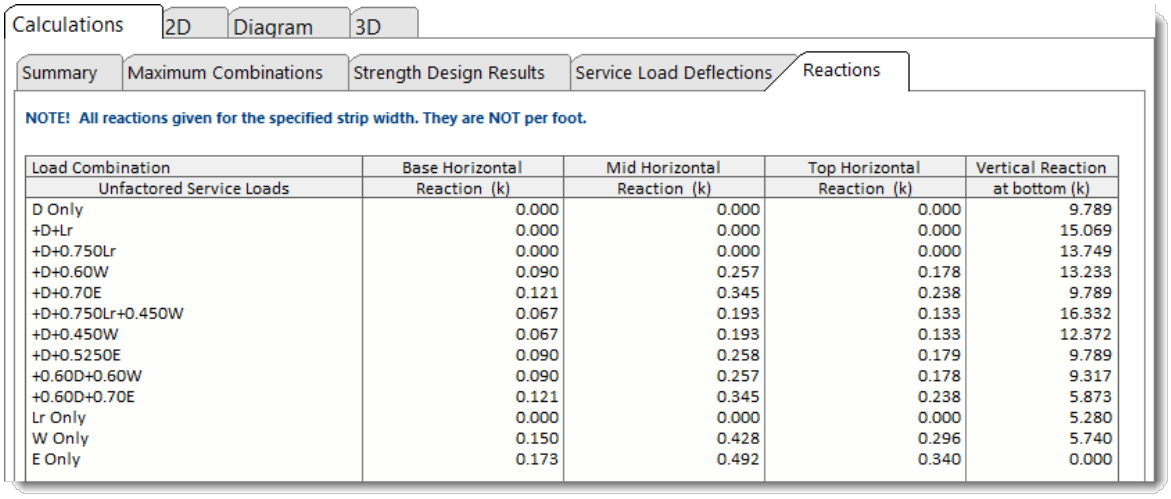


Load Combination / Wall Segment	Axial Load (k)	Moment (k-ft) Mcr	Moment (k-ft) Mactual	Inertia (in^4) Igross	Inertia (in^4) Icr	Inertia (in^4) Ieff	Iterated Defl. (in) Deflection	Iterated Defl. (in) Defl Ratio
Overall MAXimum Envelope								
D Only								
+D+Lr								
+D+0.750Lr								
+D+0.60W								
+D+0.70E								
29.50 to 30.00			-0.251	561.52	135.34	135.34	0.0004	999.0
29.00 to 29.50	0.898		-0.154	561.52	135.42	135.42	0.0014	999.0
28.50 to 29.00	0.948		-0.095	561.52	135.50	135.50	0.0025	999.0
28.00 to 28.50	0.998		-0.041	561.52	135.58	135.58	0.0037	999.0
27.50 to 28.00	1.048		0.008	561.52	135.66	135.66	0.0050	999.0
27.00 to 27.50	1.097		0.052	561.52	135.75	135.75	0.0063	999.0
26.50 to 27.00	1.147		0.090	561.52	135.83	135.83	0.0075	999.0

© 2026 ENERCALC, LLC

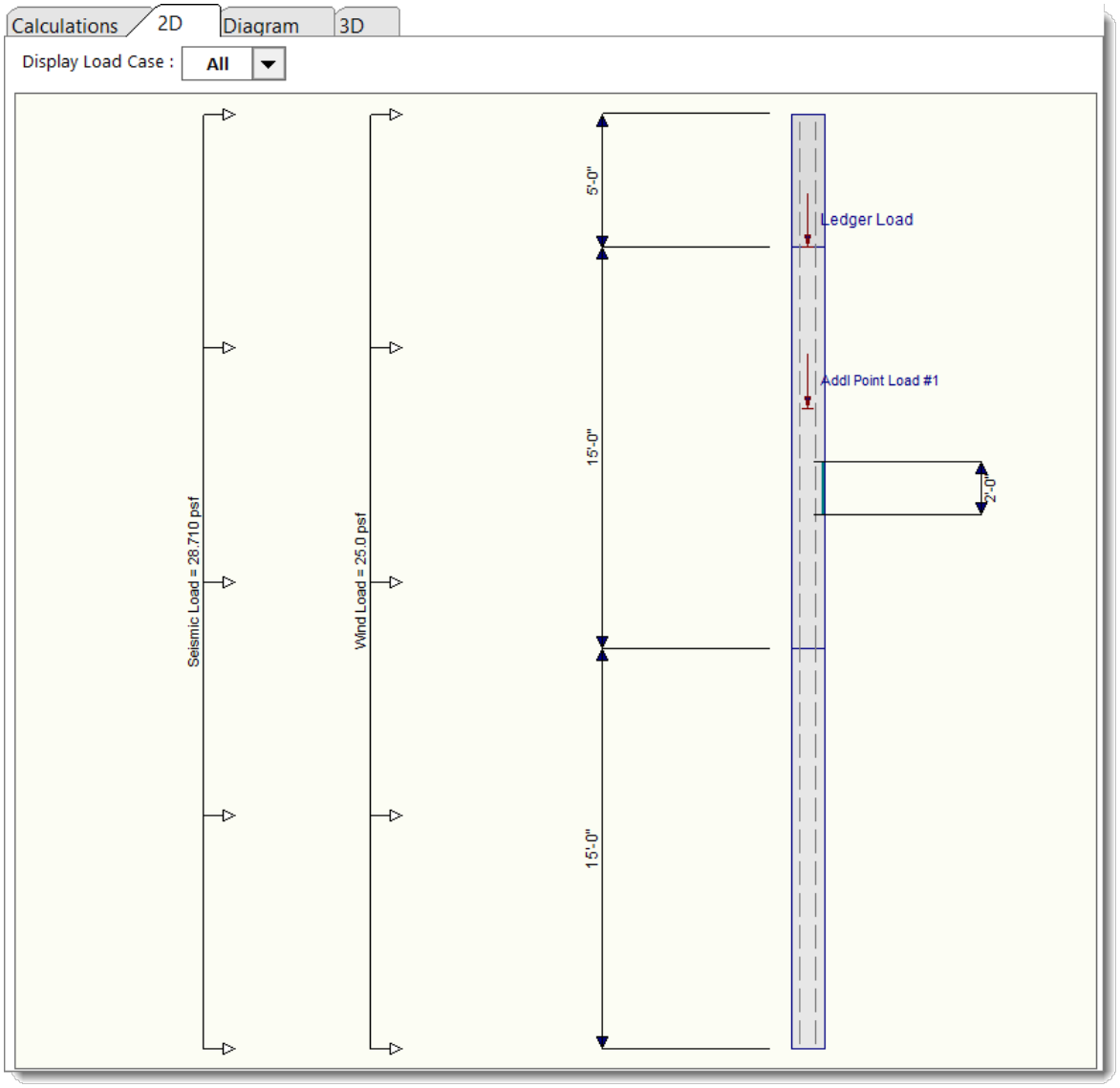
Reactions

This tab gives a summary of out-of-plane and vertical base reactions for each service load combination.



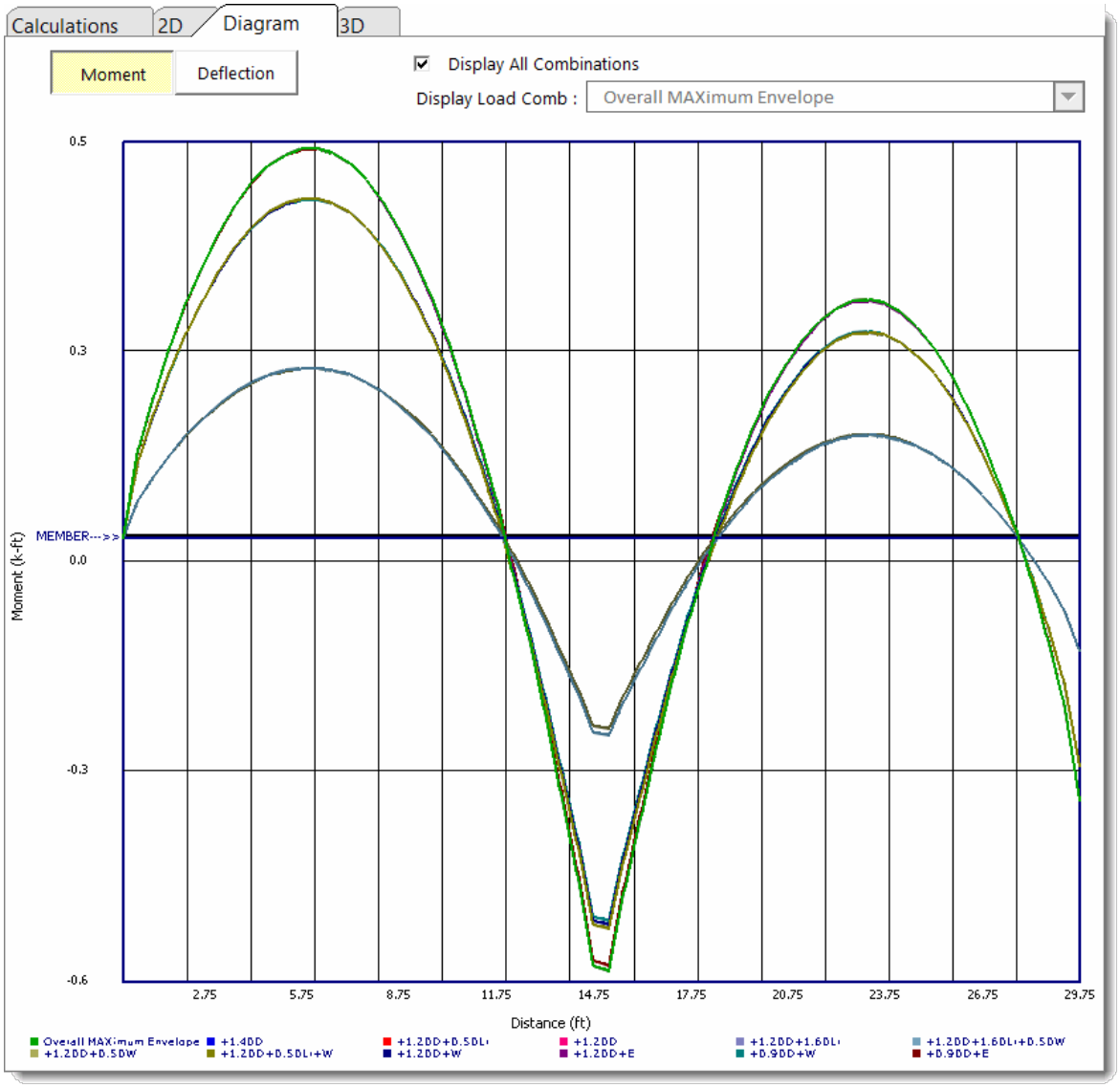
© 2026 ENERCALC, LLC

2D Sketch



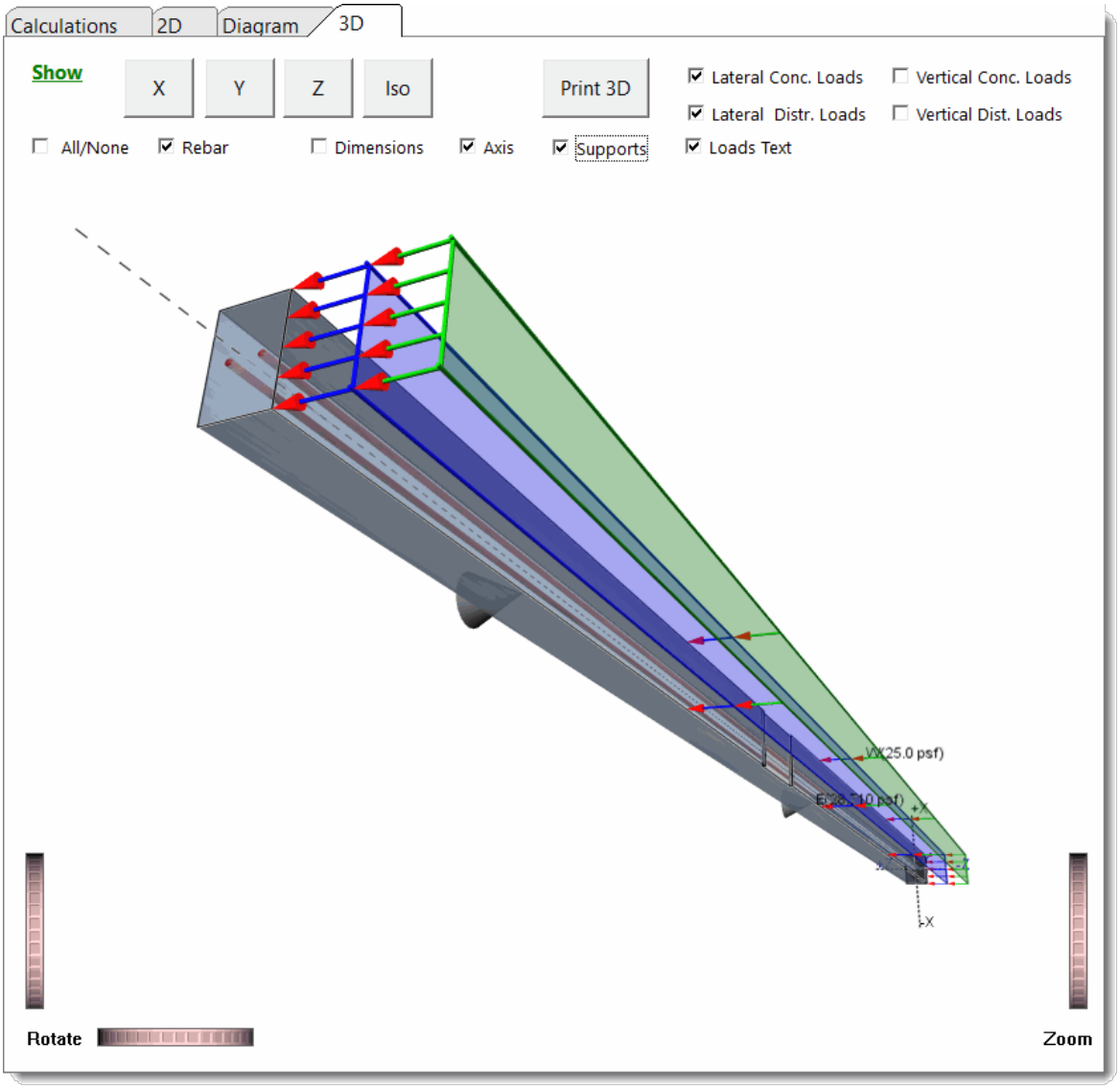
© 2026 ENERCALC, LLC

Diagram



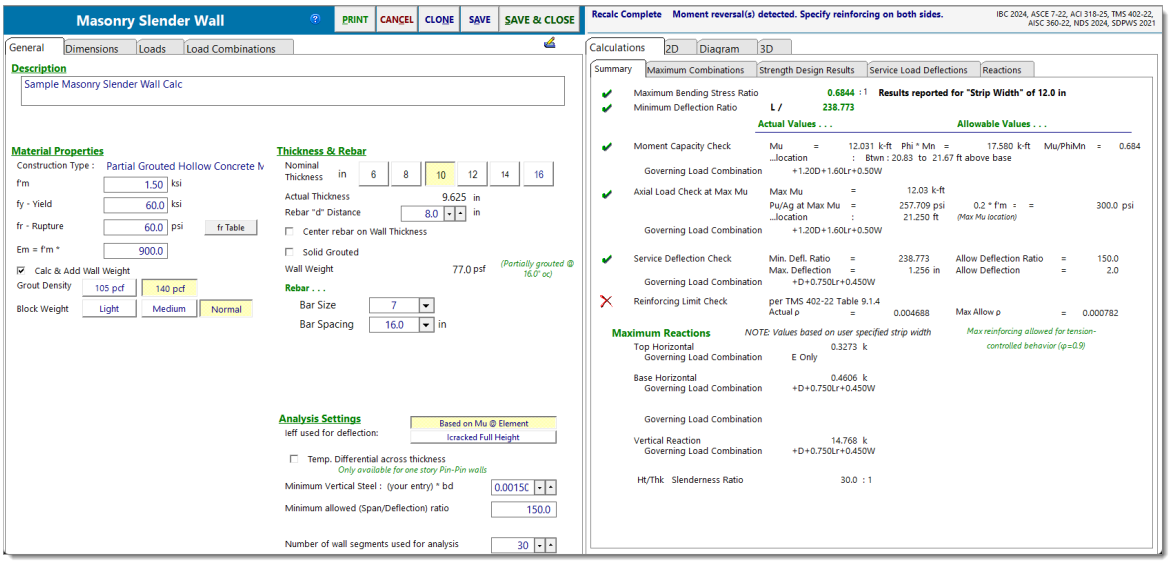
© 2026 ENERCALC, LLC

3D Rendering



13.7.1.2 Masonry Slender Wall

This module provides design and analysis according to the new provisions for design of masonry walls, using the P-Delta deflection considerations now included in the IBC.



This method lifts the restriction on H/t ratios, and performs wall analysis using the principles of ultimate strength design. The design method is very similar to that of the Concrete Slender Wall module.

This module uses a variable width strip of wall section to represent a typical section of wall. The module has the ability to apply a lateral wind load, seismic load, partial length uniform lateral load, and a lateral point load to the clear span of the wall section. This variety of loadings should take care of almost every lateral loading case possible.

The user may specify masonry and reinforcing strengths, seismic factor, wind load, vertical and lateral loads, vertical load eccentricities, and wall construction. The module determines the wall capacity, actual deflections considering P-Delta effects, and solves for the final moments obtained through iteration of the P-Delta effects. Deflection analysis is provided for both service and factored load cases.

The user reaches a final design by modifying wall thickness, rebar size, and rebar spacing until no overstress condition exists and the deflection limits prescribed in the code are satisfied.

This module uses basic principles of structural mechanics to model the wall as a series of beam segments. For each segment the actual moment is used to calculate member stiffness using the $I_{effective}$ equations developed by Peter H. Bischoff. Since these changes to wall stiffness affect the wall deflection profile, the program performs an iterative analysis of calculating moments (including P-Delta effects). The results are deflection curves almost exactly matching the SCCACI-SEAOSC test results. This makes this module far more accurate at calculating wall deflections and P-Delta effects than the simple equations in the ACI code.

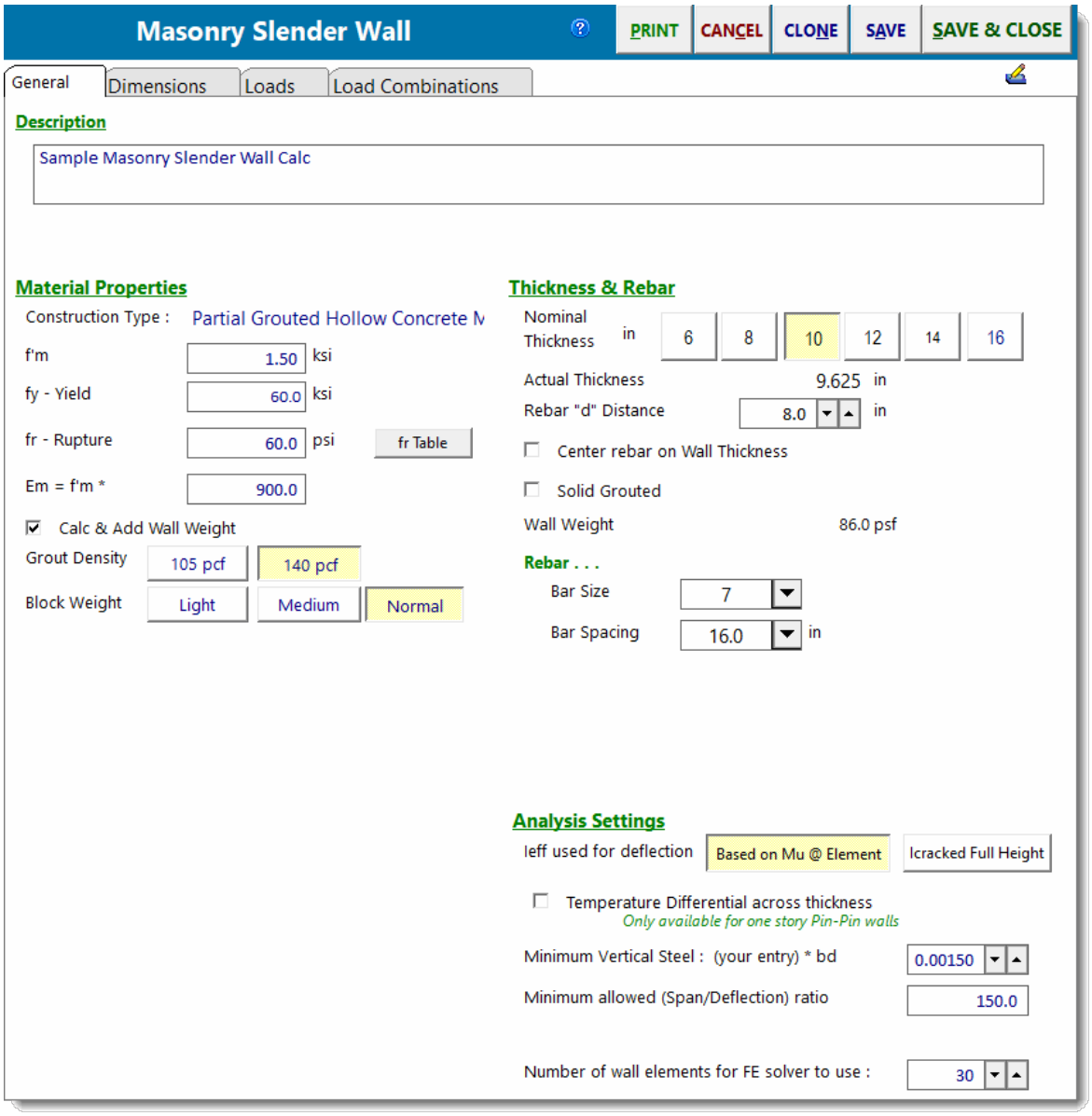
© 2026 ENERCALC, LLC

Capabilities

This module provides these capabilities:

- One or two story slender masonry walls
- Iterative process accounts for P-delta
- Optional parapet
- Axial loads with optional eccentricities
- Wind, seismic and user defined lateral loads creating bending on the wall panel
- Variable strip width to model the wall panel
- Temperature differential can be specified across thickness of wall to add curvature

General



© 2026 ENERCALC, LLC

Material Properties

f'm

Enter the allowable masonry strength to be used in the analysis. The allowable bending and axial stresses calculated from f'm are outlined in a later section.

fy

Yield point stress of reinforcing.

fr - Rupture & fr-Table

Modulus of rupture for the masonry wall system. Note: This module does not support fully ungrouted walls; however, fr values for ungrouted walls are available as a conservative reference for partially grouted walls.

Table 9.1.9.2 - Strength Design TMS 402-22

Fr : Modulus of Rupture

Click button to select

	Portland Cement/Lime or Mortar Cement		Masonry Cement or Air Entrained Portland Cement/Lime	
	Type M or S	Type N	Type M or S	Type N
Tensile stress normal to joint				
Solid Masonry Units	133	100	80	51
Hollow Units - Ungrouted *	84	64	51	31
Hollow Units - Fully Grouted *	163	158	153	145
Tensile stress parallel to joint				
Solid Masonry Units	267	200	160	100
Hollow Units - Ungrouted and Partially Grouted	167	127	100	64
Hollow Units - Fully Grouted	267	200	160	100

All values in psi

* Note: The TMS permits interpolation between partial and fully grouted cross sections. This module does not support fully ungrouted walls, but Fr values for ungrouted walls are provided for reference.

Close

Em = f'm * [value]

The modulus of elasticity of the masonry wall system is specified by this value acting as a multiplier to f'm.

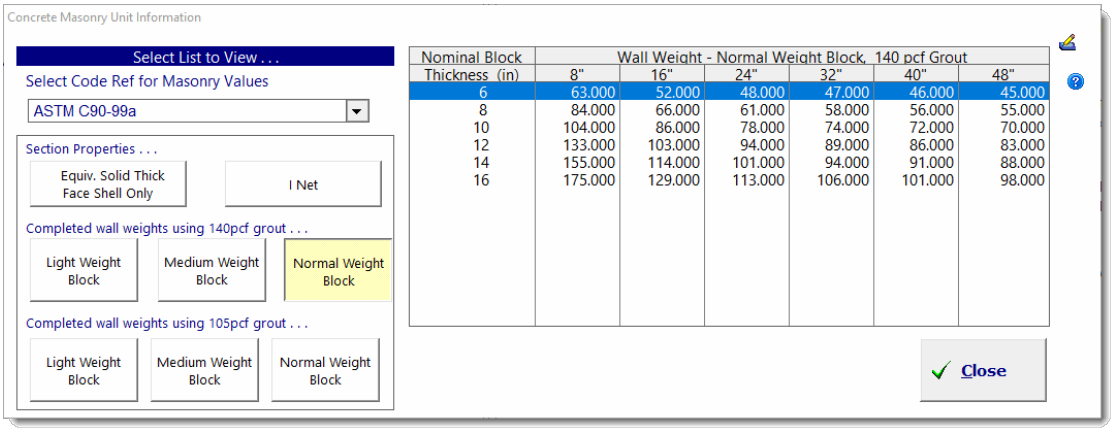
Grout Density

Choose from two different options for the density of the grout.

© 2026 ENERCALC, LLC

Block Weight

Select light, medium and normal weight block. The weight of a completed wall is determined from the masonry database, depending on the block weight, grout density, and grouted cell spacing. To view the database values click **Databases > Concrete Masonry Unit Data** from the main menu. Here is what you will see:



Thickness & Rebar

Nominal Thickness

Select the nominal thickness for the concrete masonry units used in the wall construction. This selection will pull the values for wall weight, equivalent solid thickness and Igross from the masonry database (see above).

Actual Thickness

The actual wall thickness for the nominal selection.

Rebar "d" Distance

Enter the distance from the extreme compression fiber to the center of the rebar.

Solid Grouted

Select this checkbox if the wall is to be grouted solid . If unchecked the module will calculate the wall weight considering that grouted cells only occur at the spacing of the reinforcing.

Wall Weight

Weight of wall as retrieved from the masonry database. Value is based on specified wall thickness, grout density, block type, and grouting frequency.

Rebar Size & Spacing

Enter the rebar size and spacing.

© 2026 ENERCALC, LLC

Analysis Settings

Ieff used for deflection

The module always performs an iterative analysis for moments and deflections using progressively greater wall deflections due to increasing P-Delta effects.

Temperature Differential across thickness

This input is used to describe the temperature change between each face of the wall. A temperature change induces a slight curvature into the wall because the hotter side expands, resulting in a slightly higher out-of-plane deflection.

Enter temperature differentials as positive values. The effects of a specified temperature differential are always additive with the bending and deflection resulting from other applied loads.

Minimum Vertical Steel: %/100

Minimum steel percentage as a portion of the gross wall area.

Minimum Allowed (Span/Deflection) Ratio

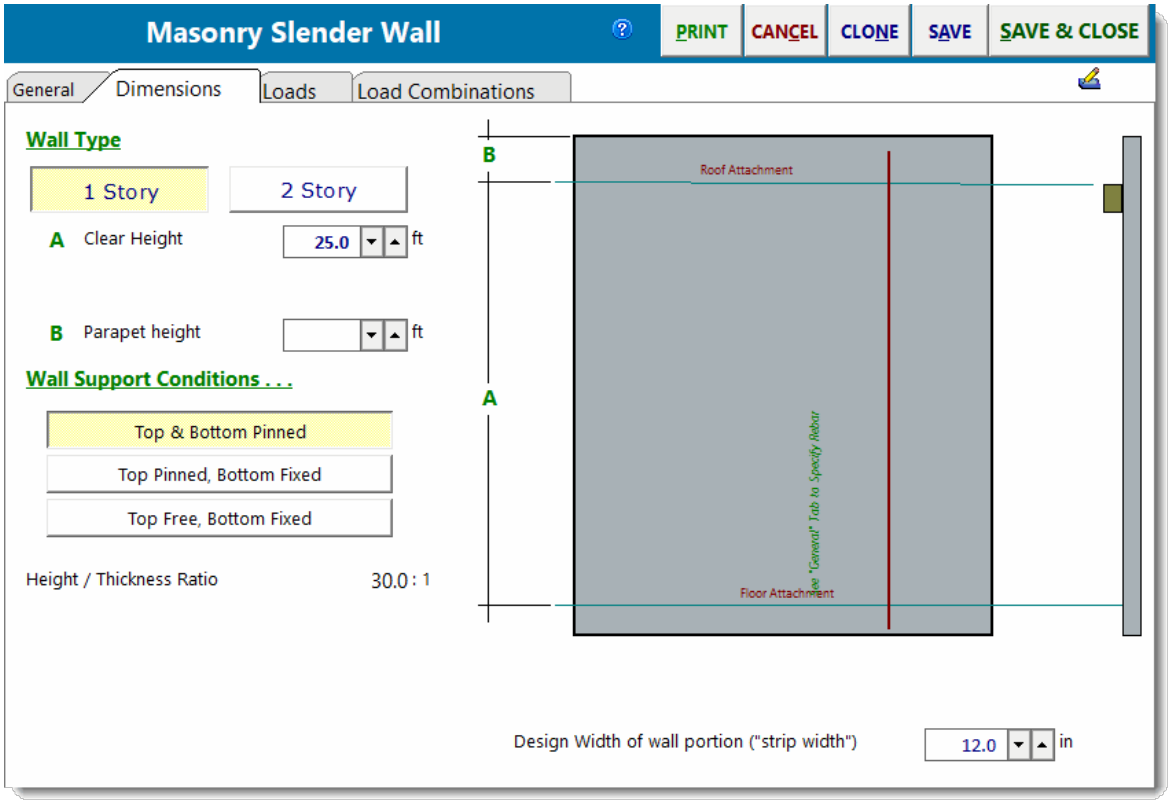
This setting establishes the minimum allowable ratio of span length to service load deflection. If a lower actual Span/Deflection ratio occurs (meaning greater deflection), a warning message will be displayed.

Number of wall elements for FE solver to use

This module divides the wall design strip into segments from the base to the top for analysis purposes. Use this entry to define the number of segments to use. Experience demonstrates that approximately 30-40 segments gives a good balance between the iterative P-Delta analysis reaching convergence and excessive calculation time.

© 2026 ENERCALC, LLC

Dimensions



Clear Height

Span of the wall between the base and the first lateral support. For one story walls this is the top support. For 2 story walls this is this prompt will change to be "1st story height".

Parapet Height

Distance the wall extends above the topmost lateral support (clear height for one story wall, 2nd story height for 2 story walls)

Wall Support Conditions

Controls how the top and bottom of the wall are restrained for moments and lateral movement.

[Top & Bottom Pinned]

Base of wall is restrained against movement out of plane and vertically, rotates freely. Top of wall restrained against out of plane movement and can move vertically and rotate freely.

[Top Pinned, Bottom Fixed]

Base of wall is restrained against movement about all three axes. Top of wall restrained against out of plane movement and can move vertically and rotate freely.

© 2026 ENERCALC, LLC

[Top Free, Bottom Fixed]

Base of wall is restrained against movement about all three axes. Top of wall is completely free making this a cantilevered wall.

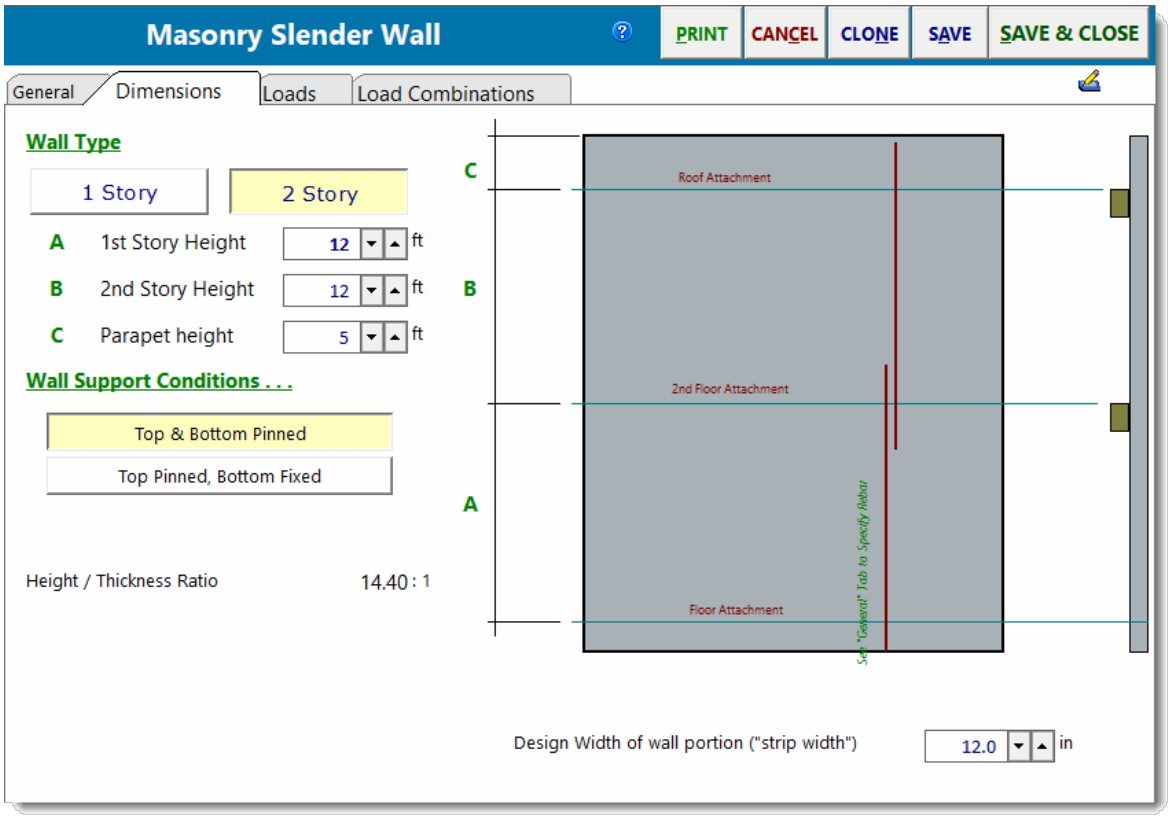
Design Width of wall portion ("strip width")

This module performs its analysis for this width. Results are for either for this width or a 12" width as noted where the results are provided.

Note that applied loads either are applied to the entire strip width (as for concentrated vertical and lateral loads) or are entered on a per-foot basis when they are uniform loads.

Two Story...

When a two story wall is selected this tab changes slightly to provide the 2nd story height and remove the Fixed-Free support option.



1st Story Height

Distance from the bottom of the wall to the first lateral support.

2nd Story Height

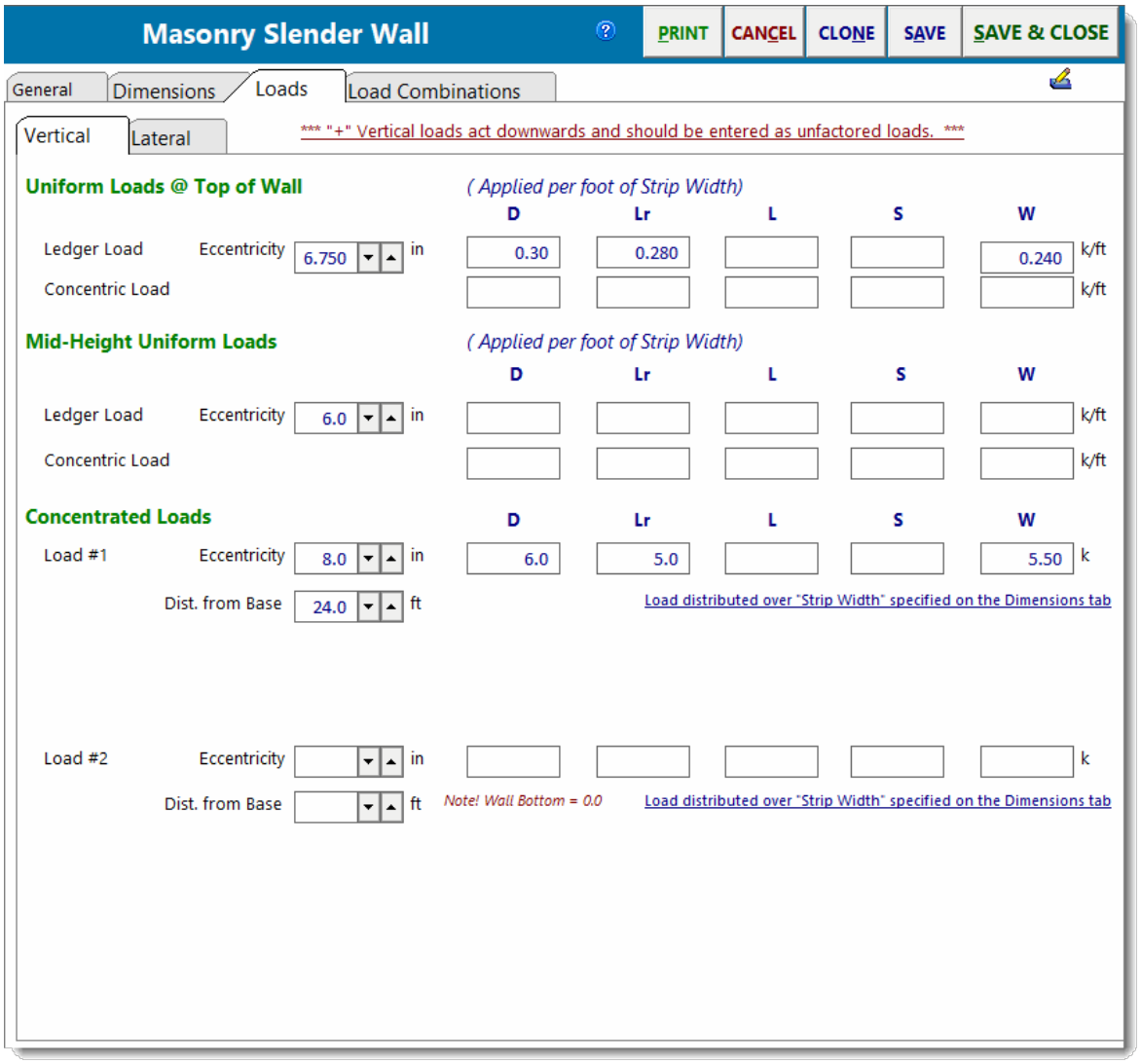
Distance from the first lateral support to the top lateral support.

© 2026 ENERCALC, LLC

Loads - Vertical

A variety of vertical loads are available. Note the hint describing whether the load is per foot or on the entire strip width.

All loads that are entered on this tab will be multiplied by the load factors specified on the Load Combination sub-tabs. So these magnitudes should be specified with those load factors in mind.



Ledger Load

This is a per-foot vertical load applied to the wall at an optional eccentricity. So if you have a 48" strip width and specify a 1 k/ft dead load then the strip will have a total of 4 kip applied due to the 1 k/ft entry.

© 2026 ENERCALC, LLC

Concentric Load

This is a per-foot vertical load applied concentrically to the wall. So if you have a 48" strip width and specify a 1 k/ft dead load then the strip will have a total of 4 kip applied due to the 1 k/ft entry.

Mid-Height Vertical Uniform Load

This load entry is only shown for 2-story walls. It allows you to specify two uniform loads applied at the "1st Story" height, one of which can have an eccentricity from the wall center.

Concentrated Loads

These are single concentrated vertical loads applied to the wall "strip width" with an optional eccentricity.

Distance from Base is the height at which the load is applied.

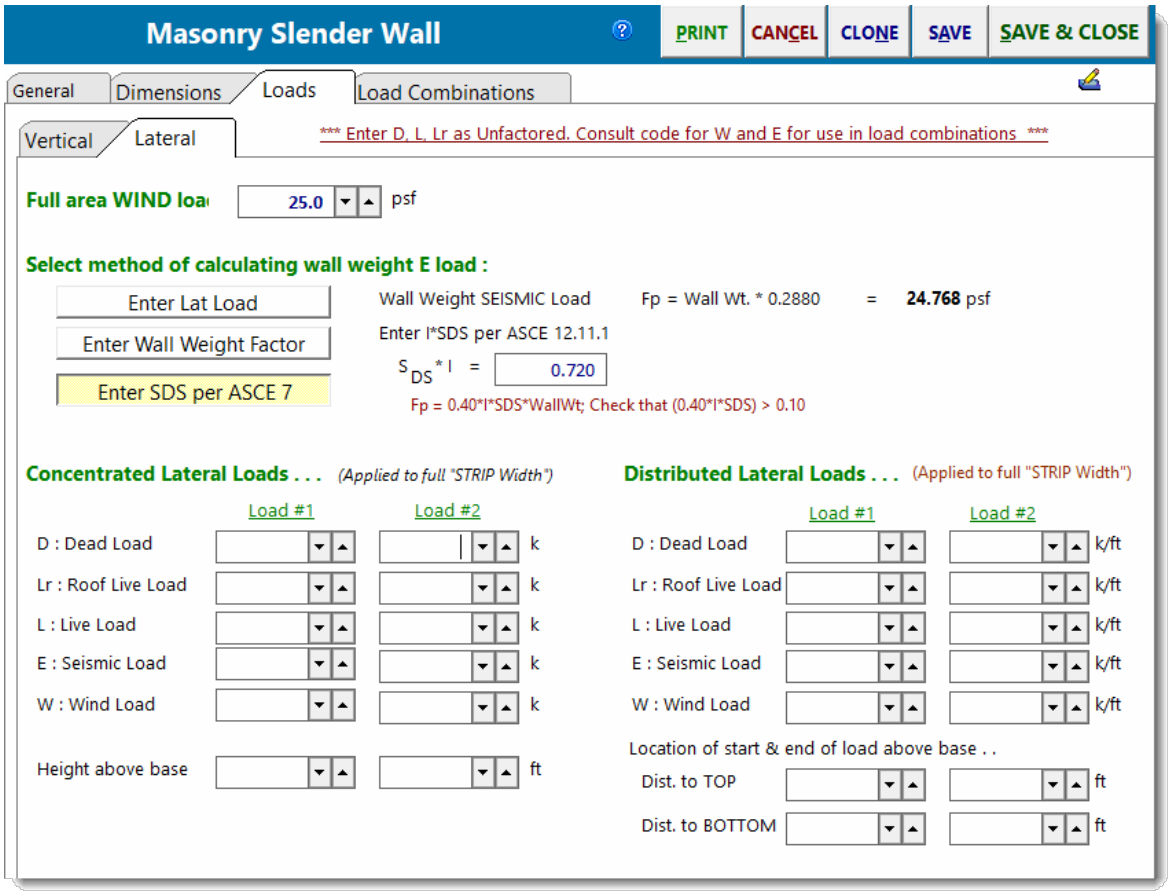
Eccentricity describes an offset from the mid-thickness of the wall panel, which is the default location of application of a vertical load. Enter this value as a positive number when the load is shifted toward the inside of the wall.

Loads - Lateral

Lateral loads are applied perpendicular to the plane of the wall and are almost always seismic or wind. These loads create out-of-plane deflection of the wall, which the module will use to develop P-Delta effects to calculate secondary moments in the wall. Recall from other explanations that the module divides the wall into small segments and calculates the allowable and actual forces and deflections for each small segment. In this way the lateral loads are properly modeled on what is a beam with variable stiffness due to the state of cracking in each segment.

© 2026 ENERCALC, LLC

All loads that are entered on this tab will be multiplied by the load factors specified on the Load Combination sub-tabs. So these magnitudes should be specified with those load factors in mind. All lateral loads must be entered as positive values.



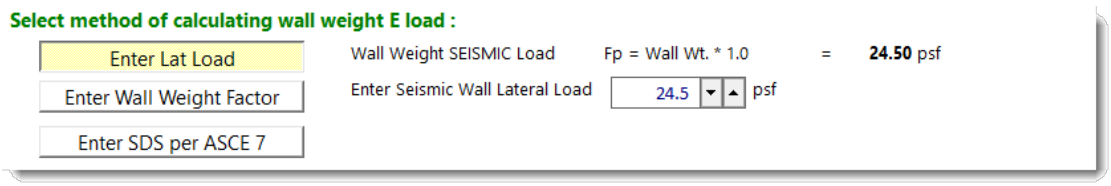
Full area WIND Load

Enter the wind load that will be applied to the wall in the out-of-plane direction. This load will only be applied to one surface of the wall, and as such, the magnitude must take into consideration both the internal and external pressures.

Wall weight seismic load

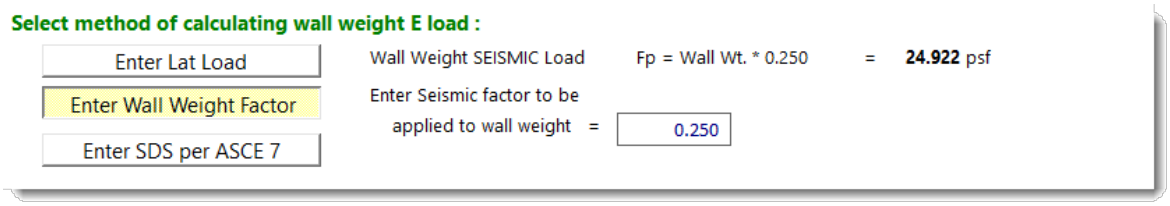
This section offers three options to specify the seismic load that will be internally applied to the wall in the out-of-plane direction:

Enter Lateral Load: This entry is a simple net load applied to the wall (but will still be factored by the load combination factors for "E").

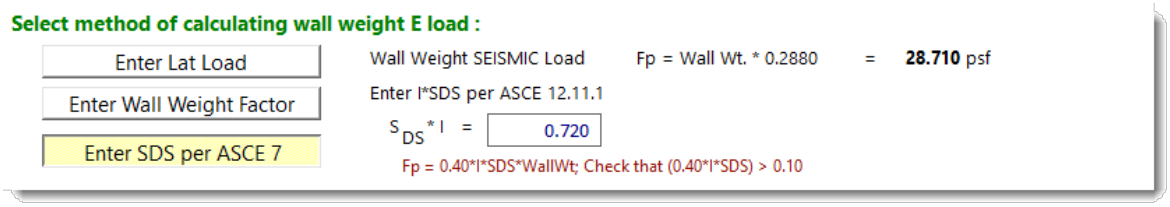


© 2026 ENERCALC, LLC

Enter Wall Weight Factor: Enter a number that will be multiplied by the self-weight of the wall. For example, if you enter 0.25 and the wall weighs 80 psf, then a 20.00 psf out-of-plane load will be calculated and applied to the wall using the load combination factors for "E".



Enter S_{DS} per ASCE 7: Enter the (S_{DS} * I) value as prescribed by the ASCE code for the building location. The minimum calculated load value of 10 psf or (0.4 * Value Entered * Wall Weight) will be applied to the wall using the load combination factors for "E".



Fp

This is the actual seismic load applied perpendicular to the plane of the wall, which represents the wall's seismic self weight load.

Concentrated Lateral Loads

This is an added lateral load applied perpendicular to the plane of the wall. It acts on the full "Strip Width" and is factored by the load combination factors corresponding to the type of load.

Distributed Lateral Loads

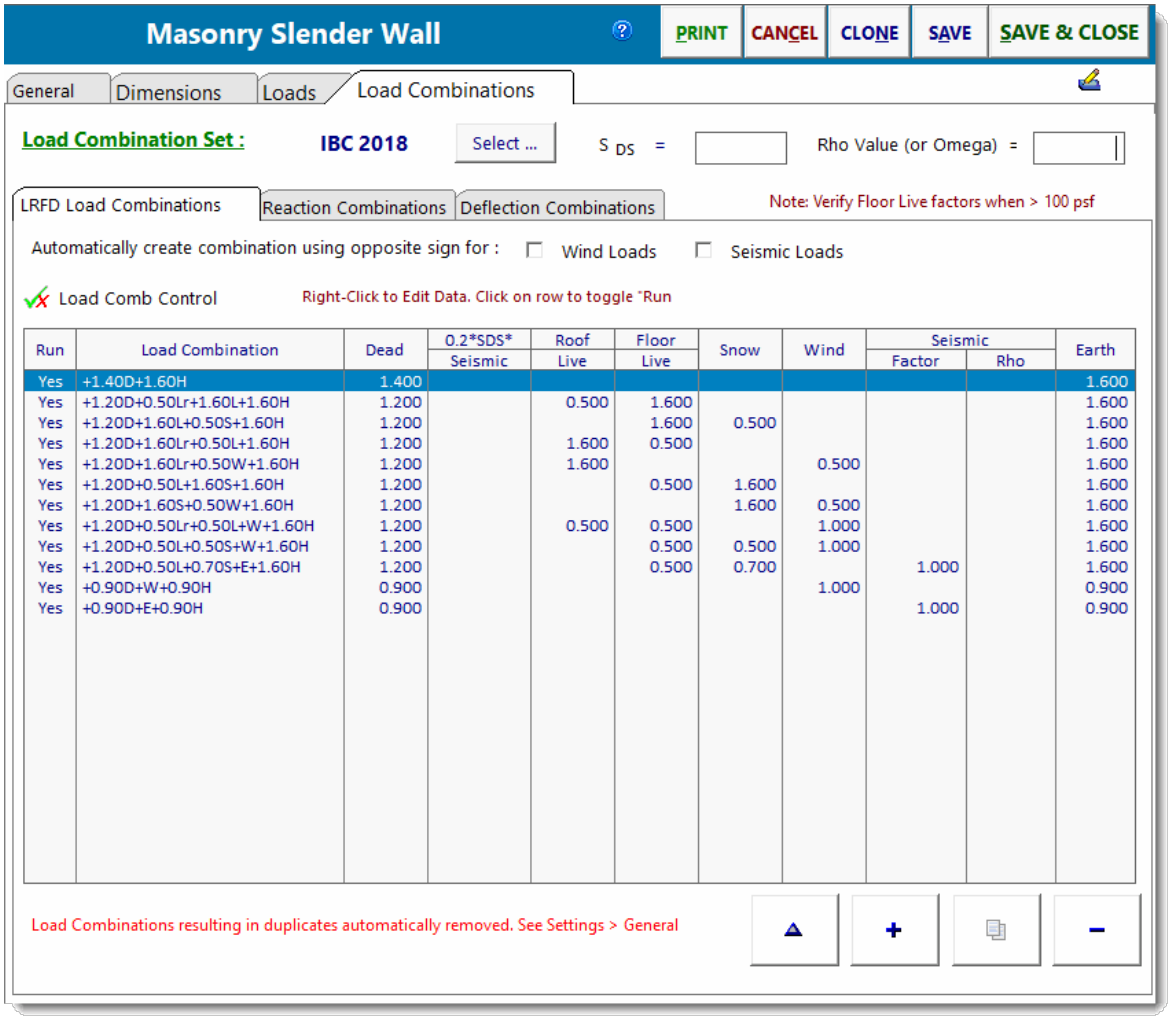
This is an added lateral uniform load applied perpendicular to the plane of the wall. It acts on the full "Strip Width" and is factored by the load combination factors corresponding to the type of load.

You also enter the start and end distance of the load extent above the base of the wall.

© 2026 ENERCALC, LLC

Load Combinations

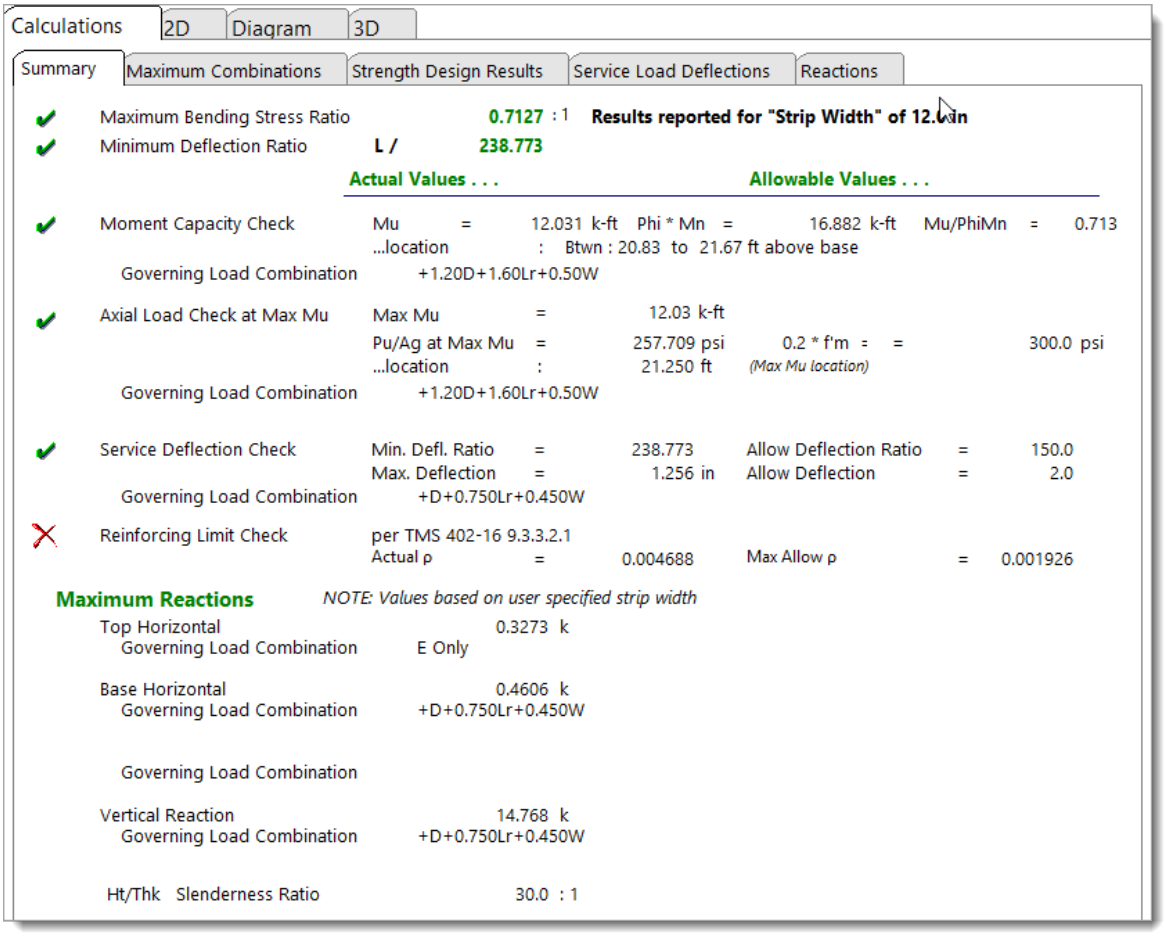
Typical load combination information as used throughout **ENERCALC SEL**.



Run	Load Combination	Dead	0.2*SDS* Seismic	Roof Live	Floor Live	Snow	Wind	Seismic Factor	Seismic Rho	Earth
Yes	+1.40D+1.60H	1.400								1.600
Yes	+1.20D+0.50Lr+1.60L+1.60H	1.200		0.500	1.600					1.600
Yes	+1.20D+1.60L+0.50S+1.60H	1.200			1.600	0.500				1.600
Yes	+1.20D+1.60Lr+0.50L+1.60H	1.200		1.600	0.500					1.600
Yes	+1.20D+1.60Lr+0.50W+1.60H	1.200		1.600			0.500			1.600
Yes	+1.20D+0.50L+1.60S+1.60H	1.200			0.500	1.600				1.600
Yes	+1.20D+1.60S+0.50W+1.60H	1.200				1.600	0.500			1.600
Yes	+1.20D+0.50Lr+0.50L+W+1.60H	1.200		0.500	0.500		1.000			1.600
Yes	+1.20D+0.50L+0.50S+W+1.60H	1.200			0.500	0.500	1.000			1.600
Yes	+1.20D+0.50L+0.70S+E+1.60H	1.200			0.500	0.700		1.000		1.600
Yes	+0.90D+W+0.90H	0.900					1.000			0.900
Yes	+0.90D+E+0.90H	0.900						1.000		0.900

Summary

This tab presents the critical results as calculated by the module.



Maximum Bending Stress Ratio

The module looks at the detailed results for ALL strength design load combinations at all "segments" in the wall and pulls out the maximum factored load bending stress ratio to present here as the governing condition.

Minimum Deflection Ratio

The module looks at the detailed results for ALL service load combinations at all "segments" in the wall and pulls out the minimum service load deflection ratio (meaning maximum deflection) to present here as the governing condition.

Moment Capacity Check

For the condition of maximum bending stress ratio, the actual applied and allowable bending moments are given along with the governing load combination.

© 2026 ENERCALC, LLC

Axial Load Check

The module checks the factored axial stress in all segments for all load combinations and gives the maximum actual stress Pu/Ag. The allowable value is the result of the wall slenderness. Per Section 9.3.5.4.2, if the slenderness is less than or equal to 30, the allowable value of factored axial stress is 0.20f'm. If the slenderness is greater than 30, the allowable value of factored axial stress is 0.05f'm.

Service Deflection Check

For the condition of minimum deflection ratio (meaning maximum deflection) the ratio, deflection, allowable minimum ratio, allowable deflection (based on allowable ratio) and governing load combination are reported.

Reinforcing Limit Check

The module checks all portions of the wall for reinforcing (including differently reinforced first and second stories) and reports the maximum reinforcing ratio and compares it with the maximum reinforcing ratio allowed by the TMS. See "Strength Design Results" below for more info.

Minimum Moment Check

ACI specifies that a wall section in bending shall have a minimum strength Mn that is greater than the cracking strength Mcr = Sgross * fr.

Maximum Reactions

This gives a summary of the maximum reactions (both out-of-plane and vertical) along with the load combination that creates them.

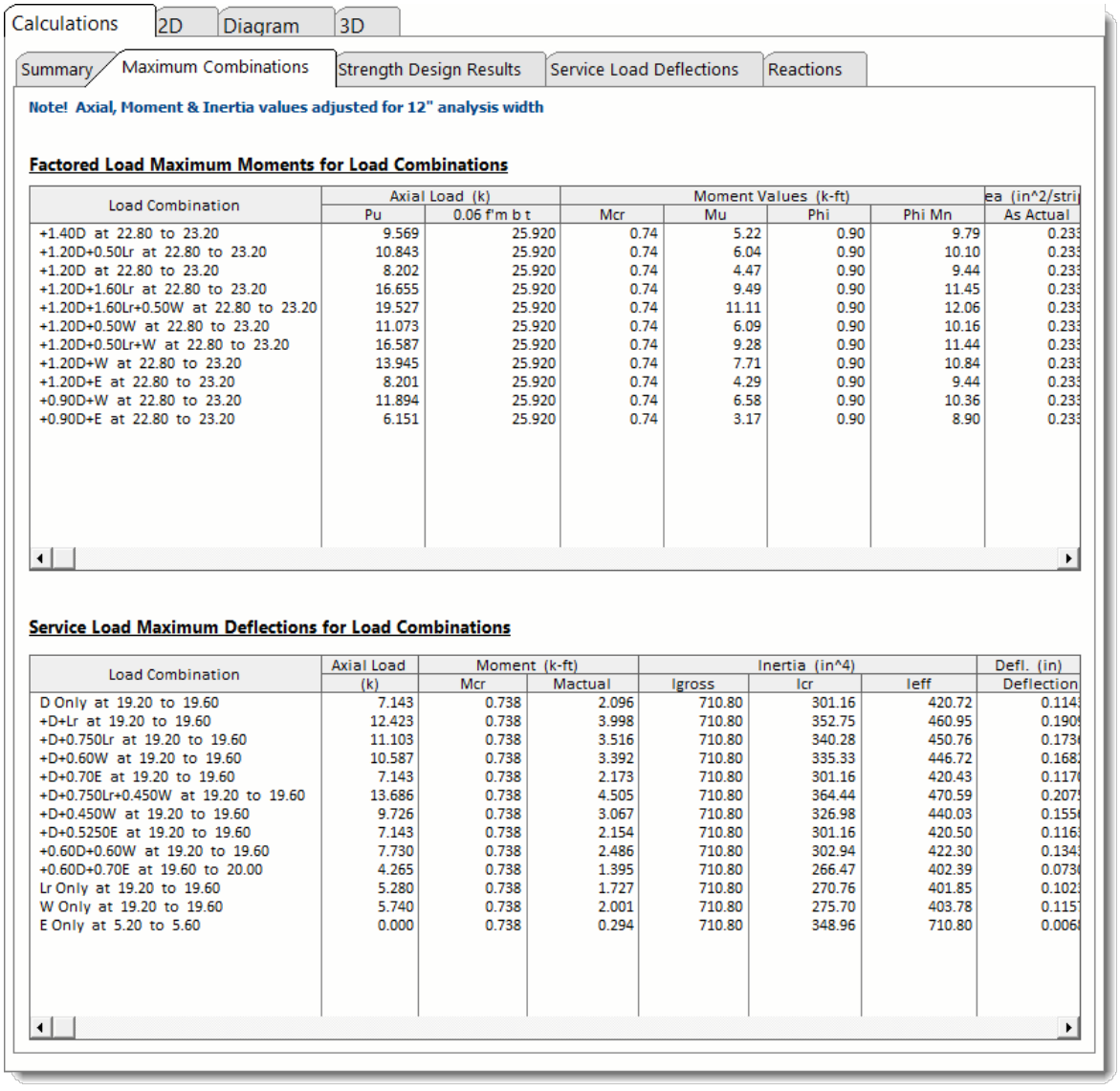
© 2026 ENERCALC, LLC

Maximum Combinations

This tab provides a summary of the governing values for each load combination for both factored load axial & bending and service load deflections.

Factored Load Maximum Moments for Load Combinations: The module looks through the result set for each load combination and identifies the location above the base of the wall at which the maximum condition is found. Note that "Aseff" is the effective area of steel and is influenced by the axial compression in that segment.

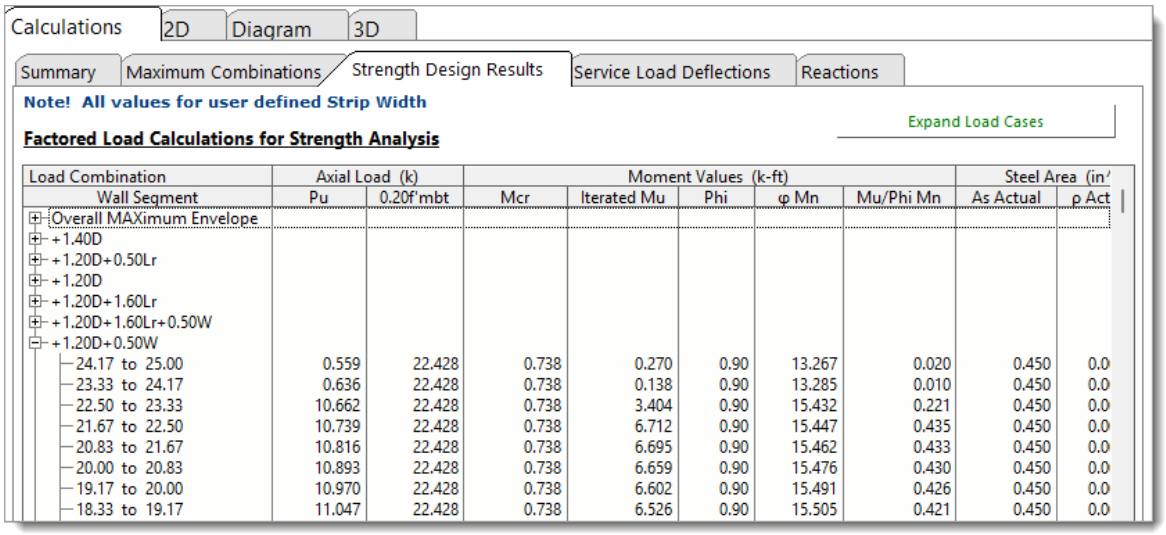
Service Load Maximum Deflections for Load Combinations: The module looks through the result set for each load combination and identifies the location above the base of the wall at which the maximum out-of-plane deflection is found. The value for "Ieff" is specific to the segment at that location and is based on the actual moment and Bischoff's equation for calculating effective moment of inertia.



© 2026 ENERCALC, LLC

Strength Design Results

This tab provides an extremely detailed summary of the factored axial load, moments, effective steel area and moment of inertia at each wall analysis segment for each load combination.



Load Combination / Wall Segment	Axial Load (k) Pu	0.20f'mbt	Moment Values (k-ft) Mcr	Iterated Mu	Phi	φ Mn	Mu/Phi Mn	Steel Area (in²) As Actual	ρ Act
Overall MAXimum Envelope									
+1.40D									
+1.20D+0.50Lr									
+1.20D									
+1.20D+1.60Lr									
+1.20D+1.60Lr+0.50W									
+1.20D+0.50W									
24.17 to 25.00	0.559	22.428	0.738	0.270	0.90	13.267	0.020	0.450	0.0
23.33 to 24.17	0.636	22.428	0.738	0.138	0.90	13.285	0.010	0.450	0.0
22.50 to 23.33	10.662	22.428	0.738	3.404	0.90	15.432	0.221	0.450	0.0
21.67 to 22.50	10.739	22.428	0.738	6.712	0.90	15.447	0.435	0.450	0.0
20.83 to 21.67	10.816	22.428	0.738	6.695	0.90	15.462	0.433	0.450	0.0
20.00 to 20.83	10.893	22.428	0.738	6.659	0.90	15.476	0.430	0.450	0.0
19.17 to 20.00	10.970	22.428	0.738	6.602	0.90	15.491	0.426	0.450	0.0
18.33 to 19.17	11.047	22.428	0.738	6.526	0.90	15.505	0.421	0.450	0.0

ρ Max Allow: This indicates the max allowable permissible flexural reinforcement ratio to ensure the net tensile strain (ε_t) limits specified by the TMS design standard are satisfied:

TMS 402-16: $\varepsilon_t = 1.5 * f_y / E_s$ *[§ 9.3.3.2.1(a)]*

TMS 402-22: $\varepsilon_t = 0.003 + f_y / E_s$ *[§ 9.3.3.2.4 and Table 9.1.4]* *Note: At this time, the Masonry Slender Wall module only supports tension-controlled masonry walls (φ = 0.9).*

Using the equations from TMS 402-16 Commentary Section 9.3.3.2, the maximum flexural reinforcement ratio for a fully grouted masonry wall can be calculated as follows:

$$\rho_{max} = \frac{A_{s,max}}{bd} = \frac{0.64 f'_m \left(\frac{\varepsilon_{mu}}{\varepsilon_{mu} + \varepsilon_t} \right) - \frac{P}{bd}}{f_y}$$

Where:

ε_{mu} = 0.0025; maximum usable compressive strain of masonry

ε_t = net tensile strain as described above

P = Axial load at the wall section considered

b = width of the design strip

d = depth to rebar as reported on the General tab

Similarly, for a partially grouted masonry wall, the maximum flexural reinforcement ratio can be calculated using:

© 2026 ENERCALC, LLC

$$\rho_{max} = \frac{A_{s,max}}{bd} = \frac{0.64f'_m\left(\frac{\varepsilon_{mu}}{\varepsilon_{mu}+\varepsilon_t}\right)\left(\frac{b_w}{b}\right) + 0.80f'_m\, t_{fs}\left(\frac{b-b_w}{bd}\right) - \frac{P}{bd}}{f_y}$$

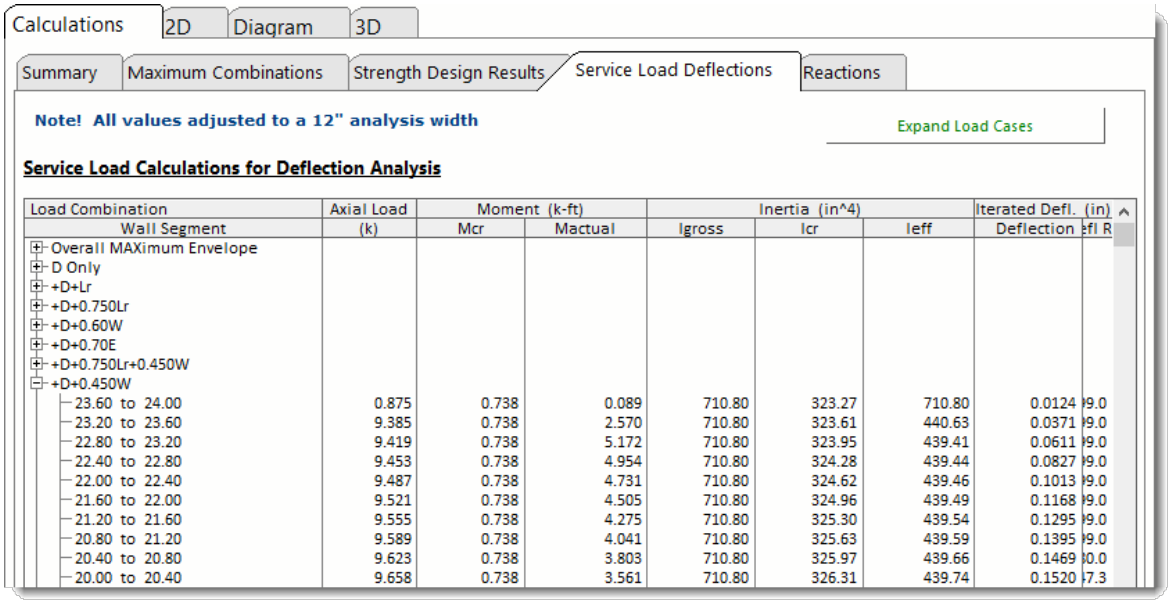
Where:

b_w = width of the design strip minus the sum of the length of ungrouted cells. See Appendix C - Masonry Wall Weights & Section Properties 1242.

t_{fs} = face-shell thickness for hollow masonry units. See Appendix C - Masonry Wall Weights & Section Properties 1242.

Service Load Deflections

This tab provides an extremely detailed summary of the service axial load, moments, effective moment of inertia and calculated deflection at each wall analysis segment for each load combination.

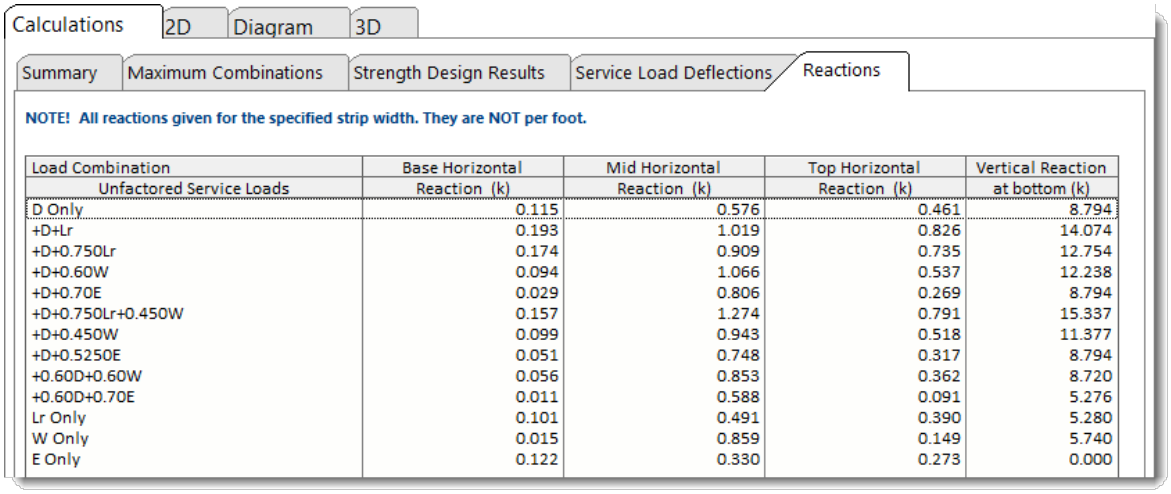


Load Combination / Wall Segment	Axial Load (k)	Moment (k-ft) Mcr	Moment (k-ft) Mactual	Inertia (in^4) Igross	Inertia (in^4) Icr	Inertia (in^4) Ieff	Iterated Defl. (in) Deflection	Iterated Defl. (in) Defl R
Overall MAXimum Envelope								
D Only								
+D+Lr								
+D+0.750Lr								
+D+0.60W								
+D+0.70E								
+D+0.750Lr+0.450W								
+D+0.450W								
23.60 to 24.00	0.875	0.738	0.089	710.80	323.27	710.80	0.0124	9.0
23.20 to 23.60	9.385	0.738	2.570	710.80	323.61	440.63	0.0371	9.0
22.80 to 23.20	9.419	0.738	5.172	710.80	323.95	439.41	0.0611	9.0
22.40 to 22.80	9.453	0.738	4.954	710.80	324.28	439.44	0.0827	9.0
22.00 to 22.40	9.487	0.738	4.731	710.80	324.62	439.46	0.1013	9.0
21.60 to 22.00	9.521	0.738	4.505	710.80	324.96	439.49	0.1168	9.0
21.20 to 21.60	9.555	0.738	4.275	710.80	325.30	439.54	0.1295	9.0
20.80 to 21.20	9.589	0.738	4.041	710.80	325.63	439.59	0.1395	9.0
20.40 to 20.80	9.623	0.738	3.803	710.80	325.97	439.66	0.1469	0.0
20.00 to 20.40	9.658	0.738	3.561	710.80	326.31	439.74	0.1520	7.3

© 2026 ENERCALC, LLC

Reactions

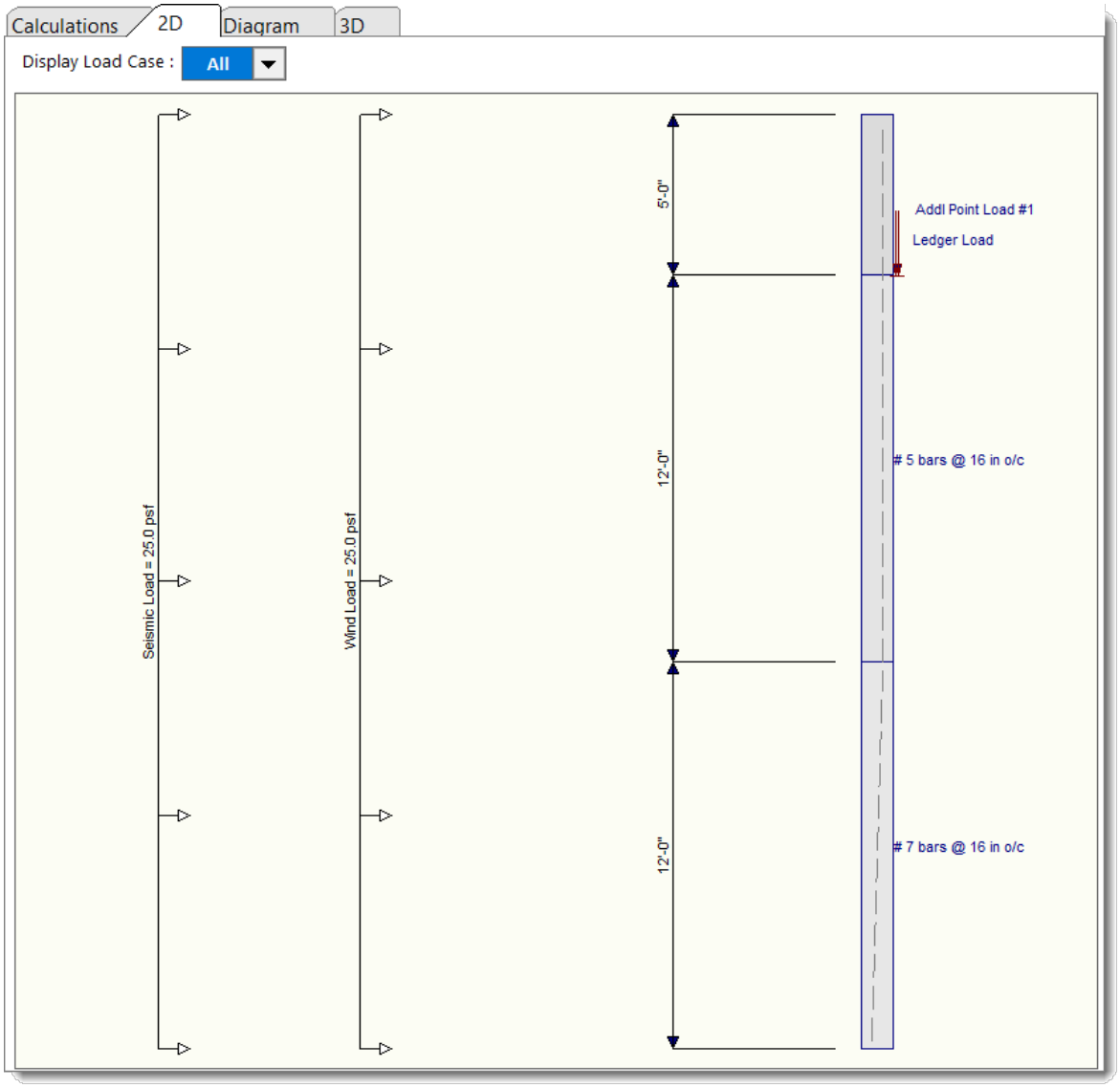
This tab gives a summary of out-of-plane and vertical base reactions for each service load combination.



Load Combination Unfactored Service Loads	Base Horizontal Reaction (k)	Mid Horizontal Reaction (k)	Top Horizontal Reaction (k)	Vertical Reaction at bottom (k)
D Only	0.115	0.576	0.461	8.794
+D+Lr	0.193	1.019	0.826	14.074
+D+0.750Lr	0.174	0.909	0.735	12.754
+D+0.60W	0.094	1.066	0.537	12.238
+D+0.70E	0.029	0.806	0.269	8.794
+D+0.750Lr+0.450W	0.157	1.274	0.791	15.337
+D+0.450W	0.099	0.943	0.518	11.377
+D+0.5250E	0.051	0.748	0.317	8.794
+0.60D+0.60W	0.056	0.853	0.362	8.720
+0.60D+0.70E	0.011	0.588	0.091	5.276
Lr Only	0.101	0.491	0.390	5.280
W Only	0.015	0.859	0.149	5.740
E Only	0.122	0.330	0.273	0.000

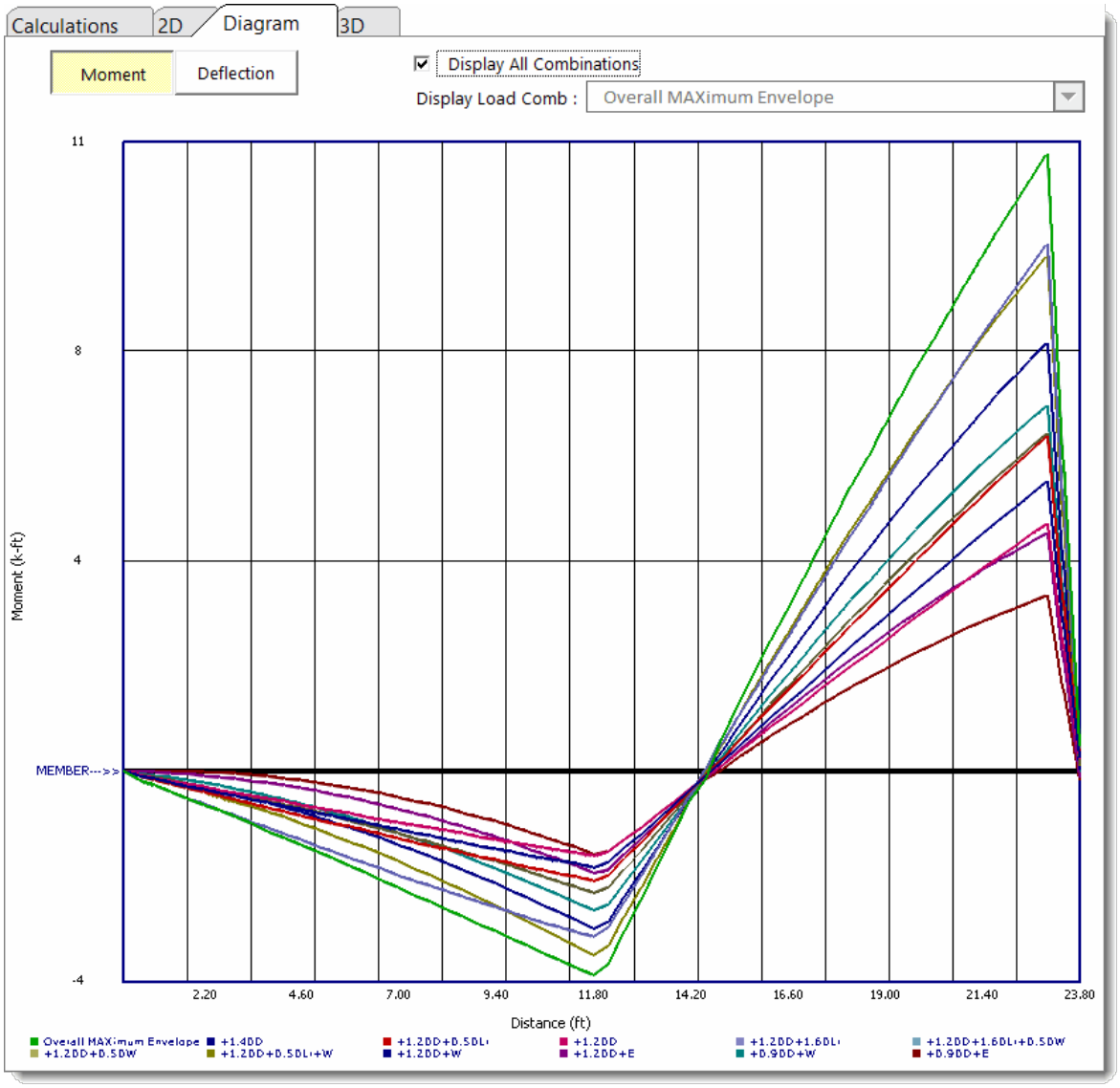
© 2026 ENERCALC, LLC

2D Sketch



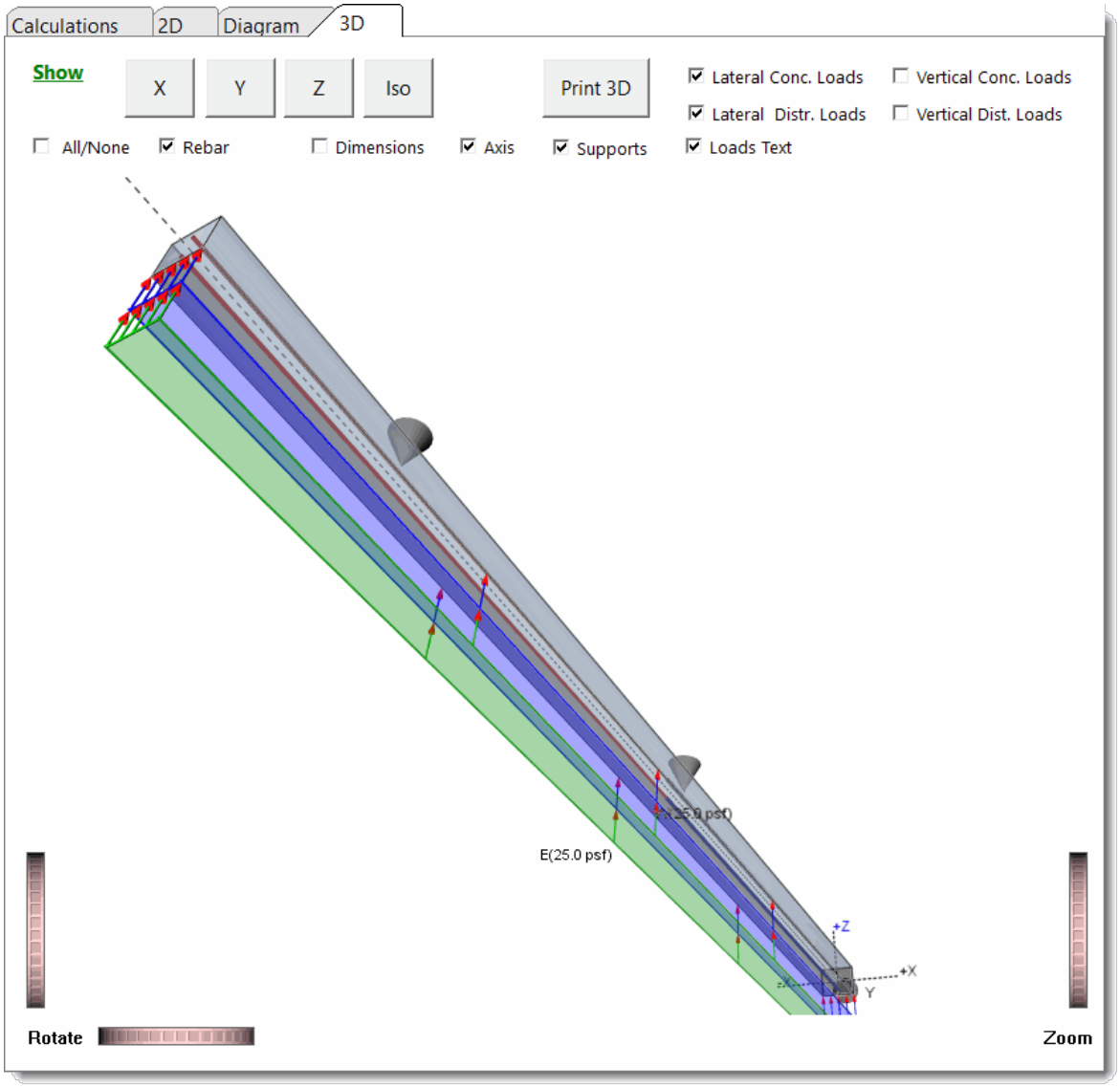
© 2026 ENERCALC, LLC

Diagram



© 2026 ENERCALC, LLC

3D Rendering



13.7.2 Shear Walls

Please select a subtopic.

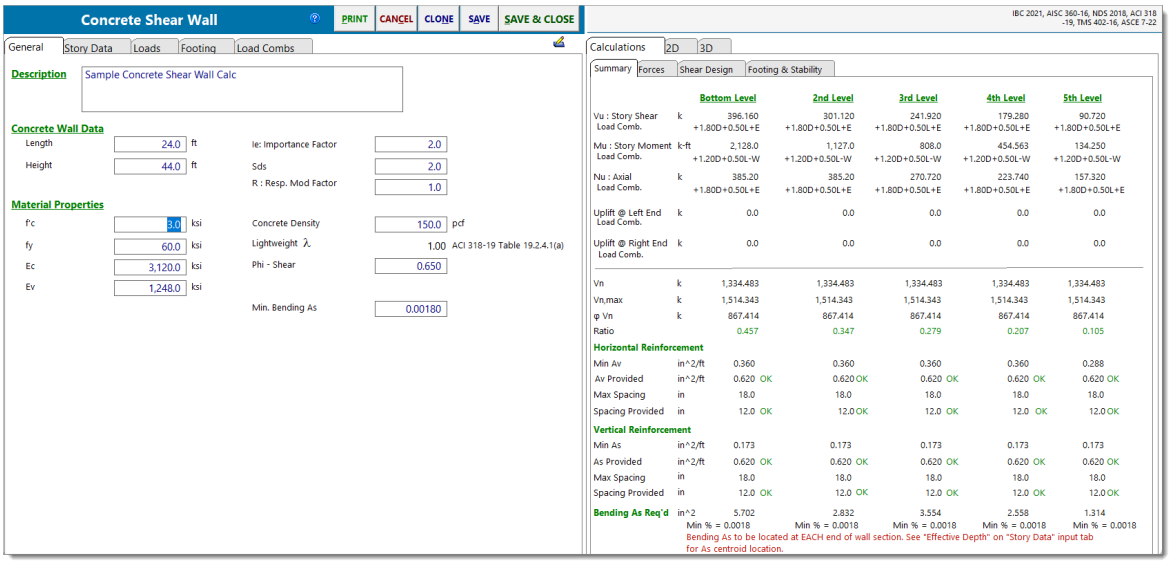
© 2026 ENERCALC, LLC

13.7.2.1 Concrete Shear Wall

This module allows the design of concrete shear walls per ACI 318 Chapter 11 including multi-story walls with no openings but with up to five levels of differing length, height and thickness.

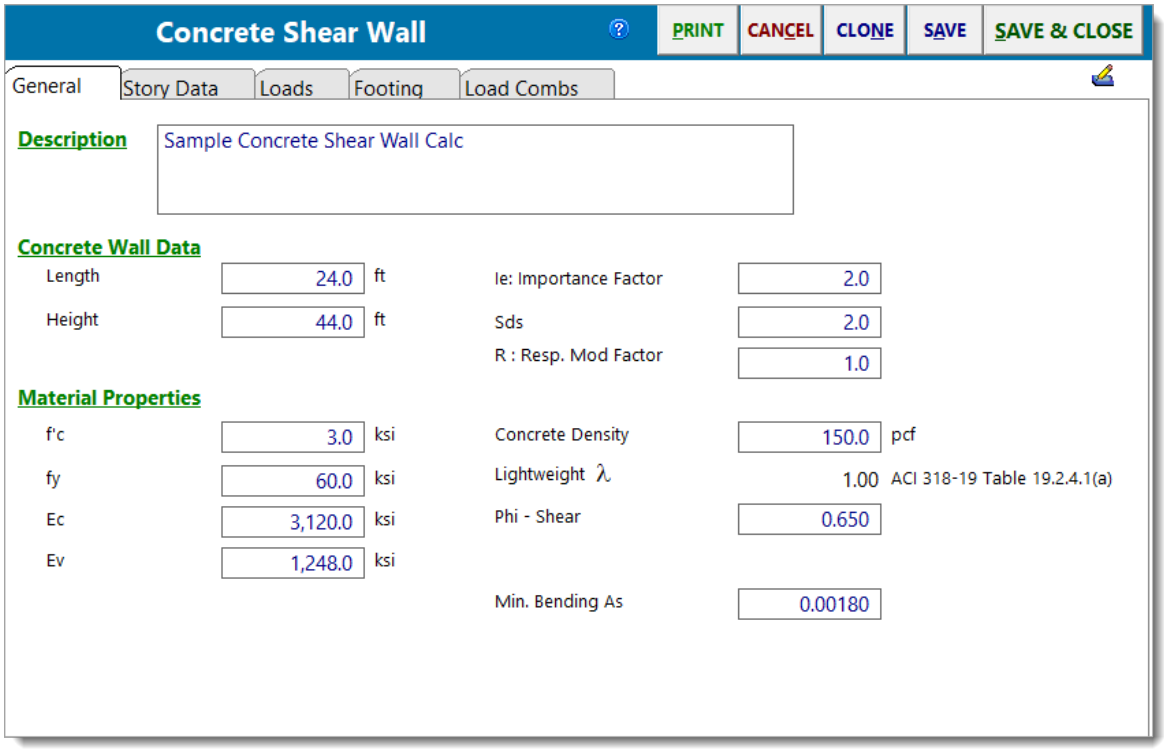
NOTE: As of February 2025, the Concrete Shear Module supports defining horizontal and vertical reinforcement for walls designed per ACI 318-19 and later. The module will check shear strength as well as prescriptive ACI reinforcement limits and detailing requirements based on the defined layout. When opening a calculation created before this update, a prompt will appear to review the Story Data tab and define the reinforcement layout.

Additionally, based on discussions with industry experts about the interpretation of the 'hw' and 'Lw' values when calculating 'αc', this module no longer allows varying wall lengths on a story-by-story basis. Wall lengths and offsets are now standardized across all stories. The shear wall aspect ratio Hw/Lw is now calculated using the global dimensions of the entire wall stack. When opening older Concrete Shear Wall calculations, a warning message will highlight these changes. If needed, legacy calculation data remains accessible in previous builds.



© 2026 ENERCALC, LLC

General



Length specifies the total length of the wall.

Height specifies the total height of the wall. On the next tab you can divide that total height into up to five different wall portions.

f'c is the concrete strength.

fy is the yield stress of rebar.

Ec is the bending modulus of elasticity.

Ev is the shear modulus of elasticity.

Ie, Sds, and **R** are used to calculate the in-plane portion of the wall weight to be applied as a seismic load.

Concrete Density is the unit weight of concrete in pounds per cubic foot (pcf).

Lightweight λ is for lightweight concrete and as calculated based on Concrete Density per ACI 318-19 Table 19.2.4.1(a).

Phi - Shear is the strength reduction factor used for shear calculations. Refer to ACI 318-19 Table 21.2.1 and section 21.2.4 for modifications pertaining to seismic force resisting elements.

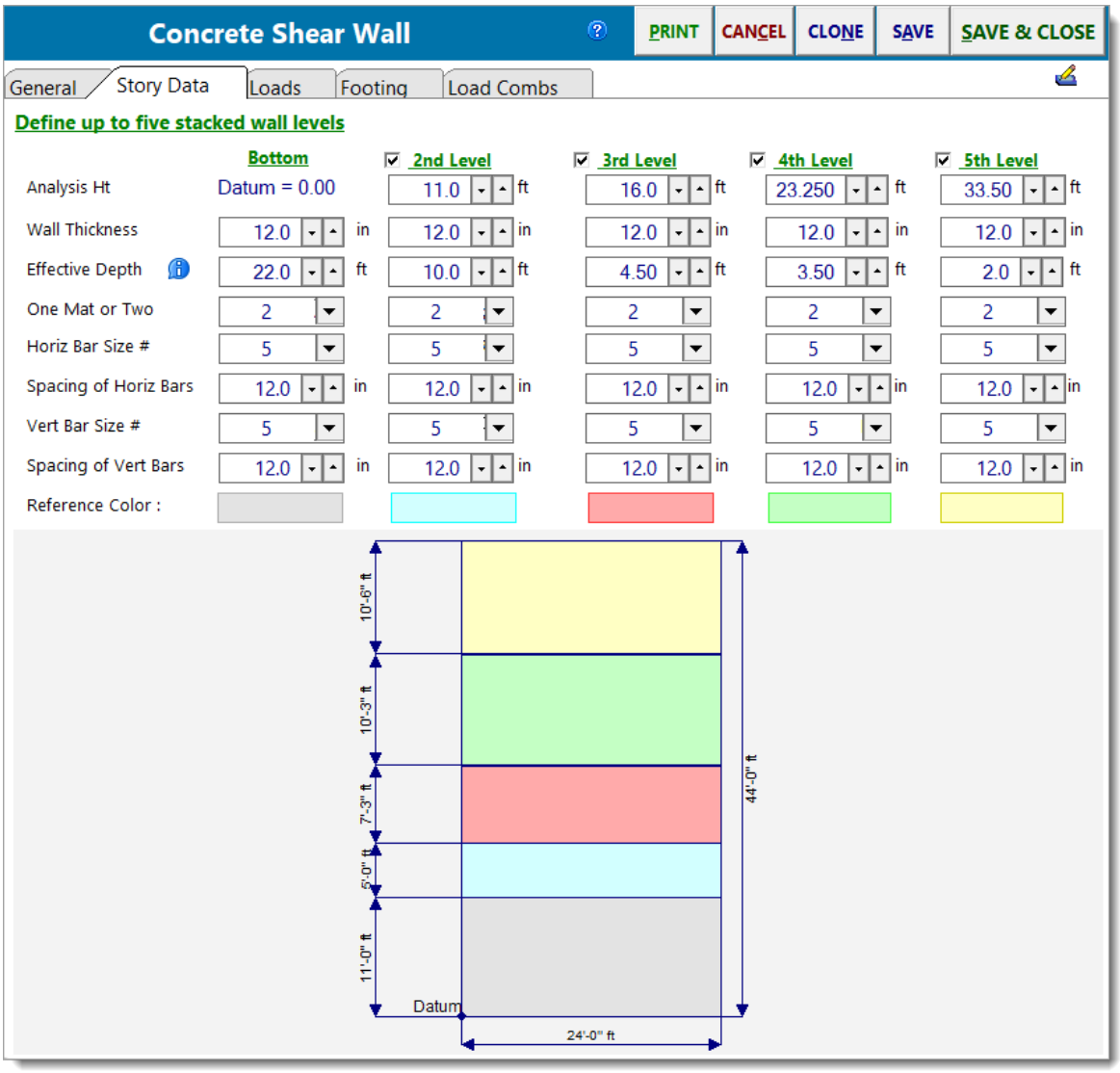
© 2026 ENERCALC, LLC

Min. Bending As is the minimum required flexural reinforcement ratio.

Default Rebar Sizes & Spacing *(ACI 318-14 only)*: specify size, spacing, and cover for horizontal and vertical rebar. These inputs are used for drawing the graphics to scale, as well as for determining limiting reinforcing percentages.

Story Data

This tab is where you specify the distinct wall levels for the wall.



Analysis Height locates the bottom edge of the wall section and is where the maximum shear and bending stress will be calculated. This is the user-defined height at which the analysis of a particular wall section will be performed. All moments, shears, and vertical loads at this height will be calculated using all applied lateral and

© 2026 ENERCALC, LLC

vertical loads and the wall self weight above this point. The other wall data items specified in the same column will be used between this analysis height and the next higher level indicated in the column to the right.

ALWAYS DEFINE ANALYSIS HEIGHTS IN INCREASING ORDER FROM LEFT TO RIGHT. This is needed due to the manner in which the module calculates the heights by comparing heights of adjacent sections.

Wall thickness is the thickness of this wall section. Enter the thickness to be used in the analysis of a particular wall section. This thickness will be used only between the Analysis Height for that section up to the analysis of the next higher section (or Total Wall Height if it is the highest section).

Effective Depth locates the tension reinforcement within the panel. For designs following ACI 318-14, it is also used to calculate the shear depth, which is required for determining actual shear stresses. Similar to beams, the Effective Depth is measured from the wall's compression edge to the centroid of the tension chord reinforcement.

Reference Color provides a visual reference color to each wall section for easy identification in the graphical sketch.

The following controls are only available for ACI 318-19 and up:

One Mat or Two specifies whether the reinforcement consists of a single layer or two layers of both horizontal and vertical reinforcement.

Horiz Bar Size # defines the size of the horizontal reinforcement bars in the shear wall, using standard rebar designations (e.g., #5, #6, etc.).

Spacing of Horiz Bars defines the center-to-center spacing of the horizontal reinforcement bars in inches.

Vert Bar Size # defines the size of the vertical reinforcement bars in the shear wall, using standard rebar designations (e.g., #5, #6, etc.).
Note: The vertical reinforcement specified here is only used for the prescriptive reinforcement limits and detailing checks per ACI 318 Chapter 11. It is not utilized for flexural reinforcement. The required flexural reinforcement is reported separately on the Summary tab as "Bending As Req'd".

Spacing of Vert Bars defines the center-to-center spacing of the vertical reinforcement bars in inches.

© 2026 ENERCALC, LLC

Loads

This main tab has four sub-tabs that allow you to enter four types of loads.

Vertical loads can be of dead, live, roof live and snow types.

Lateral loads can be of seismic and wind types.

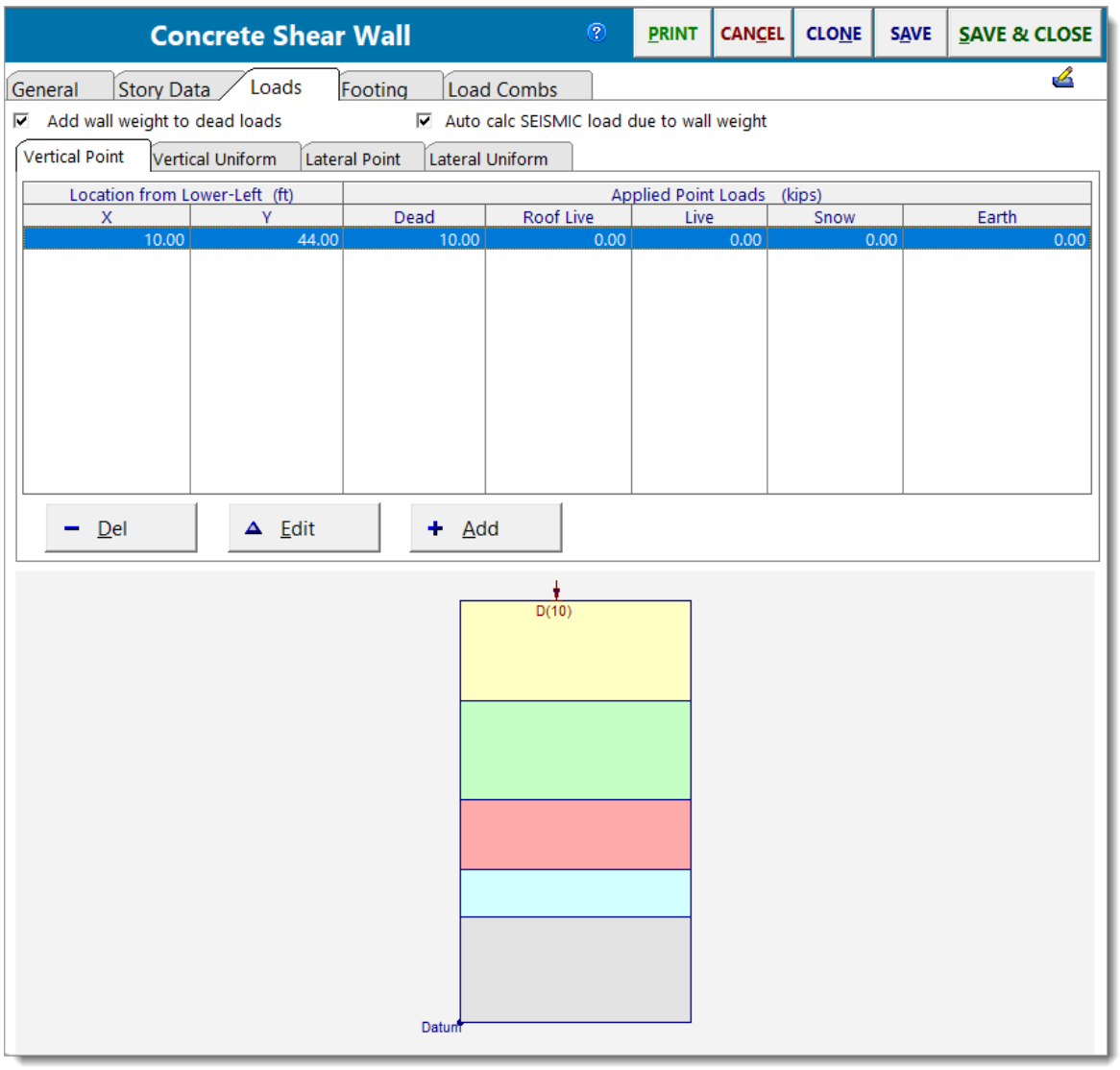
Add wall weight to dead loads will tell the module to calculate the weight of the wall above each analysis height and include it in the vertical dead loads to calculate applied axial stress. It also is used for footing design when that option is selected.

Add wall weight as SEISMIC load will calculate the wall self weight, apply the lateral weight seismic factor and "E" load combination factor. The resulting load will be applied at the wall center to calculate shear and overturning due to that portion of the wall. This is used for values at the analysis height, for the effect of that level's seismic weight on the levels below, and for footing overturning and sliding calculations.

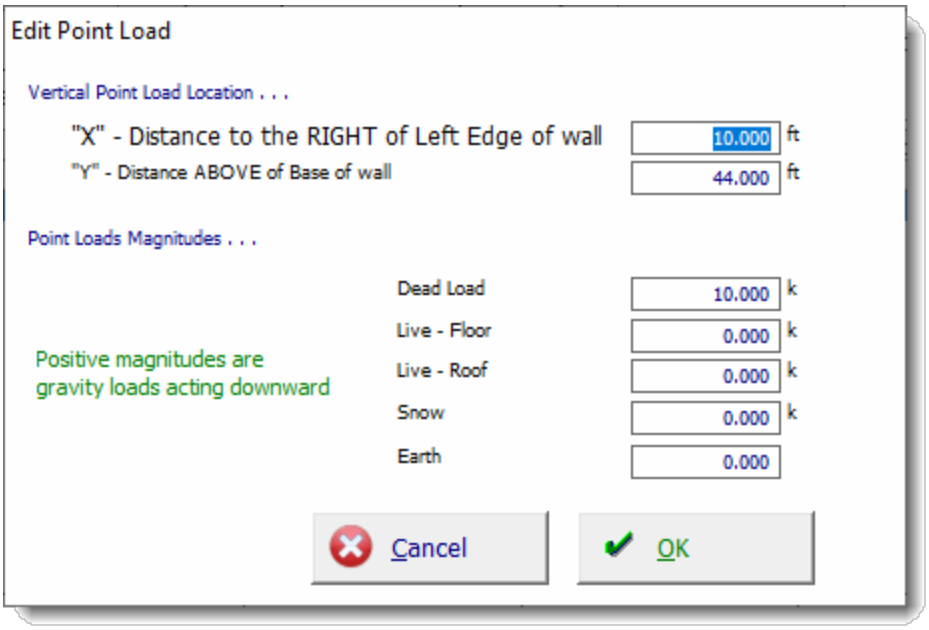
© 2026 ENERCALC, LLC

Vertical Point Loads

Use this tab to apply point loads to the wall. You can specify an "X" and "Y" distance from the lower-left corner of the lower wall so that the load can be located anywhere on the defined walls.

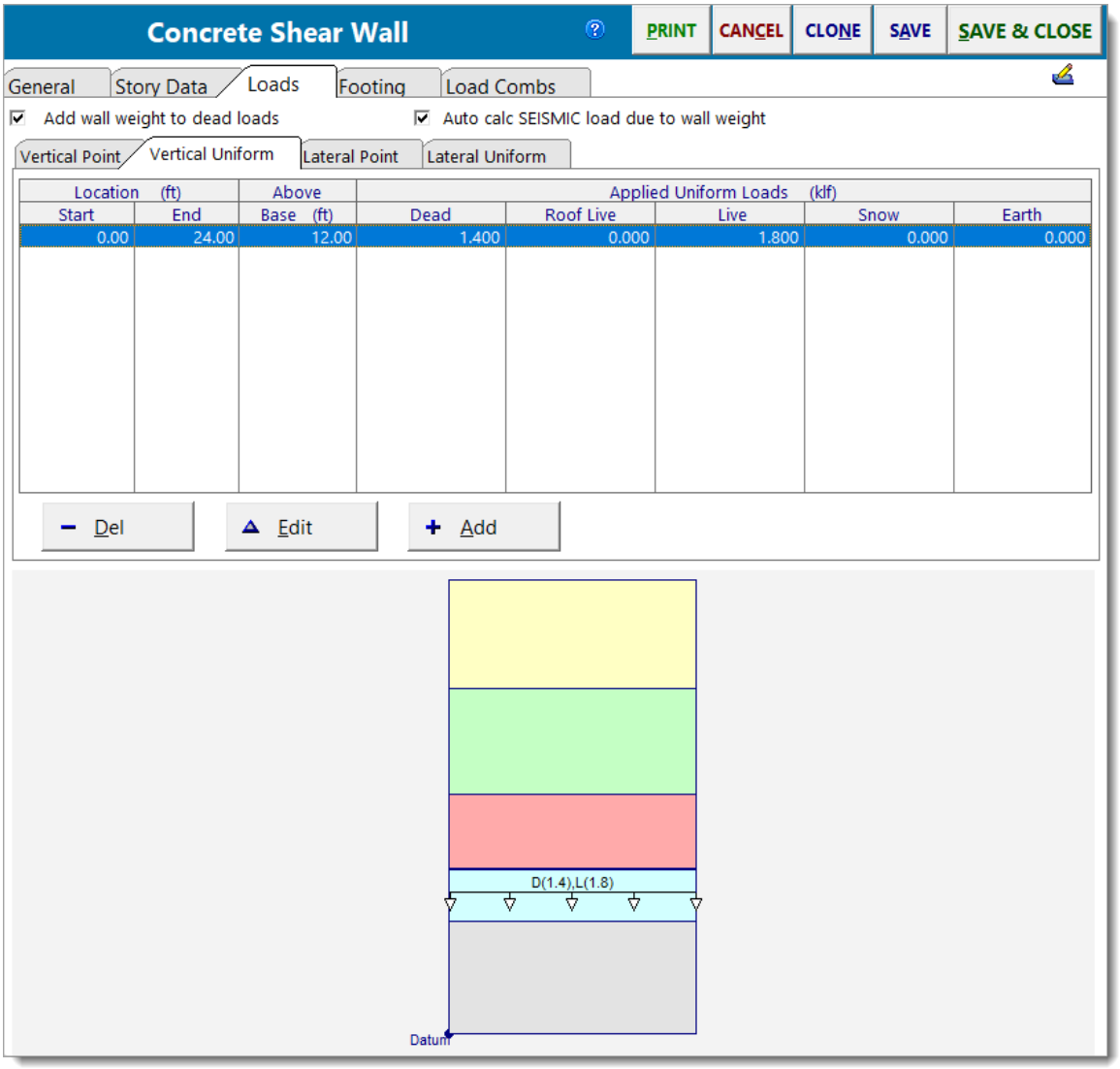


Use the [**Add**] and [**Edit**] buttons to change the values of applied loads. Clicking either button displays this window:

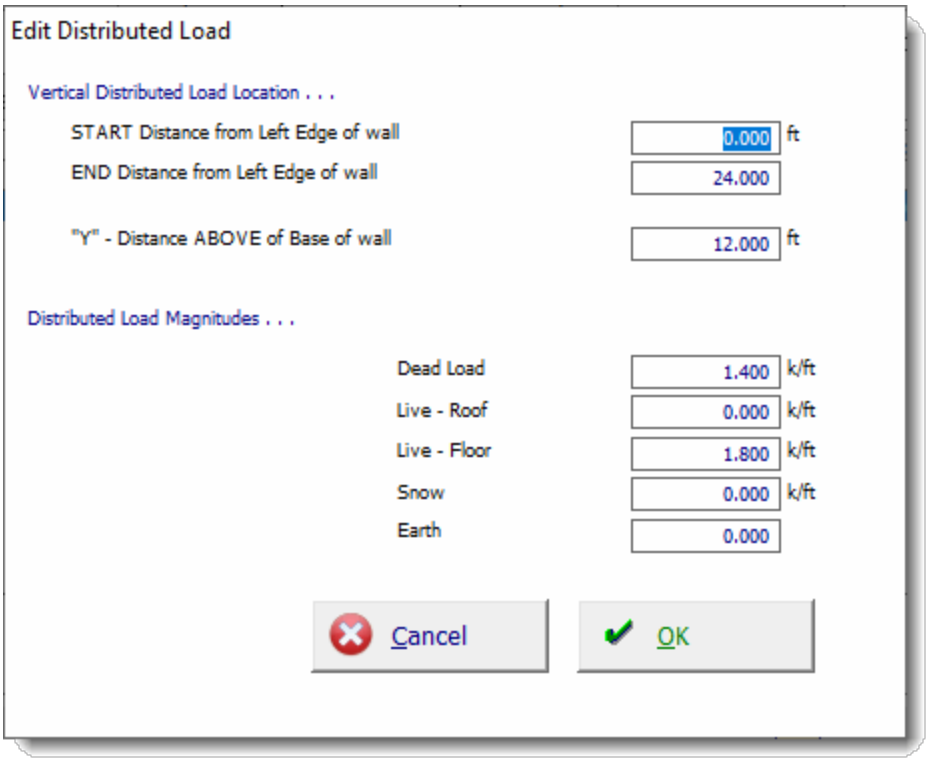


Vertical Uniform Loads

Use this tab to apply uniform loads to the wall. You can specify a "Y" distance from the bottom of the lower wall so that the load can be located at any height.



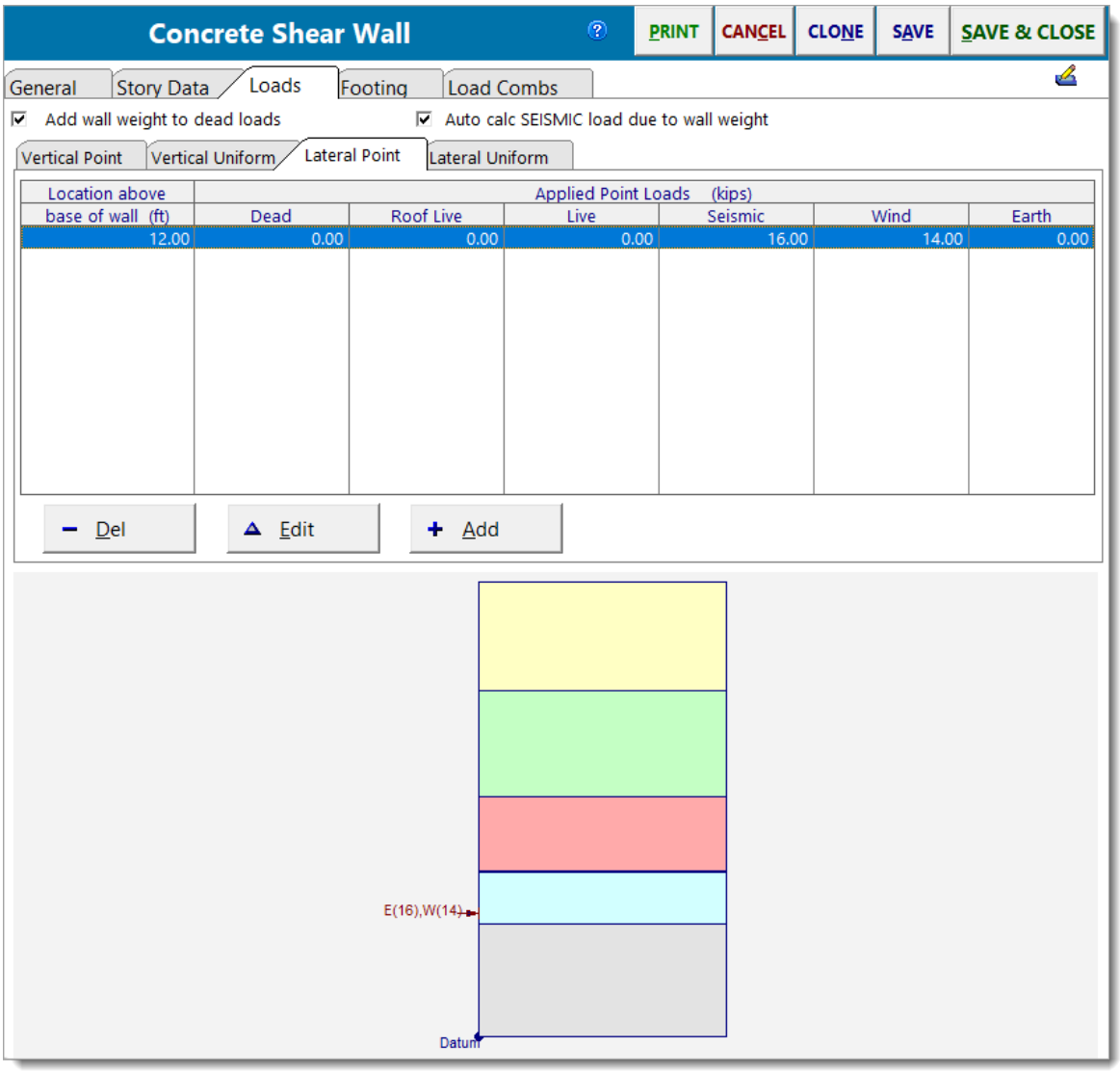
Use the [**Add**] and [**Edit**] buttons to change the values of applied loads. Clicking either button displays this window:



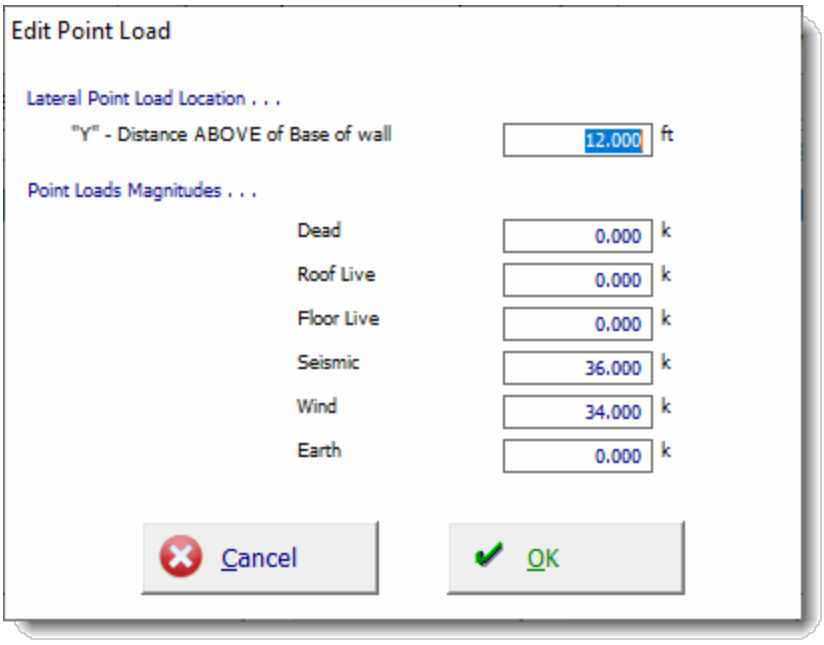
© 2026 ENERCALC, LLC

Lateral Point Loads

Use this tab to apply point lateral loads to the wall. You can specify a "Y" distance from the bottom of the lower wall so that the load can be located at any height.



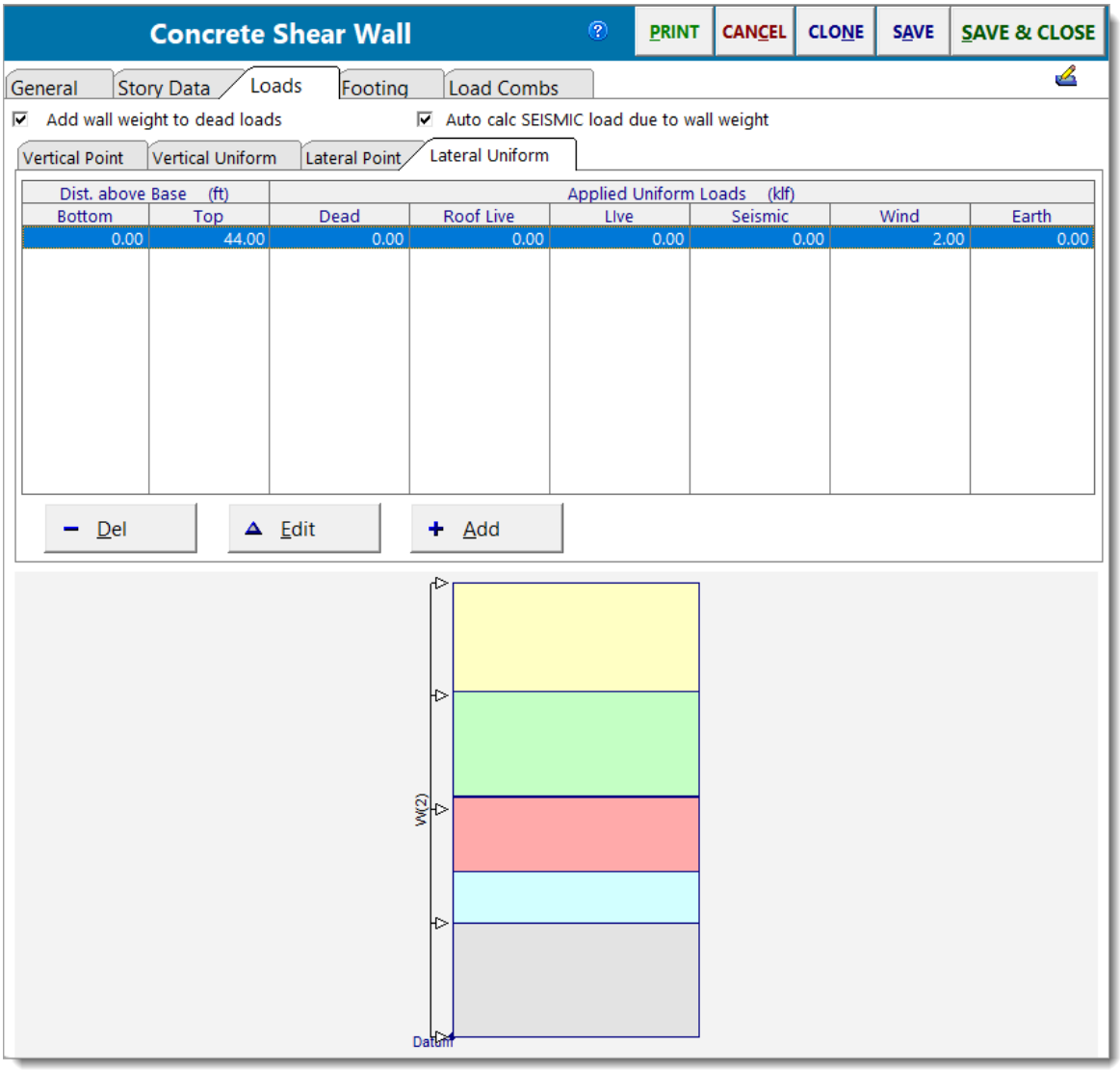
Use the [**Add**] and [**Edit**] buttons to change the values of applied loads. Clicking either button displays this window:



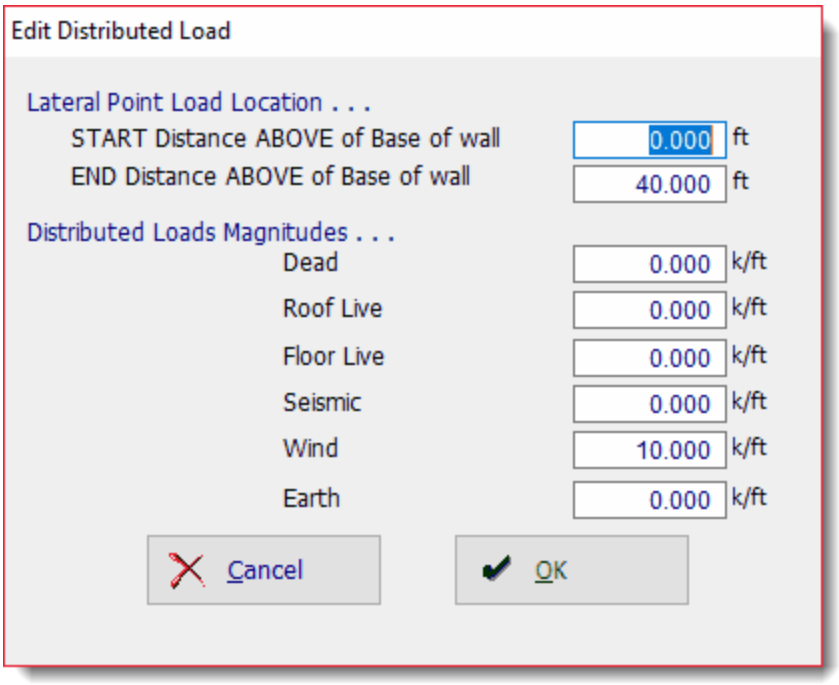
© 2026 ENERCALC, LLC

Lateral Uniform Loads

Use this tab to apply uniform lateral loads to the wall. You can specify a Start and End location to define the extent of the lateral load.



Use the [**Add**] and [**Edit**] buttons to change the values of applied loads. Clicking either button displays this window:

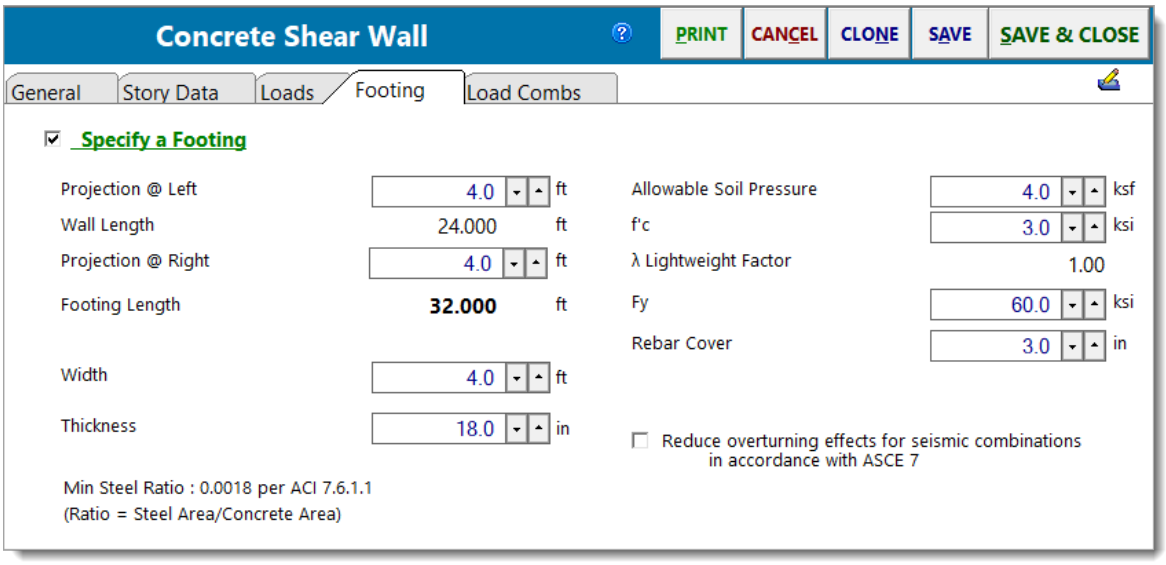


© 2026 ENERCALC, LLC

Footing

Optional tab to design a footing that behaves as a continuous grade beam that cantilevers past the end of the wall.

Specify geometry, allowable soil pressure, and material properties to arrive at checks for soil pressure and footing flexure.



Specify a Footing: If this checkbox is selected, then the input fields for defining a footing are displayed. Note: This checkbox also causes some footing-related input to be shown or hidden as appropriate on the Loads tab.

Projection @ Left: Enter the projection of the footing beyond the left end of the wall in units of ft.

Wall Length: The program reports the shear wall length in units of ft for reference.

Projection @ Right: Enter the projection of the footing beyond the right end of the wall in units of ft.

Footing Length: The program reports the footing length in units of ft for reference.

Width: Specify the width of the footing in units of ft. This is the dimension perpendicular to the wall length.

Thickness: Enter the thickness of the footing in units of inches.

Min Steel Reinforcing: The minimum flexural reinforcement ratio, per ACI 7.6.1.1, is 0.0018 times the gross area of the concrete (Width x Thickness). Commentary R7.6.1.1 also notes that minimum flexural reinforcement should be placed as close as possible to

© 2026 ENERCALC, LLC

the tension face of the concrete, unlike shrinkage and temperature reinforcement which may be distributed between the top and bottom faces of the footing.

Allowable Soil Pressure: Specify the allowable soil bearing pressure in units of ksf.

f'c: Specify the compressive stress of concrete in units of ksi.

Fy: Specify the yield stress of rebar in units of ksi.

Rebar Cover: Specify the cover over rebar in units of inches. The program will use this value and a 1/2" allowance for the rebar size to calculate the effective depth of the concrete section.

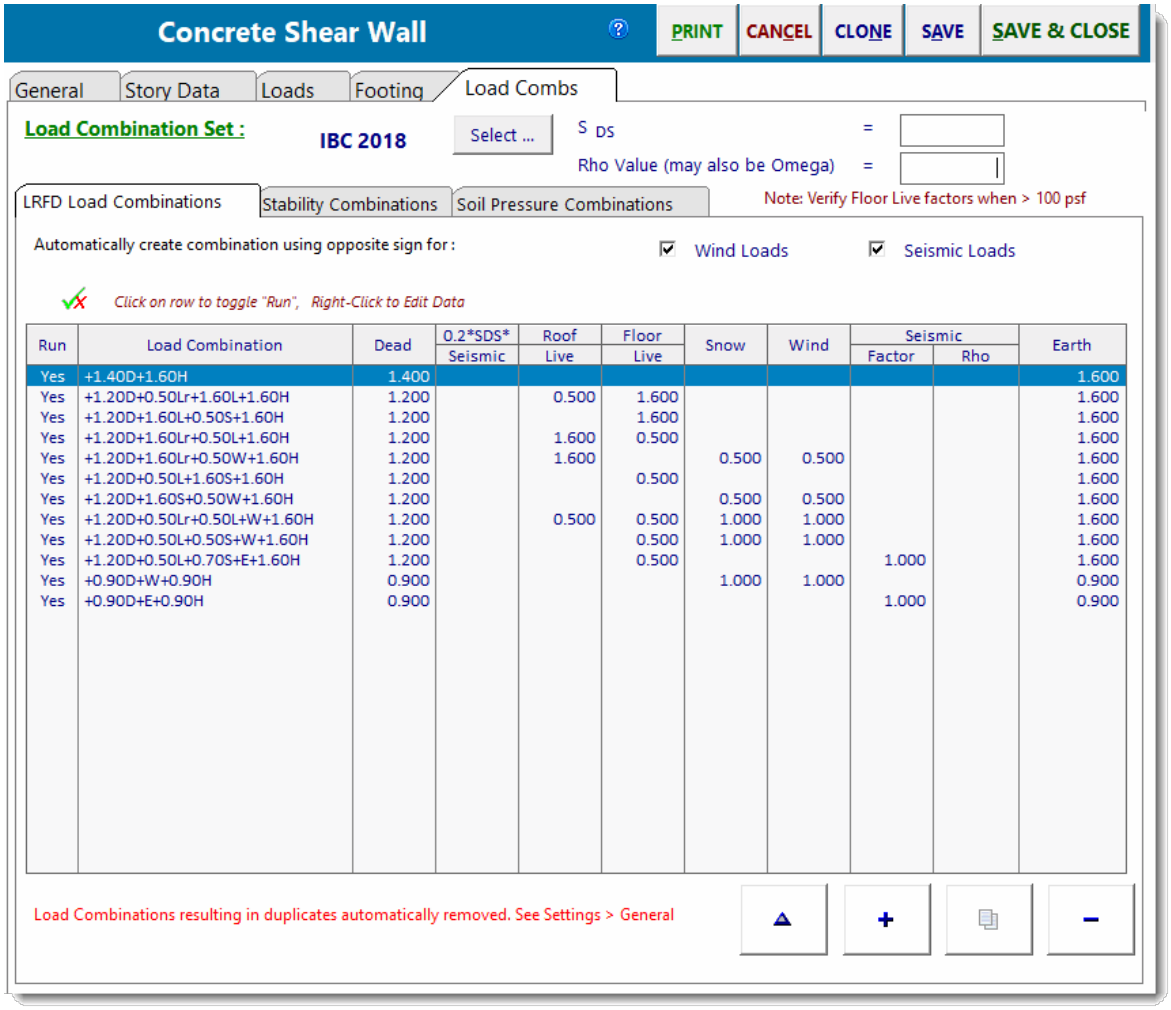
> Note: Rebar is assumed to exist only at the bottom of the footing to resist tensile forces from the vertical loads and increased pressure due to overturning forces. Tension in the top of the footing in cases where no upward soil pressure exists and the footing weight creates a downward net force IS IGNORED.

Reduce overturning effects for seismic combinations per ASCE 7-16 Section 12.13.4: Applies the referenced reduction in overturning force for seismic load combinations when selected.

© 2026 ENERCALC, LLC

Load Combinations

The typical load combination tab for strength design of concrete is provided.



Run	Load Combination	Dead	0.2*SDS* Seismic	Roof Live	Floor Live	Snow	Wind	Seismic Factor	Seismic Rho	Earth
Yes	+1.40D+1.60H	1.400								1.600
Yes	+1.20D+0.50Lr+1.60L+1.60H	1.200		0.500	1.600					1.600
Yes	+1.20D+1.60L+0.50S+1.60H	1.200			1.600					1.600
Yes	+1.20D+1.60Lr+0.50L+1.60H	1.200		1.600	0.500					1.600
Yes	+1.20D+1.60Lr+0.50W+1.60H	1.200		1.600		0.500	0.500			1.600
Yes	+1.20D+0.50L+1.60S+1.60H	1.200			0.500					1.600
Yes	+1.20D+1.60S+0.50W+1.60H	1.200				0.500	0.500			1.600
Yes	+1.20D+0.50Lr+0.50L+W+1.60H	1.200		0.500	0.500	1.000	1.000			1.600
Yes	+1.20D+0.50L+0.50S+W+1.60H	1.200			0.500	1.000	1.000			1.600
Yes	+1.20D+0.50L+0.70S+E+1.60H	1.200			0.500			1.000		1.600
Yes	+0.90D+W+0.90H	0.900				1.000	1.000			0.900
Yes	+0.90D+E+0.90H	0.900						1.000		0.900

Summary

Provides a summary of each level.

In the top portion of the window, you will see the following:

Specify a Footing: If this checkbox is selected, then the input fields for defining a footing are displayed. Note: This checkbox also causes some footing-related input to be shown or hidden as appropriate on the Loads tab.

Projection @ Left: Enter the projection of the footing beyond the left end of the wall in units of ft.

© 2026 ENERCALC, LLC

Vu: Story Shear: The governing shear force and associated load combination at each level.

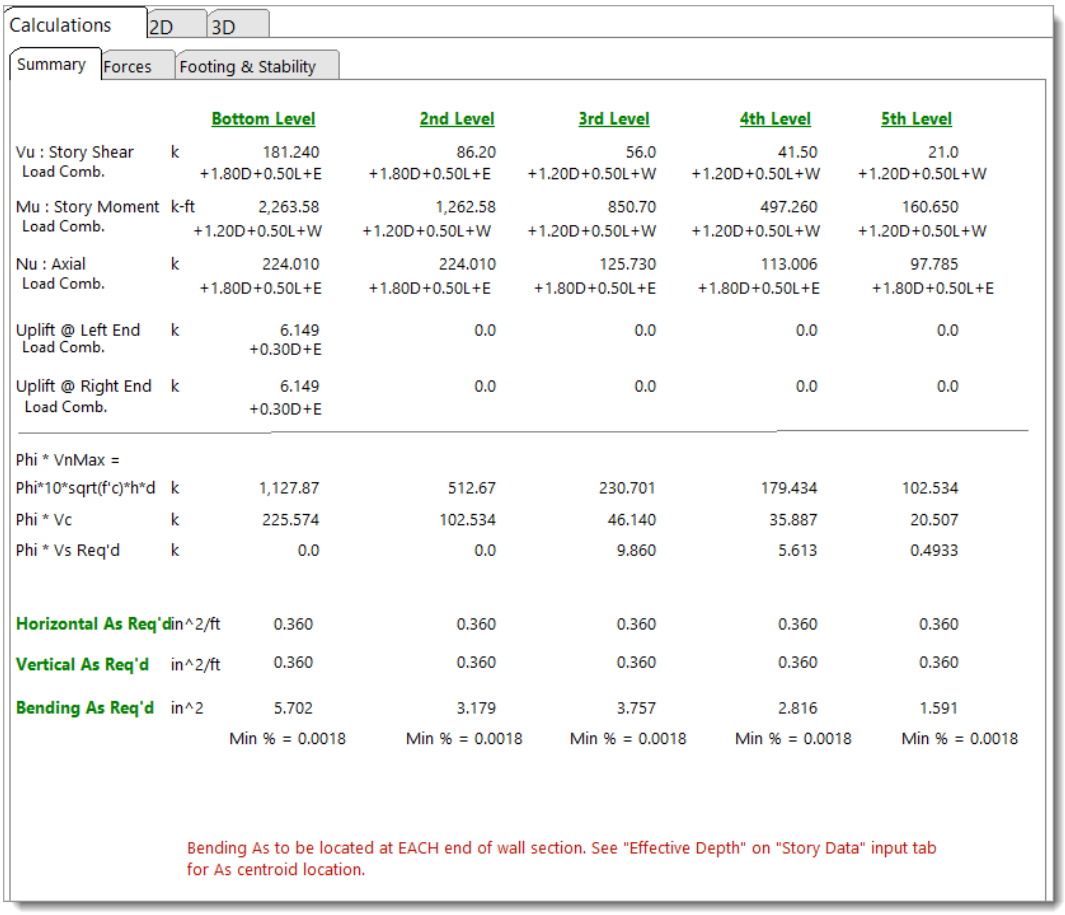
Mu: Story Moment: The governing flexural moment and associated load combination at each level.

Nu: Axial: The governing axial demand and associated load combination at each level.

Uplift @ Left End / Right End: The net uplift force at the left and right ends of the wall due to overturning moments from lateral loads and vertical loads. The centroid location for the net uplift is determined by the "Effective Depth" specified in the "Story Data" tab.

In the bottom portion of the screen, the content displayed varies depending on the governing ACI edition.

ACI 318-14 only: Shear strength, horizontal steel, vertical steel, and end reinforcing for bending tension in that wall section are reported.



Calculations | 2D | 3D

Summary | Forces | Footing & Stability

		Bottom Level	2nd Level	3rd Level	4th Level	5th Level
Vu : Story Shear	k	181.240	86.20	56.0	41.50	21.0
Load Comb.		+1.80D+0.50L+E	+1.80D+0.50L+E	+1.20D+0.50L+W	+1.20D+0.50L+W	+1.20D+0.50L+W
Mu : Story Moment	k-ft	2,263.58	1,262.58	850.70	497.260	160.650
Load Comb.		+1.20D+0.50L+W	+1.20D+0.50L+W	+1.20D+0.50L+W	+1.20D+0.50L+W	+1.20D+0.50L+W
Nu : Axial	k	224.010	224.010	125.730	113.006	97.785
Load Comb.		+1.80D+0.50L+E	+1.80D+0.50L+E	+1.80D+0.50L+E	+1.80D+0.50L+E	+1.80D+0.50L+E
Uplift @ Left End	k	6.149	0.0	0.0	0.0	0.0
Load Comb.		+0.30D+E				
Uplift @ Right End	k	6.149	0.0	0.0	0.0	0.0
Load Comb.		+0.30D+E				
Phi * VnMax =						
Phi*10*sqrt(f'c)*h*d	k	1,127.87	512.67	230.701	179.434	102.534
Phi * Vc	k	225.574	102.534	46.140	35.887	20.507
Phi * Vs Req'd	k	0.0	0.0	9.860	5.613	0.4933
Horizontal As Req'd	in^2/ft	0.360	0.360	0.360	0.360	0.360
Vertical As Req'd	in^2/ft	0.360	0.360	0.360	0.360	0.360
Bending As Req'd	in^2	5.702	3.179	3.757	2.816	1.591
		Min % = 0.0018	Min % = 0.0018	Min % = 0.0018	Min % = 0.0018	Min % = 0.0018

Bending As to be located at EACH end of wall section. See "Effective Depth" on "Story Data" input tab for As centroid location.

Phi * 10 * sqrt(f'c) * h * d: The maximum nominal shear strength of the wall section per ACI 318-14 Section 11.5.4.3.

© 2026 ENERCALC, LLC

Phi * Vc: The shear contribution from concrete alone per ACI 318-14 Section 11.5.4.5 or Table 11.5.4.6 if the "Detailed" method is enabled on the General tab..

Phi * Vs Req'd: The shear contribution required from reinforcement to resist applied shear forces per ACI 318-14 Eqn. 11.5.4.4 and Eqn. 11.5.4.8.

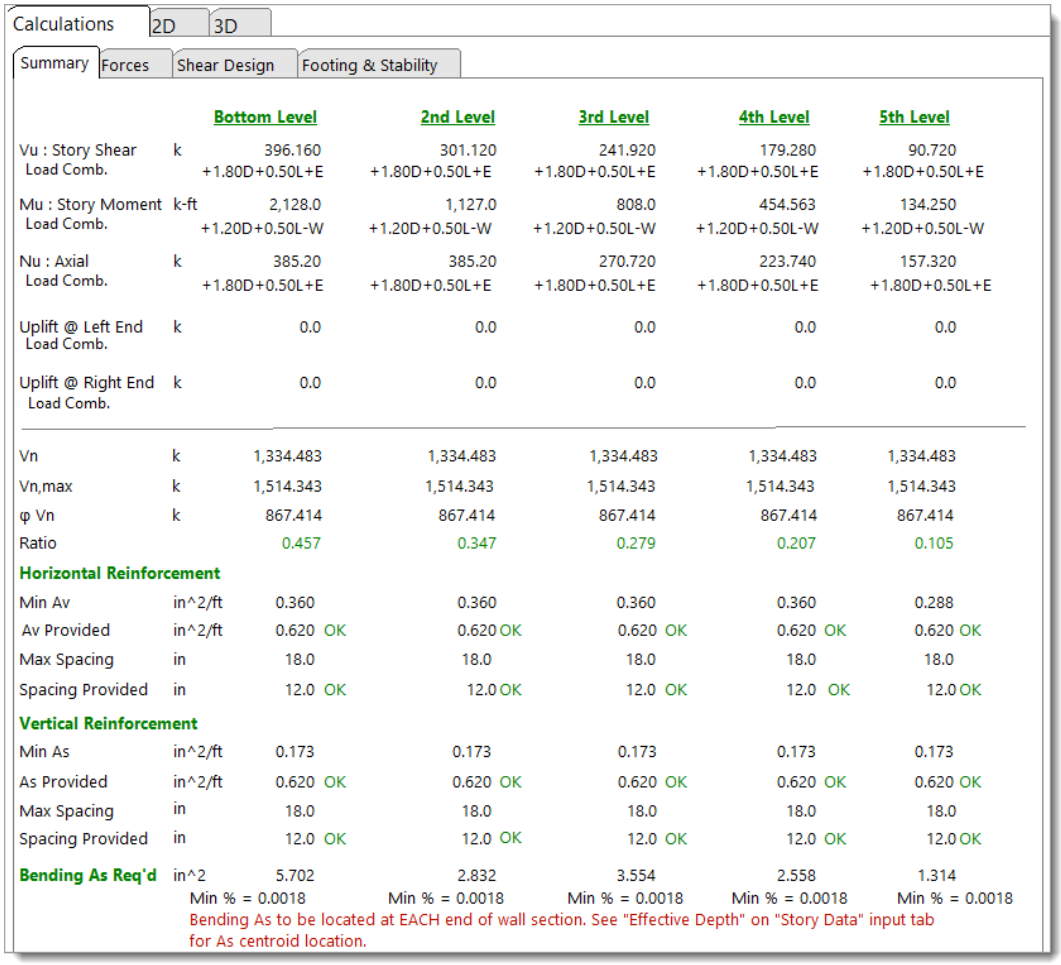
Horizontal As Req'd: The required horizontal reinforcement area per unit length of the wall (in^2/ft).

Vertical As Req'd: The required vertical reinforcement area per unit length of the wall (in^2/ft). Note: This is used only for prescriptive detailing per ACI 318 Chapter 11, not for flexural reinforcement.

Bending As Req'd: The total required flexural reinforcement area (in^2) at each level. This reinforcement must be placed at each end of the wall section. The centroid location for this reinforcement is determined by the "Effective Depth" specified in the "Story Data" tab.

© 2026 ENERCALC, LLC

ACI 318-19 only: Shear strength, horizontal steel area and spacing, vertical steel area and spacing, and end reinforcing for bending tension in that wall section are reported.



Calculations | 2D | 3D

Summary | Forces | Shear Design | Footing & Stability

		Bottom Level	2nd Level	3rd Level	4th Level	5th Level
Vu : Story Shear Load Comb.	k	396.160 +1.80D+0.50L+E	301.120 +1.80D+0.50L+E	241.920 +1.80D+0.50L+E	179.280 +1.80D+0.50L+E	90.720 +1.80D+0.50L+E
Mu : Story Moment Load Comb.	k-ft	2,128.0 +1.20D+0.50L-W	1,127.0 +1.20D+0.50L-W	808.0 +1.20D+0.50L-W	454.563 +1.20D+0.50L-W	134.250 +1.20D+0.50L-W
Nu : Axial Load Comb.	k	385.20 +1.80D+0.50L+E	385.20 +1.80D+0.50L+E	270.720 +1.80D+0.50L+E	223.740 +1.80D+0.50L+E	157.320 +1.80D+0.50L+E
Uplift @ Left End Load Comb.	k	0.0	0.0	0.0	0.0	0.0
Uplift @ Right End Load Comb.	k	0.0	0.0	0.0	0.0	0.0
Vn	k	1,334.483	1,334.483	1,334.483	1,334.483	1,334.483
Vn,max	k	1,514.343	1,514.343	1,514.343	1,514.343	1,514.343
φ Vn	k	867.414	867.414	867.414	867.414	867.414
Ratio		0.457	0.347	0.279	0.207	0.105
Horizontal Reinforcement						
Min Av	in^2/ft	0.360	0.360	0.360	0.360	0.288
Av Provided	in^2/ft	0.620 OK	0.620 OK	0.620 OK	0.620 OK	0.620 OK
Max Spacing	in	18.0	18.0	18.0	18.0	18.0
Spacing Provided	in	12.0 OK	12.0 OK	12.0 OK	12.0 OK	12.0 OK
Vertical Reinforcement						
Min As	in^2/ft	0.173	0.173	0.173	0.173	0.173
As Provided	in^2/ft	0.620 OK	0.620 OK	0.620 OK	0.620 OK	0.620 OK
Max Spacing	in	18.0	18.0	18.0	18.0	18.0
Spacing Provided	in	12.0 OK	12.0 OK	12.0 OK	12.0 OK	12.0 OK
Bending As Req'd	in^2	5.702 Min % = 0.0018	2.832 Min % = 0.0018	3.554 Min % = 0.0018	2.558 Min % = 0.0018	1.314 Min % = 0.0018

Bending As to be located at EACH end of wall section. See "Effective Depth" on "Story Data" input tab for As centroid location.

Shear Strength (Vn, Vn,max, φVn, Ratio): This section reports the governing shear strength demand-capacity ratio due to the applied shear force, as well as the associated shear strengths. More detail is provided in the description for the "Shear Design" tab below.

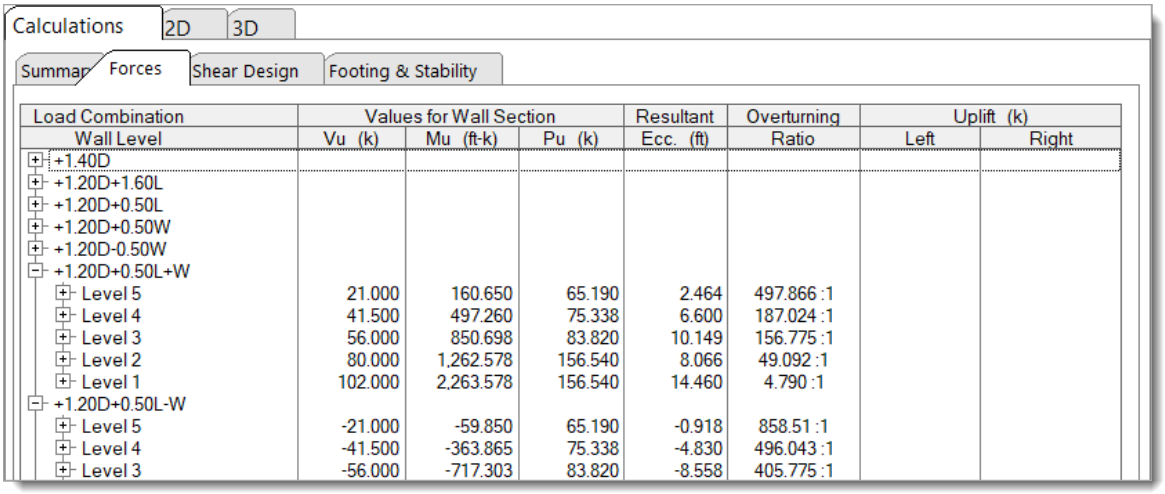
Horizontal/Vertical Reinforcement: This section summarizes the governing horizontal and vertical reinforcement checks, including minimum reinforcement checks per ACI 318-19 Section 11.6 and maximum bar spacing limits per Section 11.7. Detailed explanations can be found in the "Shear Design" tab description below.

Bending As Req'd: The total required flexural reinforcement area (in²) at each level. This reinforcement must be placed at each end of the wall section. The centroid location for this reinforcement is determined by the "Effective Depth" specified in the "Story Data" tab.

© 2026 ENERCALC, LLC

Forces

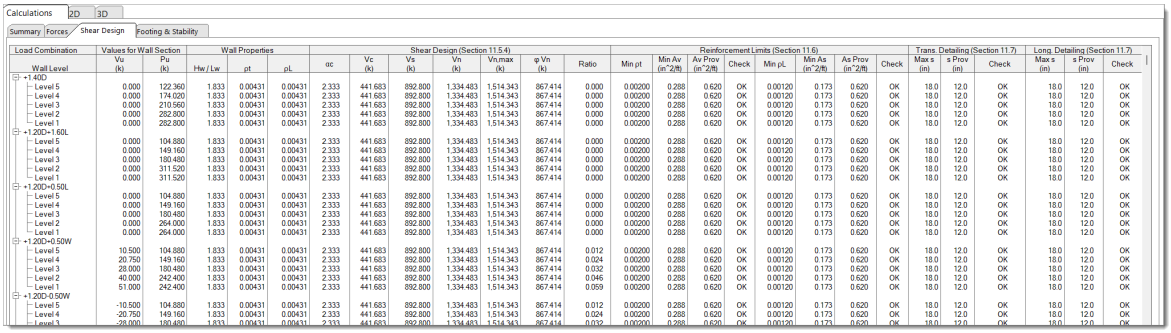
This tab provides the detailed force values for each wall level and for each load combination.



Load Combination / Wall Level	Values for Wall Section Vu (k)	Mu (ft-k)	Pu (k)	Resultant Ecc. (ft)	Overturning Ratio	Uplift (k) Left	Right
+1.40D							
+1.20D+1.60L							
+1.20D+0.50L							
+1.20D+0.50W							
+1.20D-0.50W							
+1.20D+0.50L+W							
Level 5	21.000	160.650	65.190	2.464	497.866 :1		
Level 4	41.500	497.260	75.338	6.600	187.024 :1		
Level 3	56.000	850.698	83.820	10.149	156.775 :1		
Level 2	80.000	1,262.578	156.540	8.066	49.092 :1		
Level 1	102.000	2,263.578	156.540	14.460	4.790 :1		
+1.20D+0.50L-W							
Level 5	-21.000	-59.850	65.190	-0.918	858.51 :1		
Level 4	-41.500	-363.865	75.338	-4.830	496.043 :1		
Level 3	-56.000	-717.303	83.820	-8.558	405.775 :1		

Shear Design

This tab provides the detailed shear design results for each wall level and for each load combination. This tab is only visible for ACI 318-19 and above.



Values for Wall Section

Vu (k): The applied shear force at each wall level under the given load combination.

Pu (k): The applied axial force at each wall level due to vertical loads.

Wall Properties

Hw/Lw: The wall aspect ratio of the entire wall stack.

pt: The total transverse (horizontal) reinforcement ratio in the wall.

© 2026 ENERCALC, LLC

pL: The total longitudinal (vertical) reinforcement ratio in the wall.

Shear Design (Section 11.5.4)

αc: Coefficient for the relative contribution of concrete strength to nominal wall shear strength, calculated per ACI 318-19 Section 11.5.4.3.

Vc (k): The shear strength provided by the concrete alone, calculated as $\alpha_c \lambda \sqrt{f'_c} A_{cv}$ per ACI 318-19 Eqn. 11.5.4.3.

Vs (k): The shear strength provided by the defined horizontal reinforcement, calculated as $\rho_t f_{yt} A_{cv}$ per ACI 318-19 Eqn. 11.5.4.3.

Vn (k): The total nominal shear strength, which is the sum of Vc and Vs per ACI 318-19 Eqn. 11.5.4.3.

Vn,max (k): The maximum allowable nominal shear strength for the wall section, calculated as $8\sqrt{f'_c} A_{cv}$ per ACI 318-19 Section 11.5.4.2.

φVn (k): The total factored shear strength, taken as φ times the minimum of Vn and Vn,max.

Ratio: The ratio of applied shear force Vu to factored shear strength φVn. A value of 1.0 or less indicates the wall is adequate for shear.

Reinforcement Limits (Section 11.6)

Min pt: The minimum required transverse (horizontal) reinforcement ratio to satisfy prescriptive reinforcement detailing requirements per ACI 318-19 Section 11.6.

Min Av (in²/ft): The minimum required area of horizontal reinforcement per unit length of the wall, calculated as [Min pt] * [Wall Thickness] * 12".

Av Prov (in²/ft): The provided horizontal reinforcement area per unit length.

Check: Indicates whether the provided horizontal reinforcement meets the prescriptive reinforcement limits.

Min pL: The minimum required longitudinal (vertical) reinforcement ratio to satisfy prescriptive reinforcement detailing requirements per ACI 318-19 Section 11.6.

Min As (in²/ft): The minimum required vertical reinforcement per unit length of the wall, calculated as [Min pL] * [Wall Thickness] * 12".

As Prov (in²/ft): The provided vertical reinforcement per unit length.

© 2026 ENERCALC, LLC

Check: Indicates whether the provided vertical reinforcement meets the prescriptive reinforcement limits.

Transverse Detailing (Section 11.7)

Max s (in): The maximum allowable spacing for transverse (horizontal) reinforcement per ACI 318-19 Section 11.7.3.

s Prov (in): The provided spacing of horizontal reinforcement.

Check: Indicates whether the provided spacing meets prescriptive reinforcement detailing requirements.

Longitudinal Detailing (Section 11.7)

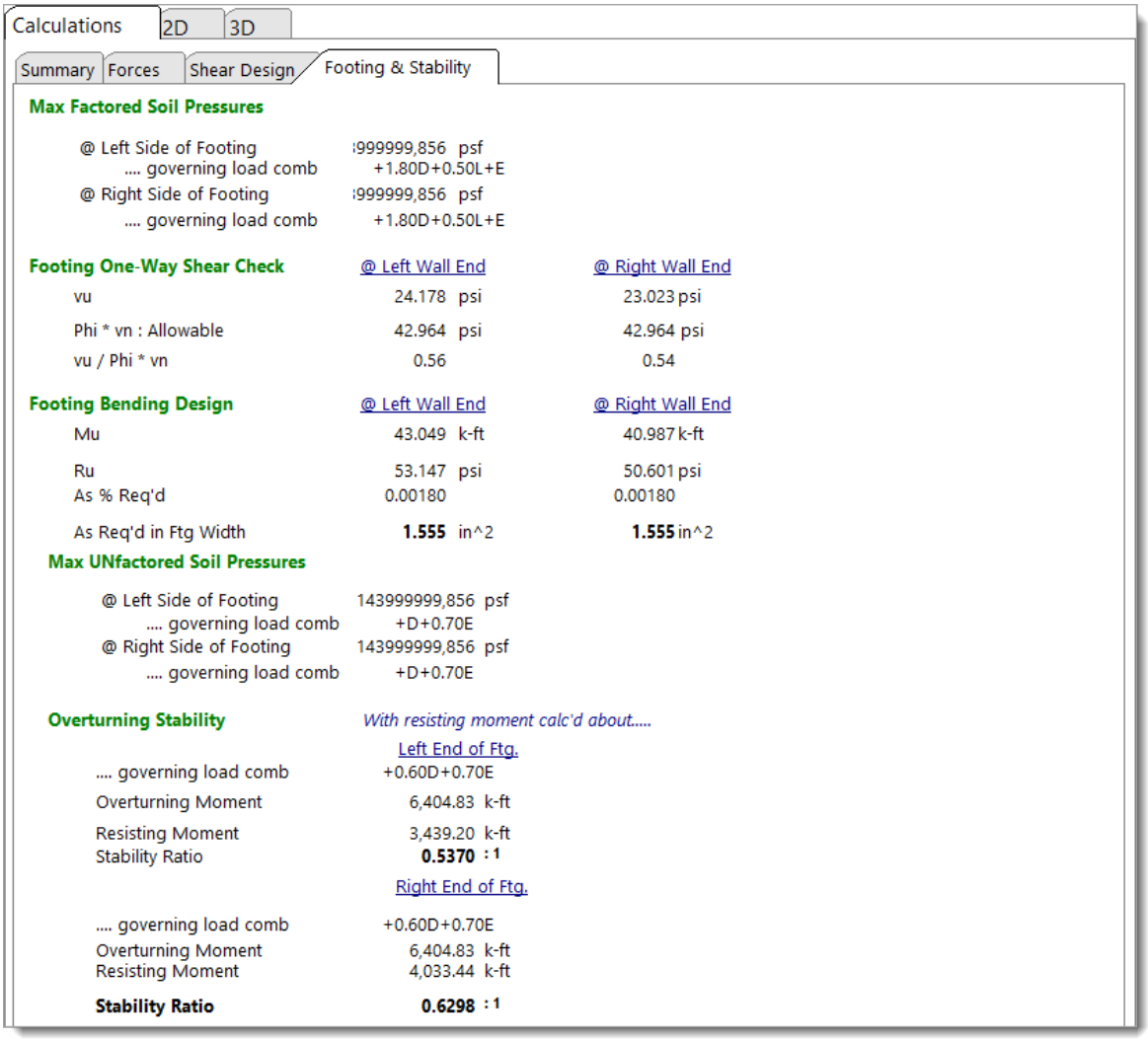
Max s (in): The maximum allowable spacing for longitudinal (vertical) reinforcement per ACI 318-19 Section 11.7.2.

s Prov (in): The provided spacing of vertical reinforcement.

Check: Indicates whether the provided spacing meets prescriptive reinforcement detailing requirements.

© 2026 ENERCALC, LLC

Footing & Stability



Calculations | 2D | 3D

Summary | Forces | Shear Design | Footing & Stability

Max Factored Soil Pressures

@ Left Side of Footing	:999999,856	psf
.... governing load comb	+1.80D+0.50L+E	
@ Right Side of Footing	:999999,856	psf
.... governing load comb	+1.80D+0.50L+E	

Footing One-Way Shear Check	@ Left Wall End		@ Right Wall End	
vu	24.178	psi	23.023	psi
Phi * vn : Allowable	42.964	psi	42.964	psi
vu / Phi * vn	0.56		0.54	

Footing Bending Design	@ Left Wall End		@ Right Wall End	
Mu	43.049	k-ft	40.987	k-ft
Ru	53.147	psi	50.601	psi
As % Req'd	0.00180		0.00180	
As Req'd in Ftg Width	**1.555**	in^2	**1.555**	in^2

Max UNfactored Soil Pressures

@ Left Side of Footing	143999999,856	psf
.... governing load comb	+D+0.70E	
@ Right Side of Footing	143999999,856	psf
.... governing load comb	+D+0.70E	

Overturning Stability	*With resisting moment calc'd about.....*	
	Left End of Ftg.	
.... governing load comb	+0.60D+0.70E	
Overturning Moment	6,404.83	k-ft
Resisting Moment	3,439.20	k-ft
Stability Ratio	**0.5370**	:1
	Right End of Ftg.	
.... governing load comb	+0.60D+0.70E	
Overturning Moment	6,404.83	k-ft
Resisting Moment	4,033.44	k-ft
Stability Ratio	**0.6298**	:1

Footing Design Combination Pressures: Reports the maximum factored soil pressures at the left and right sides of the footing, along with the load combination responsible for producing each of those pressures. These are used in the reinforced concrete design calculations for the footing.

Footing One-Way Shear Check: Reports the calculated one-way shear at both ends of the footing.

For footings designed per ACI 318-14 and earlier, and ACI 318-25 and later:

The nominal shear strength follows the one-way shear provisions of ACI 318-14 §22.5.5.1 / ACI 318-25 §13.2.6.2:

$\varphi v_n = \varphi * 2 * \lambda * \sqrt{(f'_c)}$, where $\varphi = 0.75$.

For footings designed per ACI 318-19:

The nominal shear strength follows the one-way shear provisions of §22.5 and Table 22.5.5.1 (referenced by §7.5.3.1):

© 2026 ENERCALC, LLC

$\varphi v_n = \varphi * 8 * \lambda_s * \lambda * (\rho_w)^{1/3} * \sqrt{(f'_c)}$, where $\varphi = 0.75$.

Note: The one-way shear capacity assumes no transverse reinforcement in the footing. Therefore, $A_v < A_{v,min}$, and equations (a) and (b) from Table 22.5.5.1 are not applicable.
It is also acceptable to neglect the size effect factor (λ_s) for one-way shallow foundations per §13.2.6.3, so $\lambda_s = 1.0$.

This value is also subject to additional limits per §22.5.5.1.1:
Lower bound: $\varphi * \lambda * \sqrt{(f'_c)}$
Upper bound: $\varphi * 5 * \lambda * \sqrt{(f'_c)}$

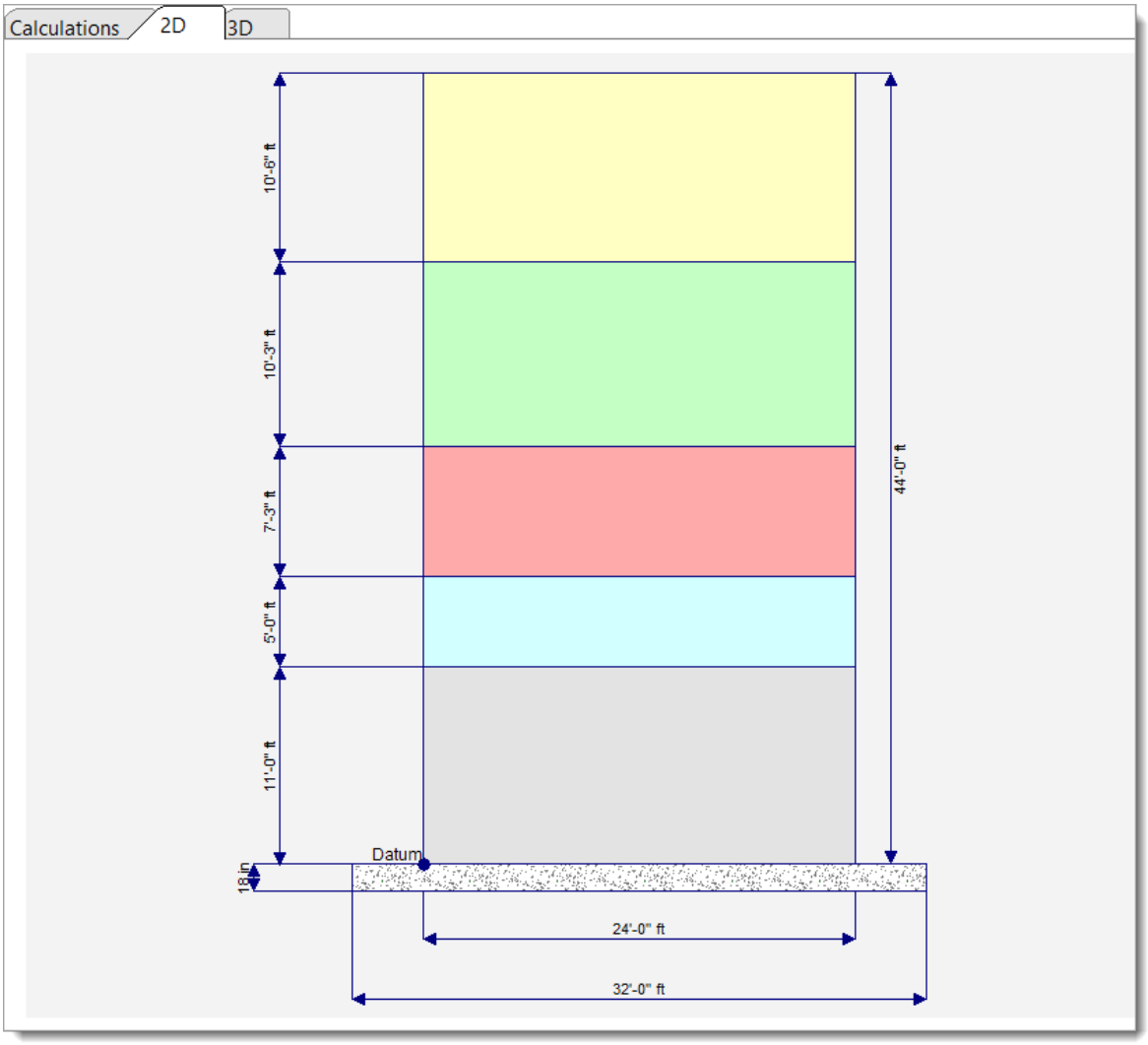
Footing Bending Design: Reports the results of the flexural analysis and design including recommended reinforcing options to satisfy the required area of steel.

Soil Pressure Combination Pressures: Reports the maximum unfactored soil pressures at the left and right sides of the footing, along with the load combination responsible for producing each of those pressures. Unfactored soil pressures are compared to the allowable soil bearing pressure provided by the user.

Overturning Stability: Reports the overturning analysis performed about each end of the footing including the overturning moment, the resisting moment, the stability ratio, and the governing load combination.

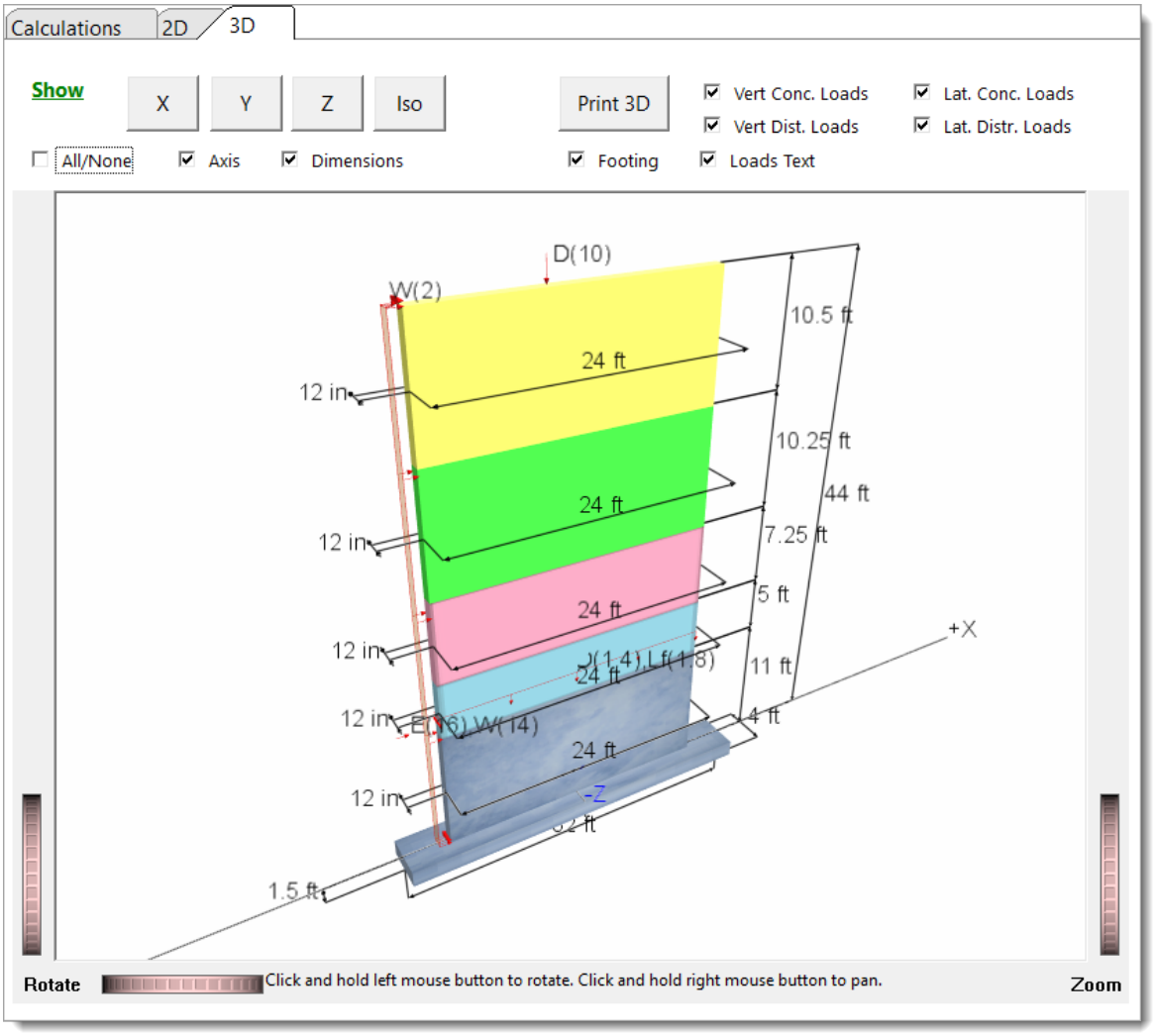
© 2026 ENERCALC, LLC

2D Sketch



© 2026 ENERCALC, LLC

3D Rendering



13.7.2.2 Masonry Shear Wall

General

This module allows the design of Ordinary Reinforced Masonry Shear Walls including multi-story walls with no openings but with up to five levels of differing length, height, thickness, grouting and reinforcing patterns.

The wall will be composed of two zones:

Chord Zone: The areas at both ends of each level that contain chord rebar in every cell. (Always solid grouted.)

Field of Wall: The area between chord zones that may have a different reinforcing/grouting pattern. (May be partially-grouted or solid-grouted.)

Convenience Features

© 2026 ENERCALC, LLC

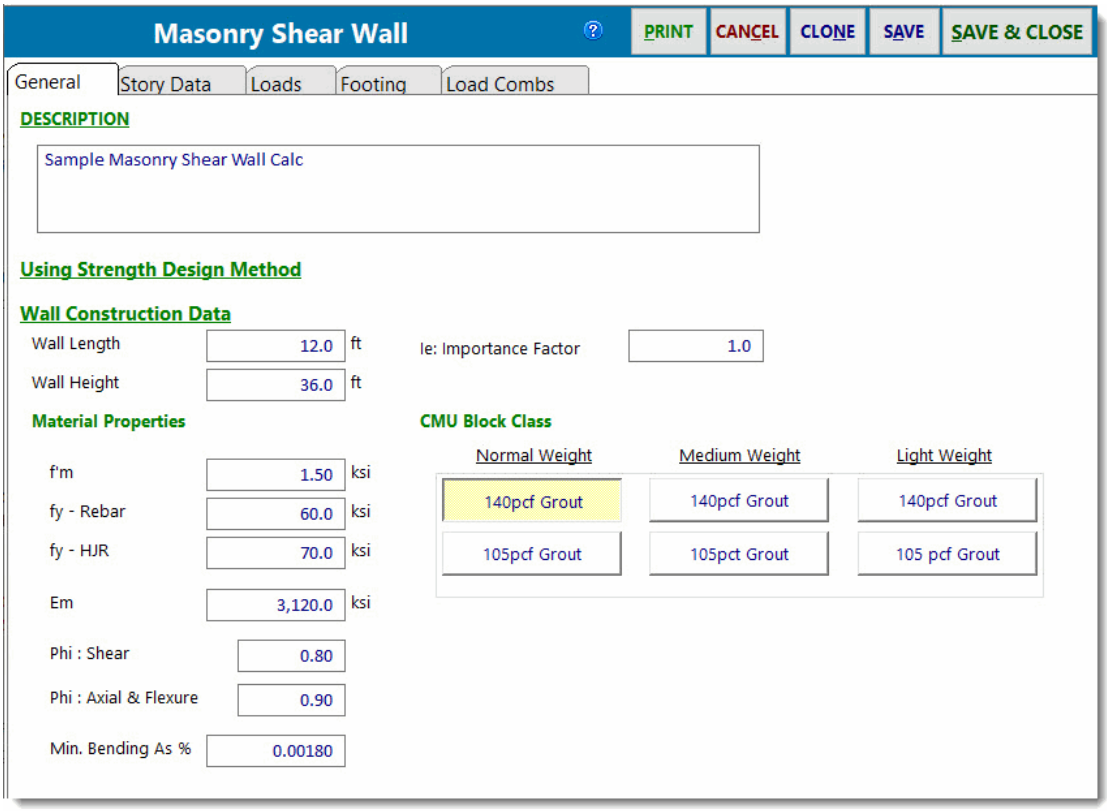
The module collects data on a level-by-level basis to allow the user to account for varying reinforcing and grouting patterns. It also allows some convenience features such as the ability to specify:

- Reinforcing and grouting that can vary from one story to the next,
- Wall offsets to account for conditions where the length of wall at one story is shorter than the length of the wall below,
- A continuous footing,
- Solid grouting,
- Chord rebar that is different from the rebar in the field of the wall,
- Bond beams and/or horizontal joint reinforcing for shear reinforcing.

Limitations

Masonry Shear Wall does not allow for the design of openings. It only support Ordinary Reinforced Shear Walls and does not allow for the design of special boundary elements, but it does incorporate the checks to verify that special boundary elements are or are not required.

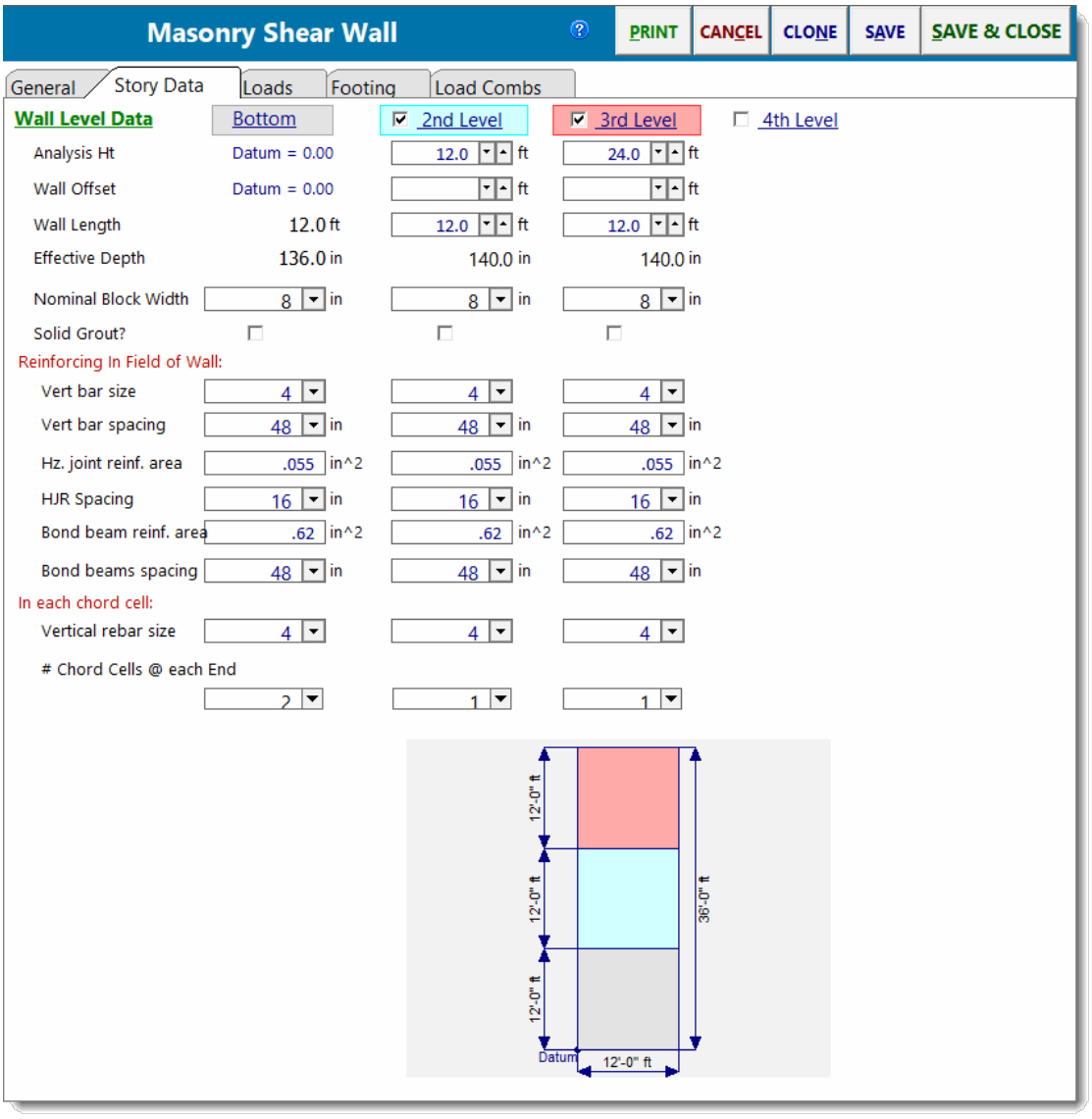
General



Collects overall wall length and height, material properties, and strength reduction factors.

© 2026 ENERCALC, LLC

Story Data



Select checkboxes as necessary to specify the number of stories. Shear wall is assumed to be laterally braced at each defined story level.

Analysis Height: The heights at which the story framing is assumed to brace the shear wall.

Wall Offset: A lateral offset dimension that can be used to shift the left edge of the wall at the selected level.

Wall Length: The distance from the left end of the wall to the right end, at the selected level. This can be used to shift the right edge of the wall.

© 2026 ENERCALC, LLC

Effective Depth: The dimension from the compression edge of the wall to the centroid of the chord steel.

Nominal Block Width: Use the dropdown to select the block width.

Solid Grout: Reinforced cells will always be assumed to be grouted, but this option provides a way to tell the program that ALL cells in the selected level will be grouted, regardless of whether they contain rebar or not.

Vertical Bar Size: Use the dropdown to select the size of the vertical rebar that will be used in the field of the wall.

Vertical Bar Spacing: Specify the spacing of vertical rebar in the field of the wall.

Horizontal Joint Reinforcing (HJR) Area: Specify the effective cross-sectional area of one piece of horizontal joint reinforcing, if it is to be considered as shear reinforcing.

Horizontal Joint Reinforcing (HJR) Spacing: Specify the vertical spacing of horizontal joint reinforcing, if it is to be considered as shear reinforcing.

Bond Beam Reinforcing Area: Specify the effective cross-sectional area of rebar in one bond beam, if it is to be considered as shear reinforcing.

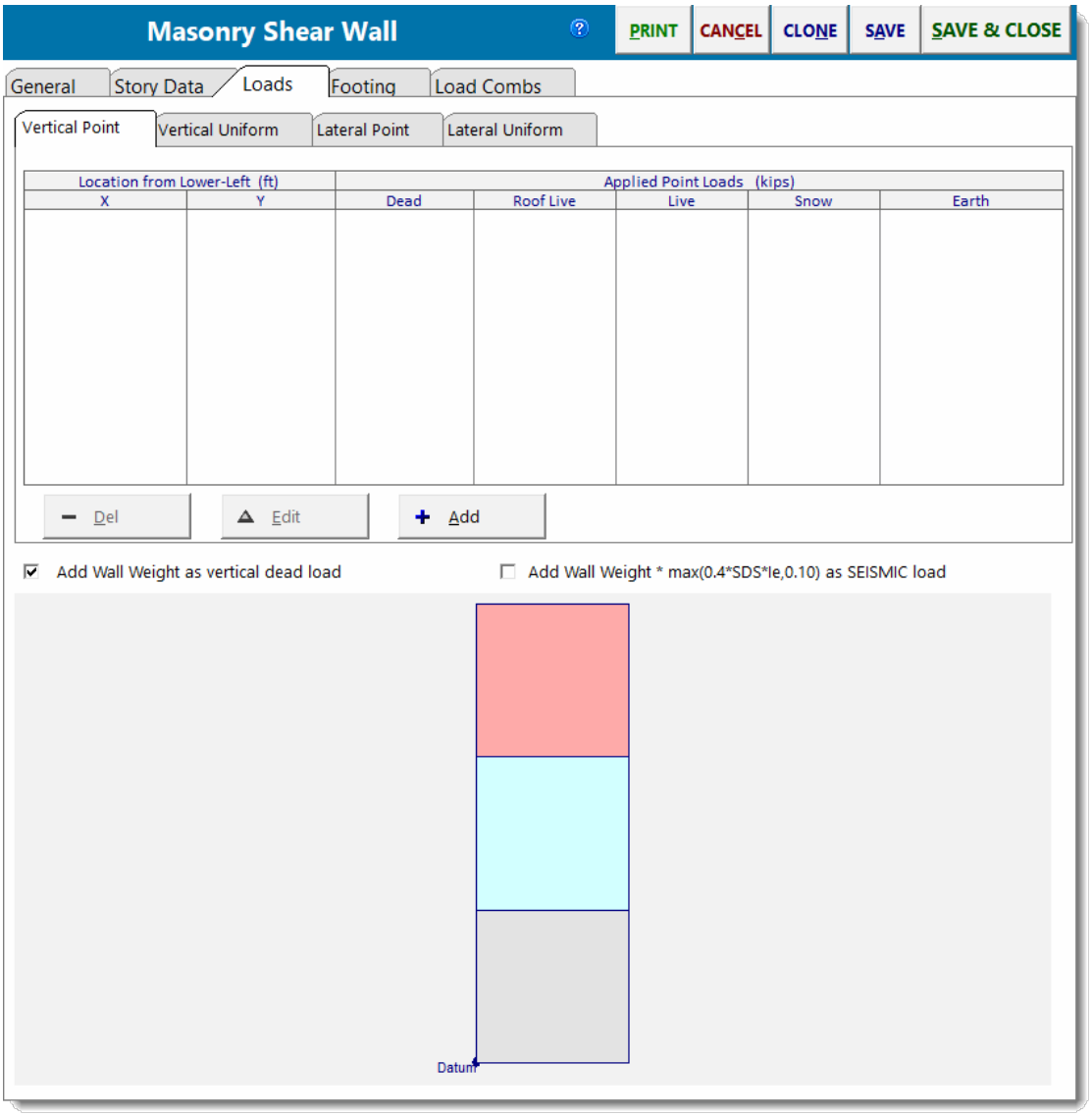
Spacing of Bond Beams: Specify the vertical spacing of bond beam reinforcing, if it is to be considered as shear reinforcing.

Vertical Rebar Size (Chords): Use the dropdown to select the size of the vertical rebar that will be used in each solid-grouted chord cell.

 # Chord Cells @ Each End: Specify the number of solid-grouted reinforced chord cells at each end of the selected level.

Loads

© 2026 ENERCALC, LLC



Use the tabs to specify magnitude and location of:

- Vertical Point Loads
- Vertical Uniform Loads
- Lateral Point Loads, and
- Lateral Uniform Loads.

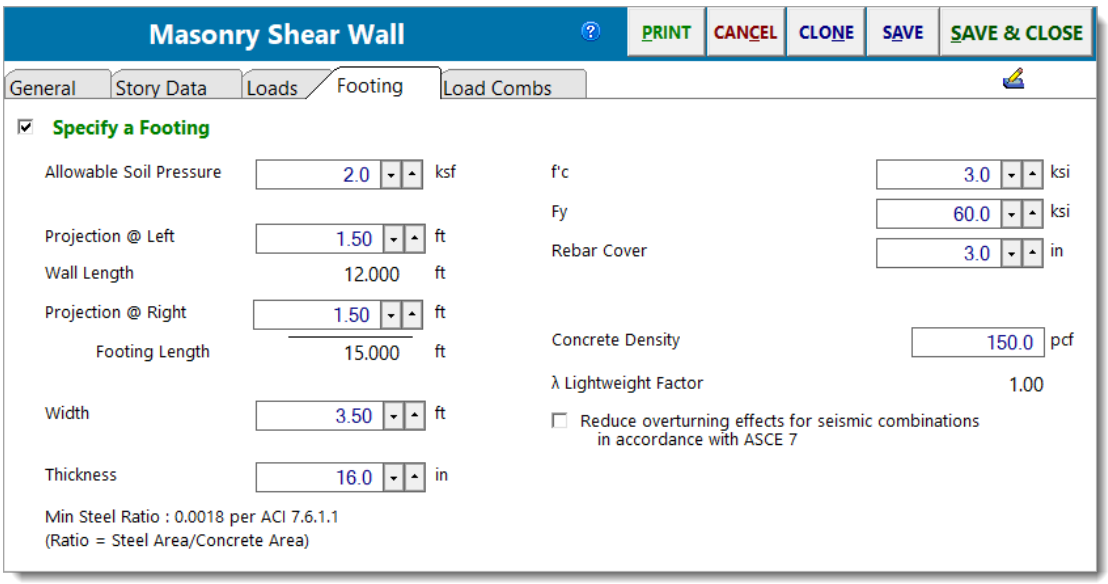
Also offers the option to automatically consider wall weight as vertical dead load and/or automatically calculate seismic load due to wall weight.

Footing

Optional tab to design a footing that behaves as a continuous grade beam that cantilevers past the end of the wall.

© 2026 ENERCALC, LLC

Specify geometry, allowable soil pressure, and material properties to arrive at checks for soil pressure and footing flexure.



Specify a Footing: If this checkbox is selected, then the input fields for defining a footing are displayed. Note: This checkbox also causes some footing-related input to be shown or hidden as appropriate on the Loads tab.

Allowable Soil Pressure: Specify the allowable soil bearing pressure in units of ksf.

Projection @ Left: Enter the projection of the footing beyond the left end of the wall in units of ft.

Wall Length: The program reports the shear wall length in units of ft for reference.

Projection @ Right: Enter the projection of the footing beyond the right end of the wall in units of ft.

Footing Length: The program reports the footing length in units of ft for reference.

Width: Specify the width of the footing in units of ft. This is the dimension perpendicular to the wall length.

Thickness: Enter the thickness of the footing in units of inches.

Min Steel Reinforcing: The minimum flexural reinforcement ratio, per ACI 7.6.1.1, is 0.0018 times the gross area of the concrete (Width x Thickness). Commentary R7.6.1.1 also notes that minimum flexural reinforcement should be placed as close as possible to the tension face of the concrete, unlike shrinkage and temperature reinforcement which may be distributed between the top and bottom faces of the footing.

© 2026 ENERCALC, LLC

f'c: Specify the compressive stress of concrete in units of ksi.

Fy: Specify the yield stress of rebar in units of ksi.

Rebar Cover: Specify the cover over rebar in units of inches. The program will use this value and a 1/2" allowance for the rebar size to calculate the effective depth of the concrete section.

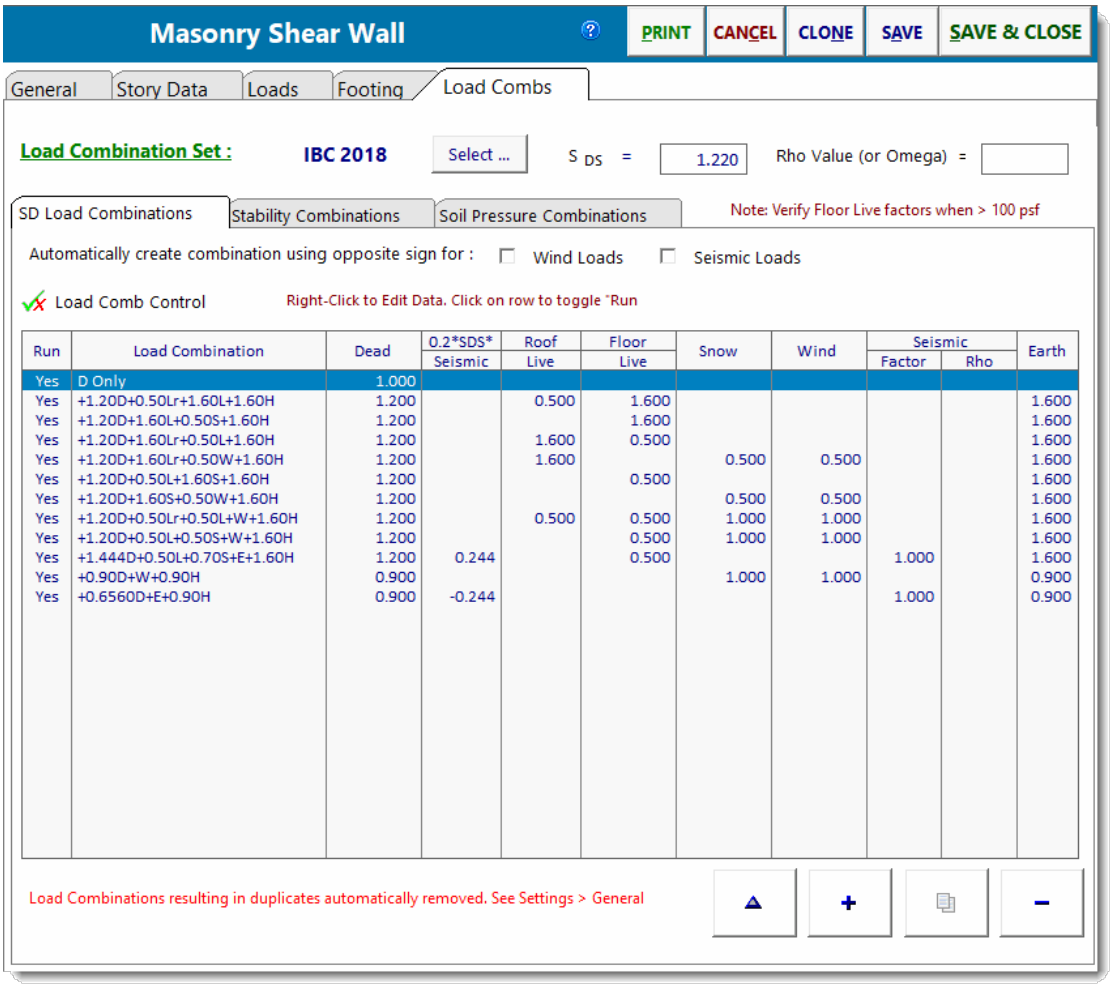
Note: Rebar is assumed to exist only at the bottom of the footing to resist tensile forces from the vertical loads and increased pressure due to overturning forces. Tension in the top of the footing in cases where no upward soil pressure exists and the footing weight creates a downward net force IS IGNORED.

Concrete Density: Specify the density of concrete in units of pcf.

Reduce overturning effects for seismic combinations per ASCE 7-16 Section 12.13.4: Applies the referenced reduction in overturning force for seismic load combinations when selected.

Load Combinations

© 2026 ENERCALC, LLC



Run	Load Combination	Dead	0.2*SDS* Seismic	Roof Live	Floor Live	Snow	Wind	Seismic Factor	Seismic Rho	Earth
Yes	D Only	1.000								
Yes	+1.20D+0.50Lr+1.60L+1.60H	1.200		0.500	1.600					1.600
Yes	+1.20D+1.60L+0.50S+1.60H	1.200			1.600					1.600
Yes	+1.20D+1.60Lr+0.50L+1.60H	1.200		1.600	0.500					1.600
Yes	+1.20D+1.60Lr+0.50W+1.60H	1.200		1.600		0.500	0.500			1.600
Yes	+1.20D+0.50L+1.60S+1.60H	1.200			0.500					1.600
Yes	+1.20D+1.60S+0.50W+1.60H	1.200				0.500	0.500			1.600
Yes	+1.20D+0.50Lr+0.50L+W+1.60H	1.200		0.500	0.500	1.000	1.000			1.600
Yes	+1.20D+0.50L+0.50S+W+1.60H	1.200			0.500	1.000	1.000			1.600
Yes	+1.444D+0.50L+0.70S+E+1.60H	1.200	0.244		0.500			1.000		1.600
Yes	+0.90D+W+0.90H	0.900				1.000	1.000			0.900
Yes	+0.6560D+E+0.90H	0.900	-0.244					1.000		0.900

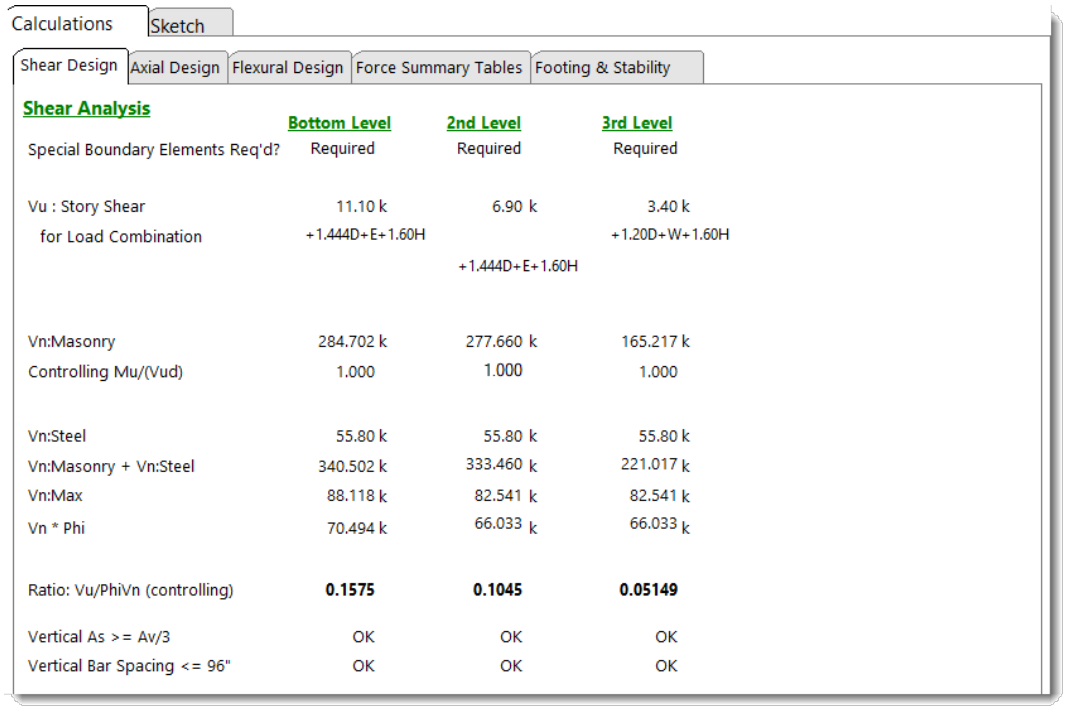
Collects settings that control how load combinations will be generated and applied to the analysis/design of:

- Masonry and reinforcing
- Footing and reinforcing
- Stability
- Soil Pressure

Output

Shear Design

© 2026 ENERCALC, LLC



Calculations | Sketch

Shear Design | Axial Design | Flexural Design | Force Summary Tables | Footing & Stability

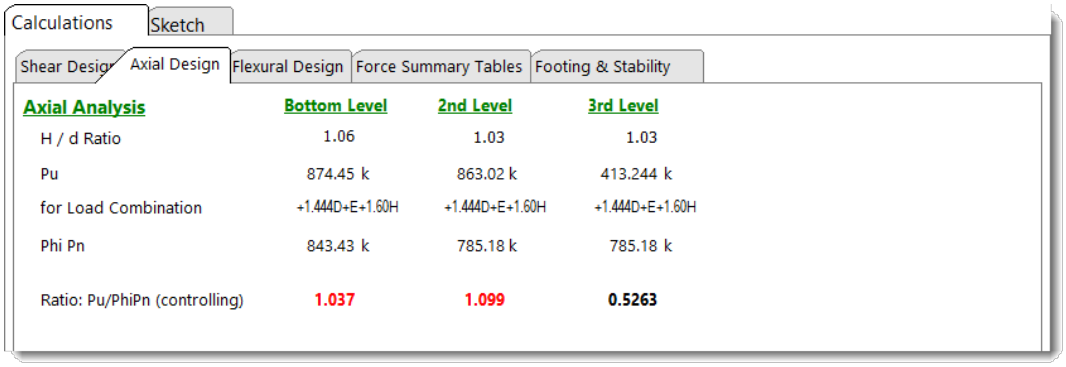
Shear Analysis

	Bottom Level	2nd Level	3rd Level
Special Boundary Elements Req'd?	Required	Required	Required
Vu : Story Shear	11.10 k	6.90 k	3.40 k
for Load Combination	+1.444D+E+1.60H	+1.444D+E+1.60H	+1.20D+W+1.60H
Vn:Masonry	284.702 k	277.660 k	165.217 k
Controlling Mu/(Vud)	1.000	1.000	1.000
Vn:Steel	55.80 k	55.80 k	55.80 k
Vn:Masonry + Vn:Steel	340.502 k	333.460 k	221.017 k
Vn:Max	88.118 k	82.541 k	82.541 k
Vn * Phi	70.494 k	66.033 k	66.033 k
Ratio: Vu/PhiVn (controlling)	**0.1575**	**0.1045**	**0.05149**
Vertical As >= Av/3	OK	OK	OK
Vertical Bar Spacing <= 96"	OK	OK	OK

Reports:

- Need for special boundary elements
- Factored shear force and controlling load combination
- Shear strength from masonry
- Shear strength from reinforcing
- Limiting shear strength
- Design shear strength
- Design ratio
- Code requirements for area and spacing of vertical rebar.

Axial Design



Calculations | Sketch

Shear Design | Axial Design | Flexural Design | Force Summary Tables | Footing & Stability

Axial Analysis

	Bottom Level	2nd Level	3rd Level
H / d Ratio	1.06	1.03	1.03
Pu	874.45 k	863.02 k	413.244 k
for Load Combination	+1.444D+E+1.60H	+1.444D+E+1.60H	+1.444D+E+1.60H
Phi Pn	843.43 k	785.18 k	785.18 k
Ratio: Pu/PhiPn (controlling)	**1.037**	**1.099**	**0.5263**

Reports:

- H/d ratio
- Factored axial load and controlling load combination

© 2026 ENERCALC, LLC

- Design axial load
- Design ratio

Flexural Design

Calculations | Sketch

Shear Design | Axial Design | Flexural Design | Force Summary Table | Footing & Stability

Flexural Analysis	Bottom Level	2nd Level	3rd Level
"a" : Flexural compression	2.623 in	1.311 in	1.311 in
Length of defined chord zone is >= the "a" dimension of the masonry (the compression zone)	OK	OK	OK
"d" : Eff depth to tension reinf	136.0 in	140.0 in	140.0 in
Pu	58.800 k	50.784 k	19.392 k
for Load Combination	+D+0.75L+0.525E	+D+0.75L+0.525E	+D+0.75L+0.525E
As-Flex	0.400 in^2	0.200 in^2	0.200 in^2
As-max	6.423 in^2	6.774 in^2	7.297 in^2
As-flex < As-max ? * Max flexural reinforcement assumes ordinary reinforced shear wall.	OK	OK	OK
Mu	226.80 k-ft	151.20 k-ft	84.0 k-ft
for Load Combination	+1.444D+E	+1.20D+W	+1.20D+W
Phi Mn	242.439 k-ft	125.410 k-ft	125.410 k-ft
Ratio: Mu/PhiMn (controlling)	**0.9355**	**1.2060**	**0.6698**

"a" : Flexural compression: Reports the actual depth of the flexural compression zone.

Length of defined chord zone is >= the "a" dimension of the masonry (the compression zone): Checks whether the defined solid grouted chord zone is deep enough to fully contain the required compression block ("a"). If the required depth ("a") exceeds the grouted region and extends into an ungrouted portion of the wall, this condition will report "NG" (Not Good). In such cases, increase the number of grouted chord cells at the wall ends or consider fully grouting the wall.

"d" : Eff depth to tension reinforcement: Reports the effective depth from the extreme compression fiber to the centroid of the chord reinforcement at each wall end.

Pu for Load Combination +D+0.75L+0.525E: Displays the factored axial load considered when calculating the maximum allowable steel area, As-max, per section 9.3.3.2.1.

As-flex: Reports the actual total area of chord reinforcement in tension.

© 2026 ENERCALC, LLC

As-max: Reports the maximum permissible area of chord reinforcement in tension, based on the following formula (from TMS 402-16, Section 9.3.3.2 Commentary):

$$\rho_{max} = \frac{A_{s-max}}{bd} = \frac{0.64 f'_m \left(\frac{\varepsilon_{mu}}{\varepsilon_{mu}+\varepsilon_t}\right) - \frac{P_u}{bd}}{f_y}$$

Where:

- f'_m = Specified compressive strength of masonry
- f_y = Yield strength of reinforcement
- $\varepsilon_{mu} = 0.0025$; Maximum usable compressive strain in masonry (per Section 9.3.2)
- ε_t = Net tensile strain in reinforcement (see below)
- $E_s = 29{,}000ksi$; Modulus of elasticity of steel
- P_u = Axial load (from combination +D+0.75L+0.525E)
- b = Wall thickness
- d = Effective depth to reinforcement centroid

This formula is used to calculated the maximum reinforcement, As-max, by considering the prescribed strain distribution, determining the corresponding stress and force distribution, and using statics to sum axial forces.

TMS 402-16: $\varepsilon_t = 1.5 * f_y / E_s$.

TMS 402-22: ε_t is based on the user-defined "Phi : Axial & Flexure" value from the General tab, per Table 9.1.4:

- For $\varphi \geq 0.9$: $\varepsilon_t = 0.003 + f_y / E_s$
- For $\varphi \leq 0.65$: $\varepsilon_t = f_y / E_s$
- For $0.65 < \varphi < 0.9$: $\varepsilon_t = \frac{\varphi - 0.65}{0.25} * 0.003 + f_y / E_s$

As-flex < As-max? : Reports whether the provided reinforcement (As-flex) is within the allowable limit (As-max).

TMS 402-16: "NG" indicates As-flex exceeds the limit per Section 9.3.3.2. Reduce reinforcement to meet ductility requirements.

TMS 402-22: "NG" means As-flex exceeds the limit based on the user-supplied ϕ value. To correct this, reduce reinforcement or lower the ϕ factor to increase As-max.

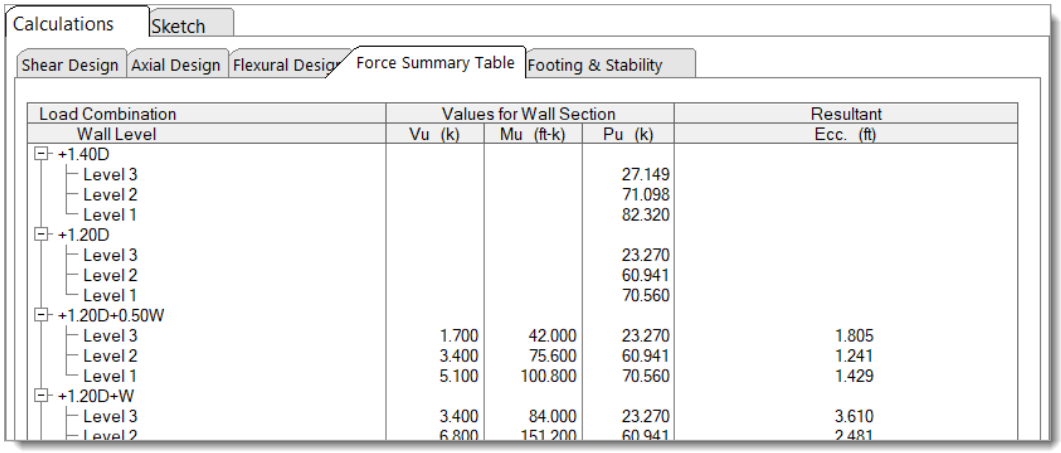
Mu for Load Combination: Reports the factored moment and its controlling load combination.

© 2026 ENERCALC, LLC

Phi Mn: Reports the design moment capacity.

Ratio: Mu/PhiMn (controlling): Reports the ratio of factored moment to design moment capacity (Mu / Phi Mn).

Force Summary Table



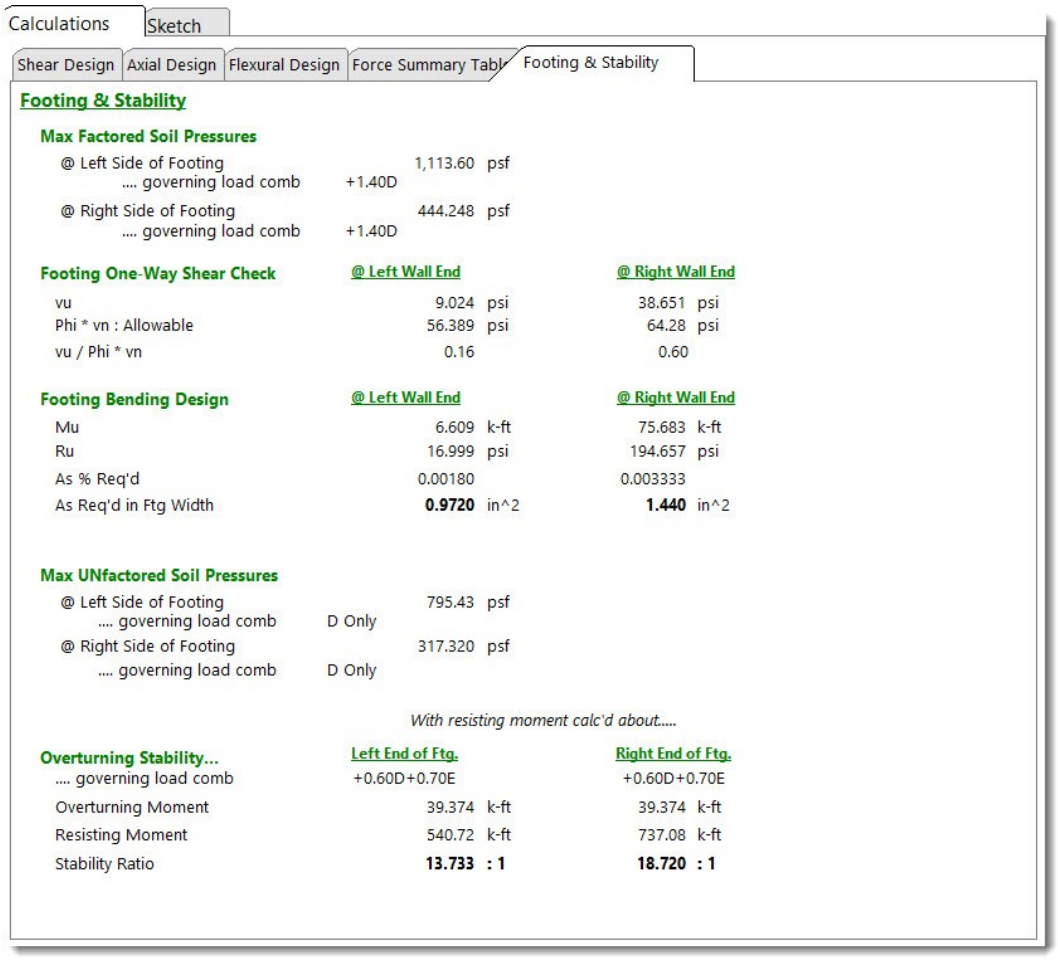
Load Combination / Wall Level	Values for Wall Section Vu (k)	Mu (ft-k)	Pu (k)	Resultant Ecc. (ft)
+1.40D				
Level 3			27.149	
Level 2			71.098	
Level 1			82.320	
+1.20D				
Level 3			23.270	
Level 2			60.941	
Level 1			70.560	
+1.20D+0.50W				
Level 3	1.700	42.000	23.270	1.805
Level 2	3.400	75.600	60.941	1.241
Level 1	5.100	100.800	70.560	1.429
+1.20D+W				
Level 3	3.400	84.000	23.270	3.610
Level 2	6.800	151.200	60.941	2.481

For each load combination, this table reports:

- Vu
- Mu
- Pu
- Eccentricity
- Overturning Ratio
- Uplift

Footing & Stability

© 2026 ENERCALC, LLC



Calculations | Sketch

Shear Design | Axial Design | Flexural Design | Force Summary Tabl | Footing & Stability

Footing & Stability

Max Factored Soil Pressures

@ Left Side of Footing		1,113.60	psf
.... governing load comb	+1.40D		
@ Right Side of Footing		444.248	psf
.... governing load comb	+1.40D		

Footing One-Way Shear Check	@ Left Wall End		@ Right Wall End	
vu	9.024	psi	38.651	psi
Phi * vn : Allowable	56.389	psi	64.28	psi
vu / Phi * vn	0.16		0.60	

Footing Bending Design	@ Left Wall End		@ Right Wall End	
Mu	6.609	k-ft	75.683	k-ft
Ru	16.999	psi	194.657	psi
As % Req'd	0.00180		0.003333	
As Req'd in Ftg Width	**0.9720**	in^2	**1.440**	in^2

Max UNfactored Soil Pressures

@ Left Side of Footing		795.43	psf
.... governing load comb	D Only		
@ Right Side of Footing		317.320	psf
.... governing load comb	D Only		

With resisting moment calc'd about.....

Overturning Stability...	Left End of Ftg.		Right End of Ftg.	
.... governing load comb	+0.60D+0.70E		+0.60D+0.70E	
Overturning Moment	39.374	k-ft	39.374	k-ft
Resisting Moment	540.72	k-ft	737.08	k-ft
Stability Ratio	**13.733**	: 1	**18.720**	: 1

Footing Design Combination Pressures: Reports the maximum factored soil pressures at the left and right sides of the footing, along with the load combination responsible for producing each of those pressures. These are used in the reinforced concrete design calculations for the footing.

Footing One-Way Shear Check: Reports the calculated one-way shear at both ends of the footing.

For footings designed per ACI 318-14 and earlier, and ACI 318-25 and later:

The nominal shear strength follows the one-way shear provisions of ACI 318-14 §22.5.5.1 / ACI 318-25 §13.2.6.2:

$\varphi v_n = \varphi * 2 * \lambda * \sqrt{(f'_c)}$, where $\varphi = 0.75$.

For footings designed per ACI 318-19:

The nominal shear strength follows the one-way shear provisions of §22.5 and Table 22.5.5.1 (referenced by §7.5.3.1):

$\varphi v_n = \varphi * 8 * \lambda_s * \lambda * (\rho_w)^{1/3} * \sqrt{(f'_c)}$, where $\varphi = 0.75$.

© 2026 ENERCALC, LLC

Note: The one-way shear capacity assumes no transverse reinforcement in the footing. Therefore, $A_v < A_{v,min}$, and equations (a) and (b) from Table 22.5.5.1 are not applicable.
It is also acceptable to neglect the size effect factor (λ_s) for one-way shallow foundations per §13.2.6.3, so $\lambda_s = 1.0$.

This value is also subject to additional limits per §22.5.5.1.1:
Lower bound: $\varphi * \lambda * \sqrt{(f'_c)}$
Upper bound: $\varphi * 5 * \lambda * \sqrt{(f'_c)}$

Footing Bending Design: Reports the results of the flexural analysis and design including recommended reinforcing options to satisfy the required area of steel.

Soil Pressure Combination Pressures: Reports the maximum unfactored soil pressures at the left and right sides of the footing, along with the load combination responsible for producing each of those pressures. Unfactored soil pressures are compared to the allowable soil bearing pressure provided by the user.

Overturning Stability: Reports the overturning analysis performed about each end of the footing including the overturning moment, the resisting moment, the stability ratio, and the governing load combination.

13.7.2.3 Wood Shear Wall

Basis

The NDS Special Design Provisions for Wind & Seismic allows for three approaches to wood shear wall design:

1. Force Transfer Shear Walls
2. Perforated Shear Walls
3. Individual Full-Height Wall Segment Shear Walls.

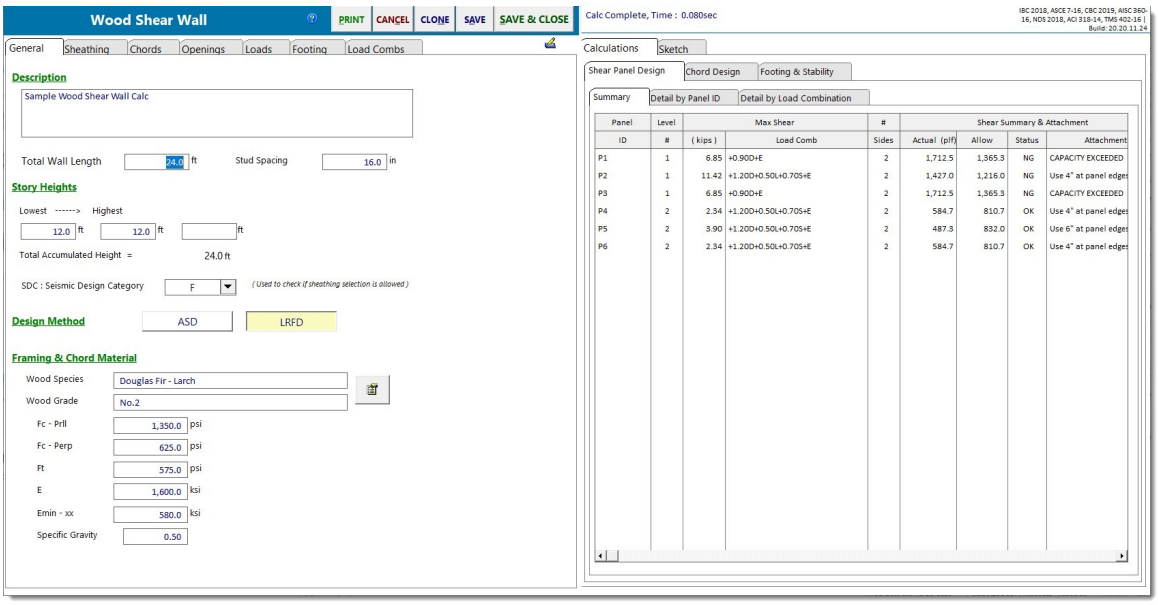
Force Transfer Shear Walls and Perforated Shear Walls allow the designer to use more favorable analysis assumptions, and offers some benefits in terms of construction details. But it comes at a cost of the designer having to perform more calculations and designs to follow the load path through all of the headers, jambs, sills, etc., and their connections.

At present, the Wood Shear Wall module implements the Individual Full-Height Wall Segment Shear Walls method. This is the most straightforward method from the design standpoint. This method still requires the evaluation of h/b ratios to decide whether to eliminate slender segments or suffer the imposed reduction on the nominal shear capacity based on the slenderness of each panel in the wall. This method does not require consideration of the reduction factor (Co) that applies to Perforated Shear Walls.

Section 4.3.2.1 of SDPWS 2021 refers to Section 4.3.3, for the reduction due to aspect ratio. But the implementation within ENERCALC SEL goes a little further by allowing the user to specify openings. The module then takes the defined openings and considers only the solid stacked portions of remaining wall as being effective at resisting lateral loads. So the module actually applies the 2b/h reduction as permitted in Exception 1 in

© 2026 ENERCALC, LLC

Section 4.3.5.5.1, rather than using the aspect ratio formula in Section 4.3.3. It is always more conservative than the adjustment in 4.3.3, but it allows the module to handle shear walls in a line because, by definition within the module, all segments will be “sheathed with the same materials and construction”.



Overview

The Wood Shear Wall module allows the user to define overall geometry, openings (if any), sheathing type, chord member species, grade, and size, applied loads, and a footing (if desired). The module then evaluates the resulting shear panels, chords, and footing (if defined).

Sheathing is evaluated for aspect ratio and unit shear due to load combinations that include either wind or seismic. The module considers the selected sheathing type, sheathing thickness, fastener size, blocking condition, and the species of the supporting framing. It then automatically incorporates any necessary adjustments to the nominal unit shear values from the Special Design Provisions for Wind and Seismic on the basis of sheathing type, aspect ratio, and the species of the supporting framing. The result is a required fastener spacing for each panel in the wall.

Chords are evaluated for tension and compression due to load combinations that include either wind or seismic. The moment in a given shear panel is assumed to be coupled out at the location of the chords, resulting in tension and compression forces. Any applied vertical loads that may be present are combined with the wind or seismic chord forces, and the resulting loads on the chords are evaluated as per the requirements of the NDS.

Footings (if defined) are evaluated for soil bearing pressure, overturning, and one-way shear and flexure of the cantilevered end projection.

There are certain items that are not explicitly evaluated by this module, and the user should be aware of the following exclusions: bending or shear in the top plate, bearing on the top plate, out of plane design of the wall sheathing or framing, gravity-only loading on

© 2026 ENERCALC, LLC

the common studs or chords, bearing on the bottom plate, design of anchorage hardware or connections.

Workflow Process

The general workflow proceeds as follows:

1. The overall height and length of the shear wall are defined on the General tab, along with selections such as the Seismic Design Category, the Design Method, and the Framing and Chord Species and Grade.
2. The sheathing type, thickness, fastener size and blocking conditions is selected on the Sheathing tab. This tab also allows for the specification of sheathing on the second side of the framing when necessary.
3. The chord size is specified on a level-by-level basis on the Chords tab. This tab also allows for the specification of the bracing assumption to be applied in the compression design of the chords.
4. If openings are present in the wall, they can be defined on the Openings tab.
5. Loads of many types can be defined and applied on the Loads tab.
6. If a continuous footing design is desired, the footing geometry and material properties can be entered on the Footing tab.
7. The Load Combinations tab allows for the definition of the load combinations that will be used for the design.
8. The results for the shear panels, chords, and footing (if designed) can be reviewed in the Results panel in the lower portion of the screen.

General

Description: Enter a free-form description of the current wall design for reference.

Total Wall Length: Enter the overall length of the shear wall in units of feet.

Stud Spacing: Enter the typical stud spacing in inches.

Story Heights: Enter the heights of up to five stories in units of feet. Whenever a value is entered, the input field for the next story height is displayed in case it is needed. The total accumulated height is automatically reported.

Seismic Design Category: Enter the appropriate Seismic Design Category for the wall being designed. This is used to check the selected sheathing type to be sure it is permissible for use in that SDS according to the Special Design Provisions for Wind and Seismic.

Design Method: Select ASD or LRFD to dictate which method will be applied when designing shear panels and chords. Footing design (when requested) is always by LRFD methods.

Framing & Chord Material: Use the icon to access the Wood Reference Design Values database in order to select the species and grade of wood used for the chords and common framing in the wall.

© 2026 ENERCALC, LLC

Sheathing

Select SDPWS Construction Table: Choose the table from which the sheathing will be selected.

Select Main Sheathing: Select a tabular entry to represent the sheathing type, sheathing thickness, fastener size/penetration, and in some cases the blocking condition.

Nominal Shear Capacities: The program displays the nominal shear capacities for seismic design and for wind design directly from the selected table. These are nominal values which still need to be modified for use in design, such as with a phi factor or factor of safety and with any applicable adjustments such as for aspect ratio or specific gravity of the supporting framing.

Table 4.3A Footnote 2 is applicable: (Only visible for some sheathing selections) Use this checkbox to indicate if a specific condition exists as described in detail in the referenced footnote in Table 4.3A of the Special Design Provisions for Wind and Seismic.

Sheathing is Blocked: (Only visible for some sheathing selections) Use this checkbox to indicate if the sheathing is blocked or not. This setting has an effect on some of the allowable aspect ratios.

Sheathing on 2nd Side: Use this checkbox to indicate that there is sheathing of some sort on the other side of the framing.

Use Same as Main Sheathing: A convenience option that automatically sets the Sheathing on 2nd Side to be identical to the sheathing on the Main side.

Note: If sheathing is specified on the second side, and if the sheathing comes from the same Construction Table as the Main Sheathing, then the blocking setting for the 2nd side will automatically be assumed to be the same as the blocking setting for the Main side.

Chords

The Chords tab will automatically display one row of chord definition data for each story that was defined on the General tab.

Chord Member Size: Use the drop-down list box to select the size of the sawn lumber member(s) that will be used at the chord locations for the given level. Note that the program will automatically determine the number of chord members required at each location. So for example, if the wall was generally going to be constructed of 2x4 framing, then this input should be set to "2x4". Once the analysis and design has been completed, the program will report how many 2x4s should be ganged together to safely resist the imposed chord forces at each location.

CF: Size Factor: Enter the appropriate Size Factor for the chord member size, species, and grade being entered.

© 2026 ENERCALC, LLC

Area per Chord: The program reports that cross sectional area of one member of the selected size for reference.

Maximum Chord Stress Ratio: Enter the maximum permissible chord stress ratio. Typically 1.0 using current design methods and load combinations.

Wood Chord Strength Calculation: This setting offers two options for defining the bracing of chord members when the allowable compression stress is calculated:

- The option named *"Treat all chords as fully braced about both axes"* implies that all chords are braced against column buckling. The physical model for this option might be a situation where chord members always occur at "L" or "T" shaped intersections, such that the sheathing prevents buckling in the plane of the shear wall, and the intersecting wall prevents buckling out of the plane of the shear wall. (These conditions are probably not common.)
- The option named *"Assume all chords unbraced out of plane of wall for story height"* implies that the sheathing prevents buckling in the plane of the shear wall, but nothing prohibits column buckling of the chord member out of the plane of the wall. This is the condition for the chords at the end of an isolated straight panel of wall where no perpendicular walls intersect the shear wall at the chord locations.

Openings

The Openings tab allows openings to be defined in the shear wall.

Add: The Add button opens the Add Opening dialog. The dialog automatically numbers the openings that are added, and it collects the geometric information necessary to locate and size the opening. The Add Opening dialog also checks the alignment of defined openings to ensure that all jambs are aligned.

Edit: By selecting an existing opening in the list, the Edit button allows the selected opening to be revised.

Delete: By selecting an existing opening in the list, the Delete button allows the selected opening to be removed.

Renumber: The Renumber button allows existing openings to be renumbered in a logical order by working from left to right and then from bottom to top.

The graphic display offers checkboxes to independently display or hide:
- Openings
- Shear Panel callouts
- Chord graphics and callouts
- Panel and opening dimensions
- All loads

Note: This module is predicated on the design assumptions of the Individual Full-Height Wall Segment Shear Walls method. This method disregards the shear resistance of

© 2026 ENERCALC, LLC

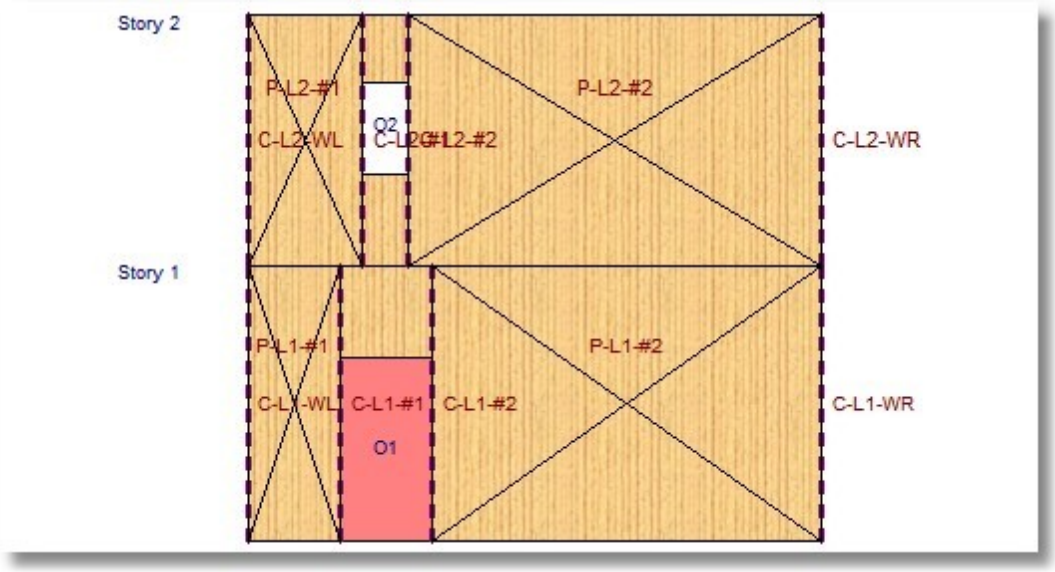
portions of shear walls above and below openings. This leads to two **important** considerations regarding the use of openings:

1. In multi-story models, any opening in one story must not ***partially*** overlap an opening in a different story. If they overlap at all, they must match ***exactly***. Envision vertical lines at the jambs of an opening, and project them to the top and bottom of the entire wall. Those jamb reference lines must not cross any other opening. If anything, they can only align with other jambs. The Add Opening dialog will warn if this condition is being violated by having partially-overlapping openings in any story. This may require enlarging some openings so that they respect the jamb locations of adjacent openings, or it may require applying the module only to the solid portions of the wall that remain after the openings have been omitted.

2. The presence of an opening within a given story, creates a zone that is ineffective at resisting shear. The ineffective zone is defined by the width of the opening and it extends for the full height of that story. This means that it is not necessary to model stacked openings *within a given story*. In fact, doings so will cause an error.

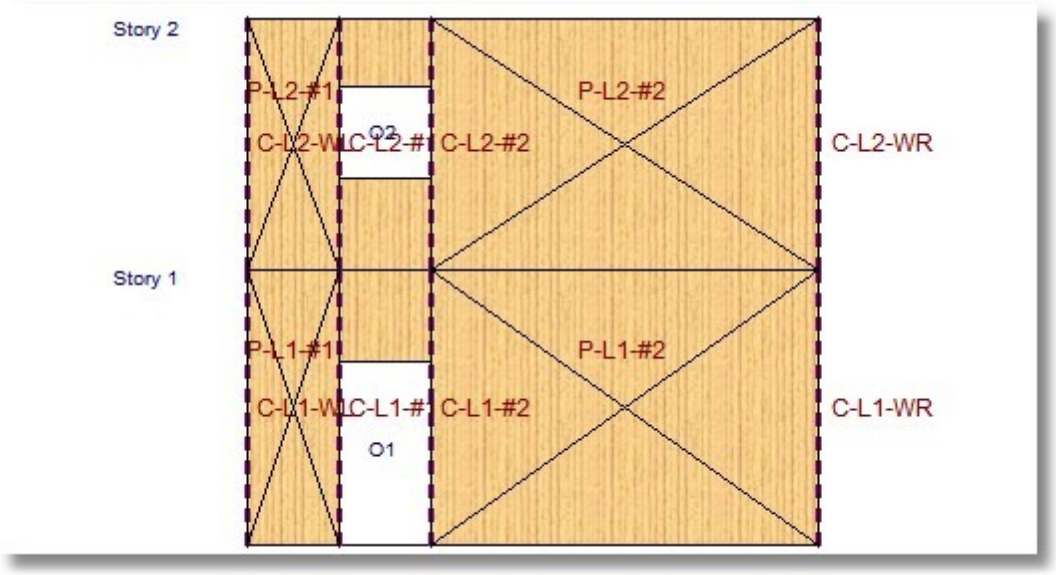
See the following diagrams for some acceptable and unacceptable opening geometries:

© 2026 ENERCALC, LLC



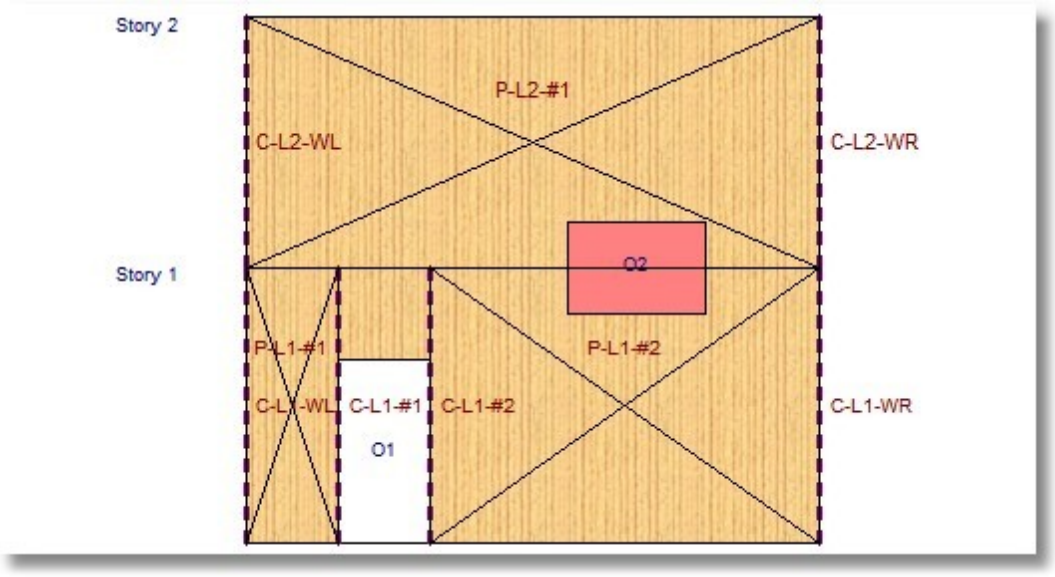
NG - Jambs of openings do NOT align

Here is how to fix it:



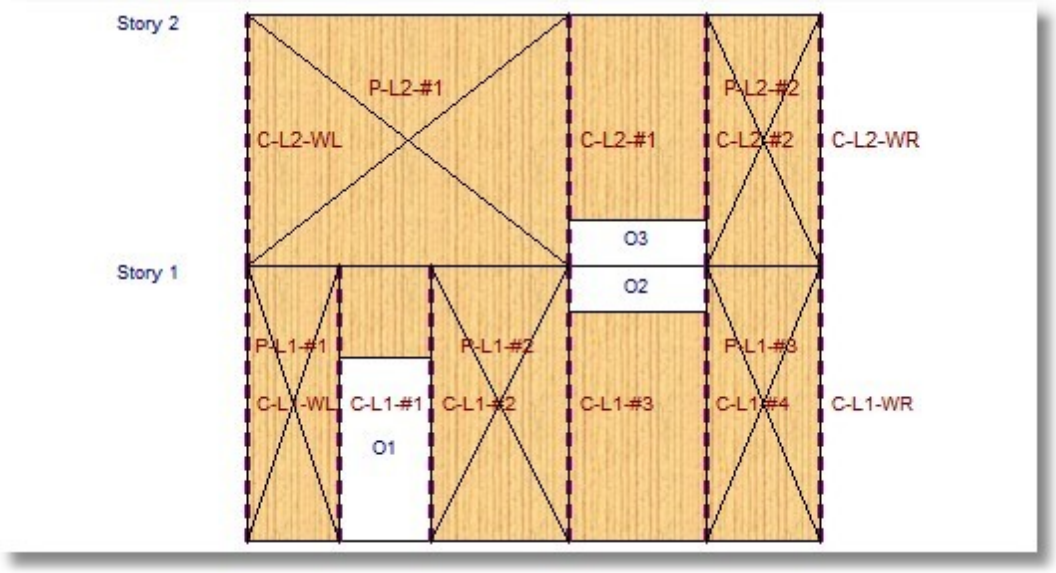
OK - Jambs of openings align properly

© 2026 ENERCALC, LLC



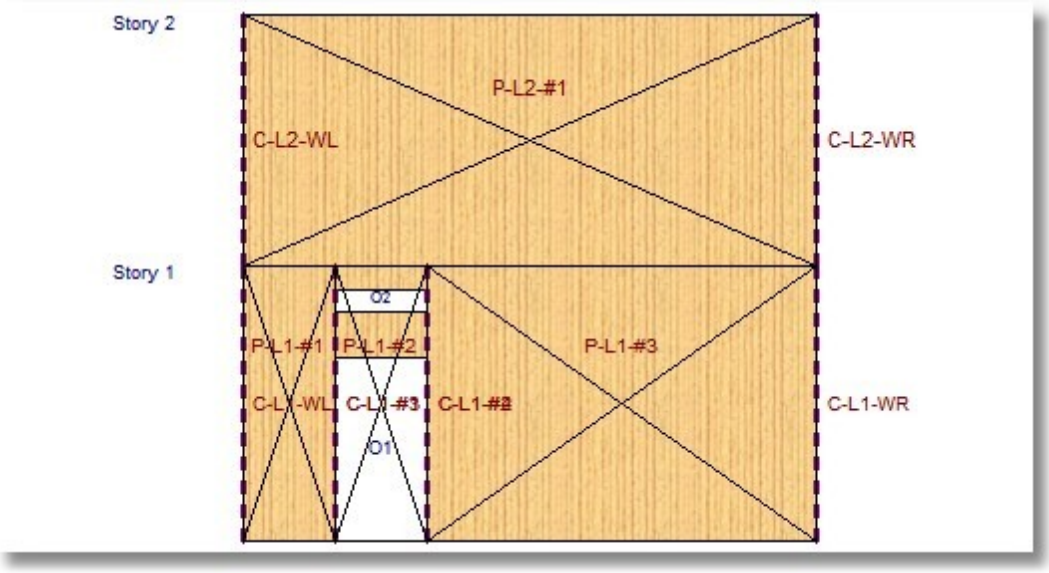
NG - Opening crosses top of story

Here is how to fix it:



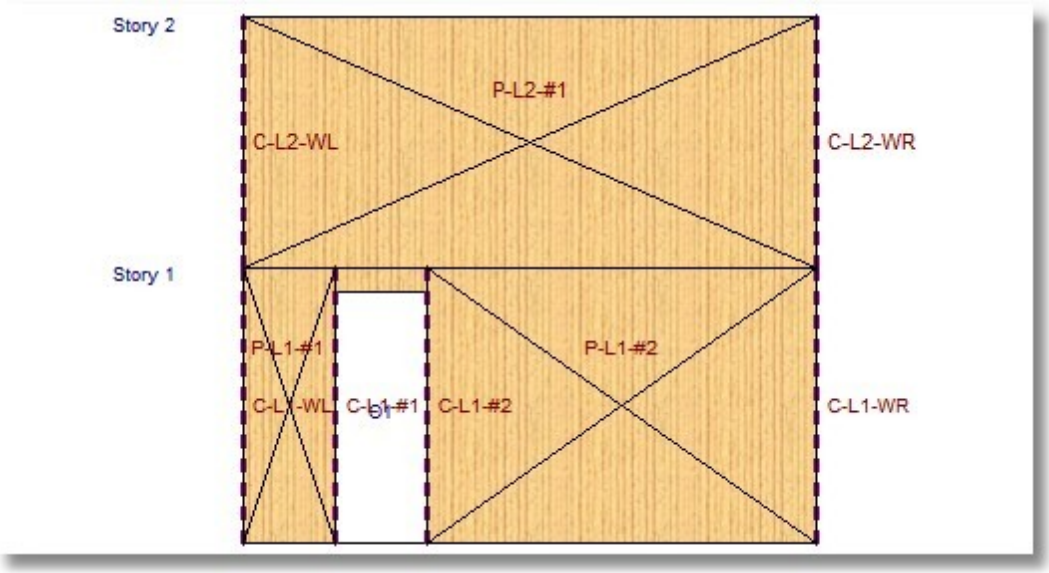
OK - Large opening modeled as 2 separate openings

© 2026 ENERCALC, LLC



NG - Stacked openings within a story

Here is how to fix it:



OK - One opening accomplishes the task

Note: When using the self-weight functions for Dead Load and for seismic load, the program does not deduct any weight for openings. In other words, the program assumes that the unit weight of the window, door, louver, etc. in the opening, is approximately the same as the unit weight of the wood-framed construction it is replacing.

Loads

The Loads tab provides several sub-tabs for the convenient definition of many different types of loads that could potentially act on a shear wall system.

© 2026 ENERCALC, LLC

General
The Wall Self Weight category offers inputs for the following:

Consider Wall Weight: This category provides options to automatically consider the wall self-weight:
- As vertical Dead load - a way to request that the program use the unit weight described above to calculate a dead load
- As Seismic load - a way to request that the program use the unit weight described above to calculate a lateral seismic load using the value of Cs described below

Weight: (Only displayed when one of the options above has been selected.) Enter the unit weight of shear wall construction in psf.

Cs: Seismic Response Coefficient: (Only displayed when the "As Seismic load" option above has been selected.) Collects the value of Cs by which the unit weight would be multiplied to determine the seismic load due to the self-weight of the wall if that option is selected.

Soil over Footing: category is only displayed if the "Specify a Footing" option is selected on the Footing tab. This category offers inputs for the following:

Soil depth over footing: Enter the depth of soil above the top of the footing in units of inches. This value is used to determine loading on concrete footings when one is designed. This value will affect the reinforced concrete design, the soil bearing pressure, and the overturning ratio.

Soil density: Enter the density of soil above the top of the footing in units of pcf. This value is used to determine loading on concrete footings when one is designed. This value will affect the reinforced concrete design, the soil bearing pressure, and the overturning ratio. If no footing design is requested, this value has no influence on the remaining calculations.

Added Overburden Load over Footing: category is only displayed if the "Specify a Footing" option is selected on the Footing tab. This category allows the user to specify superimposed load on the top of the footing for all of the common load cases. This value will affect the reinforced concrete design, the soil bearing pressure, and the overturning ratio.

Vertical Point

Add: The Add button opens the Add Point Load dialog. The dialog allows vertical point loads of all load cases to be defined and located with respect to the lower left corner of the wall. Note: Positive magnitudes are assumed to act downward.

Edit: By selecting an existing load in the list, the Edit button allows the selected load to be revised.

© 2026 ENERCALC, LLC

Delete: By selecting an existing load in the list, the Delete button allows the load opening to be removed.

Vertical Uniform

Add: The Add button opens the Add Uniform Load dialog. The dialog allows vertical uniform loads of all load cases to be defined and located with respect to the lower left corner of the wall. Note: Uniform loads can be specified as partial-length loads by specifying the start and end locations of the load with respect to the left edge of the wall. Positive magnitudes are assumed to act downward.

Edit: By selecting an existing load in the list, the Edit button allows the selected load to be revised.

Delete: By selecting an existing load in the list, the Delete button allows the load opening to be removed.

Lateral Point

Add: The Add button opens the Add Point Load dialog. The dialog allows lateral point loads of all load cases to be defined and located with respect to the lower edge of the wall. Note: Positive magnitudes are assumed to act to the right.

Edit: By selecting an existing load in the list, the Edit button allows the selected load to be revised.

Delete: By selecting an existing load in the list, the Delete button allows the load opening to be removed.

Lateral Uniform

Add: The Add button opens the Add Uniform Load dialog. The dialog allows lateral uniform loads of all load cases to be defined and located with respect to the lower edge of the wall. Note: Uniform loads can be specified as partial-height loads by specifying the start and end locations of the load with respect to the bottom edge of the wall. Positive magnitudes are assumed to act to the right.

Edit: By selecting an existing load in the list, the Edit button allows the selected load to be revised.

Delete: By selecting an existing load in the list, the Delete button allows the load opening to be removed.

The graphic display offers checkboxes to independently display or hide:

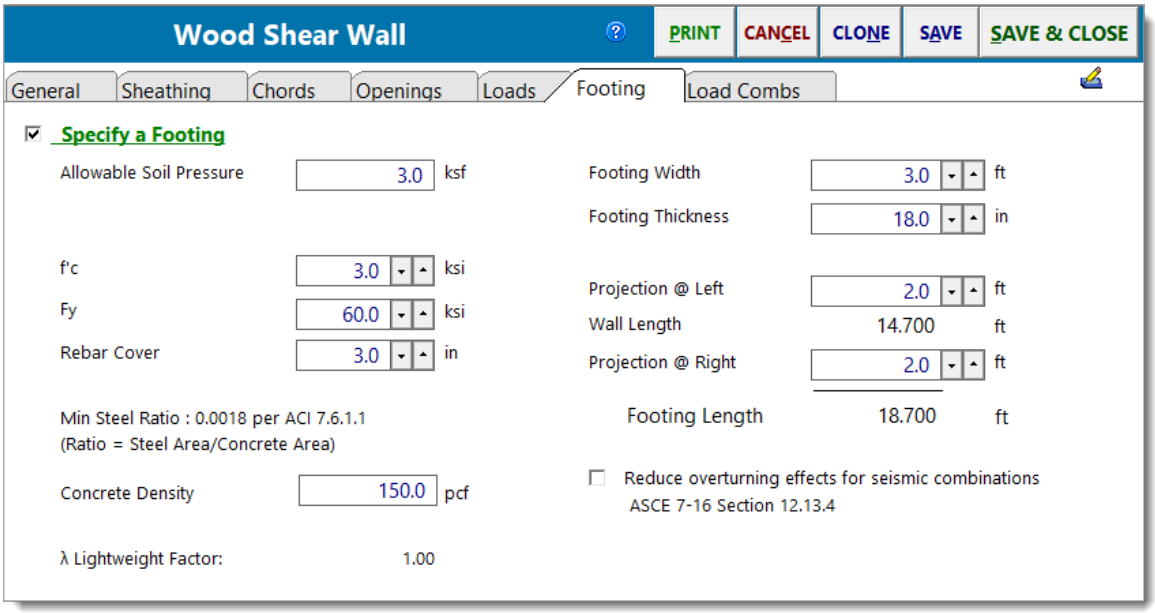
- Openings
- Shear Panel callouts
- Chord graphics and callouts
- Panel and opening dimensions
- All loads - otherwise only the loads associated with the selected tab are displayed

© 2026 ENERCALC, LLC

Footing

Optional tab to design a footing that behaves as a continuous grade beam that cantilevers past the end of the wall.

Specify geometry, allowable soil pressure, and material properties to arrive at checks for soil pressure and footing flexure.



Specify a Footing: If this checkbox is selected, then the input fields for defining a footing are displayed. Note: This checkbox also causes some footing-related input to be shown or hidden as appropriate on the Loads tab.

Allowable Soil Pressure: Specify the allowable soil bearing pressure in units of ksf.

f'c: Specify the compressive stress of concrete in units of ksi.

Fy: Specify the yield stress of rebar in units of ksi.

Rebar Cover: Specify the cover over rebar in units of inches. The program will use this value and a 1/2" allowance for the rebar size to calculate the effective depth of the concrete section.

> Note: Rebar is assumed to exist only at the bottom of the footing to resist tensile forces from the vertical loads and increased pressure due to overturning forces. Tension in the top of the footing in cases where no upward soil pressure exists and the footing weight creates a downward net force IS IGNORED.

Min Steel Reinforcing: The minimum flexural reinforcement ratio, per ACI 7.6.1.1, is 0.0018 times the gross area of the concrete (Width x Thickness). Commentary R7.6.1.1 also notes that minimum flexural reinforcement should be placed as close as possible to

© 2026 ENERCALC, LLC

the tension face of the concrete, unlike shrinkage and temperature reinforcement which may be distributed between the top and bottom faces of the footing.

Concrete Density: Specify the density of concrete in units of pcf.

Footing Width: Specify the width of the footing in units of ft. This is the dimension perpendicular to the wall length.

Footing Thickness: Enter the thickness of the footing in units of inches.

Projection @ Left: Enter the projection of the footing beyond the left end of the wall in units of ft.

Wall Length: The program reports the shear wall length in units of ft for reference.

Projection @ Right: Enter the projection of the footing beyond the right end of the wall in units of ft.

Footing Length: The program reports the footing length in units of ft for reference.

Reduce overturning effects for seismic combinations per ASCE 7-16 Section 12.13.4: Applies the referenced reduction in overturning force for seismic load combinations when selected.

Load Combinations

The Load Combinations tab indicates the currently selected load combination set. The load combination set can be changed by clicking the Select button.

The Load Combinations tab has up to four sub-tabs:

- LRFD Load Combinations or ASD Load Combinations (depending upon the selected Design Method),
- Stability Combinations (only shown if a footing is designed)
- Soil Pressure Combinations (only shown if a footing is designed), and
- Footing Design Combinations (only shown if a footing is designed).

LRFD Load Combinations or ASD Load Combinations
This tab offers strength-level or service level combinations from the selected load combination set. These load combinations will be used for wood design.

A selected checkbox in the "Run" column indicates that the associated load combination will be considered. The "Run" button offers quick convenience options for changing the selection status of many load combinations at one time.

The C_D column is only visible when the ASD Design Method is selected. It indicates the value of the Load Duration Factor for each load combination. Click the C_D button to

© 2026 ENERCALC, LLC

quickly set the values of C_D for all load combinations based on the load case with the shortest duration.

The Lambda column is only visible when the LRFD Design Method is selected. It indicates the value of the Time Effect Factor for each load combination. Click the Lambda button to quickly set the values of Lambda for all load combinations based on Table N3.

Auto Reverse Wind: Select this button to instruct the module to also consider the algebraic negative of the defined Wind loads. (This may be useful for quickly creating a load combinations that reverse the direction of application of applied wind loads by creating a "sister" load combination that uses the negative version of the wind component for each load combo that normally incorporates +W.)

Auto Reverse Seismic: Select this button to instruct the module to also consider the algebraic negative of the defined Seismic loads. (This may be useful for quickly creating a load combinations that reverse the direction of application of applied seismic loads by creating a "sister" load combination that uses the negative version of the seismic component for each load combo that normally incorporates +E.)

Stability Combinations (only shown if a footing is designed)
This tab offers service level combinations from the selected load combination set. These load combinations will be used for evaluating sliding and overturning of a footing.

Soil Pressure Combinations (only shown if a footing is designed)
This tab offers service level combinations from the selected load combination set. These load combinations will be used for evaluating soil bearing pressure.

Footing Design Combinations (only shown if a footing is designed)
This tab offers strength-level combinations from the selected load combination set. These load combinations will be used for concrete footing design.

The right-hand portion of the screen offers the following options for reviewing results:

Shear Panel Design

Summary
On a panel-by-panel, level-by-level basis, the Summary tab presents the following information:

Max Shear: Reports the maximum shear force and the load combination associated with that maximum shear force.

# of Sides: Indicates the number of sides to which sheathing has been applied. Will either be 1 or 2.

Shear Summary & Attachment: Reports the actual unit shear, the allowable unit shear, the design status, and the required attachment pattern.

© 2026 ENERCALC, LLC

Height/Width Ratio: Reports the actual height-to-width ratio, the allowable height-to-width ratio for Side 1 (Main Sheathing), the allowable height-to-width ratio for Side 2 (2nd Side Sheathing), and notes regarding the status of the height-to-width ratio and any necessary adjustments.

Detail by Panel ID
On a panel-by-panel, load combination-by-load combination basis, the Detail by Panel ID tab presents the following information:

Panel Data: Reports the story in which the panel exists, the distance from the left edge of the overall wall to the left edge of the panel, the width of the panel, the distance from the bottom edge of the overall wall to the bottom edge of the panel, the height of the panel, and the height-to-width ratio of the panel.

Shear Forces: Reports the "Tributary Width" = (Panel Width/∑Panel Widths)*Overall Wall Length, Tributary % = Panel Width/∑Panel Widths, Shear tributary to each panel, panel width, and the maximum unit shear.

Capacity Factors: Reports phi for LRFD designs or the reciprocal of the Factor of Safety for ASD designs, a capacity adjustment factor due to aspect ratio of the sheathing on the Main Sheathing side of the wall, a capacity adjustment factor due to aspect ratio of the sheathing on the 2nd Side of the wall, and a capacity adjustment factor due to the specific gravity of the framing.

Adjusted Allowable Shear: Reports the allowable unit shears for all relevant fastener spacings incorporating any applicable capacity adjustment factors. In situations where two sides of sheathing are being considered, this table will correctly determine the capacity considering both sides of sheathing by applying the rules of 4.3.3.3 and 4.3.3.3.2.

Panel Moment: Reports the moment in the panel.

Chord ID: Reports the identifying labels for the chords at the left and at the right end of each panel.

Detail by Load Combination
On a load combination-by-load combination, panel-by-panel basis, the Detail by Load Combination tab presents the following information:

"Tributary Width" = (Panel Width/∑Panel Widths)*Overall Wall Length

Tributary % = Panel Width/∑Panel Widths

Shear Force = shear tributary to each panel

Chord Design

Chord Data
For each chord in the wall, the Chord Data tab presents the following information:

© 2026 ENERCALC, LLC

Location: Reports the Level in which the chord occurs and the distance from the left edge of the overall wall to the chord.

Chord Design: Reports the chord force associated with the controlling design ratio, the load combination responsible for producing the controlling design ratio, then number of chord members required to resist the applied load, the size of the chord member indicated by the user, the governing design ratio, the governing design consideration (tension or compression), and the design status.

Chord Compression Stress: Reports the maximum chord compression force, the load combination responsible for producing the maximum chord compression force, the maximum compressive stress, and the allowable compressive stress.

Chord Tension Stress: Reports the maximum chord tension force, the load combination responsible for producing the maximum chord tension force, the maximum tensile stress, and the allowable tensile stress.

Chord Forces by Chord ID
For each chord in the wall, on a load combination-by-load combination basis the Chord Forces by Chord ID tab presents the following information:

Story: The story in which the chord occurs.

ID: The identification label assigned to the chord.

Axial: Reports the load in the chord due to the sum of applied vertical loads, the load in the chord that can be added or subtracted due to the overturning moment in the panel, the force with the largest (tendency for) tension, and the force with the largest (tendency for) compression.

Chord Location: Reports the distance from the left edge of the overall shear wall to the chord location, the distance from the bottom edge of the overall shear wall to the bottom of the chord, and the distance from the bottom edge of the overall shear wall to the top of the chord.

Footing & Stability

Footing Design Combination Pressures: Reports the maximum factored soil pressures at the left and right sides of the footing, along with the load combination responsible for producing each of those pressures. These are used in the reinforced concrete design calculations for the footing.

Footing One-Way Shear Check: Reports the calculated one-way shear at both ends of the footing.

For footings designed per ACI 318-14 and earlier, and ACI 318-25 and later:
The nominal shear strength follows the one-way shear provisions of ACI 318-14 §22.5.5.1 / ACI 318-25 §13.2.6.2:
$\varphi v_n = \varphi * 2 * \lambda * \sqrt{(f'_c)}$, where $\varphi = 0.75$.

© 2026 ENERCALC, LLC

For footings designed per ACI 318-19:

The nominal shear strength follows the one-way shear provisions of §22.5 and Table 22.5.5.1 (referenced by §7.5.3.1):

$\varphi v_n = \varphi * 8 * \lambda_s * \lambda * (\rho_w)^{1/3} * \sqrt{(f'_c)}$, where $\varphi = 0.75$.

Note: The one-way shear capacity assumes no transverse reinforcement in the footing. Therefore, $A_v < A_{v,min}$, and equations (a) and (b) from Table 22.5.5.1 are not applicable.

It is also acceptable to neglect the size effect factor (λ_s) for one-way shallow foundations per §13.2.6.3, so $\lambda_s = 1.0$.

This value is also subject to additional limits per §22.5.5.1.1:

Lower bound: $\varphi * \lambda * \sqrt{(f'_c)}$

Upper bound: $\varphi * 5 * \lambda * \sqrt{(f'_c)}$

Footing Bending Design: Reports the results of the flexural analysis and design including recommended reinforcing options to satisfy the required area of steel.

Soil Pressure Combination Pressures: Reports the maximum unfactored soil pressures at the left and right sides of the footing, along with the load combination responsible for producing each of those pressures. Unfactored soil pressures are compared to the allowable soil bearing pressure provided by the user.

Overturning Stability: Reports the overturning analysis performed about each end of the footing including the overturning moment, the resisting moment, the stability ratio, and the governing load combination.

Sketch

The sketch tab provides a convenient graphical view of the shear wall with options to selectively view or hide the following:

- Openings
- Shear Panel callouts
- Chord graphics and callouts
- Panel and opening dimensions
- All loads

© 2026 ENERCALC, LLC

Aspect Ratio Adjustment Options

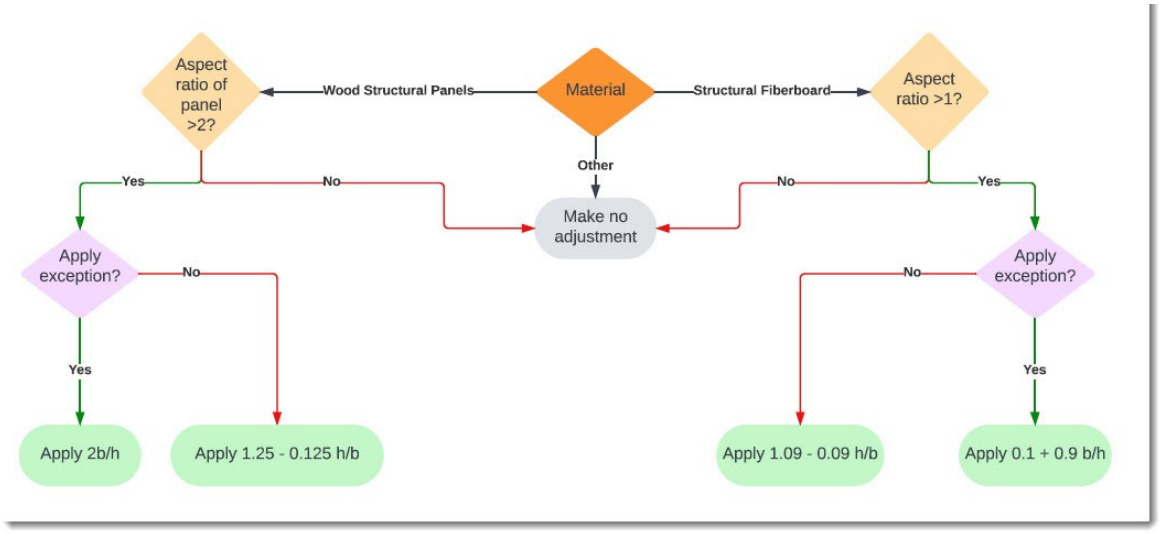
NDS Aspect Ratio Adjustment Options

NDS is clear that Wood Structural Panels must have an aspect ratio adjustment factor applied when the aspect ratio (h/b) of the effective panel is greater than 2.

Structural Fiberboard panels must have an aspect ratio adjustment factor applied when the aspect ratio (h/b) of the effective panel is greater than 1.

NDS also provides an exception to the normal adjustment formulas that allows the designer to distribute shear to multiple panels in a wall in proportion to the individual shear capacities of the full height segments (as opposed to proportioning in terms of stiffness).

As of 7/24/2023, the Wood Shear Wall module does not have the ability to consider stiffness. So in walls with openings, it may make sense to use the exception to avoid having to do additional hand calculations. In walls with no openings, it may be permissible to use the standard aspect ratio adjustment (more liberal).



13.8 Earth Retention

As of Build 20, ENERCALC SEL includes all retaining wall modules that were in Build 11 of RetainPro, along with the Retaining Wall Equation editor, making short work of your retaining wall analysis and design tasks.

© 2026 ENERCALC, LLC

13.8.1 All Retaining Walls

The following topics generally apply to all wall types except where noted specifically:

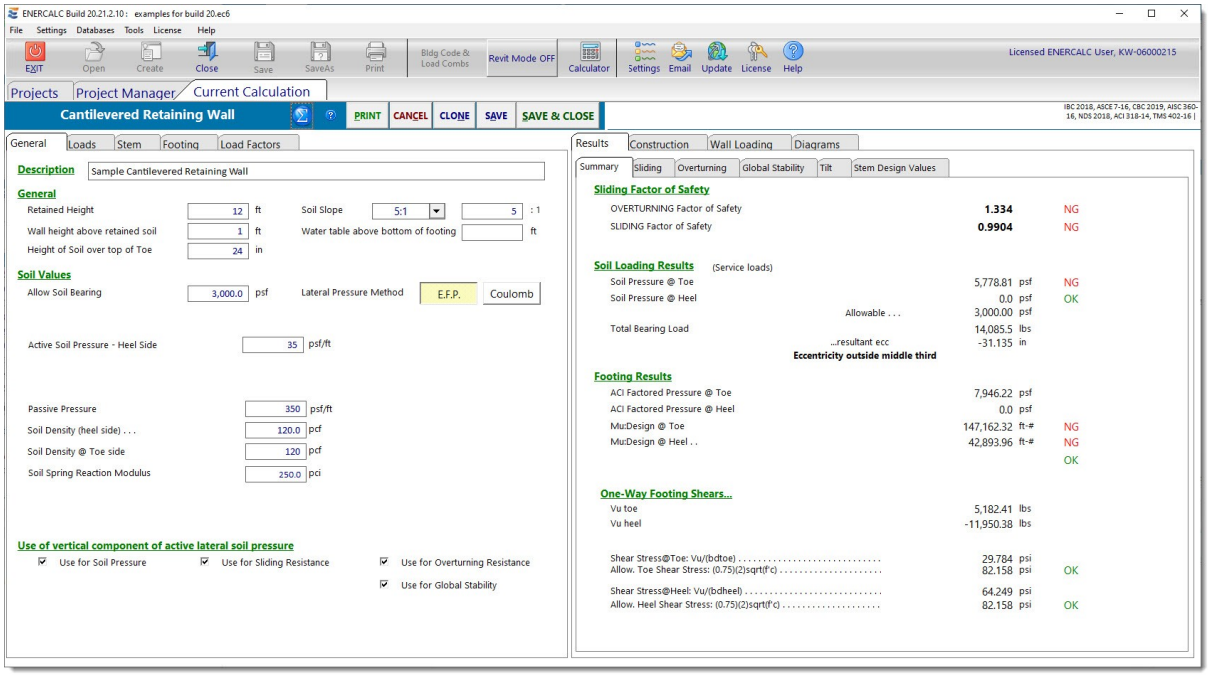
- General Tab 1089
  - General Data 1089
  - Soil Values 1091
  - Use of Vertical Component (Not in Restrained Retaining Wall module) 1095
- Loads Tab 1096
  - Loads 1096
  - Seismic Loads 1101
- Stem Tab 1105
  - Stem Tab for Cantilevered Retaining Wall 1106
    - Summary Section of Stem Tab 1111
  - Stem Tab for Tapered Stem Retaining Wall 1118
  - Stem Tab for Gravity Retaining Wall 1121
  - Stem Tab for Restrained Retaining Wall 1123
- Footing Tab 1131
  - Footing Design Sub-tab 1131
  - Key Design & Sliding Options 1135
  - Pier Design 1139
- Load Factors 1144
- Results Tabs 1145
  - Summary 1146
  - Sliding > Resisting Forces 1149
  - Sliding > Sliding Forces 1150
  - Overturning > Resisting Moments 1151

© 2026 ENERCALC, LLC

  - Overturning > Overturning Moments 1152
  - Global Stability 1153
  - Wall Tilt 1153
  - Stem Design Values 1154
- Stability Tab (Restrained Walls only) 1155
- Construction Tab 1156
- Wall Loading Tab 1157
- Methodology / Analysis & Design Assumptions 1158

13.8.1.1 General Tab

The General tab collects basic wall geometry, soil values, and certain design assumptions.



General Data

Retained Height:

This is the height of retained earth measured from top of footing to the top of soil behind the stem (over the heel). When the backfill is sloped, the soil will slope away and upwards from this height.

The actual retained height used for overturning and soil pressure calculations will be the retained height projected at the vertical plane of the back of the heel, but for stem moments, no such increase will be made.

After each entry you can press the tab key to advance to the next entry, or use your mouse to position the cursor in the next input field.

© 2026 ENERCALC, LLC

Wall Height above Retained Soil:

Use this entry to specify if the wall extends above the retained height. This entry is typically used to define a "screen wall" projection. This extension can be used as a weightless "Fence" or a concrete or masonry stem section without any soil retained behind it. You can enter wind load on this projection using the entry "Wind on Stem above soil" on the "Loads" tab. We'll handle the fence when we get to the STEM design screen. TOTAL HEIGHT OF WALL = “RETAINED HEIGHT” + WALL HEIGHT ABOVE RETAINED SOIL”.

Height of Soil over Toe:

Measured from top of footing to top of soil on toe side, this may vary from a few inches to a few feet depending upon site conditions. (Note that it is input in inches.) It is used to calculate passive soil resistance (but its effective depth can be modified by the "Ht. to Neglect" entry on the Footing > Key Dimensions & Sliding tab). This depth of soil is also used to calculate a resisting moment, and reduce net lateral sliding force. You can negate the latter effects on the Options screen.

Top Lateral Restraint Height (For the Restrained Retaining Wall module only):

Define the height from top of footing to the elevation of the lateral support.

Soil Slope (Not in Restrained Retaining Wall module):

You may enter any backfill slope behind the wall. Use the drop-down menu or type the slope as a ratio in the form of Horiz/Vert. The soil must be level or slope upward. Negative backfill slopes (grade sloping downward, away from the wall) are not allowed.

The program will use this slope to 1) include the weight of a triangular wedge of soil over the heel as vertical load, and 2) compute overturning based upon an assumed vertical plane at the back face of the footing extending from the bottom of the footing to ground surface – a steeper slope will result in a higher overturning moment. The program will not accept a backfill slope steeper than the angle of internal friction.

When the EFP method is used, the program will NOT change the EFP based on soil slope. All it does with the slope is:

- calculate the retained height at the back of the heel, which might be greater because of the sloped soil, and
- add a surcharge due to the weight of the triangular prism of soil on top.

When the Coulomb or Rankine methods are used, the final calculated pressures do include the effect of the slope on those equations.

© 2026 ENERCALC, LLC

Water Table Height over Heel (Not in Restrained Retaining Wall module):

If a portion of the retained height is below a water table, the active pressure of the saturated soil will increase below that level. This additional pressure for the saturated soil is equal to the pressure of water, plus the submerged weight of the soil (its saturated weight - 62.4), plus the surcharge of the soil above the water table. The program does not collect a saturated weight of soil, so instead it conservatively approximates the buoyant or submerged weight of a soil as 62% of its dry unit weight.

If you want to design for a water table condition, enter the maximum height from ***bottom*** of footing to water table level. The program will then compute the added pressures for saturated soil on the heel side of the footing, including buoyancy effect, to calculate increased moments and shears on the stem, and overturning. Don't enter a height more than the retained height, and keep in mind that this feature automatically assumes that the liquid is water. If the water table is near the top of the retained height, it may be advisable to use the saturated soil density and active pressure for the full retained height instead of specifying a water table height.

Soil Values

Allowable Soil Bearing:

The maximum allowable soil bearing pressure for static conditions. Using the spin buttons you can vary the value in increments. Usual values for this vary from 1,000 psf to 4,000 psf or more.

Lateral Pressure Method:

Here you can choose between E.F.P., Rankine or Coulomb formula. EFP refers to "Equivalent Fluid Pressure," where you can enter a lateral soil pressure in psf per foot of depth. "Coulomb" or "Rankine" instructs the program to use the respective method to calculate active and passive soil pressures using an entered angle of internal friction for the soil.

© 2026 ENERCALC, LLC

When the EFP Method is selected:

Active Soil Pressure - Heel Side:

Enter the equivalent fluid pressure (EFP) for the soil being retained that acts to overturn and slide the wall toward the toe side. This pressure acts on the stem for stem section calculations, and on the total footing+wall+slope height for overturning, sliding, and soil pressure calculations.

Commonly used values, assuming an angle of internal friction of 34°, are 30 pcf for a level backfill; 35 pcf for a 4:1 slope; 38 pcf for a 3:1 slope; 43 pcf for a 2:1 slope; and 55 pcf for a 1.5:1 slope. These values are usually provided by the geotechnical engineer.

When the EFP method is used the value entered is the horizontal component of the active earth pressure, commonly called the lateral earth pressure. Because active pressure is always due to an active soil wedge there are horizontal and vertical components. Using the specified horizontal component and the soil density, the program iterates for a value of an effective soil friction angle (“Phi”, the angle of internal friction) using the Coulomb equation. Once Phi is known, the program can calculate a vertical component of the active pressure and provide options to have this vertical component applied at the plane of retained earth, which is always considered to be at the rear of the heel. The user can choose to apply this force for overturning resistance, sliding resistance, and/or for soil pressure calculations, by checking the boxes in the category named "Use of vertical component of active lateral soil pressure".

Passive Pressure:

This is the resistance of the soil in front of the wall and footing to being pushed against to resist sliding. Its value is in psf per foot of depth (pcf). This value is usually obtained from the geotechnical engineer. Its value usually varies from 100 pcf to about 350 pcf.

Soil Density (heel side):

Enter the soil density for all earth (or water if applicable) above the heel of the footing. This weight is used to calculate overturning resistance forces and soil pressures using the weight of the soil block over the projecting heel of the footing. When surcharges are applied over the soil, the surcharges are transformed to equivalent uniform lateral loads acting on the wall by the ratio force = (Surcharge/ Density)*Lateral Load. Input this value in lbs. per cubic foot. Usual values are 110 pcf to 120 pcf. More if saturated soil. Water is usually assumed to be 64 pcf.

Soil Density (toe side):

Enter the soil density on the toe side, which may be different than the heel side. When surcharges are applied over the soil on the toe side, the surcharge is transformed to equivalent uniform lateral loads acting on the wall by the ratio force = (Surcharge/ Density) *Lateral Load. Input this value in lbs. per cubic foot. Usual values are 110 pcf to 120 pcf.

© 2026 ENERCALC, LLC

© 2026 ENERCALC, LLC

When the Coulomb or Rankine Method is selected:

Soil Friction Angle:

This value is entered in degrees and is the angle of internal friction of the soil. This value is usually provided by a geotechnical engineer from soils tests, but can also be found in reference books or building codes for various typical soil classifications. This value is used along with Soil Density within the standard equations to determine "Ka" and "Kp" multipliers of density to give active and passive soil pressure values.

Active Pressure (or At-Rest Pressure for Restrained Walls):

This value will be computed using the respective formulas for Coulomb or Rankine method. This represents the lateral earth pressure acting to slide and overturn the wall toward the toe side. The result will be presented in units of psf/ft. This pressure acts on the stem for stem section calculations, and on the total footing+wall+slope height for overturning, sliding, and soil pressure calculations.

When the retained soil is sloped, the Rankine method determines a vertical component of the lateral earth pressure over the heel. It is applied vertically downward in the plane of the back of the footing. You can choose to apply this force for overturning resistance, sliding resistance, and/or for soil pressure calculations, by checking the boxes on the General tab.

There is always a vertical component when the Coulomb active pressure method is selected. And again you can choose to apply this force for overturning resistance, sliding resistance, and/or for soil pressure calculations, by checking the boxes on the General tab.

Passive Soil Pressure:

This value will also be computed using the respective formulas based on the selection of Coulomb or Rankine active pressure. This is the resistance of the soil in front of the wall to being pushed against to resist sliding. Its value is in psf per foot of depth (pcf). Common values usually vary from 100 pcf to about 350 pcf.

Soil Density (heel side):

Enter the soil density for all earth (or water if applicable) above the heel of the footing. This weight is used to calculate overturning resistance forces and soil pressures using the weight of the soil block over the projecting heel of the footing. When surcharges are applied over the soil, the surcharges are transformed to equivalent uniform lateral loads acting on the wall by the ratio force = (Surcharge/ Density)*Lateral Load. Input this value in lbs. per cubic foot. Usual values are 110 pcf to 120 pcf. More if saturated soil. Water is usually assumed to be 64 pcf.

Soil Density (toe side):

© 2026 ENERCALC, LLC

Enter the soil density on the toe side, which may be different than the heel side. When surcharges are applied over the soil on the toe side, the surcharge is transformed to equivalent uniform lateral loads acting on the wall by the ratio force = (Surcharge/ Density) *Lateral Load. Input this value in lbs. per cubic foot. Usual values are 110 pcf to 120 pcf.

Soil Spring Reaction Modulus

Enter the spring constant to be used to determine the tilt of the wall.

Use of Vertical Component (Not in Restrained Retaining Wall module)

Use of vertical component of active lateral soil pressure

This category offers options for considering the vertical component of active lateral soil pressure:

- Use for Soil Pressure & Footing Design
- Use for Sliding Resistance
- Use for Overturning Resistance
- Use for Global Stability Check

When used, the vertical component of the lateral pressure is applied as:

When EFP is used, the program will backsolve the Rankine equation to find the equivalent internal friction angle, phi. And the vertical component will be applied at the end of the heel.

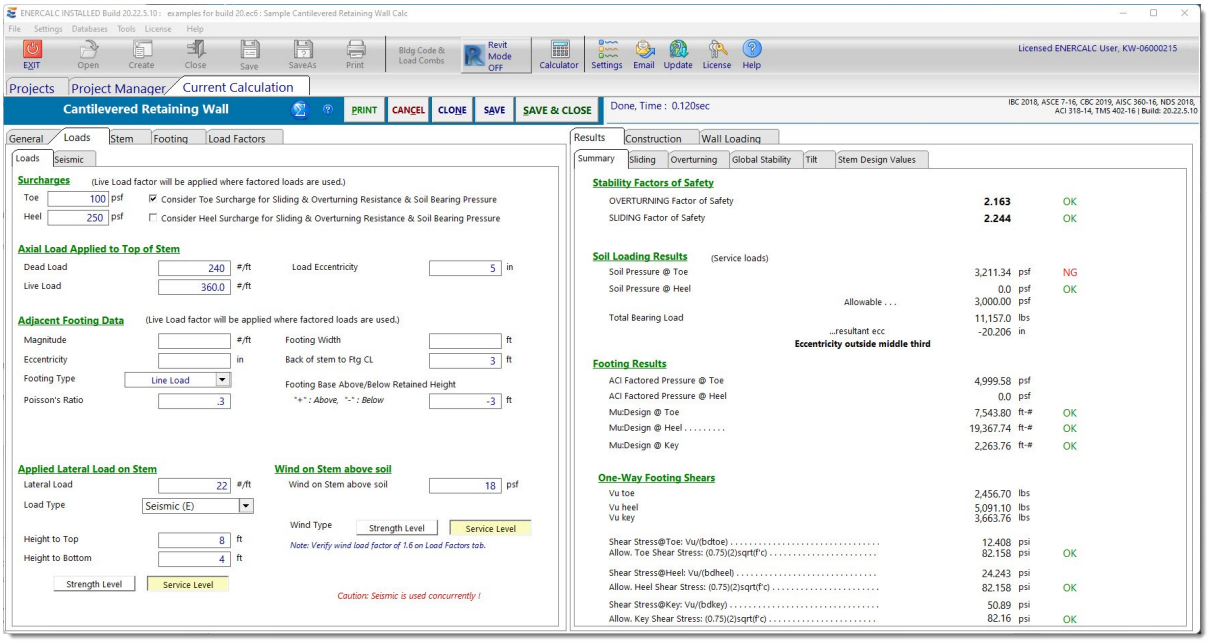
When the Coulomb method is used, the phi angle is specified by the user. And the vertical component will be applied at the back of the wall.

When the Rankine method is used, the phi angle is specified by the user. And the vertical component will be applied at the end of the heel.

© 2026 ENERCALC, LLC

13.8.1.2 Loads Tab

The Loads tab collects the data required to define the applicable vertical and lateral loads, and the seismic design criteria, if applicable.



Loads

Surcharges

This surcharge is treated as additional soil weight – if the surcharge is 240 psf and the density is 120 pcf, then the program uses two feet of additional soil. Similarly, if 50 psf is added for the weight of a slab over the footing, this will be equivalent to 0.41 feet of soil (50 / 120). This surcharge will affect sliding resistance and passive pressure at the toe. Consider this if modeling a point load toe surcharge.

When a heel surcharge is defined, it is considered to be uniformly applied to the top surface of the soil over the heel. It may be entered whether or not the ground surface is sloped. This surcharge is always taken as a vertical force. This surcharge is divided by the soil density and multiplied by the Active Pressure coefficient to create a uniform lateral load applied to the wall. You can choose to use this surcharge to resist sliding and overturning by checking the option box adjacent to the load input field. Typical live load surcharges are 100 psf for light traffic and parking, and 250 psf for highway traffic.

Both the toe surcharge and the heel surcharge have associated checkboxes that can be used to dictate whether the respective surcharges should be considered as resisting sliding and overturning of the wall.

© 2026 ENERCALC, LLC

Axial Load Applied to Top of Stem

These loads are considered uniformly distributed along the length of the wall. They are applied to the top of the topmost stem section. The dead and live loads are used to calculate stem design values and factored soil reaction pressures used for footing design. Only the dead load is used to resist overturning and sliding of the retaining wall. AVOID A HIGH AXIAL LOAD (say over 3 kips plf Total Load) SINCE IT COULD CAUSE A REVERSAL OF BENDING IN THE HEEL.

Since slenderness ratios (h/t) for retaining walls are generally small, usually less than 10, and axial stresses are low, slenderness effects are checked but usually have a small effect.

Consider a point load (such as a beam reaction) applied to the top of a wall. The intensity of that point load will decrease at locations that are more distant from the point of application, because the lateral distribution width will increase as one moves away from the point of application. For this reason, the intensity of the axial load felt at the base of the stem will be significantly less than the intensity immediately beneath the beam bearing. To account for this effect, the magnitude of the axial point load entered should be reduced proportionately (since the input actually represents a uniformly distributed load along the length of the wall). But the top of the wall may need to be checked for localized stress by appended calculations.

The input for axial load applied to the top of the stem allows the load magnitudes to be defined as either Dead Load or Live Load. The load will be factored accordingly. This type of load also allows the specification of an eccentricity value, where the eccentricity is defined with respect to the centerline of the uppermost stem section. Positive values of eccentricity move the load toward the toe, causing bending moments that are additive to those caused by the lateral soil pressure over the heel. Negative eccentricities are accepted in the Restrained Retaining Wall module, where tension is already expected on the toe side. But negative eccentricities are not accepted in the Cantilevered Retaining Wall module.

Adjacent Footing Data

This entry gives you the option of placing a footing (line or square) adjacent and parallel to the back face of the wall, and have its effect on the wall included in both the vertical and horizontal forces on the wall and footing. Refer to the Reference Diagram for locations where input measurements should be taken.

Adjacent Footing Loads will be factored by the Live Load factor for strength design.

For "Line (Strip) Load" the entry is the total load per ft. parallel to the wall (not psf).

If the adjacent footing is specified as "Square Footing" (not line load), the load entered should be the adjacent footing load divided by its dimension parallel to the wall, giving a pounds per lineal foot value, as for a continuous (line) footing.

A Boussinesq analysis is used to calculate the vertical and lateral pressures acting on the stem and footing. The program uses equation (11-20a) in Bowles' *Foundation Analysis and Design*, 5th Edition, McGraw-Hill, page 630.

© 2026 ENERCALC, LLC

When the Boussinesq analysis is used, the program may require additional computing time (hundreds of internal calculations are done after each entry), depending upon the speed of your computer. To avoid this delay (which occurs any time any entry is changed) we suggest you use a vertical load of zero until your data entry is nearly finalized. Then enter the actual footing load and modify your final values.

For adjacent truck or highway loading, it may be preferable to use a heel surcharge (uniform) of 250 psf (or more) instead of treating it as an "adjacent footing."

It is generally not necessary to use this feature if the adjacent footing load is farther from the stem than the retained height, less the depth of the adjacent footing below the retained height, since at this distance it will not have significant effect on the wall.

Footing Width: Width of the adjacent footing measured perpendicular to the wall. This is necessary to create a one-foot long by Width wide area over which the load is applied.

Footing Eccentricity: This entry is provided in case the soil pressure under the adjacent footing is not uniform. Enter the eccentricity of the resultant force under the adjacent footing from the centerline of the adjacent footing. Positive eccentricity is toward the toe, resulting in greater pressure at the side of the adjacent footing closest to the stem. (An eccentricity value of zero means that the adjacent footing load will be considered to act at the center of the adjacent footing.) The program will use the vertical load and eccentricity and create a trapezoidal pressure distribution under the adjacent footing for use with the Boussinesq analysis of vertical and lateral pressures.

Wall to Footing Centerline Distance: This is the distance from the center of the adjacent footing to the back face of the stem at the retained height. The nearest edge of the footing should be at least a foot away from the wall face – otherwise suggest using an equivalent heel surcharge instead. Do not use a horizontal distance greater than the vertical distance from the top of the footing to the bottom of the adjacent footing, since the effect on the wall will be insignificant.

Footing Type: This drop down menu selection allows you to enter either an isolated footing using the "Square Footing" selection, or a continuous footing using the "Line Load" selection.

Footing Base Above/Below Retained Height: Use this entry to locate the bottom of the adjacent footing with respect to the Retained Height. Entering a negative number places the footing below the elevation of the soil measured at the back of the wall. A positive entry would

© 2026 ENERCALC, LLC

typically only be used when the soil is sloped and the footing resides "uphill" from the retained height elevation. To insert a negative number, first type the number, then press the "-" (minus) sign.

Note: If the "Adjacent Footing" is another retaining wall at a higher elevation, the Boussenesq analysis may be used for the vertical load applied to the soil from the adjacent retaining wall footing, however the design must also consider the lateral (sliding) loads from that adjacent wall. This load could be applied as "Added Lateral Load", however this is at the discretion of the designer and is not within the scope of the program. Caution is urged for this condition. See discussion in the companion book: Basics of Retaining Wall Design. ***The designer should be advised that the program does not incorporate any form of global stability analysis.***

Poisson's Ratio: Since the resulting pressures are sensitive to Poisson's Ratio, there is an entry allowing you to specify a ratio from 0.30 to 0.55. This value should be provided by the geotechnical engineer. A value of 0.50 is often assumed.

Applied Lateral Load on Stem

This input allows you to specify an additional uniformly distributed lateral load applied to the stem. This is generally not the preferred method of applying seismic load. Use the Seismic sub-tab instead.

This entry can be useful for a point load, such as due to an impact of a car or similar force. When used in this way, it may be easiest to enter the load as a one-foot high increment, and specify the "Height to Bottom" and "Height to Top" to define a one-foot high strip of application.

This load will be factored by whatever value is specified in the adjacent Load factor input. To apply an additional factor (such as an impact factor), increase the applied load proportionately (e.g. an impact load of 1000 lbs requiring an impact factor of 2.0 would be entered as 2,000 lbs). You may need to do several designs to check multiple load combinations.

Use engineering judgment when applying a point lateral load. The magnitude may be able to be reduced to account for the fact that the load distributes horizontally at levels below the point of application, so its intensity reduces at elevations below the point of application.

Height to Top: Defines the upper extent of the applied lateral load measured from the top of the footing. Do not enter a dimension higher than the top of the wall ("Retained Height" plus "Wall height above retained soil").

Height to Bottom: Defines the lower extent of the applied lateral load measured from the top of the footing.

© 2026 ENERCALC, LLC

Load Factor: Will be applied to the lateral load when performing strength design checks. It is not applied for service level load checks such as sliding, overturning, or soil bearing pressure checks.

Wind on Stem above Soil: Will be applied to that part of the stem projecting above the retained height defined by the entry "Wall height above retained soil." It is used to generate sliding force, overturning moment, stem design moment and shear, and soil pressures.

Wind Type: Note that recent building codes have started to determine wind forces at the strength level as opposed to at the traditional service level. Consequently the program allows the user to indicate whether the specified wind pressure is at the Strength-Level or at the Service-Level.

When performing a design based on IBC 2012 / CBC 2013 or later:

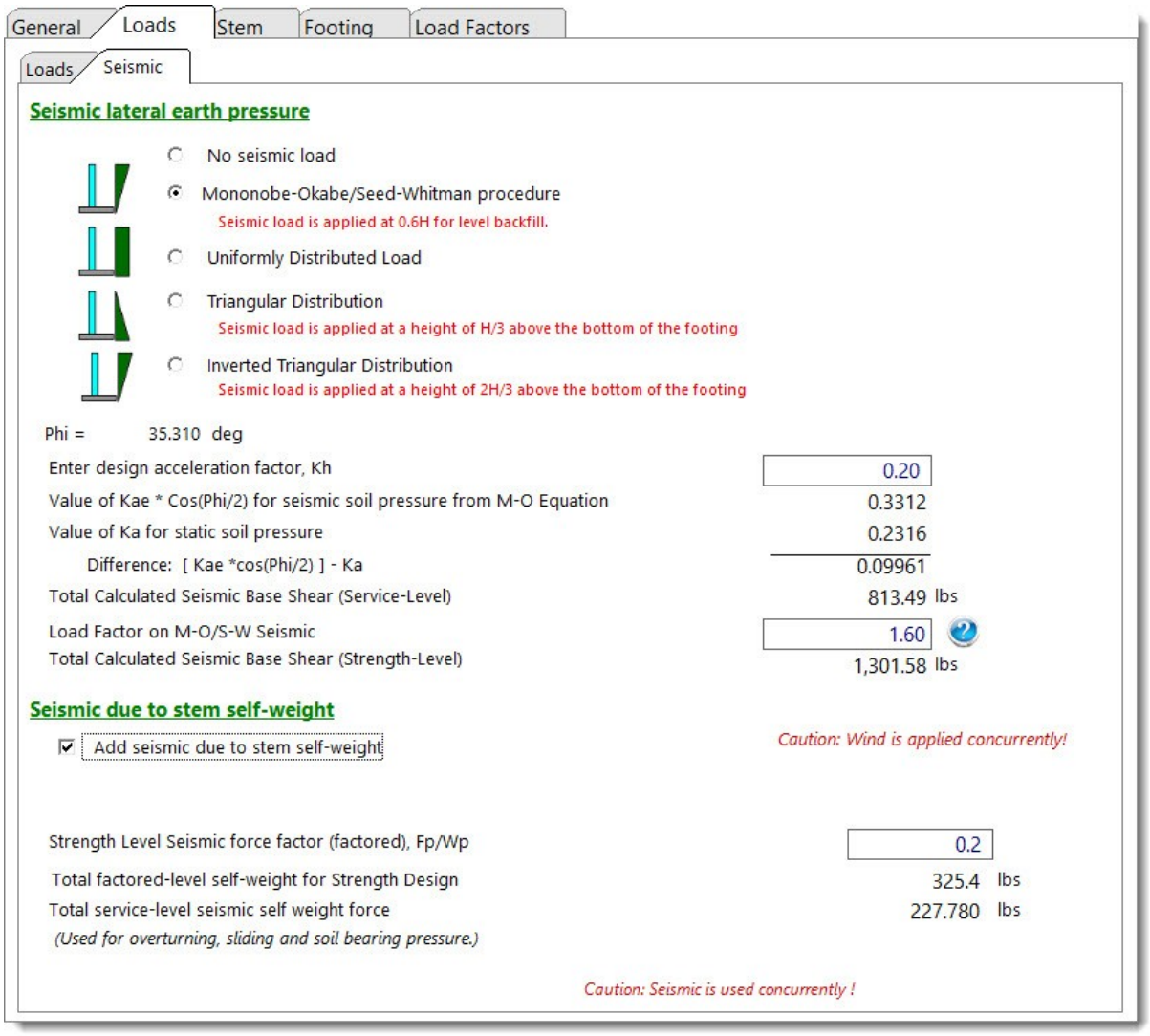
- The wind should be entered as a Strength-Level load.
- When designing a masonry stem by strength design methodology or a concrete stem, the wind load factor (which should be 1.0) will be applied to the specified wind loads.
- When designing a masonry stem by ASD methodology, the wind load factor is not used, and the specified wind loads will be reduced to a service level by multiplying the specified pressure by 0.6.
- Regardless of the stem construction, when determining service-level soil bearing pressure and when performing sliding and overturning checks, the specified wind loads will be reduced to a service level by multiplying the specified pressure by 0.6.

When performing a design based on codes *earlier than* IBC 2012 / CBC 2013:

- The wind should be entered as a Service-Level load.
- When designing a masonry stem by strength design methodology or a concrete stem, the wind load factor (which should be 1.6) will be applied to the specified wind loads.
- When designing a masonry stem by ASD methodology, the wind load factor is not used, and the specified wind loads will be used exactly as specified.
- Regardless of the stem construction, when determining service-level soil bearing pressure and when performing sliding and overturning checks, the specified wind loads will be used exactly as specified.

© 2026 ENERCALC, LLC

Seismic Loads



You can choose to apply seismic force from lateral earth pressure and/or from wall self-weight.

Seismic Lateral Earth Pressure

This category is used to specify whether seismic lateral earth pressure is to be considered or not. If it is to be considered, the program offers the option of two different methods:

- Mononobe-Okabe/Seed-Whitman procedure, or
- Simplified procedure per Geotechnical report

© 2026 ENERCALC, LLC

Mononobe-Okabe/Seed-Whitman Procedure

By entering k_h the program will calculate K_{AE} and K_A using the Mononobe-Okabe/Seed-Whitman equations for a yielding wall (cantilevered).

If it is a non-yielding wall (restrained) the added lateral force per square foot is computed using $F_w = k_h$(density)(retained height), in psf. Common k_h values range from 0.05 to 0.30, depending upon area seismicity. Some sources indicate that $k_h = S_{DS} / 2.5$, but jurisdictions and interpretations vary.

Both the static soil pressure component and the added seismic component will be displayed. The resultant seismic component is assumed to act at 0.6 x retained height. The seismic component is assumed to vary from an intensity of X at the bottom of footing to an intensity of 4X at the top of the retained height.

Mononobe-Okabe/Seed-Whitman Methodology

Two excellent references on the subject include *Basics of Retaining Wall Design* 11th Edition by Hugh Brooks and *Seismic Earth Pressures on Deep Building Basements* by Lew, et al.

The program computes K_{AE} (coefficient for combined active and earthquake forces) per the Coulomb formula, modified by Mononobe-Okabe/Seed-Whitman, to account for earthquake loading, where the term θ is the angle whose tangent is the horizontal ground acceleration. (Note that if $K_h = 0$, $\theta = 0$, then $K_{AE} = K_A$.) Vertical acceleration is neglected, resulting in a more conservative K_{AE}.

K_{AE} = active earth pressure coefficient, static+seismic

$$= \frac{\sin^2(\alpha+\theta-\phi)}{\cos\theta' \sin^2\alpha \sin(\alpha+\theta+\delta)\left[1+\sqrt{\frac{\sin(\phi+\delta)\sin(\phi-\theta-\beta)}{\sin(\alpha+\delta+\theta)\sin(\alpha-\beta)}}\right]^2}$$

Where $\theta = \tan^{-1} K_h$, α = wall slope to horizontal (90 degrees for a vertical face), ϕ = angle of internal friction, β = backfill slope, and δ = wall friction angle.

For a vertical wall face and δ assumed to be $\phi/2$, K_{AE} becomes:

© 2026 ENERCALC, LLC

$$K_{AE} = \frac{\sin^2(90+\theta-\phi)}{\cos\theta \sin^2 90 \sin(90+\theta+\phi/2 \left[1+\sqrt{\frac{\sin 1.5\phi \sin(\phi-\theta-\beta)}{\sin(90+\phi/2+\theta)\sin(90-\beta)}}\right]^2}$$

The values K_{AE} and K_A are displayed.

For the horizontal component, the forces are multiplied by cos δ (wall/soil interface angle).

Total force (active and seismic) = $P_{AE} = 0.5(\gamma) K_{AE} H^2$ where γ = soil density and H = retained height.

Since the total force P_{AE} consists of two components, static (P_A, as previously computed for static forces) with triangular distribution and the earthquake (P_{AE} - P_A) with an inverted semi-triangular distribution with an assumed point of application at 0.60 x height, the combined (static and EQ) point of application is determined by

$$\bar{y} = \frac{P_A(H/3) + (P_{AE} - P_A)\,0.6H}{P_{AE}}$$

which is displayed as "Ht. to static + EQ point of appl."

Total base shear for both static force and added seismic force are displayed.

From *Seismic Earth Pressures on Deep Building Basements* by Lew, et al:

If the Mononobe-Okabe analysis is used to determine the lateral seismic earth pressure, the lateral earth pressure should consist of the static active earth pressure and the seismic increment of earth pressure as discussed in the previous section. Presumably, the load factor of 1.6 in Eq. (8) would be applicable to the total earth pressure in this case. However, as noted above, a reduced load factor would be appropriate when considering the transitory nature of the seismic component and the low likelihood of the load maxima occurring simultaneously. Accordingly a lower load factor of 1.0 is proposed to be applied to the seismic increment component of earth pressure while the 1.6 load factor is applied to the static active pressure component. To facilitate such loading combination the geotechnical engineers would have to separate earth pressure components attributable to the active earth pressure condition and the seismic increment of earth pressure when using the M-O method.

© 2026 ENERCALC, LLC

For Cantilevered Retaining Wall Only:

Other options include:
- Uniformly Distributed Load
- Triangular Distribution
- Inverted Triangular Distribution

For Gravity Retaining Wall and Tapered Retaining Wall:

Simplified procedure per Geotechnical report

Use this method if a geotechnical report specifies added seismic load as a factor multiplied by the retained height, such as X*H, where X is the multiplier and H is the retained height, enter that multiplier here. Using this method, the seismic lateral force will be applied uniformly over the retained height. Since this is a factored force it will be reduced by 0.7 for use in sliding, overturning, and soil bearing calculations.

For Cantilevered Retaining Wall, Gravity Retaining Wall and Tapered Retaining Wall:

Seismic due to stem Self-Weight

If you indicate that you want the program to consider the seismic effect due to the self-weight of the stem, then you will specify a value for the factor Fp/Wp, which will be used to calculate a uniform seismic force in psf (Fp/Wp x (wall weight)). If the wall has multiple stem sections, each will be calculated separately and accumulated for the base shear and moment.

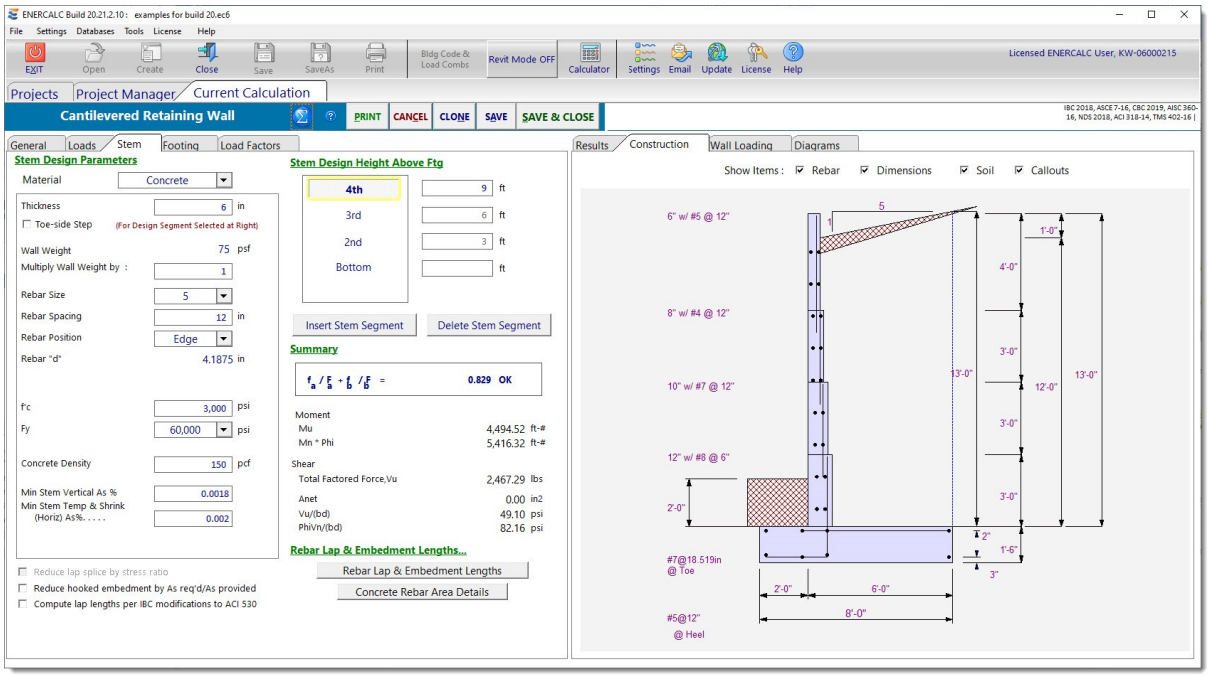
NOTE: The k_h values entered are the <u>design</u> accelerations (not necessarily peak ground acceleration as may be given in a geotechnical report) and must be determined per procedures in the applicable code. The program then applies the appropriate Load Factors (1.0 for concrete design and 0.7 for serviceability checks).

© 2026 ENERCALC, LLC

13.8.1.3 Stem Tab

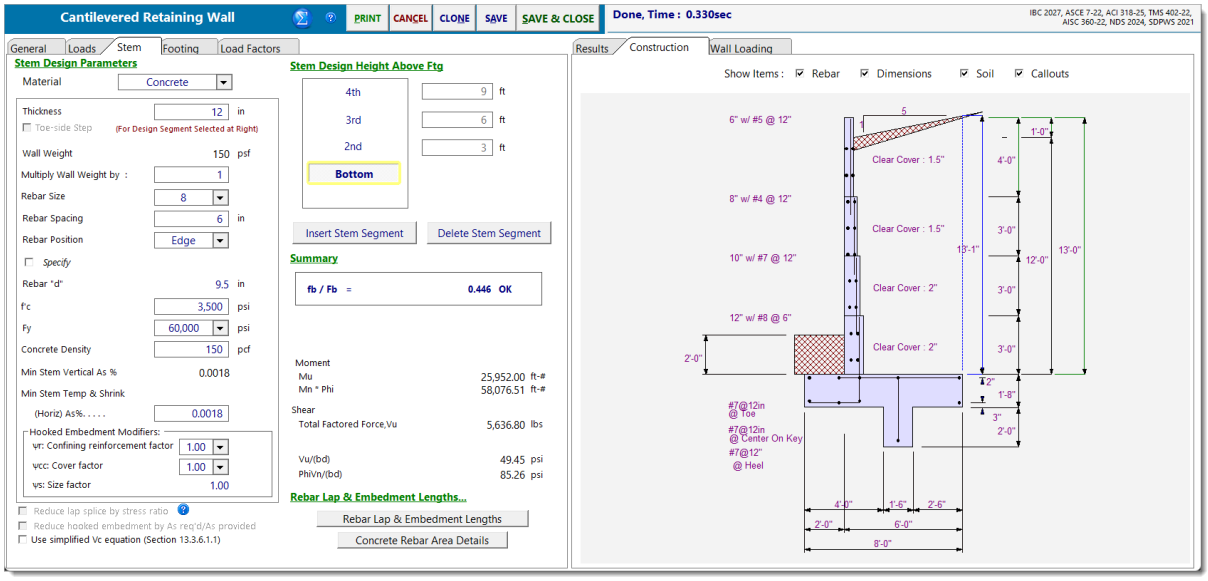
The Stem tab collects the data required to define the stem geometry, reinforcing, and design heights.

The layout and content of this tab will vary depending upon which type of wall is being designed. Refer to the subsequent topics in this section of the User's Manual for wall-specific details on the various parameters that are collected on this tab.



Stem Tab for Cantilevered Retaining Wall

When a Cantilevered Retaining Wall is defined, the Stem tab will appear as shown below:



Stem Design Parameters

Material: Use the drop-down list box to select Masonry, Concrete, Fence, or None. Fence is only allowed on top of the wall, higher than the Retained Height, and is considered weightless. Use None to disable the stem section.

Thickness: Use the spinners to set the thickness of Concrete wall segments. Use the drop-down list box to set the thickness of Masonry wall segments. For segments defined as "Fence" the thickness input is unavailable.

Wall Weight: This displayed value is based upon wall data within the program. A multiplier input field is provided if it becomes necessary to adjust the data. See Appendix C for masonry wall weights.

Design Method: When a masonry stem section is chosen, this allows a choice of ASD or LRFD (Allowable Stress Design or Load and Resistance Factor Design). When the latter is selected the input notations change (e.g. f_s to f_y) and all calculations are based upon LRFD.

Rebar Size: Make your selection from the drop-down list box for bar sizes #3 to #10. “Soft Metric” sizes will be displayed in parentheses.

Rebar Spacing: Use the spinners to set the rebar spacing in Concrete wall segments. Use the drop-down list box to set the rebar spacing in Masonry wall segments. For segments defined as "Fence" the rebar spacing input is unavailable.

© 2026 ENERCALC, LLC

Rebar Position: Chose between Center or Edge. If Center is chosen, the rebar d distance will be 1/2 the actual wall thickness. If Edge is chosen the rebar will be located at the heel side of the stem as defined below.

For masonry wall segments, the program contains a table of the appropriate "d" values to use for various block sizes and center/edge locations, as shown in the table below.

Default Values of Rebar Position for Masonry Wall Segments

	Rebar Depth	
Thickness	***Center***	***Edge***
6"	***2.8125"***	***2.8125"***
8"	***3.8125"***	***5.25"***
10"	***4.8125"***	***7.25"***
12"	***5.8125"***	***9.0"***
14"	***6.8125"***	***11.0"***
16"	***7.8125"***	***13.0"***

For concrete wall segments, the "edge" rebar depth is always stem thickness less 1.5" for #5 and smaller bars (stem thickness less 2" for #6 or larger), less one-half the bar diameter.

Specify Position: Click this box to enter an explicit "d" value for the particular stem segment.

f'm: For Masonry stem segments, enter the compressive strength of masonry in units of psi. This input is not applicable to Concrete stem segments.

f'c: For Concrete stem segments, enter the compressive strength of concrete in units of psi. This input is not applicable to Masonry stem segments.

F_S: For ASD masonry design, select the allowable steel stress, based on working stress design, which should be used for design of the masonry stem segment. The drop-down list box allows quick selection of common values. This input is not applicable to LRFD masonry design or to concrete design.

Fy: For LRFD masonry design and for concrete design, select the rebar yield stress to used for design of the indicated stem segment. The

© 2026 ENERCALC, LLC

drop-down list box allows quick selection of common values. This input is not applicable to ASD masonry design.

CMU weight type: (Applies to Masonry stem segments only.) This input provides a drop-down list box that offers the common CMU weights.

Concrete Density: (Applies to Concrete stem segments only.) This input provides spinners to define the unit weight of the concrete for a particular stem segment.

Solid Grouting: This applies to masonry only, and if this box is checked the weight of the wall will be based upon industry standard values for the weights of solid-grouted walls of lightweight, medium weight, or normal weight block based on the selection for CMU weight type.

If this box is not checked, the program will calculate the weight based upon grouting of only cells containing reinforcing.

This also affects equivalent solid thickness for stem shear calculations, and area for axial stress calculations (combined with moment for masonry stems).

Em = f'm *: This input collects the value by which the compressive strength of masonry is multiplied to arrive at the value of the modulus of elasticity for masonry. TMS 402 specifies $E_m = 900 * f'_m$ which is the default value.

"n", Modular Ratio: This is calculated by the program as Es/Em.

Equivalent Solid Thickness (EST): For partially-grouted masonry stem segments (those where solid grouting has not been specified) the equivalent solid thickness is generated from an internal database. See Appendix C - Masonry Wall Weights & Section Properties 1242.

MaxAs: (Only for Masonry Stems designed according to SD) This indicates the max allowable permissible cross-sectional area of flexural tensile reinforcement to ensure the net tensile strain (ε_t) limits specified by the TMS design standard are satisfied:

TMS 402-16: $\varepsilon_t = 1.5 * f_y / E_s$ *[§ 9.3.3.2.1(a)]*

TMS 402-22: $\varepsilon_t = 0.003 + f_y / E_s$ *[§ 9.3.3.2.4 and Table 9.1.4]* *Note: At this time, the Restrained Retaining Wall module only supports tension-controlled masonry stems (φ = 0.9).*

© 2026 ENERCALC, LLC

Using the equations from TMS 402-16 Commentary Section 9.3.3.2, the maximum flexural reinforcement ratio for a fully grouted masonry wall stem can be calculated as follows:

$$\rho_{max} = \frac{A_{s,max}}{bd} = \frac{0.64f'_m\left(\frac{\varepsilon_{mu}}{\varepsilon_{mu}+\varepsilon_t}\right) - \frac{P}{bd}}{f_y}$$

Where:

ε_{mu} = 0.0025; maximum usable compressive strain of masonry
ε_t = net tensile strain as described above
P = Axial load at the stem section considered
b = 12"; width of compression section
d = depth to rebar as reported on the Stem tab

Similarly, for a partially grouted masonry wall stem, the maximum flexural reinforcement ratio can be calculated using:

$$\rho_{max} = \frac{A_{s,max}}{bd} = \frac{0.64f'_m\left(\frac{\varepsilon_{mu}}{\varepsilon_{mu}+\varepsilon_t}\right)\left(\frac{b_w}{b}\right) + 0.80f'_m\, t_{fs}\left(\frac{b-b_w}{bd}\right) - \frac{P}{bd}}{f_y}$$

Where:

b_w = width of the compression section minus the sum of the length of ungrouted cells. See Appendix C - Masonry Wall Weights & Section Properties 1242.
t_{fs} = face-shell thickness for hollow masonry units. See Appendix C - Masonry Wall Weights & Section Properties 1242.

Finally, the maximum permissible reinforcement area can be determined using:

$$A_{s,max} = \rho_{max} bd$$

Asprovided: (Only for Masonry Stems designed according to SD) This indicates the actual provided cross-sectional area of flexural tensile reinforcement.

Stem Design Height Above Footing

IMPORTANT! The term “Stem Design Height” refers to a height above the top of the footing (i.e. above the base of the stem). It is the height above the bottom of the stem where you want the program to compute moments and shears.

© 2026 ENERCALC, LLC

You can divide the stem into a maximum of five segments (increments of height). Each increment can represent a change in material (concrete, masonry, or fence), thickness, reinforcing size or spacing.

For most walls, only two or three changes in stem sections are used. For example, it would be logical to place a Stem Design Height at the top of the dowels projecting into the stem from the footing and perhaps at another location farther up the wall where a more economical section is desired.

Bottom: You must start your stem design here, at the base (height above footing = 0.00), where the stem moment and shear is maximum. You can manipulate the bar sizes, spacing, and position, as well as the wall material and thickness until the Summary box indicates an acceptable stress ratio (the higher and closer to 1.0, the more efficient).

To check the wall at a higher Design Height, such as where reinforcing or thickness can be reduced, click the Insert Stem button and enter the next higher design height. Advance the spin button to the desired height above the top of the footing or enter it by typing. This will create a new 2nd section that you can now design.

Continue this way, clicking Insert Stem after each stem section design is completed, up to a maximum of five heights. A new Design Height should only be entered when you want to change the material, thickness, or reinforcing, and should never be less than about two-foot intervals.

Summary

The summary box indicates the design shears and moments in the selected stem segment, and the interaction ratio for that segment.

For stem segments of Masonry that are designed according to ASD, the Summary indicates actual and allowable moments, total applied shear force, applied shear stress and allowable shear stress, and rebar lap splice lengths.

For stem segments of Concrete or of Masonry that are designed according to LRFD, the Summary indicates factored applied moment and the nominal moment capacity, the total applied shear force, the factored applied shear stress and the nominal shear stress, and rebar lap splice lengths.

See additional detail in the section named "Summary Section of Stem Tab".

Design Options

The last section offers the following design options:

Hooked Embedment Modifiers — These are the various hooked embedment modifiers used in the hooked development length calculations. For more information, see ACI 318 Table 25.4.3.2 and Summary Section of Stem Tab - Hooked Embedment of Rebar in Concrete[1115].

© 2026 ENERCALC, LLC

Reduce lap splice by stress ratio This option is provided for informational purposes only. Use engineering judgment with the application of this option, as it is contrary to current building codes and design standards.

Reduce hook embedment by As req'd/As provided When this option is selected, the program reduces the calculated hooked embedment depth by applying the ratio of required steel area to provided steel area. This setting is unavailable for ACI 318-19 and later editions as Section 25.4.10 does not permit reducing hooked development length by the ratio of ($A_{s,required}$ / $A_{s,provided}$).

Compute lap lengths per IBC modifications to ACI 530 Masonry stems only. As of build 20.22.10.9, this is automatically selected, because IBC is clear that this is mandatory.

Use simplified Vc equation (Section 13.3.6.1.1) For cantilever retaining walls, ACI 318-25 Section 13.3.6.1.1 allows a simplified shear equation ($V_c = 2 \lambda \sqrt{(f'_c)}\ b_w\ d$) as an alternative to the one-way shear provisions in Section 22.5.5.1. Enabling this option usually gives a higher shear capacity. For walls with higher reinforcement ratios, Section 22.5.5.1 can still produce larger values, which happens when this option is disabled.

Summary Section of Stem Tab

The summary section indicates the results of the Stem design at-a-glance.

Interaction Ratio: The interaction ratio indicates the efficiency of your design, not to exceed 1.0.

For masonry using ASD this is the computed ratio of $f_a/F_a + f_b/F_b$.

© 2026 ENERCALC, LLC

For concrete and masonry using LRFD it is $M_{actual}/M_{allowable}$.

The weight of the stem will be included only if there is added axial load. For masonry stems, F_a is calculated by considering the wall as unsupported with "K" = 2.0. Since even a very small axial load will activate the unsupported height/slenderness calculation for masonry stems, we suggest you do not enter an axial load unless it is significant (e.g. greater than, say, 3000 plf.).

Actual Moment: This is the maximum moment due to the lateral pressures and applied loads above the "Design Height" location entered. Note that when concrete is used, all soil pressures and loads are factored per default Load Factors for evaluation of moments and shears.

Allowable Moment: This is the allowable moment capacity. It is Allowable Stress Design (ASD) for masonry, or based upon Strength Design for concrete and when LRFD is specified for masonry. For concrete strength design, the maximum reinforcing steel percentage is controlled by equilibrium at the prescribed strain limits.

Total Force: This is the total lateral force from loads applied above the "Check Design at Height" location entered. This force is factored for concrete and masonry using the LRFD method. Forces applied to compute overturning, sliding, and soil pressure are not factored.

Actual Shear: For masonry stems, the shear stress is calculated as Total Shear Force / An. For concrete stems, the shear stress is calculated as Total Shear Force / (12" * 'd').

Allowable Shear: For masonry stems designed by ASD, the allowable shear stress is calculated per TMS 402-16 Eqn. 8-26 as follows:

$$F_{vm} = \frac{1}{2}\left[\left(4.0 - 1.75\left(\frac{M}{Vd_v}\right)\right)\sqrt{f'_m}\right] + 0.25\frac{P}{A_n} \geq 0$$

where $\frac{M}{Vd_v}$ need not exceed 1.0

Since no contribution of shear strength is assumed from reinforcing steel in a retaining wall, nor is the stem a partially grouted shear wall, TMS 402-16 Eqn. 8-22 reduces to $F_v = F_{vm}$

An upper bound limit $F_{v,max}$ is imposed on F_v such that $F_v \leq F_{v,max}$ where:

a) $F_{v,max} \leq 3\sqrt{f'_m}$ where $M/(Vd_v) \leq 0.25$

b) $F_{v,max} \leq 2\sqrt{f'_m}$ where $M/(Vd_v) \geq 1.0$

© 2026 ENERCALC, LLC

c) and $F_{v,max}$ is linearly interpolated for values of $M/(Vd_v)$ between 0.25 and 1.0.

For masonry stems designed by LRFD, the nominal shear strength is calculated per TMS 402-16 Eqn. 9-20 as follows:

$$v_{nm} = \left[4.0 - 1.75\left(\frac{M_u}{V_u d_v}\right)\right]\sqrt{f'_m} + 0.25\frac{P_u}{A_n} \geq 0$$

where $\frac{M_u}{V_u d_v}$ need not exceed 1.0

Since no contribution of shear strength is assumed from reinforcing steel in a retaining wall, nor is the stem a partially grouted shear wall, TMS 402-16 Eqn. 9-17 reduces to $v_n = v_{nm}$

An upper bound limit $v_{n,max}$ is imposed on v_n such that $v_n \leq v_{n,max}$ where:

a) $v_{n,max} \leq 6\sqrt{f'_m}$ where $M_u/(V_u d_v) \leq 0.25$
b) $v_{n,max} \leq 4\sqrt{f'_m}$ where $M_u/(V_u d_v) \geq 1.0$
c) and $v_{n,max}$ is linearly interpolated for values of $M_u/(V_u d_v)$ between 0.25 and 1.0.

For concrete stems designed per ACI 318-14 and earlier, or ACI 318-25 and later with *Use simplified Vc equation (Section 13.3.6.1.1)* 1111 enabled, the nominal shear strength is calculated as follows:

$\varphi v_n = \varphi * 2 * \lambda * \sqrt{f'_c}$, where $\varphi = 0.75$.

For concrete stems designed per ACI 318-19, or ACI 318-25 and later with *Use simplified Vc equation (Section 13.3.6.1.1)* 1111 disabled, the nominal shear strength follows the one-way shear provisions of §22.5 and Table 22.5.5.1 (referenced by §13.3.6.1 and §7.5.3.1):

$\varphi v_n = \varphi * 8 * \lambda_s * \lambda * (\rho_w)^{1/3} * \sqrt{f'_c}$, where $\varphi = 0.75$.

Note: The one-way shear capacity assumes no transverse reinforcement in the stem. Therefore, $A_v < A_{v,min}$, and equations (a) and (b) from Table 22.5.5.1 are not applicable.

This value is also subject to additional limits per §22.5.5.1.1:

Lower bound: $\varphi * \lambda * \sqrt{f'_c}$ (ACI 318-25 and later only)

Upper bound: $\varphi * 5 * \lambda * \sqrt{f'_c}$

© 2026 ENERCALC, LLC

Rebar Lap & Embedment Lengths:

Regardless of the stem material, there are two fundamental lengths to calculate: lap splice length and development length. These values are summarized in the "Rebar Lap & Embedment Lengths" table which can be accessed from the button on the Stem Design Values tab.

The following presents the formulas used and the limits applied to generate the values in that table:

Straight Development Length of Rebar in Concrete: (Applies to all referenced codes)

$l_{d\ calc}$ = (3/40) * (fy / sqrt(f'c)) * (psi_s / 2.5) * (bar size / 8)

psi_s = 0.8 for bar sizes #6 and smaller, 1.0 for bar sizes #7 and larger

$l_{d\ report}$ = $l_{d\ calc}$ but not less than 12 inches

(This is Eq. (25.4.2.3a) from ACI 318-14 and Eq. (25.4.2.4a) from ACI 318-19 with appropriate assumptions for bar location, clear cover, spacing, transverse reinforcing, and epoxy coating.)

Lap Splice Length of Rebar in Concrete: (Applies to all referenced codes)

l_s = 1.3 * $l_{d\ calc}$ but not less than 12 inches

© 2026 ENERCALC, LLC

Hooked Embedment of Rebar in Concrete:

For ACI 318-14:

$$\ell_{dh} = \left(\frac{f_y \psi_e \psi_c \psi_r}{50\lambda\sqrt{f'_c}} \right) d_b$$ but not taken less than 8 * d_b or 6" per Section 25.4.3.1

Where:

f_y = Rebar yield stress for the bottom stem segment.
ψ_e = Epoxy factor per Table 25.4.3.2 (taken as 1.0).
ψ_c = Cover factor per Table 25.4.3.2 (user-specified as 0.7 or 1.0).
ψ_r = Confining Reinforcement factor per Table 25.4.3.2 (user-specified as 0.8 or 1.0).
λ = Lightweight factor per Table 25.4.3.2 (taken as 1.0).
f'_c = Concrete compressive strength of the footing.
d_b = Rebar diameter for the bottom stem segment.

Users may optionally choose to reduce the calculated ℓ_{dh} by the ratio of required-to-provided rebar area ($A_{s,required}$ / $A_{s,provided}$) by clicking the associated check box on the Stem tab.

For ACI 318-19:

$$\ell_{dh} = \left(\frac{f_y \psi_e \psi_r \psi_o \psi_c}{55\lambda\sqrt{f'_c}} \right) d_b^{1.5}$$ but not taken less than 8 * d_b or 6" per Section 25.4.3.1

Where:

f_y = Rebar yield stress for the bottom stem segment.
ψ_e = Epoxy factor per Table 25.4.3.2 (taken as 1.0).
ψ_r = Confining Reinforcement factor per Table 25.4.3.2 (user-specified as 1.0 or 1.6).
ψ_o = Location factor per Table 25.4.3.2 (user-specified as 1.0 or 1.25).
ψ_c = Concrete Strength factor per Table 25.4.3.2 (calculated as the minimum of f'_c/15,000 + 0.6 and 1.0).
λ = Lightweight factor per Table 25.4.3.2 (taken as 1.0).
f'_c = Concrete compressive strength of the footing.
d_b = Rebar diameter for the bottom stem segment.

For ACI 318-25:

$$\ell_{dh} = \left(\frac{f_y \psi_e \psi_s \psi_{cc} \psi_r}{50\lambda\sqrt{f'_c}} \right) d_b$$ but not taken less than 8 * d_b or 6" per Section 25.4.3.1

Where:

f_y = Rebar yield stress for the bottom stem segment.

© 2026 ENERCALC, LLC

ψ_e = Epoxy factor per Table 25.4.3.2 (taken as 1.0).
ψ_s = Size factor per Table 25.4.3.2 (taken as 1.0 for #9 bars and smaller, 1.15 for #10 and #11 bars, 1.3 for #14 bars, and 1.5 for #18 bars).
ψ_{cc} = Cover factor per Table 25.4.3.2 (user-specified as 0.7 or 1.0).
ψ_r = Confining Reinforcement factor per Table 25.4.3.2 (user-specified as 0.8 or 1.0).
λ = Lightweight factor per Table 25.4.3.2 (taken as 1.0).
f'_c = Concrete compressive strength of the footing.
d_b = Rebar diameter for the bottom stem segment.

Note: Section 25.4.10 for ACI 318-19 and later editions does not permit reducing hooked development length by the ratio of ($A_{s,required}$ / $A_{s,provided}$):

25.4.10 *Reduction of development length for excess reinforcement*

25.4.10.1 Reduction of development lengths defined in 25.4.2.1(a), 25.4.6.1(a), 25.4.7.1(a), and 25.4.9.1(a) shall be permitted by use of the ratio $(A_{s,required})/(A_{s,provided})$, except where prohibited by 25.4.10.2. The modified development lengths shall not be less than the respective minimums specified in 25.4.2.1(b), 25.4.6.1(b), 25.4.7.1(b), and 25.4.9.1(b).

25.4.10.2 A reduction of development length in accordance with 25.4.10.1 is not permitted for (a) through (f)

(a) At noncontinuous supports

(b) At locations where anchorage or development for f_y is required

(c) Where bars are required to be continuous

(d) For hooked, headed, and mechanically anchored deformed reinforcement

(e) In seismic-force-resisting systems in structures assigned to Seismic Design Categories C, D, E, or F

(f) Anchorage of concrete piles and concrete filled pipe piles to pile caps in structures assigned to Seismic Design Categories C, D, E, or F

© 2026 ENERCALC, LLC

Development Length of Rebar in Masonry designed by ASD: (Applies to all referenced codes)

$l_{d\ calc}$ = (0.002) * (bar size / 8) * f_s

f_s = actual stress in rebar

$l_{d\ report}$ = $l_{d\ calc}$ but not less than 12 inches

(This is the IBC equation.)

Lap Splice Length of Rebar in Masonry designed by ASD: (Applies to all referenced codes)

l_s = Factor * $l_{d\ calc}$ but not less than 12 inches or 40 bar diameters

Factor = 1.5 in regions where design tensile stresses in reinforcement are greater than 0.8 * f_s, otherwise 1.0.

(As of build 20.22.10.9, the program conservatively assumes a value of 1.5 for the "Factor" referenced above.)

Development Length of Rebar in Masonry designed by LRFD: (Applies to all referenced codes)

$l_{d\ calc}$ = (0.13) * (bar size / 8)^2 * fy * gamma / (K * sqrt(f'm))

gamma = 1.0 for #3 through #5 bars, 1.3 for #6 through #7 bars, and 1.5 for #8 through #9 bars

K = 1.5 for #3 through #5 bars, 2.0 for #6 through #9 bars

$l_{d\ report}$ = $l_{d\ calc}$ but not less than 12 inches

(This is the ACI equation by direct reference from IBC. The value of K has conservatively been set to the required clear cover for the selected bar exposed to earth.)

Lap Splice Length of Rebar in Masonry designed by LRFD: (Applies to all referenced codes)

l_s = 1.0 * $l_{d\ calc}$ but not less than 12 inches and need not be GREATER than 72 bar diameters

© 2026 ENERCALC, LLC

General Notes on Rebar Lap & Embedment Lengths:

For concrete stems, a Class B lap splice is assumed, therefore the lap length is the bar development length x 1.3. Concrete is assumed to be normal weight, and bars are assumed to be plain (not epoxy coated).

Concrete development lengths are computed per ACI 318.

For the bottom Design Height only (Ht. = 0.00), this displays the required hooked bar embedment into the footing. It assumes a bar with a 90° bend and at least a 12-diameter extension.

The minimum footing thickness required is based upon this embedment depth plus the clearance you have specified below the bar (usually 3 inches). If this totals more than the footing thickness you have chosen, a warning message will be displayed.

Note that if the bar extends straight down into a key, it must be embedded by a depth equal to the development length.

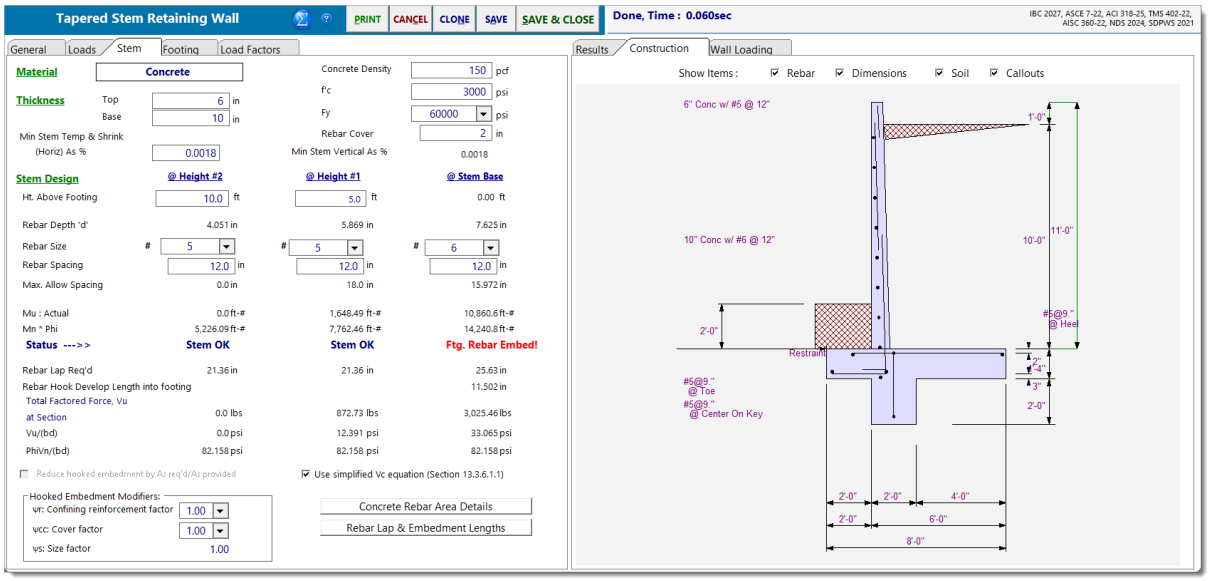
The program does not reduce embedment length by stress level unless the user selects the checkbox labeled Reduce Hook Embedment by Percent Rebar Stress.

The program never reduces lap splice lengths by the stress ratio. It is not permitted by the referenced codes.

Stem Tab for Tapered Stem Retaining Wall

Tapered Stem Retaining Walls are cantilevered retaining walls where the soil face is battered to achieve a variable thickness from the base of the stem to the top of the stem.

When a Tapered Stem Retaining Wall is defined, the Stem tab will appear as shown below:



Note: Taper can only apply to the inside face (the face against the soil).

© 2026 ENERCALC, LLC

Material: The Material will automatically be defined as Concrete, since masonry cannot be tapered.

Thickness: Top and Base: Enter the stem thickness at the top and at the bottom.

Concrete Density: Use the spinners to set the unit weight of the concrete.

f'c and Fy: Enter concrete strength and rebar yield stress.

Rebar Cover: Select the rebar clear cover to consider in the design.

Stem Design: Stem design will automatically be performed at the bottom of the stem (interface with the footing). In addition, you can specify two additional heights above the base to check moments and shears. These are identified as "@ Height #2" and "@ Height #1", where the latter is the lower height.

Ht. Above Footing: Specify two heights above the top of footing elevation where a stem design should be performed (such as where it would be desirable to change the rebar pattern or size for economy). Height #2 is highest and Height at Stem Base will be fixed at 0.00. The #1 height should be located at a distance above the top of the footing that is at least equal to the lap splice length for the dowels.

Rebar Depth "d": This will be computed based upon the heights you have chosen, the specified wall taper, and the specified Rebar Cover. The program automatically uses the specified clear cover and one-half of a bar diameter when determining "d".

Rebar Size: Use the drop-down list box to select the desired rebar size.

Rebar Spacing: Use the spinners to set the desired rebar spacing. (The maximum permissible spacing is 18 inches, which is in accordance with ACI.)

Max. Permissible Spacing: This is the maximum permissible spacing for the rebar size selected. This is based on the strength calculation, but it will stop at an upper limit of 18 inches in accordance with ACI.

$\mathbf{M_u}$: These are factored moments at the heights you have selected. These will be based on the load factors that you specify on the Load Factors tab. Compare these values with Design Moment as described below, to verify adequacy of your design at the selected height location.

$\phi\mathbf{M_n}$: This is the design moment strength, which will be based upon the bar sizes and spacings you established, along with wall geometry, concrete strength, etc.

© 2026 ENERCALC, LLC

Status: This indicates whether each stem design is OK at the specified height. If there is a problem, this will display a descriptive message such as "M_u > Phi * M_n" or "A_s < min" or "A_s > max" or "Ftg. Rebar Embed!".

Rebar Lap Req'd: This is the lap splice length required based on the bar size used at the specified Design Height. It is the development length of the bar multiplied by 1.3 (assuming a Class B splice) and without adjustment for stress level.

Rebar Hooked Embedment Length: This is the hooked development length that is required into the concrete footing for the bar size specified at the stem base. For more information, see Summary Section of Stem Tab - Hooked Embedment of Rebar in Concrete[1115].

Shear at Section: This is the total factored shear at the indicated height.

V_u: Factored shear stress at designated height computed by Shear at Section / (12 * "d").

ϕV_n: For concrete stems designed per ACI 318-14 and earlier, or ACI 318-25 and later with *"Use simplified Vc equation (Section 13.3.6.1.1)"* enabled, the nominal shear strength is calculated as follows:

$\varphi v_n = \varphi * 2 * \lambda * \sqrt{(f'_c)}$, where $\varphi = 0.75$.

For concrete stems designed per ACI 318-19, or ACI 318-25 and later with *"Use simplified Vc equation (Section 13.3.6.1.1)"* enabled, the nominal shear strength follows the one-way shear provisions of §22.5 and Table 22.5.5.1 (referenced by §13.3.6.1 and §7.5.3.1):

$\varphi v_n = \varphi * 8 * \lambda_s * \lambda * (\rho_w)^{1/3} * \sqrt{(f'_c)}$, where $\varphi = 0.75$.

Note: The one-way shear capacity assumes no transverse reinforcement in the stem. Therefore, $A_v < A_{v,min}$, and equations (a) and (b) from Table 22.5.5.1 are not applicable.

This value is also subject to additional limits per §22.5.5.1.1:

Lower bound: $\varphi * \lambda * \sqrt{(f'_c)}$ (ACI 318-25 and later only)

Upper bound: $\varphi * 5 * \lambda * \sqrt{(f'_c)}$

Reduce hooked embedment by As req'd/As provided: When this option is selected, the program reduces the calculated hooked embedment depth by applying the ratio of required steel area to provided steel area. This setting is unavailable for ACI 318-19 and later editions as Section

© 2026 ENERCALC, LLC

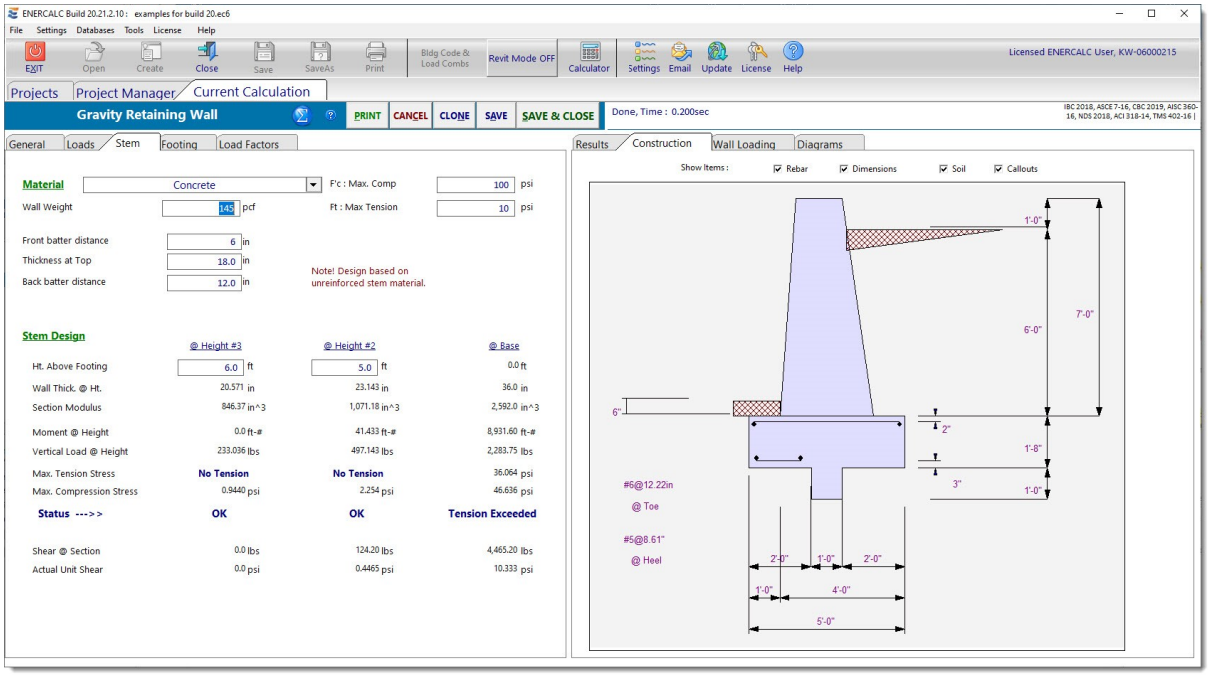
25.4.10 does not permit reducing hooked development length by the ratio of ($A_{s,required}$ / $A_{s,provided}$).

Use simplified Vc equation (Section 13.3.6.1.1): For cantilever retaining walls, ACI 318-25 Section 13.3.6.1.1 allows a simplified shear equation ($V_c = 2 \lambda \sqrt{(f'_c)}\ b_w\ d$) as an alternative to the one-way shear provisions in Section 22.5.5.1. Enabling this option usually gives a higher shear capacity. For walls with higher reinforcement ratios, Section 22.5.5.1 can still produce larger values, which happens when this option is disabled.

Hooked Embedment Modifiers: These are the various hooked embedment modifiers used in the hooked development length calculations. For more information, see ACI 318 Table 25.4.3.2 and Summary Section of Stem Tab - Hooked Embedment of Rebar in Concrete 1115.

Stem Tab for Gravity Retaining Wall

When a Gravity Retaining Wall is defined, the Stem tab will appear as shown below:



Gravity walls may have one or both sides tapered and are assumed to be proportioned such that no reinforcing is required since every section is primarily in compression. Any solid homogeneous material may be used. Reinforcing can be added if there is any tension in the cross section, but the program does not compute this requirement.

Material: Use this drop-down list box to specify the material being considered.

Wall Weight: Enter the weight of the wall material in pcf. Generally this will be the weight of concrete or rubble (approximately 145 pcf).

Front Batter Distance: Enter the offset of the front face at top of the wall from the front face at the base. Value should be greater than or equal to zero.

Thickness at Top: Enter the thickness of the top of the wall.

Back Batter Distance: Enter the offset of the back face at the top of the wall from the back face at the base. Value should be greater than or equal to zero.

F'c Max. Compression: Enter your criteria for the maximum permissible compressive stress on the wall. Usually varies from 100 psi to over 700 psi.

Ft Max. Tension: Enter your criteria for the maximum permissible tensile stress on the wall. Usually varies from about 15 psi to 40 psi. Generally gravity walls are designed such that there is no tension – the full cross section is in compression.

Stem Design

Stem design will automatically be performed at the bottom of the stem (interface with the footing). In addition, you can specify two additional heights above the base to check stresses. These are identified as "@ Height #2" and "@ Height #1", where the latter is the lower height.

Height Above Footing: Specify two heights above the top of footing elevation where stem stresses should be checked. Height #2 is highest and @Stem Base will be fixed at 0.00.

Wall Thickness @ Height: Displays the calculated values of wall thickness at the heights you have specified for analysis.

Section Modulus: Displays the computed section modulus at the heights selected for analysis.

Moment @ Height: Displays the moment at the designated design heights.

Vertical Load @ Height: Displays the summation of vertical loads above designated height.

© 2026 ENERCALC, LLC

Maximum Tension / Compression Stress: Displays extreme tension and compression stresses based on interaction formulas.

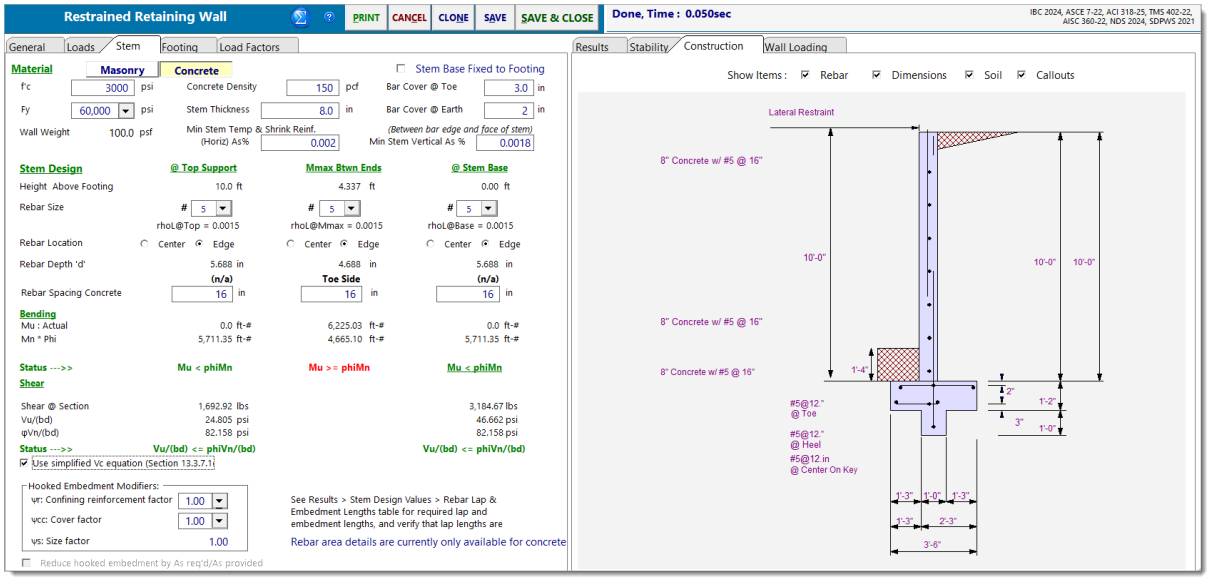
Status: Indicates "OK" if not tension exists. If tension exists but it does not exceed the user-specified threshold then the status indicates "Tension Exists". If tension exists to a degree that exceeds the user-specified threshold then the status indicates "Tension Exceeded". If compression exists to a degree that exceeds the user-specified threshold then the status indicates "Compression Exceeded".

Shear @ Section: Displays total shear force at the designated height.

Actual Unit Shear: Displays the calculated shear stress at the designated height. Compare this with the allowable shear for the material you have selected.

Stem Tab for Restrained Retaining Wall

When a Restrained Retaining Wall is defined, the Stem tab will appear as shown below:



Note: The lateral support should be near the top of the wall, although some extension of the wall above the support is permitted by the program. You have the option of fixing the base (as for a cantilevered wall) or assuming it pinned. Intermediate degrees of fixity are not permitted. The program will compute moments, shears, and stresses at three locations: base (negative moment if fixed; zero moment if pinned), maximum positive moment between base and lateral support, and at the point of lateral support.

Material

Select Masonry or Concrete. Only one material can be used, and must be of constant thickness.

© 2026 ENERCALC, LLC

Stem Base Fixed to Footing: Clicking this box will model the stem as being fully fixed at the base (connection to the footing). If unchecked, the stem will be considered pinned to the footing (no moment fixity).

Stem Thickness: The program only permits a constant thickness throughout the height of the wall. Enter the stem thickness of a concrete stem, or select the thickness of a masonry stem from the drop-down list box offering common CMU sizes.

Design Method: (Only applies to Masonry stems) Select the design method to be used, either ASD or LRFD.

Block Weight Multiplier: (Only applies to Masonry stems) Provides a multiplier input field in case it becomes necessary to adjust the data. See Appendix C for masonry wall weights.

Solid Grouted Block: (Only applies to Masonry stems) If this box is checked the weight of the wall will be based upon industry standard values for the weights of solid-grouted walls of lightweight, medium weight, or normal weight block based on the selection for CMU weight type. If this box is not checked, the program will look up the weight based upon grouting of only cells containing reinforcing. This also affects equivalent solid thickness for stem shear calculations, and area for axial stress calculations (combined with moment for masonry stems).

f'm: For Masonry stem segments, enter the compressive strength of masonry in units of psi. This input is not applicable to Concrete stem segments.

f'c: For Concrete stem segments, enter the compressive strength of concrete in units of psi. This input is not applicable to Masonry stem segments.

F_S: For ASD masonry design, select the allowable steel stress, based on working stress design, which should be used for design of the masonry stem segment. The drop-down list box allows quick selection of common values. This input is not applicable to LRFD masonry design or to concrete design.

Fy: For LRFD masonry design and for concrete design, select the rebar yield stress to used for design of the indicated stem segment. The drop-down list box allows quick selection of common values. This input is not applicable to ASD masonry design.

Em = f'm *: This input collects the value by which the compressive strength of masonry is multiplied to arrive at the value of the modulus of elasticity for masonry. IBC specifies $E_m = 900*f'_m$ which is the default value.

© 2026 ENERCALC, LLC

CMU Type: (Applies to Masonry stem segments only.) This input provides a drop-down list box that offers the common CMU weights.

Concrete Density: (Applies to Concrete stems only.) This input provides spinners to define the unit weight of the concrete for the stem.

Rebar Cover: This appears if a concrete stem is chosen and lets you enter desired cover on toe and earth side. The cover is used to calculate the "d" dimension when the rebar is specified to be in the "Edge" position of Concrete stems in the Stem Design category, which is explained in more detail below. When the rebar is specified to be in the "Edge" position of Masonry stems, the program uses tabular data on the geometry of various CMU sizes to calculate the "d" dimension. (Refer to the "Stem Tab for Cantilevered Retaining Wall" topic for detailed information regarding the calculated "d" dimension for Masonry stems.)

Stem Design

This allows you to design or check wall moment and shear at three locations: @ Top Support, @ M_{max} Between Ends, and @ Stem Base. If base is pinned, the entries under @ Stem Base will be zero or dimmed.

Ht. Above Footing: This displays, from left to right, the distance from the top of footing up to the lateral support, the distance from the top of footing up to the point of maximum positive moment, and it displays 0.00 ft to represent the design that is performed at the base of the stem.

Rebar Size: Make your selection from the drop-down list box for bar sizes #4 to #10.

Rebar Location: Choose Center or Edge placement.

Rebar Depth "d": For concrete wall segments, the "edge" rebar depth is always stem thickness less 1.5" for #5 and smaller bars (stem thickness less 2" for #6 or larger), less one-half the bar diameter.

For masonry wall segments, the program contains a table of the appropriate "d" values to use for various block sizes and center/edge locations, as shown in the table below.

Default Values of Rebar Position for Masonry Wall Segments

	Rebar Depth	
Thickness	***Center***	***Edge***
6"	***2.8125"***	***2.8125"***

© 2026 ENERCALC, LLC

	Rebar Depth	
8"	3.8125"	5.25"
10"	4.8125"	7.25"
12"	5.8125"	9.0"
14"	6.8125"	11.0"
16"	7.8125"	13.0"

Rebar Placement: Serves as a convenient reminder to indicate which side of the wall the specified rebar is considered to be placed on.

Rebar Spacing: For Concrete stems, use the spinners to increment the rebar spacing. For Masonry stems, use the drop-down list box to select a modular spacing.

Mu: (Only for Concrete Stems and for Masonry Stems designed according to LRFD) Displays factored moments at the indicated locations with (+) and (-) as applicable. For concrete stems and for masonry stems designed according to LRFD, the moments will be factored by the load factors specified on the Load Factors tab.

Actual Moment: (Only for Masonry Stems designed according to ASD) Displays actual moments at the indicated locations with (+) and (-) as applicable.

ϕMn: (Only for Concrete Stems and for Masonry Stems designed according to LRFD) This is the design moment strength, which will be based upon the bar sizes and spacings you established, along with wall geometry, concrete strength, etc.

Allowable Moment: (Only for Masonry Stems designed according to ASD) This is the allowable moment capacity based upon the bar sizes and spacings you established, along with wall geometry, concrete strength, etc.

Status: This indicates whether the stem design is OK for flexure at the specified height. If there is a problem, this will display a descriptive message such as "Mu > Phi * Mn" or "As < min" or "As > max" or "Ftg. Rebar Embed!".

MaxAs: (Only for Masonry Stems designed according to SD) This indicates the max allowable permissible cross-sectional area of flexural tensile reinforcement to ensure the net tensile strain (ε_t) limits specified by the TMS design standard are satisfied:

TMS 402-16: $\varepsilon_t = 1.5 * f_y / E_s$ *[§ 9.3.3.2.1(a)]*

TMS 402-22: $\varepsilon_t = 0.003 + f_y / E_s$ *[§ 9.3.3.2.4 and Table 9.1.4]* *Note: At this time, the Restrained Retaining Wall*

© 2026 ENERCALC, LLC

module only supports tension-controlled masonry stems (φ = 0.9).

Using the equations from TMS 402-16 Commentary Section 9.3.3.2, the maximum flexural reinforcement ratio for a fully grouted masonry wall stem can be calculated as follows:

$$\rho_{max} = \frac{A_{s,max}}{bd} = \frac{0.64f'_m\left(\frac{\varepsilon_{mu}}{\varepsilon_{mu}+\varepsilon_t}\right) - \frac{P}{bd}}{f_y}$$

Where:

ε_{mu} = 0.0025; maximum usable compressive strain of masonry
ε_t = net tensile strain as described above
P = Axial load at the stem section considered
b = 12"; width of compression section
d = depth to rebar as reported on the Stem tab

Similarly, for a partially grouted masonry wall stem, the maximum flexural reinforcement ratio can be calculated using:

$$\rho_{max} = \frac{A_{s,max}}{bd} = \frac{0.64f'_m\left(\frac{\varepsilon_{mu}}{\varepsilon_{mu}+\varepsilon_t}\right)\left(\frac{b_w}{b}\right) + 0.80f'_m\, t_{fs}\left(\frac{b-b_w}{bd}\right) - \frac{P}{bd}}{f_y}$$

Where:

b_w = width of the compression section minus the sum of the length of ungrouted cells. See Appendix C - Masonry Wall Weights & Section Properties 1242.
t_{fs} = face-shell thickness for hollow masonry units. See Appendix C - Masonry Wall Weights & Section Properties 1242.

Finally, the maximum permissible reinforcement area can be determined using:

$$A_{s,max} = \rho_{max} bd$$

Asprovided: (Only for Masonry Stems designed according to SD) This indicates the actual provided cross-sectional area of flexural tensile reinforcement.

Rebar Lap Req'd: For masonry, the lap required is 48 bar diameters for F_S = 32,000 psi and 40 diameters for F_S = 20,000 psi. For concrete, a Class B splice is assumed, which multiplies the development length by 1.3 (See ACI

© 2026 ENERCALC, LLC

12.15.2), and excludes reduction for stress level. Note: The program does not compute or display bar cut-off points, which must be done manually, or extend positive reinforcing so it is acceptable.

Rebar Hooked Embedment Length: This is the hooked development length that is required into the concrete footing for the bar size specified at the stem base. For more information, see Summary Section of Stem Tab - Hooked Embedment of Rebar in Concrete[1115].

Shear at Section: This is the total shear force at the indicated height (factored for concrete or masonry designed according to LRFD).

Factored Shear Stress: (or Applied Shear Stress for Masonry Stems designed according to ASD) For masonry stems, the shear stress is calculated as Total Shear Force / An. For concrete stems, the shear stress is calculated as Total Shear Force / (12" * 'd').

Design Shear Strength: (or Allowable Shear Stress for Masonry Stems designed according to ASD)

For masonry stems designed by ASD, the allowable shear stress is calculated per TMS 402-16 Eqn. 8-26 as follows:

$$F_{vm} = \frac{1}{2}\left[\left(4.0 - 1.75\left(\frac{M}{Vd_v}\right)\right)\sqrt{f'_m}\right] + 0.25\frac{P}{A_{net}} \geq 0$$

where $\frac{M}{Vd_v}$ need not exceed 1.0

Since no contribution of shear strength is assumed from reinforcing steel in a retaining wall, nor is the stem a partially grouted shear wall, TMS 402-16 Eqn. 8-22 reduces to $F_v = F_{vm}$

An upper bound limit $F_{v,max}$ is imposed on F_v such that $F_v \leq F_{v,max}$ where:

a) $F_{v,max} \leq 3\sqrt{f'_m}$ where $M/(Vd_v) \leq 0.25$
b) $F_{v,max} \leq 2\sqrt{f'_m}$ where $M/(Vd_v) \geq 1.0$
c) and $F_{v,max}$ is linearly interpolated for values of $M/(Vd_v)$ between 0.25 and 1.0.

For masonry stems designed by LRFD, the nominal shear strength is calculated per TMS 402-16 Eqn. 9-20 as follows:

$$v_{nm} = \left[4.0 - 1.75\left(\frac{M_u}{V_u d_v}\right)\right]\sqrt{f'_m} + 0.25\frac{P_u}{A_{net}} \geq 0$$

where $\frac{M_u}{V_u d_v}$ need not exceed 1.0

© 2026 ENERCALC, LLC

Since no contribution of shear strength is assumed from reinforcing steel in a retaining wall, nor is the stem a partially grouted shear wall, TMS 402-16 Eqn. 9-17 reduces to $v_n = v_{nm}$

An upper bound limit $v_{n,max}$ is imposed on v_n such that $v_n \leq v_{n,max}$ where:

a) $v_{n,max} \leq 6\sqrt{f'_m}$ where $M_u/(V_u d_v) \leq 0.25$

b) $v_{n,max} \leq 4\sqrt{f'_m}$ where $M_u/(V_u d_v) \geq 1.0$

c) and $v_{n,max}$ is linearly interpolated for values of $M_u/(V_u d_v)$ between 0.25 and 1.0.

For concrete stems designed per ACI 318-14 and earlier, or ACI 318-25 and later with *"Use simplified Vc equation (Section 13.3.7.1c)"* enabled, the nominal shear strength is calculated as follows:

$\varphi v_n = \varphi * 2 * \lambda * \sqrt{f'_c}$, where $\varphi = 0.75$.

For concrete stems designed per ACI 318-19, or ACI 318-25 and later with *"Use simplified Vc equation (Section 13.3.7.1c)"* enabled, the nominal shear strength follows the one-way shear provisions of §22.5 and Table 22.5.5.1 (referenced by §13.3.6.1 and §7.5.3.1):

$\varphi v_n = \varphi * 8 * \lambda_s * \lambda * (\rho_w)^{1/3} * \sqrt{f'_c}$, where $\varphi = 0.75$.

Note: The one-way shear capacity assumes no transverse reinforcement in the stem. Therefore, $A_v < A_{v,min}$, and equations (a) and (b) from Table 22.5.5.1 are not applicable.

This value is also subject to additional limits per §22.5.5.1.1:

Lower bound: $\varphi * \lambda * \sqrt{f'_c}$ (ACI 318-25 and later only)

Upper bound: $\varphi * 5 * \lambda * \sqrt{f'_c}$

Status: This indicates whether the stem design is OK for shear at the specified height.

© 2026 ENERCALC, LLC

Use simplified Vc equation (Section 13.3.7.1c): For basement walls, ACI 318-25 Section 13.3.7.1(c) allows a simplified shear equation ($V_c = 2 \lambda \sqrt{(f'_c)}\ b_w\ d$) as an alternative to the one-way shear provisions in Section 22.5.5.1. Enabling this option usually gives a higher shear capacity. For walls with higher reinforcement ratios, Section 22.5.5.1 can still produce larger values, which happens when this option is disabled.

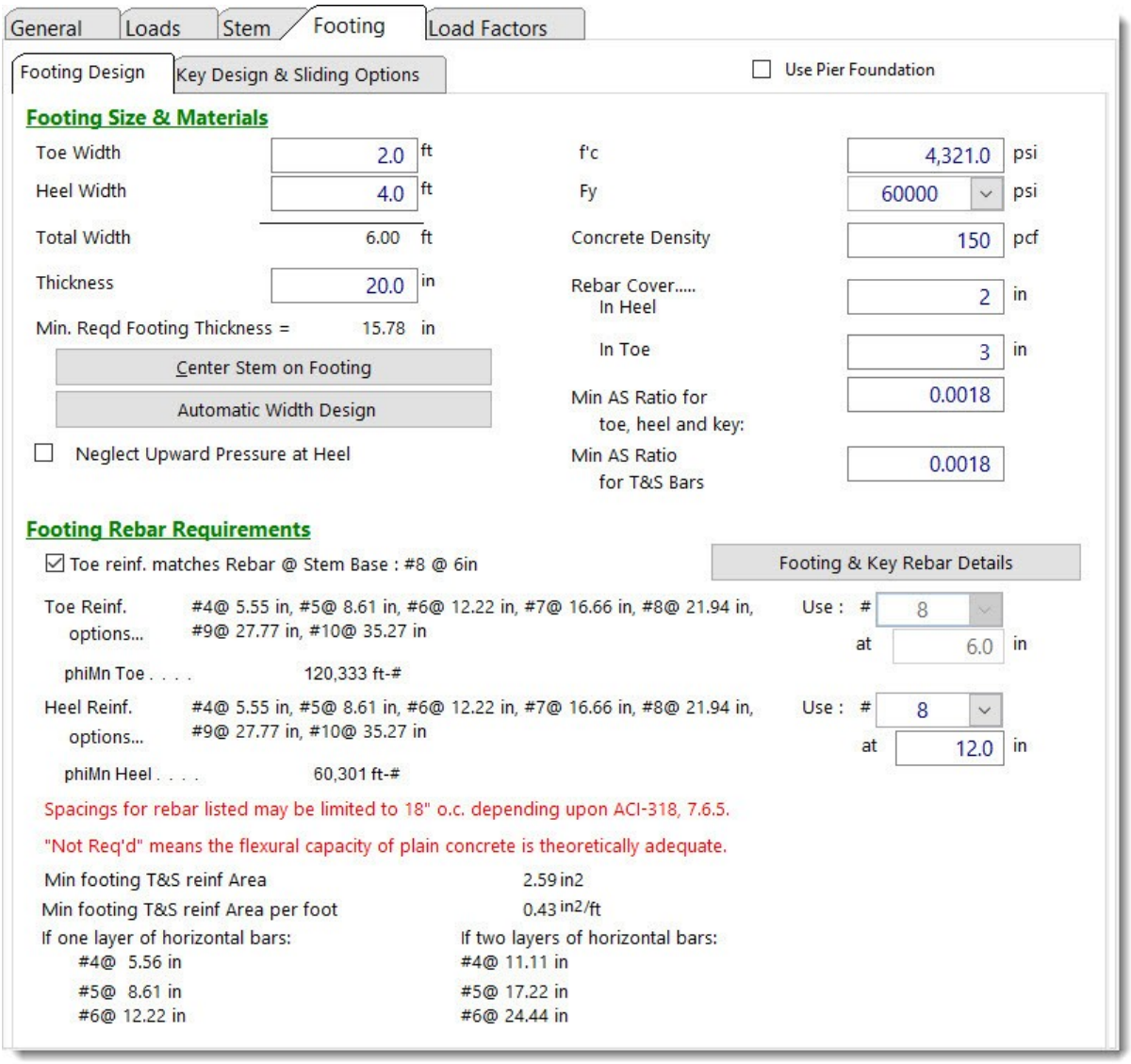
Hooked Embedment Modifiers: These are the various hooked embedment modifiers used in the hooked development length calculations. For more information, see ACI 318 Table 25.4.3.2 and Summary Section of Stem Tab - Hooked Embedment of Rebar in Concrete[1115].

Reduce hooked embedment by As req'd/As provided: When this option is selected, the program reduces the calculated hooked embedment depth by applying the ratio of required steel area to provided steel area. This setting is unavailable for ACI 318-19 and later editions as Section 25.4.10 does not permit reducing hooked development length by the ratio of ($A_{s,required}$ / $A_{s,provided}$).

© 2026 ENERCALC, LLC

13.8.1.4 Footing Tab

The Footing tab collects the data required to define the footing geometry and reinforcing, and the key geometry and reinforcing if one is present. This is also where certain design decisions can be made regarding how the program handles the sliding calculations.



Footing Design Sub-tab

Footing Size & Materials

Toe Width: This is the width of the Toe of the footing, and is measured from the front edge of the footing to the front face of the stem. Can be set to 0.00 for a property line condition. All overturning and resisting moments are taken about the bottom-front edge of the toe.

Heel Width: Distance from front face of stem to back of heel projection. If a dimension is entered that is less than the stem width at the base, the

© 2026 ENERCALC, LLC

program will automatically reset the heel dimension to at least the stem width. For a property line at the rear face of the stem, set this dimension to be equal to the stem width.

Total Footing Width: The calculated width of the footing, Toe Width + Heel Width.

Thickness: Total footing thickness, NOT including the key depth (if used). For bending and shear design of the footing, the rebar depth "d" is taken as Footing Depth - Rebar Cover - 0.5 * Bar Diameter. If footing thickness is inadequate for shear capacity a red warning indicator will appear.

The footing thickness must be greater than the hooked rebar embedment length required for the bottom stem reinforcing + rebar cover. The program adds the calculated hooked bar embedment from the Stem screen and the rebar cover you have specified for the toe (bottom of the footing, usually 3"). If the specified thickness is inadequate, increase the thickness, or change the stem dowels.

Center Stem on Footing: Clicking this bar will adjust the toe and heel widths you have entered so the stem is centered on the footing but the overall footing width remains the same.

Automatic Width Design: Clicking this button will cause the program to iterate footing widths until the soil pressure and footing strength are acceptable. ***(Note: This function does not optimize for overturning stability or sliding stability, so those will need to be checked after using this option.)*** You can select either a fixed toe or heel distance, or balance the toe and heel dimensions. You can also select whether the resultant must be within the middle third of the footing. After clicking “Design,” the widths required will be displayed. Automatic footing design is not available for Restrained Walls, Gravity Walls, or Segmental Walls.

f'c: Enter concrete compressive stress for footing.

F_y: Allowable rebar yield stress to be used for design of footing bending reinforcement.

Rebar Cover in Heel/Toe: Distance from the face of concrete to edge of rebar. The program will add 0.5 * Bar Diameter to this value and subtract the result from the footing thickness to determine the bending "d" distance.

© 2026 ENERCALC, LLC

Minimum A_s Ratio for Toe, Heel & Key:

Enter the absolute minimum permissible ratio of working steel in the footing. If the % steel required by stress analysis is less than $200/F_y$, the minimum of ($200/F_y$ -or- 1.333 * bending percentage required) is calculated and compared with the Minimum A_s% entered here, and the greater of the two is used to calculate rebar spacing requirements.

Minimum A_s Ratio for Temperature & Shrinkage Bars:

Enter the minimum steel percentage to address temperature and shrinkage requirements in the footing (commonly 0.0018 Ag for F_y= 60,000 psi). This value will be multiplied by the gross cross-sectional area of the footing to produce recommendations for longitudinal reinforcing in the footing (perpendicular to a section view of the footing).

Neglect Upward Pressure at Heel:

For heel calculations you may choose to neglect the upward soil pressure, typically resulting in greater heel moment. If this box is checked the M_u for upward loads will be zero.

Footing Rebar Requirements

Rebar at Stem Base: This is a reminder of the size and spacing of the reinforcing used at the bottom of the stem, to make it easier to select toe reinforcing to match (toe reinforcing is usually the bottom stem dowel bars bent toward the toe).

© 2026 ENERCALC, LLC

Toe Reinforcing Options: This list provides options for reinforcing sizes and spacing for the toe bars (located in the bottom of the footing). Typically the toe bars are extensions of the stem dowels, which are bent out toward the toe. Therefore, you will probably just want to verify that the stem dowel bar size and spacing would also be adequate for use in the toe.

Toe shear stress is calculated using the sum of upward and downward forces acting on the toe. The summation only considers those forces that are farther than the d dimension from the face of the stem.

NOTE: If structural plain concrete is appropriate in the designer's judgment, the rebar size can be set to "None". This will trigger the program to calculate the moment capacity for structural plain concrete using the footing modulus of rupture $F_r = \Phi 5\lambda(f'_c)^{1/2}$ times the section modulus, with 2" deducted from the thickness for crack allowance per code. The design status will report on the adequacy of the design. However, the designer may consider it prudent to add reinforcing regardless of the theoretical flexural capacity. ***Be sure to check code limitations on the use of structural plain concrete.***

Heel Reinforcing Options: This list provides options for acceptable sizes and spacing for heel bars (located in the top of the footing). It is desirable to select a spacing that is modular with the stem dowel bars for ease of construction. Note: The program does not calculate the heel bar development length inward from the back face of the stem (where the moment is maximum). You can refer to Appendix B for development lengths in concrete, which can be adjusted for the stress level in the heel bars. When detailing footing reinforcing it is important to consider and specify development lengths for both toe and heel bars.

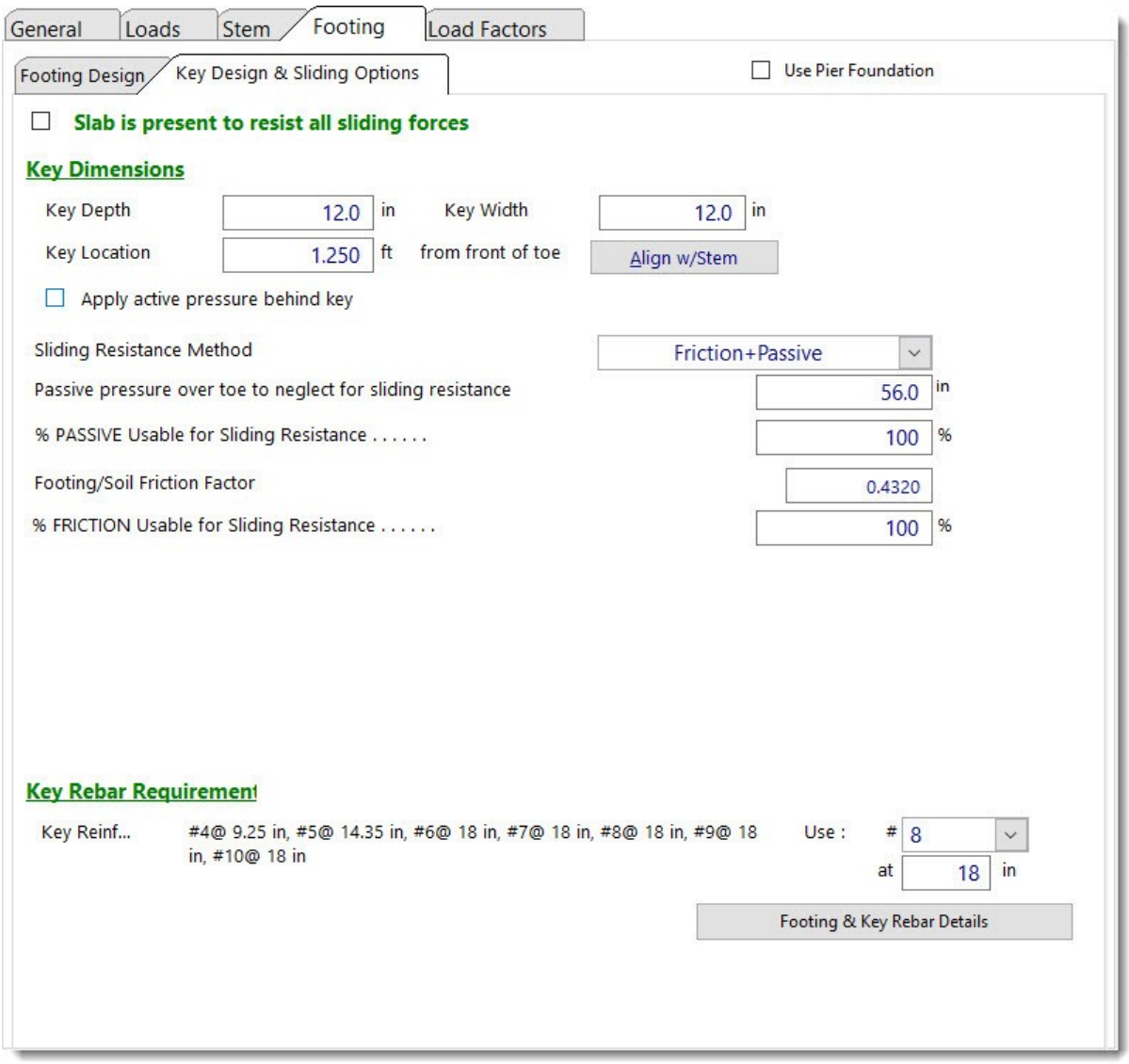
NOTE: If structural plain concrete is appropriate in the designer's judgment, the rebar size can be set to "None". This will trigger the program to calculate the moment capacity for structural plain concrete using the footing modulus of rupture $F_r = \Phi 5\lambda(f'_c)^{1/2}$ times the section modulus, with 2" deducted from the thickness for crack allowance per code. The design status will report on the adequacy of the design. However, the designer may consider it prudent to add reinforcing regardless of the theoretical flexural capacity. ***Be sure to check code limitations on the use of structural plain concrete.***

© 2026 ENERCALC, LLC

Rebar Selections: Use these three size and spacing entries to select your toe, heel, and if applicable, key reinforcing. The "Max @" message tells you the maximum spacing allowed for the bar selected.

Key Design & Sliding Options

This screen is used to indicate whether a key is to be used, and if so, specify its dimensions. This screen also collects information about the design intent for the sliding check, and presents a summary of the sliding forces.



Slab is present to resist all sliding forces: Provides a way to communicate to the program that sliding is not a design consideration, because in the designer's judgment, sliding is completely precluded, such as by a slab on grade on the toe side of the wall that prevents sliding altogether. If this option is selected, the

© 2026 ENERCALC, LLC

lateral sliding force is displayed for checking the resistance offered by the slab, and the slab is assumed to be at the top of the footing, but not higher.

Key Dimensions

Key Depth: Depth of the key below the bottom of footing. The bottom of the key is used as the lower horizontal plane for determining the size of the passive pressure block from the soil in front of the footing. Adjust this depth so the sliding safety factor is acceptable, but not less than 1.5.

Key Width: Width of the key, measured along the same direction as the footing width. This is usually 12"-14", but generally not less than one-half the key depth so flexural stresses in the key are usually minimal.

Key Location: Enter the distance from the front edge of the toe to the front of the key. Do not enter a distance greater than the footing width minus key width.

Align with Stem: Click this button to align the front edge of key with the front of the stem. If the key width is then set to a value that is reasonably close to the stem width, the stem bars may be able to be extended down into the key to facilitate rebar development.

Apply active pressure behind key: When this option is selected, the program will consider the driving force to extend all the way to the bottom of the key. If this option is NOT selected, then the driving force will not extend below the bottom of the footing.

Note: IBC 2015 was worded in such a way as to confuse this issue and imply that it was a requirement that active pressure must extend to the bottom of any key. The actual intent was expanded upon in subsequent documents to clarify that the intent was to require active pressure to extend to the bottom of a key in situations where it could be reasonably anticipated that the soil on the low side of the wall (providing passive resistance) might be removed during the service life of the wall, such as in a scour condition. This verbiage was revised in IBC 2018 to remove the ambiguity.

Sliding Resistance Method: Enter whether sliding resistance will be by friction and passive pressure or by cohesion and passive pressure.

Passive pressure over toe to neglect for sliding resistance: Since the soil over the toe of the footing may be loose and uncompacted, it

© 2026 ENERCALC, LLC

may have little or no passive resistance. This entry gives the option to neglect some portion of the Height of Soil Over Toe entered in the General tab. You can neglect the soil over toe plus the footing thickness, if desired.

% Passive Usable for Sliding Resistance: Enter a value from zero to 100% to indicate the percentage of the calculated passive pressure that will be used as resistance in the sliding calculation. This may be a stated restriction in the geotechnical report.

Footing/Soil Friction Factor: Enter the friction factor here. It usually varies from 0.25 to 0.45, and is generally provided by the geotechnical engineer.

% Friction Usable for Sliding Resistance: Enter a value from zero to 100% to indicate the percentage of the calculated friction force that will be used as resistance in the sliding calculation. This may be a stated restriction in the geotechnical report.

Summary of Sliding Forces

Lateral Force @ Base of Footing: This is the total lateral force against the stem and footing which causes the wall to slide and which must be resisted.

Less Passive Pressure Force: This uses the allowable passive pressure in pcf and the available depth ("footing thickness" plus "soil above toe" less "height to neglect") multiplied by the "percent usable" you specified to compute the total passive resistance. Weight due to toe surcharge, if applicable, will also be incorporated into the calculation of the passive force. If a key is used, the available passive pressure depth will be to the bottom of the key.

Less Friction Force: This is the total vertical reaction multiplied by the friction factor, and then multiplied by the "percent usable" you specified.

Added Resisting Force Required: If this value is indicated as 0.0 lbs., then there is no requirement for additional resisting force in order to

© 2026 ENERCALC, LLC

achieve a static balance of forces, but it does not necessarily mean that there is an adequate factor of safety against sliding. Watch the Sliding Ratio on the Results tab, Summary sub-tab for an adequate value (usually 1.5). Consider adding a key or modifying footing geometry if required.

Added Resisting Force Required for 1.5:1 Factor of Safety:

This is the additional resisting force that would be required in order to achieve a 1.5 safety factor. If this value is indicated as 0.0 lbs., then the Siding Ratio is already 1.5 or greater.

Key Rebar Requirement

Key Reinforcing: This area indicates the permissible spacing values for a variety of logical rebar sizes, and allows the user to specify the size and spacing of the rebar in the key. The program assumes that key rebar is centered on the width of the key. This is done for a couple reasons. First, placement of key reinforcement is generally less than precise in typical configurations. Second, the intent of a key is to develop some additional sliding resistance through shear. It is not to create keys so deep that they start to develop significant flexural stresses. If that much key is required, it's already asking too much of a key.

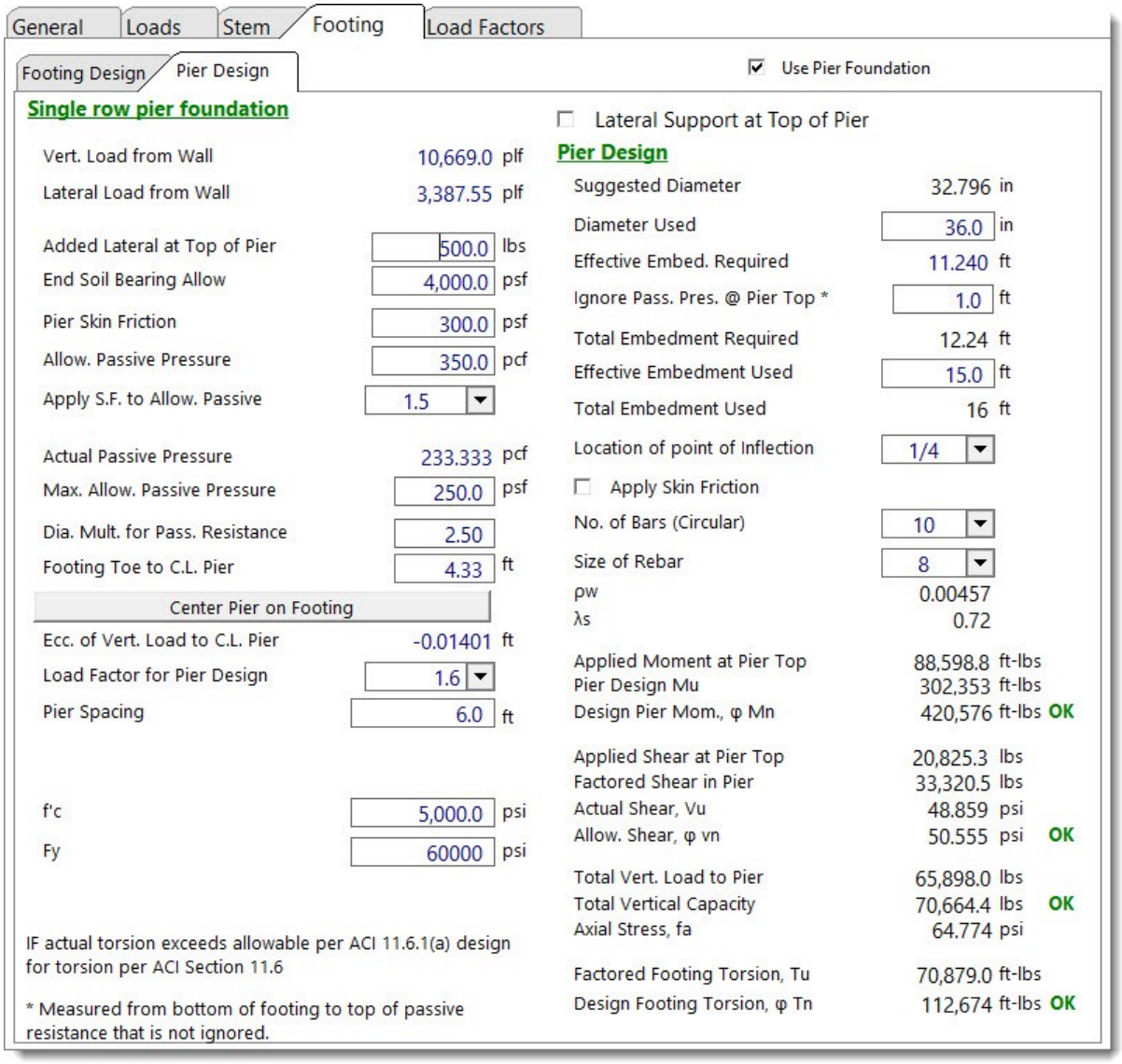
Sliding Factor of Safety: This reports the ratio of passive and friction resistance to the total lateral force. This should be at least 1.5, or 1.1 if seismic is activated.

NOTE: If lateral restraint is provided by an abutting floor slab (by checking the "Slab is present..." box), the sliding factor of safety will not be displayed, but the “Lateral Force @ Base of Footing” will be displayed for checking restraint adequacy of the slab.

© 2026 ENERCALC, LLC

Pier Design

Checking the Use Pier Foundation checkbox (on the sub-tabs under the Footing tab) will replace the Key Design & Sliding Options sub-tab with the Pier Design sub-tab as shown below:



This allows you to use drilled cast-in-place concrete piers spaced in a single row along the length of the wall. The default is without lateral support at the footing level. If lateral support is available, such as an abutting slab at the footing level, check the box labeled "Lateral support at Top of Pier". The Key Dimensions & Sliding tab is not applicable when piers are used so the Key Dimensions & Sliding Options tab is not displayed when the Use Pier Foundation checkbox is checked. However, the Footing Design tab does remain active, so you can adjust the footing dimensions as necessary for the piers, and adjust as needed for torsion resistance (see below).

© 2026 ENERCALC, LLC

Lateral Support at Top of Pier: Provides a way to specify that there is lateral restraint at or near the top of the pier. If this option is checked, the program will offer a related item named "Assumed Fixity Below Embed"

Vert. Load from Wall, plf: Displays the total vertical load imposed upon the piers from the wall above, including the footing weight. It matches the total vertical load from the Resisting Moment summary.

Lateral Load from Wall, plf: Displays the net sliding force and matches the total force shown on the Overturning Moments summary for the wall.

Added Lateral at Top of Pier, lbs: The geotechnical engineer may recommend an added lateral force at or near the top of the pier (sometimes termed “creep”). This may be a triangular force but for simplicity it is assumed to act at the top of the pier.

End Soil Bearing Allow, psf: Allowable end bearing pressure at bottom of pier.

Pier Skin Friction, psf: If applicable, enter the allowable skin friction on the pier for added vertical load capacity. This may require conversion from a friction angle.

Allow. Passive Pressure, pcf: This is used to define the variation in allowable passive pressure with depth.

Apply S.F. to Allow. Passive: Allows the user to use a drop-down list box to select a safety factor that will be applied to the calculated passive pressure.

Actual Passive Pressure, pcf: Reports the value of Allowable Passive Pressure in pcf divided by the safety factor selected above.

Max. Allow. Passive Pressure, psf: Specifies the upper limit on the allowable passive pressure. The allowable passive pressure will increase with depth until it reaches this value, at which point the allowable passive pressure will remain constant at this value.

Dia. Mult. for Pass. Resistance: The geotechnical engineer may permit a multiplier to the diameter for greater effective passive resistance. The default is 1.0.

Footing Toe to CL Pier, ft: Distance from the toe to the centerline of the pier.

© 2026 ENERCALC, LLC

Center Pier on Footing (button): Convenient way to automatically center the pier on the footing.

Eccentricity of Vertical Load to CL Pier, ft: Distance from the centerline of the pier to the resultant vertical load.

Load Factor for Pier Design: Select a single load factor that will be applied to all pier loads to perform the reinforced concrete design.

Pier Spacing, ft: Center to center spacing of piers.

f'c, psi: Specified compressive strength of concrete.

Fy, psi: Yield strength of rebar.

Suggested Diameter, in: Diameter required based upon applied Vertical Load and the allowable end soil bearing pressure.

Diameter Used, in: If skin friction is used (activated by checkbox below) the diameter can be adjusted provided Total Bearing Capacity exceeds Total Vert. Load to Pier.

Effective Embed. Required, ft: This uses the “pole embedment” equations per Section 1807.3 of IBC to determine the required pier embedment depth based upon the passive pressure entered and the applied moment to pier. The embedment depth will vary depending upon whether the checkbox for lateral support at top is checked.

Ignore Pass. Pres. @ Pier Top, ft: Since the soil near the top of a drilled pier may be disturbed and uncompacted, it may have little or no passive resistance. This entry gives the option to neglect the passive pressure over the specified height at the top of the drilled pier.

Total Embedment Required, ft: Displays the sum of "Effective Embedment Required" plus "Ignore Passive Pressure from Pier Top".

Effective Embedment Used, ft: Input a depth of embedment considered to be effective below the section where passive pressure is being ignored.

Total Embedment Used, ft: Displays the sum of "Effective Embedment Used" plus "Ignore Passive Pressure from Pier Top".

© 2026 ENERCALC, LLC

Location of point of inflection: Use the drop-down list box to select the ratio of depth-to-inflection to effective embedment depth. Tests suggest 1/6 is reasonable; 1/3 is conservative. This will be used to calculate the maximum moment applied to pier. The resulting length will be measured below the zone where passive pressure is ignored (if any).

Apply Skin Friction (with option to ignore some length of skin friction): Check this if skin friction is to be used to increase vertical capacity of pier. If selected, there is an entry for depth to be ignored for skin friction.

No. of Bars (circular): Select the number of bars. They are assumed to be in a circular pattern.

Size of Rebar: Select size of bars to use in the circular pattern.

ρw: Pier reinforcement ratio calculated as A_s / (b_w d), where:

- A_s = Sum of the areas of longitudinal bars located more than two-thirds of the overall member depth away from the extreme compression fiber per ACI 318-19 R22.5.2.2. Due to the uncertainty in how the longitudinal bars are arranged in relation to the extreme compression fiber, two bar arrangements are examined: one with the bars in a circular pattern without rotation, and another where the circular pattern is rotated by an angle of θ = 0.5 * (360 degrees / "No. of Bars"). The arrangement that results in the fewest longitudinal bars beyond two-thirds of the distance from the extreme compression fiber is used in the calculation for A_s.
- b_w = "Diameter Used" per ACI 318-19 §22.5.2.2.
- d = 0.8 * "Diameter Used" per ACI 318-19 §22.5.2.2.

λs: Size effect modification factor per ACI 318-19 §22.5.5.1.3

Applied Moment at Pier Top, ft-lbs: Displays the wall overturning moment multiplied by the pier spacing.

© 2026 ENERCALC, LLC

Pier Design Mu, ft-lbs: This is the total factored design moment applied to the pier.

Design Pier Mom., φ Mn, ft-lbs: Displays the design moment capacity of the pier using the strength values input and a phi factor of 0.90. This uses the Whitney Approximation method which is slightly conservative.

Applied Shear at Pier Top, lbs: Displays the Lateral Load from Wall multiplied by the pier spacing.

Factored Shear in Pier, lbs: Displays the total factored design shear applied to the pier. It includes lateral load from the wall, and additional shear due to the pier reacting out the applied moment.

Actual Shear, V_u, psi: Displays factored shear stress determined using Whitney Equivalent Rectangular Section.

Width = b = 0.8 * Diam.
Area of Whitney Equivalent Rectangular Section = Area of circular pier
H = Area / b
d = 0.67 * H
Aeff = b * d

Allow. Shear, φ v_n, psi: For piers designed per ACI 318-14 and earlier, the nominal shear strength is calculated as follows:

$\varphi v_n = \varphi * 2 * \lambda * \sqrt{(f'_c)}$, where $\varphi = 0.75$.

For piers designed per ACI 318-19 and later, the nominal shear strength follows the one-way shear provisions of §22.5 and Table 22.5.5.1 (referenced by §7.5.3.1):

$\varphi v_n = \varphi * 8 * \lambda_s * \lambda * (\rho_w)^{1/3} * \sqrt{(f'_c)}$, where $\varphi = 0.75$.

Note: Piers are assumed to be unreinforced for shear. Therefore, $A_v < A_{v,min}$, and equations (a) and (b) from Table 22.5.5.1 are not applicable.

This value is also subject to additional limits per §22.5.5.1.1:

Lower bound: $\varphi * \lambda * \sqrt{(f'_c)}$ (ACI 318-25 and later only)

Upper bound: $\varphi * 5 * \lambda * \sqrt{(f'_c)}$

© 2026 ENERCALC, LLC

Total Vert. Load to Pier, lbs: Displays the Vertical Load from Wall multiplied by the pier spacing.

Total Vertical Capacity, lbs: This combines both end bearing capacity and skin friction, as applicable.

Axial Stress, fa, psi: This is the total vertical load / pier area. This is for reference only since it is not considered a critical design consideration.

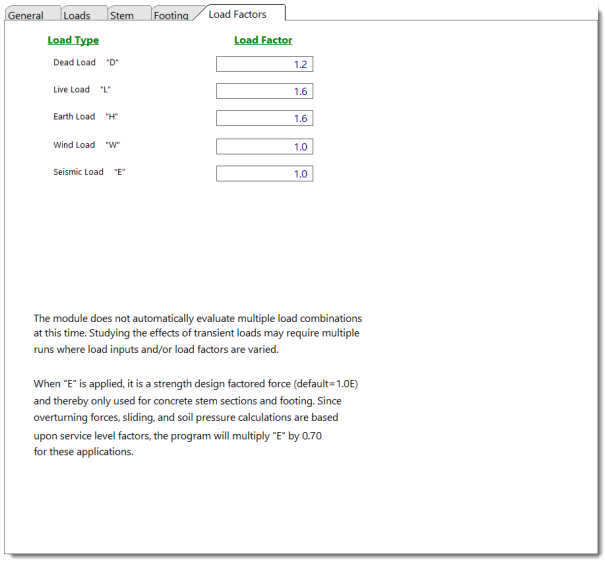
Footing Torsion, Tu, ft-lbs: Displays factored torsional force in footing, which is calculated as moment from wall multiplied by one-half pier spacing.

Footing Torsion Allow., φ Tn, ft-lbs: Displays torsional design strength of the footing based on Section 22.7.4 from ACI 318-14 or -19.

For more information on pier foundation design see *Basics of Retaining Wall Design*, 11th Edition

13.8.1.5 Load Factors

This tab allows the code-specified load factors to be reviewed and edited if necessary.



© 2026 ENERCALC, LLC

Load Type / Load Factors

For each type of load (DL, LL, etc) the default factor will be displayed. ***Remember to review these factors for each new design since they are editable.***

Strength-Level Checks
In this tab, the user defines Load Factors, which ENERCALC SEL then uses to generate a single factored load combination for Strength Design checks. This combination is used in:

- Concrete stem design
- Concrete footing design
- Masonry design when SD (Strength Design) is selected

For example, if the following load factors are input: D = 1.2 | L = 1.6 | H = 1.6 | W = 1.0 | E = 1.0
ENERCALC SEL will apply this combination: 1.2D + 1.6L + 1.6H + 1.0W + 1.0E
This combination is applied automatically for concrete stem and footing design, and for masonry design when SD (Strength Design) is selected.

Service-Level Checks
Service-level checks include:

- Soil bearing pressure
- Sliding
- Overturning
- Masonry stem design when ASD (Allowable Stress Design) is selected

ENERCALC SEL evaluates these checks using a single service-level load combination. The program internally converts strength-level wind and seismic loads to service-level equivalents using the following factors: Seismic (E): 0.7, Wind (W): 0.6, All other loads (D, L, H): 1.0
This results in the following combination: 1.0D + 1.0L + 1.0H + 0.6W + 0.7E

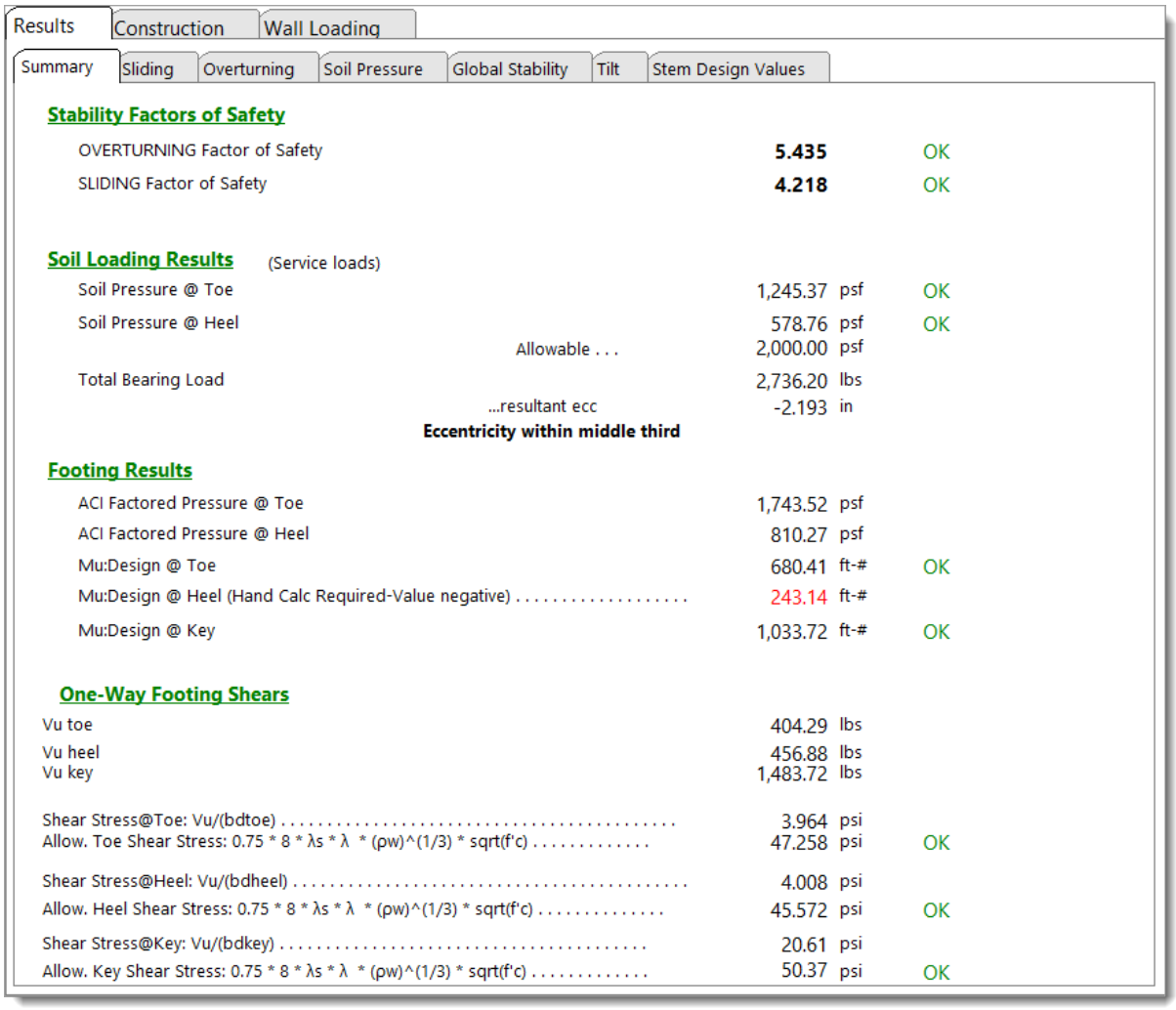
Special Consideration: Wind and Seismic Loads
The building code does not require simultaneous consideration of wind and seismic loads. Therefore, ENERCALC SEL provides alerts when both are present in a given calculation. These alerts indicate that you may need to separate the calculations to properly address independent wind and seismic conditions.

13.8.1.6 Results Tabs

© 2026 ENERCALC, LLC

Summary

This screen summarizes the footing/soil bearing results obtained from previous screens, including a message whether the resultant is within or outside the middle third of the footing.



This is not an input screen. It's strictly for your review, and it does not appear if a pier is specified.

Stability Ratios: These are displayed for both overturning and sliding.

Soil Loading Results

Soil Pressure @ Toe and Heel: This is the resulting soil pressure for both the toe and heel based on service loads. If the eccentricity is outside the middle third, the heel pressure will show 0.00, and the program will calculate the toe pressure assuming no tension at the heel.

© 2026 ENERCALC, LLC

Allowable Soil Pressure: This is for reference as entered on the General tab.

Total Bearing Load: This is the sum of all vertical forces.

Resultant Eccentricity: Distance from center of footing to the resultant of the soil pressure distribution.

Eccentricity Within/Outside Middle Third: If the eccentricity is greater than one-sixth the footing width, the resultant is outside the middle third. (If outside the middle third, the program computes the toe soil pressure assuming no tension at the heel.)

Footing Results

ACI Factored Soil Pressure @ Toe and Heel: ACI load factors are applied to all loads to determine total vertical load for soil pressure used in calculating footing moments and shears. This load is then applied at the same eccentricity calculated for service load soil pressures to yield the factored soil pressures for footing design using LRFD design principles.

Note that since factored vertical loads are applied at the non-factored resultant eccentricity, a true 1.6 load factor applied to lateral earth pressure is not used for footing design. ACI load factors are intended to give conservative results for design. Calculation of a factored load eccentricity would give soil pressure diagrams that would not always represent the actual soil pressure distribution under the footing, and yield unreasonable results. Factored lateral earth pressure, however, is always used for concrete stem design.

M_u Design @ Toe/Heel: These are the factored moments at face of stem for toe and heel moments. Since neither can be greater than the stem base moment (factored if concrete stem), the latter may govern. These moments will be reduced if you choose to neglect the upward soil pressure on the Footing tab. A message will indicate which controls.

The program assumes that the tension face and associated reinforcement will be at the bottom face of the toe and the top face of the heel. If the loading does not meet these conditions, the Mu value will be displayed in red, and a warning message

© 2026 ENERCALC, LLC

will indicate: "Hand Calc Required - Value Negative." These conditions are beyond the scope of this module and require a separate hand calculation to verify adequacy.

Shear @ Toe and Heel: These items report the factored shear stress from the one-way action in the footing. The toe shear stress is calculated at a distance "d" (footing thickness - rebar cover) from the face of the bottom stem segment. (If "d" is greater than the projecting toe length, then the one-way toe shear is reported as zero.) The heel shear stress is calculated at the face of the stem.

Allowable Footing Shear: <u>For footings designed per ACI 318-14 and earlier, and ACI 318-25 and later:</u>

The nominal shear strength follows the one-way shear provisions of ACI 318-14 §22.5.5.1 / ACI 318-25 §13.2.6.2:

$\varphi v_n = \varphi * 2 * \lambda * \sqrt{(f'_c)}$, where $\varphi = 0.75$.

<u>For footings designed per ACI 318-19:</u>

The nominal shear strength follows the one-way shear provisions of §22.5 and Table 22.5.5.1 (referenced by §7.5.3.1):

$\varphi v_n = \varphi * 8 * \lambda_s * \lambda * (\rho_w)^{1/3} * \sqrt{(f'_c)}$, where $\varphi = 0.75$.

Note: The one-way shear capacity assumes no transverse reinforcement in the footing. Therefore, $A_v < A_{v,min}$, and equations (a) and (b) from Table 22.5.5.1 are not applicable. It is also acceptable to neglect the size effect factor (λ_s) for one-way shallow foundations per §13.2.6.3, so λ_s is taken as 1.0.

This value is also subject to additional limits per §22.5.5.1.1:

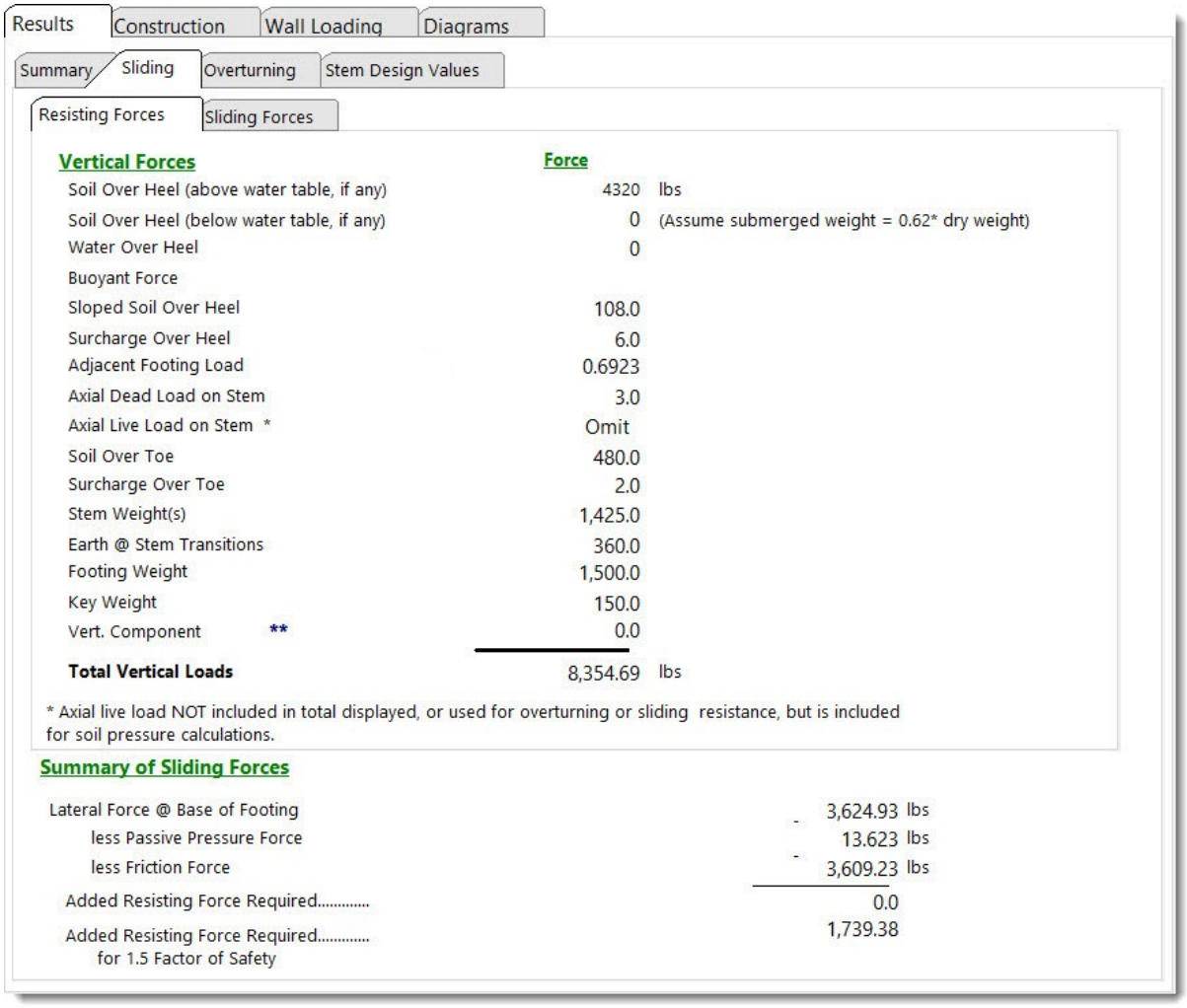
Lower bound: $\varphi * \lambda * \sqrt{(f'_c)}$

Upper bound: $\varphi * 5 * \lambda * \sqrt{(f'_c)}$

© 2026 ENERCALC, LLC

Sliding > Resisting Forces

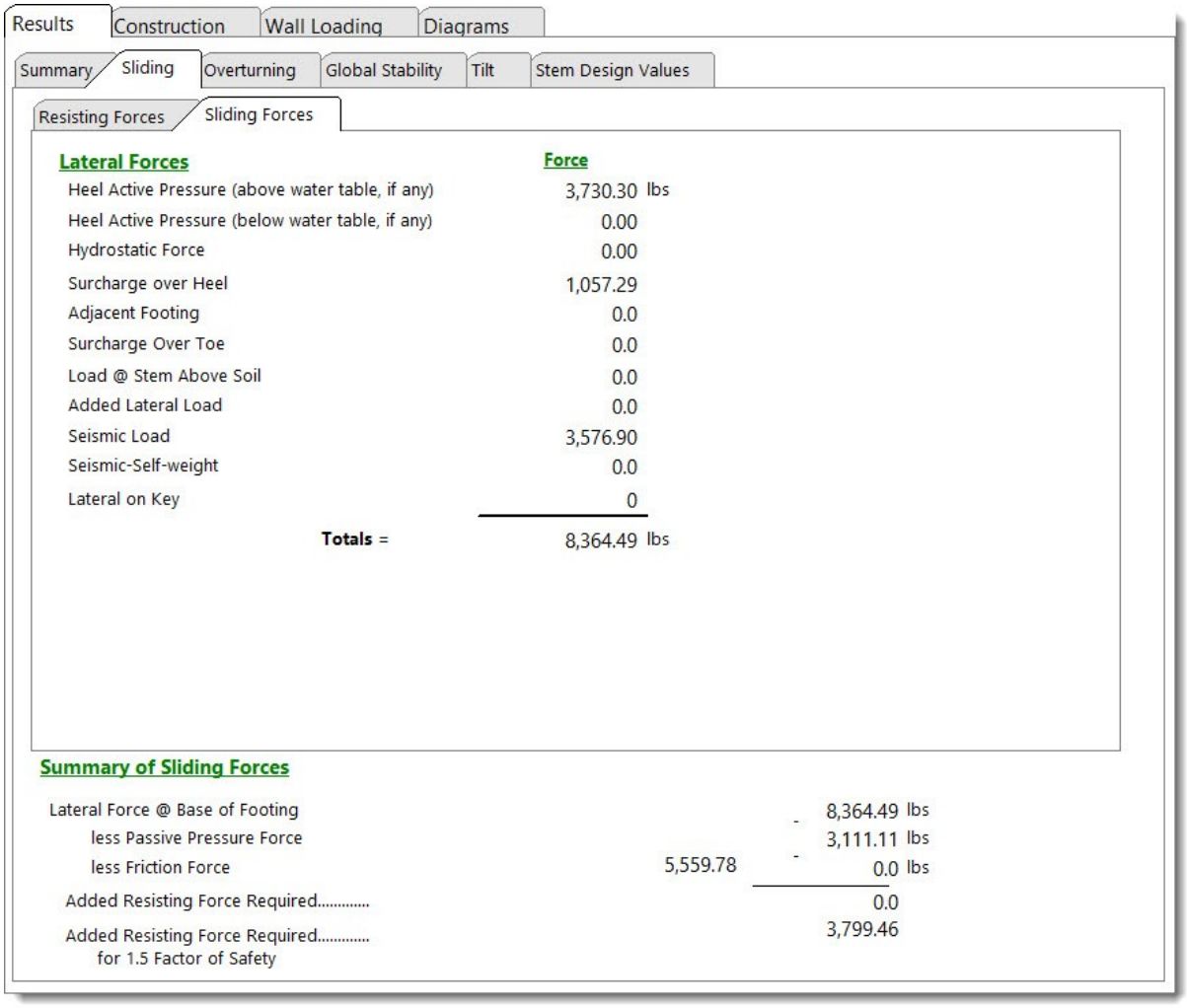
This screen presents in tabular form each component contributing to resistance to sliding.



© 2026 ENERCALC, LLC

Sliding > Sliding Forces

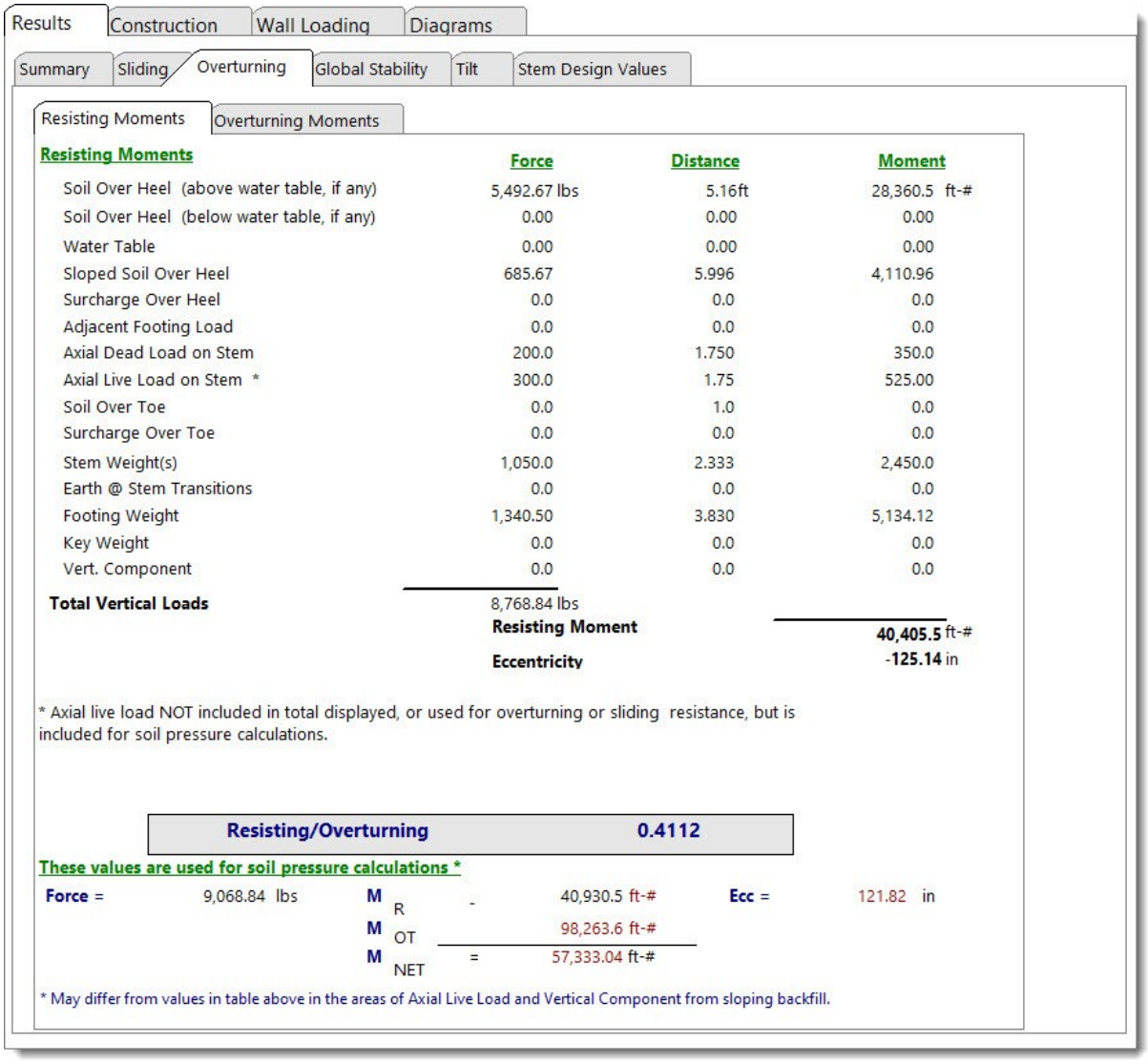
This screen presents in tabular form each component contributing to sliding force.



© 2026 ENERCALC, LLC

Overturning > Resisting Moments

This screen presents in tabular form each component contributing to resisting moment, giving weights and moment arms from the front edge of the toe to the centroid of the force.



Results | Construction | Wall Loading | Diagrams

Summary | Sliding | Overturning | Global Stability | Tilt | Stem Design Values

Resisting Moments | Overturning Moments

Resisting Moments	Force	Distance	Moment
Soil Over Heel (above water table, if any)	5,492.67 lbs	5.16ft	28,360.5 ft-#
Soil Over Heel (below water table, if any)	0.00	0.00	0.00
Water Table	0.00	0.00	0.00
Sloped Soil Over Heel	685.67	5.996	4,110.96
Surcharge Over Heel	0.0	0.0	0.0
Adjacent Footing Load	0.0	0.0	0.0
Axial Dead Load on Stem	200.0	1.750	350.0
Axial Live Load on Stem *	300.0	1.75	525.00
Soil Over Toe	0.0	1.0	0.0
Surcharge Over Toe	0.0	0.0	0.0
Stem Weight(s)	1,050.0	2.333	2,450.0
Earth @ Stem Transitions	0.0	0.0	0.0
Footing Weight	1,340.50	3.830	5,134.12
Key Weight	0.0	0.0	0.0
Vert. Component	0.0	0.0	0.0
Total Vertical Loads	8,768.84 lbs		
	Resisting Moment		**40,405.5** ft-#
	Eccentricity		**-125.14** in

* Axial live load NOT included in total displayed, or used for overturning or sliding resistance, but is included for soil pressure calculations.

Resisting/Overturning **0.4112**

These values are used for soil pressure calculations *

Force = 9,068.84 lbs **M** R - 40,930.5 ft-# **Ecc** = 121.82 in

M OT 98,263.6 ft-#

M NET = 57,333.04 ft-#

* May differ from values in table above in the areas of Axial Live Load and Vertical Component from sloping backfill.

Resisting/Overturning ratio is displayed.

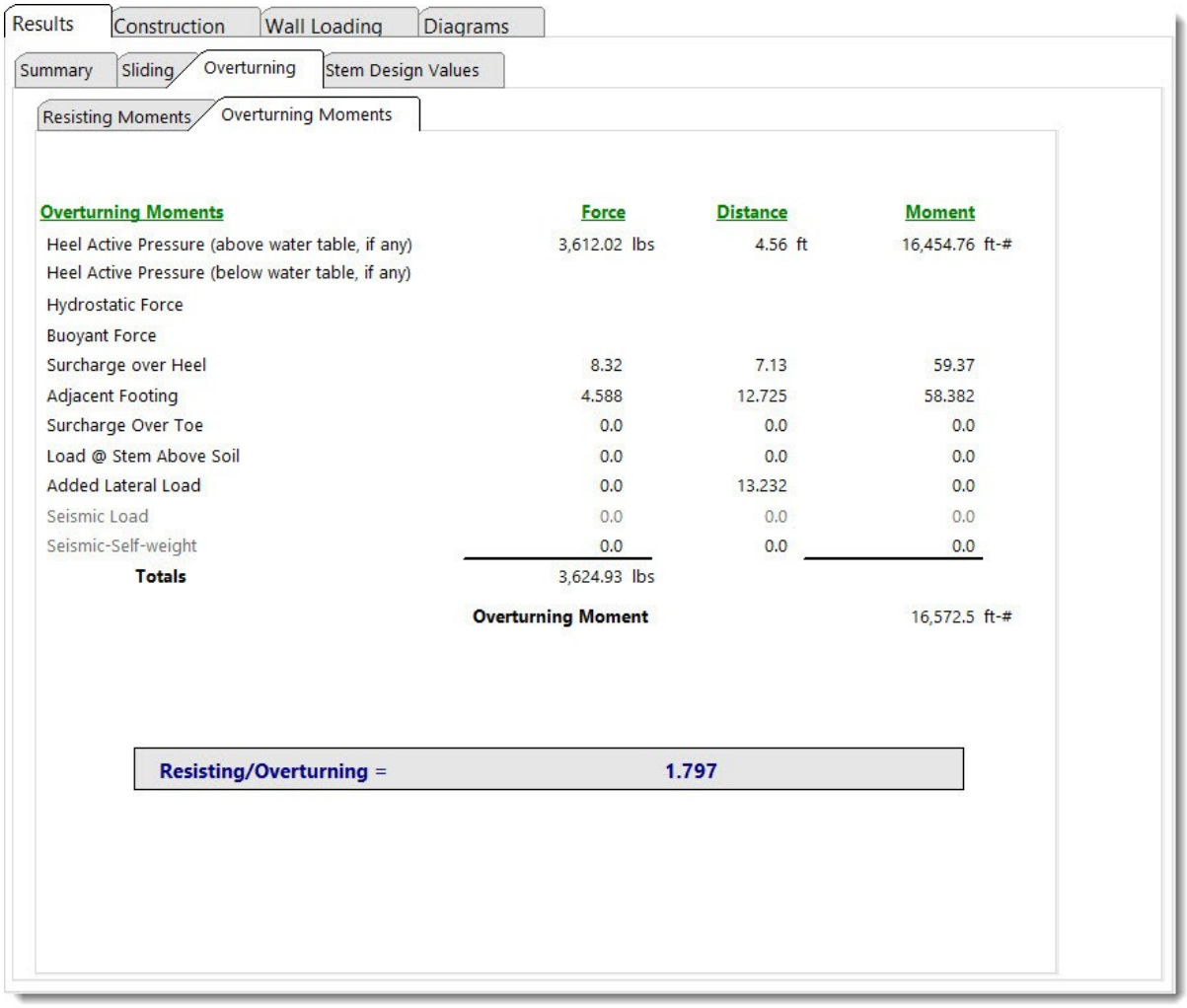
The force and moment displayed at the bottom accounts for deduction of effect of vertical component, if box on the General tab has been checked.

For calculating the vertical component, if checked on the General tab, and if the EFP method was chosen, the program will back-solve using the Rankine formula to obtain an equivalent internal friction angle.

© 2026 ENERCALC, LLC

Overturning > Overturning Moments

This screen presents in tabular form each component acting horizontally to overturn the wall/footing system. The centroid of each force is multiplied by its distance up from the bottom of the footing. The Heel Active Pressure includes the effect of surcharges and water table, if applicable, and its Distance is to the centroid of the total lateral force.



Overturning Moments	Force	Distance	Moment
Heel Active Pressure (above water table, if any)	3,612.02 lbs	4.56 ft	16,454.76 ft-#
Heel Active Pressure (below water table, if any)			
Hydrostatic Force			
Buoyant Force			
Surcharge over Heel	8.32	7.13	59.37
Adjacent Footing	4.588	12.725	58.382
Surcharge Over Toe	0.0	0.0	0.0
Load @ Stem Above Soil	0.0	0.0	0.0
Added Lateral Load	0.0	13.232	0.0
Seismic Load	0.0	0.0	0.0
Seismic-Self-weight	0.0	0.0	0.0
Totals	3,624.93 lbs		
	Overturning Moment		16,572.5 ft-#

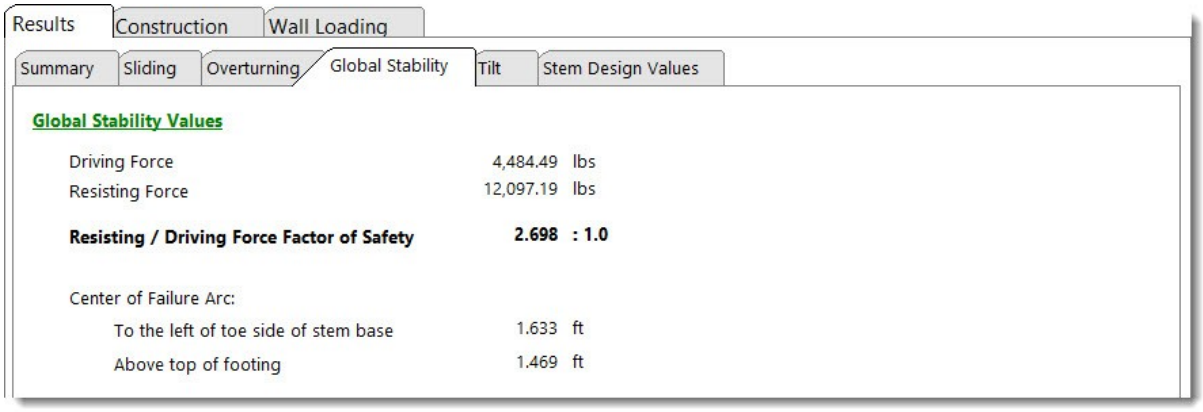
Resisting/Overturning = **1.797**

The total overturning moment is displayed along with the Resisting/Overturning ratio.

© 2026 ENERCALC, LLC

Global Stability

This screen presents the driving force tending to cause a global stability (scooping) failure, the force tending to resist the failure, and the ratio.

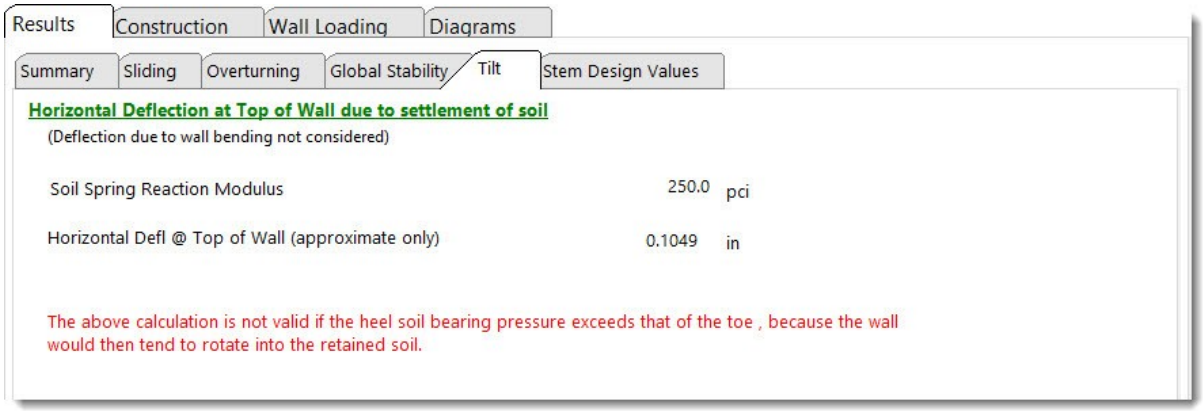


The Global Stability check follows the procedure in Principles of Foundation Engineering, 2nd Edition, Das, where various circular failure planes are tested. Each circle is broken into slices of soil behind the wall and in front of the wall. The weight of the soil slices is used to calculate a total driving moment, tending to upset the stability. And it is also used to determine tangential frictional forces on each slice, tending to restore stability.

This tab will **not** be displayed if a water table is specified, or if a pier foundation is specified.

Wall Tilt

This computes the horizontal displacement at the top of a wall caused by rotation due to compression of the soil under the toe.



You must enter the modulus of subgrade reaction on the General tab. The program divides the soil bearing stress in psi by the soil modulus (psi/inch) to quantify the displacement at the footing. Then, assuming the wall and footing are rigid, the program determines the horizontal displacement at the top of the wall based on the amount of rotation experienced at the footing.

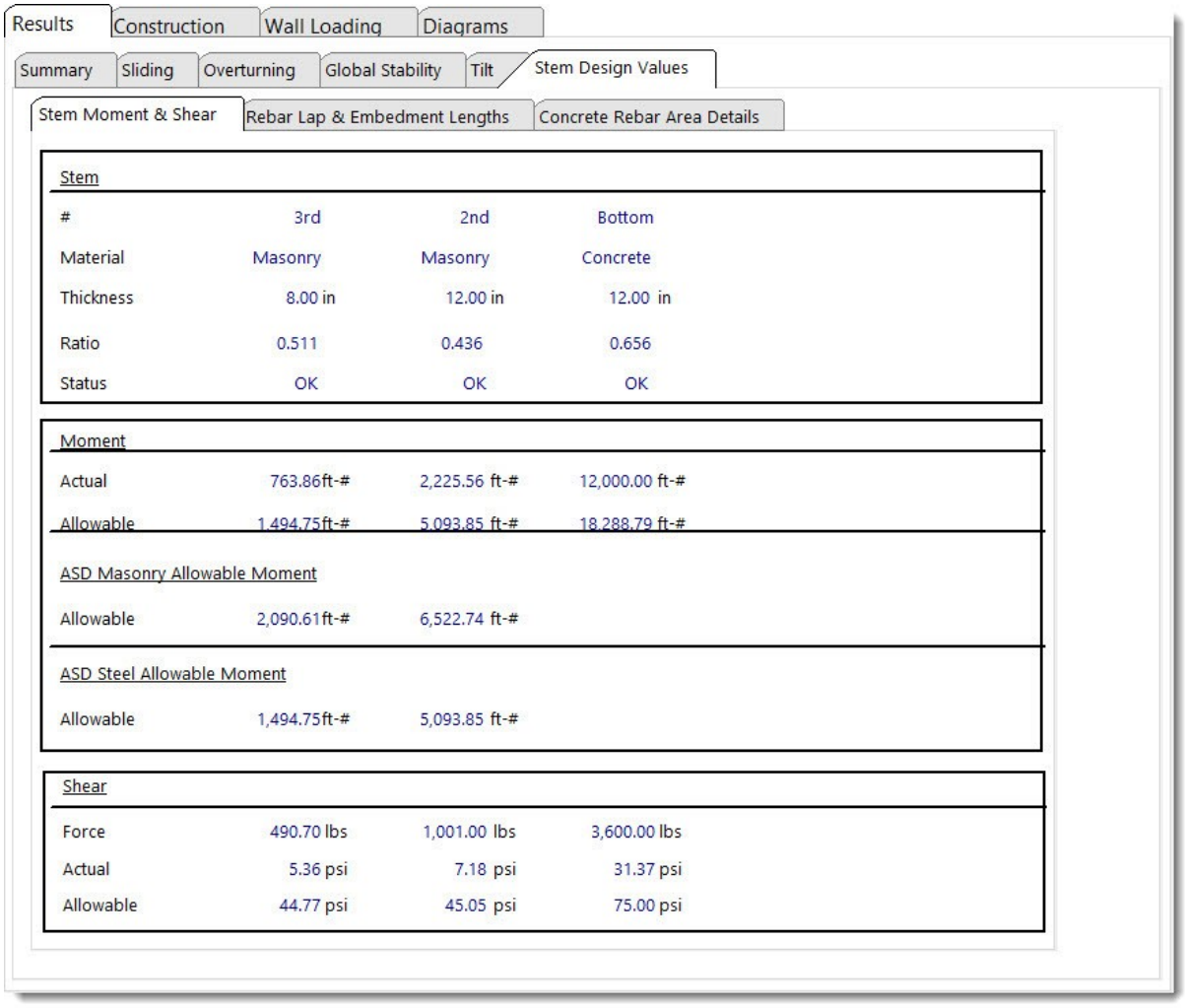
© 2026 ENERCALC, LLC

Note: This is approximate due to variation in soil pressure under the footing, and does not include deflection of the stem due to lateral earth pressures. (The latter is usually less than the "tilt" deflection, and if desired, must be done by hand calculation, requiring investigation of cracked and uncracked moments of inertia.)

To mobilize the active pressure in retained earth, it is often considered that the deflection at top must be greater than or equal to 0.005 x H_{total}.

Stem Design Values

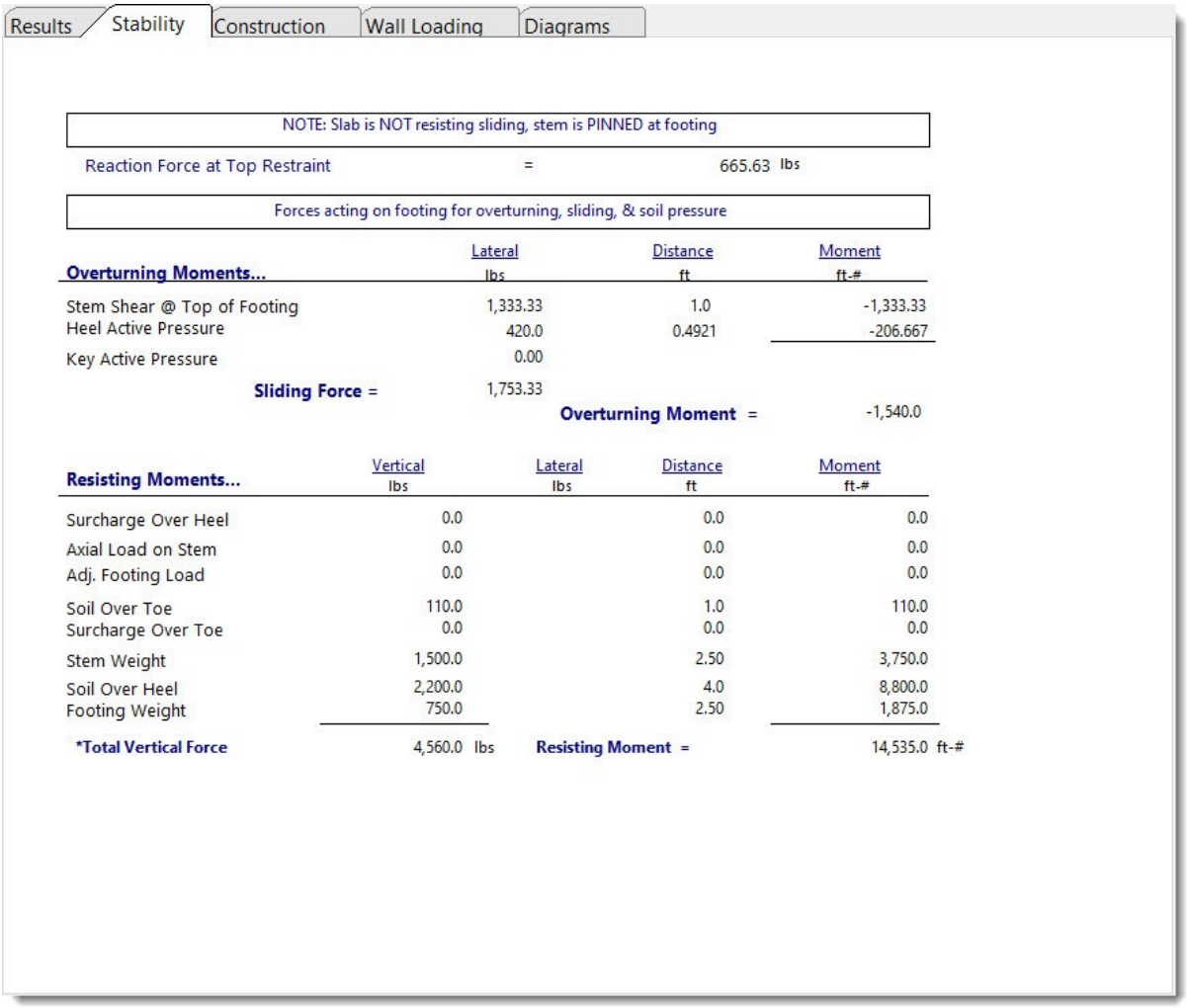
This tab was introduced in order to make it possible to view stem design values while the input focus is on tabs other than the Stem tab. This makes it easier to view the effects on stem moment and shear while changing other parameters such as retained height, backfill density, etc. It also includes subtabs to view the Rebar Lap & Embedment Length table and the Concrete Rebar Area Details table.



© 2026 ENERCALC, LLC

13.8.1.7 Stability Tab (Restrained Walls only)

For Restrained Walls the Stability sub-tab will appear, summarizing the conditions regarding base fixity and base lateral restraint.



A banner displays whether a slab is present to resist base sliding (box checked on Footing > Key Design tab) and whether fixed or pinned at base, as previously selected on the Stem tab.

The reaction at the top restraint is displayed.

The Sliding forces are displayed.

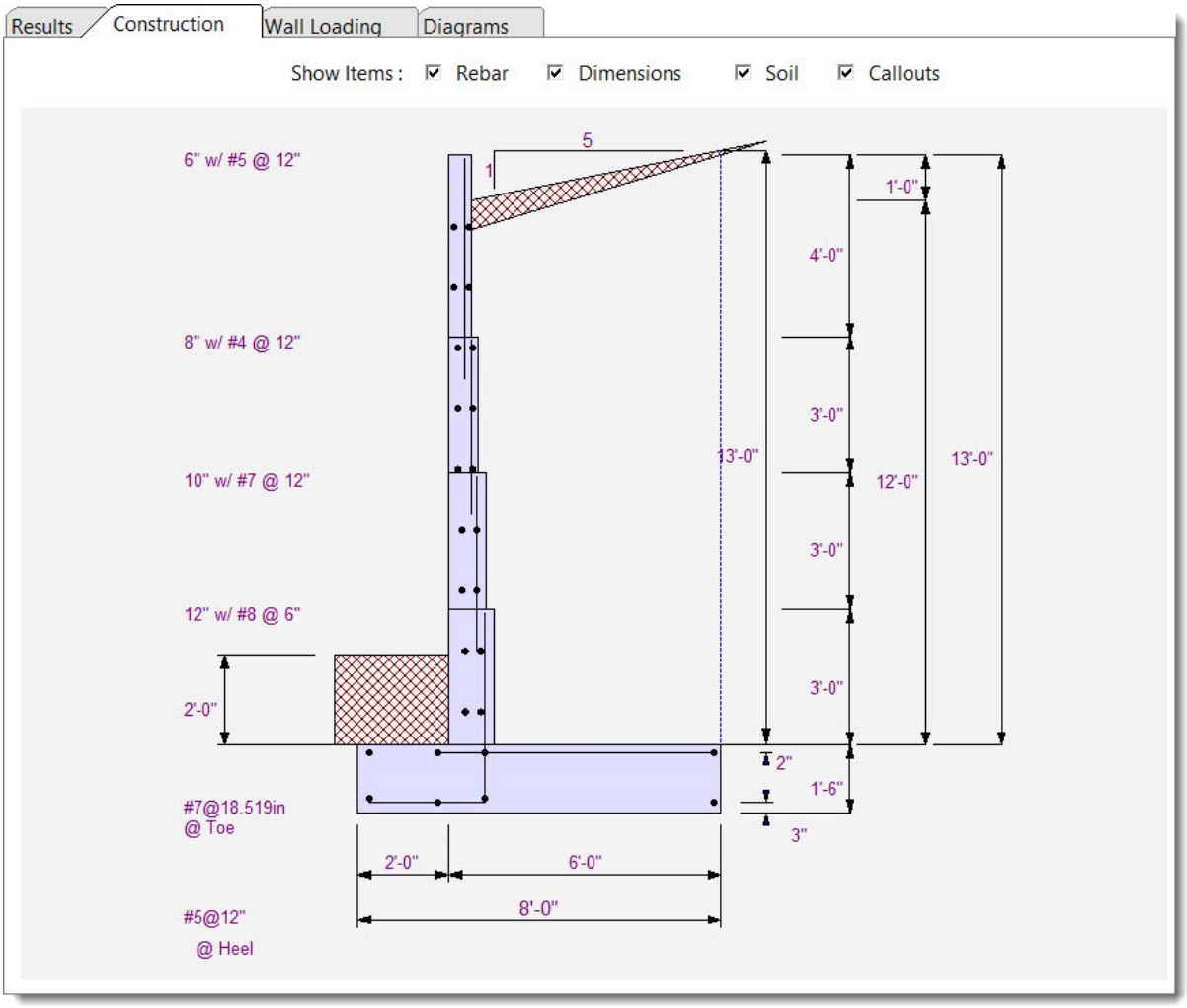
For analyzing the stem, if it is assumed “pinned” at the bottom (option is located on the Stem tab), and a slab is not present to resist sliding, then the theoretical overturning of the footing due to the reaction at the base of the stem, is the horizontal reaction at the bottom of the stem times the thickness of the footing.

© 2026 ENERCALC, LLC

If slab restraint is provided, the moment applied to the footing is the total vertical load times its eccentricity from the center of the footing. This moment is displayed (on the Stability tab) and is used to compute soil pressure.

13.8.1.8 Construction Tab

This graphics screen displays a construction drawing showing the pertinent construction data for the wall as you have entered it. It is automatically included in printed reports. This graphic is intended as a check of your input and is not editable.



To print, use Print button at top left. Layers of information can be turned on and off by checkboxes across the top of the drawing view.

This drawing will not depict the wall in a graphically correct way until sufficient data has been entered. Only a default graphic will appear initially.

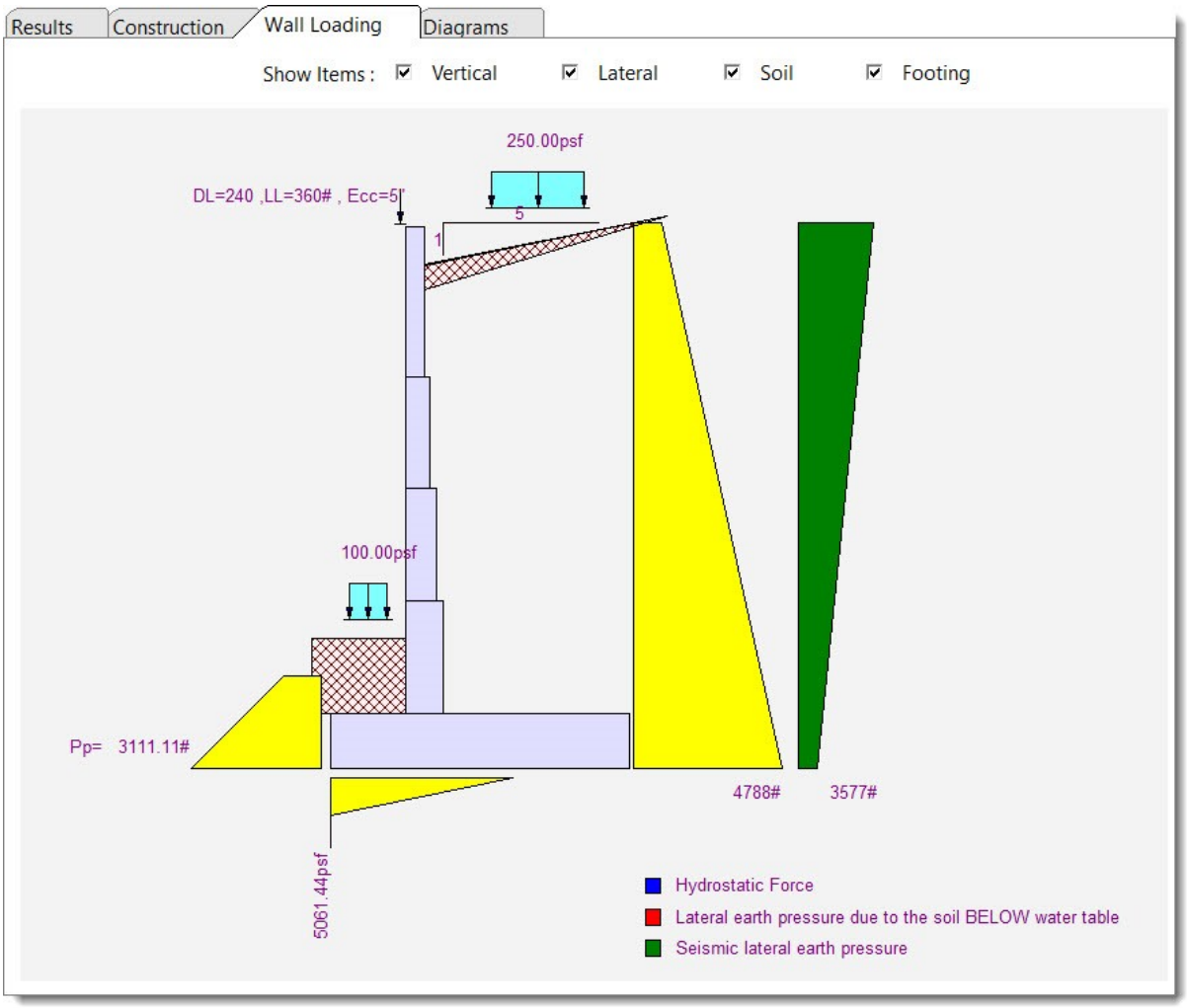
© 2026 ENERCALC, LLC

13.8.1.9 Wall Loading Tab

This diagram displays the active or at-rest pressure distribution, the passive pressure distribution, any applied loads that have been defined, and the maximum soil pressure distribution.

Loads are color-coded and may be turned on and off by using the checkboxes across the top of the diagram.

It is automatically included in printed reports.



This feature not available for segmental walls

This drawing will not depict the wall in a graphically correct way until sufficient data has been entered. Only a default graphic will appear initially.

© 2026 ENERCALC, LLC

13.8.1.10 Methodology / Analysis & Design Assumptions

GENERAL:

For cantilever walls the stem is fixed to the footing, the footing is free to rotate on the supporting soil, and no lateral restraint can exist at or near the top of the wall (otherwise it is not a cantilevered wall).

For restrained ("basement" or "tie-back") walls, the program assumes either 100% fixity at the base, or pinned (zero rotational fixity). Lateral support is at or near the top, and moment/shears are computed at the base, maximum positive, and at the upper support. The program does not check flexural stress reduction for axial loads (the unity interaction formula) since in most cases of basement walls the h/t ratio is below about 10 for masonry walls and somewhat higher for concrete, and axial stresses are low. If axial stresses are considered significant (say over 1000 lbs. per ft. length of wall), the interaction should be checked at the point of maximum positive moment.

For restrained walls, the program assumes that the restraint at or near the top is provided by a continuous line of restraint, such as could be provided by continuous connection to a slab or other diaphragm. If the connection between the retaining wall and the restraining diaphragm occurs only at discrete points, the horizontal span of the wall *between* those tieback points may become a design consideration. This potential failure mode would have to be checked by supplemental hand calculations, as the program does not consider this type of behavior.

References used for the development of this program are listed in Appendix E.

Stem design material is limited to concrete or concrete masonry. Design strength of concrete and masonry may be specified.

Conventional "heel" and "toe" terminology is used, whereby the "heel" side of the wall supports the retained earth. In this program, the "heel" distance is measured from the front face of the stem.

Concrete design for stem and footing is based upon ultimate strength design using factored loads. Factors for various building codes will be displayed on the Load Factors page, and may be edited. Since they are editable, be sure to check them before starting a design since you may have changed them.

Masonry design is based upon the Allowable Stress Design (ASD) or Strength Design (SD), as selected.

A geotechnical engineer will typically have determined design criteria (equivalent fluid pressure, allowable soil bearing pressure, sliding coefficient, etc.). If this is not the case, you can enter the angle of internal friction for the soil, and the program will compute the corresponding active pressure, using the Coulomb formulas based upon the soil density and backfill slope you have specified. If the Coulomb method is chosen, passive pressure will be based upon the Rankine Formula, assuming a level toe-side backfill.

Global stability is checked and reported for certain wall types.

© 2026 ENERCALC, LLC

Weight of concrete block masonry can be lightweight, medium weight, or normal weight, per the table in this User's Manual. Refer to Appendix C.

Horizontal temperature/shrinkage reinforcing is at the discretion of the designer. For horizontal temperature and shrinkage reinforcing for various stems see Appendix A.

Axial loads may be applied to the top of the stem but it is recommended that they do not exceed about 3,000 lbs to avoid reversal of heel bending moment. Slenderness interaction reductions for cantilevered walls are not calculated since h/t ratios are typically less than about 12. Only "positive" eccentricities from the centerline of the top stem are accepted (i.e. toward the toe), since negative eccentricity could lead to unconservative results.

Excessively high axial loads are not anticipated by the program and should not be applied if they would cause tension in the bottom of the footing heel – the program assumes typical retaining wall conditions where the heel moment causes tension at the top of the footing. If a design requires a very high axial load, say, over 3 kips/lf, it is suggested to use footing design software or hand calculations.

Concrete block thicknesses of 6", 8", 10", 12", 14", and 16" are allowed in the program.

Bond stress masonry for masonry stems. Flexural bond is a slipping (grip) stress between reinforcing and grout, resulting from the incremental change in moment from one point to another, and is a function of the total shear at the section. The program does not specifically check bond stress, but does use the formula $\mu = M / (j\ d\ \pi\ d_b)$, and compares this with the allowable development length. The formula for bond, relating to shear, is: $\mu = V / (\Sigma_o\ j\ d)$, where Σ_o is the perimeter of the bar(s) per linear foot. “j” and “d” are the familiar terms. This can be re-written to be approximately: $\mu = 0.35\ V\ s / d_b\ j\ d$, where “s” is the bar spacing in feet and d_b is the bar diameter, if the designer wishes to check to the bond.

Bond stress in masonry retaining walls is of questionable significance since the bars are customarily cast in grout which by code must be at least 2,000 psi, therefore comparable to embedment in concrete. Furthermore, Amrein (see bibliography) quotes a research study concluding the bond stress could be 400 psi based upon experimental studies showing minimum achieved stresses of 1,000 psi, thereby giving the former value a safety factor of 2.5.

This is probably a moot issue since rarely would bond stresses govern over shear stresses, particularly if the stress level in the reinforcing is factored in. Additionally, development lengths for reinforcing in masonry, and code required lap lengths, are considered quite conservative.

Stem reinforcing may be #4 through #10 bars.

Critical section for bending in the footing is at the face of the stem for concrete and 1/4 nominal thickness within the wall for masonry stems. For shear, for both concrete and masonry stems, the critical section is a distance "d" from the face of the stem toward the toe, and at the face of the stem for the heel. The program does not calculate toe or heel bar development lengths inward from the face of the stem (where the moment is maximum). When selecting and detailing the arrangement of toe and heel bars this should be considered.

© 2026 ENERCALC, LLC

Refer to Appendix B for development lengths in concrete, which can be adjusted for the stress level.

The program calculates the bending in the key and determines whether reinforcing is required. For determining section modulus, 3" is deducted from the key width per ACI recommendation. If reinforcing is required, a message will appear. You can then change the key dimensions until the message disappears, or use the rebar suggestions displayed. The key moment and shear is produced by the passive resisting pressure acting against the key.

Slab restraint at the base can be specified on the Footing > Key Design & Sliding Options tab. The program only allows this restraint to occur at the top of the footing – not higher.

RESTRAINED WALLS:

A vertical component of active pressure is not activated, whether or not it is checked on the General tab, since the top of the wall is assumed not to deflect and thereby not activate such force. Overturning moment is not applicable, and is therefore not displayed, since overturning stability is by restraint at or near the top of the wall.

When 100% Fixity @ Base is selected soil pressures are assumed to be completely uniform.

When 100% Fixity @ Base is **not** selected, and if slab restrains sliding, the soil pressures are calculated considering the following effects:

- Moment on soil due to eccentricity of vertical loads.

When 100% Fixity @ Base is **not** selected, and if no slab restrains sliding, the soil pressures are calculated considering the following effects:

- Moment on soil due to shear times the footing thickness.
- Moment on soil due to eccentricity of vertical loads.

Shear at base of stem is computed based on the summation of all lateral force above that point.

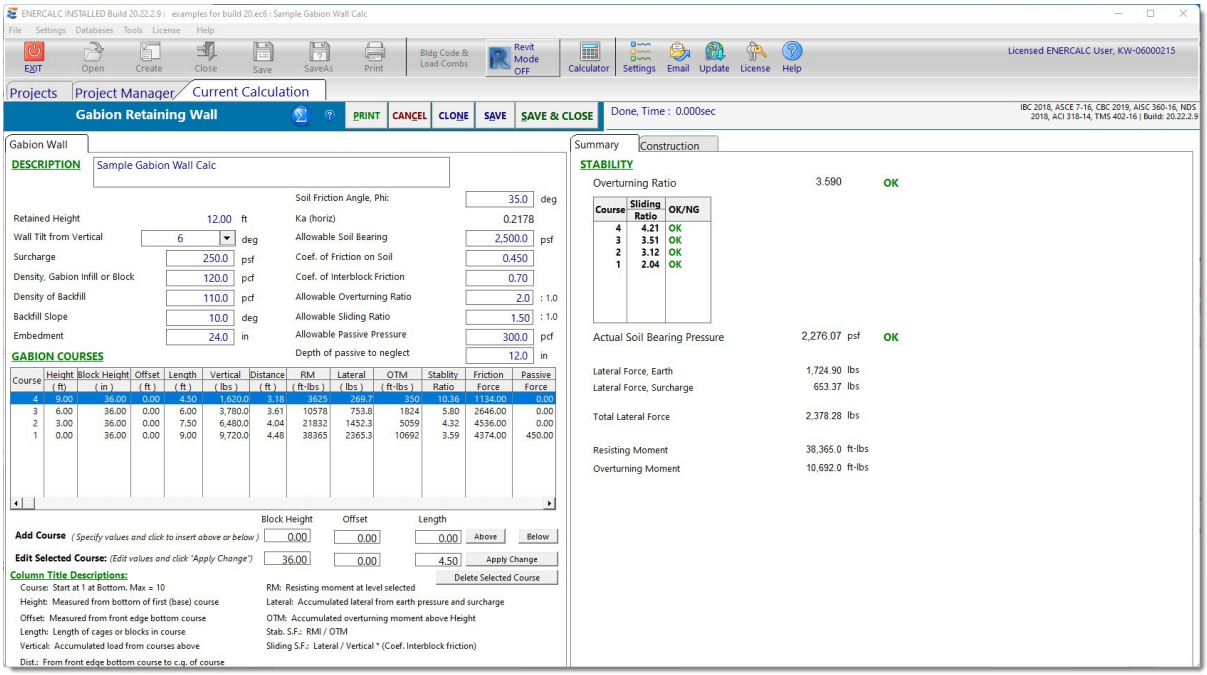
13.8.2 Gabion Retaining Wall

A gabion wall is a gravity wall constructed using prefabricated steel wire cages filled with rock. The cages are often 3 ft on a side and are infilled with stone as specified by the designer. In lieu of rock filled gabion cages, large precast concrete blocks may be used.

This program assumes all cages or blocks to be of uniform height and infill density. They can either be assembled vertically or tilted backward by selecting either 3° or 6° tilt. Maximum allowed height is 18 ft. A rule of thumb for the length of the bottom course is 75% of the retained height. The retained height is assumed to be the same height as the wall. The Coulomb equation is used for determining lateral earth pressure. As of July 2016 the Gabion Wall module uses the following values in the Coulomb equation: Angle of soil face of wall is equal to 90 degrees plus the user-specified wall tilt value, and the soil-wall friction angle is equal to (2/3) Phi.

© 2026 ENERCALC, LLC

This Gabion Wall program does not handle MSE (mechanically stabilized earth) walls, which use geogrids.



Notes:

1. All courses are of the same height and infill density.
2. Concrete blocks may be used in lieu of gabion cages.
3. Coulomb equation is used for active pressure.
4. This design is not valid for reinforced soils (Mechanically Stabilized Earth). Consider using Segmental Retaining Wall module instead.

General Input:

Course Height (Gabion/Block), In: Height of the gabion cages or block in inches. This can vary from one course to the next.

Retained Height, ft: Retained height in ft. which is also assumed to be the top of the wall.

Wall Tilt from Vertical, deg: Select "None", 3°, or 6° backward tilt.

Surcharge, psf: Surcharge load if applicable.

Density, Gabion Infill or Block, pcf: Density of the infill or block. A rock infill is typically 120pcf and concrete block is typically 140pcf.

Density of Backfill, pcf: Density of the backfill material, typically provided by the geotechnical engineer.

© 2026 ENERCALC, LLC

Backfill Slope, deg: If applicable, enter the backfill slope in degrees.

Embedment, in: Specifies the depth of embedment of the first course.

Soil Friction Angle, Phi: Obtain this from the geotechnical engineer.

Ka (horiz): Computed using the Coulomb equation with variables being phi, backfill slope and with wall/soil friction angle assumed to be 0°.

Allowable Soil Bearing, psf: Obtain this value from the geotechnical engineer.

Coef. of Friction on Soil: As determined by the geotechnical engineer. Typically 0.25-0.50. Used for the sliding check of the first course.

Coef. of Interblock Friction: Coefficient of friction to resist sliding between cages or blocks. A value 0.70 is often used. Used for the sliding checks *above* the first course.

Allowable Overturning Ratio: The minimum permissible ratio of resisting moment divided by overturning moment.

Allowable Sliding Ratio: The minimum permissible ratio of sliding resistance divided by sliding force.

Allowable Passive Pressure, pcf: (When a nonzero embedment is specified) Allowable passive resistance to sliding (when an embedment is specified) as an equivalent fluid pressure.

Depth of Passive to Neglect, in: (When a nonzero embedment is specified) An option to specify a top layer of passive pressure to disregard in the resistance calculation, as in cases where it may be unreliable due to frost disturbance.

Gabion Courses Table:

Course: These are numbered in ascending order and cannot exceed 10.

Height: Measured from bottom of first (base) course.

Block Height: Height of the selected course. Can vary from one course to another.

Offset: Horizontal offset measured from front edge of bottom course.

Length: Length of cages or blocks in course.

© 2026 ENERCALC, LLC

Vertical: Accumulated vertical load from courses above.

Distance: Horizontal distance from front edge of bottom course to centroid of the referenced course.

RM: Resisting moment at referenced course.

Lateral: Accumulated lateral force from earth pressure and surcharge at referenced course.

OTM: Accumulated overturning moment above referenced course.

Stability Ratio: RM / OTM

Friction Force: Friction force at the specified course calculated as Vertical / Friction Factor, where Friction Factor is Coefficient of Friction on Soil for the calc at Course 1 and Coefficient of Interblock Friction for the calc at courses *above* Course 1.

Passive Force: Allowable resisting force that can be generated from passive pressure that is not neglected. **Only considered to benefit Course 1.**

Friction + Passive: Simple sum of Friction Force plus Passive Force.

Sliding Ratio: Friction + Passive / Lateral.

Add, edit or delete courses using the buttons and input fields below the table. The first value entered will automatically be the bottom layer. To delete a course highlight it and click Delete.

Summary Tab:

Overturning Ratio: Reports the minimum Stability Ratio as described above.

Sliding Table: Reports the Sliding Ratio and an OK/NG status for each course.

Act. Soil Bearing Pressure, psf: Computed using conventional statics and appears in red if it exceeds the allowable soil bearing specified.

Lateral Force, Earth: The total lateral force attributable to lateral earth pressure.

Lateral Force, Surcharge: The total lateral force attributable to an applied surcharge.

Total Lateral Force: Simple sum of Lateral Force, Earth plus Lateral Force, Surcharge.

© 2026 ENERCALC, LLC

Resisting Moment: The value of RM at Course 1.

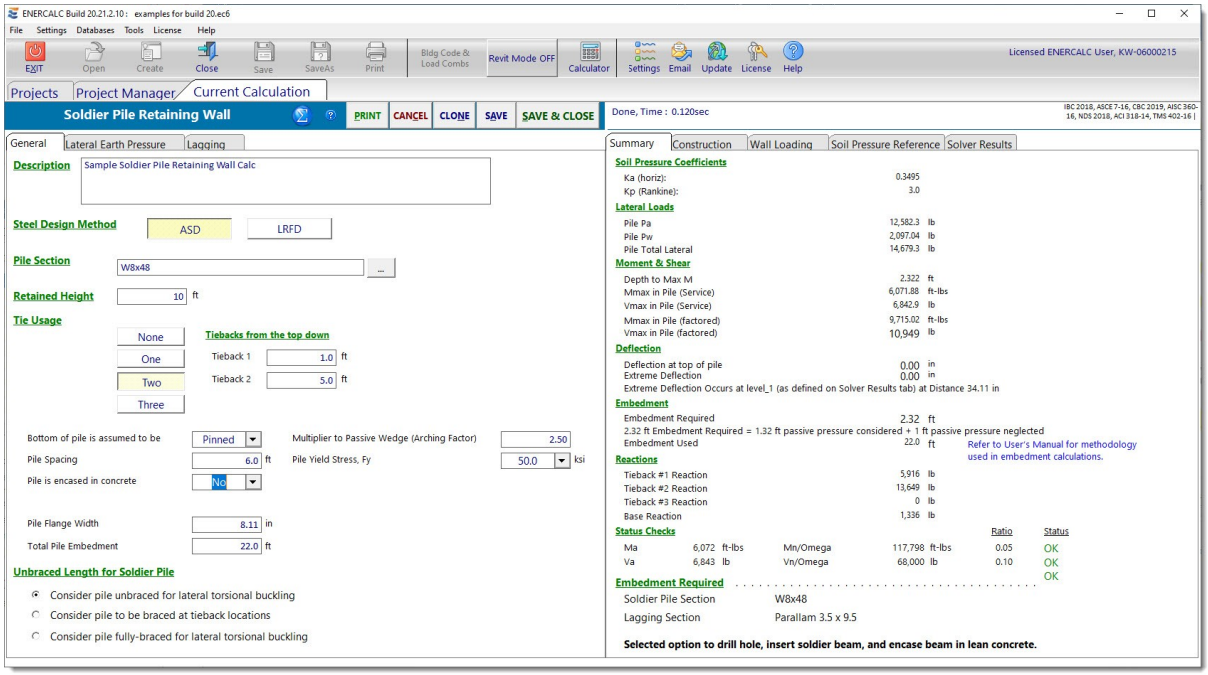
Overturning Moment: The value of OTM at Course 1.

13.8.2.1 Methodology / Analysis & Design Assumptions

References used for the development of this program are listed in Appendix E.

13.8.3 Soldier Pile Retaining Wall

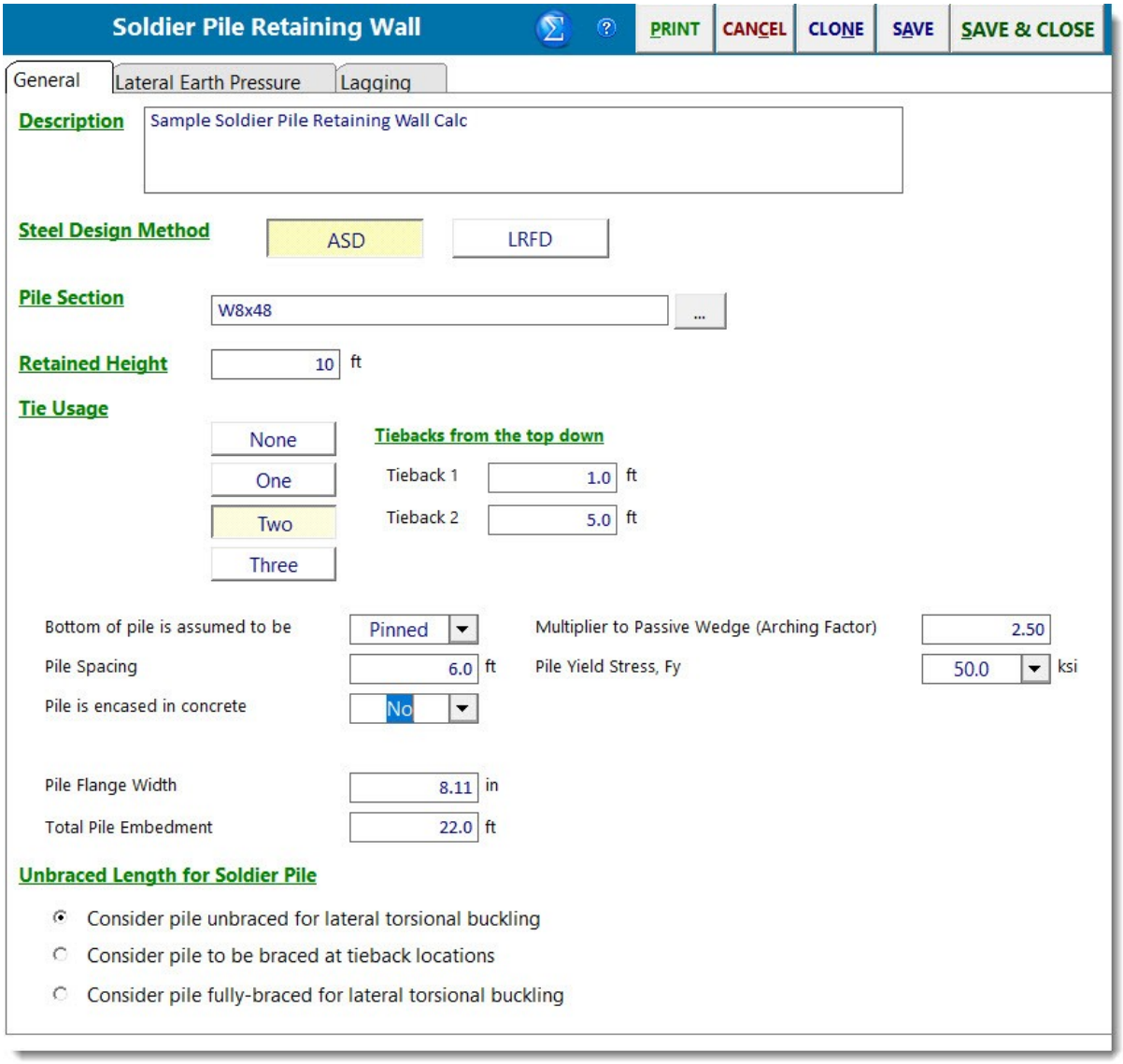
Soldier pile retaining walls, also called soldier beam walls, are generally used at construction sites for temporary shoring. Steel piles are driven into the ground, or placed in drilled holes filled with lean concrete, at a spacing such that lagging can be placed between the piles, and the excavation can proceed down to the level of the finished grade on the low side. The stability of a soldier pile retaining wall depends upon the active earth pressure being resisted by passive pressure on the embedded section of the pile. Pile spacing is typically 6 – 10 feet on center.



This module is suitable for cantilevered soldier piles (no tie-backs) or soldier piles with tie-backs.

© 2026 ENERCALC, LLC

13.8.3.1 General Tab



Steel Design Method: Select ASD or LRFD methods for design of steel pile

Pile Section: Opens the Steel Section Database, allowing the selection of common pile sections.

Retained Height, ft: Distance between the final excavated grade and the retained height at the top grade.

Tie Usage: Select none, one, two, or three. If any tie-backs are used, they are located by their distance from the top of the pile.

Bottom of pile is assumed to be: This is a conditional input. It is only available when tie-backs **are** used, which triggers a stiffness analysis using ENERCALC SEL's 2D Frame program in the

© 2026 ENERCALC, LLC

background. The pile is modeled in segments from spanning between tie-backs. The bottom segment spans from the lowest tie-back to the bottom of the embedment depth. This setting controls whether that lowest support is considered fixed or pinned with regard to rotation.

Pile Spacing, ft: Center to center spacing of piles, typically 6 ft to 10 ft.

Pile is encased in concrete: Select whether the steel pile is driven into the soil or placed into a drilled hole and encased in lean concrete.

Pile Flange Width / Diameter of Encasement, in: If the pile is driven, enter the flange width. If the pile is set in lean concrete in a drilled hole, enter the hole diameter.

Total Pile Embedment, ft: Actual embedment, usually rounded from the required embedment reported below.

Multiplier to Passive Wedge: This is an arching factor. It takes the form of a multiplier from 1.0 to 3.0 to be applied to the pile flange width or drilled hole diameter due to wedging action and is usually provided by the geotechnical engineer.

Pile Yield Stress, Fy: Yield stress of the selected pile section.

Unbraced Length for Soldier Pile: Allows the user to specify the unbraced length for steel design purposes. This is a conditional input.

If the pile is **not** tied back, the user can select:

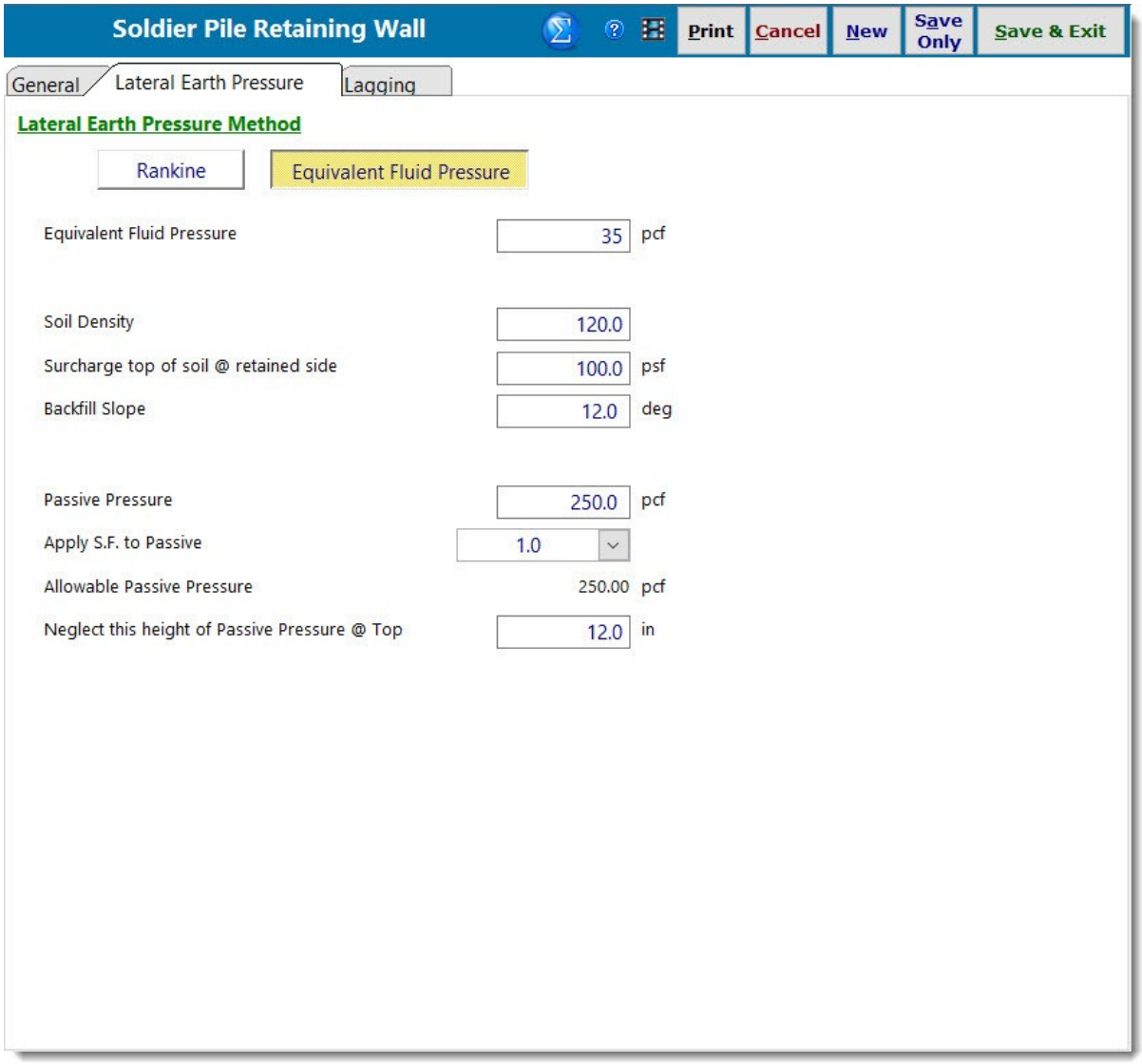
- Consider pile unbraced for lateral torsional buckling
- Consider pile fully-braced for lateral torsional buckling

If the pile **is** tied back, the user can select:

- Consider pile unbraced for lateral torsional buckling
- Consider pile to be braced at tieback locations
- Consider pile fully-braced for lateral torsional buckling

© 2026 ENERCALC, LLC

13.8.3.2 Lateral Earth Pressure Tab



Lateral Earth Pressure Method: This is a conditional input. It is only available when tie-backs are **not** used. When it is available, it allows the use of Rankine Active Pressure or Equivalent Fluid Pressure methods. When tie-backs are used, the pressure is applied as described on the Soil Pressure Reference tab.

EFP: Equivalent Fluid Pressure of the retained material, usually obtained from the geotechnical engineer.

Soil Phi, degrees: Angle of internal friction of the retained material, usually obtained from the geotechnical engineer.

Soil Density, pcf: Density of the retained soil, usually 100 to 130 pcf.

© 2026 ENERCALC, LLC

Surcharge top of soil at retained side, psf: Surcharge on upper grade such as for equipment, materials, or contingencies.

Backfill Slope, degrees: Slope of the backfill measured in degrees from horizontal.

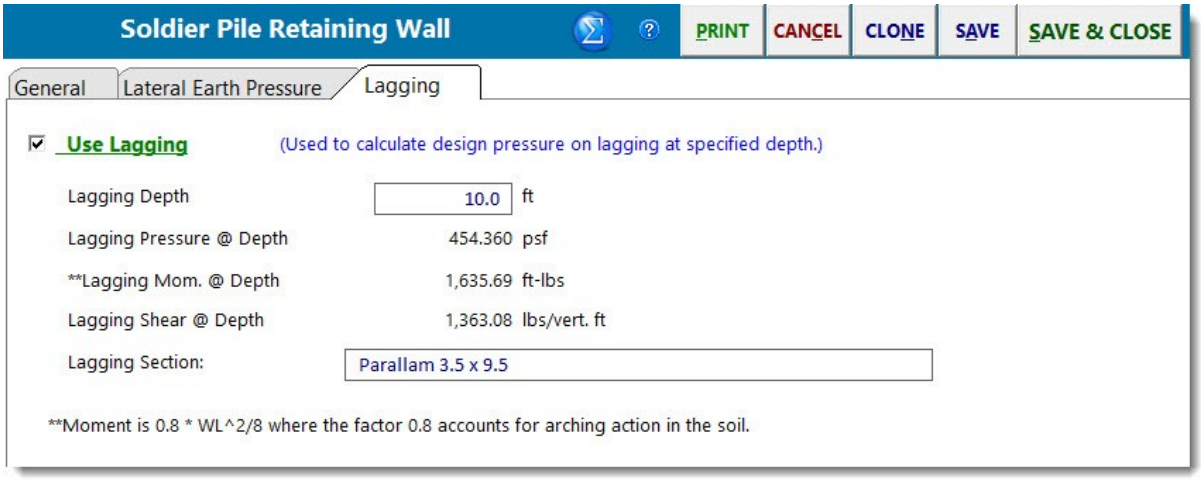
Passive Pressure, psf/ft: Passive pressure in pcf. When using Rankine active soil pressure method, passive pressure is calculated as (1/Ka) *(soil density). When using EFP, the passive pressure is a user input.

Apply S.F. to Passive Pressure: Safety factor that will be applied to the above passive pressure, typically suggested by the geotechnical engineer.

Neglect this height of Passive Pressure @ Top: Allows a specified height of passive pressure to be ignored for resistance, such as if the soil was likely to be frost disturbed or otherwise unsuitable to resist lateral pressure. Note that this input is specified in units of inches.

Multiplier on Ka*Gamma*H: This is a conditional input. It is only available when tie-backs **are** used, and it allows the user to fine tune the pressure value. The Soil Pressure Reference tab shows how the soil pressure distribution is developed.

13.8.3.3 Lagging Tab



Use Lagging: Check this box if lagging is to be considered between piles.

© 2026 ENERCALC, LLC

Lagging Depth, ft: Depth below grade at which lagging pressure is to be calculated based on the active soil pressure.

Lagging Pressure @ Depth, psf: Pressure used in the design of horizontal wood lagging between piles.

Lagging Moment @ Depth, ft-lbs: Moment computed assuming arching action and using moment = $0.8 * wl^2/8$.

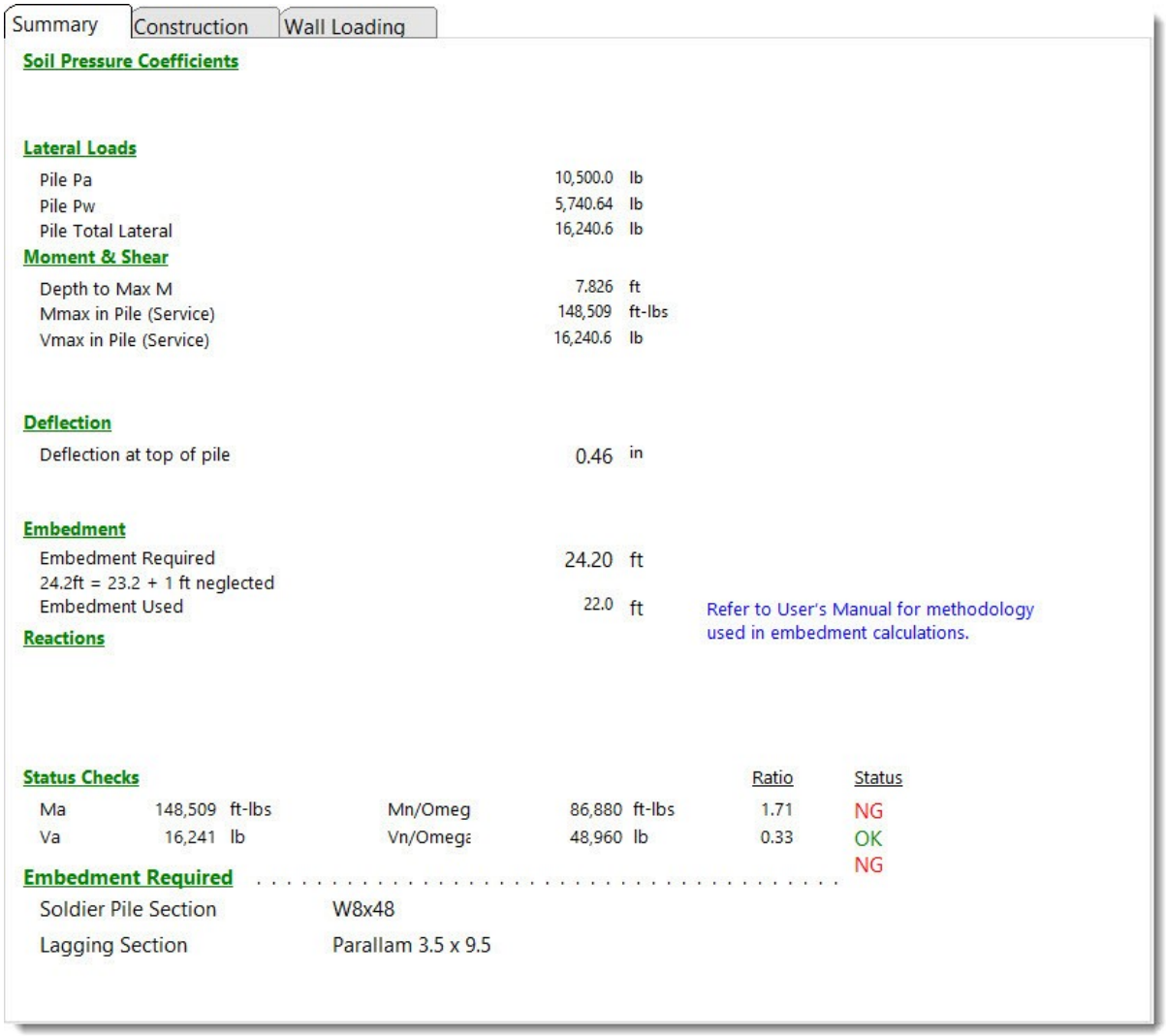
Lagging Shear @ Depth, lb/vertical ft: Shear computed using wl/2 where w is the Lagging Pressure @ Depth determined above.

Lagging selection: Wood lagging selection, such as 4 in x 12 in. No design is performed based on this entry, but it is printed in the calculation report for documentation purposes.

13.8.3.4 Results Tabs

© 2026 ENERCALC, LLC

Summary Tab



Soil Pressure Coefficients:

Ka (horiz) and Kp (Rankine) are computed automatically when using the Rankine method. Kp (Rankine) is calculated as 1/Ka for a backfill slope of zero.

© 2026 ENERCALC, LLC

Lateral Loads:

Pile Pa, lbs: Total lateral force due to earth pressure.

Pile Pw, lbs: Total lateral force due to surcharge if applicable.

Pile Total lateral, lbs: Sum of Pa + Pw

Moments & Shears:

Depth to Max M, ft: Distance below lower grade to point of maximum moment in the pile.

Mmax in Pile (Service), ft-lbs: Maximum service level moment in the pile.

Vmax in Pile (Service), ft-lbs: Maximum service level shear in the pile.

Mmax in Pile (Factored), ft-lbs: Maximum factored level moment in the pile. A factor of 1.6 is used.

Vmax in Pile (Factored), ft-lbs: Maximum factored level shear in the pile. A factor of 1.6 is used.

Deflection: This is a conditional result.

If the pile has **no** tie-backs, the result will report the deflection at the top of the pile.

If the pile **has** tie-backs, the result will report the deflection at the top of the pile, as well as the extreme deflection and the location where the extreme deflection was found to occur. Refer to the Solver Results tab for more information on how to locate the point of extreme deflection.

Embedment:

Reports the required pile embedment based upon allowable passive pressure, the specified safety factor and the applied active pressure. The result is further itemized into the length of embedment that is considered effective for passive resistance and the portion that is neglected (if any). Finally, the results report the embedment used for documentation purposes.

© 2026 ENERCALC, LLC

Reactions: This is a conditional result.

If the pile has **no** tie-backs, this area will be blank.

If the pile **has** tie-backs, the result will report the tie-back reactions and the base reaction as determined by the stiffness analysis.

Status Checks:

Reports Mu (or Ma), phiMn (or Mn/Omega), the ratio, and an OK/NG status.

Reports Vu (or (Va), phiVn (or Vn/Omega), the ratio, and an OK/NG status.

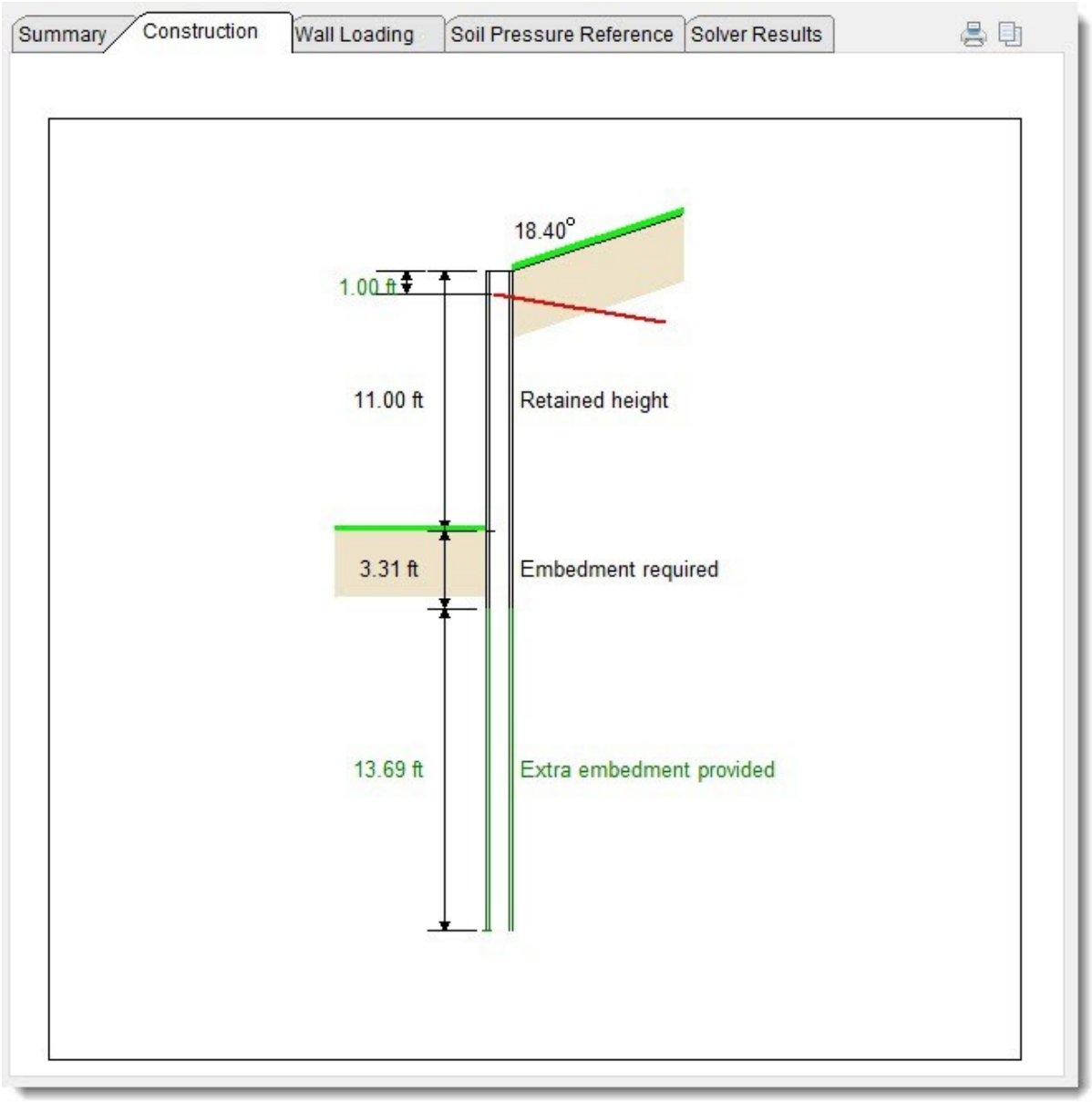
Reports an OK/NG status on the Embedment.

Soldier Pile Section documents the selected section for the pile.

Lagging Section documents the selected section for the lagging.

© 2026 ENERCALC, LLC

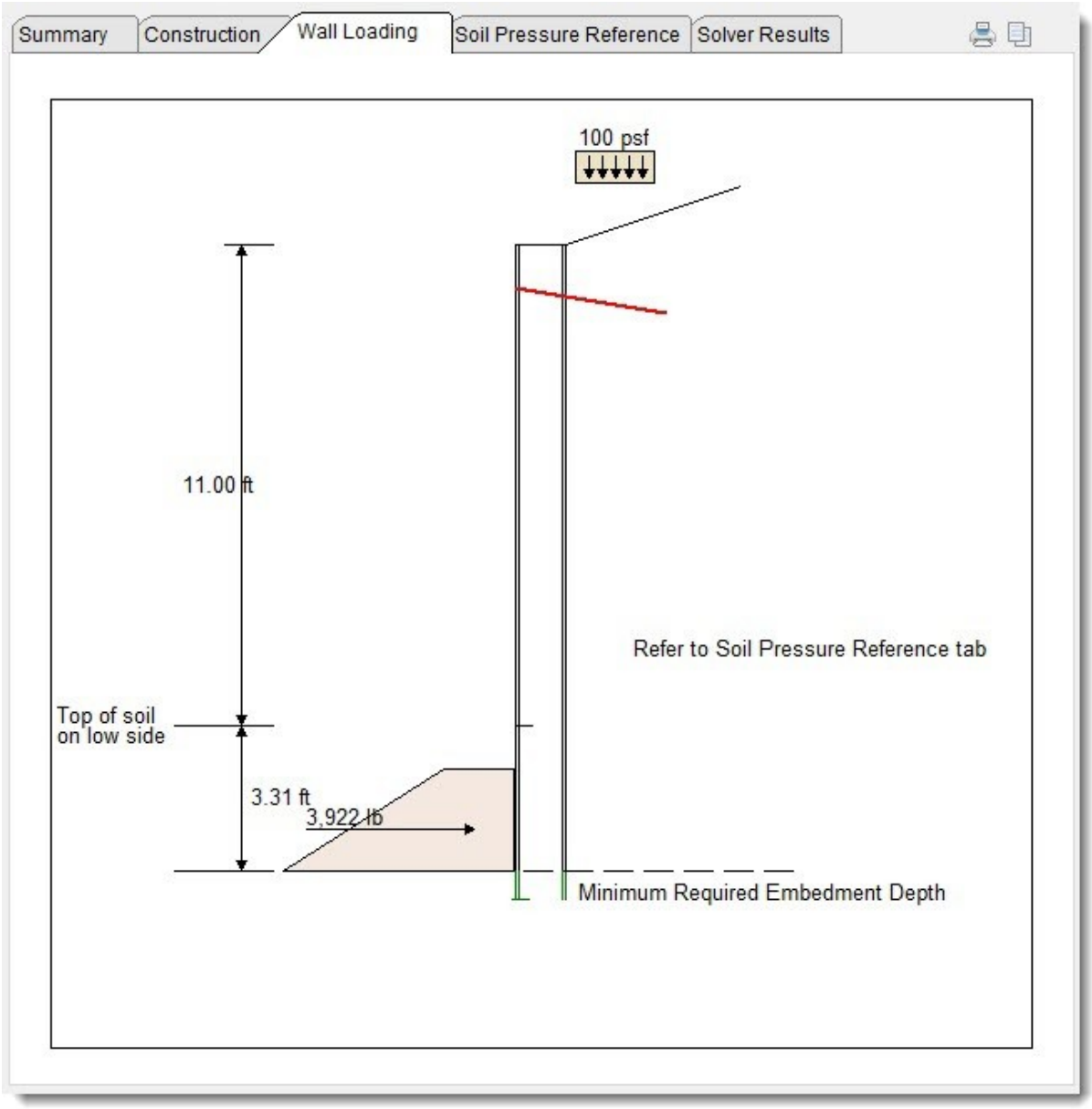
Construction Tab



Displays the soldier pile, the retained soil, tie-backs (if any), and critical dimensions.

© 2026 ENERCALC, LLC

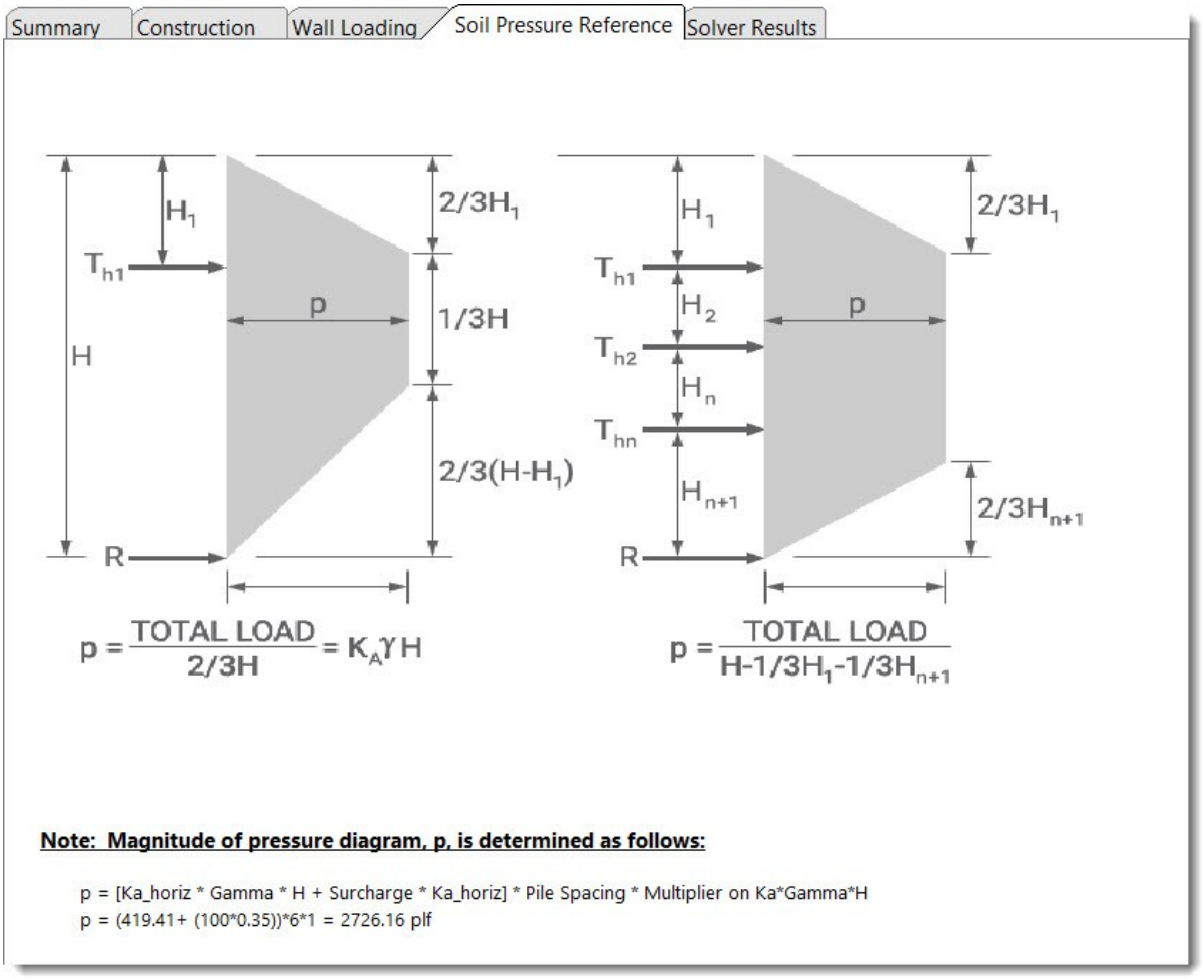
Wall Loading Tab



Displays the soldier pile, the retained soil, tie-backs (if any), surcharge load (if any), and passive pressure diagram.

© 2026 ENERCALC, LLC

Soil Pressure Reference



Displays the loading diagram that is applied to the 2D Frame analysis model, as well as the derivation of the pressure values.

H: Retained height

Th1...Thn: Tie-back reactions

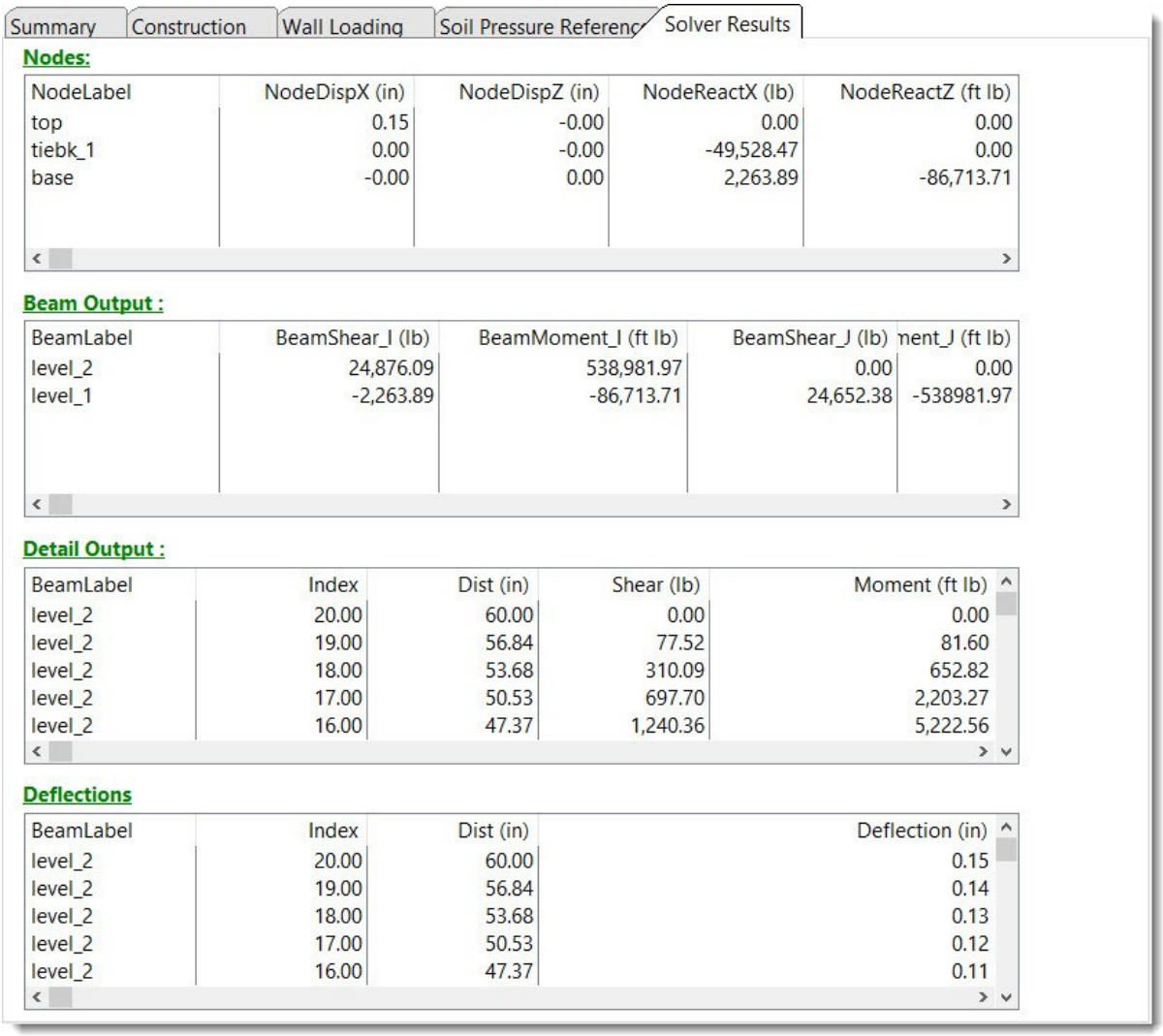
H1...Hn: Dimensions used to locate tiebacks

R: Lateral reaction at base

p: Maximum pressure intensity as determined by [Ka_horiz * Gamma * H + Surcharge * Ka_horiz] * Pile Spacing * Multiplier on Ka * Gamma * H

© 2026 ENERCALC, LLC

Solver Results



Displays the results from the 2D Frame analysis model.

Nodes: Displays the node list as well as displacements and reactions

Beam Output: Displays list of beam segments with shears and moments

Detail Output: Displays list of beam segments with shears and moments at small increments along the height of the pile

Deflections: Displays list of beam segments with deflection values at small increments along the height of the pile

13.8.3.5 Methodology / Analysis & Design Assumptions

References used for the development of this program are listed in Appendix E.

© 2026 ENERCALC, LLC

A basic design requires these steps:

There are many theories presented for calculating embedment depths (Plum method and others) and that shown in figure below is one that closely agrees with other methods.

Determine the driving forces, that is, forces imposed by any construction surcharges and the active soil pressure tributary to each pile beam. Use the Rankine equation to calculate Ka. Several designs may be done to optimize the beam spacing based upon lagging selection, embedment depths, and beam sizes.

Referring to the figure below, after Pa and Pv, have been calculated, the depth of embedment must be determined. This will be a function of the allowable passive pressure and arching factor allowed to increase the effective flange width, or hole diameter if pre-drilling is used. The arching factor. A. can be taken as 0.08 * phi, but should not exceed about 2.5. This means that the effective pressure width in front of a 24" diameter drilled and concrete filled beam encasement, with a phi of 32° would be 0.08 * 32 = 2.56. but use 2.5. Thus the effective passive pressure would be a width of 2.0*2.5 = 5.00 feet which will considerably reduce embedment depth and moment applied to the beam.

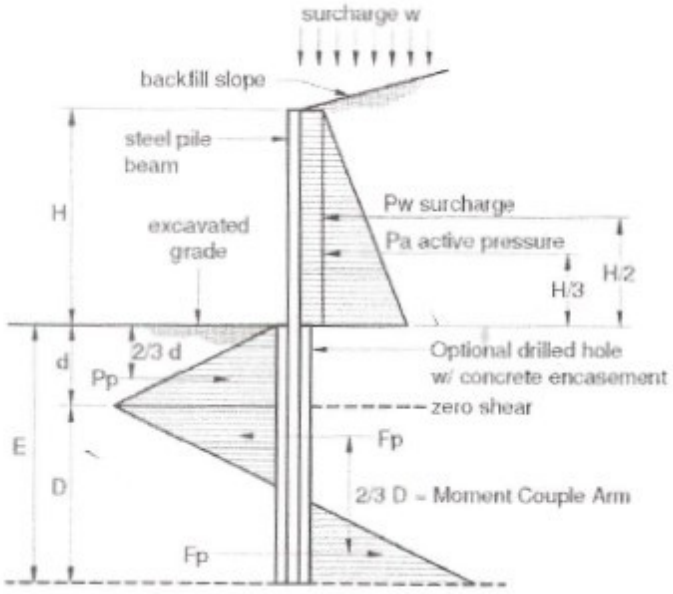


Figure illustrates theorized forces on cantilevered soldier beam in sandy soil

For determining the embedment depth to zero shear (where beam bending is maximum), designated "d", the following equation can be used:

$$d = \sqrt{\frac{P_P \times SF \times 2.}{p \times Dia \times A}}$$

© 2026 ENERCALC, LLC

Where Pp is equal to and counteracting to Pw + Pa; SF is the safety factor applied to allowable passive pressure; A is the arching factor multiplier; Dia is the hole diameter or flange width, whichever applicable; and p is the allowable passive pressure in pcf at “d".

1. The maximum beam moment is then determined by summing moments above the point of zero shear. Mathematically the result is equivalent to:

$$Mmax = Pw (0.50H + 0.67d) + Pa (0.33H + 0.67d)$$

2. The maximum moment is resisted by a passive pressure couple consisting of Fp * 0.67D. Therefore the required depth D can be estimated from the following equation:

$$D = \sqrt{\frac{(\text{max. moment}) \times SF}{(p \times \text{Dia} \times A \times d \times 0.25)0.67}}$$

The required depth of embedment is then (d + D). As a rule-of-thumb for sandy soils this is usually in the range of 1.3 H to 1.5 H. It is conservative to add 20% - 30% depth to calculated embedment.

3. After the maximum moment has been computed, convert it to LRFD (Load Resistance Factor Design) by multiplying by the usually applicable load factor of 1.6. Then select several beam options from AISC latest edition, LRFD Steel Design Handbook.

4. Select the lagging. Treated lumber should be used. Calculate the lateral pressure at various depths, Hy,(to determine changing lagging thicknesses) which is Ka * Soil Density * Hy. When the simple span moment is calculated it is common to multiply by 0.8 because of arching action of the soil between pile beams. Lagging is typically either 3" or 4" by 12” treated wood. Lagging ends should bear against the beam flange a minimum of 3". Allow about 1" between each lagging for drainage.

In addition, it is important to understand that the tied-back version of the module is based on a stiffness analysis from the 2D Frame module within ENERCALC SEL. It uses beam elements spanning from the top of the retained height to the top tie-back, which is modeled as a lateral support. If there is more than one tieback, the subsequent tie-backs are modeled as lateral supports. The lowest beam segment spans from the lowest tie-back to a "base" node that is modeled at the bottom of the embedment depth.

The top node is free to rotate and deflect. Nodes representing tie-backs are free to rotate and deflect vertically, but are constrained laterally. The "base" node cannot translate at all, but the user can specify whether it is able to rotate or if it should be treated as rotationally fixed.

The applied loading is as shown on the Soil Pressure Reference tab.

Reactions and displacements are reported by the solver. The user will probably need to increment the embedment depth until the module reports that the embedment is acceptable for the modeled conditions.

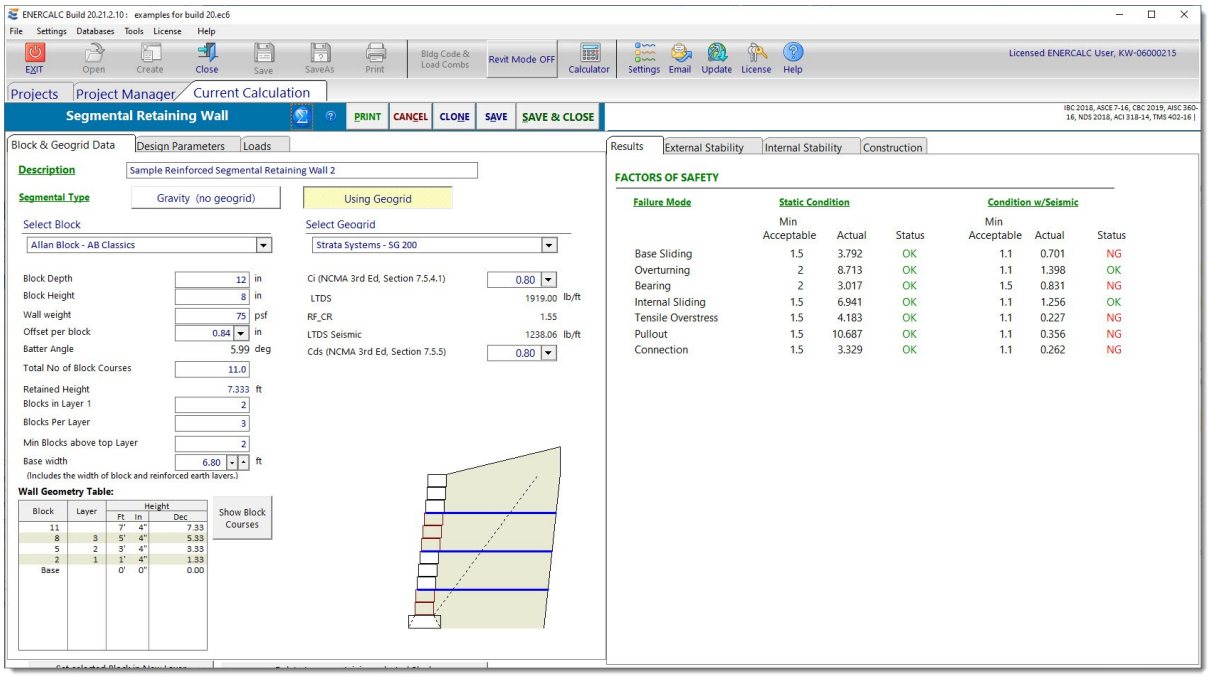
© 2026 ENERCALC, LLC

Steel design is performed at many increments along the length of the pile, taking into consideration the steel yield strength, the section forces, and the bracing assumptions specified by the user.

13.8.4 Segmental Retaining Walls

13.8.4.1 Segmental Wall Overview

Segmental walls are constructed of stacked masonry blocks, usually of proprietary configurations, without steel rebar, grouting, or mortar. They are dry-stacked, either vertically or with offsets at each block such that the wall is slightly battered and leans into the earth. When geogrids are used in segmental retaining walls, they are placed in horizontal layers separated by some vertical distance as the wall is constructed and backfilling progresses. Their purpose is to reinforce the earth behind the wall such that the reinforced earth zone acts en masse with the wall to resist sliding and overturning, hence no conventional foundation is required. (These walls are also called MSE – Mechanically Stabilized Earth walls.) The geogrids extend beyond the failure plane and resist pullout by friction resistance due to the weight of soil above. Connection to the wall blocks is achieved through friction between blocks and sometimes by proprietary connection devices.



13.8.4.2 Design Assumptions for Geogrid Reinforced Segmental Walls

When working with the Geogrid Reinforced Segmental Retaining Wall module, the input screens and output report vary from the conventional cantilevered and restrained retaining walls.

In general, methodology used conforms to NCMA's *Design of Segmental Retaining Walls, 3rd Edition.*

© 2026 ENERCALC, LLC

Since segmental geogrid reinforced retaining walls can be highly complex, some simplifying design assumptions have been implemented to make the program easier to use and still cover most conditions encountered. These assumptions are:

1. All masonry units are the same size (height, width, depth) and single wythe.
2. Offsets between blocks are uniform for the full height of the wall.
3. Spacing of geogrid layers may be specified (number of blocks between layers), but spacing is constant except for lowest layer and above uppermost layer.
4. Lengths of geogrids are constant for all layers.
5. Same geogrid material is used for all layers.
6. Coulomb method is used for determining lateral earth pressures.
7. Overall wall height is limited to 30 feet.
8. Setting base is assumed to be gravel or crushed stone, 6” thick, and extending 6” beyond each edge of the bottom block.
9. Block dimensions are obtained from vendor websites or literature.
10. Weight of wall is assumed to be 120 pcf for depth of block.
11. Geogrid Long Term Design Strength and connection values have been obtained from vendor websites or vendor ES-ICC Evaluation Reports. Verification with vendor is recommended.

13.8.4.3 Design Assumptions for Gravity Segmental Retaining Walls

When working with the Geogrid Reinforced Segmental Retaining Wall module, the input screens and output report vary from the conventional cantilevered and restrained retaining walls.

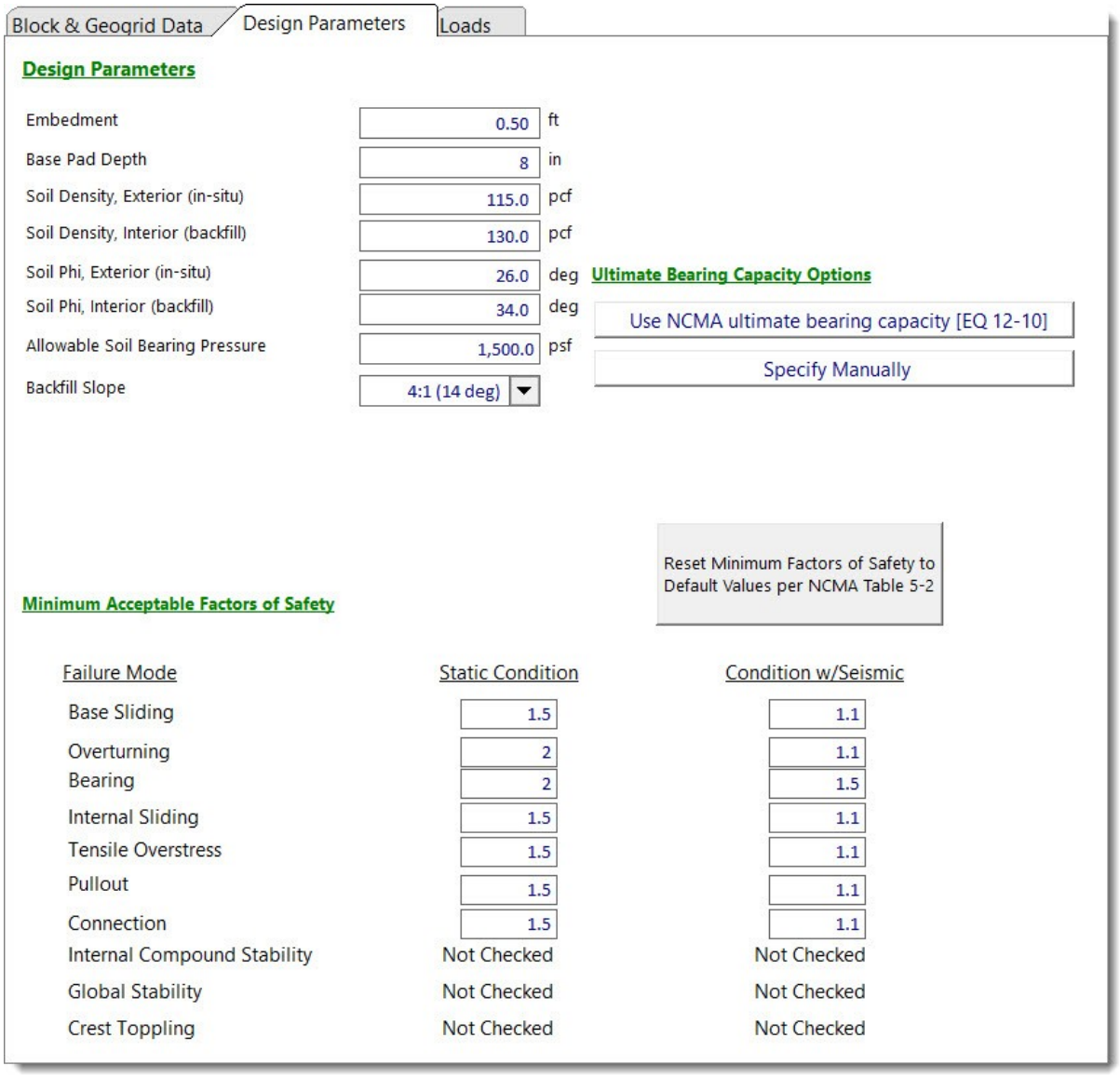
In general, methodology used conforms to NCMA’s *Design of Segmental Retaining Walls, 3rd Edition.*

Since segmental retaining walls can be highly complex, some simplifying design assumptions have been implemented to make the program easier to use and still cover most conditions encountered. These assumptions are:

1. All masonry units are the same size (height, width, depth) and single wythe.
2. Offsets between blocks are uniform for the full height of the wall.
3. Coulomb method is used for determining lateral earth pressures.
4. Setting base is assumed to be gravel or crushed stone, 6” thick, and extending 6” beyond each edge of the bottom block.
5. Block dimensions are obtained from vendor websites or literature.
6. Weight of wall is assumed to be 120 pcf for depth of block.

© 2026 ENERCALC, LLC

13.8.4.4 Design Parameters Tab



Retained Height: Displays the retained height based on block coursing and geometry entered on the Block & Geogrid tab.

Embedment: Depth below grade (on the low side) to top of setting pad. Usually one block course or 1'-0".

Base Pad Depth: Thickness of the base pad.

Soil Density, Exterior (in-situ): Enter the density of the native soil beyond the backfill zone and under the base.

© 2026 ENERCALC, LLC

Soil Density, Interior (backfill): Enter the density of the backfill material (typically granular soil or gravel).

Soil Phi, Exterior (in-situ): Enter the angle of internal friction of in-situ soil.

Soil Phi, Interior (backfill): Enter the angle of internal friction of the backfill soil.

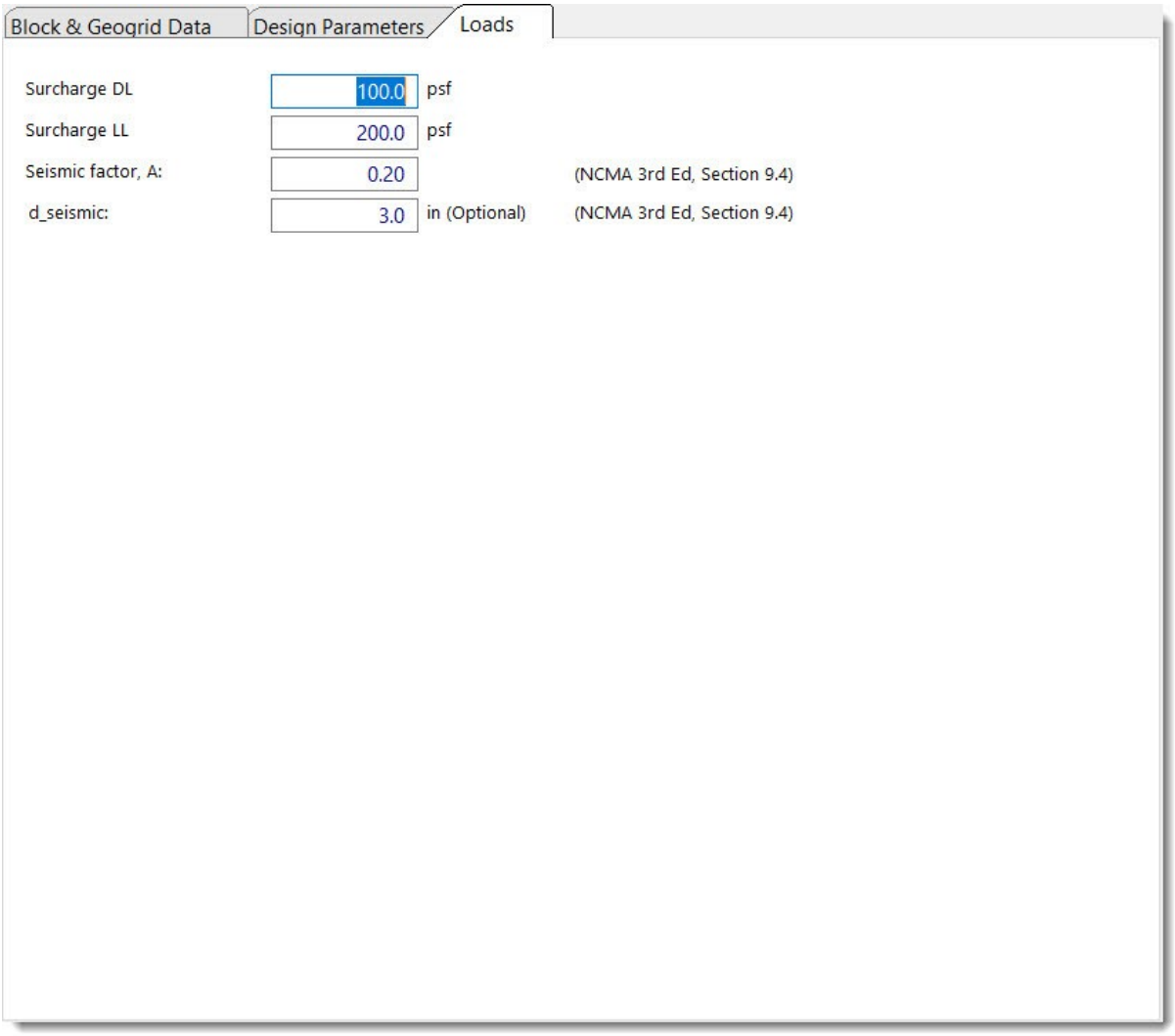
Allowable Soil Bearing Pressure: Enter the allowable soil bearing pressure.

Backfill Slope: Select the backfill slope from the drop-down list box.

Minimum Acceptable Factors of Safety: Specify the minimum acceptable factors of safety for each of the different failure modes, or click the reset button to easily reset all values to defaults as per NCMA Table 5-2.

© 2026 ENERCALC, LLC

13.8.4.5 Loads Tab



Surcharge DL, psf: Enter the dead load surcharge.

Surcharge LL, psf: Enter the live load surcharge – it will not be used to resist overturning or sliding.

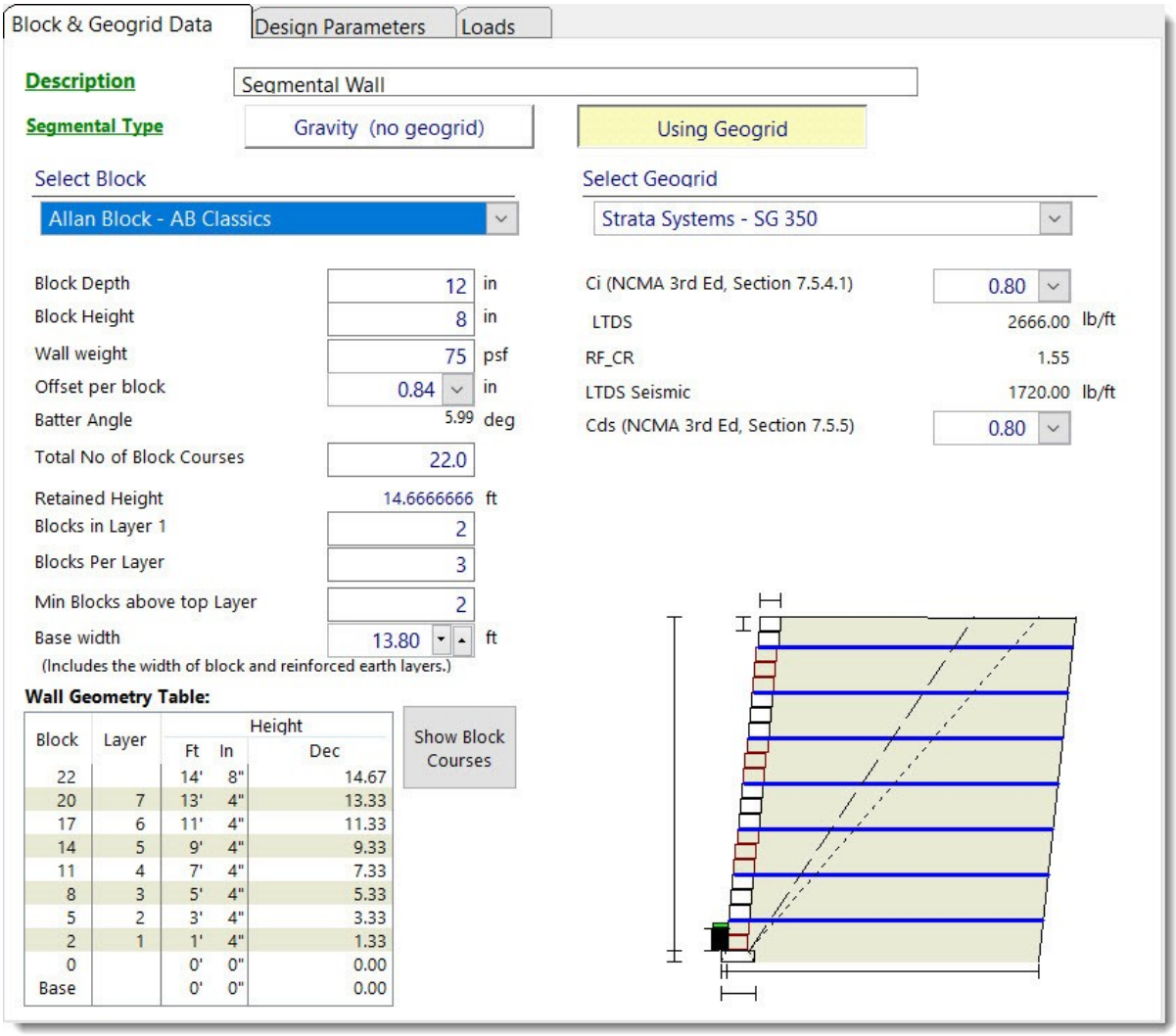
Seismic A factor: Specified horizontal peak ground acceleration expressed as a fraction of the gravitational constant, g. The site-specific values of A represent a 90% probability of that value not being exceeded in 50 years. See NCMA 3rd Edition, Section 9.4.

$d_{seismic}$: The lateral deflection that the retaining wall can withstand during a seismic event, in units of inches. See NCMA 3rd Edition, Section 9.4. Note that this input is optional. If a non-zero value is entered, the program will apply the formulas that consider the maximum permissible deformation. If the value is left zero (blank), the program will apply the formulas that are independent of the deformation.

© 2026 ENERCALC, LLC

13.8.4.6 Geogrid Reinforced Segmental Retaining Walls

Block & Geogrid Data Tab (for Geogrid Reinforced Walls)



Segmental Type: Select either Gravity or Geogrid.

Select Block: From the drop down menu select the vendor and block you want to use. Selecting a block will insert its values into the criteria below.

Block depth, in: This will be automatically input based upon block selection.

Block height, in: This will be automatically input based upon block selection.

Wall weight, psf: This will be automatically input. Note that the full block depth is assumed to be in-filled and an average density of 120 pcf is used. Note that block weight is in units of psf. Think of looking at an

© 2026 ENERCALC, LLC

elevation of a constructed wall. It is the weight of one square foot of finished wall. It is calculated by knowing the weight of the solid portion of the block and assuming that any hollow portions are filled with material that weighs 120 pcf.

Offset per block, in: Select this value from the drop down menu – it is vendor-dependent.

Batter, degrees: This angle will be computed and displayed based upon offset and block height entered.

Total No of Block Courses: Enter the number of block courses.

Retained Height: The retained height is automatically calculated and reported.

Blocks in Layer 1: Enter the number of courses below the first layer of geogrid.

Blocks per Layer: Enter the typical number of courses between layers of geogrid.

Min Blocks above top Layer: Enter the minimum number of courses above the top layer of geogrid.

Base Width: Enter the full base width including wall depth. (usually 60% - 70% of retained height).

Select Geogrid: From the drop down menu select the geogrid vendor and the specific geogrid. Design parameters will be displayed below.

Ci factor: Select the coefficient of interaction for pullout (usually 0.70 – 0.90). Note: The program will not permit a layer of geogrid to be placed under the first course, so the value of Cds, coefficient of direct sliding is hard coded to 1.0 for external sliding checks.

LTDS, lbs/ft: Long Term Design Strength will be automatically inserted based upon the vendor/geogrid selection.

RF_{CR}: Creep Reduction Factor will be automatically inserted based upon the vendor/geogrid selection.

$LTDS_{seismic}$, lbs/ft: Long Term Design Strength for seismic loading will be automatically calculated as LTDS / RF_{CR}.

C_{DS}: Select the coefficient of direct sliding. Reference NCMA 3rd Edition, Section 7.5.5.

© 2026 ENERCALC, LLC

Wall Geometry Table:

Block: Displays the total number of block courses.

Layer: Displays the geogrid layer numbers in ascending order from bottom.

Height: Displays block and layer heights in ft-inches and decimals.

Show Block Courses / Show Grid Layers: Toggles between the display of block courses or grid layers.

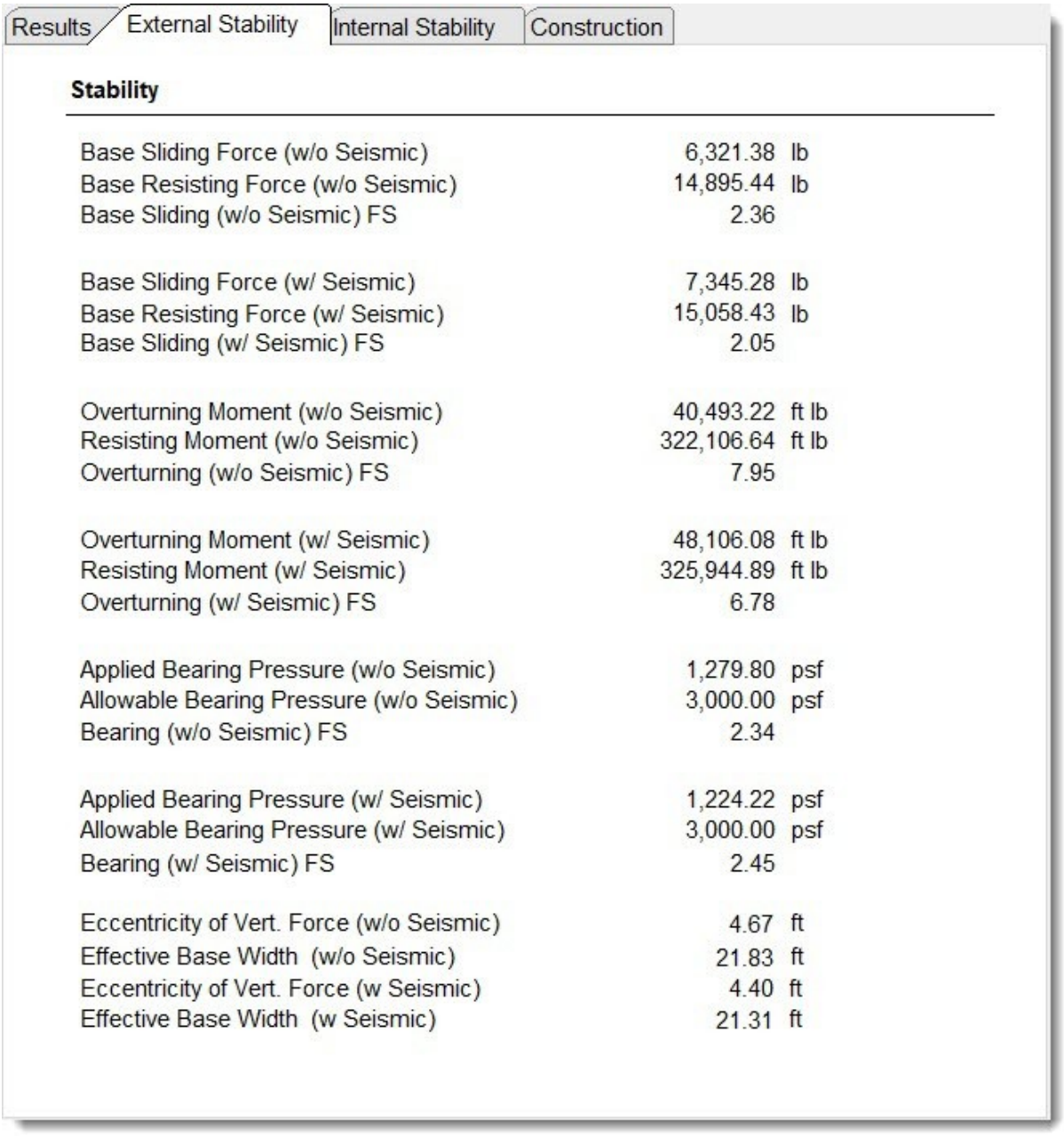
© 2026 ENERCALC, LLC

Results Tab (for Geogrid Reinforced Walls)

Displays the minimum acceptable factor of safety and the minimum actual factor of safety for all failure modes, for the static condition and for the condition that includes seismic loading.

© 2026 ENERCALC, LLC

External Stability Tab (for Geogrid Reinforced Walls)

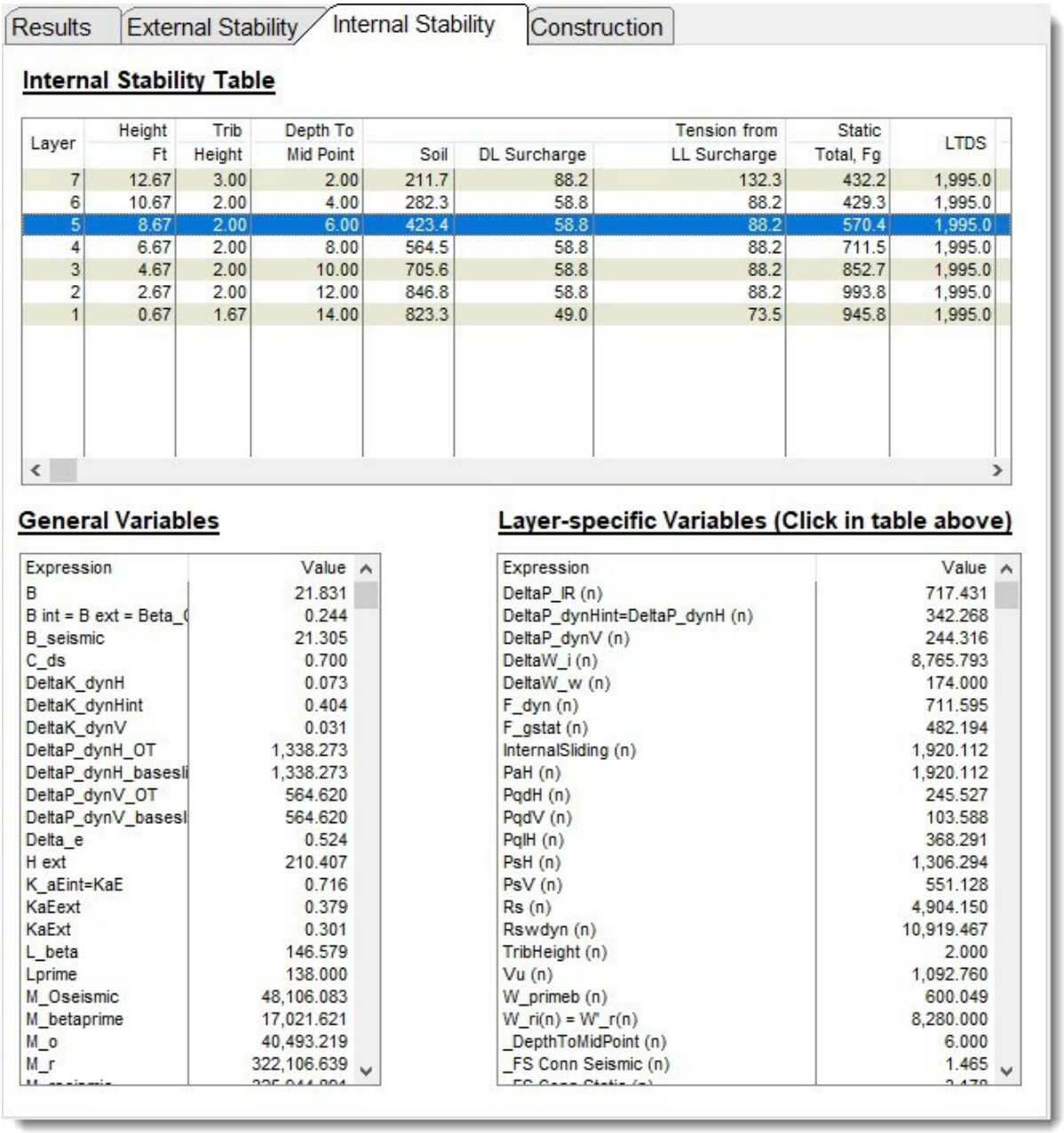


Displays overturning and sliding factors of safety as well as bearing pressure results for conditions with and without seismic loading.

Also reports eccentricity of vertical force and effective base width.

© 2026 ENERCALC, LLC

Internal Stability Tab (for Geogrid Reinforced Walls)



Displays internal stability table:

Layer: Displays geogrid layer numbers.

Height, ft: Displays the height from the base to the elevation of each geogrid layer.

Trib Height: Displays the tributary height of each geogrid layer.

Depth to Midpoint: Displays the distance from the top of the wall to each geogrid layer.

© 2026 ENERCALC, LLC

Tension from Soil: Displays the tension in each layer of geogrid due to lateral earth pressure from the soil.

Tension from DL Surcharge: Displays the tension in each layer of geogrid due to lateral earth pressure from a dead load surcharge.

Tension from LL Surcharge: Displays the tension in each layer of geogrid due to lateral earth pressure from a live load surcharge.

Static Total Tension, Fg: The sum of Tension from Soil plus Tension from DL Surcharge plus Tension from LL Surcharge.

LTDS: Echoes the Long Term Design Strength of the selected geogrid.

$LTDS_{seismic}$: Echoes the Long Term Design Strength for seismic design of the selected geogrid.

Total Tension (w/ Seismic), Fi: Total tension in the geogrid when subjected to the load condition that includes seismic loading.

FS Tensile Overstress (static): LTDS / Static Total Tension

FS Tensile Overstress (w/ seismic): $LTDS_{seismic}$ / Total Tension (w/ seismic), Fi

Pullout Strength: Ability of the geogrid to resist pullout failure.

FS Pullout (static): Pullout Strength / Static Total Tension, Fg

FS Pullout (w/ seismic): Pullout Strength / Total Tension (w/ seismic), Fi

Connection Strength: Connection strength of the selected block/geogrid combination.

FS Connection (static): Connection capcacity / Static Total Tension, Fg

FS Connection (w/ seismic): Connection capcacity / Total Tension (w/ seismic), Fi

Internal Sliding Force (static): The driving force at each layer of geogrid from static loading tending to cause the soil mass above to slide sideways.

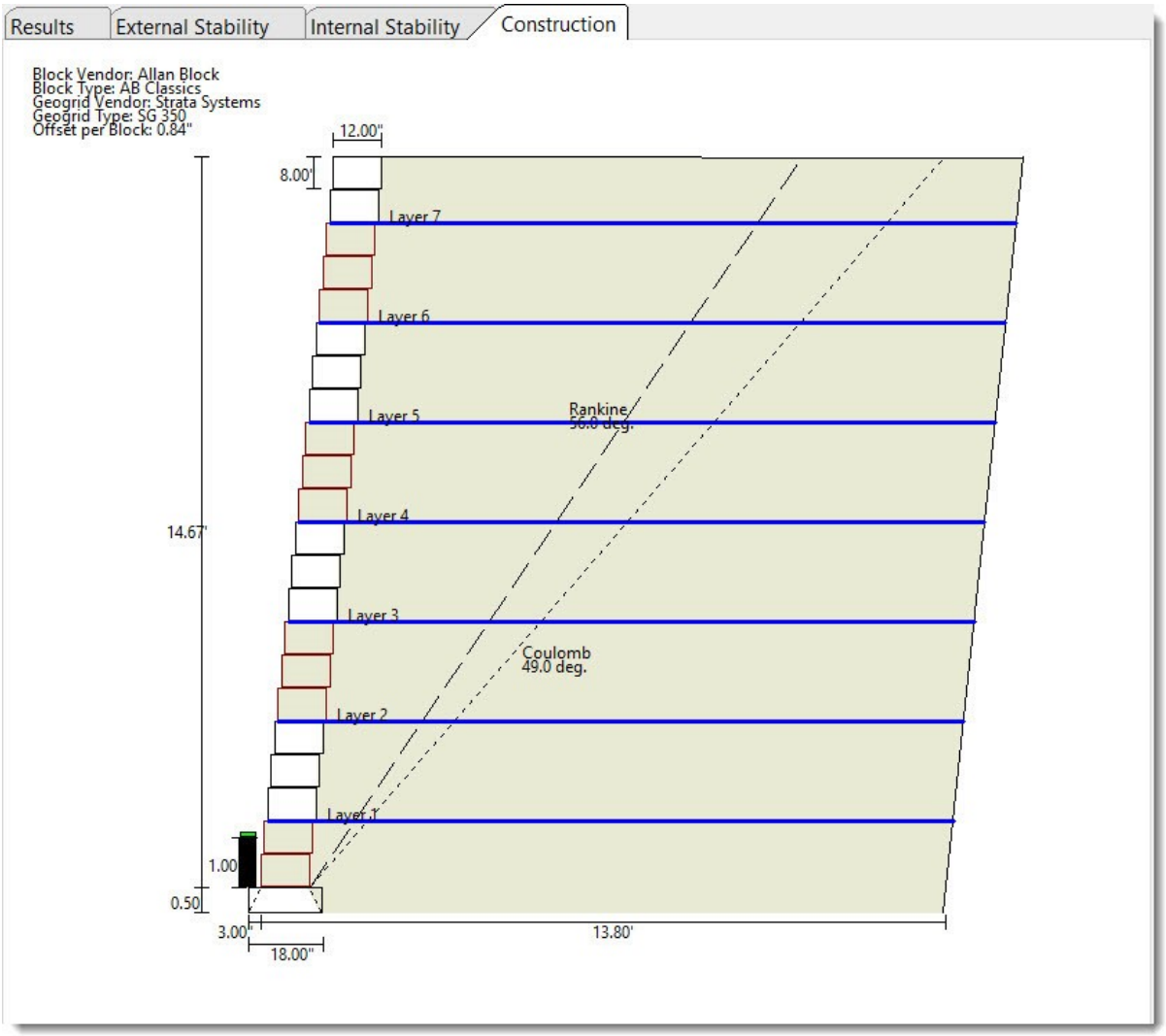
FS Internal Sliding (static): Internal sliding resistance / Static internal sliding force

Internal Sliding Force (w/ seismic):The driving force at each layer of geogrid from loading including seismic tending to cause the soil mass above to slide sideways.

FS Internal Sliding (w/ seismic): Internal sliding resistance / Internal sliding force (w/ seismic)

© 2026 ENERCALC, LLC

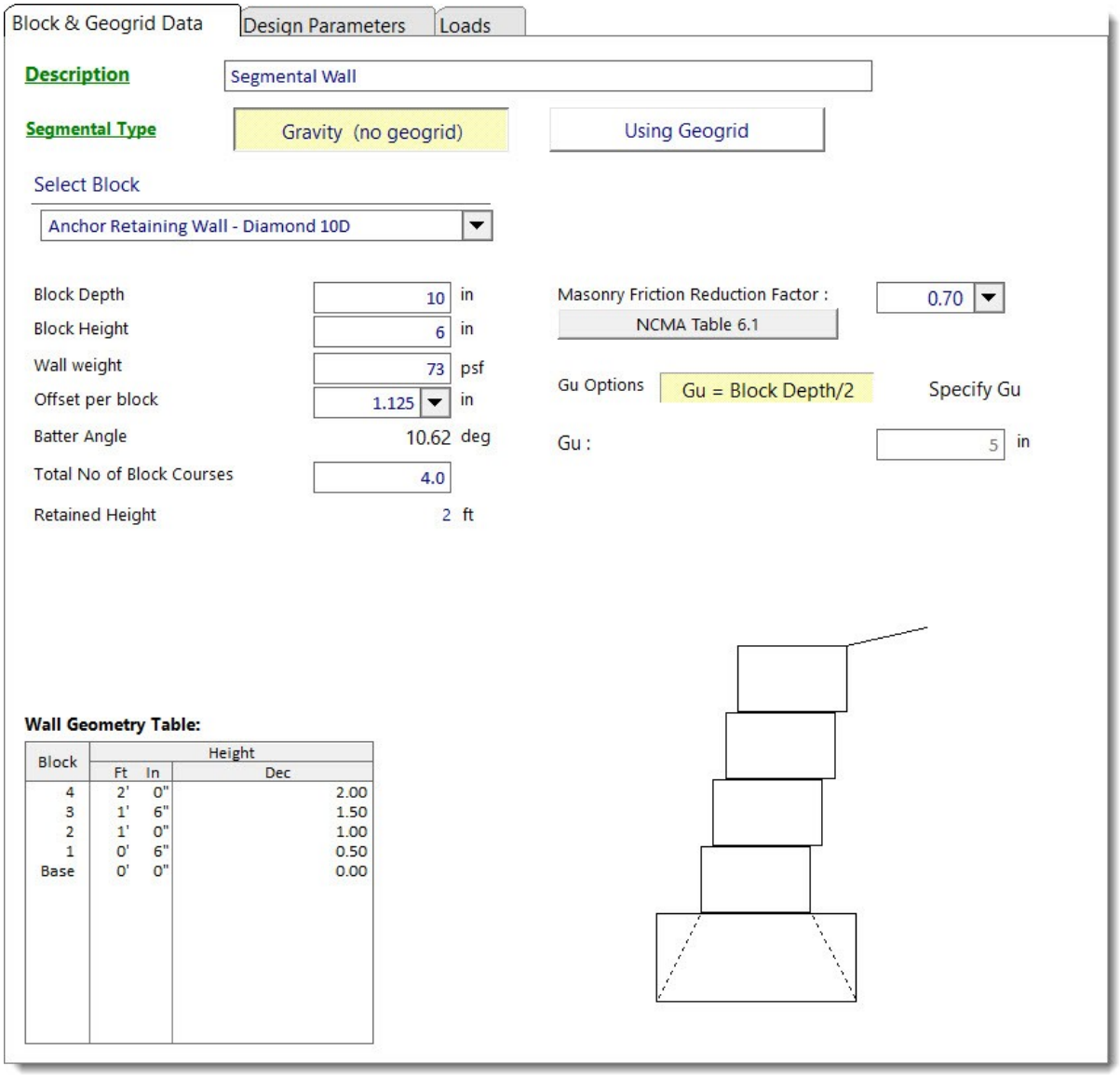
Construction Tab (for Geogrid Reinforced Walls)



Displays schematic drawing of wall, reinforced area, geogrid layers, dimensions, and failure line for Coulomb method.

13.8.4.7 Gravity Segmental Retaining Walls

Block & Geogrid Data Tab for Gravity Walls



Segmental Type: Select either Gravity or Geogrid.

Total No of Block Courses: Enter the number of block courses.

Select block: From the drop down menu select the vendor and block you want to use. Selecting a block will insert its values into the criteria below.

Block Depth, in: This will be automatically input based upon block selection.

Block Height, in: This will be automatically input based upon block selection.

© 2026 ENERCALC, LLC

Wall weight, psf: This will be automatically input. Note that the full block depth is assumed to be in-filled and an average density of 120 pcf is used. Note that block weight is in units of psf. Think of looking at an elevation of a constructed wall. It is the weight of one square foot of finished wall. It is calculated by knowing the weight of the solid portion of the block and assuming that any hollow portions are filled with material that weighs 120 pcf.

Offset per block, in: Select this value from the drop down menu – it is vendor dependent.

Batter, degrees: This angle will be computed and displayed based upon offset and block height entered.

Masonry Friction Reduction Factor: Select the desired Masonry Friction Reduction Factor. (The adjacent button offers a quick view of NCMA Table 6.1 for some guidance on this factor.)

Gu Options: Select the desired method of calculating/specifying the value of Gu.

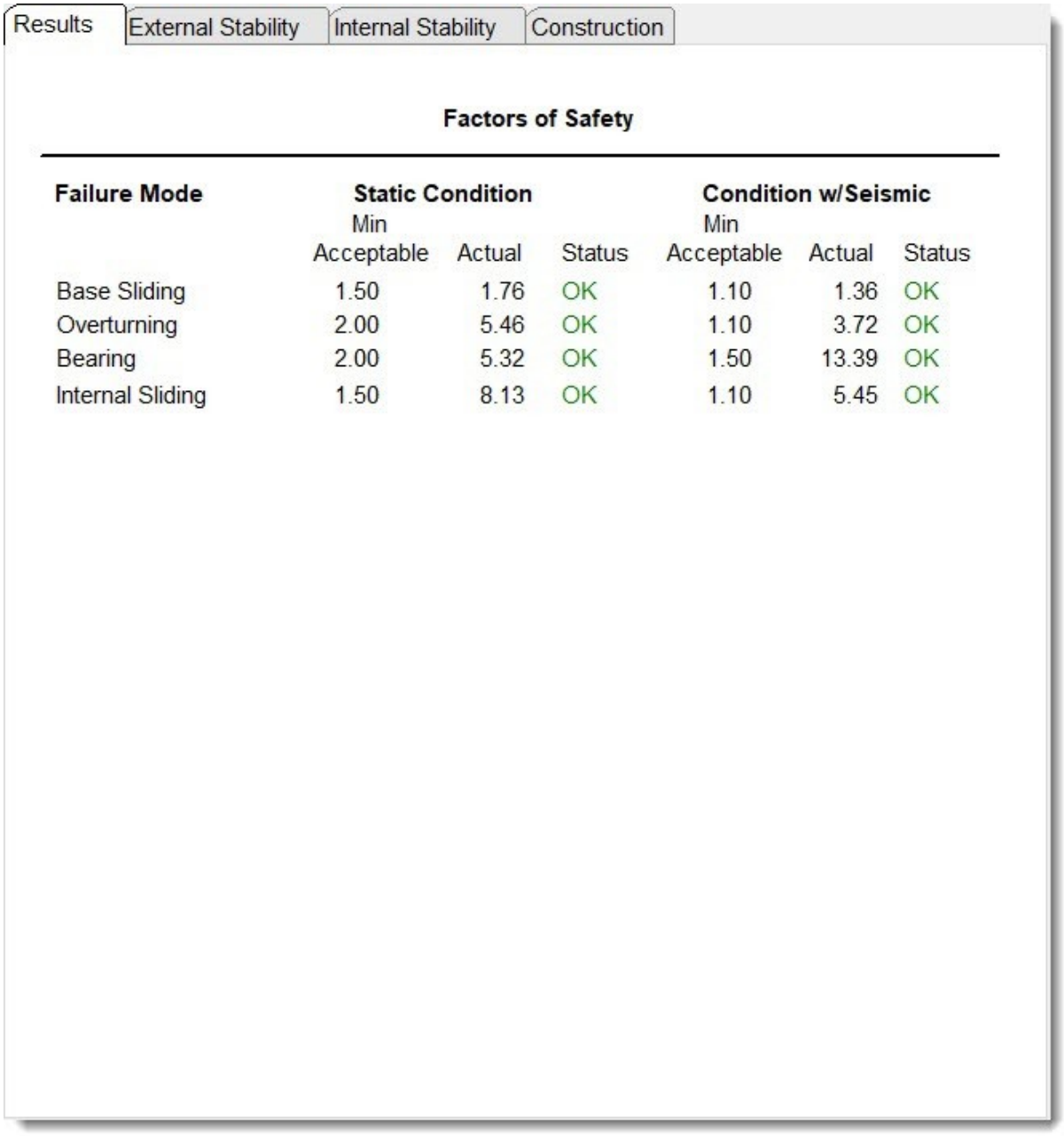
Wall Geometry Table:

Block: Displays the total number of block courses.

Height: Displays block and layer heights in ft-inches and decimals.

© 2026 ENERCALC, LLC

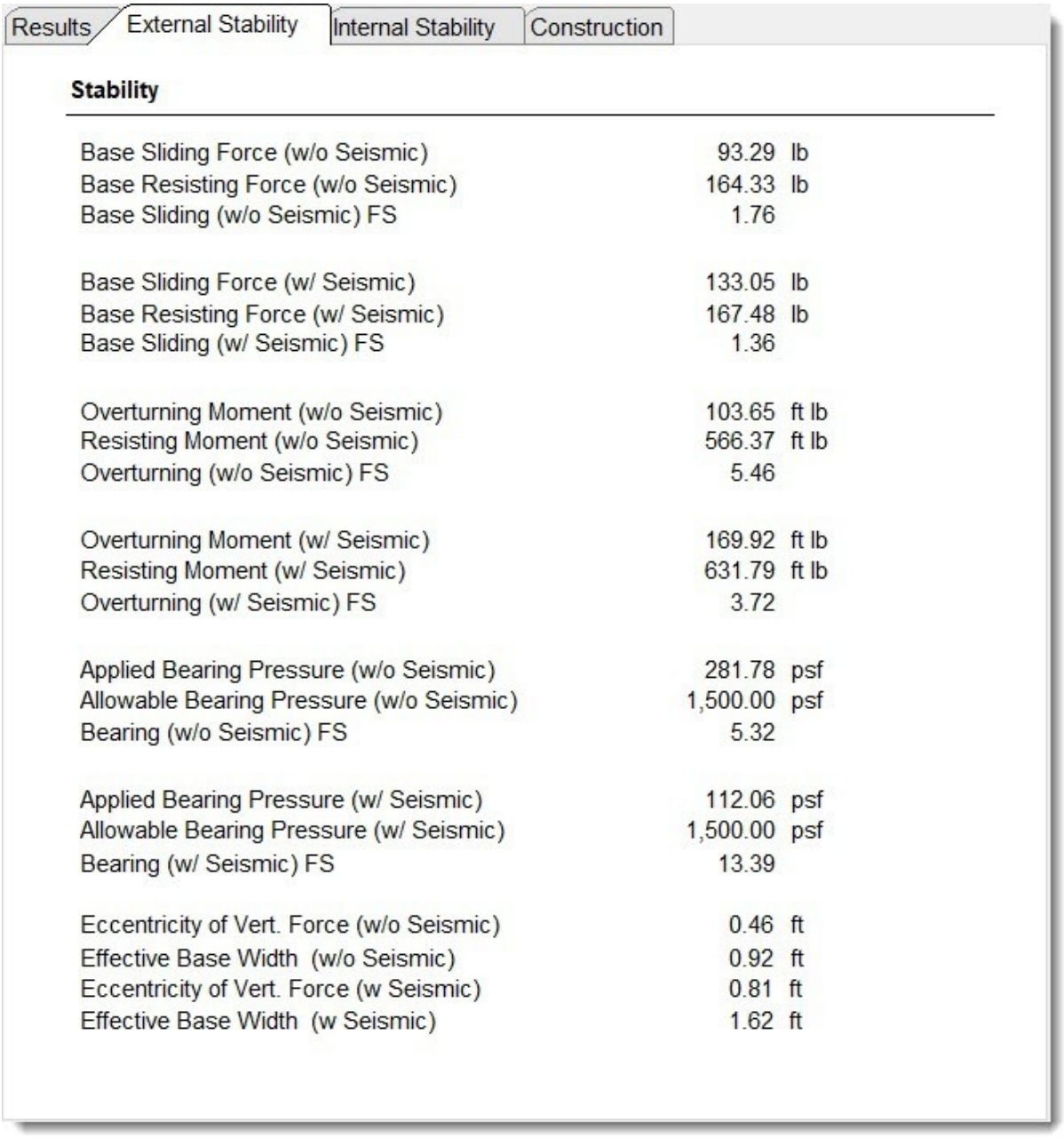
Results Tab (for Gravity Segmental Retaining Walls)



Displays the minimum acceptable factor of safety and the minimum actual factor of safety for all applicable failure modes, for the static condition and for the condition that includes seismic loading.

© 2026 ENERCALC, LLC

External Stability Tab (for Gravity Segmental Retaining Walls)

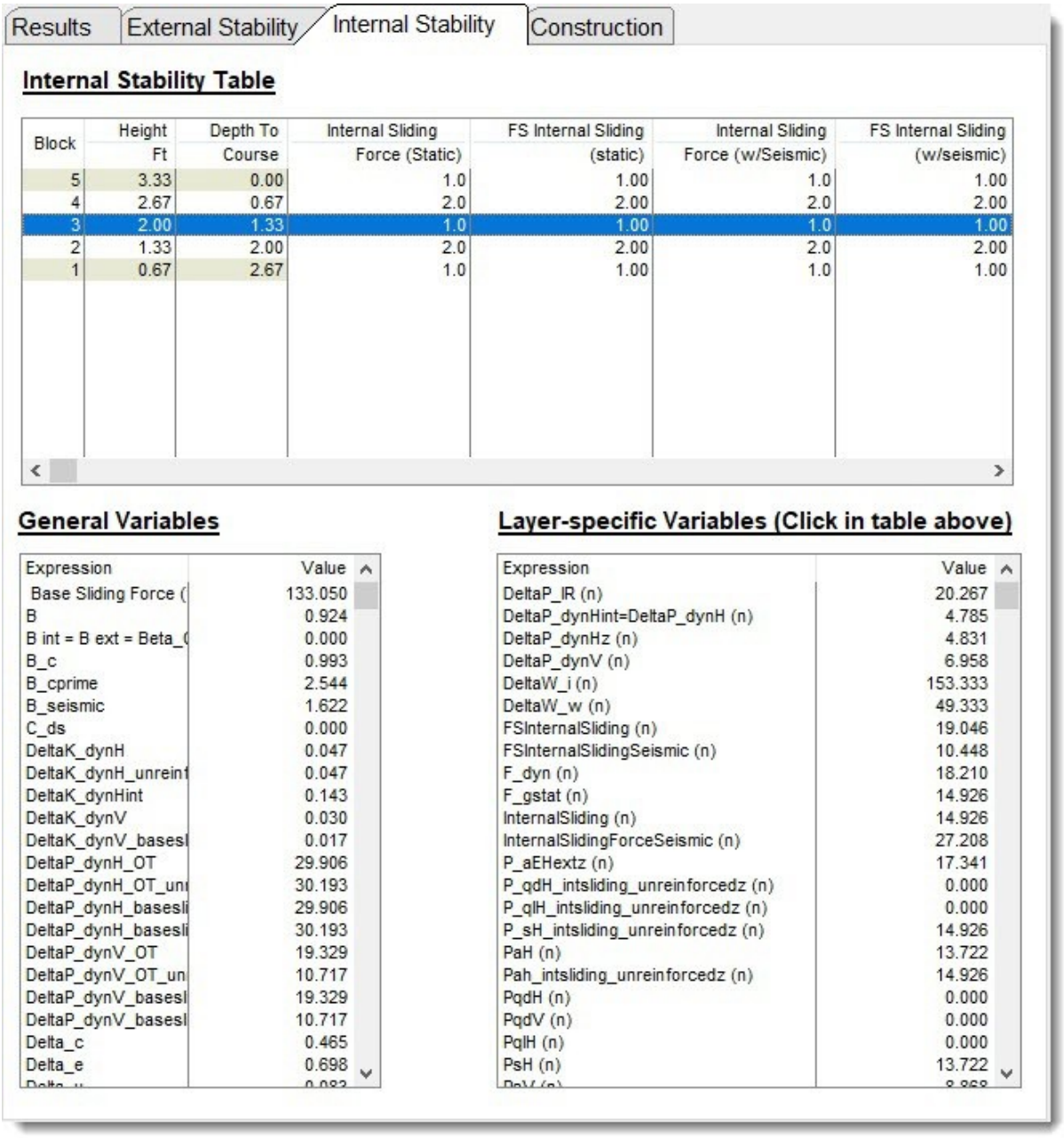


Displays overturning and sliding factors of safety as well as bearing pressure results.

Also reports eccentricity of vertical force and effective base width.

© 2026 ENERCALC, LLC

Internal Stability Tab (for Gravity Segmental Retaining Walls)



Displays internal stability table:

Block: Displays course numbers.

Height, ft: Displays the height from the base to the elevation of the top of each course.

Depth to Course: Displays the distance from the top of the wall to the top of each course.

© 2026 ENERCALC, LLC

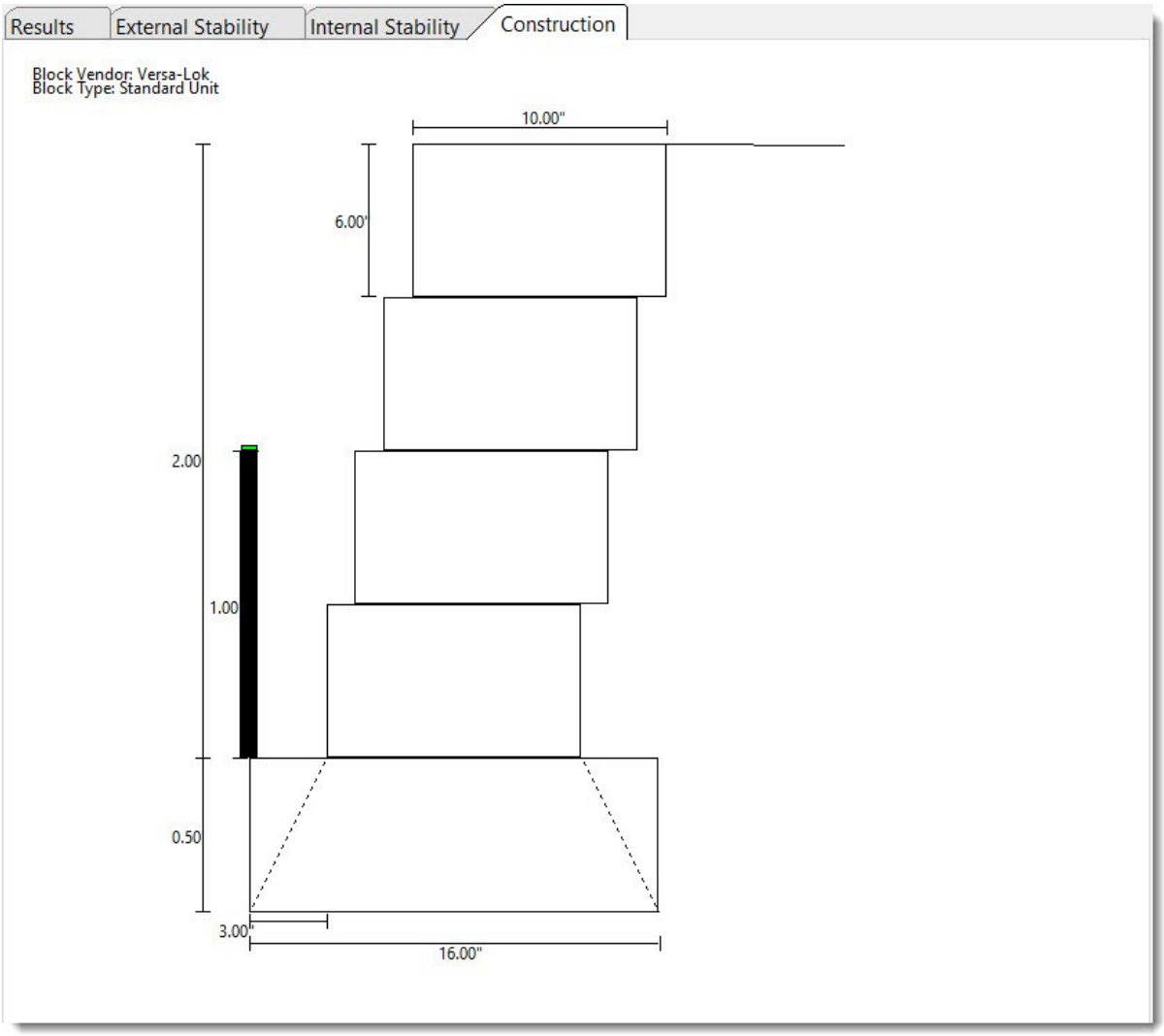
Internal Sliding Force (static): The driving force at each course from static loading tending to cause the soil mass above to slide sideways.

FS Internal Sliding (static): Internal sliding resistance / Static internal sliding force

Internal Sliding Force (w/ seismic): The driving force at each course from loading including seismic tending to cause the soil mass above to slide sideways.

FS Internal Sliding (w/ seismic): Internal sliding resistance / Internal sliding force (w/ seismic)

Construction Tab (for Gravity Segmental Retaining Walls)



Displays schematic drawing of wall.

© 2026 ENERCALC, LLC

13.8.4.8 Methodology / Analysis & Design Assumptions

References used for the development of this program are listed in Appendix E.

Surcharge can be composed of either dead load, live load, or both.

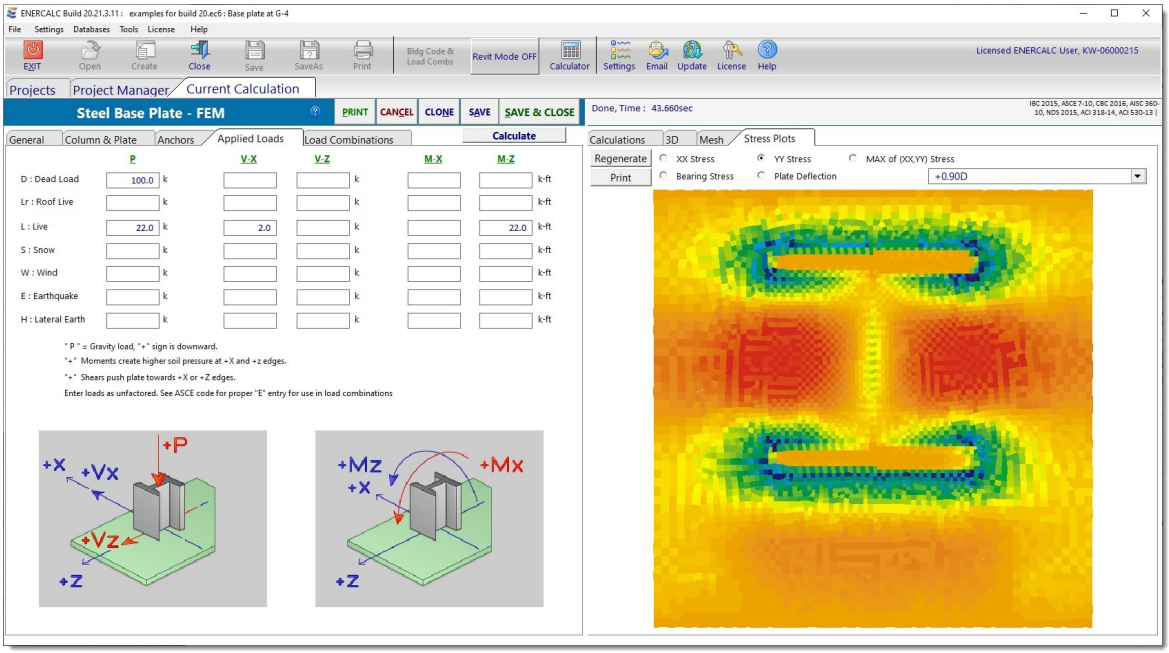
The design of segmental retaining walls generally follows the guidelines in *Design of Segmental Retaining Walls, 3rd Edition* published by the National Concrete Masonry Association (NCMA). Some assumptions have been made to simplify the program (as stated in the program), yet cover most construction practices and design requirements.

13.9 Miscellaneous

Please select a subtopic.

13.9.1 Steel Base Plate FEM

This module applies engineering principles to the design of a steel base plate though the use of a finite element analysis of the plate.



General

AISC provides solutions for steel base plates with various loads in Design Guide 1. But the design guide does make some simplifying assumptions (infinitely rigid substrate, uniform or triangular bearing pressure of a certain magnitude, extent of bearing pressure, etc). And the design guide also has some limitations (concentric loading). While the design guide has proven to be a reliable design reference over the years, there are some applications where a different tool is required. The purpose of this module is to provide an alternative to the design guide.

© 2026 ENERCALC, LLC

The methods of this module are based in engineering principles, but it should be noted that it employs Finite Element Analysis and a *different* set of assumptions (spring support, non-uniform bearing pressure) to arrive at code-compliant designs. For this reason, one should not be surprised to see that the module results in a different design than the AISC Design Guide methods, even when the same starting parameters are entered, such as for one of the design guide examples.

Flexural Design Approach

The code dictates that Mn shall be calculated with the smaller of Z or 1.6S. For a rectangular cross section, Z is always less than 1.6S.

Therefore nominal moment should always be controlled by Z * Fy. ENERCALC 3D reports plate stresses as M/S.

This means that the actual stress can be modified by dividing by the ratio of Z/S.

Z = (bd^2/4)

S = (bd^2/6)

Z/S = 6/4 = 1.5

So the stresses determined by ENERCALC 3D, the underlying Finite Element Analysis application, are then reduced by a factor of 1.5 when they are passed to this module. In this way, the stresses can be compared on par with allowable stresses determined as follows:

ASD:

Allowable stress is Fy / Omega.

LRFD:

Allowable stress is phi * Fy.

Target Mesh Element Size

Typical mesh element target size is automatically established by finding the minimum of plate width and plate height, and dividing that value by 50. The target mesh element size is automatically reduced in close proximity to the column footprint, where stress concentrations occur.

© 2026 ENERCALC, LLC

Substrate Stiffness

The plate is supported on a bed of compression-only area springs. In reality, the stiffness of the resistance under the plate is a function of the pedestal material, the footing material and configuration, and the stiffness of the subgrade. To simplify the condition to something that can reasonably be modeled and solved, the stiffness of the springs is determined by using $\delta = PL/(AE)$, where:

δ = deflection
P = axial load
L = length of material that is under compression
A = area of material that is under compression
E = modulus of elasticity

Note that this can be rearranged into the form of $P/(A * \delta) = E / L$, where:

$P / (A * \delta)$ may be easier to visualize as $(P / A) / \delta$, which is the value of the spring constant we are seeking, and
E and L are known quantities.

In this case, E is the modulus of elasticity of concrete, and L represents the length of concrete in compression. It stands to reason that a logical value for L is the length of the anchor rods, as they create the tensile portion of the couple that creates compression in the concrete.

General

Steel Design Method
Select between ASD or LRFD design methods.

Plate Material
Specify the yield strength of the base plate material.

Concrete Support Material
Specify f'c, the 28-day compressive strength of concrete used to support the base plate. Also collects Ec, the modulus of elasticity of concrete. This is used to determine the stiffness of the compression-only springs in the underlying ENERCALC 3D model. Specify nominal bearing strength of concrete. This is used to calculate an allowable bearing stress.

ASD: Omega per AISC J.8
Displays the capacity reduction factor, Omega, to be used in ASD per AISC 360 Section J.8 as follows:
IBC 2015 references AISC 360-10: Omega = 2.31
IBC 2018 references AISC 360-16: Omega = 2.31
IBC 2021 references AISC 360-16: Omega = 2.31

© 2026 ENERCALC, LLC

LRFD: Phi per AISC J.8

Displays the capacity reduction factor, Phi, to be used in LRFD per AISC 360 Section J.8 as follows:
IBC 2015 references AISC 360-10: Phi = 0.65
IBC 2018 references AISC 360-16: Phi = 0.65
IBC 2021 references AISC 360-16: Phi = 0.65

User Defined Allowable Bearing Pressure in Lieu of AISC 360 Section J.8

The module also offers the option to override the AISC 360 Section J.8 calculated values with a user-specified input value for the allowable bearing pressure. When this option is used, the user-specified value will be multiplied by phi or divided by Omega.

Column & Plate

Steel Column Properties

Type the AISC section name in the entry and press [**Tab**]. The module will look up the section in the Steel database and, if found, will retrieve the values. The name must be typed just as it appears in the AISC Steel Construction Manual.

Or click the [**Section Database**] button to display the built-in steel database and select a section.

Column Rotation

Select orientation of column on plate. Note that plate width dimension remains parallel to the X axis.

Column Offset

Specify offset of column from the center of the plate. Note that plate remains centered on the concrete support.

Base Plate Size

Specify the plate length, width and thickness.

Concrete Support

Specify the length and width of the concrete support.

Anchors

Anchor Geometry

The module provides two options for specifying anchor layout: Graphical Anchor Layout and General Anchor Layout.

When using Graphical Anchor Layout, the coordinate axes are shown, and anchor locations are specified by selecting the desired checkboxes and providing the relevant dimensions. To provide an anchor at a location, simply select the checkbox. To skip an anchor, clear the corresponding checkbox.

© 2026 ENERCALC, LLC

When using the General Anchor Layout, specify the number of anchors, then specify the X an Z coordinates of each anchor with respect to the origin at the center of the plate. For convenience, checkboxes are provided to automatically specify an anchor placed symmetrically about one or both of the axes. This makes it possible to replicate multiple anchors without actually having to enter the coordinates of every anchor.

Anchor Diameter

Specify typical anchor diameter.

Anchor Length

Specify typical anchor length. Note that the program uses this value to estimate the length of concrete that experiences significant compressive forces and therefore affects the internally calculated spring constant used in the finite element model.

Tension Capacity in Concrete

Tension capacity due to controlling concrete failure mode considering ACI 318-14 Sections 17.4.2 through 17.4.5 after all capacity factors are applied. The applied tension is compared to phi times this value to report a ratio.

Shear Capacity in Concrete

Shear capacity due to controlling concrete failure mode considering ACI 318-14 Sections 17.5.2 and 17.5.3 after all capacity factors are applied. The applied shear is compared to phi times this value to report a ratio.

Tensile Steel Strength, Fu

Used to calculate a ratio for tensile strength due to steel failure mode in accordance with ACI 318-14 Section 17.4.1. Also used to calculate a ratio for shear strength due to steel failure mode in accordance with ACI 318-14 Section 17.5.1.

Applied Loads

P - Axial Load

This column of entries specifies the axial load applied to the base plate. Note that positive values represent downward loads.

Vx and Vz - Shear

These columns of entries specify the shear applied parallel to the named axis and to be resisted by the anchor bolts.

Mx and Mz - Moment

These columns of entries specify the moment applied about the named axis and to be resisted by the anchor bolts and concrete.

© 2026 ENERCALC, LLC

Load Combinations

This tab displays the load combinations used for either the ASD or LRFD method.

Calculate Button

The Calculate button initiates the analysis and design process, which includes:

- Gather all input parameters and pass the input to ENERCALC 3D.
- Create the optimal mesh for the specified parameters.
- Assign materials, properties, supports, spring constants, releases, loads, etc. to the ENERCALC 3D model.
- Run the ENERCALC 3D analysis.
- Return analysis results from ENERCALC 3D to the Steel Base Plate module.
- Perform code checks and display results.

Click the Calculate button any time the input parameters change.

Calculations

Summary

The Summary sub-tab reports the selected design method, the governing stress ratio, and the plate dimensions. It then provides specifics on the controlling condition for plate flexure, bearing stress, and bolt tension.

Plate Bending Stresses

The Plate Bending Stresses sub-tab lists the analyzed load combinations, and reports stress on the xx edge and the zz edge of each shell in the underlying ENERCALC 3D model. It also reports the location of the center of each shell.

Plate Bearing Stresses

The Plate Bearing Stresses sub-tab lists the analyzed load combinations, and reports bearing stress for each shell in the underlying ENERCALC 3D model. It also reports the location of the center of each shell.

Anchor Forces

The Anchor Forces sub-tab lists the analyzed load combinations, and reports anchor tensile forces and geometric locations.

© 2026 ENERCALC, LLC

Plate Deflections

The Plate Deflections sub-tab lists the analyzed load combinations, and reports deflection values at the center of each shell as well as the location of the center of each shell. Negative values represent downward deflections. Note that if LRFD is used, the plate deflections will be reported based on factored loads. The main purpose of this tab is to provide the data necessary to evaluate whether the plate is behaving as rigid or flexible. So the absolute deflections are not the focus of this tab. Its purpose is to view the "trend" rather than the detail.

3D Rendering

The 3D tab displays a 3D rendering of the base plate with its supported column and supporting concrete. It includes options to selectively display other relevant items.

Mesh

The Mesh tab displays a view of the mesh that will be used in the underlying ENERCALC 3D model. The anchor numbers are shown, and they coordinate with the numbers used on the Anchor Forces tab. The Regenerate button can be used to refresh the view of the mesh if anything has changed, such as plate dimensions, column shape, column orientation, column location, etc. The mesh density gets increased in close proximity to the column. This allows the program to pick up on stress concentrations where the load is introduced to the plate. As a beneficial side effect, it also makes it possible to see the column location when viewing the mesh. This can be useful in verifying the column location and orientation.

© 2026 ENERCALC, LLC

Stress Plots

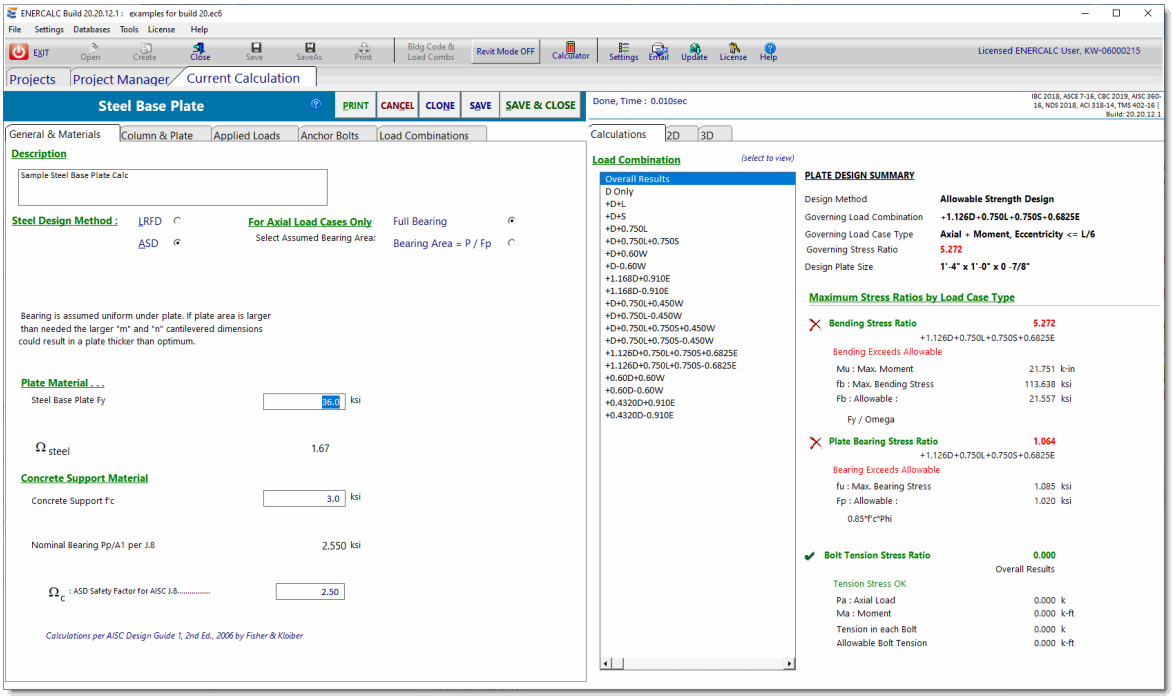
The Stress Plots tab displays a variety of stress distributions using color rendering for the selected load combination.

- XX Stress displays the top/bottom flexural stress on the xx edge of each shell for the selected load combination. The xx edge is the edge that is generally parallel to the global Z axis (perpendicular to the global X axis).
- YY Stress displays the top/bottom flexural stress on the yy edge of each shell for the selected load combination. The yy edge is the edge that is generally parallel to the global X axis (perpendicular to the global Z axis).
- Max of XX, YY Stress displays the maximum of either the XX Stress or the YY Stress on every shell for the selected load combination.
- Bearing Stress displays the bearing stress distribution for the selected load combination. Note that this module uses compression-only springs to model the interface between the steel plate and the concrete support. So certain load combinations with significant moment may show large areas with no bearing stress. This signifies that the compression-only springs in that zone have been deactivated by the iterative solution process.
- Plate Deflection displays color gradients to represent the vertical displacement of the individual shells. Red represents downward displacement. Blue represents upward displacement.

© 2026 ENERCALC, LLC

13.9.2 Steel Base Plate

This module designs steel column base plates according to the latest Edition AISC Steel Construction Manual and the AISC Design Guide 1, Second Edition.



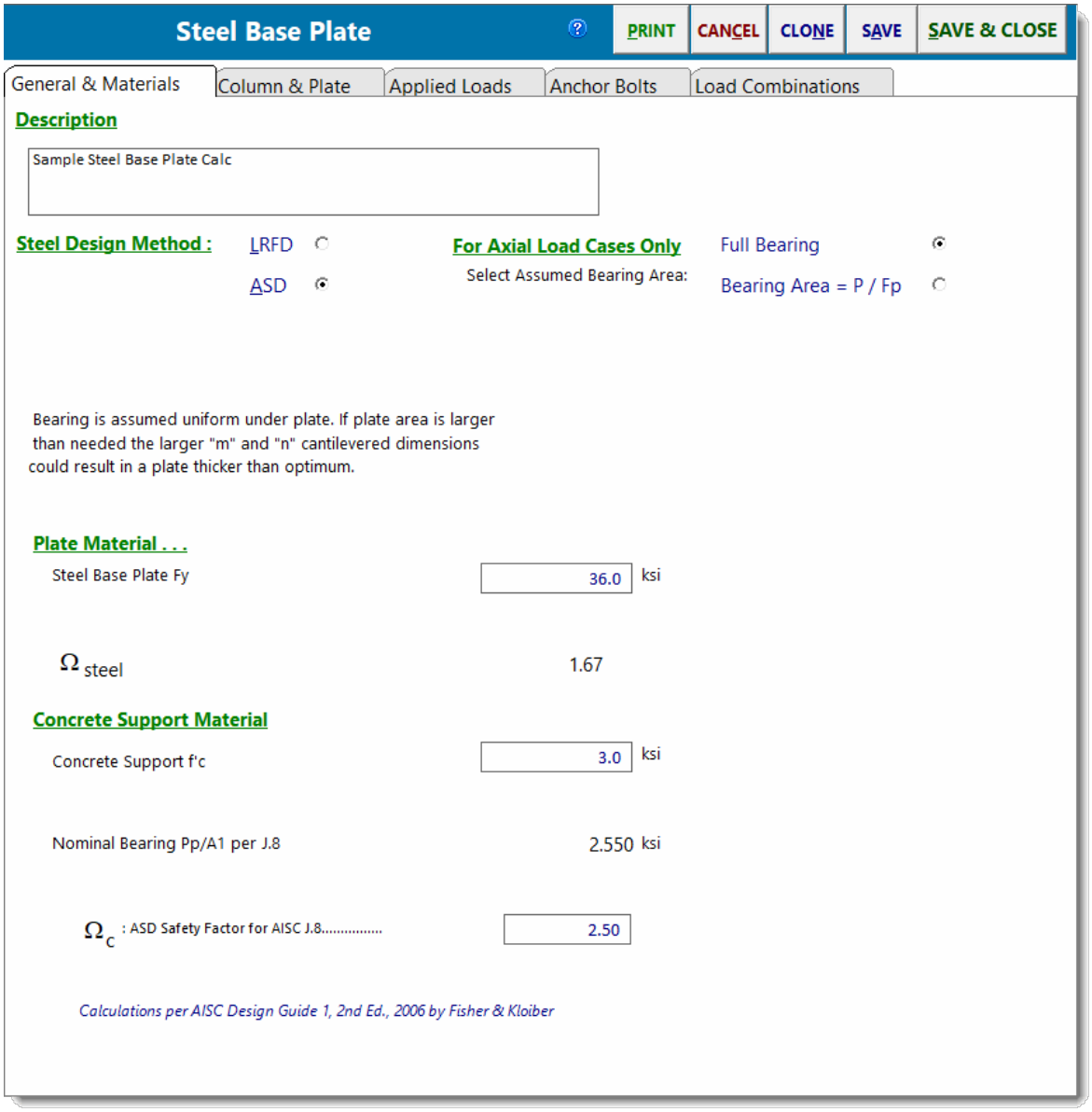
This module handles base plate design for the following conditions:

- where the resultant eccentricity is zero,
- where the resultant eccentricity is within the middle third for full bearing pressure,
- where the resultant eccentricity is outside the middle third resulting in a triangular pressure distribution on part of the base plate, and
- extreme eccentricity conditions where anchor bolts are required.

This module does not handle conditions where NET UPLIFT is present. When the summation of factored axial loads in a load combination is negative, then uplift controls and the module will not calculate. Under these conditions, a red error message will be displayed in the status bar to notify you of this condition. Consider using the Steel Base Plate by FEM module for these situations.

© 2026 ENERCALC, LLC

General & Materials



Steel Design Method

Select between ASD or LRFD design methods.

© 2026 ENERCALC, LLC

For Axial Load Cases Only....

This selection controls the upward pressure used to design the plate. A brief description of each choice is provided below the buttons for each selection.

Full Bearing: Bearing is assumed uniform under plate. If plate area is larger than needed, the larger "m" and "n" cantilevered dimensions could result in a thicker plate than would be required if the pressure was calculated by the other option.

Bearing Area = P / Fp: Base plate is considered flexible with bearing concentrated close to column. Maximum Fp is used to calculate "design" minimum plate size. Determining the bearing pressure by this method might result in thicker plates if bending stress is high between webs or within pipe or tube walls.

Steel Base Plate Fy

Specify the yield strength of the base plate material.

ASD: Omega

Enter the capacity reduction factor, Omega, to be used in ASD per AISC 360.

LRFD: Phi

Enter the capacity reduction factor, Phi, to be used in LRFD per AISC 360.

Concrete Support f'c

28-day compressive strength of concrete used to support the base plate.

ASD: Omega per AISC J.8

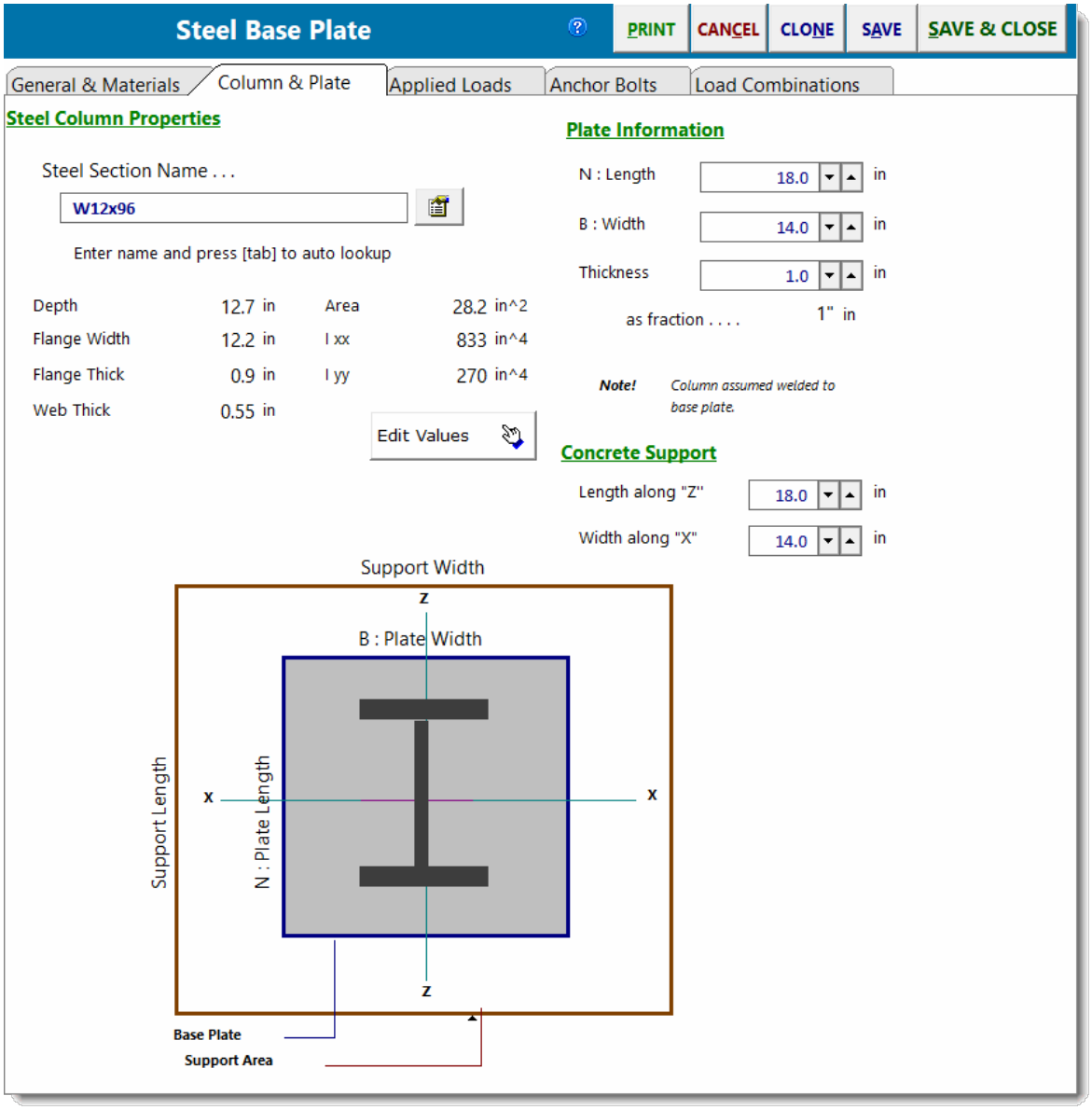
AISC 360 Section J.8 specifies Omega as 2.5. This entry allows the value to be modified.

LRFD: Phi per AISC J.8

AISC 360 Section J.8 specifies Phi as 0.6. This entry allows the value to be modified.

© 2026 ENERCALC, LLC

Column & Plate



Steel Section Name & Database Button

Type the AISC section name in the entry and press [**Tab**]. The module will look up the section in the Steel database and, if found, will retrieve the values. The name must be typed just as it appears in the AISC Steel Construction Manual.

Or click the [**Section Database**] button to display the built-in steel database and select a section.

© 2026 ENERCALC, LLC

[Edit Values] Button

Clicking this button will allow you to enter the steel properties.

Plate Information

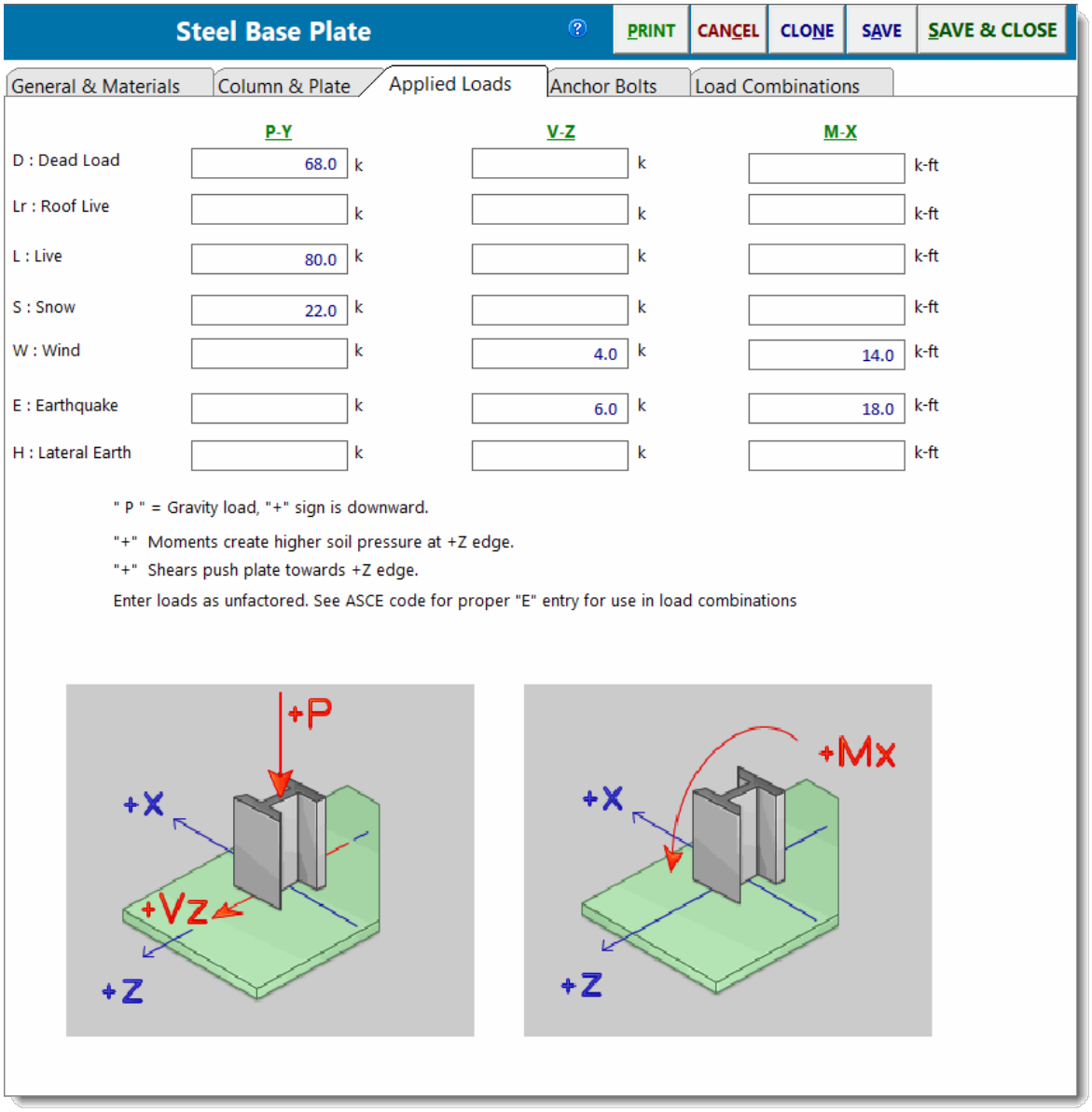
Enter the length, width and thickness of the base plate. Use the buttons to quickly change the values...the results are instantly recalculated.

Concrete Support

Enter the support dimensions that will be used to calculate the allowable bearing pressure increase, A1 * sqrt(A2/A1).

© 2026 ENERCALC, LLC

Applied Loads



Py - Axial Load

This column of entries specifies the axial load applied to the base plate. Note that positive values represent downward loads.

Vz - Shear

This column of entries specifies the shear applied parallel to the plate and to be resisted by the anchor bolts.

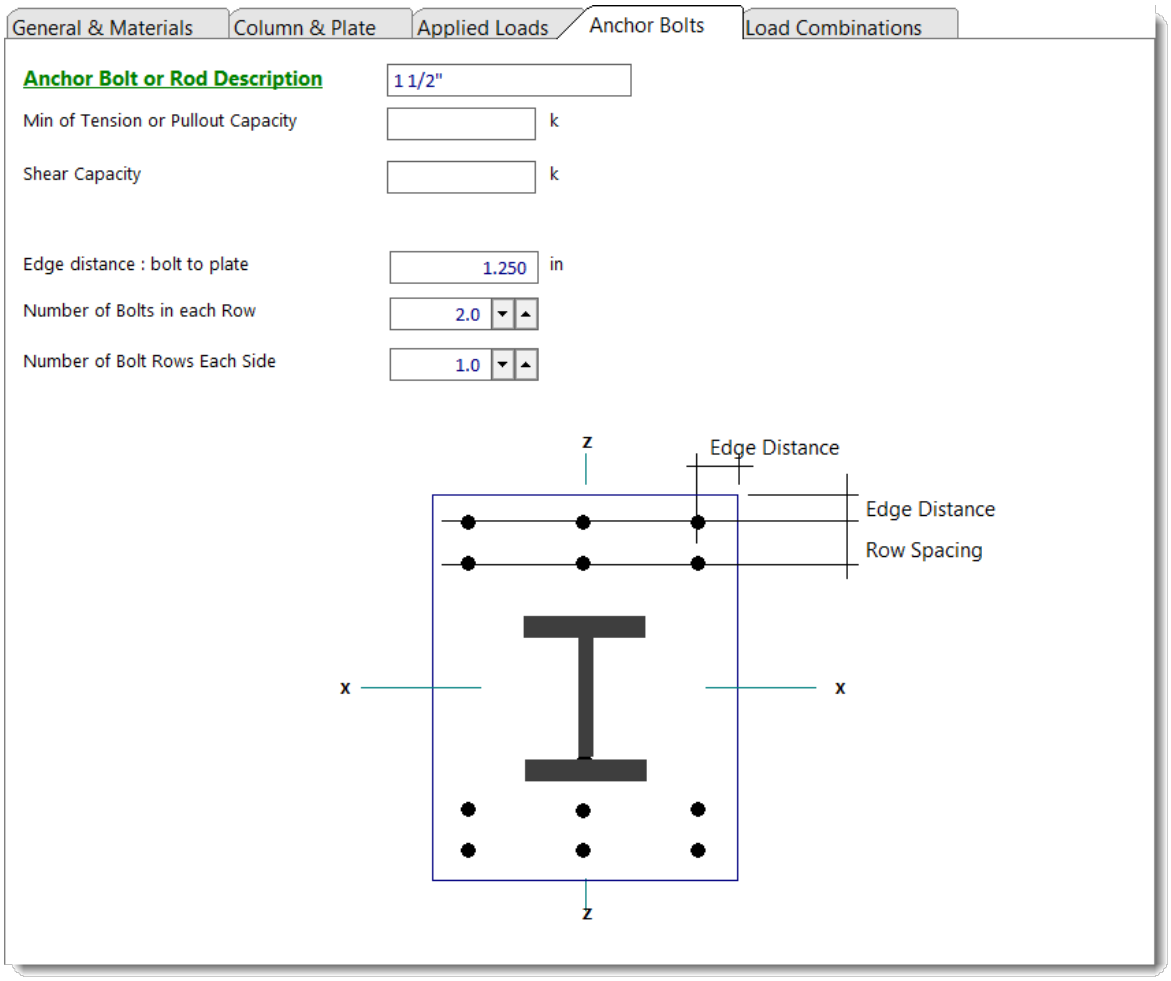
Mx - Moment

This column of entries specifies the moment that the column applies to the plate.

© 2026 ENERCALC, LLC

Anchor Bolts

Items on this tab specify the strength and location of the anchor bolts that will resist shear and tension.



Description

Text description of the bolt for your reference. Not used by module.

Tension Capacity

Net tension capacity of bolt after all capacity factors are applied.

Shear Capacity

Net shear capacity of bolt after all capacity factors are applied.

Edge Distance

Distance from edge of plate to center of bolt closest to edge.

© 2026 ENERCALC, LLC

Number of Bolts in a Row

The "Row" referred to is a row of bolts at the plate edge that will take any tension force.

Number of Bolt Rows

Number of rows of bolts on each side.

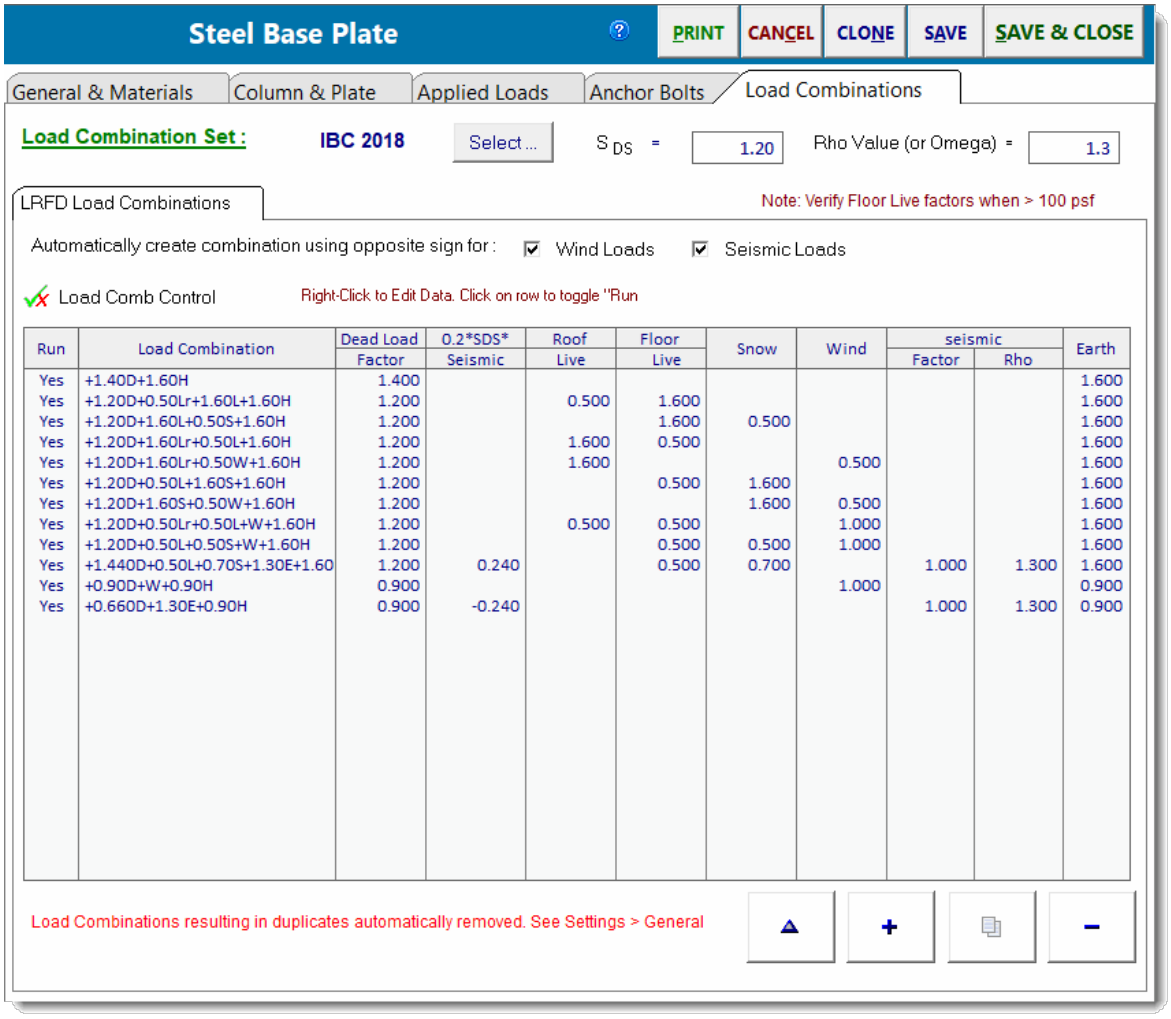
Row Spacing

Spacing of rows when more than one row is used.

Load Combinations

This tab displays the load combinations used for either the ASD or LRFD method.

The screen capture below shows a sample of the load combinations factors:

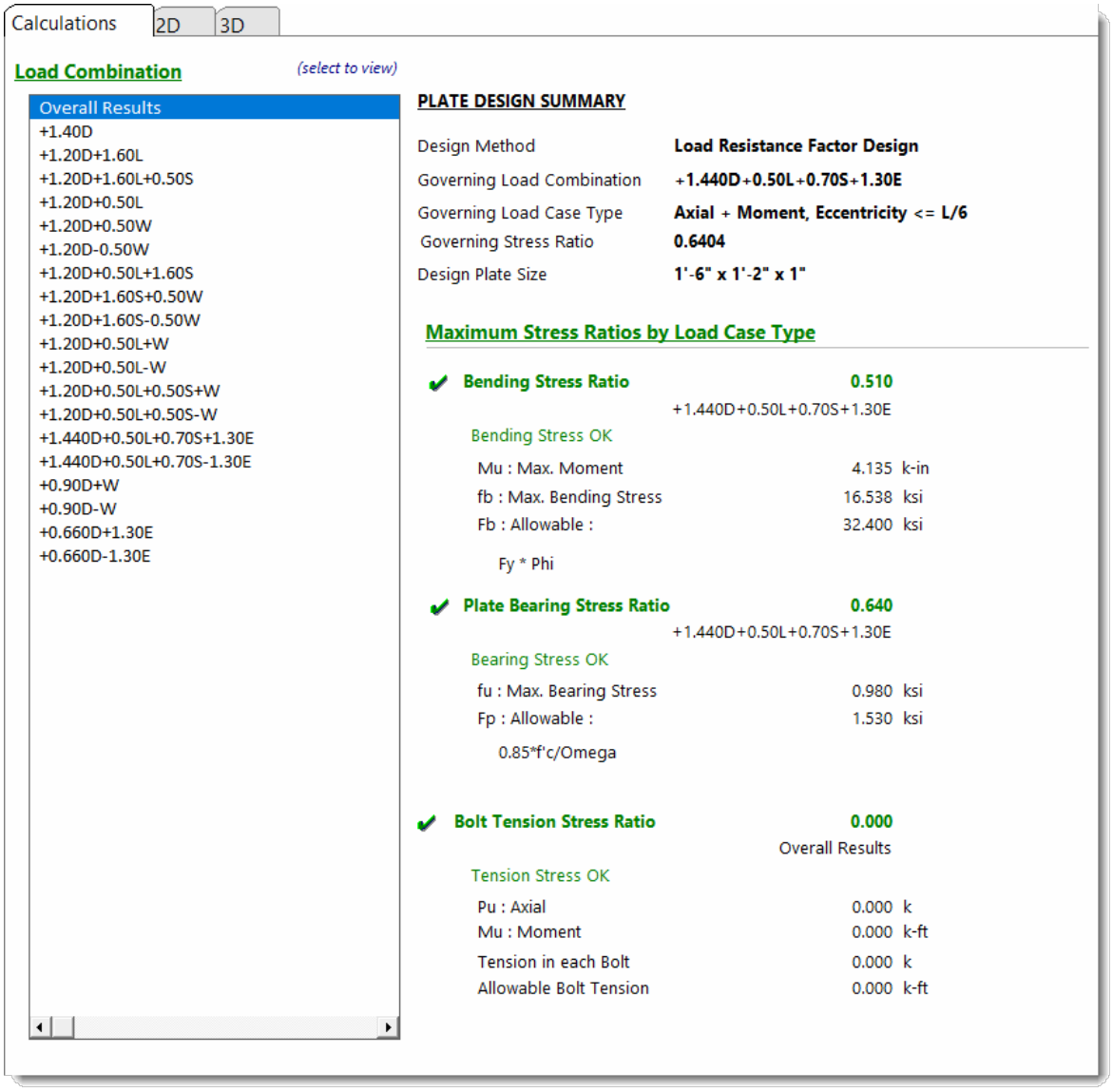


Run	Load Combination	Dead Load Factor	0.2*SDS* Seismic	Roof Live	Floor Live	Snow	Wind	seismic Factor	seismic Rho	Earth
Yes	+1.40D+1.60H	1.400								1.600
Yes	+1.20D+0.50Lr+1.60L+1.60H	1.200		0.500	1.600					1.600
Yes	+1.20D+1.60L+0.50S+1.60H	1.200			1.600	0.500				1.600
Yes	+1.20D+1.60Lr+0.50L+1.60H	1.200		1.600	0.500					1.600
Yes	+1.20D+1.60Lr+0.50W+1.60H	1.200		1.600			0.500			1.600
Yes	+1.20D+0.50L+1.60S+1.60H	1.200			0.500	1.600				1.600
Yes	+1.20D+1.60S+0.50W+1.60H	1.200				1.600	0.500			1.600
Yes	+1.20D+0.50Lr+0.50L+W+1.60H	1.200		0.500	0.500		1.000			1.600
Yes	+1.20D+0.50L+0.50S+W+1.60H	1.200			0.500	0.500	1.000			1.600
Yes	+1.440D+0.50L+0.70S+1.30E+1.60	1.200	0.240		0.500	0.700		1.000	1.300	1.600
Yes	+0.90D+W+0.90H	0.900					1.000			0.900
Yes	+0.660D+1.30E+0.90H	0.900	-0.240					1.000	1.300	0.900

© 2026 ENERCALC, LLC

Calculations - Overall

The **Calculations** tab contains a list of load combinations on the left and a summary of all the calculated values for each load combination on the right. Click one of the load combinations in the list to view results specifically for that load combination.



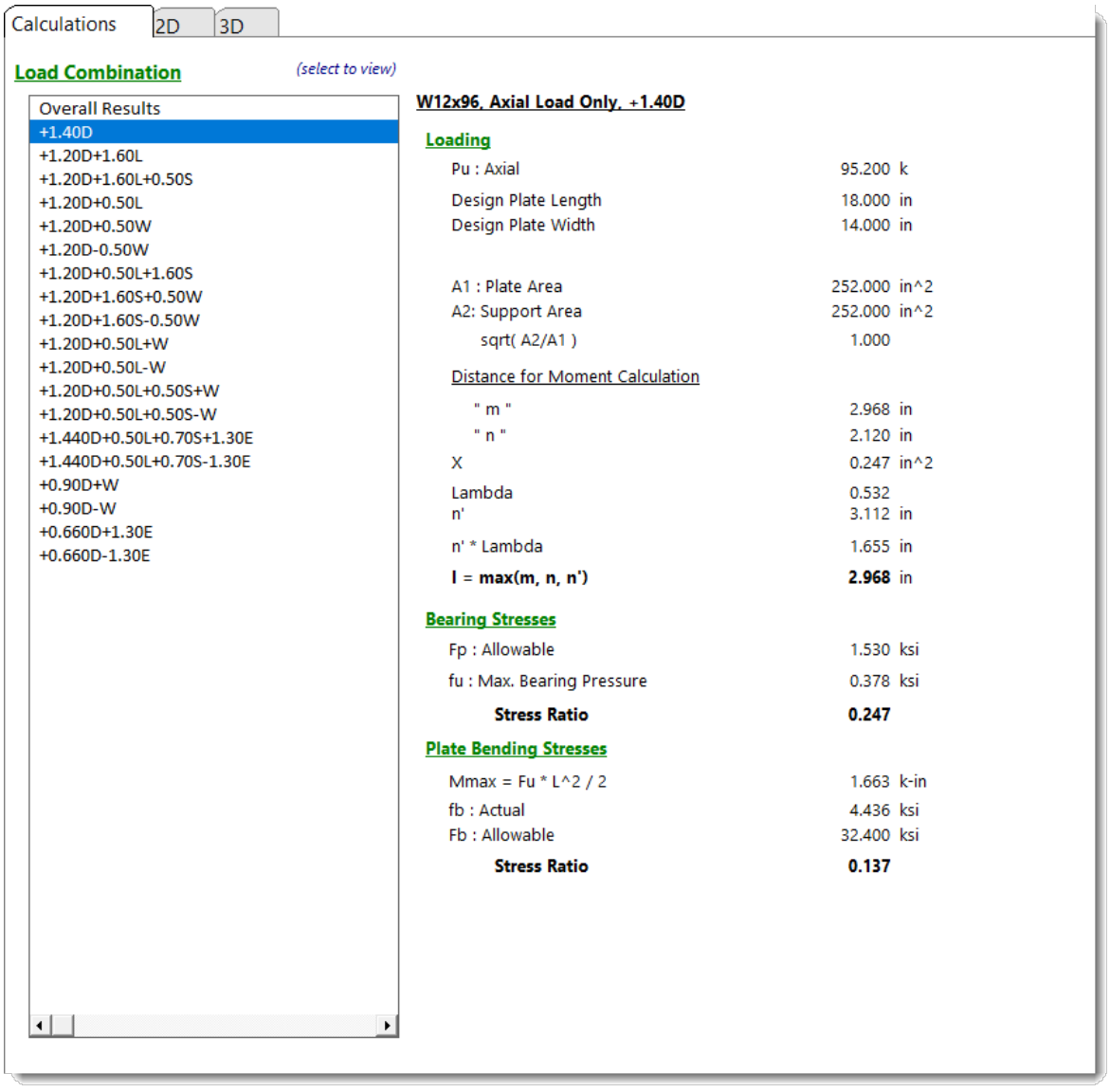
The very first item in the list will always say **Overall Results**. The module will examine the results for all of the load combinations and determine which gives the extreme condition of stress and presents it the **Overall Results** item. This is the governing case for the base plate.

Note that Maximum Bending Stress is calculated using the *plastic* section modulus, in keeping with AISC Design Guide 1, Second Edition.

© 2026 ENERCALC, LLC

Calculations - Axial Load Only

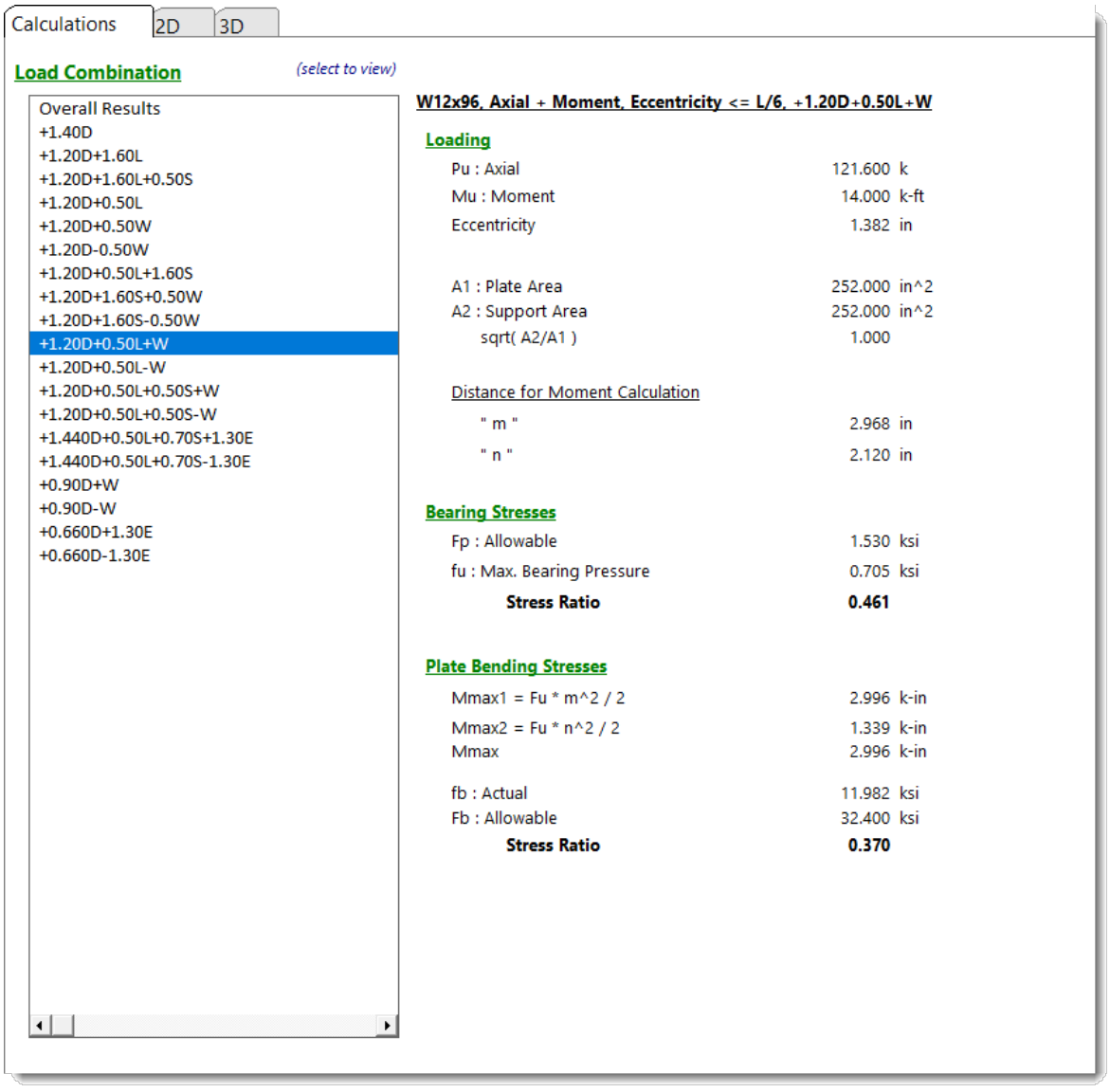
When the selected load combination results in only an axial load being applied, the summary of information below is displayed. See the AISC Design Guide #1 starting on page 4 for a description of the values present here.



© 2026 ENERCALC, LLC

Calculations - Small Eccentricity

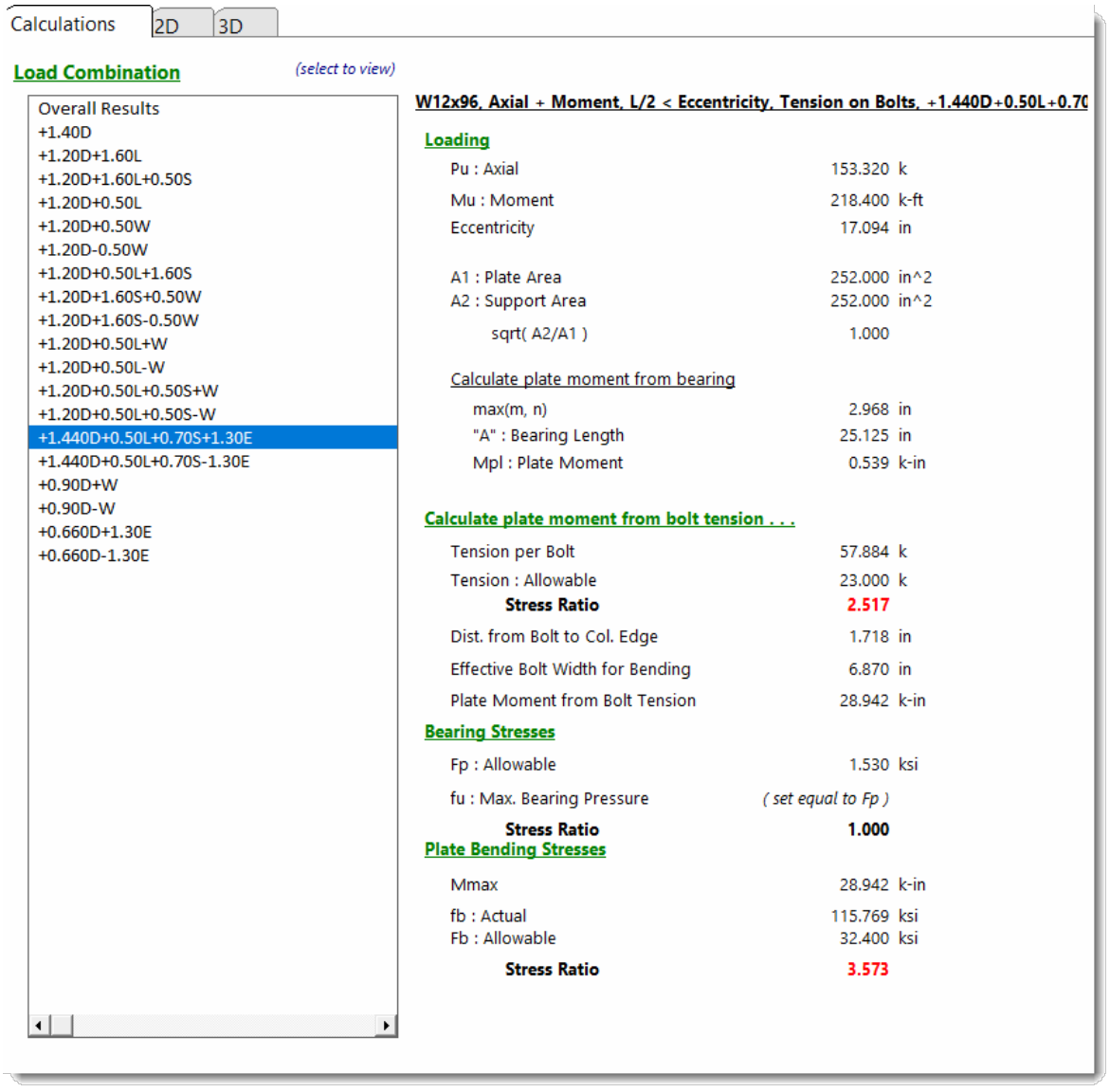
When the selected load combination results in an axial load and very small moment being applied, the summary of information below is displayed. This condition is caused when the moment causes the resultant eccentricity of the axial load to be within the middle 1/3rd of the base plate. See the AISC Design Guide #1 starting on page 19 for a description of the values present here.



© 2026 ENERCALC, LLC

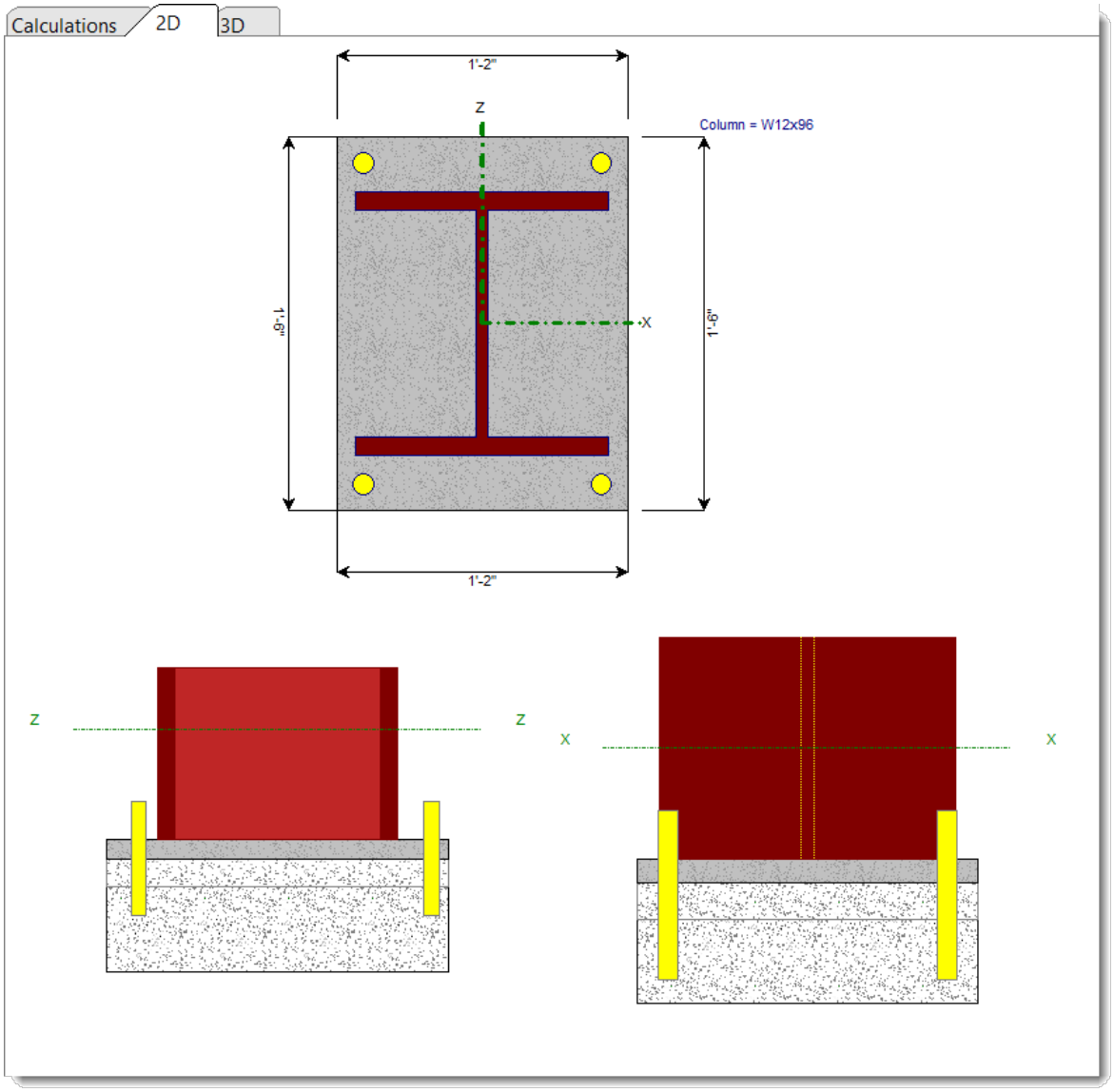
Calculations - Large Eccentricity

When the selected load combination results in an axial load and large moment being applied, the summary of information below is seen. This condition is caused when the moment causes the resultant eccentricity of the axial load to be outside the middle 1/3rd of the base plate. See the AISC Design Guide #1 starting on page 21 for a description of the values present here.



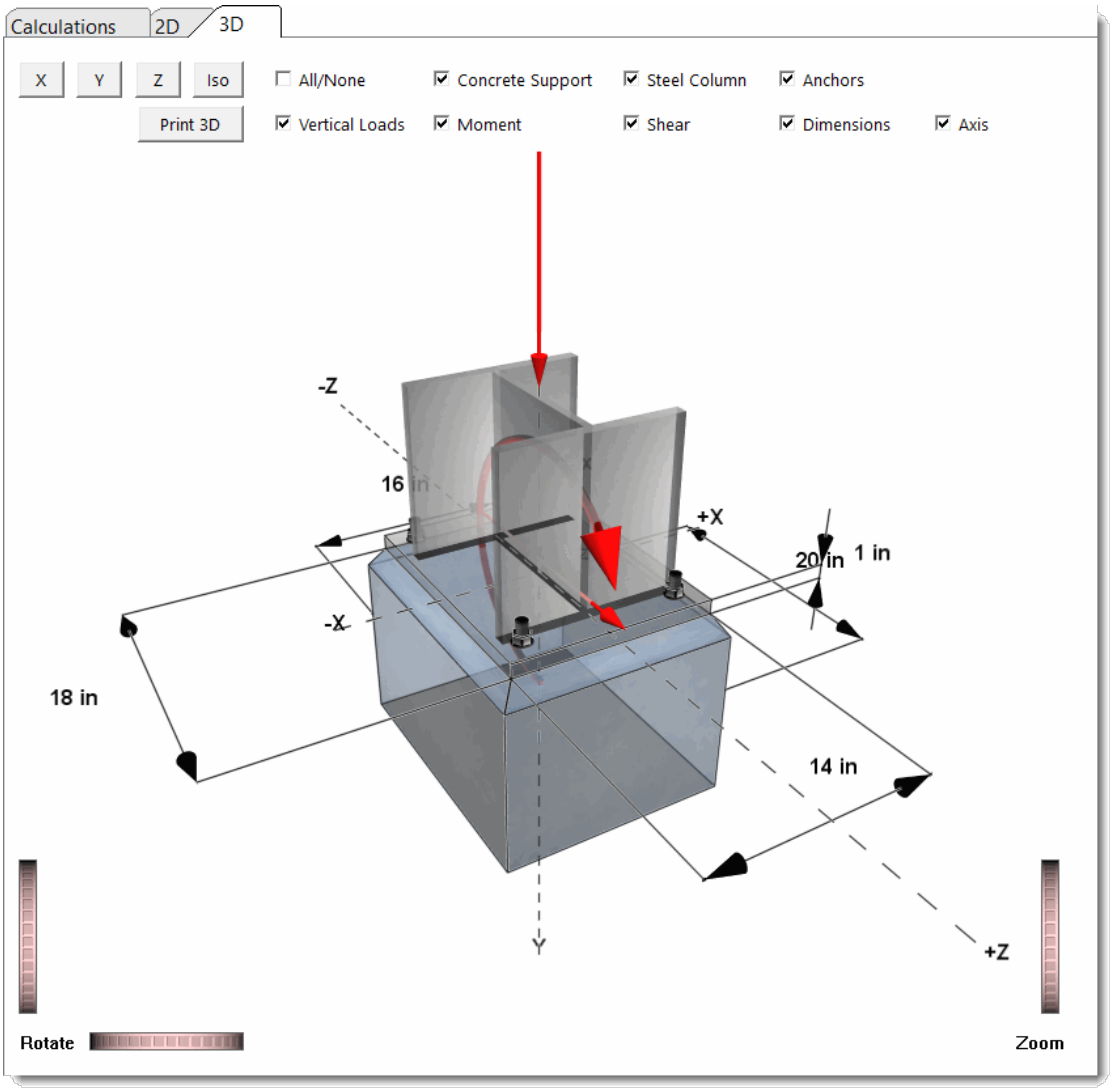
© 2026 ENERCALC, LLC

2D Sketch



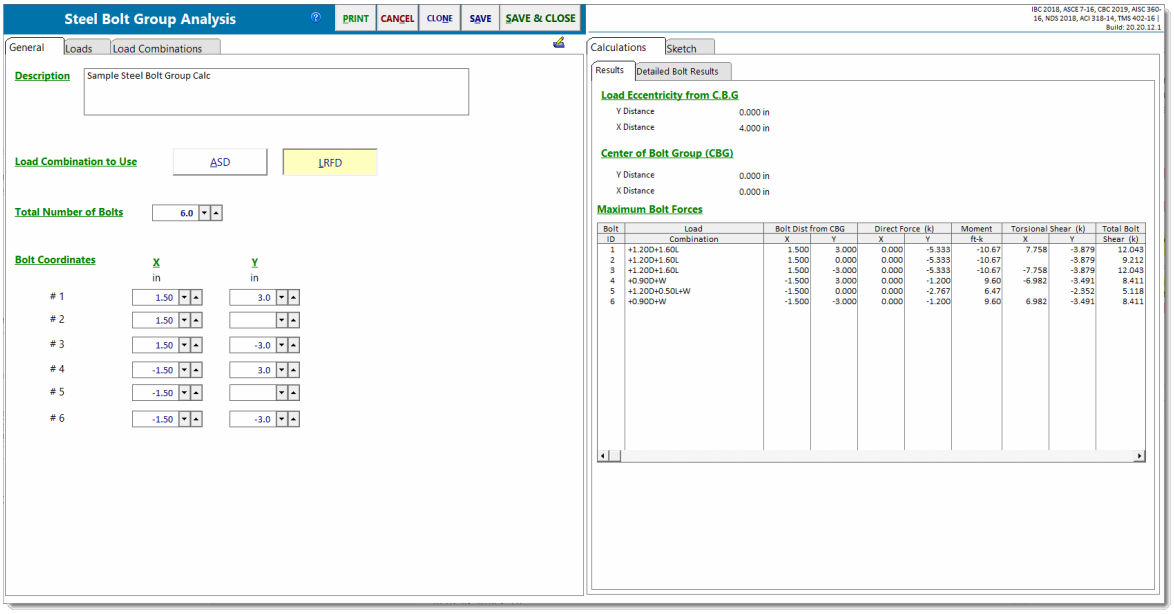
© 2026 ENERCALC, LLC

3D Rendering



13.9.3 Steel Bolt Group Analysis

This module applies the Elastic Method to determine the force distribution from loads applied to a group of up to 16 bolts.



The required input data includes vertical and horizontal load magnitudes and their location with respect to a datum point, along with the coordinates of up to sixteen bolts.

Using these force and bolt coordinates, the module calculates direct shears and torsional shears on each bolt due to its relative location within the group.

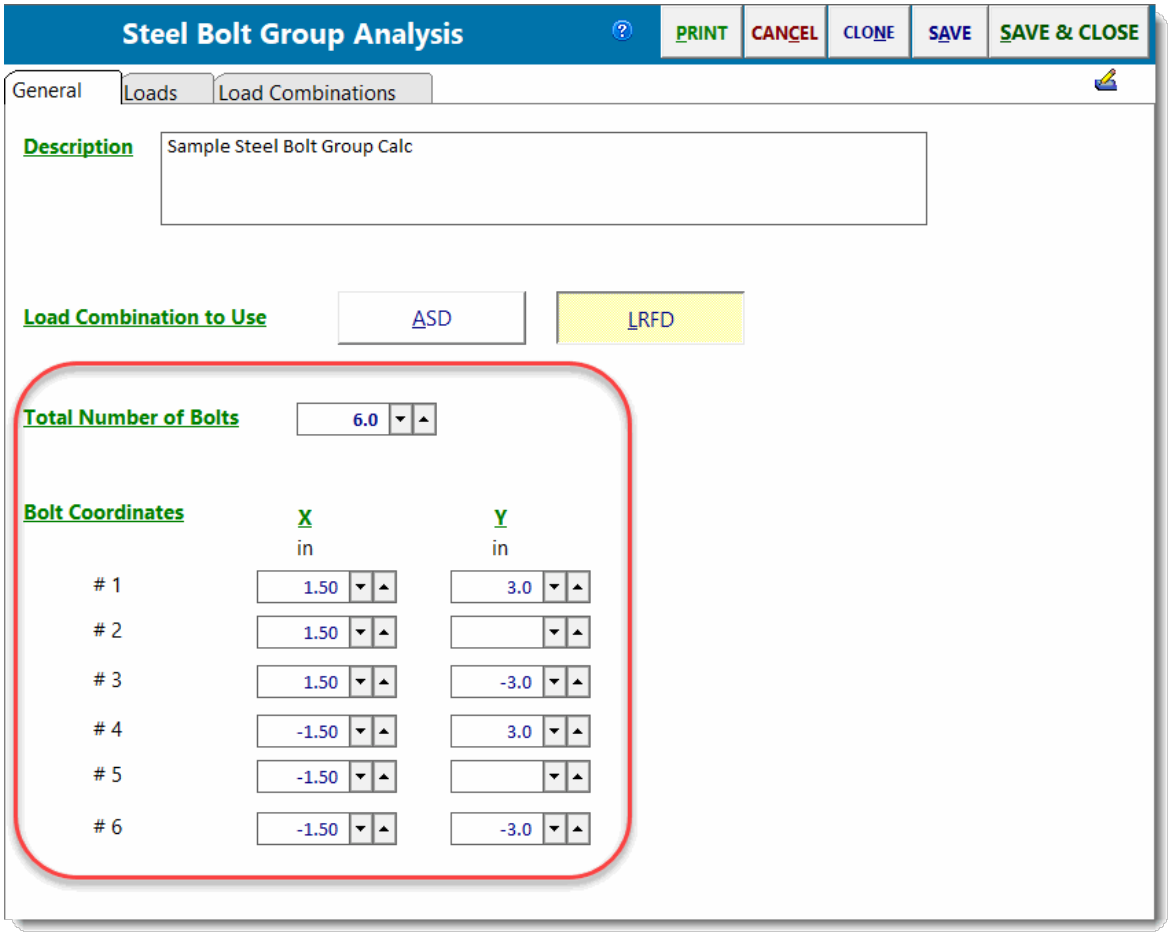
Note:

- At least two bolts should be specified.
- All bolts are assumed to be of the same deformation characteristics when loads are distributed.
- Vertical and Horizontal forces are divided by the number of fasteners to give direct shears.

© 2026 ENERCALC, LLC

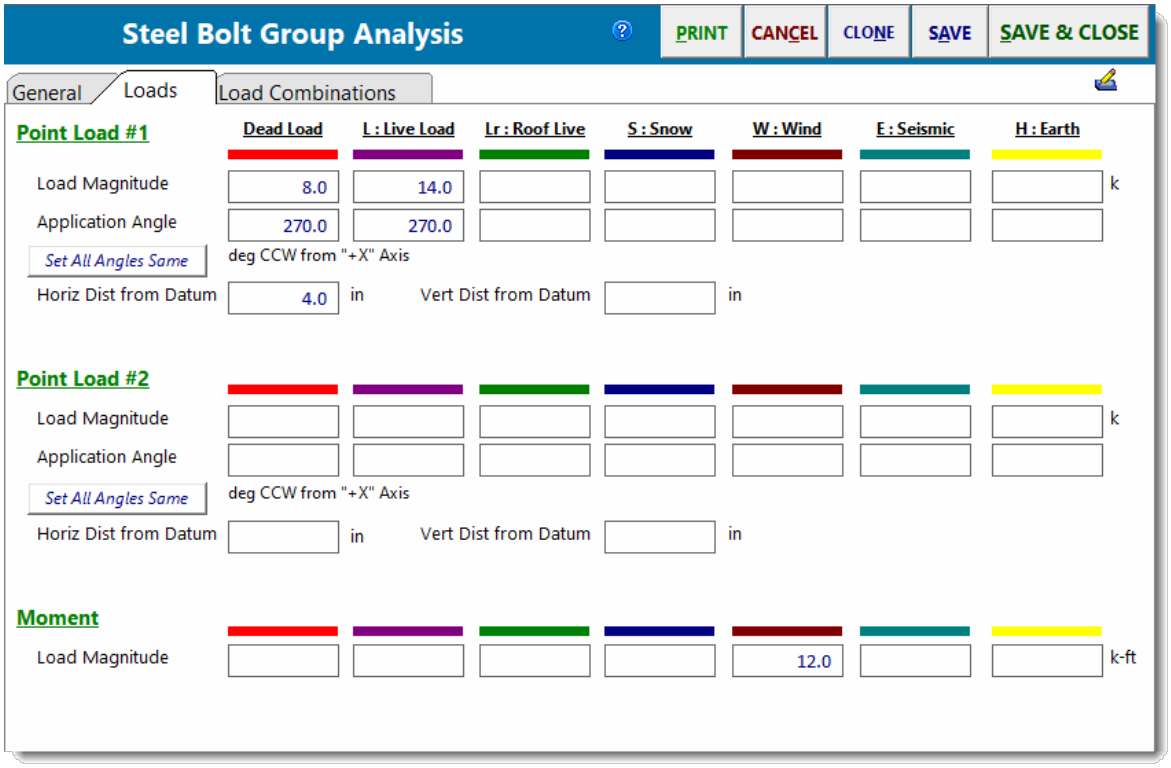
General

Specify the locations up to 16 bolts. Use the Total Number of Bolts entry to indicate bolt quantity. The correct number of data entry locations will be displayed.



Loads

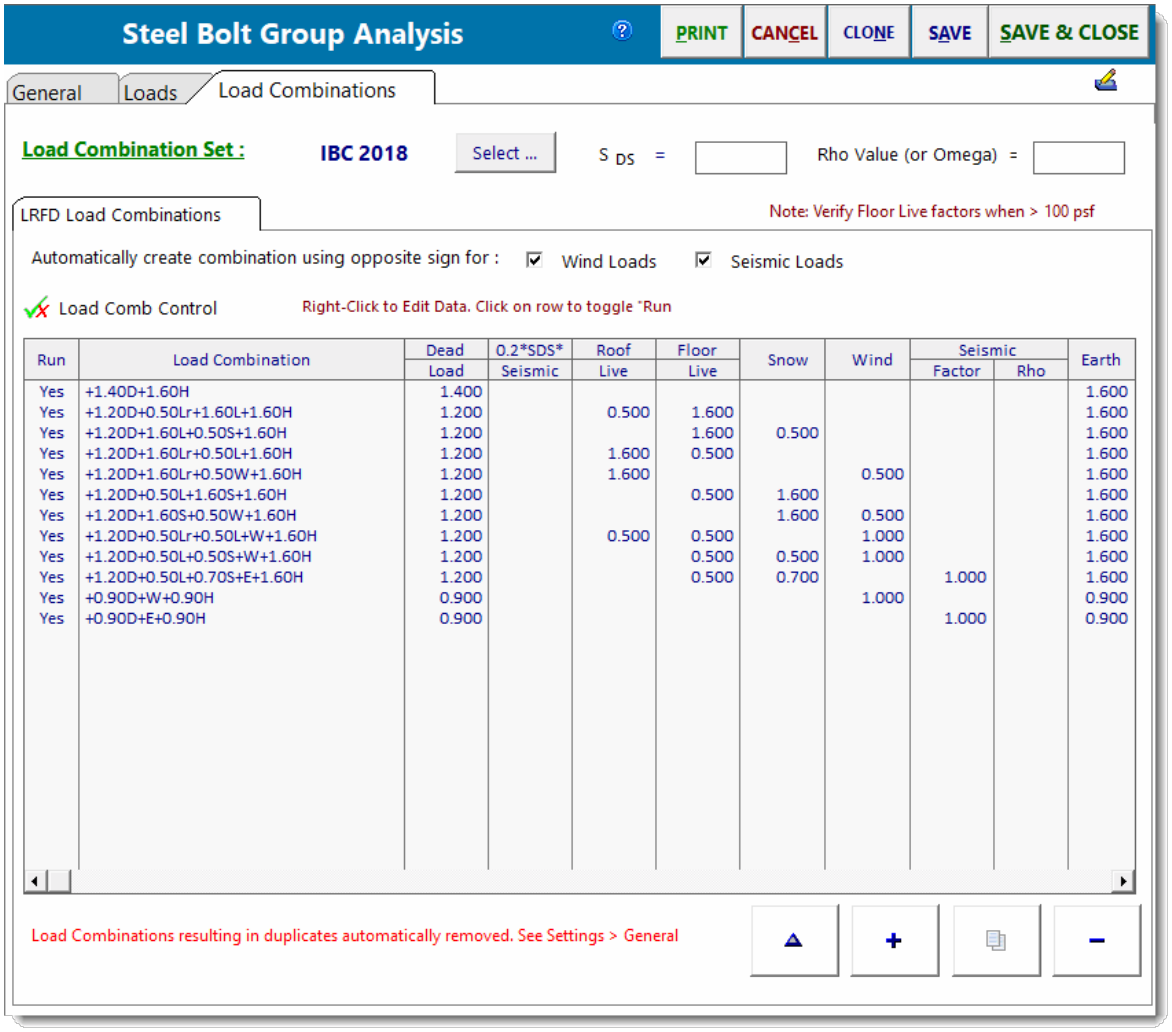
Enter the magnitude of the applied loads, the angle of load application measured CCW from Cartesian zero degrees and the location of the point of load application relative to the Datum point. The module now accepts applied moments as well.



© 2026 ENERCALC, LLC

Load Combinations

Based upon the selection of ASD or LRFD factoring, the appropriate load combinations will be displayed on the load combinations tab.

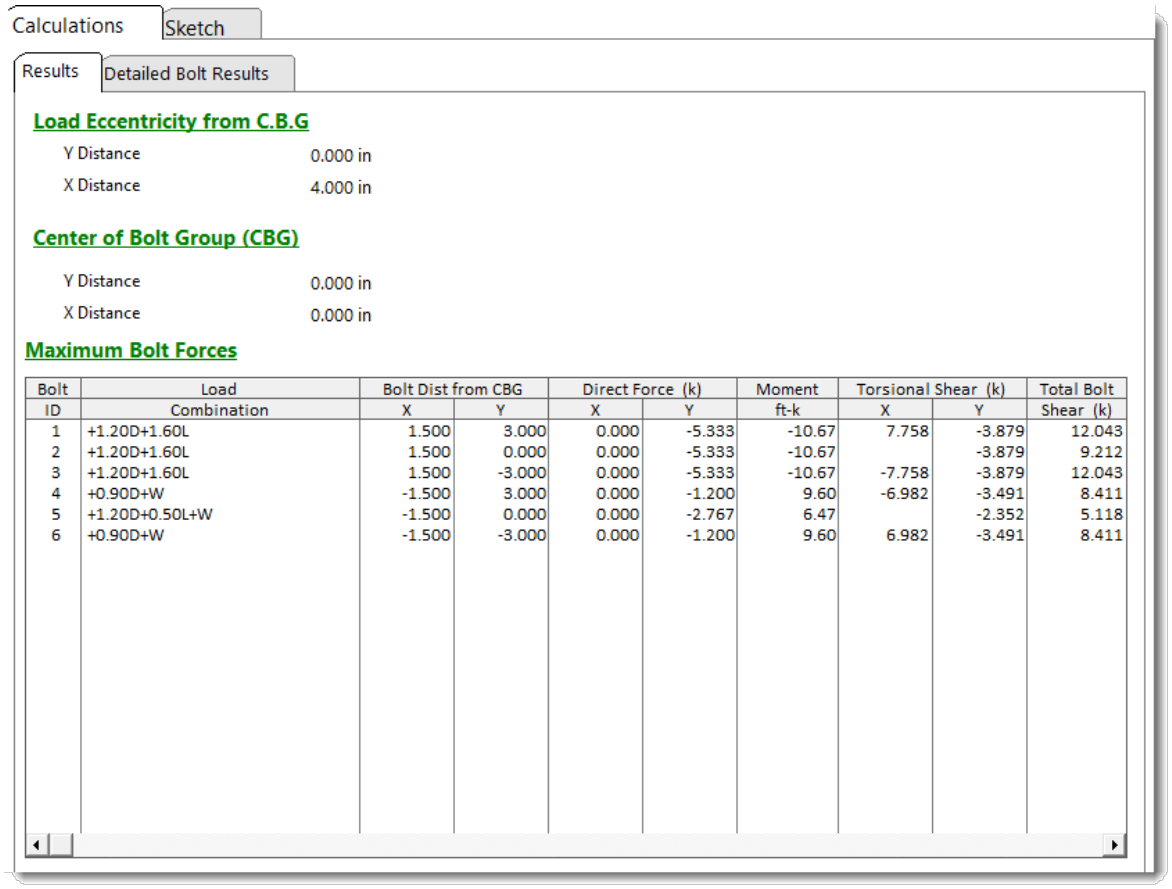


Run	Load Combination	Dead Load	0.2*SDS* Seismic	Roof Live	Floor Live	Snow	Wind	Seismic Factor	Seismic Rho	Earth
Yes	+1.40D+1.60H	1.400								1.600
Yes	+1.20D+0.50Lr+1.60L+1.60H	1.200		0.500	1.600					1.600
Yes	+1.20D+1.60L+0.50S+1.60H	1.200			1.600	0.500				1.600
Yes	+1.20D+1.60Lr+0.50L+1.60H	1.200		1.600	0.500					1.600
Yes	+1.20D+1.60Lr+0.50W+1.60H	1.200		1.600			0.500			1.600
Yes	+1.20D+0.50L+1.60S+1.60H	1.200			0.500	1.600				1.600
Yes	+1.20D+1.60S+0.50W+1.60H	1.200				1.600	0.500			1.600
Yes	+1.20D+0.50Lr+0.50L+W+1.60H	1.200		0.500	0.500		1.000			1.600
Yes	+1.20D+0.50L+0.50S+W+1.60H	1.200			0.500	0.500	1.000			1.600
Yes	+1.20D+0.50L+0.70S+E+1.60H	1.200			0.500	0.700		1.000		1.600
Yes	+0.90D+W+0.90H	0.900					1.000			0.900
Yes	+0.90D+E+0.90H	0.900						1.000		0.900

Results

This tab reports the calculated Center of Bolt Group location, calculated moment applied to the group from the load combination that creates the extreme moment and the eccentricity of the load application from the Center of Bolt Group.

In the table below are the calculated force for each bolt due to direct and torsional shears for each axis (after the applied load is resolved into X & Y components).



Bolt ID	Load Combination	Bolt Dist from CBG X	Bolt Dist from CBG Y	Direct Force (k) X	Direct Force (k) Y	Moment ft-k	Torsional Shear (k) X	Torsional Shear (k) Y	Total Bolt Shear (k)
1	+1.20D+1.60L	1.500	3.000	0.000	-5.333	-10.67	7.758	-3.879	12.043
2	+1.20D+1.60L	1.500	0.000	0.000	-5.333	-10.67		-3.879	9.212
3	+1.20D+1.60L	1.500	-3.000	0.000	-5.333	-10.67	-7.758	-3.879	12.043
4	+0.90D+W	-1.500	3.000	0.000	-1.200	9.60	-6.982	-3.491	8.411
5	+1.20D+0.50L+W	-1.500	0.000	0.000	-2.767	6.47		-2.352	5.118
6	+0.90D+W	-1.500	-3.000	0.000	-1.200	9.60	6.982	-3.491	8.411

© 2026 ENERCALC, LLC

Detailed Bolt Results

This tab provides the detailed calculations for all load combinations and all bolts.

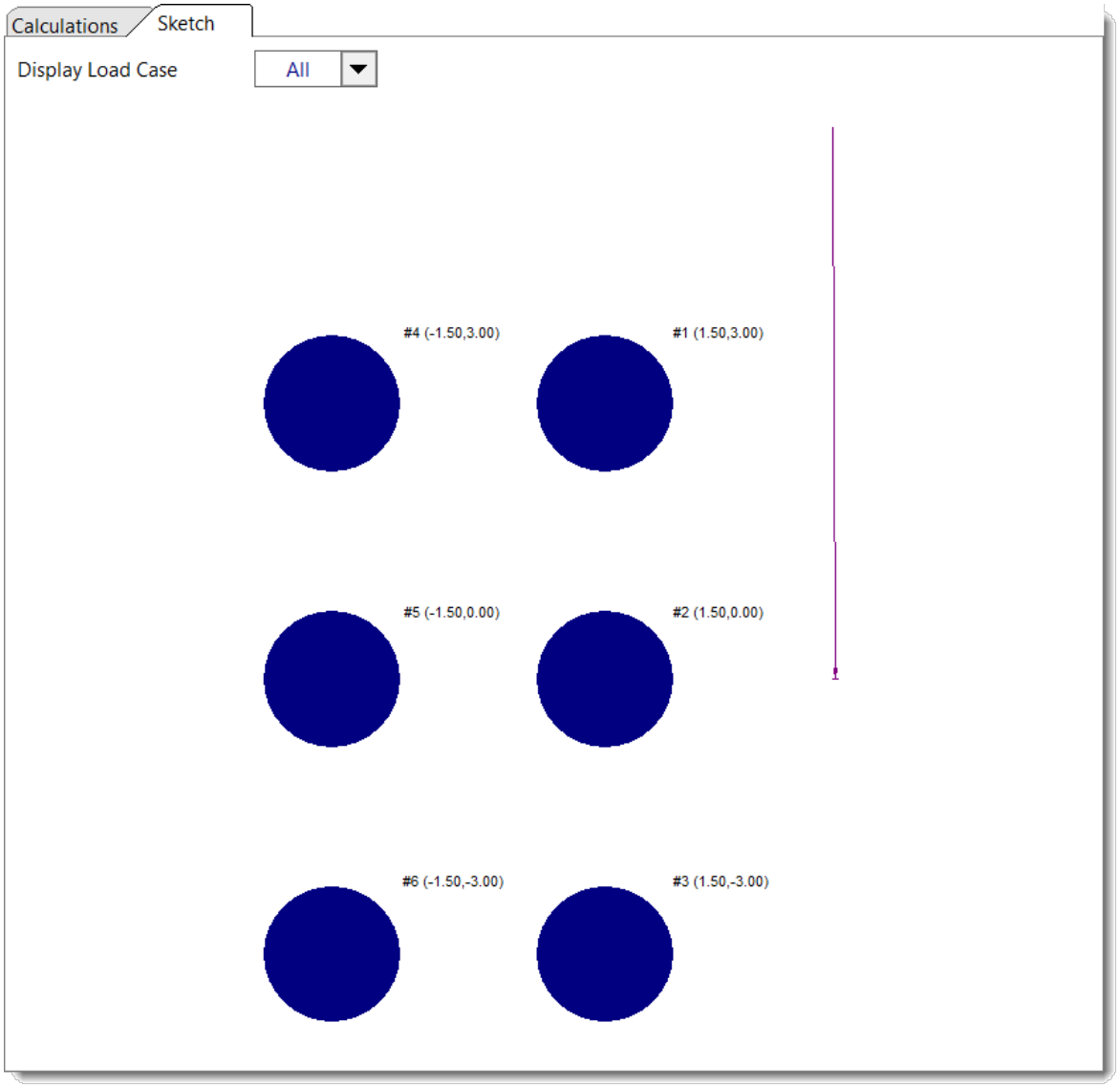
Calculations | Sketch

Results | Detailed Bolt Results

Load Combination	Bolt Dist from CBG		Direct Shear (k)		Torsional Shear (k)		Max. Bolt
Bolt #	X	Y	X	Y	X	Y	Shear (k)
+1.40D							
+1.20D+1.60L							
+1.20D+0.50L							
+1.20D+0.50W							
+1.20D+0.50L+W							
Bolt # 1	1.500	3.000	0.000	-2.767	-4.703	2.352	4.721
Bolt # 2	1.500		0.000	-2.767		2.352	0.415
Bolt # 3	1.500	-3.000	0.000	-2.767	4.703	2.352	4.721
Bolt # 4	-1.500	3.000	0.000	-2.767	-4.703	-2.352	6.951
Bolt # 5	-1.500		0.000	-2.767		-2.352	5.118
Bolt # 6	-1.500	-3.000	0.000	-2.767	4.703	-2.352	6.951
+0.90D+W							
Bolt # 1	1.500	3.000	0.000	-1.200	-6.982	3.491	7.348
Bolt # 2	1.500		0.000	-1.200		3.491	2.291
Bolt # 3	1.500	-3.000	0.000	-1.200	6.982	3.491	7.348
Bolt # 4	-1.500	3.000	0.000	-1.200	-6.982	-3.491	8.411
Bolt # 5	-1.500		0.000	-1.200		-3.491	4.691
Bolt # 6	-1.500	-3.000	0.000	-1.200	6.982	-3.491	8.411
+0.90D							
Bolt # 1	1.500	3.000	0.000	-1.200	1.745	-0.873	2.710
Bolt # 2	1.500		0.000	-1.200		-0.873	2.073
Bolt # 3	1.500	-3.000	0.000	-1.200	-1.745	-0.873	2.710
Bolt # 4	-1.500	3.000	0.000	-1.200	1.745	0.873	1.776
Bolt # 5	-1.500		0.000	-1.200		0.873	0.327
Bolt # 6	-1.500	-3.000	0.000	-1.200	-1.745	0.873	1.776

© 2026 ENERCALC, LLC

Sketch

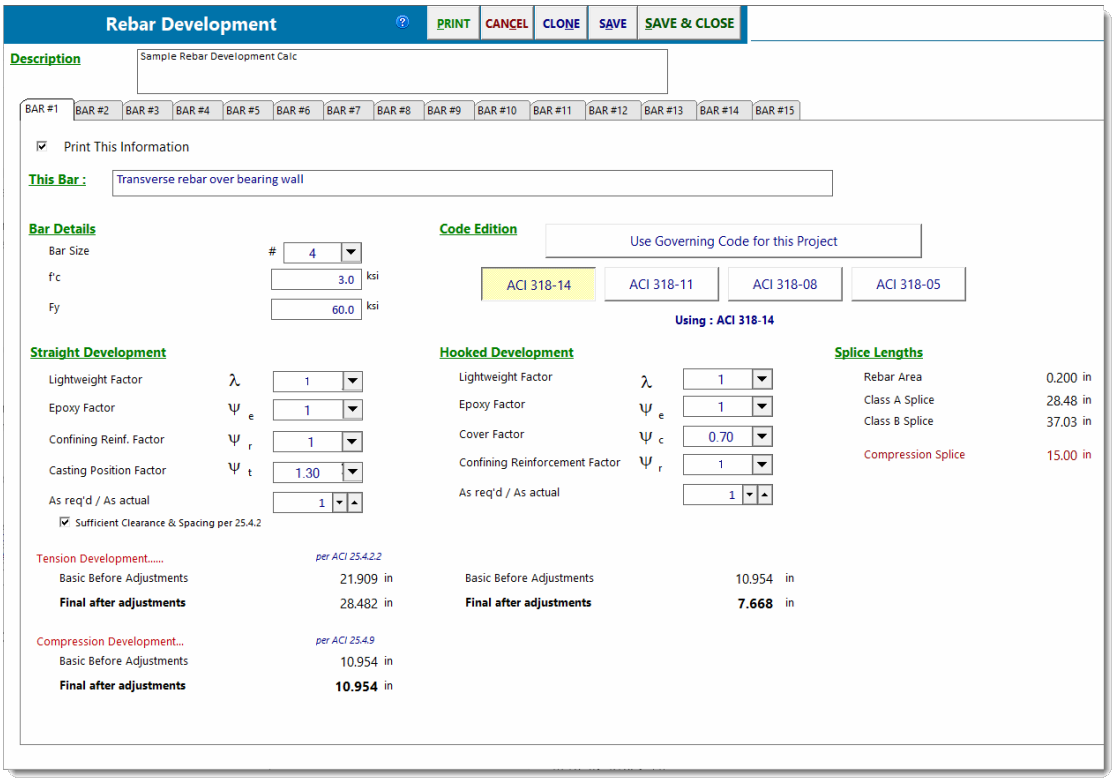


© 2026 ENERCALC, LLC

13.9.4 Rebar Development Table

This module calculates development and splice lengths for deformed steel rebar in concrete. Since typical development lengths are readily available in tables, this module is primarily for developing tables of development lengths when unusual conditions affect the lengths.

The module uses the relevant sections of ACI 318 to determine straight tension & compression development and hooked tension development lengths.



Print This Information

Check this box if you want the development calculation on the current tab to appear on the printout.

This Bar

Enter a description for the current development calc.

Bar Details

Enter the basic stress info for this bar.

Straight Development

This section calculates tension & compression bar development according to ACI. Please refer to the relevant ACI code for details.

© 2026 ENERCALC, LLC

Hooked Development

This section calculates tension hooked rebar development according to ACI. Please refer to the relevant ACI code for details.

13.10 External Items

ENERCALC SEL allows you to add non-ENERCALC items into a locally saved Project File. Until 2023, the types of items that could be added were limited to:

- MS Excel Spreadsheets
- MS Word Documents
- Adobe Acrobat PDF Files
- Scanned Images

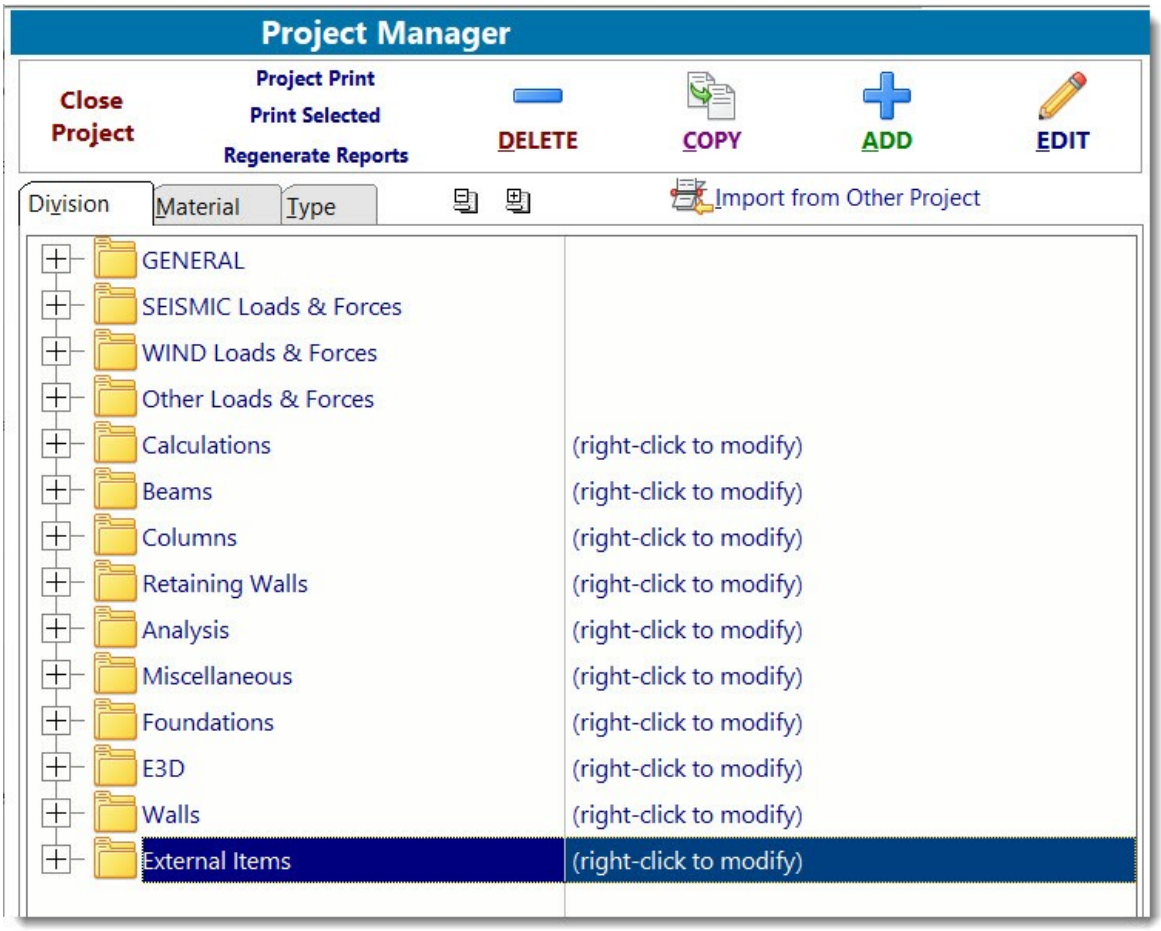
However, in January 2023, this functionality received a huge enhancement. By using the *native application* to display the contents of an External Item file, ENERCALC SEL can now support an almost limitless variety of file formats as External Items.

For example, you can now think in terms of having a RAM or RISA model as an External Item. Have some JPEG photos of as-built conditions and want to preserve them as part of an ENERCALC project file? No problem. Still prefer to use that MathCad sheet for the design of certain elements? Go for it, and then link it or embed it.

The ability to include External Items allows you to use the ENERCALC Project File as a central point of document preparation and storage for structural project calculations.

© 2026 ENERCALC, LLC

External Items can be interspersed throughout the list of calculations in the Project Manager, or they can be placed in Divisions that have been created specifically to contain the external items.



© 2026 ENERCALC, LLC

13.10.1 Relating an External Item to a Project File

Relating External Items to an ENERCALC Project File

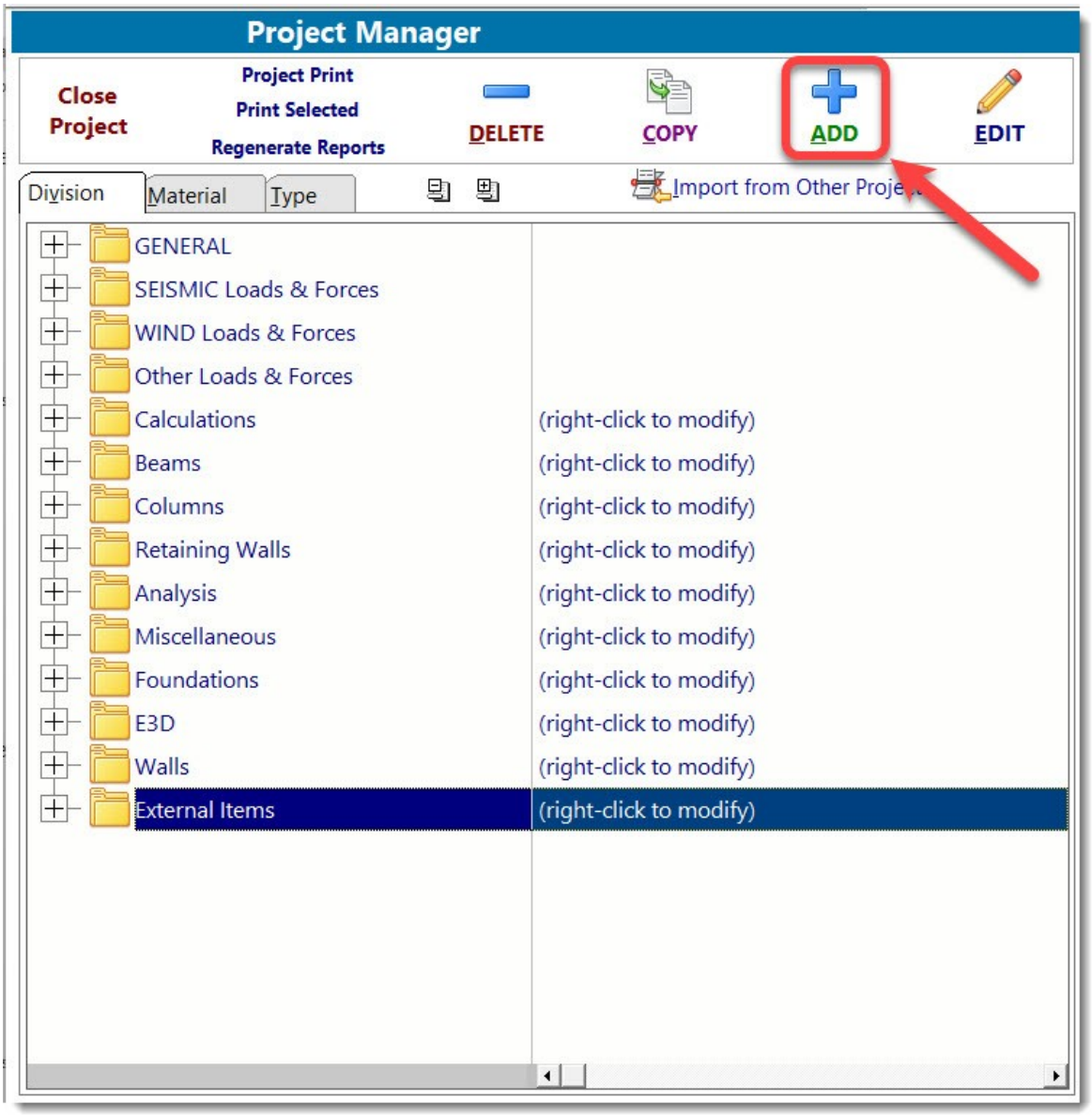
When relating External Items to an ENERCALC Project File, there are two options for the way the External Items are saved:

- Linked: Saves a *link* to the External Item in Project File. (Keeps Project File size small, but it's not as easily portable.)

- Embedded: Saves a *copy* of the External Item in what's called a "blob file" that actually becomes a part of the Project File. (Easiest for portability, but results in larger Project File size.)

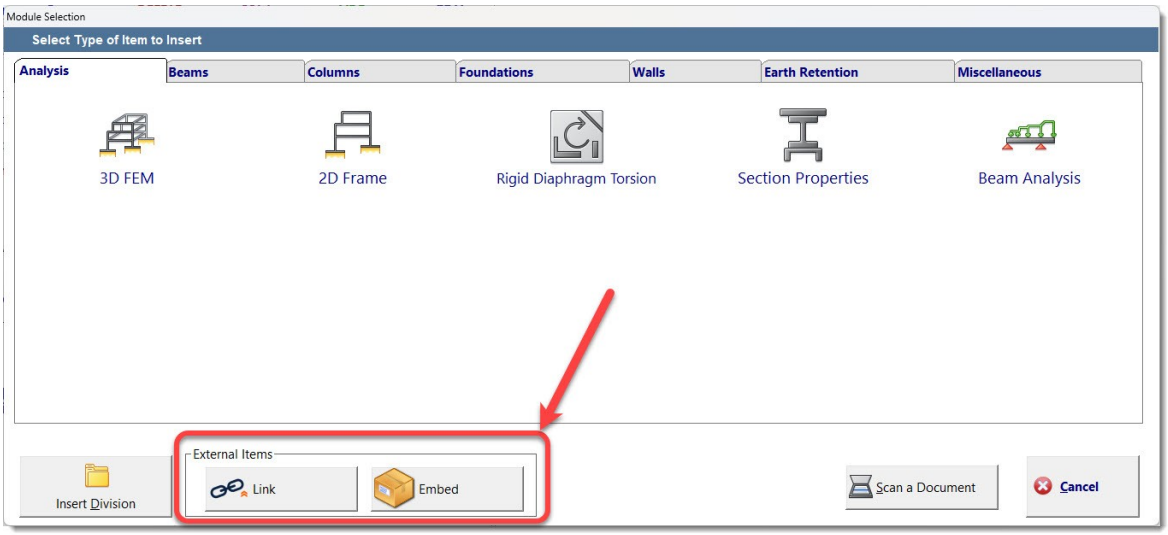
The method is the same, regardless of which option is desired. Click on the desired location of insertion in the Project Manager, and then click the Add button:

© 2026 ENERCALC, LLC

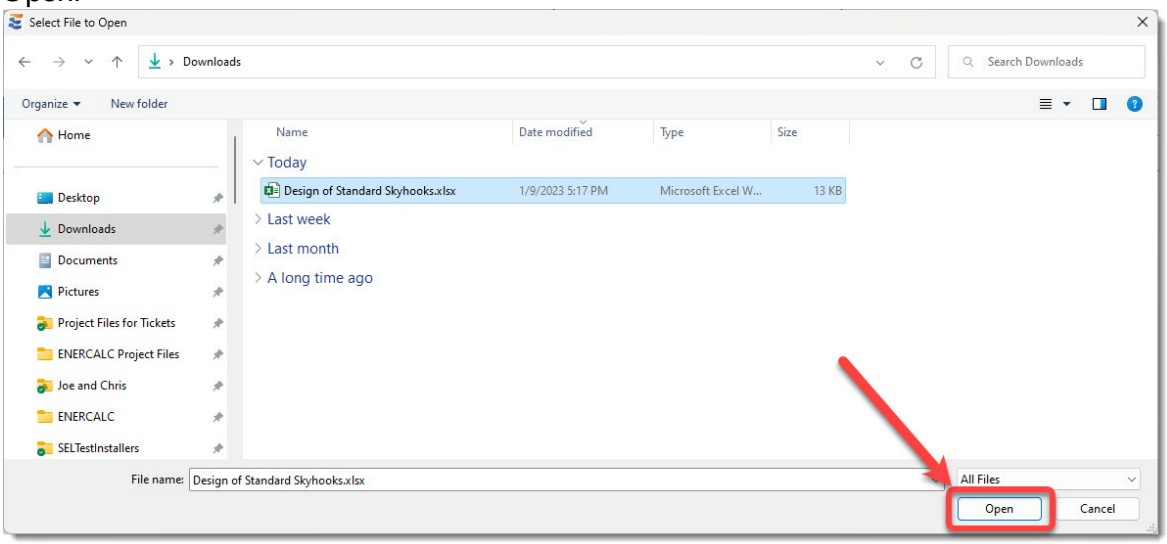


In the External Items category of the Module Selection screen, click Link or Embed:

© 2026 ENERCALC, LLC

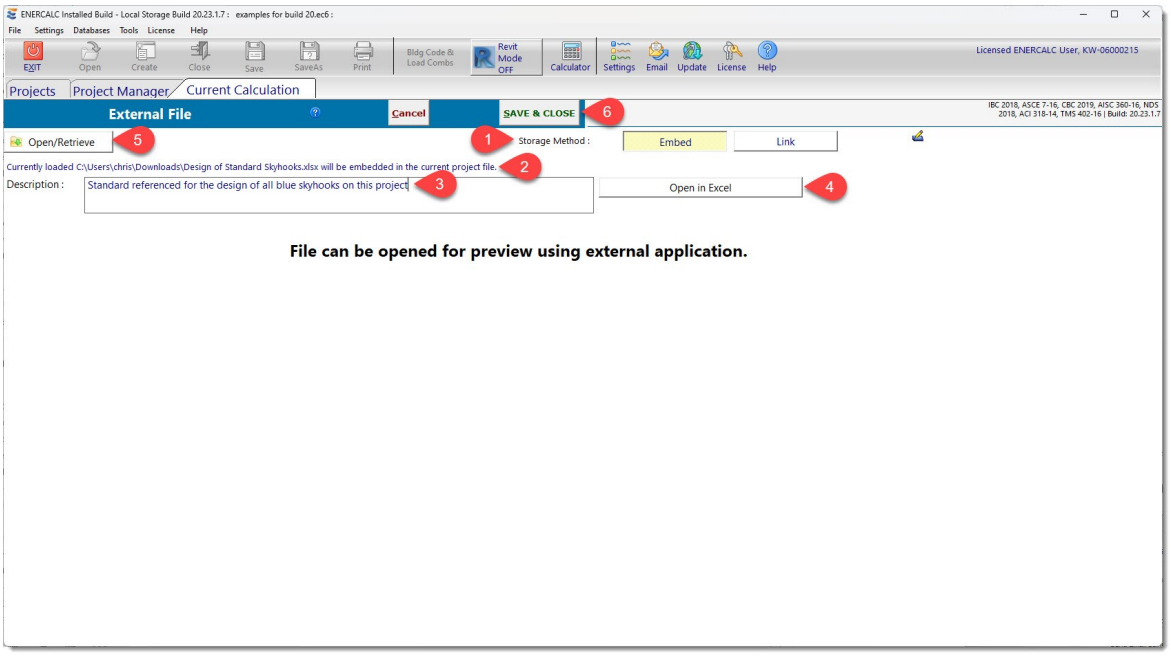


In the Select File to Open screen, browse for the desired file, click on it, and then click Open:



The ENERCALC SEL screen will appear as shown below, and note that the ***view/preview/display of the External Item no longer occurs within the ENERCALC window***. This is to be expected. The numbered features are described below the image.

© 2026 ENERCALC, LLC



1: Storage Method: Indicates the current storage method, and allows it to be modified. If an Embedded item is changed to Linked, the program will report the name and location of the resulting file.

2: External Item Location: For Embedded items, this indicates the original name and location of the item, and verifies that it is embedded in the Project File. For Linked items, this indicated the file name and location of the item

3: Description: The user can enter a description to more easily identify the item. The Description will be shown in the Project Manager and will be prepended by either "Linked" or "Embedded" for reference.

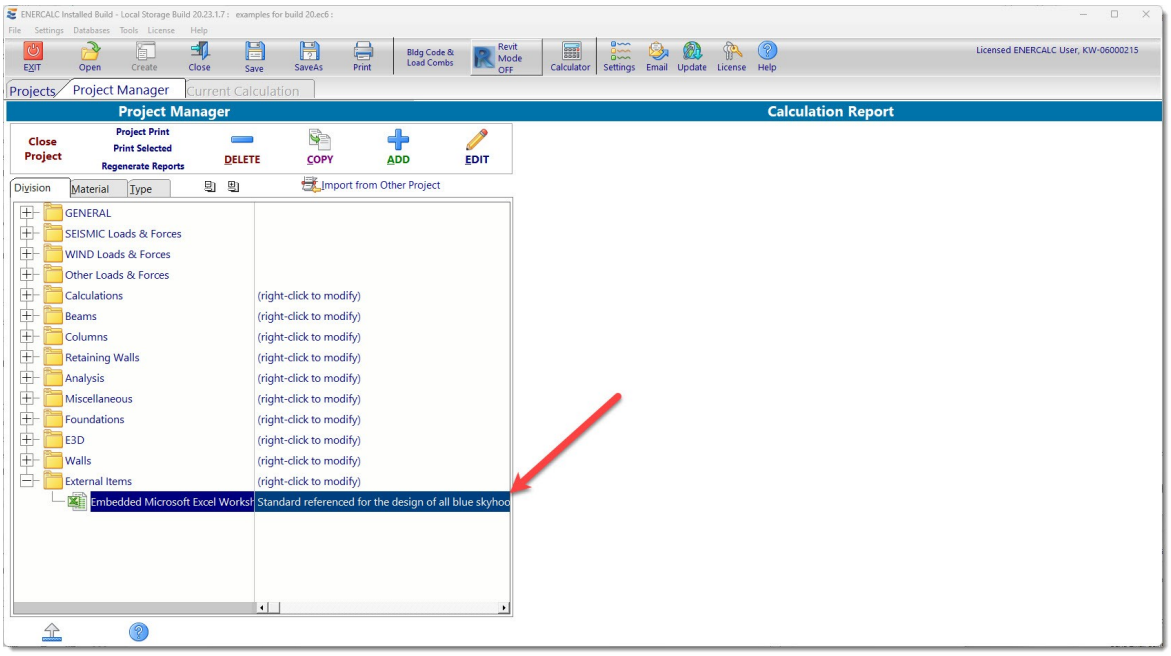
4: Open in [native application]: Button allows the External Item to be opened in its native application.

5: Open/Retrieve: Allows a different file to be selected for the current External Item.

6: Save & Close: Saves changes to the Description or Storage Method. (Any changes to the External Item can be saved when the item is open in its native application.

After Save & Close is clicked, the Project Manager will display the item like this:

© 2026 ENERCALC, LLC



13.10.2 Limitations of Working with External Items

As of January 2023, External Items are not incorporated into a Project Print when using the Project Printing Manager.

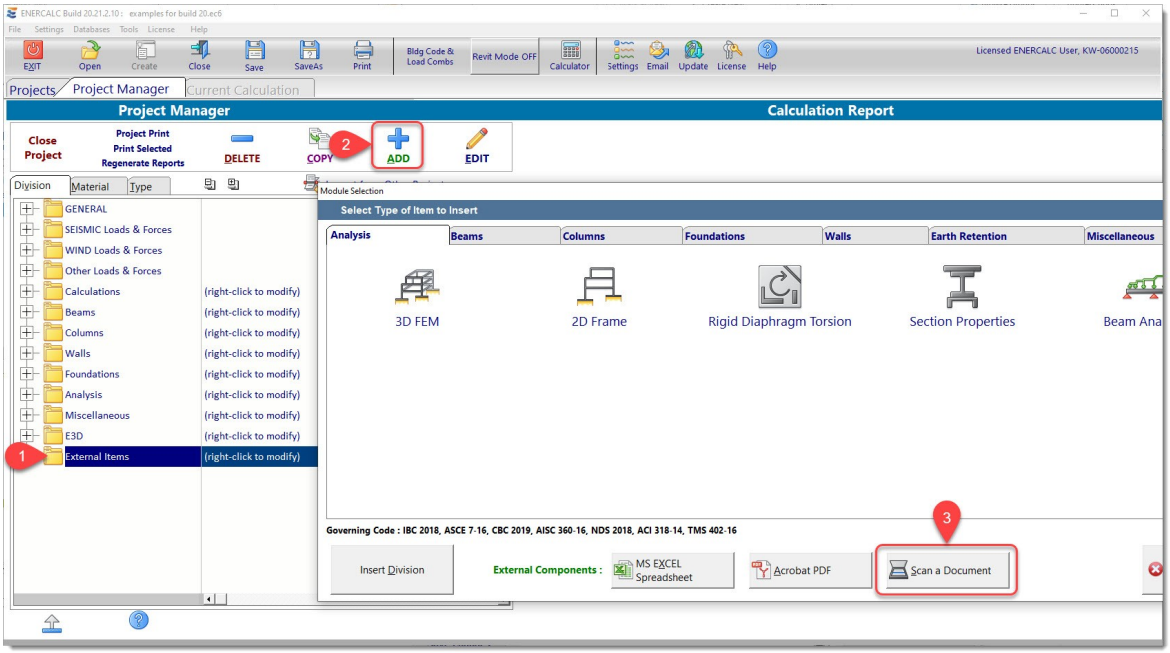
The following table summarizes the capabilities of external items within the various formats of ENERCALC SEL:

	Embedded External Items can be **Used**?	Linked External Items can be **Used**?	Embedded External Items can be **Viewed**?	Linked External Items can be **Viewed**?
Local Storage option within the Installed Version	**Yes**	**Yes**	**Yes**	**Yes**
Hosted Storage option within the Installed Version	**No**	**No**	**Yes**	**No**
Cloud Version	**No**	**No**	**No**	**No**

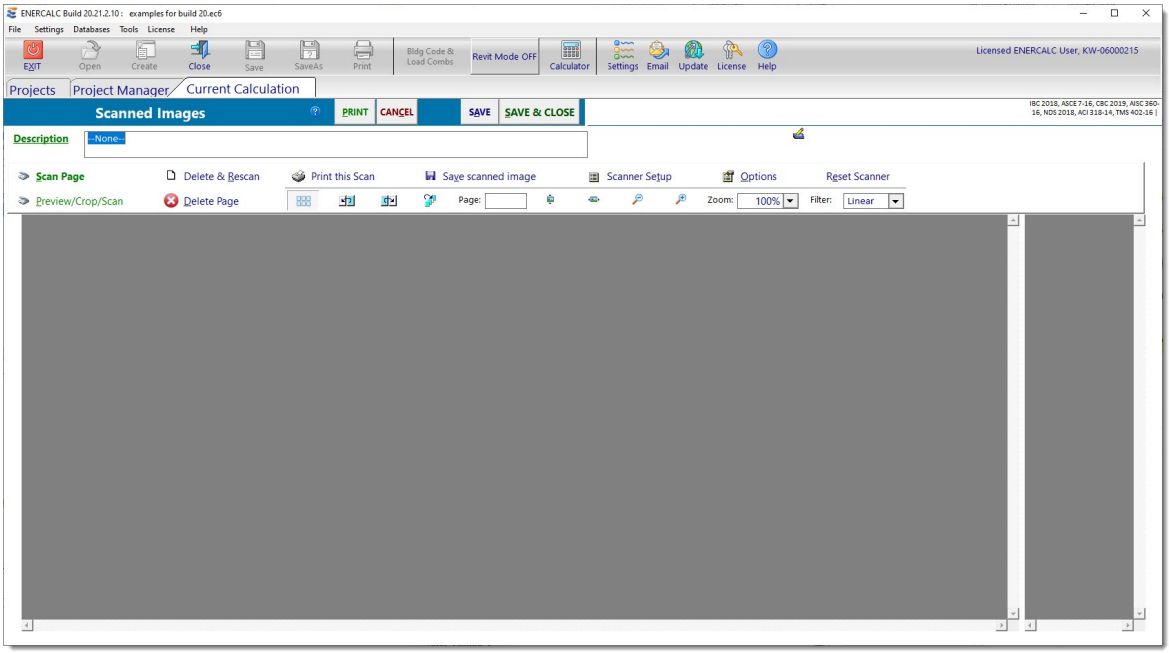
© 2026 ENERCALC, LLC

13.10.3 Scanned Document

ENERCALC SEL now offers you the ability to use your scanner to create items and insert them into your Project File. To scan an item and add it to your Project File, click on the Division where you would like to insert the scanned item, click the [**Add**] button, and then click [**Scan a Document**]:



When the module is loaded you will see the screen below:



© 2026 ENERCALC, LLC

Here is a description of the various controls in this module:

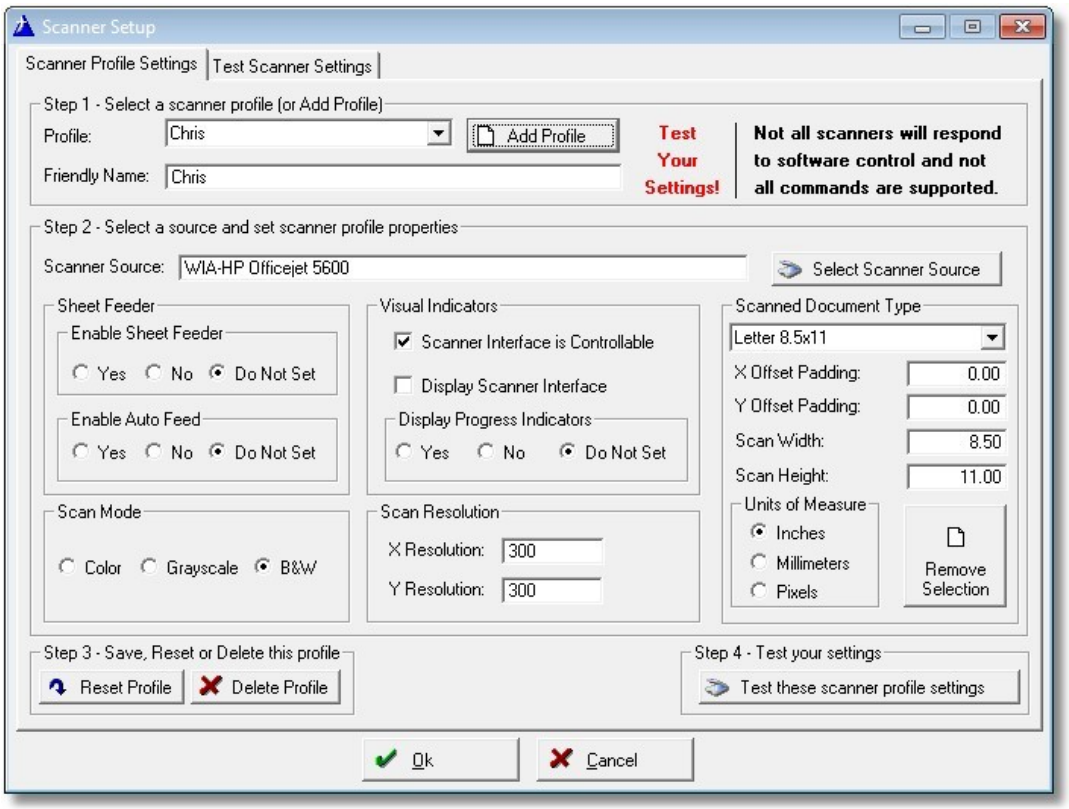
Scan Page: Using all the settings on the "Scanner Setup" page, perform a high resolution scan.

Delete & Rescan: Delete the currently visible scan and rescan the image at a different resolution.

Print this Scan: Print the currently visible image to a printer.

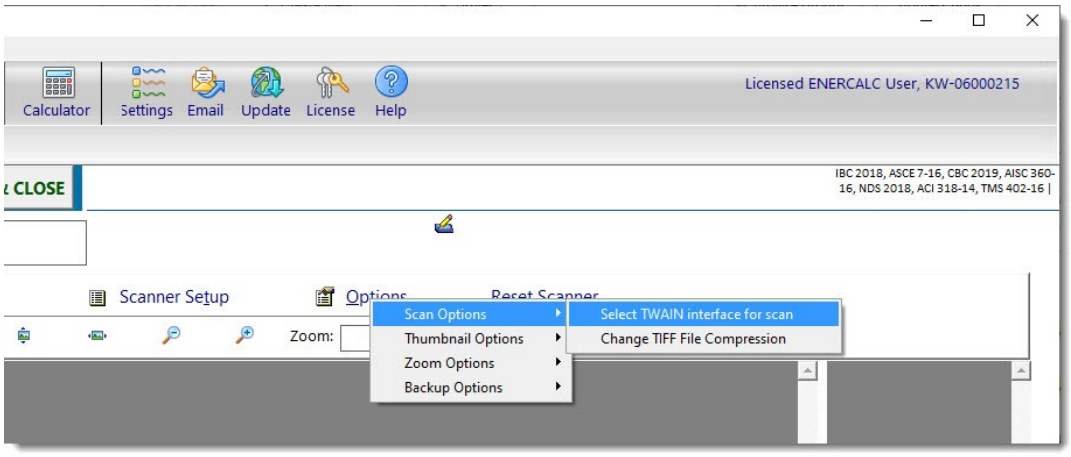
Save scanned image: Save the current scan to a file.

Scanner Setup: Access the scanner setup screen (see image below)



© 2026 ENERCALC, LLC

Options:



Reset Scanner: Send signals to the scanner to reset itself to its default state.

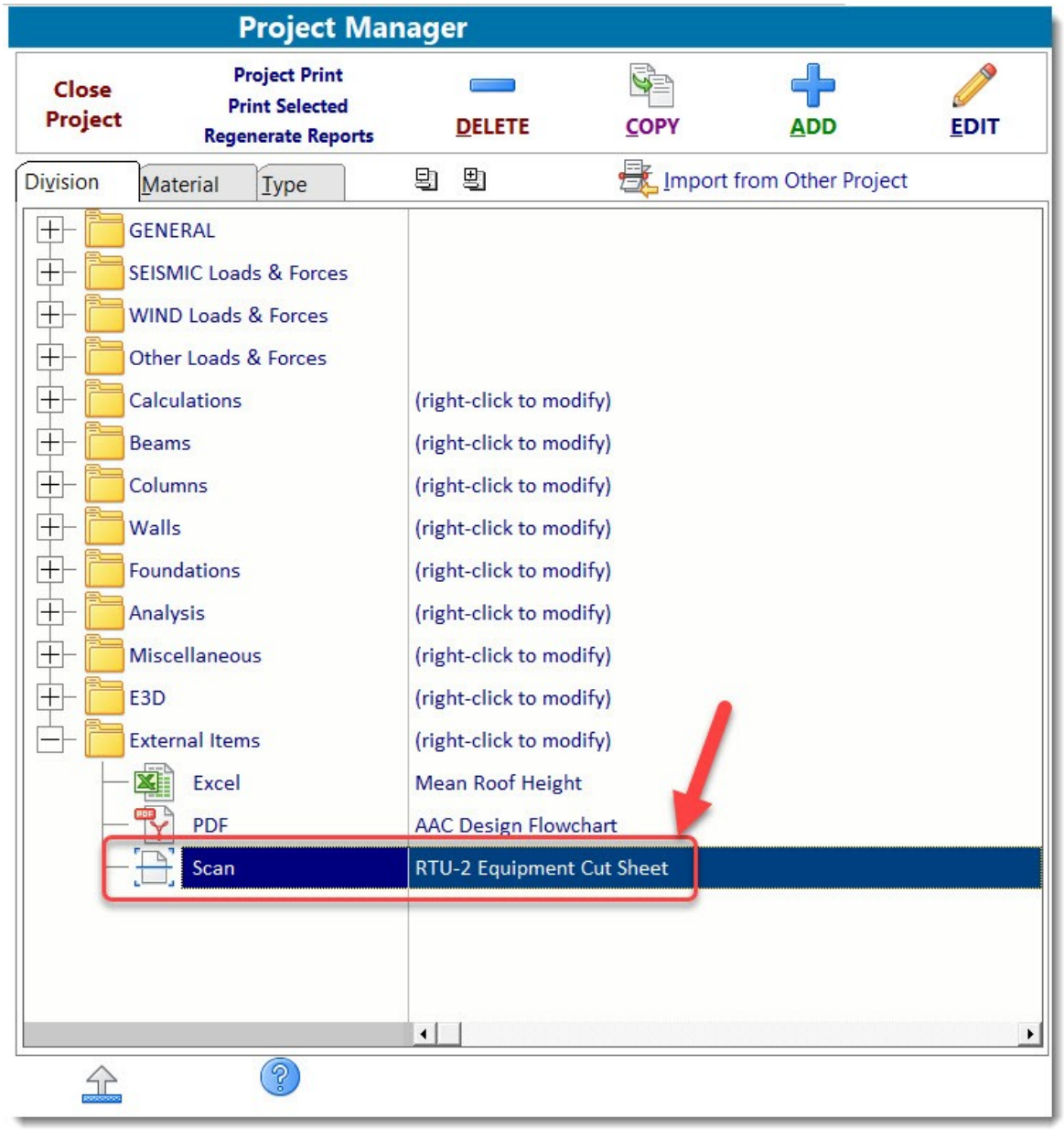
Preview/Crop/Scan: Perform a fast, low resolution "preview" scan and display it for your use in defining the final region that will be used for the high resolution scan.

Delete Page: Remove the current scan from this viewer.

Image Manipulation buttons: Allow you to change the way you view the scans in this document.

© 2026 ENERCALC, LLC

To create a scanned item, place the item on the scanner and click the [**Scan Page**] button. If desired, enter text in the Description field. Then, click the [**Save & Close**] button. The display will return to the Project Manager, and you will see an indication that the scanned item has been attached as shown in the image below:



© 2026 ENERCALC, LLC

Part XIV

14 Appendices

Appendix A 1240 - Table of Horizontal Temperature and Shrinkage Reinforcing
Appendix B 1241 - Development and Lap Lengths
Appendix C 1242 - Masonry Wall Weights & Section Properties
Appendix D 1247 - Summary of Concrete & Masonry Design Formulas
Appendix E 1248 - References Used For The Development Of This Program
Appendix F 1248 - Rankine and Coulomb Formulas

14.1 Appendix A - Table of Horizontal Temperature and Shrinkage Reinforcing

Typical Horizontal Rebar Spacing for .0007 A_g Masonry and .002 A_g for concrete						
Mat'l	**Thick**	**#3**	**#4**	**#5**	**#6**	**#7**
Concrete	6	9	17	18	18	¾
Concrete	7	8	14	18	18	¾
Concrete	8	7	12	18	18	¾
Concrete	9	6	11	17	18	¾
Concrete	10	5.5	10	15	18	¾
Concrete*	12	9	17	18	18	¾
Concrete*	14	8	14	18	18	¾
Concrete*	16	7	12	18	18	¾
CMU	6	24	48	48	48	¾
CMU	8	16	32	48	48	¾
CMU	10	16	24	32	48	¾
CMU	12	12	24	32	48	¾
CMU	16	8	16	24	40	48

* ACI 318-14 Section 11.7.2.3 requires two layers in walls over 10" thick, but "basement walls" are exempted, which presumably applies to retaining walls also. However, the above spacings assume that the specified rebar will be placed on each face.

© 2026 ENERCALC, LLC

14.2 Appendix B - Development and Lap Lengths

Lap Splice Lengths(1) and Hooked Bar Embedments (inches)

Bar Size		Masonry (2) f'_m=1500 psi		Concrete (3)		
		Grade 40	Grade 60	2000 psi	3000 psi	4000 psi
#4	L	20	24	20.9	17.1	14.8
	H(4)			9.4	7.7	6.7
#5	L	25	30	26.2	21.4	18.5
	H(4)			11.8	9.6	8.3
#6	L	30	36	31.4	25.6	22.2
	H(4)			14.1	11.5	10.0
#7	L	35	42	45.8	37.4	32.4
	H(4)			16.5	13.4	11.6
#8	L	40	48	52.3	42.7	37.0
	H(4)			18.8	15.4	13.3

(1) Min. lap for spliced bars, in., assumes f_y = 60 ksi, per ACI 318-14, Equation (25.4.2.3a).

(2) 40 bar diameters for f_y = 40 ksi and 48 diameters for f_y = 60 ksi

(3) Min. lap is development length x 1.3, assuming Class B splice. Cannot be reduced for stress level.

(4) Assumes standard hook and not reduced by ratio A_S (required) / A_S (provided).

Note that IBC 2018 Section 2107.2, offers an alternative to TMS 402-16, Section 2.1.7.7.1.1. If the alternative is used, it replaces the following development length equation in TMS 402:

$$\ell_d = \frac{0.13\, d_b^2\, f_y\, \gamma}{K\sqrt{f'_m}}$$

g = 1.0 for #3, #4, #5 bars, 1.4 for #6, #7, and 1.5 for #8
K = Smallest of (minimum masonry cover, the clear spacing between adjacent reinforcement splices, and 9 d_b

This requirement resulted in much longer lap lengths and has met with considerable objection.

© 2026 ENERCALC, LLC

14.3 Appendix C - Masonry Wall Weights & Section Properties

References:

- ASTM C90-06 - Standard Specification for Loadbearing Concrete Masonry Units
- NCMA TEK 14-01B - Section Properties of Concrete Masonry Walls
- NCMA TEK 14-13B - Concrete Masonry Wall Weights

Weights of Masonry Walls (pcf) - Using 140 pcf Grout

		Grouted Cell Spacing								
Block Type	**CMU Wall Thickness**	**8"oc (solid)**	**16"oc**	**24"oc**	**32"oc**	**40"oc**	**48"oc**	**72"oc**	**96"oc**	**120"oc**
Lightweight Block (105 pcf)	**6"**	58	41	35	32	31	30	28	27	26
	8"	78	55	47	43	40	39	36	35	34
	10"	100	68	57	51	48	46	42	41	40
	12"	121	80	66	59	55	53	48	46	44
	14"	143	93	76	67	62	59	53	51	49
	16"	164	105	85	76	70	66	59	56	54
Medium Weight Block (115 pcf)	**6"**	60	43	37	34	33	32	30	29	28
	8"	81	57	49	45	43	41	39	37	37
	10"	103	71	60	54	51	49	46	44	43
	12"	125	84	70	63	59	56	51	49	48
	14"	147	96	80	71	66	63	57	54	53
	16"	168	109	90	80	74	70	63	60	58
Normal Weight Block (135 pcf)	**6"**	64	47	41	39	37	36	34	33	32
	8"	86	63	55	51	48	47	44	43	42
	10"	109	77	66	61	58	55	52	50	49
	12"	132	90	77	70	66	63	58	56	55
	14"	154	104	87	79	74	70	65	62	60
	16"	177	118	98	88	82	78	71	68	66

© 2026 ENERCALC, LLC

Weights of Masonry Walls (pcf) - Using 105 pcf Grout

		Grouted Cell Spacing								
Block Type	**CMU Wall Thickness**	**8"oc (solid)**	**16"oc**	**24"oc**	**32"oc**	**40"oc**	**48"oc**	**72"oc**	**96"oc**	**120"oc**
Lightweight Block (105 pcf)	**6"**	49.5	36.8	32.3	30.0	29.3	28.5	27.0	26.3	25.5
	8"	66.3	49.0	43.0	40.0	37.8	37.0	34.8	34.0	33.3
	10"	84.0	59.8	51.5	47.0	44.8	43.3	40.3	39.5	38.8
	12"	100.8	69.8	59.3	54.0	51.0	49.5	45.8	44.3	42.8
	14"	118.0	80.3	67.5	60.8	57.0	54.8	50.3	48.8	47.3
	16"	134.8	90.3	75.3	68.5	64.0	61.0	55.8	53.5	52.0
Medium Weight Block (115 pcf)	**6"**	51.5	38.8	34.3	32.0	31.3	30.5	29.0	28.3	27.5
	8"	69.3	51.0	45.0	42.0	40.5	39.0	37.5	36.0	36.0
	10"	87.0	62.8	54.5	50.0	47.8	46.3	44.0	42.5	41.8
	12"	104.5	73.5	63.0	57.8	54.8	52.5	48.8	47.3	46.5
	14"	122.0	83.5	71.5	64.8	61.0	58.8	54.3	52.0	51.3
	16"	138.8	94.3	80.0	72.5	68.0	65.0	59.8	57.5	56.0
Normal Weight Block (135 pcf)	**6"**	55.8	42.8	38.3	36.8	35.3	34.5	33.0	32.3	31.5
	8"	74.3	57.0	51.0	48.0	45.8	45.0	42.8	42.0	41.3
	10"	93.0	69.0	60.8	57.0	54.8	52.5	50.3	48.8	48.0
	12"	111.5	79.8	70.0	64.8	61.8	59.5	55.8	54.3	53.5
	14"	129.3	91.5	78.8	72.8	69.0	66.0	62.3	60.0	58.5
	16"	147.8	103.0	88.0	80.5	76.0	73.0	67.8	65.5	64.0

These wall weights were adapted from TEK 14-13B, which lists values only for 140 pcf grouted walls. The weights were derived by isolating the grout component (fully-grouted wall weight minus non-grouted wall weight), scaling that grout weight by 105 pcf / 140pcf, and adding the adjusted grout weight back to the non-grouted wall weight baseline.

© 2026 ENERCALC, LLC

Net Equivalent Solid Thickness (EST) - Mortar at Face Shells Only (in)

	Grouted Cell Spacing								
CMU Wall Thickness	**8"oc (solid)**	**16"oc**	**24"oc**	**32"oc**	**40"oc**	**48"oc**	**72"oc**	**96"oc**	**120"oc**
6"	5.625	3.883	3.258	2.942	2.750	2.625	2.417	2.317	2.250
8"	7.625	5.167	4.275	3.833	3.567	3.392	3.092	2.942	2.858
10"	9.625	6.233	4.983	4.367	3.992	3.742	3.325	3.125	3.000
12"	11.625	7.275	5.683	4.892	4.408	4.092	3.558	3.300	3.133
14"	13.625	8.325	6.383	5.408	4.833	4.442	3.792	3.467	3.275
16"	15.625	9.367	7.083	5.933	5.250	4.792	4.025	3.642	3.417

Average Equivalent Solid Thickness (EST) - Mortar at Face Shells Only (in)

	Grouted Cell Spacing								
CMU Wall Thickness	**8"oc (solid)**	**16"oc**	**24"oc**	**32"oc**	**40"oc**	**48"oc**	**72"oc**	**96"oc**	**120"oc**
6"	5.625	4.108	3.633	3.392	3.250	3.158	3.000	2.917	2.867
8"	7.625	5.483	4.808	4.475	4.267	4.133	3.908	3.800	3.733
10"	9.625	6.733	5.825	5.367	5.092	4.908	4.608	4.458	4.367
12"	11.625	7.917	6.75	6.175	5.825	5.592	5.200	5.008	4.892
14"	13.625	9.108	7.683	6.975	6.550	6.267	5.792	5.558	5.417
16"	15.625	10.292	8.617	7.783	7.275	6.942	6.383	6.108	5.942

Moment of Inertia of Net Cross-Sectional Area of Masonry, I_n (in^4/ft)

	Grouted Cell Spacing								
CMU Wall Thickness	**8"oc (solid)**	**16"oc**	**24"oc**	**32"oc**	**40"oc**	**48"oc**	**72"oc**	**96"oc**	**120"oc**
6"	178.0	155.1	146.8	142.7	140.2	138.6	135.8	134.5	133.6
8"	443.3	378.6	355.3	343.7	336.7	332.0	324.3	320.4	318.0
10"	891.7	719.3	656.2	624.6	605.7	593.1	572.0	561.5	555.2
12"	1571.0	1208.9	1076.3	1010.1	970.3	943.8	899.6	877.5	864.2

© 2026 ENERCALC, LLC

	Grouted Cell Spacing								
14"	2529.4	1873.2	1632.9	1512.8	1440.7	1392.7	1312.6	1272.6	1248.6
16"	3814.7	2737.2	2342.7	2145.5	2027.1	1948.2	1816.7	1751.0	1711.5

Radius of Gyration of the Average Cross-Sectional Area of Masonry, r_{ave} (in)

	Grouted Cell Spacing								
CMU Wall Thickness	**8"oc (solid)**	**16"oc**	**24"oc**	**32"oc**	**40"oc**	**48"oc**	**72"oc**	**96"oc**	**120"oc**
6"	1.62	1.79	1.87	1.91	1.94	1.96	2.00	2.02	2.03
8"	2.20	2.43	2.53	2.59	2.63	2.66	2.71	2.74	2.76
10"	2.78	3.04	3.16	3.24	3.29	3.33	3.39	3.43	3.45
12"	3.36	3.64	3.79	3.88	3.95	3.99	4.07	4.12	4.15
14"	3.93	4.25	4.41	4.51	4.59	4.64	4.74	4.79	4.83
16"	4.51	4.84	5.02	5.14	5.22	5.28	5.39	5.45	5.49

Face-Shell Thickness of Masonry, t_{fs} (in)

Concrete Masonry Unit Wall Thickness					
6"	**8"**	**10"**	**12"**	**14"**	**16"**
1.00	1.25	1.25	1.25	1.25	1.25

Compression Section Width for Partially Grouted Walls, b_w

Grouted Cell Spacing								
8"oc (solid)	**16"oc**	**24"oc**	**32"oc**	**40"oc**	**48"oc**	**72"oc**	**96"oc**	**120"oc**
Number of open cells within the compression section, n_{open}								
0	1	2	3	4	5	8	11	14

© 2026 ENERCALC, LLC

		Grouted Cell Spacing								
CMU Wall Thickness	**Cell Width b_{cell}**	**Ratio of solid width-to-total width of the compression section, b_w / b**								
6"	6.3125"	1.000	0.605	0.474	0.408	0.369	0.342	0.299	0.277	0.264
8"	6.3125"	1.000	0.605	0.474	0.408	0.369	0.342	0.299	0.277	0.264
10"	6.125"	1.000	0.617	0.490	0.426	0.388	0.362	0.319	0.298	0.285
12"	6.125"	1.000	0.617	0.490	0.426	0.388	0.362	0.319	0.298	0.285
14"	6.125"	1.000	0.617	0.490	0.426	0.388	0.362	0.319	0.298	0.285
16"	6.125"	1.000	0.617	0.490	0.426	0.388	0.362	0.319	0.298	0.285

Definitions:

b_{cell} *= width of a single open cell*

n_{open} *= total number of open (ungrouted) cells within the unit width of compression section*
= (Grout Spacing) / 8" - 1

b *= unit width of compression section*

b_w *= unit width of the compression section minus the sum of the length of open (ungrouted) cells*
$= b - n_{open} * b_{cell}$

$\frac{b_w}{b}$ $= (b - n_{open} * b_{cell}) / b = 1 - (n_{open} * b_{cell}) / b$

© 2026 ENERCALC, LLC

14.4 Appendix D - Summary of Concrete & Masonry Design Formulas

CONCRETE (SD)

ϕ = .90 for flexure
= .75 for shear
= .55 for plain concr flexure/shear

$\rho_{bal} = .85 \frac{f_c'}{f_y} \beta \left(\frac{87{,}000}{87{,}000 + f_y} \right) [\beta = 0.85]$

$\rho_{max} = .75_{bal}$

$\rho_{min} = 200/f_y$

E_s = 29,000,000 psi

$E_c = 57{,}000 \sqrt{f_c}$

$n = E_s/E_c$

$a = \frac{A_s f_y}{.85 f_c b}$

The general solution for A_s (per CRSI) =

$$\frac{1.7 f_c' bd}{2 f_y} - \frac{1}{2} \sqrt{\frac{2.89 (f_c' bd)^2}{f_y^2} - \frac{6.8 f_c' b M_u}{\phi f_y'^2}}$$

For b = 12", f_y' = 60 ksi , this reduces to:

$A_s =$

$$0.17 f_c' d - \sqrt{.029 (f_c' 4)^2 - .0063 f_c' M_u}$$

$M_n = A_s f_y (d - a/2)$

$M_u \le \phi M_n$

$$\ell_{hb} = \frac{0.02 f_y d_b \times 0.7}{\sqrt{f_c}} \left(\frac{A_s \text{ req'd}}{A_s \text{ provided}} \right)$$

or $.8 d_b$ or 6"

*ℓ_{db} (#6 and smaller)

$$= \frac{0.024 d_b f_y}{\sqrt{f_c}} \left(\frac{A_s \text{ req'd}}{A_s \text{ provided}} \right)$$

*ℓ_{db} (#7 and larger)

$$= \frac{.03 d_b f_y}{\sqrt{f_c}} \left(\frac{A_s \text{ req'd}}{A_s \text{ provided}} \right)$$

* From ACI 12.2.3

Lap length Class B splice = 1.3 ℓ_{db}

$v_c = 2 \sqrt{f_c} bd$

Plain concr tension = $5\phi \sqrt{f_c}$

Plain concr shear = $2\phi \sqrt{f_c}$

MASONRY (WSD)

F_s = 0.5 f_y (24,000 psi max.)

E_s = 29,000,000 psi

E_m = 750 f_m

$n = E_s/E_m$

F_b = .33 f_m

$v_u = 1.0 \sqrt{f_m}$ (50 psi max)

$\rho = A_s/bd$

$k = \sqrt{(np)^2 + 2np} - np$

$j = 1 - k/3$

$M_s = F_s A_s jd$

$M_m = F_b bd^2 (kj/2) = Kdb^2$

$v_a = V/bd$

l_d = .002$d_b F_s$ (but not less than 12")

MASONRY (LRFD)

$a = A_s f_y / 0.80 f_m b$

$M_n = A_s f_y (d - a/2)$

ϕ = 0.90

© 2026 ENERCALC, LLC

14.5 Appendix E - References Used For The Development Of This Program

- ACI 318-14, published by the American Concrete Institute.
- International Building Code (IBC) 2015 and 2018, published by the International Code Council (ICC).
- Building Code Requirements for Masonry Structures, TMS 402-13/ACI 530-13/ASCE 5-13, TMS 402-16.
- Minimum Design Loads for Buildings and Other Structures, ANSI/ASCE 7-05, ANSI/ASCE 7-10, and ANSI/ASCE 7-16.
- Design of Reinforced Masonry Structures, Concrete Masonry Association of California and Nevada, 1997.
- Foundation Analysis and Design, Fifth Edition, by Joseph E. Bowles, published by McGraw-Hill.
- Reinforced Masonry Engineering Handbook, Fifth Edition, by J. Amrhein, published by the Masonry Institute of America
- CRSI Handbook, 1996, published by Concrete Reinforcing Steel Institute.
- Design Manual for Segmental Retaining Walls, 3rd Edition, NCMA.
- Reinforced Concrete Design, Sixth Edition, Wang & Salmon, published by Harper & Row.
- Principles of Foundation Engineering, 5th Edition, Braja Das, Thompson.
- Geotechnical Earthquake Engineering, Kramer, Prentice-Hall, 2003.
- The Seismic Design Handbook, 2nd. Edition, Farzad Naeim, Kluwer Academic Publishers, Boston. 2001.
- NEHRP Recommended Provisions for Seismic Regulations for Buildings and Other Structures, Parts 1 and 2, 2002. Edition.
- Foundations and Earth Structures, NAVFAC Design Manual 7.02, 1986.
- Foundation Engineering, 2nd Edition, Peck, Hansen, Thornburn, Wiley, 1974.
- Soil Mechanics in Engineering Practice, Tarzaghi and Peck, Wiley, 1967.
- Design and Performance of Earth Retaining Structures, ASCE Paper by Robert Whitman, 1990.
- Lateral Stresses & Design of Earth-Retaining Structures, ASCE Paper by Seed and Whitman, 1970.

14.6 Appendix F - Rankine and Coulomb Formulas

The three methods of inputting active soil pressure are the Equivalent Fluid Pressure (EFP) method, Rankine method and Coulomb method.

© 2026 ENERCALC, LLC

With the Equivalent Fluid Pressure (EFP) method, the soil active pressure is defined by an equivalent fluid pressure in psf per foot of depth (e.g. 35 psf).

With the Rankine or Coulomb method, you can input the angle of internal friction and the program will compute the horizontal (and vertical, if applicable) K_a by the respective formulas.

For a level backfill, both the Rankine and Coulomb formulas give the same result, except that the latter also takes into account frictional resistance of the wall surface, and inclination of the wall surface (i.e. batter).

The Rankine Formula

$$K_a = \cos\beta \frac{\cos\beta - \sqrt{\cos^2\beta - \cos^2\phi}}{\cos\beta + \sqrt{\cos^2\beta - \cos^2\phi}}$$

$$K_a = (\text{horiz.}) = \cos\beta\, K_a$$

For level backfill can be written as: $K_a = \tan^2\left(45 - \frac{\phi}{2}\right)$ or $= \frac{1-\sin\phi}{1+\sin\phi}$

The Coulomb Formula

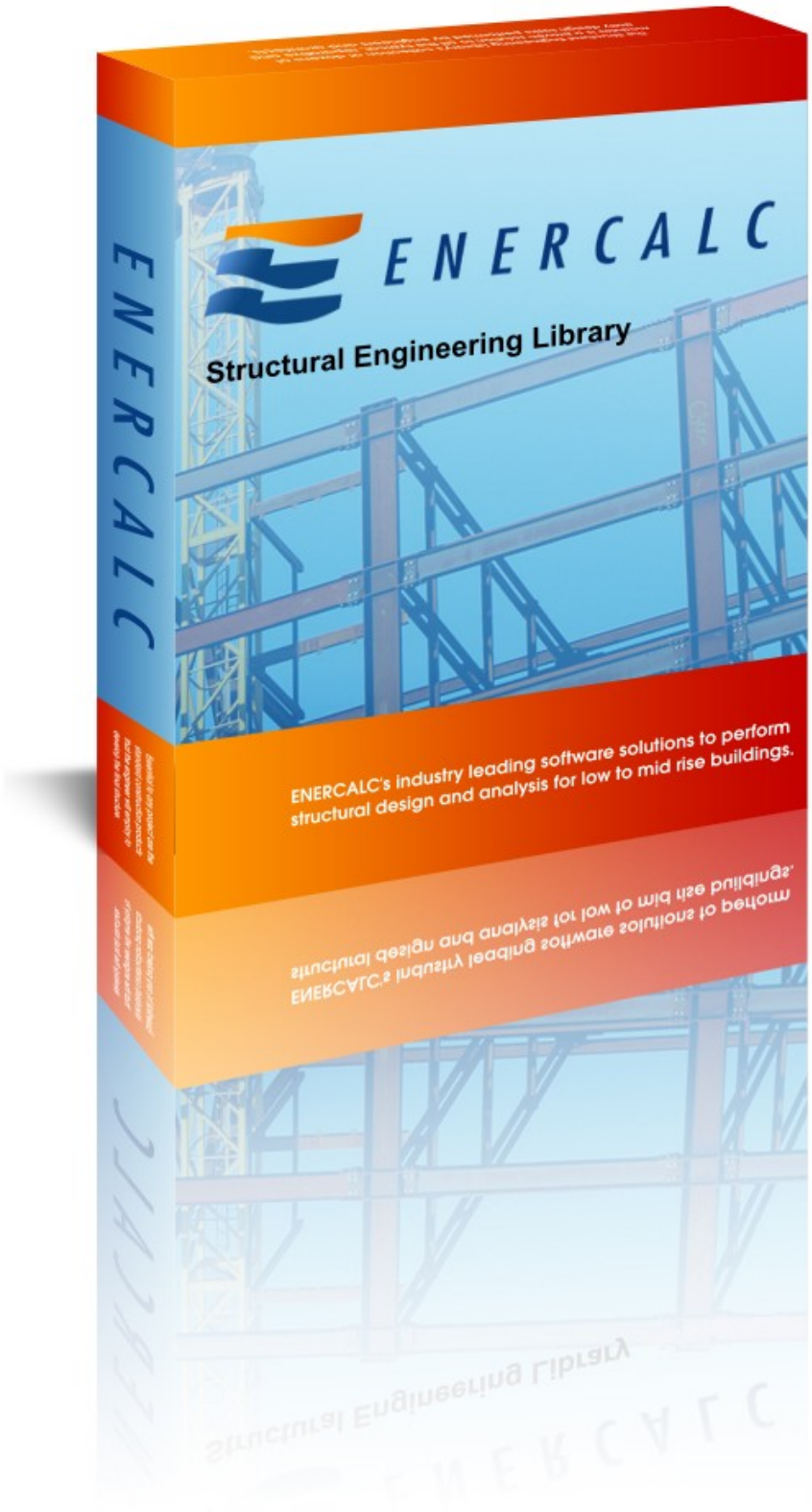
$$K_a = \frac{\sin^2(\alpha+\phi)}{\sin^2\alpha \sin(\alpha-\delta)\left[1+\sqrt{\frac{\sin(\phi+\delta)\sin(\phi-\beta)}{\sin(\alpha-\delta)\sin(\alpha+\beta)}}\right]^2}$$

$$K_a\,(\text{horiz.}) = \cos\delta\, K_a$$

For both formulas:

β = Angle of backfill slope
ϕ = Angle of internal friction
α = 90° - wall slope angle from horizontal
δ = Angle of friction between soil and wall

© 2026 ENERCALC, LLC


ENERCALC
ENERCALC
Structural Engineering Library
ENERCALC's industry leading software solutions to perform structural design and analysis for low to mid rise buildings.